9th International Conference on Multiphase Flow
May 22-27, Firenze, Italy

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

ICMF2016 Chairman
CONTENT

Paper Index

Monday I
Tuesday XIII
Wednesday XXV
Thursday XXXV
Friday XLV
Posters XLIX
Author Index LIII
Abstracts 1
## PARALLEL SESSIONS I

### Mo 9:50-11:10 Bio-Fluids

**Room:** Affari-2.B  
**NUMERICAL ANALYSIS OF THE FLOW AND PARTICLE PATTERN IN THE HUMAN NASAL CAVITY**  
Farnoud Ali, Cui Xinguang, Baumann Ingo, Guthel Eva  
**THROMBUS/BUBBLES FORMATION AND MULTIPHASE FLOW IN BLOOD VESSEL AFTER LASER IRRADIATION**  
Wu Wenzhuan, Li Dong, Chen Bin  
**NANOPARTICLE DEPOSITION ANALYSIS IN HUMAN AND RAT NASAL CAVITIES: A NUMERICAL COMPARISON STUDY**  
Dong Jingliang, Shang Yidan, Inthavong Kiao, Tu Jiyuan, Chen Rui, Chen Chunying

### Mo 9:50-11:10 Boiling, Condensation, Evaporation

**Room:** Affari-3.D  
**EVAPORATION HEAT TRANSFER COEFFICIENT OF R410A INSIDE MICRO/MINI-CHANNEL**  
Oh Jong-Taek, Chien Nguyen Ba, Choi Kwang-II  
**NUMERICAL SIMULATIONS OF NUCLEATE BOILING**  
Huber Gregory, Tanguy Sebastien, Colin Catherine  
**DIRECT NUMERICAL SIMULATION OF INTERACTING EVAPORATING DROPLETS**  
Dairay Thibault, Naso Aurore, Spelt Peter  
**RESEARCH ON MARANGONI CONDENSATION MODES OF ETHANOL-WATER MIXTURES ON A HORIZONTAL SURFACE**  
Li Yong, Wang Jinshi, Chen Nana, Yan Junjie, Xia Kai

### Mo 9:50-11:10 Cavitation/Nucleation

**Room:** Affari-3.C  
**LES MODELING OF CAVITATION FLOW IN A DIESEL INJECTOR NOZZLE**  
Chen Boxiang, Falgout Zachary, Linne Mark, Oevermann Michael  
**CAVITATION MODELLING IN MICRO CHANNELS**  
Kayakol Nuray  
**SIMULATION OF A HIGH PRESSURE FUEL PUMP, INCLUDING CAVITATION EFFECTS, WITH TWO MULTIPHASE MODELS**  
Koukouvinis Foivos C., Karathanassis Ioannis, Li Jason, Wang Lifeng, Gavaises Manolis  
**NUMERICAL SIMULATION OF VAPOR-GAS BUBBLE DYNAMICS BASED ON MOLECULAR GAS DYNAMICS**  
Nagayama Takahiro, Kon Misaki, Kobayashi Kazumichi, Watanabe Masao, Fuji Hiroyuki, Takahira Hiroyuki

### Mo 9:50-11:10 Collision, Agglomeration and Breakup

**Room:** Affari-3.A  
**TERNARY DROP COLLISIONS**  
Planchet Carole, Hinterbichler Hannes, Brenn Guenter  
**ANALYSIS OF THE MECHANISMS FOR GAS BUBBLE AND DROPLET BREAKUP IN TURBULENT FLOWS**  
Andersson Ronnie  
**NOVEL PHENOMENOLOGICAL MODELS FOR VISCUS BINARY DROPLET COLLISIONS**  
Finotello Giulia, Deen Niels G., Padding Johan, Kuipers J. A. M (hans), Jongsma Alfred, Innings Fredrik  
**MULTI-SCALE MODELING OF BUBBLE COALESCENCE AND BREAKUP IN HETEROGENEOUS BUBBLY FLOWS**  
Battistella Alessandro, Roghair Ivo, Van Sint Annaland Martin

### Mo 9:50-11:10 Computational Techniques for Multiphase Flows

**Room:** Affari-1.A  
**GENERALIZED IMMERSED BOUNDARY METHOD FOR UNSTRUCTURED MESH**  
Abdol Azis Mohd Hazmil Syahidy, Van Wachem Berend  
**A NEW APPROACH TO DEFINE A NON-ITERATIVE IMMERSED BOUNDARY METHOD FOR SPHERICAL PARTICLES OF ARBITRARY DENSITY RATIO**  
Tschisgale Silvio, Kempe Tobias, Frohlich Jochen
AN IMMERSED BOUNDARY METHOD FOR COMPLEX-SHAPED BUBBLES REPRESENTED BY SPHERICAL HARMONIC FUNCTIONS
Krull Benjamin, Tschisgale Silvio, Frohlich Jochen
3/6
IMMERSED BOUNDARY – THERMAL LATTICE BOLTZMANN METHOD FOR PARTICLE SEDIMENTATION IN NON-NEWTONIAN LIQUID
Amiri Delouei Amin, Mohsen Nazari, Mohammad Hasan Kayhani, Ahmadi Goodarz
4/1

Mo 9:50-11:10
Environmental and Geophysical Flows
Room: Affari-3.E
TANGENTIAL INLET SUPersonic SEPARATORS: A NOVEL APPARATUS FOR GAS PURIFICATION
Wen Chuang, Yang Yan, Walther Jens
4/2
CONTRAIL MODELING AND SIMULATION
Paoli Roberto
4/3
ESCOSED: OBSERVATIONS OF MORPHODYNAMICS DURING BORA AT THE MOUTH OF THE MISA RIVER
Calantoni Joseph, Sheremet Alexandru, Broccolini Maurizio, Postacchini Matteo
4/4
THE EFFECT OF BED GEOMETRY AND COHERENT VORTICES ON INCIPIENT GRANULAR BED MOTION IN OSCILLATORY FLOW: LABORATORY MEASUREMENTS AND NUMERICAL SIMULATIONS
Simeonov Julian, Frank Donya, Calantoni Joseph
4/5

Mo 9:50-11:10
Experimental Methods for Multiphase Flows
Room: Affari-4.A
STUDY OF A STATIONARY TAYLOR BUBBLE USING THE BRIGHTNESS BASED LASER INDUCED FLUORESCENCE TECHNIQUE (BBLIF)
Vasques Joao, Cherdantsev Andrey, Harr David, Hewakandamby Buddhioka, Azzopardi Barry James
4/6
ACOUSTICAL CHARACTERISTICS OF TWO-PHASE HORIZONTAL INTERMITTENT FLOW THROUGH AN ORIFICE
Lee Joohwa Sarah, Violato Danielle, Pollike Wolfgang
5/1
SPATIAL- AND TIME-RESOLVED MEASUREMENTS OF LIQUID HOLD-UP DISTRIBUTION IN A PACKED BED USING ELECTRICAL RESISTANCE TOMOGRAPHY AND VOIDAGE PROBES
Singh Brajesh Kumar, Jain Ekta, Buwa Vivek
5/2
SIMULTANEOUS BRIGHTNESS-BASED AND PLANAR LASER-INDUCED FLUORESCENCE MEASUREMENT OF DOWNWARDS ANNULAR FLOW
An Jae Sik, Cherdantsev Andrey, Zadrazil Ivan, Markides Christos N.
5/3

Mo 9:50-11:10
Industrial Applications
Room: Affari-0.A
MECHANISTIC SIMULATION OF VERTICAL SLUG FLOW WITH THE ONE-DIMENSIONAL TWO-FUID MODEL
Galleni Francesco, Issa Raad
5/4
VISCous OIL-WATER HORIZONTAL WAVY STRATIFIED FLOW: CFD AND EXPERIMENTAL DATA COMPARISON
Avila Ricardo, Arroxeas Andre, Souza De Castro Marcelo, Hernandez Rodriguez Oscar Mauricio
5/5
COMPARISON OF NUMERICAL METHODS FOR SLUG CAPTURING WITH THE TWO-FUID MODEL
Hendrix Maurice, Eskerud Smith Ivar, Van Zwieten Joost, Sanderse Benjamin
5/6
EFFECTS OF SURFACE ROUGHNESS AND INTERPARTICLE COLLISIONS ON ELBOW EROSION
Dos Santos Vinicius Vagundes, De Souza Francisco Jose, Ribeiro Duarte Carlos Antonio
6/1

Mo 9:50-11:10
Interfacial Flows
Room: Affari-2.A
A HYBRID NUMERICAL METHOD FOR TWO-PHASE FLOW WITH SOLUBLE SURFACTANT Booty Michael, Siegel Michael
6/2
LIQUID FILM FLOW WITH SOLUBLE SURFACTANTS: THEORY AND EXPERIMENT
Georgantaki Anna, Karapetsas George, Vlachogiannis Michaelis, Bontozoglou Vasileios
6/3
ON THE INFLUENCE OF SURFACTANTS ON THE WAKE AND SHAPE OF GASEOUS BUBBLES ASCENDING IN LIQUIDS
Schranner Felix, Heinz Johannes, Adams Nikolaus
6/4
FINITE VOLUME / FINITE AREA INTERFACE-TRACKING METHOD FOR SOLUBLE SURFACTANTS
Marshall Holger, Pesci Chiara, Weber Paul S., Bothe Dieter
6/5
Mo 9:50-11:10  
**Multiphase Flow in Heat and Mass Transfer**  
*Room: Affari-2.C*  
A NEW SUBMODEL FOR MOVING PARTICLES UNDERGOING PHASE CHANGE SOLID-LIQUID-GAS  
Bansal Hemant, Nikrityuk Petr  
DIRECT NUMERICAL SIMULATION (DNS) OF FLUID-SOLID MASS TRANSFER USING A GHOST-CELL BASED IMMERSED BOUNDARY METHOD  
Lu Jiangtao, Peters E.A.J.F., Kuipers J. A. M (hans)  
DIRECT NUMERICAL SIMULATION OF HEAT TRANSFER IN DENSE PARTICLE-LIQUID TWO-PHASE MEDIA  
Gu Jingchen, Kondo Katsuya, Takeuchi Shintaro, Kajishima Takeo  
DETAILED NUMERICAL SIMULATIONS OF RESOLVED MULTI-COMPONENT FUEL DROPLETS EVAPORATING IN HOMOGENEOUS ISOTROPIC TURBULENCE  
Palmore John, Desjardins Olivier  

---  

Mo 9:50-11:10  
**Particle and Drop Dynamics**  
*Room: Villa-Sala Verde*  
OBLIQUE IMPACT OF DROPLETS ON FLOWING LIQUID FILMS  
Che Zhizhao, Ismail Renad, Adebayo Idris, Matar Omar K.  
DROP IMPACT ON SOLIDS  
Josserand Christophe  
PHENOMENA OF DROP IMPACT ON HOT SURFACES  
Breitenbach Jan, Roisman Ilia V., Tropea Cameron  
SPLASH OF A LIQUID DROP IMPACTING ONTO A WALL FILM OF DIFFERENT LIQUID  
Kittel Hannah, Roisman Ilia V., Tropea Cameron  

---  

Mo 9:50-11:10  
**Particle-Laden Flows**  
*Room: Villa-Auditorium*  
TURBULENCE MODULATION IN CONCENTRATED, HIGH REYNOLDS NUMBER SOLID-LIQUID FLOWS  
Breakley David, Agrawal Yogesh, Amini Parsa, Ghaemi Sina, Nobes David, Sanders Sean  
ROLE OF PREFERENTIAL CONCENTRATION IN THE DESTABILIZATION OF SHEARED PARTICLE-LADEN FLOWS  
Kasbaoui Mohamed Housssem, Koch Donald, Desjardins Olivier  
PARTICLE CONCENTRATIONS IN RIB-ROUGHENED WALL TURBULENCE  
Zhao Lihao, Andersson Helge  
SIMULATION OF TURBULENT FLOW AND PARTICLE TRANSPORT AND DEPOSITION USING LATTICE BOLTZMANN METHOD AND RANS MODEL  
Sajjadi Hasan, Salmanzadeh Mazayr, Ahmadi Goodarz, Jafari Saeed  

---  

Mo 9:50-11:10  
**Multiphase Pipe Flows**  
*Room: Affari-3.B*  
A SLUG CAPTURING METHOD APPLIED TO UNCONVENTIONAL SCENARIOS: HIGH VISCOSOUS OILS AND ACTUAL PIPELINE GEOMETRIES  
Ferrari Marco, Bonzanini Arianna, Poesio Pietro  
A MECHANISTIC APPROACH FOR HORIZONTAL SLUG FLOW FROM TWO-PHASE (GAS-LIQUID) TO THREE-PHASE WITH THE FORMATION OF GAS HYDRATES  
Bassani Carlos, Barbuto Fausto, Sum Amadeu, Morales Rigoberto  
AN EXPERIMENTAL INVESTIGATION ON THE EVOLUTION OF HIGHLY VISCOUS OIL SLUG LENGTHS IN HORIZONTAL PIPES  
Losi Gianluca, Poesio Pietro  
LINEAR STABILITY ANALYSES OF DISCRETE REPRESENTATIONS OF THE TWO-FUID MODEL FOR STRATIFIED PIPE FLOW  
Akselsen Andreas, Nydal Ole Jørgen  

---  

Mo 9:50-11:10  
**Bubble Dynamics**  
*Room: Affari-2.D*  
EXPERIMENTAL INVESTIGATION OF THE FLOW STRUCTURE AROUND TAYLOR BUBBLES IN VERTICAL DUCTS IN THE PRESENCE OF DISPERSED BUBBLES  
Paladino Emilio, Cerqueira Rafael, Brito Ronei, Maliska Clotis, Meneghini Cristiano  
NUMERICAL STUDY OF TAYLOR BUBBLES RISING THROUGH SUDDEN EXPANSIONS AND CONTRACTIONS  
Zhong Chengsi, Knikker Ronnie, Reveillen Remi  
TAYLOR BUBBLE DESTRUCTION AT SUDDEN EXPANSIONS  
Carter William, Agunjieka Ezekiel, Azzopardi Barry James  
INTERACTION OF A BUBBLE RISING IN A THIN-GAP CELL WITH THE VORTICES RELEASED BY A COMPANION BUBBLE  
Ern Patricia, Filletta Audrey, Roig Veronique  

---
PARALLEL SESSIONS II

Mo 11:30-12:50  Bubbly Flows  
Room: Affari-2.B  
DIRECT NUMERICAL SIMULATION OF A TURBULENT PIPE FLOW WITH DEFORMABLE BUBBLES TO INVESTIGATE THE DYNAMICS OF BUBBLES  
Kim Kyung, Kim Jungwoo, Choi Haecheon  
10/2  
NUMERICAL SIMULATION OF A SINGLE RISING BUBBLE BY A VOF METHOD WITH ENHANCED CURVATURE ESTIMATION BASED ON POINT-CLOUD SAMPLING  
Kassar Bruno, Carneiro Joao, Nieckele Angela  
10/3  
NUMERICAL SIMULATION OF UNDERWATER EXPLOSION WITH GHOST FLUID METHOD  
Chen Hong, Zhu Weibing, Khoo Boo Cheong  
10/4  
EFFECT OF BUBBLE CLUSTERS ON TURBULENT FLOW STRUCTURES IN UPWARD BUBBLY FLOW  
Zhang Wenhao, Sekiguchi Yoshito, Nakanishi Hiroaki, Sakakibara Jun, Takagi Shu  
10/5

Mo 11:30-12:50  Boiling, Condensation, Evaporation  
Room: Affari-3.D  
NUMERICAL SIMULATION OF MICROLAYER FORMATION DURING THE INERTIA-CONTROLLED GROWTH PHASE OF BUBBLES IN NUCLEATE BOILING  
Guion Alexandre, Buongiorno Jacopo, Zaleski Stephane, Afkhani Shahriar  
10/6  
CONSIDERING THE SUBLAYER: A SUB MODEL OF CFD APPROACHES  
Ding Wei, Krepper Eckhard, Hampel Uwe  
11/1  
PRELIMINARY RESULTS OF SOBER-SJ10 ON SINGLE BUBBLE POOL BOILING HEAT TRANSFER IN MICROGRAVITY  
Wu Ke, Zhao Jianfu, Li Hui-Xiong, Li Kai  
11/2

Mo 11:30-12:50  Cavitation/Nucleation  
Room: Affari-3.C  
NUMERICAL ANALYSIS OF BUBBLE DYNAMICS WITH VAPOR–LIQUID BOUNDARY CONDITIONS DERIVED FROM MOLECULAR GAS DYNAMICS  
Morikawa Shotaro, Jinbo Yoshinori, Takekura Yuki, Kobayashi Kazumichi, Watanabe Masao, Fujii Hiroyuki, Sasaki Koichi  
11/3  
TESTING A BUBBLE DYNAMIC MODEL FOR BULK CAVITATION IN UNDERWATER EXPLOSION SIMULATIONS  
Wedberg Rasmus, Alin Niklas  
11/4  
MODELING CAVITATION BUBBLES IN A FOCUSED ULTRASOUND FIELD  
Maeda Kazuki, Colonius Tim, Kreider Wayne, Maxwell Adam  
11/5  
CAVITATION CONTROL BY ACTIVE MASS INJECTION THROUGH A SLOT CHANNEL IN THE SURFACE OF A 2D HYDROFOIL  
Timoshchavsky Mikhail, Pervunin Konstantin, Markovich Dmitriy  
11/6

Mo 11:30-12:50  Collision, Agglomeration and Breakup  
Room: Affari-3.A  
A NEW EXPERIMENTAL DATASET TO VALIDATE CFD MODELS OF AIRBORNE NANO PARTICLES AGGLOMERATION  
Belut Emmanuel, Christophe Theo  
12/1  
BUBBLE DYNAMICS IN PLUG FLOW REGIME FOR HORIZONTAL GAS-LIQUID FLOW  
Banerjee Jyotirmay, Thaker Jignesh  
12/2  
HYDRATES FORMATION, AGGREGATION AND SLURRY FLOW PROPERTIES IN DIFFERENT MULTIPHASE FLOW PATTERNS  
Ding Lin, Shi Bohui, Lv Xiaofang, Liu Yang, Wu Haihao, Wang Wei, Gong Jing  
12/3  
PREFERENTIAL CONCENTRATION IN COHERENT FLOW STRUCTURES: FOCUS ON PARTICLE-PARTICLE INTERACTIONS  
Kosinski Pawel, Balakin Boris V., Kartushinsky Alexander  
12/4

Mo 11:30-12:50  Computational Techniques for Multiphase Flows  
Room: Affari-1.A  
PERFORMANCE ASSESSMENT OF EULERIAN VERSUS EULERIAN-LAGRANGIAN LEVEL SET IMPLEMENTATIONS FOR THE SIMULATION OF TWO PHASE FLOWS  
Lorieul Gael, Chatelain Philippe, Bartosiewicz Yann  
12/5  
DEVELOPMENT OF A FULLY COUPLED SOLVER FOR THE EULERIAN-EULERIAN APPROACH  
Bartholomew Paul, Marquis Andrew, Denner Fabian, Van Wachem Berend  
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
Dupif Valentin, Massot Marc, Dupays Joel, Laurent Frederique, Le Touze Clement

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

Mo 11:30-12:50 Environmental and Geophysical Flows
Room: Affari-3.E

Environmental and Geophysical Flows
Room: Affari-3.E

Mo 11:30-12:50 Environmental and Geophysical Flows
Room: Affari-3.E

Mo 11:30-12:50 Experimental Methods for Multiphase Flows
Room: Affari-4.A

Experimental Methods for Multiphase Flows
Room: Affari-4.A

Mo 11:30-12:50 Experimental Methods for Multiphase Flows
Room: Affari-4.A

Mo 11:30-12:50 Industrial Applications
Room: Affari-0.A

Industrial Applications
Room: Affari-0.A

Mo 11:30-12:50 Industrial Applications
Room: Affari-0.A

Mo 11:30-12:50 Interfacial Flows
Room: Affari-2.A

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

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 Josep, Sharaf Safa, Hawakandambay Buddhikha, Azzopardi Barry James

HOMOGENEOUS FLOW IN A LARGE SCALE BUBBLE COLUMN: EXPERIMENTAL INVESTIGATION AND CFD MODELLING
Besagni Giorgio, Ziegenhein Thomas, Lucas Dirk, Inzoli Fabio

HYDRODYNAMICS OF BUBBLE COLUMNS IN THE HETEROGENEOUS REGIME
Raimundo Pedro Maximiano, Cartellier Alain, Beneventi Davide, Forret Ann, Augier Frederic

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

VORTEX STRUCTURES OF LOW STEAM QUALITY BOILING TWO-PHASE FLOW IN BUBBLE-TYPE PLATE CHANNEL
Jiang Chen, Yang Zonglin, Bai Bofeng

LAGRANGIAN TRACKING OF DROPLETS EVAPORATING IN A HIGH TURBULENCE BY USING IN-LINE DIGITAL HOLOGRAPHY
Marie Jean-Louis, Tronchin Thibaut, Grosjean Nathalie, Mees Loic, Lance Michel

EXPERIMENTALLY STUDYING OF THE EFFECT OF THE HEATING SURFACE MORPHOLOGY ON CHARACTERISTICS OF HIGHLY SUBCOOLED WATER BOILING
Vasilev Nikolay, Khodakov Konstantin, Varaksin Aleksei, Zeigarnik Yuriy

Mo 14:40-16:00 Cavitation/Nucleation
Room: Affari-3.C
CFD SIMULATION OF CAVITATION IN FUEL NOZZLES
Bretones Alvaro, De Souza Francisco Jose

EFFECTS OF MASS TRANSFER ON THE OSCILLATIONS OF CAVITATION BUBBLES
Zhang Yuning, Gao Yuhang, Du Xiaoce

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

TWO-PHASE FLOW SIMULATIONS OF THE INTER-BLADE VORTICES IN A FRANCIS TURBINE
Wack Jonas, Riedelbauch Stefan, Yamamoto Keita, Avellan Francois

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

RAPID STRETCHING OF NEWTONIAN AND NON-NEWTONIAN LIQUID BRIDGES
Roteman Ilia V., Weickgenannt Christina, Tropea Cameron

JOINT INVESTIGATION OF CLUSTERING AND SETTLING OF HEAVY SUB-KOLMOGOROV PARTICLES IN TURBULENT FLOWS
Sumbekova Sholpan, Aliseda Alberto, Bourgoin Mickael, Cartellier Alain

CLUSTERING, INTERMITTENCY OF RELATIVE VELOCITY AND CAUSTICS OF HEAVY PARTICLES UNDER GRAVITY IN ISOTROPIC TURBULENT FLOWS
Jin Guodong, Chen Jincai, He Guo-Wei

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

DIFFUSE INTERFACE SIMULATIONS OF PHASE SEPARATION OF SHEARED BINARY MIXTURES USING A FINITE VOLUME SOLVER
Mauri Roberto, Di Vitantonio Giuseppe, Galletti Chiara, Lamorgese Andrea
VIII

---

**Mo 14:40-16:00**

**Environmental and Geophysical Flows**

*Room: Affari-3.E*

- **FRAGMENTATION OF A POROUS VISCOELASTIC LIQUID BY RAPID DECOMPRESSION: IMPLICATION TO VOLCANIC ERUPTION**
  Kameda Masaharu, Ichihara Mie

- **INFLUENCE OF ANISOTROPIC PERMEABILITY ON CONVECTION IN POROUS MEDIA**
  De Paoli Marco, Zonta Francesco, Soldati Alfredo

- **CRUDE OIL PLUMES IN CROSSFLOW: EFFECT OF TURBULENT FLOW STRUCTURE ON OIL RESIDENCE IN PLUME**
  Murphy David, Xue Xinzhi, Sampath Kaushik, Katz Joseph

---

**Mo 14:40-16:00**

**Experimental Methods for Multiphase Flows**

*Room: Affari-4.A*

- **LOCAL CONCENTRATION MEASUREMENTS IN THE WAKE OF BUBBLES BASED ON IN-SITU RAMAN SPECTROSCOPY AND STATISTICAL ANALYSIS**
  Schurr Daniela, Guhathakurta Jainabally, Baroud Yousef, Simon Sven, Rinke Gunter, Dittmeyer Roland

- **THE ROLE OF INLET FLOW INSTABILITIES IN LIQUID-LIQUID FLOW**
  Morgan Rhys, Markides Christos N., Zadrazil Ivan

---

**Mo 14:40-16:00**

**Industrial Applications**

*Room: Affari-0.A*

- **A LARGE SCALE MULTI-FLUID/DISPERSED PHASE APPROACH FOR SPRAY GENERATION IN AERONAUTICAL FUEL INJECTORS**
  Zuzio Davide, Estivalezes Jean-Luc, Blanchard Ghislain

- **PRESSURE SWIRL NOZZLES WITH A CONVEX DEFLECTION BODY TO INCREASE THE SPRAY ANGLE**
  Kamplade Jens, Hohlfeld Ivette, Keiz Marcus, Thommes Markus, Walzel Peter

---

**Mo 14:40-16:00**

**Interfacial Flows**

*Room: Affari-2.A*

- **INTERFACE TREATING TECHNIQUE FOR COMPRESSIBLE MULTI-MEDIUM FLOWS**
  Khoo Boo Cheong, Chen Hong, Zhu Weibing

- **HYBRID INTERFACE CAPTURING SCHEME FOR MIXED MULTIPHASE FLOWS**
  Tandon Mohit P., Gada Vinesh H., Elias Jebin, Lo Simon

---

**Mo 14:40-16:00**

**Multiphase Flow in Heat and Mass Transfer**

*Room: Affari-2.C*

- **THE EFFECTS OF NANOPARTICLES IN SUBCOOLED FLOW BOILING IN THE DIFFERENT CHANNELS WITH SAME HYDRAULIC DIAMETER**
  Mehrzad Shams, Alimoradi Hasan, Vallizadeh Ziba

- **EXPERIMENTAL INVESTIGATION OF HIGH PRESSURE WATER-STEAM FLOW CHARACTERS IN SIMULATED CRACKS**
  Zhenhong Yang, Zhang Qiaoling, Bi Qincheng
A NEW MODEL FOR THE PREDICTION OF FRICTIONAL PRESSURE DROP OF AIR-WATER TWO-PHASE FLOWS IN VERTICALLY-DOWNWARD TUBES
Yao Chao, Li Hui-Xiong, Xue Yuqing, Hao Chenyu 25/5

EFFECT OF FLOW OBSTACLES ON THE POOL BOILING CRITICAL HEAT FLUX OF DOWNWARD-FACING METAL HEATER
Son Hong Hyun, Song Kyusang, Jeong Uija, Seo Gwang Hyeok, Jeun Gyooodong, Kim Sung Joong 25/6

Mo 14:40-16:00 Particle and Drop Dynamics
Room: Villa-Sala Verde
SURFACTANT EFFECTS ON DROPLET COALESCENCE WITH A LIQUID-LIQUID INTERFACE
Angeli Panagiota, Voulgaropoulos Victor, Wehllye Hashi Wehllye, Chinaud Maxime 26/1
EXPERIMENTAL EVALUATION OF SURFACTANT CONCENTRATION AT AN INTERFACE OF SINGLE SPHERICAL DROP
Hosokawa Shigeo, Masukura Yuya, Hayashi Kosuke, Tomiyama Akio 26/2
NUMERICAL SIMULATION OF SURFACTANT CONCENTRATION AT AN INTERFACE OF SINGLE SPHERICAL DROP
Hayashi Kosuke, Masukura Yuya, Hosokawa Shigeo, Tomiyama Akio 26/3
EFFECT OF INTERFACIAL PROPERTIES ON DROPLET-INTERFACE COALESCENCE: A COMPUTATIONAL STUDY
Li Kai, Ma Mengyu, Liu Nannan, Jin Hang, Wang Wei 26/4

Mo 14:40-16:00 Particle-Laden Flows
Room: Villa-Auditorium
MODELLING AND SIMULATION OF PARTICLE AGGLOMERATION AND DEPOSITION IN TURBULENT CHANNEL AND PIPE FLOWS
Schutte Koen, Portela Luis, Twera Aris, Henkes Ruud 26/5
NUMERICAL INVESTIGATION OF PARTICLE DEPOSITION IN TURBULENT DUCT FLOWS
Yao Jun, Fairweather Michael, Zhao Yanlin 26/6
THE EFFECTS OF NEAR WALL CORRECTIONS TO HYDRODYNAMIC FORCES ON PARTICLE DEPOSITION AND TRANSPORT IN VERTICAL TURBULENT BOUNDARY LAYERS
Reeks Michael, Jin Chunyu, Potts Ian 27/1
TWO-WAY COUPLING EFFECTS IN PARTICLE-LADEN TURBULENCE: HOW PARTICLE-TRACKING SCHEME AFFECTS PARTICLE AND FLUID STATISTICS
Horwitz Jeremy, Rahmani Mona, Geraci Gianluca, Banko Andrew, Mani Ali 27/2

Mo 14:40-16:00 Multiphase Pipe Flows
Room: Affari-4.A
OIL-WATER FLOW CHARACTERISATION IN HORIZONTAL PIPES USING SIMULTANEOUS LASER-INDUCED FLUORESCENCE AND PARTICLE IMAGE/TRACKING VELOCIMETRY
Ibarra Roberto, Zadravil Ivan, Markides Christos N., Matar Omar K. 27/3
EFFECTS OF AIR INJECTION POSITIONS ON THE FLOW STRUCTURE OF HIGH VISCOSITY OIL IN A LARGE DIAMETER PIPES
Mohammed Shara, Hasan Abbas, Azzopardi Barry James, Dimitriakis Georgios 27/4
MEASUREMENT OF WALL SHEAR STRESS IN CHURN FLOW IN A LARGE DIAMETER VERTICAL RISER
Zangana Mohammed, Azzopardi Barry James 27/5
ENTRANCE EFFECTS ON TWO-PHASE FLOW DEVELOPMENT IN A VERTICAL PIPE: EXPERIMENTS AND SIMULATIONS
Issa Raad, Galleni Francesco, Azzopardi Barry James, Hewakandamby Buddhikha, Escrig Escrig Josep 27/6

Mo 14:40-16:00 Bubble Dynamics
Room: Affari-2.D
CONTROLLED BUBBLE FORMATION BY A FORCING MECHANISM IN A PLANAR CO-FLOW CONFIGURATION
Ruiz-Rus Javier, Bolanos Jimenez Rocio, Gutierrez Montes Candido, Sevilla Alejandro, Bazan Carlos Martinez 28/1
STUDY OF A BUBBLE PRODUCTION IN SHEAR LIQUID-GAS FLOWS USING THE BRIGHTNESS BASED LASER INDUCED FLUORESCENCE TECHNIQUE (BBLIF)
Hann David, Cherdantsev Andrey, Hewakandamby Buddhikha, Azzopardi Barry James 28/2
BUBBLES RELEASE FROM HYDROPHOBIC SITE UNDER MAGNETIC FIELD
Liu Hongbo, Pan Liangming, Chen Tianming 28/3
ON THE PHYSICS OF FIZZINESS: HOW BUBBLE BURSTING CONTROLS DROPLETS EJECTION
Seon Thomas, Ghabache Elisabeth, Antkowiak Arnaud, Josserand Christophe 28/4

IX
## PARALLEL SESSIONS IV

### Mo 16:20-17:40 | Bio-Fluids

**Room: Affari-2.B**

<table>
<thead>
<tr>
<th>Title</th>
<th>Authors</th>
<th>Page</th>
</tr>
</thead>
<tbody>
<tr>
<td>Numerical Simulation of Magnetic Drug Delivery to Human Maxillary Sinus</td>
<td>Nikookar Hamed, Abouali Ormid, Ehteshad Mohammad, Ahmadz Goodarz</td>
<td>28/5</td>
</tr>
<tr>
<td>Modal Behaviour of Three-Dimensional Arterial Flow Structures</td>
<td>Prahl Wittberg Lisa, Bulusu Kartik V., Van Wyk Stevin, Fuchs Laszlo, Plesniak Michael W.</td>
<td>28/6</td>
</tr>
<tr>
<td>Platelet Margination and the Rheology of Whole Blood</td>
<td>Berg Nicolas, Fuchs Laszlo, Prahl Wittberg Lisa</td>
<td>29/1</td>
</tr>
</tbody>
</table>

### Mo 16:20-17:40 | Boiling, Condensation, Evaporation

**Room: Affari-3.D**

<table>
<thead>
<tr>
<th>Title</th>
<th>Authors</th>
<th>Page</th>
</tr>
</thead>
<tbody>
<tr>
<td>Experiments and Models for Transient Boiling</td>
<td>Baudin Nicolas, Ruyer Pierre, Colin Catherine, Sebilleau Julien</td>
<td>29/2</td>
</tr>
<tr>
<td>New Nucleation Boiling Model Devoted to High Pressure Flows</td>
<td>Mimouni Stephane, Benguiugi William, Guingo Mathieu, Lavieville Jerome, Mechtous Namane, Merigoux Nicolas</td>
<td>29/5</td>
</tr>
</tbody>
</table>

### Mo 16:20-17:40 | Cavitation/Nucleation

**Room: Affari-3.C**

<table>
<thead>
<tr>
<th>Title</th>
<th>Authors</th>
<th>Page</th>
</tr>
</thead>
<tbody>
<tr>
<td>An Euler-Lagrange Method with Volumetric Coupling for Cavitating Flows</td>
<td>Schumichen Michel, Rudiger Frank, Frohlich Jochen</td>
<td>29/6</td>
</tr>
<tr>
<td>Void Fraction Flow Field Measurements of Cavitating Flow in the Wake of a Circular Cylinder</td>
<td>Ganesh Harish, Wu Juliana, Sun Tiezhi, Ceccio Steven L.</td>
<td>30/1</td>
</tr>
<tr>
<td>Numerical Simulations of the Unsteady Cavitating Flows Around a Circular Disk and Conical Cavitator</td>
<td>Kim Minjae, Lee Kurchul</td>
<td>30/2</td>
</tr>
<tr>
<td>Relationship Between Motion of Acoustic Cavitaton-Oriented Bubbles and Flocculation Patterns, Depending on Components of Dissolved Gases in Water</td>
<td>Nomata Koji, Yanai Sayuri, Muramatsu Hiroya, Saito Takayuki</td>
<td>30/3</td>
</tr>
</tbody>
</table>

### Mo 16:20-17:40 | Collision, Agglomeration and Breakup

**Room: Affari-3.A**

<table>
<thead>
<tr>
<th>Title</th>
<th>Authors</th>
<th>Page</th>
</tr>
</thead>
<tbody>
<tr>
<td>Behavior of Droplets Impact on Dry and Wet Walls: A Numerical Study</td>
<td>Chen Ming, Zhang Baili, Lou Jing</td>
<td>30/4</td>
</tr>
<tr>
<td>The Breakup of a Viscous Liquid Drop in a High Reynolds Number Shear Flow</td>
<td>Hei Ng Chin, Aliseda Alberto</td>
<td>30/5</td>
</tr>
<tr>
<td>Aerodynamic Breakup of Non-Spherical Droplets</td>
<td>Hardalupas Yannis, Bergeles Konstantinos, Charalampous Georgios, Taylor Alex M.K. P.</td>
<td>30/6</td>
</tr>
<tr>
<td>Particle Growth in Turbulent Flow Under Dynamically Critical Stokes Conditions</td>
<td>Guala Michele, Hong Jiarong</td>
<td>31/1</td>
</tr>
</tbody>
</table>

### Mo 16:20-17:40 | Computational Techniques for Multiphase Flows

**Room: Affari-1.A**

<table>
<thead>
<tr>
<th>Title</th>
<th>Authors</th>
<th>Page</th>
</tr>
</thead>
<tbody>
<tr>
<td>A Three-Phase Numerical Model for Liquid-Gas-Gas Flows with Relaxation Processes</td>
<td>Pelanti Marica</td>
<td>31/2</td>
</tr>
<tr>
<td>The Simulation of Multilayered Media of Fluid and Solid Using the Ghost Method</td>
<td>Feng Zhi Wei, Khoo Boo Cheong, Rong Ji Li</td>
<td>31/3</td>
</tr>
<tr>
<td>A Variational Approach to Design a Numerical Scheme on an Arbitrary Moving Grid for N-Fluid Flow with Thermodynamic Consistency</td>
<td>Vazquez Gonzalez Thibaud, Llor Antoine, Fochesato Christophe</td>
<td>31/4</td>
</tr>
</tbody>
</table>
Mo 16:20-17:40

Experimental Methods for Multiphase Flows

*Room: Affari-4.A*

**FLOW MEASUREMENT IN OPAQUE FLUIDS USING ULTRASOUND IMAGING VELOCIMETRY**  
Poelma Christian, Gurung Arati  
31/6

**X-RAY RADIOSCOPY AS A POWERFUL TOOL FOR VISUALIZATION OF LIQUID METAL BUBBLY FLOWS**  
Roschchupkina Olga, Shevchenko Natalia, Strumpf Erik, Hoppe Dietrich, Eckert Sven  
32/1

**A STUDY OF LIQUID-LIQUID SECONDARY FRAGMENTATION WITH SOLIDIFICATION**  
Hadj-Achour Miloud, Rimbert Nicolas, Castrillon-Escobar Sebastian, Gradeck Michel  
32/2

Mo 16:20-17:40

Industrial Applications

*Room: Affari-0.A*

**EXPERIMENTAL AND COMPUTATIONAL INVESTIGATION OF FLOW INSIDE A MILLING CIRCUIT HYDROCYCLONE**  
Furlan John, Ke Renjie, Shingote Chinmay, Kadambi Jaikrishnan, Visintainer Robert  
32/4

**REMOVAL OF FINE PARTICLES ON A WALL BY HIGH-SPEED AIR JET EJECTED FROM A CAVITY NOZZLE (OPTIMIZATION OF NOZZLE SHAPE)**  
Wakimoto Tatsuro, Nakagawa Atsushi, Soemoto Kazuhiro, Katoh Kenji  
32/5

**NONINVASIVE TEMPERATURE DISTRIBUTION IMAGING OF HIGH VISCOSITY PLASTIC FLOW BY ELECTRICAL CAPACITANCE TOMOGRAPHY**  
Hirose Yusuke, Hata Kazuaki, Takei Masahiro  
32/6

Mo 16:20-17:40

Interfacial Flows

*Room: Affari-2.A*

**A NUMERICAL CLOSE-UP ON THE SPRAY FORMATION FROM A GAS-LIQUID MIXING LAYER**  
Ling Yue, Fuster Daniel, Tryggvason Gretar, Scardovelli Ruben, Zaleski Stephane  
33/1

**INFLUENCE OF DISSOLVED GASES ON ATOMIZATION PROCESSES.**  
Tinon Emmanuelle, Martin Charles-Etienne, Bazile Rudy, Selle Laurent  
33/2

**CONVECTIVE LOCAL HEAT TRANSFER AROUND ELONGATED BUBBLES PROPAGATING IN DOWNWARD LIQUID PIPE FLOW**  
Fershtman Adam, Barnea Dvora, Shemer Lev  
34/2

Mo 16:20-17:40

Multiphase Flow in Heat and Mass Transfer

*Room: Affari-2.C*

**EFFECT OF THE ALIGNED FLOW OBSTACLES ON SUBCOOLED FLOW BOILING IN AN INCLINED RECTANGULAR CHANNEL**  
Jeong Uiju, Son Hong Hyun, Song Kyusang, Seo Gwang Hyeok, Jeon Joon Goo, Kim Sung Joong  
33/5

**HEAT TRANSFER CHARACTERISTICS OF SUBCOOLED WATER FLOW BOILING IN CIRCULAR CHANNELS WITH NONUNIFORM HEATING FLUXES**  
Zhu Ge, Bi Qincheng, Yan Jianguo, Yuan Qizheng, Lu Hai, Pan Hui  
33/6

**HOMOGENIZATION: SIMPLIFIED DROP DEFORMATION ESTIMATE DURING TRANSITION THROUGH ORIFICES**  
Walzel Peter  
34/6

Mo 16:20-17:40

Particle and Drop Dynamics

*Room: Villa-Sala Verde*

**INVESTIGATION ON THE DEFORMATION BEHAVIOR BETWEEN OIL DROPLETS IN AQUEOUS SOLUTION**  
Liu Fangyuan, Wang Wei, Jin Hang, Li Kai, Chang Hongli, Gong Jing  
34/3

**EFFECT OF LIQUID PROPERTIES ON THE INTERFACIAL INSTABILITY AND ATOMIZATION BEHAVIOR OF THE ACOUSTICALLY LEVITATED DROPLET**  
Kono Kengo, Hasegawa Koji, Imai Hiroaki, Ohtake Hiroyasu, Goda Atsushi, Abe Yutaka  
34/4

**HOMOGENIZATION: SIMPLIFIED DROP DEFORMATION ESTIMATE DURING TRANSITION THROUGH ORIFICES**  
Walzel Peter  
34/6
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

Multiphase Pipe Flows

Room: Affari-4.A

MODELLING OF CHURN-ANNULAR FOAM FLOWS
Westende Jos, Shoeibomrani 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, Kijenak 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

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  Bubbly Flows
Room: Affari-2.B

A PENALIZATION METHOD FOR THE SIMULATION OF BUBBLY FLOWS
Morente Antoine, Lavieville Jerome, Legendre Dominique 37/1

TURBULENT FLOWS WITH COALESCING MICROBUBBLES: AN EULER-LAGRANGE APPROACH BASED ON LARGE-EDDY SIMULATION
Hoppe Felix, Breuer Michael 37/2

SIMULATION OF BUBBLY FLOW IN AN INDUSTRIAL ELECTROCHEMICAL REACTOR
Arjmand Mehdi, Vreman Bert, Bokkers Albert, Lij Johan, Pelin Kalle 37/3

EFFECT OF INLET CONDITIONS ON TWO PHASE FLOW PATTERN PREDICTION IN SMALL TUBES
Gregorc Jurij, Zun Iztok 37/4

Tu 9:50-11:10  Boiling, Condensation, Evaporation
Room: Affari-3.D

INFLUENCE OF CONVECTION IN TUBE BUNDLES DURING BOILING

SYNTHESIS OF CRUD AND ITS EFFECTS ON POOL AND SUBCOOLED FLOW BOILING
Coyle Carolyn, Phillips Bren, Lowder Sean, Buongiorno Jacopo, Mcrrell Thomas, Rubner Michael 37/6

TRANSIENT MASS TRANSFER FROM A DROPLET EVAPORATING IN GASEOUS ENVIRONMENT
Tonini Simona, Cossali Gianpietro Elvio 38/1

SUPPRESSION MEASURES OF STEAM EXPLOSIONS WITH AND WITHOUT EXTERNAL PRESSURE PULSE
Furuya Masahiro, Arai Takahiro 38/2

Tu 9:50-11:10  Cavitation/Nucleation
Room: Affari-3.C

EFFECT OF TRIPLE LINE ON SPHERICAL-SUBSTRATE-MEDIATED CONDENSATION
Singha Sanat Kumar, Das Prasanta Kumar, Malti Biswajit 38/3

EFFECT OF THE RECESS GEOMETRY ON CRYOGENIC CAVITATING FLOW THROUGH THE FLOATING-RING CLEARANCE
Oike Mamoru, Kikuchi Masataka, Takada Hitoshi, Sudo Takayuki, Takano Tomoyuki 38/4

BECKER-DÖRING MODEL FOR NON-EQUILIBRIUM STEAM CONDENSATION IN A SUPERSONIC NOZZLE
Hagmeijer Rob, Van Der Weide Edwin, Kalikmanov Vitaly 38/5

INTERFACE CAPTURING SIMULATIONS OF SHOCK-ACCELERATED BUBBLE PAIRS
Shukla Ratnesh 38/6

Tu 9:50-11:10  Collision, Agglomeration and Breakup
Room: Affari-3.A

NUMERICAL STUDY OF WATER DROPS BEHAVIOUR IN OIL UNDER ELECTRIC FIELD
Chen Bin, Huang Meng 39/1

NUMERICAL SIMULATIONS ON COPPER DROPLET COLLISIONS IN THE SLAG CLEANING PROCESS
Yang Hongkun, Wolters Jorg, Pischke Philipp, SoItner Helmuts, Frolich Jochen, Eckert Sven 39/2

EXPERIMENTAL OBSERVATION OF THREE-DIMENSIONAL PARTICLE CLUSTERING IN TURBULENT CHANNEL FLOW
Nemes Andras, Coletti Filippo, Fong Kee Onn, Sloan Nicholas 39/3

DETERMINISTIC AGGLOMERATION MODELS FOR THE EULER-LAGRANGE APPROACH USING LARGE-EDDY SIMULATION
Almohammed Naser, Breuer Michael 39/4

Tu 9:50-11:10  Computational Techniques for Multiphase Flows
Room: Affari-1.A

3D NUMERICAL SIMULATIONS OF MACROSCOPIC FLOWS WITH MOVING CONTACT LINES
Solomenko Zlatko, Spell Peter, Lance Michel, Arix Pascal, Dalmazzone Christine, Haroun Yacine 39/5

DETAILED MODELING OF SLOSHING IN SATELLITES TANK FOR LOW BOND NUMBERS
Lepilliez Mathieu, Tanguy Sebastien 39/6

LBE-DEM SIMULATION OF GAS-SOLID TWO-PHASE IMPINGING STREAM
Xu Wenkai, Gui Nan, Li Zhenlin 40/1

NUMERICAL INVESTIGATION OF DUAL BUBBLE SYSTEM
Al-Behadili Mustapha, Hargreaves David, Hekwakamtemby Buddhikha, Azzopardi Barry James 40/2
Tu 9:50-11:10 Experimental Methods for Multiphase Flows

Room: Affari-4.A
FLAPPING OF AN AXISYMMETRIC LIQUID JET
Delon Antoine, Matas Jean-Philippe, Cartellier Alain 40/3
DROPLET SIZE MEASUREMENT IN STATIC MIXERS USING REFRACTIVE INDEX MATCHING AND LASER-INDUCED FLUORESCENCE
Rudolf von Rohr Philipp, Hafeli Richard, Ruegg Oliver, Altheimer Marco 40/4
DEVELOPMENT OF AN EXPERIMENTAL APPARATUS AND MEASUREMENT TECHNIQUES FOR THE STUDY OF QUASI-REAL SLUG-FLOW IN VERTICAL DUCTS
Cercaqueira Rafael, Paladino Emilio, Brito Ronei, Maliska Clovis 40/5
EXPERIMENTAL CHARACTERIZATION OF INTERFACIAL WAVE STRUCTURE OF A FALLING LIQUID FILM IN A VERTICAL LARGE PIPE DIAMETER
Hasan Abbas, Azzopardi Barry James, Hawekandamby Buddhikha, Alaufi Yousuf, Watson Nicholas, Mohammed Shara 40/6

Tu 9:50-11:10 Industrial Applications

Room: Affari-0.A
VOID FRACTION CHARACTERIZATION THROUGH AN OPTICAL METHOD FOR AIR-WATER UPWARD FLOW ACROSS TUBE BUNDLE
Kanizawa Fabio Toshio, Ribatski Gherhardt 41/1
COMPUTATIONALLY-EFFICIENT SIMULATION OF INDUSTRIAL BUBBLE COLUMN BIOREACTORS: INVESTIGATION OF SCALE AND MIXING EFFECTS ON YIELD
Fletcher David, Mcclure Dale, Kavagnah John, Barton Geoff 41/2
TOWARDS THE NUMERICAL SIMULATION OF TWO-PHASE FLOW AND INTERFACIAL SPECIES TRANSFER IN STRUCTURED PACKINGS
Hill Simon, Acher Thomas, Deising Daniel, Hoffmann Rainer, Rarrek Andreas, Marschall Holger, Bothe Dieter, Klein Harald 41/3
MEASURING LIQUID FLOW IN STRUCTURED PACKINGS USING GAMMA AND X-RAY TOMOGRAPHY
Hoffmann Rainer 41/4

Tu 9:50-11:10 Interfacial Flows

Room: Affari-2.A
A NUMERICAL AND ANALYTICAL STUDY ON THE MICRO-JET INDUCED BY A SHOCK
Bai Xiaolong 41/5
NUMERICAL SIMULATION OF AN AIR JET IN CROSS WATER FLOW WITH APPLICATION TO AIR-LAYER DRAG REDUCTION
Moon Hojun, You Donghyun 41/6
NUMERICAL SIMULATION OF A HIGHLY-VISCOUS MICROJET GENERATED BY AN IMPULSIVE FORCE
Onuki Hajime, Yuto Oi, Tagawa Yoshiyuki 42/1
ANALYSIS OF INTERMITTENT FLOW PATTERNS OF WATER-STEAM FLOW IN SOLAR THERMAL POWER PLANTS WITH DIRECT STEAM GENERATION
Hoffmann Alexander, Hirsch Tobias, Schleicher Eckhard, Pitz-Paal Robert 42/2

Tu 9:50-11:10 Modelling of Multiphase Flows

Room: Affari-2.D
THREE DIMENSIONAL MULTI-PHASE EULERIAN MODEL FOR NON-UNIFORM SEDIMENT-LADEN FLOW
Ota Kazuyuki, Saio Takahiro, Nakagawa Hajime 42/3
THE MODELING AND EXPERIMENTS OF AN ADVANCED OZONATION SYSTEM WITH RADIAL DIFFUSER
Swiderski Karol, Heiniger Bruno, Ramoino Luca, Uttinger Walter, Lakehal Djamel, Narayanan Chidambaram 42/4
NUMERICAL MODELLING OF MULTIPHASE FLOW PHENOMENA COUPLED WITH REACTION KINETICS IN CHEMICAL LOOPING COMBUSTION PROCESS OF A FUEL REACTOR
Chen Luming, Yang Xiaogang, Li Xia, Snape Collin 42/5
TURBULENCE AND MOMENTUM INTERFACIAL TRANSFER IN AN AXISYMMETRIC BUBBLY JET FLOW
Aouadi Aroua, Bellakhal Ghazi, Chahed Jamel 42/6

Tu 9:50-11:10 Multiphase Flow in Heat and Mass Transfer

Room: Affari-2.C
THE EFFECT OF THE CHANNEL INCLINATION ON FLOW PATTERN CHARACTERISTICS DURING GAS-LIQUID FLOW IN A SMALL RECTANGULAR CHANNEL
Loyola Francisco, Ribatski Gherhardt 43/1
EFFICIENT COMPRESSED AIR ENERGY STORAGE THROUGH SPRAY-COOLING
Qin Chao, Loth Eric 43/2
VISUALIZATION AND HEAT TRANSFER OF HIGH PRESSURE CARBON DIOXIDE IN A SMALL HORIZONTAL TUBE
Lyu Haicai, Bi Qincheng, Li Hang, Guo Yajun
FLOW BOILING HEAT TRANSFER AND TWO-PHASE FLOW CHARACTERISTICS IN 4.0 MM TUBE AT MICROGRAVITY CONDITIONS
Valencia Castillo Cesar Manuel, Guglielmetti Luca, Bragelli Leonardo, Saraceno Luca, Zummo Giuseppe

Tu 9:50-11:10  

Particle and Drop Dynamics  
Room: Villa-Sala Verde
ON THE CLASSIFICATION OF DROplet BREAKUP IN AN EXTERIOR STREAM
Rimbert Nicolas, Hadj-Achour Miloud
BINARY DROPLET INTERACTIONS AND COALESCENCE – SUPERPOSITION OF HYDRODYNAMICS AND IONIC EFFECTS
Gebauer Felix, Villvock Jom, Kraume Matthias, Bart Hans-Jorg
CONTACTLESS COALESCENCE TECHNIQUE OF ACOUSTICALLY Levitated DROPLETS USING ULTRASONIC TRANSDUCER ARRAY
Watanabe Ayumu, Watahiki Soma, Niwa Motonori, Goda Atsushi, Kanagawa Tetsuya, Kaneko Akiko, Abe Yutaka

Tu 9:50-11:10  

Particle-Laden Flows  
Room: Villa-Auditorium
DIRECT NUMERICAL SIMULATION OF THE FORMATION OF SEDIMENT PATTERNS UNDER THE ACTION OF AN OSCILLATING FLOW
Mazzuoli Marco, Kidanemariam Aman, Blondeaux Paolo, Vittori Giovanna, Uhlmann Markus
STRONGLY COUPLED PARTICLE-LA DEN FLOWS IN VERTICAL CHANNELS
Fox Rodney, Capecelatro Jesse, Desjardins Olivier
INVESTIGATION OF SUBGRID SCALE EFFECTS TO PARTICLE PAIR DYNAMICS WITH COHERENT VORTEX EXTRACTION
Xiong Yan, Li Jing, Liu Zhohui, Zheng Chuguan
PREFERENTIAL ACCUMULATION AND ENHANCED RELATIVE VELOCITY OF INERTIAL DROPLETS DUE TO INTERACTIONS WITH HOMOGENEOUS ISOTROPIC TURBULENCE
Bateson Colin, Aliseda Alberto

Tu 9:50-11:10  

Multiphase Pipe Flows  
Room: Affari-3.E
COMPARISON OF VOID FRACTION MEASUREMENTS USING DIFFERENT TECHNIQUES IN DOWNWARD TWO-PHASE FLOW
Kawaji Masahiro, Kalaga Dinesh, Tumey Damon, Hernandez Freddy, Kleinbart Simon, Pulido Jorge
LOW GAS LOADING FLOWS IN DOWNWARD INCLINED PIPES
Johansen Stein Tore, Mo Sjur, Meese Ernst A.
STUDIES OF DOWNWARD GAS-LIQUID ANNULAR FLOW
Anupriya Sethu, Jayanti Sreenivas
AN EXPERIMENTAL INVESTIGATION OF INTERFACE SHAPE IN SEPARATED FLOW REGIMES INSIDE THE INCLINED TUBE.
Ahn Taehwan, Yun Byongjo, Jeong Jaejun

Tu 9:50-11:10  

Bubble Dynamics  
Room: Affari-3.B
MODELLING OF TWO SPHERICAL OSCILLATING BUBBLES IN AN ELASTIC TUBE
Takahira Hiroyuki, Kitahara Tatsuya, Ogasawara Toshiyuki
AN EXPERIMENTAL INVESTIGATION INTO THE CLUSTERING OF SPHERICAL BUBBLES RISING ALONG AN INCLINED FLAT PLATE
Ogasawara Toshiyuki, Shirai Shota, Takahira Hiroyuki
INTERACTION OF TWO BUBBLES RISING IN A THIN-GAP CELL: EXPERIMENTS AND MODEL FOR THE VERTICAL ENTRAINMENT
Roig Veronique, Filella Audrey, Em Patricia
ADHESION AND SLIDING DYNAMICS OF AIR BUBBLES ON SUPERAEROPHOBIC SURFACES
Ozbay Ridvan, Kibar Ali, Choi Chang-Hwan

PARALLEL SESSIONS VI

Tu 11:30-12:50  

Bubbly Flows  
Room: Affari-2.B
EXPERIMENTAL INVESTIGATION OF HEAT TRANSPORT IN BUBBLY FLOW
Gvozdic Biljana, Almeras Elise, Mathai Varghese, Sun Chao, Lohse Detlef
FLOW STRUCTURES ANALYSIS DOWNSTREAM GAS SPARGERS OF DIFFERENT ORIFICE PATTERN IN BUBBLE COLUMNS
Moeller Felix, Biebter Andre, Barthel Frank, Schubert Markus, Hampel Uwe, Weber Manfred, Weber Markus 46/3

ON THE FORMATION OF MESO-SCALE STRUCTURES IN DISPERSED GAS-LIQUID FLOWS
Strom Henrik, Jareteg Klas, Sasic Srdjan, Demaiziere Christophe 46/4

INVESTIGATION OF SINGLE BUBBLES AND BUBBLE CHAINS IN LIQUID METAL UNDER THE INFLUENCE OF MAGNETIC FIELDS
Strumpf Erik, Shevchenko Natalia, Roshchupkina Olga, Eckert Sven 46/5

Boiling, Condensation, Evaporation
Room: Affari-3.D

Tu 11:30-12:50

BOILING FRONT FORMATION IN FLASH DEPRESSURIZATION
Djeridi Henda, Matas Jean-Philippe, Boivin Pierre, Cartellier Alain 46/6

COMPARATIVE ASSESSMENT OF FLOW BOILING HEAT TRANSFER CORRELATIONS FOR HORIZONTAL MINI AND MICRO CHANNELS WITH CO2 AS WORKING FLUID
Gamit Jigar, Channiwala Salim 47/1

A LOOK ON RELEVANT NONDIMENSIONAL NUMBERS FOR FLOW BOILING IN MICROCHANNELS AND MINICHANNELS
Awad Mohamed, Wu Zan, Sunden Bengt 47/2

A MECHANISTIC IR CALIBRATION TECHNIQUE FOR BOILING HEAT TRANSFER INVESTIGATIONS
Richenderfer Andrea, Bucci Matteo, Mckrell Thomas, Buongiorno Jacopo 47/3

Tu 11:30-12:50

Cavitation/Nucleation
Room: Affari-3.C

EXPERIMENTAL AND NUMERICAL INVESTIGATION OF FLASH ATOMIZATION WITH CRYOGENIC FLUIDS
Luo Meng, Ramcke Thomas, Pfitzner Michael, Haidn Oskar 47/4

STRUCTURE OF A LASER-INDUCED UNDERWATER SHOCK WAVE
Tagawa Yoshiyuki, Yamamoto Shota, Hayasaka Keisuke, Kameda Masaharu 47/5

CAVITY SHEDDING MECHANISMS ON A NACA0015 HYDROFOIL
Ganesh Harish, Wu Juliana, Ceccio Steven L. 47/6

JATROPHA BIODIESEL PRODUCTION USING HYDRODYNAMIC CAVITATION
Dwivedi Prateek, Ghosh Sumana 48/1

Tu 11:30-12:50

Collision, Agglomeration and Breakup
Room: Affari-3.A

Tu 11:30-12:50

Tu 11:30-12:50

Tu 11:30-12:50

Tu 11:30-12:50

Tu 11:30-12:50

Tu 11:30-12:50

Tu 11:30-12:50

Computational Techniques for Multiphase Flows
Room: Affari-1.A

DIRECT NUMERICAL SIMULATIONS OF FLOW AROUND A NON-SPHERICAL PARTICLE
Pacha Sanjeevi Sathish, Padding Johan, Kuipers J. A. M (hans) 48/6

A COUPLED EULERIAN-LAGRANGIAN METHOD FOR SPRAY ATOMIZATION ON UNSTRUCTURED MESHES
Evrad Fabien, Denner Fabian, Serfaty Ricardo, Van Wachem Berend 49/1

MULTI-MATERIAL MODELLING FOR COMPRESSIBLE FLOWS WITH THE CONSERVATIVE SHARP-INTERFACE METHOD
Pan Shucheng, Hu Xiangyu, Adams Nikolaus 49/2

SIMULATION OF MICROSCOPIC FLUID PARTICLE MOTION ON SOLID SURFACE USING A DIFFUSE-INTERFACE MODEL
Takada Naoki, Matsumoto Junichi, Matsumoto Sohei, Kurihara Kazuma, Hokari Ryohei 49/3

Experimental Methods for Multiphase Flows
Room: Affari-4.A

EXPERIMENTAL STUDY ON SLUG FLOW DEVELOPMENT IN A RISER AND VERTICAL TO HORIZONTAL 90° BEND
Omar Rajab, Azzopardi Barry James, Hewakandamby Buddhikha 49/4
PRESSURE DROP THROUGH ORIFICES FOR SINGLE- AND TWO-PHASE VERTICALLY UPWARDS FLOW
Azzi Abdelwahid, Zeghloul Ammar, Saidi Faize, Messilem Abdelkader, Azzopardi Barry James

SOLID VOLUME FRACTION FLUCTUATIONS IN A LIQUID FLUIDIZED BED
Marafatallah Maedeh, Sanders Sean

CHARACTERIZATION OF DYNAMICS OF THREE-PHASE SLURRY FLOW USING GAS VOLUME FRACTION FLUCTUATION MEASUREMENTS
Tyagi Parul, Buwa Vivek

Tu 11:30-12:50 Industrial Applications
Room: Affari-0.A

NUMERICAL OPTIMIZATION OF THE PHASE INTERFACE IN HIGH-PRESSURE DIE CASTING
Fring Markus, Behr Marek, Elgeti Stefan

AGGLOMERATION AND TRANSPORT OF A POPULATION OF OXIDE INCLUSIONS IN AN ALUMINUM INDUCTION FURNACE
Bellot Jean-Pierre, Bansal Akshay, Chapelle Pierre, Waz Emmanuel, Delannoy Yves, Lebrun Pierre

SPREADING BEHAVIORS ON MOLten-METAL FALLING-FLOW AT ELEVATED TEMPERATURES IN NUCLEAR SEVERE ACCIDENT CONDITIONS
Mori Michitsugu, Matsumoto Tatsuki, Ogura Takahito, Miwa Shuichiro

Tu 11:30-12:50 Interfacial Flows
Room: Affari-2.A

STUDY ON THE REMOVAL OF LOW LOADING HEAVIER PHASE BY LIGHTER PHASE IN MULTIPHASE PIPE
Koguna Aminu, Lao Liyun, Yeung Hoi

HYDRODYNAMICS OF GAS ENTRAPMENT IN MICROPORITIES
Gomez-Lopez Miguel, Torabi Soroosh, Trinkle Christine, Grana-Otero Jose

DYNAMICS OF VERTICAL FALLING FILMS – AN EXPERIMENTAL AND NUMERICAL STUDY
Akesjo Anders, Gourdon Mathias, Vamling Lennart, Sasic Srdjan

Tu 11:30-12:50 Modelling of Multiphase Flows
Room: Affari-2.D

EULER-EULER MODELING OF HYDRODYNAMICS AND MASS-TRANSFER IN BUBBLY FLOWS
Rzehak Roland, Kriebitzsch Sebastian, Krepper Eckhard

A SUBGRID MODEL FOR LARGE-EDDY SIMULATION OF TURBULENT BUBBLY FLOWS
Chabi Ina Fathia, Bellakhal Ghazi, Chahed Jamal

DNS ASSISTED MODELING OF BUBBLY FLOWS
Tryggvason Gretar, Lu Ji, Daoming, Ma Ming

Tu 11:30-12:50 Multiphase Flow in Heat and Mass Transfer
Room: Affari-2.C

TWO-PHASE HEAT TRANSFER AND FLUID FLOW WITHIN PLATE HEAT EXCHANGERS
Kim Jungho, Solodych Valentin, Donghyeon Lee, Amalfi Raffaele, Thome John

HEAT TRANSFER IN FLAT PLATE PULSATING HEAT PIPES WITH ASYMMETRIC CHANNEL DESIGNS
Jang Dong Soo, Lee Jo Seong, Kim Dongwoo, Min Hanbeok, Kim Yongchan

EXPERIMENTAL INVESTIGATION OF TAYLOR BUBBLE DISSOLUTION IN MILLI-CHANNELS
Haghnejadahar Mohammadreza, Boden Stephan, Hampel Uwe

Tu 11:30-12:50 Particle and Drop Dynamics
Room: Villa-Sala Verde

AEROSOL SCAVENGING BY DROPS: DERIVING A COLLECTION KERNEL FROM MICROPHYSICAL MODELLING
Cherrier Gael, Belut Emmanuel, Taniere Anne, Rimbert Nicolas, Gerardin Fabien

ANISOTROPIC TURBULENT CLUSTERING OF CLOUD DROPLETS SETTLING IN ISOTROPIC TURBULENCE
Matsuda Keigo, Onishi Ryo, Takahashi Keiko
<table>
<thead>
<tr>
<th>Session</th>
<th>Time</th>
<th>Room</th>
<th>Title</th>
<th>Authors</th>
</tr>
</thead>
<tbody>
<tr>
<td>Particle-Laden Flows</td>
<td>Tu 11:30-12:50</td>
<td>Villa-Auditorium</td>
<td>LAGRANGIAN SUBGRID MODELLING FOR THE RELATIVE SEPARATION OF TRACERS IN TURBULENCE</td>
<td>Toschi Federico, Lanotte Alessandra Sabina, Mazzitelli Irene M.</td>
</tr>
<tr>
<td></td>
<td></td>
<td></td>
<td>FEEDBACK EFFECT IN GAS-SOLID DISPERSE FLOW THROUGH A VERTICAL CHANNEL</td>
<td>Mito Yoichi</td>
</tr>
<tr>
<td></td>
<td></td>
<td></td>
<td>MACROSCOPIC CHARACTERISTICS OF HEAVY PARTICLE RESUSPENSION OBTAINED FROM DIRECT NUMERICAL SIMULATIONS OF PRESSURE DRIVEN CHANNEL FLOW</td>
<td>Yiantsios Stergios, Saliakellis Panagiotis</td>
</tr>
<tr>
<td></td>
<td></td>
<td></td>
<td>INVESTIGATION OF IMPINGING DROPLET SPREADINGS ON HOT/COLD SURFACES WITH TR-PIV</td>
<td>Erkan Nejdet, Okamoto Koji</td>
</tr>
<tr>
<td></td>
<td></td>
<td></td>
<td>THE EFFECT OF SHEAR STRESS FORMULATION AND OF THE SHAPE FACTOR ON SLUG CAPTURING</td>
<td>Bonzanini Arianna, Ferrari Marco, Poesio Pietro</td>
</tr>
<tr>
<td></td>
<td></td>
<td></td>
<td>THE EFFECT OF PHYSICAL PROPERTIES ON CHURN FLOW IN A LARGER DIAMETER RISER</td>
<td>Damir Garave, Escrig Escrig Josep, Azzopardi Barry James, Hewakandamby Buddhikha</td>
</tr>
<tr>
<td></td>
<td></td>
<td></td>
<td>TURBULENCE CONTRIBUTION IN MOMENTUM INTERFACIAL TRANSFER IN AN UPWARD LIQUID-LIQUID PIPE FLOW</td>
<td>Rezig Mariem, Bellakhal Ghazi, Chahed Jamel</td>
</tr>
<tr>
<td>Bubble Dynamics</td>
<td>Tu 11:30-12:50</td>
<td>Affari-3.B</td>
<td>NUMERICAL SIMULATIONS OF A RISING DROP OR BUBBLE WITH SHAPE OSCILLATIONS IN THE PRESENCE OF SURFACTANTS</td>
<td>Piedfert Antoine, Risso Frederic, Masbernat Olivier, Lalanne Benjamin</td>
</tr>
<tr>
<td></td>
<td></td>
<td></td>
<td>LIFT FORCE ACTING ON SINGLE BUBBLES IN LINEAR SHEAR FLOWS</td>
<td>Aoyama Shohei, Zun Iztok, Hayashi Kosuke, Hosokawa Shigeo, Tomiyama Akio</td>
</tr>
<tr>
<td></td>
<td></td>
<td></td>
<td>THE INTERACTION OF A HIGH-RE MODERATE-WE BUBBLE WITH AN INCLINED WALL</td>
<td>Barbosa Christophe, Legendre Dominique, Zenit Roberto</td>
</tr>
<tr>
<td>PARALLEL SESSIONS VII</td>
<td></td>
<td></td>
<td></td>
<td></td>
</tr>
<tr>
<td>Boiling, Condensation, Evaporation</td>
<td>Tu 14:40-16:00</td>
<td>Affari-2.B</td>
<td>MEASUREMENT OF BUBBLE DISPERSION IN A TURBULENT BOUNDARY LAYER</td>
<td>Gabillet Celine, Bon Alexandre, Billard Jean Yves, Cazin Sebastien, Colin Catherine</td>
</tr>
<tr>
<td></td>
<td></td>
<td></td>
<td>EFFECT OF DEFORMABILITY ON BUBBLE MIGRATION IN TURBULENT GAS-LIQUID FLOWS</td>
<td>Vichansky Alexander</td>
</tr>
<tr>
<td></td>
<td></td>
<td></td>
<td>FLOW PAST A BUBBLE-GENERATING HYDROFOIL FOR SHIP DRAG REDUCTION</td>
<td>Kumagai Ichiro, Tsukahara Takahiro, Murai Yuichi</td>
</tr>
<tr>
<td></td>
<td></td>
<td></td>
<td>INVESTIGATION OF THE BUBBLE SIZE DISTRIBUTION AROUND AN UNDERWATER VEHICLE</td>
<td>Vahaji Sara, Chen Li, Cheung Sherman, Tu Jiyuan</td>
</tr>
<tr>
<td></td>
<td></td>
<td></td>
<td>VARIATIONS IN GEOMETRIC VARIABLES OF WATER DROPLET ON LOW-SURFACE-ENERGY SOLIDS</td>
<td>Yonemoto Yukihiro, Kunugi Tomoaki</td>
</tr>
</tbody>
</table>
STRATIFICATION OF INTERMITTENT AND ANNULAR R245-FA FLOW BOILING IN HORIZONTAL ORIENTATION

Layssac Thibaut, Revellin Remi, Lips Stephane

56/3

PREDICTION OF SOLID-LIQUID EQUILIBRIUM IN PARAFFINIC SYSTEMS WITH AN IMPROVED ACTIVITY COEFFICIENT METHOD FOR THE NON-IDEALITY OF SOLID PHASE

Yang Juheung, Wang Wei, Song Pan, Ma Qianli, Quan Qing, Gong Jing

56/4

A MULTISCALE APPROACH TOWARDS IN-TUBE CONVECTIVE BOILING FLOW SIMULATION

Yazdani Miad, Alahyari Abbas, Radcliff Thomas, Soteriou Marios

56/5

Tu 14:40-16:00 Cavitation/Nucleation

Room: Affari-3.C

THE EFFECTS OF NOSE SHAPES AND VISCOITY ON THE CAVITY EVOLUTION AROUND THE UNDERWATER PROJECTILE

Okuno Kohei, Matsuo Akiko

56/6

LIQUID-FOUNTAIN INSTABILITY AND CAVITATION INSIDE BEADS-FOUNTAIN ASSOCIATED WITH ULTRASONIC ATOMIZATION

Myosen Kenta, Ito Yui, Mori Yasuhige, Tsuchiya Katsumi

57/1

INFLUENCE OF SORPTION EFFECTS ON THE EFFICIENCY OF MULTIPHASE CONVEYANCE

Wagner Stefan, Luke Andrea

57/2

EXPERIMENTAL STUDY OF A CAVITATING BACKWARD FACING STEP FLOW: VAPOR PHASE INVESTIGATION USING X-RAY ATTENUATION MEASUREMENTS AND WALL PRESSURE MEASUREMENTS

Maurice Guillaume, Djeridi Henda, Barre Stephane

57/3

Tu 14:40-16:00 Collision, Agglomeration and Breakup

Room: Affari-3.A

DROPLET-PARTICLE COLLISIONS IN HOMOGENEOUS AND ISOTROPIC TURBULENCE

Charalampous Georgios, Hardalupas Yannis

57/4

PARTICLE AND AGGLOMERATE DISPERSION IN GAS FLOW BY COLLISIONS AND FLOW INDUCED STRESSES

Wollborn Tobias, Knoop Claas, Fritsching Udo

57/5

THE PARTICLE COLLISION FREQUENCY IN PARTICLE-LADEN TURBULENT CHANNEL FLOW

Kuerten Hans, Vrsman Bert

57/6

THE INFLUENCE OF STOKES NUMBER ON PARTICLE CLUSTERING AT THE EXIT OF A TURBULENT JET ISSUING FROM A LONG PIPE

Lau Timothy, Nathan Graham

58/1

Tu 14:40-16:00 Computational Techniques for Multiphase Flows

Room: Affari-1.A

THE SPARSE CARDINAL SINE DECOMPOSITION APPLIED TO STOKES INTEGRAL EQUATIONS

Alouges Francois, Aussal Matthieu, Lefebvre Lepot Aline, Pigeonneau Franck, Sellier Antoine

58/2

AN ACCURATE METHOD TO TAKE INTO ACCOUNT LUBRICATION FORCES IN NUMERICAL SIMULATION OF SUSPENSIONS.

Lefebvre Lepot Aline, Chivot Guillaume, Girardin Mathieu, Gouarin Loic, Merlet Benoit

58/3

NON-CONSERVATIVE PRESSURE-BASED COMPRESSIBLE FORMULATION FOR MULTIPHASE FLOWS WITH HEAT AND MASS TRANSFER

Labois Mathieu, Narayanan Chidambaram

58/4

NUMERICAL SIMULATIONS OF VISCOUS AND COMPRESSIBLE TWO-PHASE FLOWS USING HIGH-ORDER SCHEMES

Capuano Marion, Bogey Christophe, Spelt Peter

58/5

Tu 14:40-16:00 Experimental Methods for Multiphase Flows

Room: Affari-4.A

COMPARISON OF THREE INJECTOR GEOMETRIES IN GAS LIFT PERFORMANCE IN A LARGE DIAMETER PIPE

Ibrahim Abubakr, Hewakandamby Buddhikha, Azzopardi Barry James

58/6

COMPARISON AMONG EXPERIMENTAL TECHNIQUES FOR PARTICLE SIZE MEASUREMENTS IN FLOWS

Romano Giampaolo, Pereira Francisco Alves, Van Rijssbergen Martijn, Di Felice Fabio

59/1

MULTI-CAMERA X-RAY TOMOGRAPHY OF MULTIPHASE FLOWS

Stjælaaussen Olaf, Denis Heidi, Langsholt Morten

59/2

THIN FILM THICKNESS MEASUREMENTS USING ULTRASONIC PULSE ECHO TECHNIQUE

Alaufi Yousuf, Watson Nicholas, Azzopardi Barry James, Hewakandamby Buddhikha, Dimitrakis Georgios, Hasan Abbas, Kalashnikov Alexander

59/3
**Tu 14:40-16:00 Interfacial Flows**  
*Room: Affari-2.A*

**NUMERICAL AND EXPERIMENTAL INVESTIGATION OF LIQUID FILM FLOWS ON PACKINGS IN ABSORBERS FOR POST-COMBUSTION CO2 CAPTURE**  
Ito Yoshiyuki, Ikeda Ryosuke, Huang Jian, Saga Mariko, Nakamura Shiko, Takano Kenji  
59/4

**A COMBINED EXPERIMENTAL/COMPUTATIONAL STUDY OF THE HYDRODYNAMIC CHARACTERISTICS OF HARMONICALLY EXCITED THIN-FILM FLOWS**  
Charagiannis Alexandros, Denner Fabian, Van Wachem Berend, Kalliadasis Serafi, Markides Christos N.  
59/5

**REDUCED ORDER MODELLING OF LIQUID-GAS CHANNEL FLOWS**  
Lavalie Gianluca, Lucquiaud Mathieu, Vallun Prashant  
59/6

**GROWTH OF A LOCALIZED PERTURBATION IN A COUNTERCURRENT TWO-PHASE FLOW**  
Coppola Gennaro, Capuano Francesco, De Luca Luigi  
60/1

---

**Tu 14:40-16:00 Modelling of Multiphase Flows**  
*Room: Affari-2.D*

**A HIERARCHY OF COMPRESSIBLE TWO-FLUID MODELS WITH IDENTIFIED PHYSICAL RELAXATIONS AND RELATED NUMERICAL METHODS FOR THE SIMULATION OF SEPARATED AND DISPERSED PHASES**  
Drui Florence, Massot Marc, Kohk Samuel, Larat Adam  
60/2

**MASSIVE PARALLEL DIRECT NUMERICAL SIMULATIONS OF THREE-DIMENSIONAL JET FLOWS**  
Kahouadji Lyes, Matar Omar K., Cherqui Jalel, Juric Damir, Shin Seungwon  
60/3

**MULTISCALE SIMULATION OF LIQUID JET DISINTEGRATION AND PRIMARY ATOMIZATION USING EULERIAN-LAGRANGIAN COUPLING**  
Saeedipour Mahdi, Torin Stefan, Schneiderbauer Simon  
60/4

**DIRECT NUMERICAL SIMULATION OF PRIMARY BREAK-UP IN SWIRLING LIQUID JETS**  
Galbiati Claudio, Tonini Simon, Cossali Gianpietro Elvis, Weigand Bernhard  
60/5

---

**Tu 14:40-16:00 Multiphase Flow in Heat and Mass Transfer**  
*Room: Affari-2.C*

**LOCAL MASS TRANSFER PHENOMENA AT SINGLE TAYLOR BUBBLES IN A VERTICAL MINI CHANNEL**  
Kastens Sven, Aoki Jiro, Pesce Chiara, Marschall Holger, Hayashi Kosuke, Schlueter Michael, Bothe Dieter, Tomiyama Aki  
60/6

**STEADY STATE ANALYSIS OF FLASHING DRIVEN NATURAL CIRCULATION LOOP: INFLUENCE OF HEATER AND COOLER ORIENTATION**  
Dewangan Kush Kumar, Das Prasanta Kumar  
61/1

**HYDRODYNAMICS AND MASS TRANSFER PROPERTIES OF A ROTATING BUBBLE COLUMN**  
Lau Yuk Man, Herrmann Heber Robert, Dashiborun Amir Motamed, Schubert Markus  
61/2

**MASS TRANSFER OF SOLUTE AND SOLVENT ACROSS A DEFORMING PERMEABLE MEMBRANE**  
Takeuchi Shintaro, Miyauchi Suguru, Kaihishima Takeo  
61/3

---

**Tu 14:40-16:00 Particle and Drop Dynamics**  
*Room: Villa-Sala Verde*

**THE SOLIDIFICATION OF WATER DROPLETS IMPINGING TO A COOLING HORIZONTAL PLATE**  
Kimura Ryota, Hagiwara Yoshimichi  
61/4

**SLIDING DROPS – MOLECULAR VERSUS CONTINUUM SIMULATIONS**  
Derksen Jos  
61/5

**EFFECTS OF NANOPARTICLES ON THE DEPINNING FORCE OF A RECEDING DROPLET ON MICROPATTERNED SUPERHYDROPHOBIC SURFACES**  
Jiang Youhua, Xu Wei, Connington Kevin, Choi Chang-Hwan  
61/6

**DEPINNING OF WATER DROPLETS ON PILLARED SUPERHYDROPHOBIC SURFACES UNDER DYNAMIC ICING CONDITIONS**  
Sarshar Mohammad, Choi Chang-Hwan  
62/1

---

**Tu 14:40-16:00 Particle-Laden Flows**  
*Room: Villa-Auditorium*

**DIRECT NUMERICAL SIMULATION OF TURBULENT CHANNEL FLOW OVER AN EvOLVING SEDIMENT BED**  
Kidanemarian Aman, Uhlinmann Markus  
62/2

**CHANNEL FLOW OF PARTICLE SUSPENSIONS: MASS AND VOLUME FRACTION EFFECTS**  
Brandt Luca, Lasgari Imran, Picano Francesco, Fornari Walter  
62/3

**EXPERIMENTS ON INCipient BED LOAD UNDER A TURBULENT STREAM**  
Mendes Penteado Marcos Roberto, De Moraes Franklin Erick  
62/4
Tu 14:40-16:00  
**Multiphase Pipe Flows**  
*Room: Affari-3.E*  
A MECHANISTIC MODEL FOR GAS-LIQUID PIPE FLOW WITH SURFACTANTS  
Van Nimwegen Dries, Van Boven Gijs, Portela Luis, Henkes Ruud  
62/5  
NUMERICAL 3-D SIMULATION OF DEVELOPING STRATIFIED GAS-LIQUID FLOW IN CURVED PIPES  
Haj Ali, Mansoori Zohreh, Saffar Avval Majid, Ahmadi Goodarz  
62/6  
EXPERIMENTAL INVESTIGATION OF GAS-LIQUID FLOW IN A HILLY TERRAIN PIPELINE OF LARGE DIAMETER  
Zabkova Katerina, Luke Andrea  
63/1  

Tu 14:40-16:00  
**Bubble Dynamics**  
*Room: Affari-3.B*  
MODELLING OF BUBBLE DYNAMICS AND EULER/LAGRANGE PREDICTIONS OF BUBBLE COLUMNS  
Muniz Marcelo, Sommerfeld Martin  
63/2  
THEORETICAL MODEL OF THE BUBBLING REGIME IN A PLANAR CO-FLOW CONFIGURATION  
Bolanos Jimenez Rocio, Sevilla Alejandro, Bazan Carlos Martinez  
63/3  
NUMERICAL STUDY ON THE WETTABILTY DEPENDENT INTERACTION OF DROPLETS AND BUBBLES WITH SOLID STRUCTURES  
Woerner Martin, Cai Xuan, Marschall Holger, Deutschmann Olaf  
63/4  
BUBBLE ENTRAINMENT BEHAVIOUR DURING DROP IMPINGEMENT ONTO LIQUID SURFACE  
Kawashima Hisanobu, Fujishima Shun, Ogawa Soichiro, Ishima Tsuneaki  
63/5  

Tu 14:40-16:00  
**Memorial session dedicated to Prof. Gad Hetsroni**  
*Room: Affari-0.A*  
WAKE FEATURES OF BIRDS DURING FLAPPING FLIGHT, CASE STUDY: STARLING, SANDPIPER AND ROBIN  
Gurka Roi  
63/6  
BREAKUP OF FINITE SIZE COLLOIDAL AGGREGATES IN TURBULENT FLOW INVESTIGATED BY 3D PARTICLE TRACKING VELOCIMETRY  
Babler Matthaus, Saha Debashish, Holzner Markus, Soos Miroslav, Luthi Beat, Liberzon Alex  
64/1  

**PARALLEL SESSIONS VIII**  

Tu 16:20-17:40  
**Bubbly Flows**  
*Room: Affari-2.B*  
LINEAR VISCOELASTIC ANALYSIS OF BUBBLE SUSPENSIONS BY MEANS OF ULTRASONIC SPINNING RHEOMETRY  
Tasaka Yuji, Rapberger Richard, Dymowski Konrad, Murai Yuichi  
64/2  
THREE-DIMENSIONAL SCENARIO OF DRAG REDUCTION IN BUBBLY TAYLOR-COUETTE FLOWS  
Murai Yuichi, Ozaki Tsubasa, Tasaka Yuji  
64/3  
INVESTIGATION OF GAS PERFUSION THROUGH POROUS MEDIA WITH AND WITHOUT SUPER-HYDROPHOBIC COATINGS  
Gose James W., Golovin Kevin, Tuteja Anish, Ceccio Steven L., Perlin Marc  
64/4  

Tu 16:20-17:40  
**Boiling, Condensation, Evaporation**  
*Room: Affari-3.D*  
EFFECT OF SUBCOOLING ON FILM BOILING HEAT TRANSFER CHARACTERISTICS OF JET ON A SUPERHEATED PLATE  
Hasegawa Koji, Ohtake Hironobu, Ueno Masahiro  
64/6  
EXPERIMENTAL STUDY OF WATER POOL BOILING ON A SINGLE NUCLEATION SITE, FROM ATMOSPHERIC PRESSURE DOWN TO LOW PRESSURE  
Michaie Sandra, Rulliere Romuald, Bonjour Jocelyn  
65/1  
EFFECT OF WETTABILTY ON POOL BOILING ONSET  
Marango Marco, Bourdon Benoit, Ville Fabio, Malavasi Ileana, Riboo Romain, Bertrand Emilie  
65/2  
EFFECT OF OSCILLATORY FLOW FIELD ON BOILING ON A SINGLE NUCLEATION SITE, FROM ATMOSPHERIC PRESSURE  
Kumar Manoj, Das Arup Kumar, Das Prasanta Kumar  
65/3
<table>
<thead>
<tr>
<th>Session</th>
<th>Title</th>
<th>Authors</th>
<th>Page</th>
</tr>
</thead>
<tbody>
<tr>
<td>Tu 16:20-17:40 Collision, Agglomeration and Breakup</td>
<td>EXPERIMENTAL VALIDATION OF ANALYTICAL MODELS FOR LIQUID FILM BREAK-UP</td>
<td>Line Hakon, Steiner Hellfried, Brenn Guenter, Engesser Philipp</td>
<td>65/4</td>
</tr>
<tr>
<td></td>
<td>SPRAY IMPINGEMENT ON SPHERICAL TARGETS IN HOMOGENEOUS, ISOTROPIC TURBULENCE</td>
<td>Andrade Paul, Charalampous Georgios, Hardalupas Yannis</td>
<td>65/5</td>
</tr>
<tr>
<td></td>
<td>A STOCHASTIC APPROACH TO COLLOIDAL PARTICLE COLLISION/AGGLOMERATION</td>
<td>Maftei Radu Eduard, Bossy Mireille, Minier Jean-Pierre, Profeta Christophe</td>
<td>65/6</td>
</tr>
<tr>
<td></td>
<td>PARTICLE-WALL ADHESION MODEL FOR TURBULENT DISPERSE MULTIPHASE FLOWS WITHIN AN EULER-LAGRANGE LES APPROACH</td>
<td>Breuer Michael, Almohammed Naser</td>
<td>66/1</td>
</tr>
<tr>
<td></td>
<td>A NEW MIXTURE MODEL FOR COMPRESSIBLE MULTIPHASE FLOWS</td>
<td>Henry De Frahan Marc, Alahyari Beig Shahab, Aboulhasanzadeh Bahman, Ganesh Harish, Ceccio</td>
<td>66/3</td>
</tr>
<tr>
<td></td>
<td>NUMERICAL SIMULATIONS OF SUBCOOLED BOILING USING A TWO-FLUID/DQMOM METHODOLOGY</td>
<td>Jareteg Klas, Sasic Srdjan, Vinai Paolo, Demaziere Christophe</td>
<td>66/4</td>
</tr>
<tr>
<td></td>
<td>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</td>
<td>Essadki Mohamed, De Chaisemartin Stephane, Massot Marc, Laurent Frederique, Larat Adam, Jay Stephane</td>
<td>66/5</td>
</tr>
<tr>
<td>Tu 16:20-17:40 Experimental Methods for Multiphase Flows</td>
<td>LDV ANALYSIS OF GAS-LIQUID SLUG FLOW IN A HORIZONTAL PIPE</td>
<td>Thaker Jignesh, Banerjee Jyotirmay</td>
<td>66/6</td>
</tr>
<tr>
<td></td>
<td>EXPERIMENTAL ANALYSIS OF DOWNWARD GAS-LIQUID SLUG FLOW IN SLIGHTLY INCLINED PIPES</td>
<td>Rodrigues Romulo, Schneider Fabio, Da Silva Marco, Marcelino Neto Moises, Morales Rigoberto</td>
<td>67/1</td>
</tr>
<tr>
<td></td>
<td>DETERMINATION OF HYDRODYNAMIC PARAMETERS OF AIR-WATER SLUG FLOW USING ULTRASONIC TECHNIQUES VIA HILBERT–HUANG TRANSFORMS</td>
<td>Abbagoni Baba Musa, Yeung Hoi</td>
<td>67/2</td>
</tr>
<tr>
<td></td>
<td>EXPERIMENTAL RESEARCH OF CONDUCTANCE AND CAPACITANCE COMBINATION SENSOR FOR WATER HOLDUP MEASUREMENT IN OIL-WATER TWO-PHASE FLOW</td>
<td>Wu Hao, Tan Chao, Dong Fieg</td>
<td>67/3</td>
</tr>
<tr>
<td>Tu 16:20-17:40 Granular Media &amp; Granular Flows</td>
<td>COUPLING DIFFERENT LEVELS OF RESOLUTION IN DEM SIMULATIONS</td>
<td>Quetschagner Daniel, Lichtenegger Thomas, Pirker Stefan</td>
<td>67/4</td>
</tr>
<tr>
<td></td>
<td>CHARACTERIZATION OF NEARLY JAMMED GRANULAR FLOWS IN HIGH SHEAR SYSTEMS THROUGH MOLECULAR DYNAMICS SIMULATIONS</td>
<td>Khaliltehrani Mohammad, Sasic Srdjan, Rasmuson Anders</td>
<td>67/5</td>
</tr>
<tr>
<td></td>
<td>ON THE MOTION OF A SINGLE AIR SPOUT THROUGH A QUASI-TWO DIMENSIONAL GRANULAR BED</td>
<td>Maiti Ritwik, Das Gargi, Das Prasanta Kumar</td>
<td>67/6</td>
</tr>
<tr>
<td>Tu 16:20-17:40 Industrial Applications</td>
<td>EFFECT OF MOLTEN STEEL FLOW IN MOLD ACCORDING TO INTENSITY AND POSITION OF ELECTRO-MAGNETIC FIELD</td>
<td>Han Sang-Woo, Cho Hyun-Jin, Jin Soonyong, Seden Martin, Choi Sungmin</td>
<td>68/1</td>
</tr>
<tr>
<td></td>
<td>NUMERICAL ANALYSIS OF EROSION ON SUSPENDED HEAT TRANSFER SURFACES IN A 350MW SUPERCritical CFB BOILER</td>
<td>Tu Qiuva, Wang Haigang, Sun Yunkai</td>
<td>68/2</td>
</tr>
<tr>
<td></td>
<td>NUMERICAL INVESTIGATION OF ELECTROSTATIC EFFECTS ON PNEUMATIC TRANSPORT OF GRANULAR MATERIALS</td>
<td>Yao Jun, Zhao Yanlin, Lim Eldin Wee Chuan</td>
<td>68/3</td>
</tr>
</tbody>
</table>
Tu 16:20-17:40 **Interfacial Flows**
*Room: Affari-2.A*

**INTERFACE-RESOLVING SIMULATIONS OF GAS-LIQUID FLOWS IN A SOLID SPONGE STRUCTURE**
Cai Xuan, Woerner Martin, Deutschmann Olaf 68/4

**INFLUENCE OF ANGULAR MOMENTUM ON THE FORMATION OF AIR-CORE INTAKE VORTICES IN CYLINDRICAL WATER TANKS**
Szilágyi Nicolai, Bezecny Daniel, Richter Steffen, Hoffmann Marko, Schlueter Michael 68/5

**INFLUENCE OF PARTIALLY WETTED MICRO-PARTICLES AT A CONTAMINATED INTERFACE ON COALESCEENCE PROCESS.**
De Malmazet Erik, Risso Frederic, Masbemart Olivier, Pauchard Vincent 68/6

Tu 16:20-17:40 **Modelling of Multiphase Flows**
*Room: Affari-2.D*

**NUMERICAL STUDY OF DROPLET STATISTICS RESULTING FROM TURBULENT JET PRIMARY BREAKUP**
Movaghar Amirreza, Linne Mark, Oevermann Michael, Bravo Luis, Kerstein Alan 69/1

**PARTICLE DISPERSION IN INHOMOGENEOUS TURBULENCE: AN ANALYSIS OF PDF-BASED MODELS**
Stafford Christopher, Swailes David, Chakraborty Nilanjan 69/2

**3D RSTM MODELING FOR PARTICULATE GRID-GENERATED TURBULENT AND SHEAR FLOWS**
Kartushinsky Alexander, Hussainov Medhat, Shcheglov Igor, Tisler Sergei, Kosinski Pawel, Balakin Boris V., Laut Peep, Segeel Karl Erik 69/3

**LAGRANGIAN STOCHASTIC MODELLING IN LARGE-EDDY SIMULATION OF TURBULENT PARTICLE-LADEN FLOWS**
Innocenti Alessio, Marchioli Cristian, Chibbaro Sergio 69/4

Tu 16:20-17:40 **Multiphase Flow in Heat and Mass Transfer**
*Room: Affari-2.C*

**COLLECTING DEVICE OF VOLATILE ORGANIC COMPOUNDS IN WATER LIKE A NOSE: THIN LIQUID FILM CONTAINING ODORANT BINDING PROTEINS**
Kida Hitoshi, Tagawa Yoshiyuki, Sato Ryoichi, Kameda Masaharu 69/5

**MULTIPHASE INVESTIGATION OF INTENSIVE QUENCHING WITH WATERJETS**
Waldeck Steffen, Fritsching Udo 69/6

**A SIX-EQUATION TWO-PHASE MODEL IN WENO SCHEME FOR PHASE TRANSFORMATION FRONTS**
Wu Wangxia, Wang Bing, Zhang Wenbin 70/1

**EFFECT OF THE VAPOR FLOW ON THE DROP SPREADING AT THE IMPACT IN THE LEIDENFROST BOILING**
Casanei Guillaume, Caballina Ophelie, Lemoine Fabrice 70/2

Tu 16:20-17:40 **Particle and Drop Dynamics**
*Room: Villa-Sala Verde*

**DROP IN DROP DIAMETER FROM A VIBRATING CAPILLARY TUBE**
Duval Herve, Bertrandias Aude, Giorgi Marie-Laurence 70/3

**KINEMATIC AND DYNAMIC CHARACTERISTICS OF A FALLING DROPLET**
Bergeles Konstantinos, Hardalupas Yannis, Taylor Alex M.K. P. 70/4

**WALL EFFECT ON THE DYNAMICS OF PARTICLE MOTION IN A LAMINAR FLOW**
Feng Zhi-Gang, Gatewood Jason, Duan Yifei, Michaelides Efstathios 70/5

**STEERING ULTRASOUND CONTRAST AGENTS WITH ULTRASOUND USING THE PRIMARY BJERKNES FORCE**
Clark Alicia, Aliseda Alberto 70/6

Tu 16:20-17:40 **Particle-Laden Flows**
*Room: Villa-Auditorium*

**PARTICLE RESUSPENSION BY A PERIODICALLY-FORCED IMPINGING JET**
Soligo Giovanni, Wu Wen, Zonta Francesco, Marchioli Cristian, Soldati Alfredo, Piomelli Ugo 71/1

**SEDIMENT-LADEN FRESH WATER ABOVE SALT WATER: STABILITY ANALYSIS AND NONLINEAR SIMULATIONS**
Meiburg Eckart, Burns Peter 71/2

**MODIFICATION OF WALL TURBULENCE VIA THE EXACT REGULARIZED POINT PARTICLE APPROACH**
Messina Roberta, Battista Francesco, Guaitieri Paolo, Casciola Carlo 71/3

Tu 16:20-17:40 **Memorial session dedicated to Prof. Gad Hetsroni**
*Room: Affari-0.A*

**PARTICLE FLOW IN GAS OR LIQUID - WHAT DO THEY HAVE IN COMMON?**
Kalman Haim 71/4
SCAVENGING OF SOLUBLE ATMOSPHERIC TRACE GASES DUE TO CHEMICAL ABSORPTION BY RAIN DROPLETS IN INHOMOGENEOUS ATMOSPHERE
Elperin Tov, Fominykh Andrew, Krasovitov Boris

AN UNIVERSAL BINARY FORCE RELATION ON A PARTICLE THAT ACCOUNTS FOR THE INFLUENCE OF NEAREST NEIGHBORS
Akiki Georges, Balachandar S, Jackson Thomas L., Mehta Yash, Neal Christopher R., Thakur Siddarth

DNS OF TURBULENT CONVECTIVE FLOW BOILING IN A CHANNEL AT Re?=400
Caviezel Daniel, Swiderski Karol, Narayanan Chidambaran, Lakehal Djamel
PARALLEL SESSIONS IX

We 9:50-11:10 Bubbly Flows
Room: Affari-2.B
EXPERIMENTAL INVESTIGATIONS ON THE LIFT FORCE FOR TURBULENT FLOWS WITH LOW MORTON NUMBER
Ziegenhein Thomas, Lucas Dirk 72/2
STUDY OF BUBBLY FLOW THROUGH A PERFORATED PLATE USING A TWO-FLUID MODEL AND A DISCRETE ELEMENT METHOD
Pena Monferrer Carlos, Martínez Cuenca Raul, Munoz Cobo Gonzalez Jose Luis, Chiva Sergio 72/3
MOTION OF SURFACTANT CONTAMINATED BUBBLES IN HOMOGENEOUS SHEAR TURBULENCE
Tanaka Mitsuru 72/4
TRANSPORT AND HYDRODYNAMIC PHENOMENON IN A VIBRATING BUBBLE COLUMN
Mohagheghian Shahrouz, Elbing Brian. R 72/5

We 9:50-11:10 Boiling, Condensation, Evaporation
Room: Affari-3.D
VISUALIZATION STUDIES ON THE SPRAY COOLING BEHAVIOR OF THE MICROMETRIC MULTIPLE SUCCESSIVE DROPLETS ABOVE THE LEIDENFROST TEMPERATURE IMPACTING ONTO HOT INCLINED SURFACES
Deendarlianto Deendarlianto, Takata Yasuyuki, Kohno Masamichi, Hidaka Sumitomo, Wakui Takaaki, Indarto Indarto 72/6
IMPLEMENTATION AND DEMONSTRATION OF A BOUNDARY CONDITION WALL FUNCTION FOR DIRECT PRECIPITATION FOULING CFD MODELLING
Johnsen Sverre Gullikstad, Johansen Stein Tore, Wittgens Bernd 73/1
3D SIMULATION OF NUCLEATE POOL BOILING USING DIFFUSE INTERFACE MODEL
Tamimoto Takuma, Tsujimoto Koichi, Shakouchi Toshikiko, Ando Toshitake 73/2
MEASUREMENTS AND VISUALIZATION DURING CONDENSATION INSIDE A 3 MM DIAMETER CHANNEL
Azzolin Marco, Bortolin Stefano, Del Col Davide 73/3

We 9:50-11:10 Granular Media & Granular flow
Room: Affari-3.B
EULERIAN SIMULATION OF 3D FREELY-FALLING GRANULAR JET
Audard Francois, Fede Pascal, Belut Emmanuel, Fontaine Jean-Raymond, Simonin Olivier 73/4
THE PROPAGATION AND DEPOSITION OF A FINITE DRY GRANULAR MASS DOWN A ROUGH INCLINE
Lee Keng Lin, Yang Fuling 73/5
COLLAPSE OF A WATER-SATURATED GRANULAR COLUMN IN AIR
Aussillous Pascale, Nobili Clement 73/6
DIRECT AND INDIRECT MEASUREMENTS OF FRICTION COEFFICIENTS OF STEADY DRY GRANULAR FLOWS DOWN A ROUGH NARROW INCLINE OF ROUGH BED
Yang Fuling, Tingyen Chiu 74/1

We 9:50-11:10 Industrial Applications
Room: Affari-1.A
STUDY OF THE PROCESS OF ELECTROMAGNETIC IMPACT ON HIGH-VISCOSITY HYDROCARBON LIQUID IN A PIPELINE
Kireev Victor, Musin Ayrat, Zininatullin Rasul, Miroshnichenko Vladislav 74/2
PERFORMANCE OF HEAVY OIL VISCOSITY REDUCER (HOVR) ON CRUDE OIL EMULSION
Bonetti Darfan, Biazussi Jorge Luiz, Peres Carlos Eduardo, Monte Verde William, Guersoni Bizotto Vanessa Cristina, Bannwart Antonio Carlos 74/3
CFD-PBM MODELLING OF AGGLOMERATION AND DEPOSITION IN PETROLEUM INDUSTRY
Balakin Boris V., Kartushinsky Alexander, Hoffmann Alex Christian, Kosinski Pawel 74/4
A STUDY OF THE PERFORMANCE OF PILOT SCALE GRAVITY SORTERS USING A CFD-DEM COMPUTATIONAL FRAMEWORK
Kannan Ananda Subramani, Lund Jacob, Carstensen Jens Michael, Sasic Srdjan 74/5

We 9:50-11:10 Interfacial Flows
Room: Affari-2.A
A COMPARATIVE STUDY OF TEMPORAL EVOLUTION OF AIR AND STEAM BUBBLES EMERGING OUT FROM AN ORIFICE IN A QUIESCENT WATER MEDIA
Ahmad.S Thariq, Datta Saikat, Singha Sanat Kumar, Datta Santanu Prasad, Das Prasanta Kumar 74/6
PHASE-FIELD SIMULATIONS OF THE BUOYANCY-DRIVEN DETACHMENT OF A WALL-BOUND PENDANT DROP
Lamorgese Andrea, Mauri Roberto
75/1

TRIBO-CHARGING AND FLAME-SYNTHESIS TO CREATE SUPER-HYDROPHOBIC COATINGS WITH STRONG WATER JET RESISTANCE
Bayer lcker, Loth Eric
75/2

VALIDATION OF TURBULENCE PARAMETERS AT THE INTERFACE OF HORIZONTAL MULTIPHASE FLOWS
Hoehne Thomas
75/3

We 9:50-11:10 Mixing
Room: Affari-3.A

CFD SIMULATIONS OF A STIRRED-TANK REACTOR FOR MULTIPHASE SYSTEMS USING OPENFOAM
Hu Xiaofei, Passalacqua Alberto, Fox Rodney, Bertola Francesco
75/4

SCALE-UP OF MIXING PROPERTIES IN BUBBLE COLUMN REACTORS
Plais Cecil, Gemello Luca, Raimundo Pedro Maximiano, Forret Ann, Marchisio Daniele, Augier Frederic
75/5

INVESTIGATION OF LOCAL MIXING AND BUBBLE SIZE DISTRIBUTION IN A STIRRED BUBBLE COLUMN REACTOR
Rosseburg Annika, Schaefer Jan Erik, Wucherpfennig Thomas, Berger Martina, Schlueter Michael
75/6

We 9:50-11:10 Modelling of Multiphase Flows
Room: Affari-2.D

DIRECT NUMERICAL SIMULATION OF FLOW AROUND A ROTATING SPHERE AT HIGH MACH AND LOW REYNOLDS NUMBERS CONDITION FOR SUPERSONIC GAS-PARTICLE FLOW ANALYSIS
Nagata Takayuki, Nonomura Taku, Takahashi Shun, Mizuno Yusuke, Fukuda Kota
76/2

A PSEUDO-DRY WET CONTACT MODEL FOR DISCRETE ELEMENT SIMULATION OF IMMERSED PARTICLE-WALL COLLISIONS
Lin Cheng Chuan, Yang Fuling
76/3

A STOCHASTIC MODEL FOR THE PREDICTION OF PARTICLE/PARTICLE INTERACTIONS IN THE FRAME OF THE EULER-LAGRANGE APPROACH
Arcen Boris, Taniere Anne
76/4

ANALYSIS OF SUBFILTER EFFECTS ON INERTIAL PARTICLES IN FORCED ISOTROPIC TURBULENCE
Rosa Bogdan, Pozorski Jacek
76/5

We 9:50-11:10 Multiphase Flow in Heat and Mass Transfer
Room: Affari-2.C

NUMERICAL SIMULATION OF A SETTLING ICE BALL WITH COUPLED HEAT AND MASS TRANSFER
Chouippe Agathe, Stumpf Michael, Uhlmann Markus, Dusek Jan, Kiselev Alexei, Leisner Thomas
76/6

EXPERIMENTAL ANALYSIS ON THE CHARACTERISTIC AND MECHANISM OF THE SECOND DOMINANT FREQUENCY OF SUPERSONIC STEAM JET
Chen Weixiong, Wang Yingchun, Zhao Quanbin, Chong Daotong, Yan Junjie
77/1

ENHANCED RECOVERY OF URANIUM(VI) AND EUROPIUM(III) USING IONIC LIQUIDS IN SMALL CHANNEL
Li Qi, Tsaoulidis Dimitrios, Angeli Panagiota
77/2

SPATIOTEMPORALLY RESOLVED HEAT TRANSFER MEASUREMENTS IN WAVY LIQUID FILM-FLOWS FALLING OVER AN INCLINED HEATED-SUBSTRATE
Markides Christos N., Baptiste Heiles, Mathie Richard, Charogiannis Alexandros
77/3

We 9:50-11:10 Non-Newtonian Multiphase Flows
Room: Affari-3.C

VISCOELASTIC FLOW MODELLING FOR POLYMER FLOODING
De Shauvik, Peters Frank, Padding Johan, Kuipers J. A. M (hans)
77/4

EXPLORING THE RHEOLOGICAL BEHAVIOUR OF AETHERAL SUSPENSIONS OF ATTRACTIVE MICROPARTICLES
Murphy Eric, Lomboy Gilson, Sundararajan Sriram, Wang Kejin, Subramaniam Shankar
77/5

ATOMIZATION OF NON-NEWTONIAN FLUIDS USING AIRLESS SPRAYERS: EFFECT OF VISCOS DAMPING AND SHEAR THINNING ON LIQUID SHEET INSTABILITY AND ITS BREAKUP
Watanabe Hiroki, Fujimoto Shuhei, Ogata Masayuki, Kumagai Ichiro
77/6

COALESCENCE DURING BUBBLE FORMATION FROM TWO NEIGHBOURING PORES IN A NON-NEWTONIAN LIQUID
Oikonomou George, Passos Agathoklis, Mouza Akaterini, Paras Spiros
78/1

XXVI
We 9:50-11:10  
**Particle and Drop Dynamics**  
*Room: Villa-Sala Verde*  

**Simulation of Gasifying Particle, Interaction with Free Interface Fluid Flow**  
Moradi Nour Zeinab, Do Quang Minh, Amberg Gustav  
78/2  

**A Wall-Function Model for Non-Brownian Hydrosol Particles Deposition**  
Dupuy Magali, Kaysenh Arunvady, Waz Emmanuel, Duval Herve  
78/3  

**Break Up of Capillary Bridges: Effect of Soluble Surfactant**  
Kovalchuk Nina, Nowak Emilia, Simmons Mark  
78/4  

**Deformation and Dynamics of Compound Capsules in Simple Shear Flow**  
Luo Zhengyuan, Bai Bofeng  
78/5  

---  

**We 9:50-11:10**  
**Particle-Laden Flows**  
*Room: Villa-Auditorium*  

**Towards Stochastic Modelling of Inertial Particle Dispersion in Wall-Bounded Turbulent Flows**  
Pozorski Jacek, Knorps Maria  
78/6  

**Experimental and Numerical Investigation of a Pulsated Evaporating Spray**  
Brandle De Motta Jorge Cesar, Robbes Benjamin, Shum Kivan Francis, Boileau Matthieu, Massot Marc, Riber Eleonore  
79/1  

**Turbophoresis: The Turbulent Soret Effect**  
Haugen Nils Erland L., Mitra Dhrubaditya, Rogachevskii Igor  
79/2  

**The Influence of the Reynolds Number and the Particle Size on Particle-Laden Turbulent Channel Flow in the Dense Regime**  
Costa Pedro, Picano Francesco, Brandt Luca, Breugem Wim-Paul  
79/3  

---  

**We 9:50-11:10**  
**Memorial session dedicated to Prof. Gad Hetsroni**  
*Room: Affari-0.A*  

**Transient Changes of the Liquid Level in Stratified Flow Along the Pipe**  
Barnea Dvora, Taitel Yehuda  
79/4  

**Experimental Investigation of the Interaction Between a Stationary Rigid Sphere and a Turbulent Boundary Layer**  
Van Hout Rene, Eisma Jerke, Overmars Edwin, Elsinga Gerrit, Westerweel Jerry  
79/5  

**Bubble Dynamics at a Non-Flat Wall**  
Shmueli Havazelet, Letan Ruth, Ziskind Gennady  
79/6  

---  

**PARALLEL SESSIONS X**  

---  

**We 11:30-12:50**  
**Bubbly Flows**  
*Room: Affari-2.B*  

**Effects of Gravity and Its Orientation of Bubble Departure**  
Di Marco Paolo, Marganti Nicolo, Sacone Giacomo  
80/1  

**Flow and Shear Induced by Bubbles Rising Through Hollow-Fiber Membranes in a Parallel Arrangement**  
Tsuchiya Katsumi, Yamada Saori, Umemura Naoki, Mori Yasushige  
80/2  

**Bubbly Free and Impinging Forced Jets: Experimental Study by PIV/PFI/PTV**  
Pervunin Konstantin, Timoshesvky Mikhail, Markovich Dmitriy  
80/3  

**Hydrodynamic Studies of Cutting Bubbles in Micro-Structured Square Bubble Column Using Ultrafast X-Ray Tomography**  
Nakamura Sujatha Krushnathaj, Kuipers J. A. M (hans), Lau Yuk Man, Barthel Frank, Hampel Uwe, Schubert Markus, Deen Niels G.  
80/4  

---  

**We 11:30-12:50**  
**Boiling, Condensation, Evaporation**  
*Room: Affari-3.D*  

**Subcooled Flow Boiling Experiments: Towards Modelling the Near-Wall Behaviour**  
Estrada Perez Carlos E., Yoo Junsoo, Hassan Yassin A.  
80/5  

**Boiling Heat Transfer to Immiscible Mixtures for Advanced Cooling of Semiconductors**  
Ohta Haruhiko, Ihata Keisuke, Yamamoto Aisuke, Okayama Satoshi, Shimoto Yasuhisa  
80/6  

**Pool Nucleate Boiling for Seawater Containing Minerals**  
Uesawa Shinichiro, Koizumi Yasuo, Yoshida Hiroyuki  
81/1  

---  

**We 11:30-12:50**  
**Droplet Flows**  
*Room: Affari-0.A*  

**CFD Analysis of Oil-Water Two-Phase Flow at Flow Regime Transition in Horizontal Pipes**  
Burlutskii Evgenii, Morvan Herve, Kumar Kiran  
81/2  

---
ICMF 2016 International Conference on Multiphase Flow
Firenze, Italy, May 22 - 27, 2016

OIL-WATER PIPE FLOW DEVELOPMENT AFTER A VALVE
Schumann Heiner, Chandra Pankaj, Nydal Ole Jørgen

EXPERIMENTAL INVESTIGATION AND MODELLING OF DISPERSED OIL-WATER PIPE FLOW
Hernandez Rodriguez Iara, Hernandez Rodriguez Oscar Mauricio

THE DISPERSED OIL-WATER FLOW VELOCITY MEASUREMENT BASED ON CONTINUOUS-WAVE ULTRASONIC DOPPLER
Dong Xiaoxiao, Tan Chao, Yuan Ye, Dong Feng

We 11:30-12:50 Experimental Methods for Multiphase Flows
Room: Affari-3.C
NEW GAS HOLDUP EXPERIMENTAL DATA IN LARGE COUNTER-CURRENT BUBBLE COLUMNS
De Guido Giorgia, Besagni Giorgio, Inzoli Fabio, Pellegrini Laura

ULTRASONIC REFLECTOR RECOGNITION AND TRACKING TECHNIQUE FOR GAS BUBBLES IN LIQUID
Povolny Antonin, Kikura Hiroshiige, Ihara Tomonori

FLOW REGIME AND MASS TRANSFER STUDY IN STRUCTURED PIPE REACTORS USING ELECTROCHEMICAL MICRO-PROBES
Altheimer Marco, Becker Dustin, Rudolf von Rohr Philipp

TOMOGRAPHIC INTERROGATION OF GAS-LIQUID FLOWS IN INCLINED RISERS
Abdulkareem Lokman A., Escrig Escrig Josep, Reinecke Sebastian, Hewakandamby Buddhikha, Azzopardi Barry James

We 11:30-12:50 Granular Media & Granular flow
Room: Affari-3.B
DEVELOPMENT OF A DISCRETE GRAIN BREAKAGE MODEL TO PREDICT EROSION MECHANISM
Bharadwaj Rahul, Ropelato Karolline, Potapov Alexander, Nelson Brent

THE EFFECT OF AIR PRESSURE GRADIENTS ON THE DISCHARGE OF GRANULAR MATERIALS FROM A SILO
Zhou Yixian, Aussillous Pascale, Ruyer Pierre

DENSE GRANULAR FLOW IN A BUBBLING FLUIDIZED BED AS A VISCOPLASTIC FLUID
Sasic Srdjan, Jareteg Klas, Strom Henrik

NUMERICAL STUDY OF MULTI-SIZED GRANULAR PARTICLES IN ROTATING TUMBLER
Zhang Ziwei, Ge Liang, Gui Nan, Li Zhenlin

We 11:30-12:50 Industrial Applications
Room: Affari-1.A
A TWO-DIMENSIONAL CFD MODEL FOR GAS-LIQUID FLOWS IN A COIL-WOUND HEAT EXCHANGER
Acher Thomas, Knaup Manuel, Goll Stephan, Zander Hans-Jorg, Lenz Stefan, Kerber Christiane

SAND TRANSPORT IN SUBSEA MULTIPHASE FLOW PIPELINES
Yang Yan, Wang Shuli, Wen Chuang

CFD MODELLING OF GRAVITY SEPARATORS USING PID CONTROLLER
Vani Manoj, Das Saurish, Punekar Hemant, Mohan Srinivasa

COMPREHENSIVE STUDY OF ELECTRO-CHLORINATION PROCESS IN AN UNDIVIDED CELL: A COMPUTATIONAL FLUID DYNAMICS APPROACH
Momeni Maryam, Grann Helge, Doeleweerd Karin

We 11:30-12:50 Interfacial Flows
Room: Affari-2.A
EXPERIMENTS FOR CORE-ANNULAR FLOW (TO STUDY THE LEVITATION MECHANISM AND THE EFFECT OF BENDS)
Ingen Housz Eduard, Ooms Gijsbert, Henkes Ruud, Pourquie Mathieu, Kidess Anton

AN IMPROVED SHARP INTERFACE METHOD FOR SPH
Zhang Mingyu, Deng Xiaolong

EXPERIMENTAL STUDY ON THE DEVELOPMENT OF SURFACE GEOMETRY UNDER FINGERING INSTABILITY
Ito Takahiro, Yamashita Masatoshi, Nakagawa Kazuyuki, Tsuji Yoshiyuki

GROWTH AND SPECTRA OF GRAVITY-CAPILLARY WAVES IN COUNTERCURRENT AIR/WATER TURBULENT FLOW
Zonta Francesco, Onorato Miguel, Soldati Alfredo

We 11:30-12:50 Micro- and Nano-Scale Multiphase Flows
Room: Affari-2.C
EXPERIMENTAL INVESTIGATION OF DROPLET GENERATION IN A MICROFLUIDIC T-JUNCTION USING NEWTNIAN NANOFLOUIDS
Loizou Katerina, Lester Edward, Hewakandamby Buddhikha

XXVIII
<table>
<thead>
<tr>
<th>Session</th>
<th>Room</th>
<th>Title</th>
<th>Authors</th>
</tr>
</thead>
<tbody>
<tr>
<td>We 11:30-12:50 Mixing</td>
<td>Affari-3.A</td>
<td>PARTICLE DISTRIBUTION IN UNBAFFLED STIRRED VESSELS</td>
<td>Tamburini Alessandro, Cipollina Andrea, Scargiali Francesca, Micale Giorgio, Cicinno Michele, Brucato Alberto</td>
</tr>
<tr>
<td></td>
<td></td>
<td>EXPERIMENTAL ANALYSIS OF GAS-LIQUID DISPERSION IN MECHANICALLY STIRRED TANK BY MEANS OF QUANTITATIVE ELECTRICAL RESISTANCE TOMOGRAPHY</td>
<td>Busciglio Antonio, Montante Giuseppina, Paglianti Alessandro</td>
</tr>
<tr>
<td></td>
<td></td>
<td>A CONDITIONAL QUADRATURE METHOD OF MOMENTS FOR TURBULENT MIXING COUPLED TO A POPULATION BALANCE EQUATION</td>
<td>Passaleacqua Alberto, Madadi Kandjani Ehsan, Heylmun Jeffrey, Fox Rodney</td>
</tr>
<tr>
<td></td>
<td></td>
<td>COMPARISON BETWEEN DIFFERENT METHODS FOR THE SOLUTION OF THE BIVARIATE POPULATION BALANCE EQUATION</td>
<td>Buffo Antonio, Ville Alopaeus</td>
</tr>
<tr>
<td>We 11:30-12:50 Modelling of Multiphase Flows</td>
<td>Affari-2.D</td>
<td>MODELLING OF THE FLOW REGIME TRANSITION WITH THE SMOOTHED PARTICLE HYDRODYNAMICS</td>
<td>Olejnik Michal, Szewc Kamil, Pozorski Jacek</td>
</tr>
<tr>
<td></td>
<td></td>
<td>A COMPARATIVE ASSESSMENT OF LATTICE BOLTZMANN AND VOLUME OF FLUID (VOF) APPROACHES FOR GENERIC MULTIPHASE PROBLEMS</td>
<td>Mukherjee Siddhartha, Zanghanmi Ahad, Haringa Cees, Kenjeres Sasa, Van Den Akker Harry</td>
</tr>
<tr>
<td></td>
<td></td>
<td>EULERIAN TRAJECTORY METHOD AND IMPINGEMENT MODEL FOR NON-SPHERICAL ICE CRYSTALS</td>
<td>Nordal Ellen, Van Der Weide Edwin, Hоеijmakers Harry</td>
</tr>
<tr>
<td>We 11:30-12:50 Particle and Drop Dynamics</td>
<td>Villa-Sala Verde</td>
<td>SINGLE ELLIPSOID IN SIMPLE SHEAR FLOW: INFLUENCE OF INERTIA</td>
<td>Rosén Tomas, Kotsubo Yusuke, Do Quang Minh, Aidun Cyrus K., Lundell Fredrik</td>
</tr>
<tr>
<td></td>
<td></td>
<td>GAS ENTRAINMENT DUE TO A PLUNGING LIQUID JET IN FREE-SURFACE DOWNWARD ANNULAR-CHANNEL FLOW</td>
<td>Cavallini Barbosa Marcel, Ortiz Vital Luis Enrique, Hernandez Rodriguez Oscar Mauricio</td>
</tr>
<tr>
<td></td>
<td></td>
<td>INTRODUCING A MULTISCALE MODELLING TOOL FOR THE SIMULATION OF POLYURETHANE FOAMS</td>
<td>Ferkl Pavel, Karimi Mohtsen, Marchisio Daniele, Kosek Juraj</td>
</tr>
<tr>
<td></td>
<td></td>
<td>MODELING OF THE TRANSIENT TERRAIN-INDUCED AND SEVERE SLUGGING PROBLEMS USING THE DRIFT-FLUX MODEL</td>
<td>Sinkov Konstantin, Kharlashkin Andrey, Sponsitsev Pavel</td>
</tr>
<tr>
<td>We 11:30-12:50 Particle-Laden Flows</td>
<td>Villa-Auditorium</td>
<td>VOLUME-AVERAGED EQUATIONS FOR DIRECT NUMERICAL SIMULATION OF PARTICLE-DISPERSED FLOW UNDER A GRID RESOLUTION COMPARABLE TO PARTICLE DIAMETER AND KOLMOGOROV LENGTH SCALE</td>
<td>Fukada Toshiaki, Takeuchi Shintaro, Kaijima Takeo</td>
</tr>
<tr>
<td></td>
<td></td>
<td>MODELING AND PREDICTION OF GAS-SOLID TURBULENT MULTIPHASE FLOWS USING LAGRANGIAN AND EULERIAN APPROACHES</td>
<td>Dunn Dennis, Squires Kyle</td>
</tr>
<tr>
<td></td>
<td></td>
<td>FULLY EULERIAN SIMULATION OF 3D TURBULENT PARTICLE LADEN FLOW BASED ON THE ANISOTROPIC GAUSSIAN CLOSURE</td>
<td>Sabat Mascle, Vie Aymeric, Larat Adam, Massot Marc</td>
</tr>
<tr>
<td></td>
<td></td>
<td>A SPATIAL PARTICLE CORRELATION-FUNCTION ANALYSIS IN NON-ISOTHERMAL DILUTE PARTICLE-LADEN TURBULENT FLOWS</td>
<td>Masi Enrica, Pede Pascal, Simonin Olivier</td>
</tr>
</tbody>
</table>
PARALLEL SESSIONS XI

We 14:40-16:00  Bubbly Flows
Room: Affari-2.B

MODELING OF MACROSCALE SCLAR TRANSPORT IN BUBBLY SUSPENSIONS
Spelt Peter, Loisy Aurore, Naso Aurore

THE IMPACT LOAD AND THE POWER DENSITY DUE TO BUBBLE COLLAPSE IN 1D BUBBLY FLOWS AS INDICATORS FOR A MATERIAL EROSION MODEL
Bergeles Georgios, Gavaises Manolis

MULTIPHASE PHENOMENA IN UNDERWATER EXPLOSION NEAR THE FREE SURFACE
Zhu Weibing, Chen Hong, Khoo Boo Cheong

SPATIAL TRANSITION OF SYNTHETIC VOID WAVES IN A HORIZONTAL BUBBLY CHANNEL FLOW
Park Hyun Jin, Tasaka Yuji, Mural Yuichi

We 14:40-16:00  Boiling, Condensation, Evaporation
Room: Affari-3.D

TWO-PHASE FLOW AND HEAT TRANSFER CHARACTERISTICS IN NON-UNIFORMLY HEATED HELICAL COILED TUBES
Niu Xiao-Juan, Fan Liang-Liang, Luo Shuaishuai, Zhao Liang

A COMPUTATIONAL STUDY OF SURFACANT INDUCED ENHANCED HEAT TRANSFER IN FILM BOILING
Premnath Kannan, Welch Samuel

MEASUREMENT OF EBILLUTION CYCLE CHARACTERISTICS IN SUBCOOLED NUCLEATE BOILING OF WATER FLOWING IN A NARROW VERTICAL GAP
Anglart Henryk, Kromer Heiko, Fonellosa Caro Aleix, Laneryd Tor, Bel Fdhila Rebei

We 14:40-16:00  Droplet Flows
Room: Affari-0.A

ELECTROSTATIC CONTROL OF IMPACT DYNAMICS OF A DROPLET ON SOLID SURFACES: SUPERHYDROPHOBIC BEHAVIOR
Datta Saikat, Das Arup Kumar, Das Prasanta Kumar

A NOVEL DROPLET DRYING MODEL APPLIED TO A LABORATORY SPRAY DRYER
Darvan Ali, Sommerfeld Martin

FORMATION OF A LIQUID RING VIA DROP IMPACT
Kim Seungho, Moon Myoung-Woon, Kim Ho-Young

THE APPLICATION OF THE DEVELOPED IMAGE PROCESSING ALGORITHM TO STUDY THE BEHAVIOR OF WATER DROPLETS IMPINGING ON THE TILTED HEATED SURFACES
Kuntoro Hadiyan, Majid Akmal Irfan, Deendarlianto Deendarlianto, Takata Yasuyuki, Kohno Masamichi

We 14:40-16:00  Experimental Methods for Multiphase Flows
Room: Affari-3.C

WET GAS MEASUREMENT BY V-CONE THROTTLE DEVICE
He Denghui, Bai Bofeng

COMPARISON BETWEEN EXPERIMENTAL CHARACTERISATION OF THE MIXING OF IMMISCIBLE FLUIDS IN STATIC MIXER USING PLIF AND CFD MODELLING USING OPEN FOAM.
Alberini Federico, Forte Giuseppe, Daraio Alessio, Alexiadis Alessio, Simmons Mark

REFRACTIVE INDEX MATCHING METHOD TO STUDY GASOLINE SPRAY-WALL IMPINGEMENT
Lamarque Nicolas, Piccinni Leopardi Giacomo, Schmit Christoffer, Helie Jerome

DYNAMIC BEHAVIOUR OF TWO-PHASE FLOW THROUGH ORIFICES USING PARTICLE IMAGE VELOCIMETRY
Bamidele Olufemi Eyitope, Vines Chris, Ahmed Wael

We 14:40-16:00  Granular Media & Granular flow
Room: Affari-3.B

COMPUTATIONAL STUDY OF DENSE GRANULAR FLOWS IN STIRRED REACTORS
Bennani Lokman, Nsei Herve, Baudry Cyril, Lavieville Jerome, Fede Pascal, Simonin Olivier

LARGE EDDY SIMULATION OF SAND WAVE EVOLUTION
Khosronejad Ali, Sotropoulos Fotis

DRAG REDUCTION DUE TO ROTATION WITHIN GRANULAR MEDIA
Jung Wonjong, Choi Sung Mok, Kim Wonjung, Kim Ho-Young

ANALYTICAL AND PRACTICAL ANALYSIS OF FRICTIONAL-KINETIC MODEL FOR DENSE AND DILUTE GAS-SOLID FLOWS
Fede Pascal, Audard Francois, Belut Emmanuel, Fontaine Jean-Raymond, Simonin Olivier

XXX
We 14:40-16:00  Industrial Applications

Room: Affari-1.A

DEVELOPMENT OF AN INDUSTRIAL SCALE MICROBUBBLE GENERATOR FOR THE PURPOSES OF AEROBIC WASTE WATER TREATMENT
Majid Akmal Irfan, Deendarlianto Deendarlianto, Wiratni Wiratni, Indarto Indarto, Brata Enggar Diwangkara, Fungono Baskoro Adiputra, Tontowi Alva Edy
91/1

EFFICIENT MODELLING OF BUBBLE SIZE DISTRIBUTION IN A FERMENTATION REACTOR WITH CFD
Eppinger Thomas, Aglave Ravindra, Lo Simon
91/2

PHYSICAL AND MATHEMATICAL MODELLING OF MULTIPHASE FLOW IN THE NEAR-WALL REGION OF ENTRAINED FLOW REACTORS
Tretola Giovanni, Troiano Maurizio, Marra Francesco Saverio, Salatino Piero, Montagnaro Fabio, Solimene Roberto
91/3

EXPERIMENTAL INVESTIGATION OF AN AXIAL OIL-WATER SWIRL SEPARATOR
Mudda Robert, Hoeijmakers Harry, Ashok Anand, Hoppers Jacc
91/4

We 14:40-16:00  Micro- and Nano-Scale Multiphase Flows

Room: Affari-2.C

PRESSURE DROP FOR GAS-NON-NEWTONIAN LIQUID TWO-PHASE FLOW IN CIRCULAR MICROCHANNEL
Kawahara Akimaro, Sadatomi Michio, Law Wen Zhe, Mori Akifumi, Mansour Mohamed H.
91/5

MASS TRANSFER WITH NANOFLUIDS – THE ROLE OF MICROCONVECTION
Michaëlies Efstathios
91/6

PROPERTIES OF FINE BUBBLES AND OF NANO PARTICLES IN PURE WATER
Ueda Yoshikatsu, Tokuda Yomei, Nihei Naoto, Hamamoto Shoichiro, Ogawa Yuichi, Sugiyama Akifumi
92/1

We 14:40-16:00  Mixing

Room: Affari-3.A

USING TOMOGRAPHY TECHNIQUE TO ANALYZE THE SUSPENSION OF THE SOLID PARTICLES IN A SLURRY REACTOR
Ein Mozaffari Farhad, Mishra Prakash
92/2

BOTTOM BUBBLING CONDITIONS ON MIXING CHARACTERISTICS IN A GAS-STIRRED LADLE
Jeong Eunju, Lee Sang-Beom
92/3

MIXING INDUCED BY A HOMOGENOUS BUBBLE SWARM IN PRESENCE OF EXTERNAL TURBULENCE
Almeras Elise, Mathai Varghese, Lohse Detlef, Sun Chao
92/4

EXPERIMENTAL CHARACTERIZATION OF INTERCHANNEL MIXING OF MULTIPHASE FLOW THROUGH A NARROW GAP
Makiharju Simo, Gose James W., Buchanan John, Mychkovsky Alexander, Hogan Kevin, Lowe Kirk, Ceccio Steven L.
92/5

We 14:40-16:00  Modelling of Multiphase Flows

Room: Affari-2.D

MULTIPHASE EQUATIONS SUITABLE FOR THE NUMERICAL SIMULATION OF ICE PRODUCTION IN OCEAN
Covello Vanessa, Abba Antonella, Bonaventura Luca, Valdettaro Lorenzo
92/6

PREDICTION OF AEROSOL SAMPLING EFFICIENCY USING A SECTIONAL DROPLET DRIFT FLUX MODEL
Frederix Edo, Kuczaj Arkadiusz, Nordlund Markus, Gours Bernard
93/1

THE IMPACT OF DIFFERENT FINE GRID SIMULATIONS ON THE SUB-GRID MODIFICATION FOR GAS–SOLID DRAG
Schneiderbauer Simon, Pirker Stefan
93/2

NUMERICAL MODELING OF CONVECTIVE FLOW IN COMPLEX HYDROCARBON LIQUID UNDER RADIO-FREQUENCY HEATING
Kovaleva Liana, Kireev Victor, Musin Ayrat
93/3

We 14:40-16:00  Particle-Laden Flows

Room: Villa-Auditorium

AN EXPERIMENTAL AND NUMERICAL VALIDATION STUDY OF PARTICLE LADEN SUPERSONIC FLOWS
Rudek Arthur, Russ Gerald, Duignan Barry
93/4

PARTICLE IMAGE VELOCIMETRY MEASUREMENTS OF INJECTED PARTICLE PLUMES IN A TWO-DIMENSIONAL VERTICAL CHANNEL
Yates Matthew, Han David, Hewakandamby Buddhihka, Azzopardi Barry James
93/5

TURBULENCE ATTENUATION IN PARTICLE-LADEN FLOW
Vreman Bert
93/6
We 14:40-16:00

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

We 14:40-16:00

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

PARALLEL SESSIONS XII

We 16:20-17:40

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

We 16:20-17:40

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

We 16:20-17:40

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

XXXII
<table>
<thead>
<tr>
<th>Time</th>
<th>Session</th>
<th>Room</th>
<th>Title</th>
<th>Authors</th>
</tr>
</thead>
<tbody>
<tr>
<td>We 16:20-17:40</td>
<td>Granular Media &amp; Granular flow</td>
<td>Affari-3.B</td>
<td>AN EXPERIMENTAL STUDY OF HEAT TRANSFER MECHANISM IN GRANULAR FLOWS</td>
<td>Hsiau, Shu-San, Liu Shang-Yu, Chou Shih-Hao</td>
</tr>
<tr>
<td></td>
<td></td>
<td></td>
<td>FLUID-GRANULAR MEDIA MULTI-PHASE MODELLING USING THE SMOOTHED PARTICLE HYDRODYNAMICS METHOD</td>
<td>Szewc, Kamil</td>
</tr>
<tr>
<td></td>
<td></td>
<td></td>
<td>SATURATION LENGTH SCALE ON WALL PRESSURE DISTRIBUTION DURING A GRANULAR DISCHARGE FROM A NARROW SILO</td>
<td>Jiang, Mingqier, Yang, Fuling</td>
</tr>
<tr>
<td>We 16:20-17:40</td>
<td>Industrial Applications</td>
<td>Affari-1.A</td>
<td>COMPUTATIONAL MODELING OF DROPLET COALESCEENCE IN COMPLEX GEOMETRIES</td>
<td>Wang, Sheng, Agbaglah, Gilou, Desjardins, Olivier</td>
</tr>
<tr>
<td></td>
<td></td>
<td></td>
<td>NUMERICAL INVESTIGATION OF AN AXIAL OIL-WATER SWIRL SEPARATOR</td>
<td>Hospers, Jacco, Hoeijmakers, Harry</td>
</tr>
<tr>
<td></td>
<td></td>
<td></td>
<td>PREDICTION OF AIR RELEASE IN HYDRAULIC RESERVOIRS USING LAGRANGIAN PARTICLE TRACKING</td>
<td>Longhina, Marco, Murrenhoff, Hubertus</td>
</tr>
<tr>
<td></td>
<td></td>
<td></td>
<td>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</td>
<td>Picchi, Davide, Poesio, Pietro</td>
</tr>
<tr>
<td>We 16:20-17:40</td>
<td>Instabilities</td>
<td>Affari-3.C</td>
<td>FLOW REGIMES AND INSTABILITIES IN A T-MIXER OPERATING WITH A BINARY MIXTURE</td>
<td>Siconolfi, Lorenzo, Galletti, Chiara, Camarri, Simone, Salvetti, Maria Vittoria</td>
</tr>
<tr>
<td></td>
<td></td>
<td></td>
<td>THE CHARACTERISTIC SCALES OF THE CAPILLARY INSTABILITY</td>
<td></td>
</tr>
<tr>
<td></td>
<td></td>
<td></td>
<td>EXPERIMENTAL INVESTIGATION OF INTERFACIAL WAVES INSTABILITIES IN STRATIFIED FLOW</td>
<td>Dumouchel, Christophe, Aniszewski, Wojciek, Menard, Thibaut, Vu, Trung-Thanh</td>
</tr>
<tr>
<td></td>
<td></td>
<td></td>
<td>FARADAY INSTABILITY IN THREE FLUID MECHANICALLY OSCILLATED SYSTEMS</td>
<td>Dumouchel, Christophe, Aniszewski, Wojciek, Menard, Thibaut, Vu, Trung-Thanh</td>
</tr>
<tr>
<td>We 16:20-17:40</td>
<td>Mixing</td>
<td>Affari-3.A</td>
<td>EXPERIMENTAL STUDY ON FRAGMENTATION OF A MOLTEN JET INTO WATER</td>
<td>Hadj-Achour, Miloud, Rimbert, Nicolas, Castrillon-Escobar, Sebastian, Gradeck, Michel, Meignen, Renaud</td>
</tr>
<tr>
<td></td>
<td></td>
<td></td>
<td>GRANULAR MIXING INSIDE ROTATING CYLINDERS: PRELIMINARY RESULTS</td>
<td>De Monaco, Gaetano, Lombardi, Davide, Greco, Francesco, Maffettone, Pier Luca</td>
</tr>
<tr>
<td></td>
<td></td>
<td></td>
<td>SIMULATION OF THE DYNAMICS COMPLEX INFLUENCE ON HYDROCARBON MIXTURES</td>
<td>Liu, Nannan, Wang, Wei, Tian, Yunya, Han, Jiacheng, Zhang, Meng, Gong, Jing</td>
</tr>
<tr>
<td>We 16:20-17:40</td>
<td>Modelling of Multiphase Flows</td>
<td>Affari-2.D</td>
<td>PREDICTION OF INTERFACE LEVEL HEIGHT OF STRATIFIED LIQUID-LIQUID FLOW USING ARTIFICIAL NEURAL NETWORK</td>
<td>Awad, Mohamed, Aziz, Sadra, Ahmad, Ebrahim, Ibara, Roberto, Zadrail, Ivan, Markides, Christos, N.</td>
</tr>
<tr>
<td></td>
<td></td>
<td></td>
<td>TOWARDS LARGE-EDDY SIMULATION OF LIQUID ATOMIZATION: A-PRIORI SUBGRID ANALYSIS</td>
<td>Ketten, Sebastian, Klein, Markus</td>
</tr>
<tr>
<td></td>
<td></td>
<td></td>
<td>TOWARDS GENERAL PURPOSE LES MODEL OF ATOMIZATION</td>
<td>Hecht, Nicolas, Demoulain, Francois, Xavier, Reveillon, Julien</td>
</tr>
<tr>
<td></td>
<td></td>
<td></td>
<td>TWO-PHASE FLOWS IN SINUSOIDALLY CONSTRICTED CHANNELS USING MOVING MESH MOVING BOUNDARY TECHNIQUE</td>
<td>Anjos, Gustavo, Oliveira, Gustavo, Pontes, Jose, Mangiavacchi, Norberto, Thome, John</td>
</tr>
</tbody>
</table>
We 16:20-17:40

**Particle-Laden Flows**
*Room: Villa-Auditorium*

- **LARGE EDDY SIMULATION OF GAS-PARTICLE TURBULENT OPPOSED JET FLOW**
  Wu Dan, Li Jing, Liu Zhaohui, Zheng Chuguang
  100/1

- **LARGE EDDY SIMULATIONS OF PARTICULATE FLOW INSIDE A DIFFERENTIALLY HEATED CAVITY AT RAYLEIGH NUMBERS UP TO 10^10**
  Dehbi Abdel, Kallilainen Jarmo
  100/2

- **STUDY ON SEAWEED REMOVAL EFFICIENCY FROM THE WASTEWATER BY SWIRL FLOW SEPARATOR**
  Tsubone Hiroaki, Shimomura Masaya, Takesue Syuichi
  100/3

- **SWIRLING FLOW CURVATURE SENSITIVE TURBULENCE MODELLING OF A GAS-SOLID CYCLONE**
  Alahmadi Yaser, Nowakowski Andrzej
  100/4

We 16:20-17:40

**Reactive Multiphase Flows**
*Room: Affari-2.C*

- **UNSTEADY THREE-DIMENSIONAL NUMERICAL SIMULATIONS OF THE METHANE COMBUSTION IN DENSE FLUIDIZED BED**
  Hamidouche Ziad, Masi Enrica, Fede Pascal, Simonin Olivier, Ansart Renaud, Hemati Mehrdji
  100/5

- **NUMERICAL SIMULATION OF PROJECTILE LAUNCHED AT THE UNDERWATER USING GRANULAR PROPELLANTS**
  Asano Kei, Matsuo Akiko
  100/6

- **DIRECT NUMERICAL SIMULATION OF COMBUSTION NOISE OF A SPRAY FLAME**
  Pillai Abhishek, Kitano Tomaoki, Kurose Ryochi, Komori Satoru
  101/1

- **EFFECTS OF PERMEABILITY-DEPENDENT POROUS ELECTRODE COMPRESSED RATIO ON THE PERFORMANCE OF VANADIUM REDOX FLOW BATTERY**
  Wang Qiong, Qu Zhiguo
  101/2

We 16:20-17:40

**Turbulence in Multiphase Flows**
*Room: Villa-Sala Verde*

- **MODULATION OF TURBULENCE BY DISPERSED SOLID PARTICLES IN A SPATIALLY DEVELOPING FLAT PLATE BOUNDARY LAYER**
  Li Dong, Luo Kun, Fan Jianren
  101/3

- **ON THE EMULATION OF BUBBLE-SWARM-INDUCED TURBULENCE WITH FLEXIBLE PARTICLE GRIDS**
  Haase Katharina, Kuck Ulf, Thomling Jorg, Kaehler Christian
  101/4

- **DIRECT NUMERICAL SIMULATION MODEL INTERCOMPARISON FOR TURBULENT COLLISION STATISTICS OF INERTIAL PARTICLES**
  Rosa Bogdan, Onishi Ryo, Wang Lian-Ping
  101/5

- **MODELLING TURBULENCE IN LARGE SCALE BUBBLE PLUMES**
  Olsen Jan Erik, Johansen Stein Tore, Skjetne Paal
  101/6

We 16:20-17:40

**Fiber suspension flow (focus session)**
*Room: Affari-2.A*

- **ROTATION OF A NEUTRALLY BUOYANT SPHEROID IN A SIMPLE SHEAR AT SMALL REYNOLDS NUMBER**
  Einarsrud Jonas, Candelier Fabien, Lundell Fredrik, Angilella Jean-Regis, Mehlig Bernhard
  102/1

- **DNS OF TURBULENT CHANNEL FLOW LADEN WITH A DENSE SUSPENSION OF NEUTRALLY BUOYANT FINITE-SIZE SPHEROIDS**
  Niazi Ardekani Mehdi, Costa Pedro, Breugem Wim-Paul, Brandt Luca
  102/2

- **INVESTIGATION OF DRAG FORCE ON FIBRES OF BONDED SPHERICAL ELEMENTS USING A COUPLED CFD-DEM APPROACH**
  Jensen Anna Lyhne, Soerensten Henrik, Rosendahl Lasse, Adamsen Per, Lykholt Ustrup Flemming
  102/3

- **SUSPENSIONS OF INERTIAL SPHEROIDAL PARTICLES IN TURBULENT VERTICAL CHANNEL FLOW**
  Andersson Helge, Challabotla Niranjan Reddy, Zhao Lihaao
  102/4
## THURSDAY, May 26, 2016

### PARALLEL SESSIONS XIII

#### Thu 9:50-11:10 Computational Techniques for Multiphase Flows

**Room: Villa-Auditorium**

<table>
<thead>
<tr>
<th>Title</th>
<th>Authors</th>
<th>Page</th>
</tr>
</thead>
<tbody>
<tr>
<td>Uncertainty Quantification of Injected Droplet Size in Mono-Disperse Eulerian Simulations</td>
<td>Lancien Thea, Dumont Nicolas, Gicquel Olivier, Vicquelin Ronan</td>
<td>102/5</td>
</tr>
<tr>
<td>Two-Size Moment Eulerian Multi-Fluid Method Describing the Statistical Trajectory Crossing: Modeling and Numerical Scheme</td>
<td>Boileau Matthieu, Dupin Valentin, Laurent Frederique, Lagarde Julien, Massot Marc</td>
<td>102/6</td>
</tr>
<tr>
<td>A Comparative Study of the Surface Compression and Interface Reconstruction Algorithms for the VOF Approach</td>
<td>Cifani Paolo, Geurts Bernard, Kuerten Hans, Van Der Geld Cees, Michalek Witko, Priems Giel</td>
<td>103/1</td>
</tr>
<tr>
<td>IsoAdvector: A VOF Algorithm for Solving the Sharp Interface Advection Problem on Arbitrary Meshes</td>
<td>Roenby Johan, Bredmose Henrik, Jasak Hrvoje</td>
<td>103/2</td>
</tr>
</tbody>
</table>

#### Thu 9:50-11:10 Droplet Flows

**Room: Affari-3.D**

<table>
<thead>
<tr>
<th>Title</th>
<th>Authors</th>
<th>Page</th>
</tr>
</thead>
<tbody>
<tr>
<td>Dynamics of Double Emulsion Droplets in Shear and Channel Flows</td>
<td>Kim Sangkyu (Calvin), Dabiri Sadegh</td>
<td>103/3</td>
</tr>
<tr>
<td>Drop Breakup Models in Spray Simulations</td>
<td>Kekesi Timea, Amberg Gustav, Prabhu Wittberg Lisa</td>
<td>103/4</td>
</tr>
<tr>
<td>Levitation of Water Droplet Inside Ferrofluid: Computational Simulation and Experimental Demonstration</td>
<td>Singh Chamkor, Das Arup Kumar, Das Prasanta Kumar</td>
<td>103/6</td>
</tr>
</tbody>
</table>

#### Thu 9:50-11:10 Fluidization

**Room: Affari-3.A**

<table>
<thead>
<tr>
<th>Title</th>
<th>Authors</th>
<th>Page</th>
</tr>
</thead>
<tbody>
<tr>
<td>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</td>
<td>Municchi Federico, Radt Stefan, Goniva Christoph</td>
<td>104/1</td>
</tr>
<tr>
<td>Fluidizing Pulverised Coal with a Very Wide Size Distribution</td>
<td>Azizpandi Barry James, Li Xia, Escrib Escrig Josep, Hanne David</td>
<td>104/2</td>
</tr>
<tr>
<td>Uncertainties and Fluctuations in the Upscaling of Dense Inertial Particle Flows</td>
<td>Icardi Matteo</td>
<td>104/3</td>
</tr>
<tr>
<td>Study on the Compression, Flow and Permeation of Fine and Ultra-Fine, Cohesive and Compressible Powders</td>
<td>Kamranian Marnani Abbas, Bueck Andreas, Antonyuk Sergey, Thevenin Dominique, Tomas Jurgen</td>
<td>104/4</td>
</tr>
</tbody>
</table>

#### Thu 9:50-11:10 Industrial Applications

**Room: Affari-1.A**

<table>
<thead>
<tr>
<th>Title</th>
<th>Authors</th>
<th>Page</th>
</tr>
</thead>
<tbody>
<tr>
<td>Towards High Order Accuracy Cavitation Simulation with the Discontinuous Galerkin Spectral Element Method</td>
<td>Hempert Fabian, Boblest Sebastian, Hoffmann Malte, Offenhaeuser Philipp, Iben Uwe, Munz Claus-Dieter</td>
<td>104/5</td>
</tr>
<tr>
<td>Evaluation of Void Fraction Around a Tube in Two-Phase Flow Across Horizontal Tube Bundle</td>
<td>Murakawa Hideki, Baba Misaki, Sugimoto Katsumi, Takenaka Nobuyuki</td>
<td>104/6</td>
</tr>
<tr>
<td>CFD-Informed Unified Closure Relation for the Rise Velocity of Taylor Bubbles in Pipes with Liquid Flow</td>
<td>Lizarraga-Garcia Enrique, Buongiorno Jacopo, Al Safran Eissa, Lakehal Djamel</td>
<td>105/1</td>
</tr>
<tr>
<td>Numerical Analysis of Flow Characteristics Inside Hydraulic Coupling at Various Operational Conditions</td>
<td>Kwak Minhee, Hur Nahmkeon, Chang Choon-Soo, Kang Nam-Won</td>
<td>105/2</td>
</tr>
</tbody>
</table>
Thu 9:50-11:10  Instabilities

**Room:** Affari-3.C

**STABILITY OF STRATIFIED TWO-PHASE FLOWS IN INCLINED CHANNELS**  
Barmak Ilya, Gelfgat Alexander, Ullmann Amos, Brauner Neima  
105/3

**THREE-DIMENSIONAL STUDY OF DISTURBANCE WAVES FORMATION IN DOWNWARD FLOW OF GAS-SHEARED LIQUID FILM**  
Cherdantsev Mikhail, Isaenko Sergey, Cherdantsev Andrey, Markovich Dmitry, Alekseenko Sergey  
105/4

**INVESTIGATION OF PARALLEL CHANNEL INSTABILITIES WITH VAPOR BACKFLOW USING AN IMPROVED DYNAMIC MICROCHANNEL EVAPORATOR MODEL**  
Saenen Tom, Thome John  
105/5

**INSTABILITIES IN STRATIFIED TWO THERMOVISCOUS LIQUIDS FLOW IN A PLANE CHANNEL**  
Kireev Victor, Nizamova Adelina, Urmanceev Said  
105/6

---

Thu 9:50-11:10  Micro- and Nano-Scale Multiphase Flows

**Room:** Affari-2.B

**CFD SIMULATION OF GROWING NANOPARTICLES IN A FSP FLAME USING DIRECT QUADRATURE METHOD OF MOMENTS**  
Bianchi Pedro, Nonier Dirceu, Meier Henry França, Fritsching Udo, Meierhofer Florian  
106/1

**STUDY OF NATURAL CONVECTION IN AN ENCLOSURE WITH WATER AND a AL2O3/WATER NANOFLUID FOR DIFFERENT ASPECT RATIOS USING STATISTICAL ANALYSIS**  
Chowdhury Rajesh, Subudhi Sudhakar  
106/2

**EXPERIMENT AND COMPUTATIONAL SIMULATIONS OF LIQUID-LIQUID INTERFACIAL INSTABILITIES UPON FLOW DISPLACEMENT IN MICROCHANNELS**  
Lu Yu, Simmons Mark  
106/3

**CFD SIMULATION OF WATER REMOVAL FROM GAS FLOW CHANNELS OF PEMFC USING WETTABILITY GRADIENT**  
Khan Shadman, Ghosh Sumana  
106/4

---

Thu 9:50-11:10  Modelling of Multiphase Flows

**Room:** Affari-2.D

**MOFFAT VORTICES IN THE VICINITY OF A CONTACT LINE**  
Febres Soria Mijail, Legendre Dominique  
106/5

**WET GAS FLOW REGIMES: EXPERIMENTS AND TRANSITION MODEL BASED ON DIMENSIONLESS NUMBERS**  
Van Putten Dennis, Hagmeijer Rob  
106/6

**VAPORIZATION AND EXPANSION OF MOLTEN FUEL IN A SODIUM FAST REACTOR: A COMBINED NON-EQUILIBRIUM THERMODYNAMICS AND DIMENSIONAL ANALYSIS METHODOLOGY**  
Manchon Xavier, Bertrand Frederic, Marie Nathalie, Lance Michel, Schmitt Damien  
107/1

**CONDENSATION IN SUPERSONIC STEAM EJECTORS: COMPARISON OF THEORETICAL AND NUMERICAL MODELS**  
Mazzelli Federico, Brezgin Dmitry, Murmanskii Iilia, Nikolay Zhelonkin, Milazzo Adriano  
107/2

---

Thu 9:50-11:10  Particle-Laden Flows

**Room:** Affari-3.B

**EXPERIMENTAL STUDY OF PARTICLES BEHAVIOR IN TORNADO-LIKE GAS-SOLID FLOW**  
Protasov Mikhail, Vasilev Nikolay, Varaksin Aleksei  
107/3

**PARTICLE-LADEN FLOW IN MODELS OF ASTHOMATIC AIRWAYS**  
Koullapis Pantelis, Kassinos Stavros  
107/4

**SIMULATION OF A LIQUID-SOLID FLUIDIZED BED CRYSTALLIZER USING CFD-DEM**  
Kerst Kristin, Medeiros De Souza Luis, Seidel-Morgenstern Andreas, Janiga Gábor  
107/5

**THERMODYNAMICALLY CONSISTENT TWO-WAY COUPLED MODELLING OF GAS-PARTICLE FLOWS WITH PARTICLE TRAJECTORY CROSSING USING EULERIAN MOMENT METHODS**  
Massot Marc, Dupif Valentin, Dupays Joel, Laurent Frederique  
107/6

---

Thu 9:50-11:10  Reactive Multiphase Flows

**Room:** Affari-2.C

**TRANSITION BETWEEN ROBUST EQUILIBRIUM DESIGN POINTS FOR STEADY STATE PROCESSES OF PULVERIZED COAL COMBUSTION IN DOWNERS**  
Eluk Tal, Levy Avi, Korytnyi Efim, Bar Kohany Tal  
108/1

**PHYSICAL AND COMBUSTION CHARACTERIZATION OF BIOMASS PARTICLES PULVERIZED BY DIFFERENT MILLING PROCESSES FOR SUSPENSION FIRING IN UTILITY BOILERS**  
Yin Chungen, Momeni Maryam, Kaer Soren Knudsen  
108/2

**EXPERIMENTAL UNCERTAINTY RECONCILIATION IN DROP TUBE CHARACTERIZATION OF REACTING STREAMS OF PARTICLES**  
Galletti Chiara, Caposcitti Gianluca, lavarone Salvatore, Parente Alessandro, Tognotti Leonardo  
108/3

---

XXXVI
THE INFLUENCE OF REACTOR ENCLOSURE AND GAS ENTRAINMENT ON FLAME SPRAYS WITH APPLICATION IN NANOPARTICLE SYNTHESIS
Meierhofer Florian, Buss Lizoel, Nonier Dirceu, Meier Henry França, Fritsching Udo 108/4

Thu 9:50-11:10 Turbulence in Multiphase Flows
Room: Villa-Sala Verde
MEASUREMENTS OF VOID FRACTION, PRESSURE DROP AND HEAT TRANSFER IN HORIZONTAL AND DOWNWARD INCLINED GAS-LIQUID STRATIFIED FLOW
Bhagwat Swanand, Ghajar Afshin 108/5
STUDY OF A TURBULENCE DAMPING PROCEDURE NEAR THE INTERFACE OF GAS-LIQUID STRATIFIED FLOW
Gada Vinesh H., Punde Pranav, Tandon Mohit P., Vikulov Roman 108/6
MODELING INTER-PHASE MOMENTUM COUPLING VIA THE EXACT REGULARIZED POINT PARTICLE METHOD
Gualtieri Paolo, Battista Francesco, Casciota Carlo Massimo 109/1
DNS OF EVAPORATING DROPLETS IN DECAYING ISOTROPIC TURBULENCE
Dodd Michael, Ferrante Antonino 109/2

Thu 9:50-11:10 Fiber suspension flow (focus session)
Room: Affari-2.A
LONG FLEXIBLE FIBERS IN TURBULENT FLOWS
Sulaiman Mostafa, Climent Eric, Delmotte Blaise, Fede Pascal, Plouraboue Franck, Verhille Gautier 109/3
NUMERICAL SIMULATION AND VALIDATION OF ELONGATED FIBRES DISPERSION AND TRANSPORT
Lain Santiago, Quintero Brian, Sommerfeld Martin 109/4
NUMERICAL SIMULATION OF FLEXIBLE FIBER CONCENTRATION DISTRIBUTION IN A TWO-DIMENSIONAL ANALYTICAL FLOW FIELD FOR DIFFERENT STOKES NUMBERS
Martinez Manuel, Pallares Jordi, Vernet Anton 109/5
DEViatoric Stresses in Suspensions of Flexible or Curved Fibers
Lindstrom Stefan, Andric Jelena, Sasic Srdjan,Nilsson Hakan 109/6

PARALLEL SESSIONS XIV

Thu 11:30-12:50 Colloidal and Suspension Dynamics
Room: Affari-3.B
BASIC EQUATIONS OF SEPARATION PROCESSES IN SUSPENSIONS WITH GRAVITY, CENTRIFUGAL AND CORIOLIS FORCES
Schneider Wilhelm 110/1
SIMULATIONS TO explore the Stability of Nanoparticle Suspensions Under Hydrodynamic Shear
Vo Minh, Papavassiliou Dimitrios 110/2
SEDIMENTATION OF DILUTE SUSPENSIONS OF SPHERES AT FINITE REYNOLDS NUMBERS
Bergougnoux Laurence, Guazzelli Elisabeth 110/3
A NUMERICAL STUDY FOR SUSPENSION PARTICLES MIXED WITH DROP FLOWS
Chen Ming, Zhang Baili, Lou Jing 110/4

Thu 11:30-12:50 Computational Techniques for Multiphase Flows
Room: Villa-Auditorium
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
Vu Trung-Thanh, Menard Thibaut, Dumouchel Christophe 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
Piscaglia Federico, Montorfano Andrea, Helle Jerome, Demoulin Francois-Xavier 110/6
A LES-DEM-VOF METHOD FOR TURBULENT THREE PHASE FLOWS
Pozzetti Gabriele, Peters Bernhard 111/1
3D, GPU-ACCELERATED AND MPI-PARALLEL SIMULATIONS OF TWO-FLUID FLOWS INTERACTING WITH MOVING RIGID BODIES – APPLICATION IN RENEWABLE ENERGY SYSTEMS
Raessi Mehdi, Pathak Ashish 111/2

Thu 11:30-12:50 Droplet Flows
Room: Affari-3.D
COALESCEnce AND BREAK-UP OF LARGE, DEFORMABLE DROPLETS WITH DIFFERENT ViscosITIES IN A TURBULENT CHANNEL FLOW
Roccon Alessio, Zonta Francesco, Soldati Alfredo 111/3

XXXVII
### Fluidization

**Room: Affari-3.A**

1. Numerical Analysis of Dense Fluidization: A Benchmark for Particle Resolved Simulations of Liquid-Solid and Gas-Solid Fluidized Beds
   - Esteghamatian Amir, Bernard Manuel, Hammouti Abdelkader, Wachs Anthony, Uhlmann Markus, Lance Michel
   - 112/1

2. Numerical Simulation of Periodic Circulating Fluidized Bed of Binary Mixture of Particles – Budget Analysis
   - Chevrier Solene, Fede Pascal, Simonin Olivier
   - 112/2

3. A New Drag Correlation from Particle-Resolved Direct Numerical Simulation of a Gas-Fluidized Bed
   - Tan Junhua, Luo Kun, Fan Jianren
   - 112/3

4. Drag Force Modelling in Dilute to Dense Particle-Laden Flows with Monodisperse or Binary Mixture of Solid Particles
   - Simonin Olivier, Chevrier Solene, Audard Francois, Fede Pascal
   - 112/4

---

### Instabilities

**Room: Affari-3.C**

1. Instability Due to Interfacial Tension in Stratified and Core-Annular Flow
   - Hernandez Rodriguez Oscar Mauricio, Rocha Douglas Martins
   - 113/5

2. A Numerical Investigation of Flow Boiling Instability in Horizontal Tube
   - Wang Yufei, Zhang Liang, Yu Zi-Tao
   - 113/6

   - Liu Jialun, Li Hui-Xiong, Li Liangxing, Zhang Qian
   - 113/1

4. Viscous Vortex Reconnection in the Cavitating Swirl Flow
   - Tsoy Mikhail, Shirkob Eygen, Alekseenko Sergey, Kuibin Pavel
   - 113/2

---

### Modelling of Multiphase Flows

**Room: Affari-2.D**

1. Droplet Condensation/Evaporation in a Turbulent Free Shear Flow: Mixing-Induced Supersaturation and Inertial Droplet Dynamics at the Turbulent/Non-Turbulent Interface
   - Keedy Ryan, Aliseda Alberto
   - 114/3

2. Multiple Holdup Problem in Developing Laminar-Laminar Stratified Flow in an Inclined Channel
   - Thibault Daniel, Line Alain, Belt Roel
   - 114/4

   - Salinas Pablo, Pavlidis Dimitrios, Xie Zhuhua, Pain Christopher, Jackson Matthew
   - 114/5

4. Statistical Modelling of Turbulence Interface Interactions with the Conservative Level-Set Method
   - Wiedawczyk Tomasz
   - 114/6

---

### Reactive Multiphase Flows

**Room: Affari-2.C**

1. Numerical Simulation of Bubbly Flows in an Aeration Tank with Biochemical Reactions
   - Khateeb Noor Ul Huda, Takagi Shu, Gong Xiaobo, Matsumoto Yoichiro
   - 114/1

   - Hlawitschka Mark W., Drefenstedt Sebastian, Schafer Jan, Bart Hans-Jorg
   - 114/2

3. Modelling of Slurry Staged Bubble Column for CO2 Capture in Carbonates Solution with Enzyme Catalysis
   - Marzocchella Antonio, Russo Maria Elena, Bareschino Piero, Olivieri Giuseppe, Chirone Riccardo, Salatino Piero
   - 114/3

4. Influence of Bubble Bouncing on Mass Transfer and Chemical Reaction
   - Timmermann Jens, Hoffmann Marko, Schlüter Michael
   - 114/4

---

XXXVIII
Thursday 11:30-12:50 | Turbulence in Multiphase Flows

**Room: Villa-Sala Verde**

- Bubble generated turbulence in two fluid simulation of bubble flow
  - Schwarz Mark Philip, Feng Yuqing
  - 114/5

- The wake effect of solid particles and combusting droplets on turbulence
  - Zhou Lixing, Li Ke, Zeng Zhuxiong
  - 114/6

- Drag reduction in dispersed multiphase Taylor-Couette turbulence: a numerical approach
  - Spandan Vamshi Arza, Verzicco Roberto, Lohse Detlef
  - 115/1

- Drag reduction in turbulent pipe flow induced by superhydrophobic surface
  - Costantini Roberta, Battista Francesco, Cascoli Carlo Massimo
  - 115/2

Thursday 11:30-12:50 | CFD modelling of multiphase flow

**Room: Affari-2.B**

- Eulerian multi-fluid simulations of local liquid spreading in a trickle bed
  - Buwa Vivek, Jindal Arpit
  - 115/3

- A novel multiscale modelling approach for flow through a cylindrical packed-bed reactor filled with porous non-spherical particles
  - Das Saurish, Kupers J. A. M (Hans), Deen Niels G.
  - 115/4

- Free surface flow improvements for poor mesh
  - Gupta Vinay Kumar, Punekar Hemant, Kvas Srikanth
  - 115/5

- Eulerian modelling of liquid flow redistribution in a random packed bed
  - Asendrych Dariusz, Niegodajew Pawel
  - 115/6

Thursday 11:30-12:50 | Fiber suspension flow (focus session)

**Room: Affari-2.A**

- Simulation of fiber suspension in a turbulent mixing layer with an Euler-Lagrangian method
  - Zhou Kun, Yang Wei
  - 116/1

- Turbulent dispersion of ellipsoidal particles in a wall-bounded turbulent flow
  - Ouchene Rafik, Hamidouche Souria, Arcen Boris, Khalij Mohammed, Taniere Anne
  - 116/2

- Modelling of fibre suspensions in turbulent flow: a pseudo-homogeneous approach
  - Rasteiro Maria Graca, Cotas Carla, Gracia Fernando, Asendrych Dariusz
  - 116/3

- MRI of cellulose fibre suspensions in a straight pipe: a new methodology for measuring the Reynolds stress components
  - Mackenzie Jordan, Soderberg Daniel, Lundell Fredrik, Swerin Agne
  - 116/4

Thursday 11:30-12:50 | Finite size particles/droplets

**Room: Affari-1.A**

- Finite-size particle transport by turbulent channel flow
  - Vinkovic Ivana, Yu Wenchao, Buffat Marc
  - 116/5

- Effect of Reynolds number and concentration on modulation of turbulence by finite size neutrally buoyant particles
  - Wang Guiguang, Abbas Micheline, Climent Eric
  - 116/6

- Particle resolved simulation of a 3D periodic Couette dense fluid-particle flow
  - Scorsim Oliver, Fede Pascal, Simonin Olivier
  - 117/1

- Analyzing the relation between finite-size resolved particles and coherent structures in forced homogeneous-isotropic turbulence
  - Uhlmann Markus, Chouippe Agathe
  - 117/2

PARALLEL SESSIONS XV

Thursday 15:10-16:30 | Colloidal and Suspension Dynamics

**Room: Affari-3.B**

- Dynamics of shear-induced migration of spherical particles in pipe flow
  - Guazzelli Elisabeth, Snook Braden, Butler Jason
  - 117/3

- Shear flow-induced enhancement in effective thermal conductivity of nanoparticle suspensions
  - Sun Chengzhen, Lu Wenchao, Bai Bofeng
  - 117/4

- Evidence of shear-induced migration in a pipe flow of a dense emulsion
  - Abbas Micheline, Pouplin Amelie, Masbernat Olivier, Line Alain, Decarre Sandrine
  - 117/5

- Dynamics of a shear-induced aggregation process by a combined Monte Carlo - Stokesian Dynamics approach
  - Frungieri Graziano, Vanni Marco
  - 117/6

XXXIX
Thu 15:10-16:30 Computational Techniques for Multiphase Flows
Room: Villa-Auditorium
EVALUATION OF GENTOP CONCEPT FOR THE BUOYANCY-DRIVEN MOTION OF A SINGLE BUBBLE
Schlegel Fabian, Gauss Frederike, Krepper Eckhard, Lucas Dirk 118/1
AN IMPLICIT HIGH ORDER DISCONTINUOUS GALERKIN LEVEL SET METHOD FOR TWO-PHASE FLOW PROBLEMS
Ta Thi Thanh Mai, Pigeonneau Francck, Saramito Pierre 118/2
DETAILED NUMERICAL SIMULATION OF SWIRLING PRIMARY ATOMIZATION USING LEVEL SET METHOD
Shao Changxiao, Luo Kun, Fan Jianren 118/3
A MULTI-SCALE EUERIERIAN-LAGRANGIAN METHOD FOR EFFICIENT DNS SIMULATION OF THE ASSISTED ATOMIZATION OF A PLANAR LIQUID SHEET
Estivalezes Jean-Luc, Zuzio Davide 118/4

Thu 15:10-16:30 Fluidization
Room: Affari-3.A
DIRECT NUMERICAL SIMULATION OF FLUIDIZATION OF SOLID PARTICLES DUE TO LIFT FORCES
Mratti Rohit, Peters Frank, Padding Johan, Kuipers J. A. M (hans) 118/5
WALL BOUNDARY CONDITIONS FOR NUMERICAL SIMULATION OF BUBBLING FLUIDIZED BEDS
Haghgoo Mohammad Reza, Bergstrom Donald J., Spiteri Raymond 118/6
LAGRANGIAN MEASUREMENT OF FORCE ON COARSE OBJECTS IN GAS-SOLID FLUIDIZED BED
Yoshimori Wataru, Ikegai Tomoki, Higashida Kyohei, Rai Kenta, Tsujitakuya, Oshitan Jun, Harada Shusaku 119/1
EVALUATION OF FLOWABILITY OF COHESIVE POWDER USING POWDER DISCHARGE METHOD BY AIR FLOW
Ogata Koichiro, Ouchi Koki, Fujita Seiji 119/2

Thu 15:10-16:30 Fluid-Structure Interactions
Room: Affari-3.D
COMPARATIVE STUDIES OF SHOCK INTERACTION WITH STATIC AND MOVING CYLINDERS THROUGH IMMERSED BOUNDARY METHOD
Luo Yujuan, Luo Kun, Jin Tai, Fan Jianren 119/3
INVESTIGATION OF GAS-LIQUID TWO-PHASE FLOW INDUCED VIBRATION CHARACTERISTICS
Miwa Shuichiro, Liu Yang, Takashi Hibiki, Mori Michtisugu 119/4
A FLUID-STRUCTURE INTERACTION SOLVER COUPLED TO A VOLUME OF FLUID METHOD
Cerroni Daniele, Fancellu Luca, Manservisi Sandro, Menghini Filippo, Scardovelli Ruben 119/5
MULTIPHASE FORCES ON BEND STRUCTURES
Nennie Erik, Belfroid Stefan 119/6

Thu 15:10-16:30 Industrial Applications
Room: Affari-1.A
LAY DOWN SIMULATION OF VISCOELASTIC FLUIDS USING THE HYBRID IMMERSED-BOUNDARY METHOD
Mark Andreas, Ingelsten Simon, Edelvik Fredrik 120/1
EFFECTS OF AZIMUTHAL ANGLE OF AERATION HOLES ON FLOWS INSIDE AND OUTSIDE A BUBBLE DIFFUSER PIPE
Sato Ryo, Miyayoshi Tatsuya, Hayashi Kosuke, Tomiyama Akio 120/2
INVESTIGATION OF AIR INJECTION PARAMETERS TOWARD OPTIMUM FUEL SAVING EFFECT FOR SHIPS
Lee Inwon, Park Seong Hyeon, Chun Ho Hwan 120/3
VALIDATION OF CFD PREDICTION OF SLOSHING EFFECTS FOR NUCLEAR REACTOR SEISMIC SAFETY
Myrillas Konstantinos, Planquart Philippe, Buchlin Jean-Marie, Schyns Marc 120/4

Thu 15:10-16:30 Instabilities
Room: Affari-3.C
RECONNECTION OF CAVITATING VORTEX TUBES WITH FORMATION OF VORTEX RINGS IN A SWIRL FLOW
Skripkin Sergey, Kutbin Pavel, Shtork Sergey, Alekseenko Sergey 120/5
LIGAMENTS FORMATION IN CO-FLOWING JETS WITH VISCOSITY STRATIFICATION
Loiseau Jean-Christophe, Tammiola Outi, Brandt Luca 120/6
JUMPING HOOPS ON LIQUID
Yang Eunjin, Kim Ho-Young 121/1
SIMULATIONS OF A SHOCK WAVE - PARTICLE CLOUD INTERACTION IN A SHOCK TUBE
Rollin Bertrand, Annamalai Subramanian, Ouellet Frederick, Koneru Rahul 121/2
### Thu 15:10-16:30 Micro- and Nano-Scale Multiphase Flows

**Room:** Affari-2.D

1. **Influence of DC Electric Field Upon the Production of Oil-In-Water-In-Oil Double Emulsions in MM-Scale Channels.**
   - Alberini Federico, Enjalbert Romain, Van Crombrugge Yann, Dapelo Davide, Simmons Mark
   - 121/3

2. **Oil-Water Plug Flow Formation in Small Channels**
   - Chinate Maxime, Roumpea Evangelia, Kahoudjdi Lyes, Matar Omar K., Angeli Panagioti
   - 121/4

3. **Effect of Solid Foam on Segmentation for CO2 Dissolution in Milli-Channel Taylor Flow**
   - Simoens Serge, Zhou Guanyu, Philippe Regis, Zanotta Marie-Line, Valiorgue Pierre
   - 121/5

4. **Experimental Investigation of Air Water Hydrodynamics in PEMFC Micro Gas Channels**
   - Malhotra Sneha, Ghosh Sumana
   - 121/6

### Thu 15:10-16:30 Reactive Multiphase Flows

**Room:** Affari-2.C

1. **Direct Numerical Simulation of a Single Rising Bubble with Interfacial Species Transfer and Chemisorption in the Liquid Phase**
   - Bezecny Daniel, Deising Daniel, Eller Sebastian, Marschall Holger, Schlueter Michael, Bothe Dieter
   - 122/1

2. **What Is the Effect of Turbulence on the Heterogeneous Reaction Rates in the High and Low Damköhler Number Regimes?**
   - Krueger Jonas, Haugen Nils Erland L., Lovas Terese
   - 122/2

3. **Large-Eddy Simulation of Sodium Release from Pulverized-Coal Jet Flames**
   - Wan Kaidi, Xia Jun, Wang Zhihua, Wrobel Luiz, Cen Kefa
   - 122/3

4. **Flow Pattern Visualization of Heated Kerosene at Sub- and Super-Critical Pressures**
   - Liu Zhaohui, Pan Hui, Feng Song, Yang Zhiqiang, Wang Fei, Bi Qinchen
   - 122/4

### Thu 15:10-16:30 Turbulence in Multiphase Flows

**Room:** Villa-Sala Verde

1. **Eulerian Multiphase Predictions of Turbulent Bubbly Flow in a Step-Expansion Channel**
   - Papoulas Dimitrios, Splawski Andrew, Vichansky Alexander, Lo Simon
   - 122/5

2. **Dispersion of Finite Size Droplets and Solid Particles in Isotropic Turbulence**
   - Rossso Michele, Wang Hengjie, Elghobashi Said
   - 122/6

3. **Large-Eddy Simulation of Particle Transport in a Volcanic Plume**
   - Suto Hitoshi, Hattori Yasuo, Toshida Kiyoshi
   - 123/1

4. **DNS of a Turbulent Steam/Water Bubbly Flow in a Vertical Channel**
   - Bois Guillaume, Mathieu Benoit, Fauchet Gauthier, Toutant Adrien
   - 123/2

### Thu 15:10-16:30 CFD modelling of multiphase flow

**Room:** Affari-2.B

1. **CFD Modelling of Liquid-Liquid Slug Flow in Capillaries**
   - Figueras Rosette Juliethe, Burns Alan
   - 123/3

2. **A Multi-Fluid Modelling Approach for the Air Entrainment of Ventilated Partial Cavity**
   - Xiang Min, Tu Jiyuan, Liu Longbin
   - 123/4

3. **CFD Modelling of Downward Two Phase Pipe Flow**
   - Krepper Eckhard, Lucas Dirk, Rzehak Roland
   - 123/5

4. **Simulation of Two Phase Flow Inside Air Conditioning Evaporators**
   - Schlottke Jan, Kuhnel Wolfram
   - 123/6

### Thu 15:10-16:30 Fiber suspension flow (focus session)

**Room:** Affari-2.A

1. **Rheological Characterization of Suspensions of Rigid Fibers: The Strong Effect of Confinement**
   - Butler Jason, Snook Braden, Guazzelli Elisabeth, Pouliquen Olivier
   - 124/1

2. **Flow Rheology of Fiber-Laden Aqueous Foams**
   - Koponen Antti, Selenius Pasi, Jasberg Ari
   - 124/2

3. **Dynamics and Rheology of a Dilute Viscoelastic Suspension of Spheroids in an Unbounded Shear Flow**
   - D Avino Gaetano, Greco Francesco, Maffettone Pier Luca
   - 124/3

4. **Browian Diffusion of Nano-Fibers**
   - Tian Lin, Ahmadi Goodarz, Tu Jiyuan
   - 124/4
<table>
<thead>
<tr>
<th>Time</th>
<th>Parallel Sessions XVI</th>
</tr>
</thead>
<tbody>
<tr>
<td>Thu 16:50-18:10</td>
<td><strong>Colloidal and Suspension Dynamics</strong></td>
</tr>
<tr>
<td></td>
<td><strong>Room: Affari-3.B</strong></td>
</tr>
<tr>
<td></td>
<td>INERTIA-DRIVEN MIGRATION AND MIXING OF PARTICLES IN A SUSPENSION CHANNEL FLOW</td>
</tr>
<tr>
<td></td>
<td>Loisel Vincent, Abbas Micheline, Masbernat Olivier, Climent Eric</td>
</tr>
<tr>
<td></td>
<td>SETTLING VELOCITY AND MICROSTRUCTURE OF SOLID PARTICLES IN SUSPENSIONS SIMULATIONS</td>
</tr>
<tr>
<td></td>
<td>WITH LATTICE-BOLTZMANN METHOD</td>
</tr>
<tr>
<td></td>
<td>Huang Yi, Li Jing, Zha Lu, Liu Zhaoxiao, Zheng Chuang</td>
</tr>
<tr>
<td></td>
<td>THE MOTION OF A CLOUD OF SOLID SPHERICAL PARTICLES FALLING IN A CELLULAR FLOW FIELD</td>
</tr>
<tr>
<td></td>
<td>AT LOW STOKES NUMBER</td>
</tr>
<tr>
<td></td>
<td>Marchetti Benjamin, Bergognaux Laurence, Bouchet Gilles, Guazzelli Elisabeth</td>
</tr>
<tr>
<td></td>
<td>TUMBLING DYNAMICS OF NON-SPHERICAL PARTICLES FALLING IN A TURBULENT FLOW</td>
</tr>
<tr>
<td></td>
<td>Gustavsson Kristian, Mehlig Bernhard</td>
</tr>
</tbody>
</table>

| Thu 16:50-18:10 | **Computational Techniques for Multiphase Flows**                                     |
|                | **Room: Villa-Auditorium**                                                             |
|                | FIDELITY REQUIREMENTS FOR THE LARGE EDDY SIMULATION OF DIESEL SPRAY                    |
|                | Doisneau Francois, Arienti Marco, Oeffelein Joseph                                      |
|                | SIMULATION OF EVAPORATING DROPLETS WITHIN A DISCONTINUOUS GALERKIN MULTI-SCALE FRAMEWORK |
|                | Hitz Timon, Fechter Stefan, Munz Claus-Dieter                                          |
|                | THERMOCAPILLARY DROPLET ACTUATION ON A WALL                                            |
|                | Fath Anja, Bothe Dieter                                                                |
|                | IMPLEMENTATION OF THE IMPLICIT-CONTINUOUS-EULERIAN SCHEME FOR MULTIPHASE FLOWS IN THE PARALLEL DUNE FRAMEWORK |
|                | Fochesatto Christophe, Zabiego Magali                                                   |

| Thu 16:50-18:10 | **Fluidization**                                                                        |
|                | **Room: Affari-3.A**                                                                   |
|                | EFFECT OF FRICTIONAL PACKING LIMIT ON NUMERICAL SIMULATION OF HYDRODYNAMICS OF DENSE GAS-SOLID FLOWS |
|                | Sahu Akhilesh Kumar, Vasudevan Raghavan, Bhamid Prasad                                  |
|                | THE DENSE DISCRETE PHASE MODEL FOR SIMULATION OF BUBBLING FLUIDIZED BEDS: VALIDATION AND VERIFICATION |
|                | Cloete Schalk, Amini Shahriar                                                          |
|                | FLUID PERMEABILITY THROUGH FIBROUS BED WITH TORTUOUS STRUCTURE                         |
|                | Otomo Ryoko, Kazuki Mori, Sho Sakai                                                    |
|                | MACROSCOPIC MODELLING OF PARTICLE FLUCTUANT KINETIC ENERGY IN A LIQUID FLUIDIZED BED   |
|                | Ozel Ali, Brandle De Motta Jorge Cesar, Masbernat Olivier, Simonin Olivier              |

| Thu 16:50-18:10 | **Fluid-Structure Interactions**                                                       |
|                | **Room: Affari-3.D**                                                                   |
|                | MEASUREMENT OF LIQUID FILM THICKNESS IN TWO PHASE FLOW OF HFC134A GAS-ETHANOL LIQUID IN A VERTICAL TUBE |
|                | Mori Shoji, Tanaka Yuta, Okuyama Kunito                                                 |
|                | DIRECT NUMERICAL SIMULATIONS OF TURBULENT BOUNDARY LAYER OVER HEMISPHERICAL ROUGH WALL |
|                | Liu Xiaofei, Zhao Hui, Wei Anyang, Luo Kun, Fan Jianren                                |
|                | LIQUID-GAS DYNAMIC SYSTEM UNDER VIBRATION                                              |
|                | O Hern Timothy, Torczynski John, Clausen Jonathan                                       |
|                | THE SWIRLING ANNULAR TWO-PHASE FLOW INSIDE A CIRCULAR PIPE - INTERFACIAL PHENOMENA, FLOW MECHANISM AND MODELLING |
|                | Bai Bofeng, Liu Wen, Liu Li                                                             |

<p>| Thu 16:50-18:10 | <strong>Micro- and Nano-Scale Multiphase Flows</strong>                                              |
|                | <strong>Room: Affari-2.D</strong>                                                                   |
|                | STEADY FILM FLOW OVER 2D TOPOGRAPHY WITH AIR INCLUSION                                |
|                | Varchanis Stylianos, Pettas Dionisios, Dimakopoulos Yiannis, Tsamopoulos John          |
|                | ON THE EFFECTS OF INERTIA IN TAYLOR FLOW IN RECTANGULAR MILLI-CHANNELS OF MODERATE ASPECT RATIO |
|                | Legendre Dominique, Abadie Thomas, Aubin Joelle                                        |
|                | THE INFLUENCE OF FLUID VISCOSITY ON SIZE-BASED PARTICLE SEPARATION IN THE MICROCHANNEL WITH SYMMETRIC SHARP CORNER STRUCTURES |
|                | Fan Liang-Liang, He Xukun, Niu Xiao-Juan, Zhe Jiang, Zhao Liang, Yan Qing              |</p>
<table>
<thead>
<tr>
<th>Time</th>
<th>Session</th>
<th>Title</th>
<th>Authors</th>
<th>Page</th>
</tr>
</thead>
<tbody>
<tr>
<td>Thu 16:50-18:10</td>
<td>Reactive Multiphase Flows</td>
<td>Yielding the Yield-Stress Analysis: A Study Focused on the Effects of Elasticity on the Settling of a Single Particle</td>
<td>Svensson Martin, Mark Andreas, Martensson Gustaf, Karawajczyk Andrzej, Edelvik Fredrik</td>
<td>128/1</td>
</tr>
<tr>
<td></td>
<td></td>
<td>A Combined LES-PBE-PDF Method for Modelling Polydispersed Particle Formation in Turbulent Reacting Flows</td>
<td>Sewerin Fabian, Rigopoulos Stelios</td>
<td>129/3</td>
</tr>
<tr>
<td></td>
<td></td>
<td>Turbulent Thermal Convection Induced by the Heating of a Particle Suspension</td>
<td>Zamansky Remi, Coletti Filippo, Massot Marc, Mani Ali</td>
<td>129/4</td>
</tr>
<tr>
<td></td>
<td></td>
<td>DNS of Co-Current Turbulent Stratified Two-Phase Pipe Flow</td>
<td>Breugem Wim-Paul, Niazi Ardekan Mehdi, Birvalski Milos, Tummers Mark, Henkes Ruud</td>
<td>129/5</td>
</tr>
<tr>
<td>Thu 16:50-18:10</td>
<td>CFD modelling of multiphase flow</td>
<td>Evaporation in Supersonic CO2 Ejectors: Analysis of Theoretical and Numerical Models</td>
<td>Giacomelli Francesco, Mazzelli Federico, Milazzo Adriano</td>
<td>129/6</td>
</tr>
<tr>
<td></td>
<td></td>
<td>A Physically Consistent Subgrid Scale Boiling Approach for Multiphase CFD</td>
<td>Baglietto Emilio</td>
<td>130/1</td>
</tr>
<tr>
<td></td>
<td></td>
<td>CFD Study of Water Droplets in a Top Rotating Wall Doughnut Cell</td>
<td>Vardin Patrick, Brown Lloyd</td>
<td>130/2</td>
</tr>
<tr>
<td></td>
<td></td>
<td>Towards a More General CFD Modeling for all Flow Regimes</td>
<td>Mimouni Stephane, Fleau Solene, Vincent Stephane</td>
<td>130/3</td>
</tr>
<tr>
<td>Thu 16:50-18:10</td>
<td>Fiber suspension flow (focus session)</td>
<td>Energy Efficient High Solids Processing of Fiber Suspensions Using MRI</td>
<td>Cardona Maria, Tozzi Emilio, Karuna Nardrapee, Jeoh Tina, Powell Robert, Mccarthy Michael</td>
<td>130/4</td>
</tr>
<tr>
<td></td>
<td></td>
<td>Fibers Settling in Cellular Flow Field</td>
<td>Lopez Diego, Bergougnoux Laurence, Bouchet Gilles, Guazzelli Elisabeth</td>
<td>130/5</td>
</tr>
<tr>
<td></td>
<td></td>
<td>Experimental Study on Dispersion of Spherical Particles and Fibres in a Jet Penetrating a Crossflow</td>
<td>Pasternak Lars, Sommerfeld Martin</td>
<td>130/6</td>
</tr>
<tr>
<td></td>
<td></td>
<td>Modelling of Gas-Solid Flows with Non-Spherical Particles</td>
<td>Van Wachem Berend, Currant Thomas, Evrard Fabien, Wilson Robert</td>
<td>131/1</td>
</tr>
</tbody>
</table>
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 Casar, 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**  
**Colloidal and Suspension Dynamics**  
*Room: Affari-3.B*

- SETTLING OF FINITE-SIZE PARTICLES IN TURBULENT ENVIRONMENTS AT DIFFERENT GALILEO NUMBERS  
  Fornari Walter, Picano Francesco, Brandt Luca  
  131/6

- SCALE INDEPENDENCE OF COLLECTIVE PARTICLE SETTLING IN FRACTAL CHANNELS  
  Kurose Yudai, Ishizawa Kanami, Harada Shusaku  
  132/1

- PARTICLE FLUXES AND IRREVERSIBILITY DUE TO SHEAR FLOW IN A BIDISPERSE SUSPENSION  
  Howard Amanda, Maxey Martin, Yeo Kyongmin  
  132/2

**Fri 9:50-11:10**  
**Fluidization**  
*Room: Affari-3.A*

- VALIDATION OF THE TWO-PHASE EULER-FOAM SOLVER FOR JET INLET FLUIDIZED BEDS  
  Tretoli Giovanni, Marra Francesco Saverio  
  132/3

- GRANULAR TEMPERATURE AND PARTICLE VELOCITY-CONCENTRATION CROSS-CORRELATION IN A LIQUID FLUIDIZED BED  
  Ballussen Maike, Van Der Hoef Martin, Abbas Michelene, Masbernat Olivier  
  132/5

**Fri 9:50-11:10**  
**Micro- and Nano-Scale Multiphase Flows**  
*Room: Affari-2.D*

- AEROSOL SAMPLING OF SUB-MICRON PARTICLES USING A PORTABLE ELECTROSTATIC IONIZER: EFFECT OF BULK FLOW VELOCITY ON COLLECTION EFFICIENCY  
  Ramin Imani Jajarmi, Etienne Robert, Laila Ladhani, Van Der Wijngaart Wouter  
  133/1

- CONTINUOUS PRODUCTION OF ORGANIC POLYMERS IN MICROSTRUCTURE REACTORS: FEASIBILITY AND SCALE-UP  
  Brandner Juergen J., Kuesters Christof, Spasova Berta, Stengel Bruno  
  133/3

**Fri 9:50-11:10**  
**Non-Newtonian Multiphase Flows**  
*Room: Affari-3.C*

- HYDRODYNAMIC NON-NEWTONIAN LIQUID-SOLID FLOW IN COMPOUND TAYLOR-COUETTE FLOW  
  Aleksandrov Anatoly, Devisilov Vladimir, Sharay Elena  
  133/5

- DYNAMICS OF A RISING BUBBLE IN A VISCOELASTIC LIQUID  
  Niethammer Matthias, Marschall Holger, Kunkelmann Christian, Bothe Dieter  
  133/6

**Fri 9:50-11:10**  
**Turbulence in Multiphase Flows**  
*Room: Affari-2.A*

- TWO-FLUID MODEL PREDICTION OF GAS-LIQUID FLOW IN A VERTICAL PIPE USING AN ALGEBRAIC SECOND MOMENT CLOSURE  
  Adoo N.A., Islam A. S. M. Atiquel, Bergstrom Donald J.  
  134/3

- PREDICTION OF GAS VOLUME FRACTION FOR POLY-DISPERSE GAS-LIQUID TURBULENT FLOW IN A VERTICAL PIPE  
  Islam A. S. M. Atiquel, Adoo N.A., Bergstrom Donald J.  
  134/4
EXPERIMENTAL STUDY ON STRUCTURE OF A DILUTE GAS-PARTICLE AXISYMMETRIC OPPOSED JETS FLOW
Li Jing, Wang Hanfeng, Liu Zhaohui, Zheng Chuguang
TURBULENT STRUCTURES IN LEAD-BISMUTH FLOWS MEASURED BY USING INTRUSIVE PROBE METHODS
Ariyoshi Gen, Ito Daisuke, Saito Yasushi, Mishima Kaichiro

Fri 9:50-11:10
CFD modelling of multiphase flow
Room: Affari-2.B
CFD MODELLING OF POLYDISPERSE BUBBLY FLOWS
Vichansky Alexander
HIGH-ORDER MOMENT-CONSERVING METHOD OF CLASSES IN CFD CODE.
Jama Mohamed Ali, Buffo Antonio, Ville Alotapeus
CFD MODELLING OF PHASE SEPARATION AND COALESCENCE IN A MICELLAR THREE-PHASE SYSTEM
Wagner Lisa-Maria, Hohl Lena, Medeiros De Souza Luis, Thevenin Dominique, Kraume Matthias
CFD STUDY ON AGGLOMERATION IN A COUNTER-CURRENT SPRAY DRYING TOWER DUE TO PARTICLE-WALL INTERACTIONS
Ali Muzammil, Mahmud Tariq, Heggs Peter, Ghadiri Mojtaba, Bayly Andrew, Martindejuan Luis

Fri 9:50-11:10
Finite size particles/droplets
Room: Affari-2.C
FULLY RESOLVED SIMULATION OF FLUID FLOWS WITH SUSPENDED PARTICLES
Prosperetti Andrea, Sierakowski A., Willen D., Wang Y.
FULLY RESOLVED NUMERICAL SIMULATION OF INTER-PHASE HEAT TRANSFER IN GAS-SOLID TURBULENT FLOW
Xia Junjie, Luo Kun, Fan Jianren
DNS ANALYSIS OF WEAKLY-COMPRESSIBLE VAPORIZING TURBULENT TWO-PHASE FLOWS
Duret Benjamin, Menard Thibaut, Reveillon Julien, Demoulin Francois-Xavier
DIRECT NUMERICAL SIMULATION WITH HEAT TRANSFER OF GAS-PARTICLE-MULTIPHASE FLOW CONTAINING SHOCK WAVES BY IMMERSED BOUNDARY METHOD
Mizuno Yusuke, Takahashi Shun, Nonomura Taku, Nagata Takayuki, Fukuda Kota

PARALLEL SESSIONS XVIII

Fri 11:30-12:50
Computational Techniques for Multiphase Flows
Room: Affari-2.B
SHOCK-INDUCED COLLAPSE OF A SINGLE CAVITATION BUBBLE ATTACHED TO AN ELASTIC-PLASTIC SOLID
Turangan Cary
NUMERICAL INVESTIGATIONS OF SHOCK-INDUCED FLOW PATTERNS IN INHOMOGENEOUS MEDIA
Nowakowski Andrzej, Nicolleau Franck
A COMPUTATIONAL MODELLING APPROACH TO MINIMIZE PARAMETER SPACE AND GEOMETRY ADAPTABILITY WHEN INVESTIGATING CLEANROOM PARTICULATE LEVELS
Baker Nazar, O Sullivan Paul, Kelly Ger

Fri 11:30-12:50
Droplet Flows
Room: Affari-3.B
EFFECTS OF ANISOKINETIC SAMPLING IN STEAM QUALITY MEASUREMENTS FOR STERILIZATION PROCESSES
Serra Sonia
MODULATION OF SPRAY DROPLET NUMBER DENSITY AND SIZE DISTRIBUTION BY AN ACOUSTIC FIELD
Achury Javier, Polifke Wolfgang
DIRECT NUMERICAL SIMULATION OF MULTI-COMPONENT DROPLET INTERACTIONS
Schlotkke Karin, Meister Christian, Karch Grzegorz, Sadlo Filip, Erl Thomas, Weigand Bernhard

Fri 11:30-12:50
Fluid-Structure Interactions
Room: Affari-3.A
MORPHODYNAMICS AROUND A SLENDER CYLINDER UNDER A WAVY BOUNDARY LAYER
Corvaro Sara, Broccolini Maurizio, Miozzi Massimo, Pereira Francisco Alves, Lorenzoni Carlo, Mancinelli Alessandro, Tondi Cristina
THREE-PHASE ROUGHNESS EFFECTS ON SUPERHYDROPHOBIC SURFACES FOR TURBULENT DRAG REDUCTION
Golovin Kevin, Gose James W., Perlin Marc, Coccio Steven L., Tuteja Anish

XLVI
FLUID DYNAMICS OF SELF-PROPELLED DROPLET JUMPING ON SUPERHYDROPHOBIC SURFACES
Figueiredo Patric, Wasserfall Joram, Pischke Philipp, Kneer Reinhold
ON THE INTERACTION BETWEEN WAVES AND A MODULAR OSCILLATING WAVE SURGE CONVERTER
Abadie Thomas, Dias Frederic

Fri 11:30-12:50  Instabilities
Room: Affari-3.C
ENERGY-BASED MODEL FOR THE PREDICTION OF MINI-CHANNELS DRY-OUT CONDITION
Giannetti Niccolo, Saito Kyoshi, Yamaguchi Seiichi
TOWARDS A 3D GLOBAL LINEAR STABILITY ANALYSIS FOR BUOYANT FLUID PARTICLES
Kromer Johannes, Bothe Dieter
THE ROLE OF VISCOSITY STRATIFICATION ON THE GLOBAL STABILITY OF TWO-PHASE JETS
Tammisola Outi, Loiseau Jean-Christophe, Brandt Luca

Fri 9:50-11:10  Micro- and Nano-Scale Multiphase Flows
Room: Affari-2.D
NUMERICAL STUDY ON EFFECTS OF ORIENTATION AND GRAVITY ON TWO-PHASE FLOW OF MINICHANNEL OF PEM FUEL CELLS
Moosa Ashrafi, Mehrzad Shams, Bozorgnezhad Ali, Ahmadi Goodarz
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
Olesen Anders Christian, Kaer Soren Knudsen
DYNAMICS OF GAS-LIQUID SEGMENTED FLOW IN POROUS MEDIA: MONOLITHIC OPEN CELL SOLID FOAMS VERSUS DENSE PACKED BED
Serres Marion, Zanotta Marie-Line, Bornette Frederic, Philippe Regis, Vidal Valerie

Fri 11:30-12:50  Reactive Multiphase Flows
Room: Affari-2.A
LES OF A REACTIVE, MULTI-PHASE FLOW IN THE INJECTOR NEAR-FIELD OF AN ENTRAINEMENT-FLOW GASIFIER
INTERACTION OF FLUID DYNAMICS AND CHEMICAL KINETICS IN A THREE-PHASE DEHYDROGENATION REACTION
Heublein Norbert, Sattelmayer Thomas
NUMERICAL STUDY ON DROPLET EVAPORATION AND AUTO-IGNITION OF HYPERGOLIC PROPELLANTS
Tani Hiroumi, Unemura Yutaka, Terashima Hiroshi, Kanno Nozomu, Koshi Mitsuo

Fri 11:30-12:50  Finite size particles/droplets
Room: Affari-2.C
FLOW TOPOLOGY AND ATTRACTORS FOR THE MOTION OF FINITE-SIZE PARTICLES IN A THREE-DIMENSIONAL STEADY CAVITY FLOW
Kuhlmann Hendrik, Romano Francesco, Albergoeder Stefan
TURBULENT CHANNEL FLOW WITH SUSPENDED BI-DISPERSE FINITE SIZE NEUTRALLY BUOYANT SPHERES
Picano Francesco, Lastgari Iman, Costa Pedro, Breugem Wim-Paul, Brandt Luca
DNS OF TURBULENT BOUNDARY LAYER WITH FULLY RESOLVED PARTICLES AT LOW VOLUME FRACTION
Wu Fan, Luo Kun, Fan Jianren
FULLY RESOLVED SIMULATIONS OF SUSPENSION OF SPHERICAL PARTICLES IN SHEAR FLOWS
Rahmani Mona, Wachs Anthony

XLVII
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 Shaiful, 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-Elisabella 140/6

**NUMERICAL STUDY OF MASS TRANSFER OF A RISING BUBBLE WITH BOUNDARY LAYER APPROACH**
Islam Shaiful, 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-Elisabella 140/6

**Boiling, Condensation, Evaporation**

MEASUREMENTS OF LOCAL HEAT TRANSFER IN SUBCOOLED POOL BOILING USING INFRARED THERMOMETRY
Jung Jaehoon, Kim Sung Jin, Kim Jungho, Lee Hyeeseung 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
Mahayekhpour 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
Solvik Jannike, Jakobsen Hugo A. 143/2
Computational Techniques for Multiphase Flows
OPTIMAL FREQUENCY OF ELECTRICAL-IMPEDANCE MEASUREMENT IN THREE-PHASE MIXTURES
Hernandez Cely Marlon, Gomez Bernal Sergio, Velasco Pena Hugo Fernando, Hernandez Rodriguez Oscar Mauricio
REMAINING SWIRLING MOTION ON THE GAS OUTLET DUCT OF CYCLONE SEPARATORS: ALTERNATIVES TO INCREASE EFFICIENCY
Salvo Ricardo, De Souza Francisco Jose, Martins Diego
FINITE VOLUME / FINITE AREA INTERFACE-TRACKING METHOD FOR TWO-PHASE FLOWS WITH SOLUBLE SURFACTANTS AND MASS TRANSFER
Pesoci Chiara, Weber Paul Stephan, Falcone Manuel, Marschall Holger, Bothe Dieter
A CONSERVATIVE LEVEL SET METHOD FOR TWO-PHASE FLOWS
Moghaddam Amin Mahmoudi, Shafieefar Mehdi, Panahi Roozbeh

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

Fluidization
IBD – INTENSIFIED BY DESIGN
Mihajlovic Milan, Roghair Ivo, Van Sint Annaland Martin

FluidStructure Interactions
CHARACTERISTICS OF TURBULENT BOUNDARY LAYER AFFECTED BY A SPHERE NEXT TO A WALL
Zhao Hui, Liu Xiaofei, Li Dong, Wu Fan, Luo Kun, Fan Jianren

Granular Media & Granular flow
THE PROPAGATION OF SHOCK WAVES IN GAS-WATER-SATURATION BULK MEDIA
Zhurov Andrei, Akmetov Alfir, Urmancheev Said
DENSITY WAVES IN THE GRAVITATIONAL FLOW OF GRANULAR MATERIAL IN NARROWPIPES
Alvarez Carlos, De Moraes Franklin Erick

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

Micro and NanoScale Multiphase Flows
LIQUID INFILTRATION CHARACTERISTICS INTO CLOSED END STRUCTURES UNDER THE EXTERNAL PRESSURE
Yamaguchi Er, Nozaki Hiroshi, Sanada Toshiyuki
**ADSORPTION OF ALCOHOLIC ADDITIVES IN WATER/LIBRARY AQUEOUS SOLUTION AT LOW PRESSURE**  
Lonardi Federico, Luke Andrea  
146/4

**Mixing**  
EXPERIMENT STUDY OF THE INSITU DROP SIZE DISTRIBUTIONS AT DIFFERENT SPATIAL REGIONS IN STIRRED TANK  
Zhang Meng, Liu Nannan, Han Jiacheng, Wang Wei, Gong Jing  
146/5

NUMERICAL STUDY ON MIXING CHARACTERISTICS IN SPHERICAL FLASK WITH MAGNETIC STIR BAR  
Lee Sang Hyuk, Cho Wonsam, Kim Dae-Hwan, Lee Taehyun, Ryu Kyungha  
146/6

**Modelling of Multiphase Flows**  
MODELING OF SCATTERING INTENSITY OF MINERAL AEROSOLS WITH A GAUSSIAN BEAM  
Tang Hong, Lin Jianzhong  
147/2

SIMULATION OF TURBULENT LIQUID-LIQUID DISPERSIONS: VALIDATION OF DIFFERENT BREAKAGE AND COALESCENCE KERNELS AND COMPARISON BETWEEN DIFFERENT MODELLING APPROACHES  
De Bona Jeremias, Li Dongyue, Buffo Antonio, Vanni Marco, Podgorska Wioletta, Marchisio Daniele  
147/3

MODELING OF SCATTERING INTENSITY OF MINERAL AEROSOLS WITH A GAUSSIAN BEAM  
Tang Hong, Lin Jianzhong  
147/4

A LATTICE BOLTZMANN MODEL TO SIMULATE LIQUID-LIQUID SLUG FLOW IN 90° BEND  
Jain Shakendra, Ghosh Sunana  
147/5

DETERMINISTIC AND STATISTICAL MODELLING OF PARTICLE-PARTICLE COLLISIONS IN SUPERSONIC GAS-SOLID FLOWS PAST BODIES  
Reviznikov Dmitry, Sposobin Andrey, Varaksa Aleksei  
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  
Gonzalez Eric, Carneiro Joao, Nieckelea Angela  
148/1

A KINETIC WAX DEPOSITION MODEL FOR W/O EMULSION WITH THE INFLUENCE OF CRYSTALLIZED WAX ADHERING TO OIL-WATER INTERFACE  
Song Pan, Wang Wei, Yang Juheng, Gong Jing  
148/2

OPTIMAL FILTER WIDTH FOR CONTINUUM MODELING OF DISCRETE PHASES  
Bateman Samuel, Simeonov Julian, Calantoni Joseph  
148/3

NUMERICAL SIMULATION OF ANNULAR FLOW IN HORIZONTAL PIPES USING THE TWO FLUID MODEL  
Souto Izabel, Nieckelea Angela  
148/4

THE NUMERICAL MODELLING OF MULTIPHASE FLOW INSTABILITIES IN 3D CYLINDRICAL COORDINATES  
Oud Guido  
148/5

MULTIPHASE SIMULATION FOR WATER COOLING TURBINE EXHAUST  
Jiang Minyee, Vinh David  
148/6

**Multiphase Flow in Heat and Mass Transfer**  
A LAGRANGIAN MULTISTAGE SPRAY DRYING MODEL FOR MULTIPHASE CFD APPLICATIONS  
Hriberek Matjaz, Sagadin Gregor, Zadavec Matej, Ravnik Jure  
149/1

THERMOPHORETIC DEPOSITION OF NANOPARTICLES IN A 90° SQUARE BEND TUBE  
Yin Zhaoqin, Dai Zhongping  
149/2

EXPERIMENTAL STUDY AND MECHANISTIC MODELLING OF HEAT TRANSFER IN AIR-WATER SLUG FLOW UNDER COOLING PROCESS  
Wang Xin, Dong Chuanshuai, Zhang Xiaoliang, Yue Xiaoqing, Wang Zhaoting, He Limin  
149/3

**Non-Newtonian Multiphase Flows**  
RHEOLOGICAL EFFECTS ON FLUID INVASION INTO CEMENTED ANNULI  
Zare Marjan, Ward Matthew, Quentin Lindfield Roberts, Kamran Alba, Frigaard Ian  
149/4

A NOVEL VISCO-PLASTICALLY LUBRICATED METHOD FOR CORE-ANNULAR PIPELINE FLOW  
Sarmadi Parisa, Frigaard Ian, Hormoz Sarah  
149/5

**Particle and Drop Dynamics**  
NUMERICAL SIMULATION OF INERTIAL MIGRATION OF ELASTIC PARTICLE IN POISEUILLE FLOWS  
He Xukun, Fan Liang-Liang, Zhe Jiang, Zhao Liang  
149/6

EXPERIMENTAL STUDY ON THE COALESCENCE BEHAVIORS UNDER VARIOUS SURFACTANT CONCENTRATIONS BETWEEN DROP-INTERFACE  
Ma Mengyu, Li Kai, Voulgaropoulos Victor, Jin Hang, Wang Wei, Gong Jing  
150/1
NUMERICAL INVESTIGATION OF THE GROUPING BEHAVIOUR WHEN THE PARTICLES ARE COAXIALLY SETTLING
Nie Deming, Chen Rongqian
150/2

NUMERICAL INVESTIGATION OF HEAT TRANSFER AND FLUID FLOW BEHAVIOR AROUND AN OBLATE SPHEROID
Razgallah Intissar, Ouchene Rafik, Khalij Mohammed, Arcen Boris, Taniere Anne
150/3

ABILITY OF ANSYS FLUENT® V 16.0 TO COMPUTE THE DRAG FORCE ACTING ON NON-SPHERICAL PARTICLES LIKE CYLINDRICAL DISKS
Sierra Del Rio Jorge, Taniere Anne, Khalij Mohammed, Hincapie Diego, Velez Sebastian
150/4

DRIFT AND MIXING BY AIR BUBBLES CROSSING AN INTERFACE OF A STRATIFIED MEDIUM AT MODERATE RE
Diaz Lamberto, Ruiz Angulo Angel, Zenit Roberto
150/5

NUMERICAL ANALYSIS OF MICROBUBBLE DEFORMATION NEAR A HIGH-VISCOSITY LIQUID IN ULTRASOUND FIELD
Mori Kengo, Sugiura Toshihiko
150/6

**Particle Laden Flows**

A PRELIMINARY STUDY ON THE STABILITY OF PARTICLE LADEN JETS THROUGH A FULLY COUPLED CFD-DEM SOLVER
Peters Bernhard, Pozzetti Gabriele
151/1

NANOPARTICLE COAGULATION AND DISPERSION IN A TURBULENT PLANAR JET WITH CONSTRAINTS
Tu Chengxu, Liu Song, Bao Fubing, Yin Zhaoqin
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
Mehrabadi Mohammad, Subramaniam Shankar
151/3

ANALYSIS OF THE EFFECT OF PARTICLE-WALL AND INTER-PARTICLE COLLISION ON EROSION USING OPENFOAM®
Novelletto Ricardo Guilherme A., Sgrott Junior Oscar, Lain Santiago, Sommerfeld Martin
151/4

LARGE-EDDY SIMULATION TO STUDY PARTICLE DEPOSITION IN A CURVED PASSAGE
Sinha Nityanand, Ghodke Chaitanya, Khot Ambarish, Apte Sourabh
151/5

**Bubble dynamics**

SIMULATION OF SINGLE RISING BUBBLES IN A LINEAR SHEAR FLOW WITH THE COLOR-GRADIENT LATTICE BOLTZMANN METHOD
Malli Phillip, Sommerfeld Martin
151/6
<table>
<thead>
<tr>
<th>Name</th>
<th>Pages</th>
</tr>
</thead>
<tbody>
<tr>
<td>Anna Georgantaki</td>
<td>6/3, 6/3</td>
</tr>
<tr>
<td>Luca Gemello</td>
<td>75/5, 75/5</td>
</tr>
<tr>
<td>Alexander Gelfgat</td>
<td>105/3</td>
</tr>
<tr>
<td>Felix Gebauer</td>
<td>43/6, 43/6</td>
</tr>
<tr>
<td>Daniel Gauthier</td>
<td>16/2, 16/2</td>
</tr>
<tr>
<td>Frederike Gauss</td>
<td>118/1</td>
</tr>
<tr>
<td>Jason Gatewood</td>
<td>70/5, 70/5</td>
</tr>
<tr>
<td>Yuhang Gao</td>
<td>20/6, 20/6</td>
</tr>
<tr>
<td>Harish Ganesh</td>
<td>30/1, 47/6, 66/3, 30/1, 47/6</td>
</tr>
<tr>
<td>Jigar Gamit</td>
<td>47/1, 47/1</td>
</tr>
<tr>
<td>Francesco Galleni</td>
<td>5/4, 27/6</td>
</tr>
<tr>
<td>Christoph Gateosato</td>
<td>31/4, 125/6, 31/4, 125/6</td>
</tr>
<tr>
<td>Andrew Fominykh</td>
<td>71/5, 71/5</td>
</tr>
<tr>
<td>Caro Aleix Fonellosa</td>
<td>88/6, 88/6</td>
</tr>
<tr>
<td>Uwe Hampel</td>
<td>11/1, 46/3, 52/4, 80/4</td>
</tr>
<tr>
<td>Ziad Hamidouche</td>
<td>100/5, 100/5</td>
</tr>
<tr>
<td>Ali Haj</td>
<td>62/6, 62/6</td>
</tr>
<tr>
<td>Mohammadreza Haghnegahdar</td>
<td>52/4, 52/4</td>
</tr>
<tr>
<td>Candido Gutierrez Montes</td>
<td>28/1, 28/1</td>
</tr>
<tr>
<td>Kristian Gustavsson</td>
<td>125/2, 125/2</td>
</tr>
<tr>
<td>Laura Gui Nan</td>
<td>118/1, 118/1</td>
</tr>
<tr>
<td>Michele Guala</td>
<td>31/1, 31/1</td>
</tr>
<tr>
<td>Paulo Guattieri</td>
<td>35/1, 71/3, 109/1, 35/1, 71/3, 109/1</td>
</tr>
<tr>
<td>Elisabeth Guazzelli</td>
<td>110/3, 117/3, 124/1, 125/1, 125/1, 124/1, 125/1, 125/1, 124/1, 125/1</td>
</tr>
<tr>
<td>Marie-Laurence Giorgi</td>
<td>70/3, 70/3</td>
</tr>
<tr>
<td>Paolo Gualtieri</td>
<td>35/1, 71/3, 109/1, 35/1, 71/3, 109/1</td>
</tr>
<tr>
<td>Afshin Ghajar</td>
<td>108/5, 108/5</td>
</tr>
<tr>
<td>Sina Ghaidemi</td>
<td>62/6, 62/6</td>
</tr>
<tr>
<td>Sina Ghaidemi</td>
<td>62/6, 62/6</td>
</tr>
<tr>
<td>Athanasios Fraggedakis</td>
<td>128/1, 128/1</td>
</tr>
<tr>
<td>Christine Frances</td>
<td>48/3, 48/3</td>
</tr>
<tr>
<td>Donya Frank</td>
<td>4/5, 4/5</td>
</tr>
<tr>
<td>Edouard Frederex</td>
<td>93/1, 93/1</td>
</tr>
<tr>
<td>Ian Frigaard</td>
<td>128/3, 149/4, 149/5, 128/3, 149/4, 149/5</td>
</tr>
<tr>
<td>Markus Frings</td>
<td>50/2</td>
</tr>
<tr>
<td>Udo Fritsching</td>
<td>57/5, 69/6, 105/1, 108/4, 111/6, 57/5, 69/6, 106/1, 108/4, 111/6</td>
</tr>
<tr>
<td>Christof Frohlich</td>
<td>3/5, 3/6, 29/6, 39/2, 95/5</td>
</tr>
<tr>
<td>Graziano Frunigeri</td>
<td>117/6, 117/6</td>
</tr>
<tr>
<td>Laslo Fuchs</td>
<td>28/6, 29/1, 28/6, 29/1</td>
</tr>
<tr>
<td>Hiroki Fujii</td>
<td>2/5, 113/6, 15/6, 2/5, 113/6, 15/6, 2/5</td>
</tr>
<tr>
<td>Shuhei Fujimoto</td>
<td>77/6, 77/6</td>
</tr>
<tr>
<td>Satoko Fujikawa</td>
<td>142/3</td>
</tr>
<tr>
<td>Shun Fujishima</td>
<td>63/5, 63/5</td>
</tr>
<tr>
<td>Seiji Fujita</td>
<td>119/2, 119/2</td>
</tr>
<tr>
<td>Yoshiki Fukada</td>
<td>87/2, 87/2</td>
</tr>
<tr>
<td>Kota Fukuda</td>
<td>76/2, 136/2, 76/2, 136/2</td>
</tr>
<tr>
<td>Maria-Fernanda Fullana</td>
<td>84/6, 84/6</td>
</tr>
<tr>
<td>John Furian</td>
<td>32/4, 32/4</td>
</tr>
<tr>
<td>Masahiro Furuya</td>
<td>38/2, 38/2</td>
</tr>
<tr>
<td>Daniel Fuster</td>
<td>33/1, 33/1</td>
</tr>
<tr>
<td>Celine Gabillet</td>
<td>55/4, 55/4</td>
</tr>
<tr>
<td>Vinesh Gada</td>
<td>25/1, 108/6, 25/1, 108/6</td>
</tr>
<tr>
<td>Claudio Galbiati</td>
<td>60/5, 60/5</td>
</tr>
<tr>
<td>Cedric Galizzi</td>
<td>142/1, 142/1</td>
</tr>
<tr>
<td>Francesco Galleni</td>
<td>5/4, 27/6</td>
</tr>
<tr>
<td>Chiara Galletti</td>
<td>22/2, 98/1, 108/3, 22/2, 98/1, 108/3</td>
</tr>
<tr>
<td>Jigar Gamit</td>
<td>47/1, 47/1</td>
</tr>
<tr>
<td>Harish Ganesh</td>
<td>30/1, 47/6, 66/3, 30/1, 47/6, 66/3</td>
</tr>
<tr>
<td>Xin Gao</td>
<td>146/1, 146/1</td>
</tr>
<tr>
<td>Hung Gao</td>
<td>20/6, 20/6</td>
</tr>
<tr>
<td>Jason Gatewood</td>
<td>70/5, 70/5</td>
</tr>
<tr>
<td>Frederike Gauss</td>
<td>118/1, 118/1</td>
</tr>
<tr>
<td>Daniel Gauthier</td>
<td>16/2, 16/2</td>
</tr>
<tr>
<td>Manolis Gavaises</td>
<td>2/4, 88/1, 2/4, 88/1</td>
</tr>
<tr>
<td>Liang Ge</td>
<td>83/1, 83/1</td>
</tr>
<tr>
<td>Felix Gebauer</td>
<td>43/6, 43/6</td>
</tr>
<tr>
<td>Alexander Gelfgat</td>
<td>105/3</td>
</tr>
<tr>
<td>Luca Gemello</td>
<td>75/5, 75/5</td>
</tr>
<tr>
<td>Anna Geogantaki</td>
<td>6/3, 6/3</td>
</tr>
<tr>
<td>Gianluca Geraci</td>
<td>27/2, 27/2</td>
</tr>
<tr>
<td>Fabien Gerardin</td>
<td>52/6, 52/6</td>
</tr>
</tbody>
</table>
Han Jiacheng 99/1, 144/2, 146/5, 99/1, 144/2, 146/5
Han Sang-Woo 50/5, 68/1, 50/5, 68/1
Hann David 4/6, 15/5, 28/2, 93/5, 104/2
Hao Chenyu 25/5, 25/5
Harada Shusaku 119/1, 132/1, 119/1, 132/1
Harasawa Noriaki 15/6
Hargreaves David 4/3, 4/3
Haringa Cees 85/2, 86/1, 85/2, 86/1
Haroun Yacine 39/5, 39/5
Hasan Abbas 27/4, 40/6, 59/3
Hasegawa Koji 34/4, 64/6, 34/4, 64/6
Hassan Yassin A. 23/4, 80/5, 23/4, 80/5
Hata Kazuaki 32/6, 32/6
Hattori Yasuhiro 4/2, 4/2
Hayasaka Keisuke 47/5, 47/5
Hayashi Kosuke 26/2, 26/2, 52/5, 55/1, 120/5, 26/2, 26/2, 52/5, 55/1, 120/5
He Limin 149/3, 149/3
Heggs Peter 135/4, 135/4
Hei Ng Chin 30/5, 30/5
Heinz Johannes 6/4, 6/4
Helgans Brian 94/1
Helie Jerome 119/1, 119/1
Hendriks Maurice 5/6, 5/6
Henkes Ruud 26/5, 26/5, 83/6, 129/5, 26/5, 26/5, 83/6, 129/5
Henry De Frahan Marc 66/3, 66/3
Hernandez Cely Marlon 143/3, 143/3
Hernandez Freddy 44/6, 44/6
Hernandez Gonzalez Oscar 5/5, 5/5, 18/3, 81/4, 86/5, 112/5, 112/5, 143/3, 145/4
Herrmann Heber Robert 61/2, 61/2
Herrmann Marcus 94/3, 94/3
Heubel Norbert 34/1, 34/1, 34/1, 34/1, 34/1, 34/1, 34/1, 34/1, 34/1, 34/1, 34/1
Hewakandamby Buddhika 4/6, 4/6, 4/6, 4/6, 4/6, 4/6, 4/6, 4/6, 4/6, 4/6, 4/6, 4/6, 4/6
Hewitt Geoffrey 16/4, 16/4
Heylmun Jeffrey 85/4, 85/4
Heyward A.M. 134/3, 134/3, 134/3, 134/3, 134/3, 134/3
Heywood John 4/6, 4/6, 4/6, 4/6, 4/6, 4/6, 4/6, 4/6, 4/6, 4/6, 4/6, 4/6, 4/6, 4/6, 4/6, 4/6, 4/6, 4/6, 4/6, 4/6, 4/6, 4/6, 4/6, 4/6
Hewitt Geoffrey 16/4, 16/4
Heylmun Jeffrey 85/4, 85/4
Heyward A.M. 134/3, 134/3, 134/3, 134/3, 134/3
Heywood John 4/6, 4/6, 4/6, 4/6, 4/6, 4/6, 4/6, 4/6, 4/6, 4/6, 4/6, 4/6, 4/6, 4/6, 4/6, 4/6, 4/6, 4/6, 4/6, 4/6, 4/6, 4/6, 4/6, 4/6
<table>
<thead>
<tr>
<th>Name</th>
<th>Pages</th>
</tr>
</thead>
<tbody>
<tr>
<td>Lain Santiago</td>
<td>109/4, 151/4, 109/4, 151/4</td>
</tr>
<tr>
<td>Lakehal Djamel</td>
<td>16/4, 42/4, 72/1, 105/1</td>
</tr>
<tr>
<td>Lakshmanan Susithra</td>
<td>14/4, 14/4</td>
</tr>
<tr>
<td>Lalanne Benjamin</td>
<td>36/4, 54/6, 36/4, 54/6</td>
</tr>
<tr>
<td>Larmargue Nicolas</td>
<td>90/1, 90/1</td>
</tr>
<tr>
<td>Lamborgese Andrea</td>
<td>22/2, 75/1, 22/2, 75/1</td>
</tr>
<tr>
<td>Lampja Aljoscha</td>
<td>111/6, 111/6</td>
</tr>
<tr>
<td>Lance Michel</td>
<td>20/3, 39/5, 107/1, 112/1</td>
</tr>
<tr>
<td>Lancien Theo</td>
<td>102/5, 102/5</td>
</tr>
<tr>
<td>Laneryd Tor</td>
<td>88/6, 88/6</td>
</tr>
<tr>
<td>Langsholt Morten</td>
<td>59/2, 59/2</td>
</tr>
<tr>
<td>Lanotte Alessandra Sabina</td>
<td>53/4, 53/4</td>
</tr>
<tr>
<td>Lao Liyun</td>
<td>36/2, 50/6, 30/2, 50/6</td>
</tr>
<tr>
<td>Larat Adam</td>
<td>60/2, 68/5, 87/4, 60/2, 66/5, 68/5, 78/4</td>
</tr>
<tr>
<td>Lauk Peep</td>
<td>69/3, 69/3</td>
</tr>
<tr>
<td>Laupsien David</td>
<td>76/1</td>
</tr>
<tr>
<td>Laurent Frederique</td>
<td>13/1, 66/5, 107/6, 107/6, 106/5, 107/6</td>
</tr>
<tr>
<td>Lay Ephraim Nemati</td>
<td>141/4, 142/2, 141/4, 142/2</td>
</tr>
<tr>
<td>Layssac Thibaut</td>
<td>56/3, 56/3</td>
</tr>
<tr>
<td>La Bot Cedric</td>
<td>133/2, 133/2</td>
</tr>
<tr>
<td>Le Clercq Patrick</td>
<td>139/1, 139/1</td>
</tr>
<tr>
<td>Le Men Claude</td>
<td>76/1</td>
</tr>
<tr>
<td>Le Merrer Marie</td>
<td>15/4, 15/4</td>
</tr>
<tr>
<td>Le Sauze Nathalie</td>
<td>18/1, 18/1</td>
</tr>
<tr>
<td>Le Touze Clement</td>
<td>13/1, 13/1</td>
</tr>
<tr>
<td>Lebon Michel</td>
<td>19/1, 19/1</td>
</tr>
<tr>
<td>Lebrun Pierre</td>
<td>50/3, 50/3</td>
</tr>
<tr>
<td>Lee Hyeseung</td>
<td>141/6, 141/6</td>
</tr>
<tr>
<td>Lee Inwon</td>
<td>120/3, 120/3</td>
</tr>
<tr>
<td>Lee Joo Seong</td>
<td>52/3, 52/3</td>
</tr>
<tr>
<td>Lee Joohwa Sarah</td>
<td>5/1, 5/1</td>
</tr>
<tr>
<td>Lee Juhn</td>
<td>24/3</td>
</tr>
<tr>
<td>Lee Jun Ho</td>
<td>19/6, 19/6</td>
</tr>
<tr>
<td>Lee Keng Lin</td>
<td>73/5, 73/5</td>
</tr>
<tr>
<td>Lee Kunchul</td>
<td>30/2, 30/2</td>
</tr>
<tr>
<td>Lee Sang Hyuk</td>
<td>146/6, 146/6</td>
</tr>
<tr>
<td>Lee Sang-Beom</td>
<td>92/3, 92/3</td>
</tr>
<tr>
<td>Lee Taehyun</td>
<td>146/6, 146/6</td>
</tr>
<tr>
<td>Lefebvre Leopot Aline</td>
<td>58/2, 58/3, 58/2, 58/3</td>
</tr>
<tr>
<td>Legendre Dominique</td>
<td>371/5, 552/6, 127/4, 371/5, 552/6, 127/4</td>
</tr>
<tr>
<td>Leisner Thomas</td>
<td>76/6, 76/6</td>
</tr>
<tr>
<td>Lemoine Fabrice</td>
<td>70/2, 70/2</td>
</tr>
<tr>
<td>Lenz Stefan</td>
<td>83/2, 83/2</td>
</tr>
<tr>
<td>Lepilliez Mathieu</td>
<td>39/6, 39/6</td>
</tr>
<tr>
<td>Lepoutre Guillaume</td>
<td>17/5, 17/5</td>
</tr>
<tr>
<td>Lester Edward</td>
<td>84/4, 84/4</td>
</tr>
<tr>
<td>Letan Ruth</td>
<td>79/6, 79/6</td>
</tr>
<tr>
<td>Lettieri Paola</td>
<td>129/1, 129/1</td>
</tr>
<tr>
<td>Levy Avi</td>
<td>108/1</td>
</tr>
<tr>
<td>Li Cheng</td>
<td>111/4, 111/4</td>
</tr>
<tr>
<td>Li Dong</td>
<td>101/3, 145/1, 101/3, 145/1</td>
</tr>
<tr>
<td>Li Dong</td>
<td>1/2, 1/2</td>
</tr>
<tr>
<td>Li Dongyue</td>
<td>147/3</td>
</tr>
<tr>
<td>Li Hang</td>
<td>43/3, 43/3</td>
</tr>
<tr>
<td>Li Hui-Xiong</td>
<td>112/2, 172/2, 26/4, 34/3, 34/3</td>
</tr>
<tr>
<td>Li Jason</td>
<td>2/4, 2/4</td>
</tr>
<tr>
<td>Li Jing</td>
<td>44/4, 100/1, 124/6, 134/5</td>
</tr>
<tr>
<td>Li Kai</td>
<td>11/2, 172/2, 26/4, 34/3, 34/3, 150/1, 112/2, 172/2, 26/4, 34/3, 150/1</td>
</tr>
<tr>
<td>Li Ke</td>
<td>114/6, 114/6</td>
</tr>
<tr>
<td>Author Name</td>
<td>Page Numbers</td>
</tr>
<tr>
<td>------------------------------</td>
<td>--------------</td>
</tr>
<tr>
<td>Nakagawa Hajime</td>
<td>42/3, 42/3</td>
</tr>
<tr>
<td>Nakagawa Kazuyuki</td>
<td>84/2, 84/2</td>
</tr>
<tr>
<td>Nakamura Shiko</td>
<td>59/4, 59/4</td>
</tr>
<tr>
<td>Nakaniishi Siroki</td>
<td>10/5, 10/5</td>
</tr>
<tr>
<td>Narayanan Chidambaram</td>
<td>16/4, 42/4, 58/4, 72/1</td>
</tr>
<tr>
<td>Narayanan Ranga</td>
<td>98/4, 98/4</td>
</tr>
<tr>
<td>Naso Aurore</td>
<td>1/6, 87/6, 94/2, 1/6, 87/6, 94/2</td>
</tr>
<tr>
<td>Nathan Graham</td>
<td>58/1, 58/1</td>
</tr>
<tr>
<td>Nayak Ameesha Kumar</td>
<td>133/4, 133/4</td>
</tr>
<tr>
<td>Neal Christopher R.</td>
<td>71/6, 71/6</td>
</tr>
<tr>
<td>Neau Herve</td>
<td>90/3, 90/3</td>
</tr>
<tr>
<td>Nemes Andras</td>
<td>39/3, 39/3</td>
</tr>
<tr>
<td>Nennie Erik</td>
<td>35/5, 119/6</td>
</tr>
<tr>
<td>Ngo Long</td>
<td>31/5</td>
</tr>
<tr>
<td>Ohta Mitsuhiro</td>
<td>134/1, 134/1</td>
</tr>
<tr>
<td>Onishi Ryo</td>
<td>53/1, 101/5, 53/1, 101/5</td>
</tr>
<tr>
<td>Onorato Miguel</td>
<td>84/3, 84/3</td>
</tr>
<tr>
<td>Onuki Hajime</td>
<td>42/1, 42/1</td>
</tr>
<tr>
<td>Omura Gismo</td>
<td>83/6, 83/6</td>
</tr>
<tr>
<td>Ortiz Vidal Luis Enrique</td>
<td>86/5, 86/5</td>
</tr>
<tr>
<td>Oshitanu Jun</td>
<td>119/1, 119/1</td>
</tr>
<tr>
<td>Ota Kazuyuki</td>
<td>42/3, 42/3</td>
</tr>
<tr>
<td>Otomo Ryoko</td>
<td>25/2, 25/2, 25/2, 25/2, 25/2</td>
</tr>
<tr>
<td>Ouchene Rafik</td>
<td>116/2, 116/2, 116/2, 116/2</td>
</tr>
<tr>
<td>Ouchi Koki</td>
<td>119/2, 119/2</td>
</tr>
<tr>
<td>Oud Guido</td>
<td>148/5, 148/5</td>
</tr>
<tr>
<td>OuULLET Frederick</td>
<td>121/2, 121/2</td>
</tr>
<tr>
<td>Oumerswift Edwin</td>
<td>79/5, 79/5</td>
</tr>
<tr>
<td>Ozaki Tsubasa</td>
<td>64/3, 64/3</td>
</tr>
<tr>
<td>Ozbay Ridvan</td>
<td>46/1, 46/1</td>
</tr>
<tr>
<td>Ozel Ali</td>
<td>126/4, 126/4</td>
</tr>
<tr>
<td>Pachia Sanjeevi Sathish</td>
<td>48/6, 48/6</td>
</tr>
<tr>
<td>Paddding Johan</td>
<td>3/2, 48/6, 77/4, 118/5, 3/2, 48/6, 77/4, 118/5</td>
</tr>
<tr>
<td>Pagliantii Alessandro</td>
<td>85/3, 85/3</td>
</tr>
<tr>
<td>Pain Christopher</td>
<td>22/1, 113/5, 22/1, 113/5</td>
</tr>
<tr>
<td>Pakhomov Maksim</td>
<td>16/3, 16/3</td>
</tr>
<tr>
<td>Paladino Emilio</td>
<td>9/4, 9/4</td>
</tr>
<tr>
<td>Pallares Jordi</td>
<td>108/5, 108/5</td>
</tr>
<tr>
<td>Palmon John</td>
<td>7/3, 7/3</td>
</tr>
<tr>
<td>Pan Hui</td>
<td>33/6, 33/6, 33/6, 33/6, 33/6, 33/6</td>
</tr>
<tr>
<td>Pan Liangming</td>
<td>28/3, 28/3</td>
</tr>
<tr>
<td>Pan Shucheng</td>
<td>49/2, 49/2</td>
</tr>
<tr>
<td>Panahi Roodbeh</td>
<td>143/6, 143/6</td>
</tr>
<tr>
<td>Pang Mingjun</td>
<td>141/2, 141/2</td>
</tr>
<tr>
<td>Paolii Roberto</td>
<td>4/3, 4/3</td>
</tr>
<tr>
<td>Papavassiliou Dimitrios</td>
<td>110/2, 110/2</td>
</tr>
<tr>
<td>Papoulis Dimitrios</td>
<td>122/5, 122/5</td>
</tr>
<tr>
<td>Paras Spisores</td>
<td>78/1, 140/5, 78/1, 140/5</td>
</tr>
<tr>
<td>Parente Alessandro</td>
<td>108/3, 108/3</td>
</tr>
<tr>
<td>Park Chan Woo</td>
<td>29/4, 29/4</td>
</tr>
<tr>
<td>Park Chansoo</td>
<td>24/3, 24/3</td>
</tr>
<tr>
<td>Park Hyun Jin</td>
<td>88/3, 88/3</td>
</tr>
<tr>
<td>Park Hyungmin</td>
<td>19/6, 19/6</td>
</tr>
<tr>
<td>Park Kyeong Hyeon</td>
<td>24/6, 24/6</td>
</tr>
<tr>
<td>Park Seong Hyeon</td>
<td>120/3, 120/3</td>
</tr>
<tr>
<td>Passalaqueta Alberto</td>
<td>75/4, 75/4, 75/4, 75/4, 75/4</td>
</tr>
<tr>
<td>Passos Agathoklis</td>
<td>78/1, 140/5, 78/1, 140/5</td>
</tr>
<tr>
<td>Pasternaks Lars</td>
<td>130/6, 130/6</td>
</tr>
<tr>
<td>Patnak Ashish</td>
<td>111/2, 111/2</td>
</tr>
<tr>
<td>Pauchard Vincent</td>
<td>68/6, 68/6</td>
</tr>
<tr>
<td>Pavlidi Dimitrios</td>
<td>22/1, 113/5, 22/1, 113/5</td>
</tr>
<tr>
<td>Pelanti Marica</td>
<td>31/2, 31/2</td>
</tr>
<tr>
<td>Pelin Kalle</td>
<td>37/3, 37/3</td>
</tr>
<tr>
<td>Pellegrini Laura</td>
<td>81/6, 81/6</td>
</tr>
<tr>
<td>Pena Monferrer Carlos</td>
<td>72/3, 72/3</td>
</tr>
<tr>
<td>Peng Cheng</td>
<td>35/3, 35/3, 35/3, 35/3, 35/3</td>
</tr>
<tr>
<td>Penico A.</td>
<td>54/1, 54/1</td>
</tr>
<tr>
<td>Pereira Francisco Alves</td>
<td>59/1, 137/3, 59/1, 137/3</td>
</tr>
<tr>
<td>Perkins Richard</td>
<td>14/6, 14/6</td>
</tr>
<tr>
<td>Peres Carlos Eduardo</td>
<td>74/3, 74/3</td>
</tr>
<tr>
<td>Perin Marc</td>
<td>64/5, 64/5, 64/5, 64/5, 64/5</td>
</tr>
<tr>
<td>Perrier Hugo</td>
<td>13/6, 13/6</td>
</tr>
<tr>
<td>Pervunin Konstantin</td>
<td>11/6, 80/3, 11/6, 80/3</td>
</tr>
<tr>
<td>Pesci Chiara</td>
<td>6/5, 60/6, 143/5, 6/5, 60/6, 143/5</td>
</tr>
<tr>
<td>Peters Bernhard</td>
<td>111/1, 151/1</td>
</tr>
<tr>
<td>Peters E.A.J.F.</td>
<td>7/1, 7/1</td>
</tr>
<tr>
<td>Peters Frank</td>
<td>77/4, 118/5, 77/4, 118/5</td>
</tr>
<tr>
<td>Petersen Alec</td>
<td>35/4, 35/4</td>
</tr>
<tr>
<td>Pettas Dioniosos</td>
<td>127/3, 127/3</td>
</tr>
<tr>
<td>Pfister Michael</td>
<td>47/4, 47/4</td>
</tr>
<tr>
<td>Philippe Regis</td>
<td>121/5, 121/5, 121/5, 121/5, 121/5</td>
</tr>
<tr>
<td>Phillips Bren</td>
<td>37/6, 37/6</td>
</tr>
<tr>
<td>Pialago Edward Joshua</td>
<td>29/4, 29/4</td>
</tr>
<tr>
<td>Picchi Davide</td>
<td>97/6, 97/6</td>
</tr>
<tr>
<td>Piccinini Leopard Giacomo</td>
<td>90/1, 90/1</td>
</tr>
<tr>
<td>Piedfert Antoine</td>
<td>54/6, 54/6</td>
</tr>
</tbody>
</table>
Pigeonneau Franck 22/3, 58/2, 118/2, 22/3, 58/2, 118/2
Pilai Abhishek 101/1, 101/1
Piomelli 71/1, 71/1
Pirker Stefan 22/4, 60/4, 67/4, 93/2
Piscaglia Federico 110/6, 110/6
Pischke Philipp 39/2, 137/5, 39/2, 137/5
Pitz-Paal Robert 42/2, 42/2
Pivello Marcio 55/3, 55/3
Pias Cecile 75/5, 75/5
Planckette Carole 2/6, 2/6
Planquart Philippe 120/4, 120/4
Plesniak Michael W. 28/6, 28/6
Plouraboue Franck 109/3, 109/3
Podgorska Wioletta 147/3
Poelma Christian 31/6
Poesio Pietro 8/6, 9/2, 54/3, 97/6, 8/6, 9/2, 54/3, 97/6
Polifke Wolfgang 5/1, 137/1, 5/1, 137/1
Pons Audrey 133/2, 133/2
Pontes Jose 99/6, 99/6
Popinet Stephane 84/6, 84/6
Portela Luis 28/5, 28/5, 85/1, 26/5, 62/5, 85/1
Postacchini Matteo 4/4, 4/4
Potapov Alexander 82/4, 82/4
Potts Ian 27/1, 27/1
Pouliquen Olivier 124/1, 124/1
Pouplin Amelie 117/5, 117/5
Pourique Mathieu 83/6, 83/6
Povolny Antonin 82/1, 82/1
Powell Robert 130/4, 130/4
Pozorski Jacek 76/5, 76/5, 76/5, 76/5, 76/5, 76/5
Pozzetti Gabriele 111/1, 111/1
Pradhan Aniruddha 141/5, 141/5
Prahl Wittberg Lisa 28/6, 29/1, 103/4, 28/6, 29/1, 103/4
Prasetya Rubby 21/1, 21/1
Premnath Kannan 88/5, 88/5
Priem Giel 103/1, 103/1
Profeta Christophe 65/6, 65/6
Prosperetti Andrea 135/5, 135/5
Protasov Mikhail 107/3, 107/3
Pujara Nirmish 94/5, 94/5
Pulido Jorge 44/6, 44/6
Punde Pranav 108/6, 108/6
Punekar Hemant 83/4, 83/4, 83/4, 115/5
Purwono Baskoro Adiputro 91/1, 91/1
Qian Lijuan 96/2, 96/2
Qin Chao 43/2, 43/2
Qiu Guizhi 132/6, 132/6
Qu Chen 19/2, 19/2
Qu Zhiguo 101/2, 101/2
Quan Qing 56/4, 56/4
Quentin Lindfield Roberts 149/4, 149/4
Quetschner Daniel 67/4, 67/4
Quintero Brian 109/4, 109/4
Rachner Michael 139/1, 139/1
Radcliff Thomas 56/5, 56/5
Rad Stefan 101/4, 101/4
Raessi Mehdi 111/2, 111/2
Rahmani Mona 27/2, 140/1, 27/2, 140/1
Rai Kenta 119/1, 119/1
Raimundo Pedro Maximiano 19/5, 19/5, 19/5, 19/5, 75/5
Rameche Thomas 47/4, 47/4
Ramini Imani Jajarmi 133/1, 133/1
Ramoeno Luca 42/4, 42/4
Rana Basanta Kumar 15/6
Rapberger Richard 64/2
Rarrek Andreas 41/3, 41/3
Rasimone Andris 67/5
Rasteiro Maria Graca 116/3, 116/3
Ravnik Jure 149/1
Razgallah Intissar 150/3, 150/3
Reeks Michael 27/1, 27/1
Reinecke Sebastian 82/3, 82/3
Ren Xiaoxin 103/5, 103/5
Revellon Julien 99/1, 136/1, 99/1, 136/1
Revellon Remi 9/5, 9/5, 9/5, 9/5
Revzinkov Dmitry 147/6, 147/6
Rezig Mariem 54/5, 54/5
Ribatski Gherhardt 41/1, 43/1
Ribeiro Duarte Carlos Antonio 6/1
Riber Eleonore 79/1, 79/1
Richter Andrey 47/3, 47/3
Richter David 94/1
Richter Steffen 68/5, 68/5
Riedelbauch Stefan 21/2, 21/2
Rigopoulos Stelios 129/3, 129/3
Rimbert Nicolas 32/3, 43/5, 52/6, 98/5, 32/3, 43/5, 52/6, 98/5
Rinke Gunter 23/3
Riiboo Romain 65/2, 65/2
Risner Frederic 36/4, 54/6, 68/6, 36/4, 54/6, 68/6
Robbes Benjamin 79/1, 79/1
Roccon Alessio 113/3, 113/3
Rocha Douglas Martins 18/3, 18/3, 18/3, 18/3, 12/5
Rodrigues Romulo 67/1, 67/1
Roerby Johan 103/2, 103/2
Rogachevski Igor 79/2, 79/2
Roghair Ivo 3/3, 132/4, 144/3, 3/3, 132/4, 144/3
Roig Veronique 101/1, 101/1, 101/1
Roisman Ilia V. 7/6, 8/1, 17/1, 21/4, 7/6, 8/1, 17/1, 21/4
Rollin Bertrand 121/2, 121/2
Roma Alexandre 55/3, 55/3
Romano Francesco 131/2, 131/2, 131/2, 131/2, 131/2
Romano Giampaolo 59/1, 59/1
Romper Carsten 146/1, 146/1
Rong Li Ji 31/3, 31/3
Ropelato Karolline 82/4, 82/4
Rosa Bogdan 76/5, 76/5, 76/5, 76/5, 101/5
Rosen Tomas 86/4, 86/4
Rosendahl Lasse 102/3, 102/3
Roshchupka Olga 32/1, 46/5, 32/1, 46/5
Rosseburg Annika 75/6, 75/6
Rossa Michele 122/6, 122/6
Roumpa Evangelia 124/1, 124/1
Roustaie Ali 128/3, 128/3
Roy Shantanu 32/2, 32/2
Rubner Michael 37/6, 37/6
Rudek Arthur 93/4
Rudiger Frank 29/6, 29/6
Rudolf von Rohr Philipp 40/4, 40/4
Ruegg Oliver 40/4, 40/4
Ruiz Angel Angel 150/5, 150/5
Ruiz-Rus Javier 28/1, 28/1
Rulliere Romuald 65/1, 65/1
Russ Gerald 93/4
Russo Maria Elena 114/3, 114/3
Ruyer Pierre 29/2, 82/5, 29/2, 82/5
Ryu Kyungha 146/6, 146/6
Rzehak Roland 51/4, 123/5, 51/4, 123/5
Sabat Macrole 87/4, 87/4
Sabban Lilach 95/1, 95/1
Sabino Renzo 145/6, 145/6
Saccone Giacomo 80/1, 80/1
Sadatomi Michio 91/5, 91/5
Sadilo Filip 134/2, 137/2, 134/2, 137/2
Saeidipour Mahdi 60/4, 60/4
Saen Tom 105/5, 105/5
Saenger Alexander 139/1, 139/1
Saffar Avval Majid 62/6, 62/6
Sagari Manico 59/4, 59/4
Sagadin Gregor 149/1
Saha Debashish 64/1, 64/1
Sahu Akhilesh Kumar 126/1, 126/1

ICMF 2016 International Conference on Multiphase Flow
Firenze, Italy, May 22 - 27, 2016
<table>
<thead>
<tr>
<th>Name</th>
<th>Page Numbers</th>
</tr>
</thead>
<tbody>
<tr>
<td>Yamamoto Shota</td>
<td>47/5, 47/5</td>
</tr>
<tr>
<td>Yamamoto Yasufumi</td>
<td>25/2, 25/2</td>
</tr>
<tr>
<td>Yamashita Masatoshi</td>
<td>84/2, 84/2</td>
</tr>
<tr>
<td>Yamazaki Isako</td>
<td>142/3</td>
</tr>
<tr>
<td>Yan Jianguo</td>
<td>33/6, 33/6</td>
</tr>
<tr>
<td>Yan Junjie</td>
<td>2/1, 77/1, 2/1, 77/1</td>
</tr>
<tr>
<td>Yan Qing</td>
<td>127/5, 127/5</td>
</tr>
<tr>
<td>Yanai Sayuri</td>
<td>30/3</td>
</tr>
<tr>
<td>Yang Eunjin</td>
<td>121/1, 121/1</td>
</tr>
<tr>
<td>Yang Fuling</td>
<td>73/5, 74/1, 76/3, 97/2, 73/5, 74/1, 76/3, 97/2</td>
</tr>
<tr>
<td>Yang Guang</td>
<td>144/1, 144/1</td>
</tr>
<tr>
<td>Yang Hongkun</td>
<td>39/2, 39/2</td>
</tr>
<tr>
<td>Yang Juheng</td>
<td>56/4, 148/2, 56/4, 148/2</td>
</tr>
<tr>
<td>Yang Junfeng</td>
<td>16/4, 16/4</td>
</tr>
<tr>
<td>Yang Wei</td>
<td>116/1</td>
</tr>
<tr>
<td>Yang Wuqiang</td>
<td>132/6, 132/6</td>
</tr>
<tr>
<td>Yang Xiaogang</td>
<td>42/5, 42/5</td>
</tr>
<tr>
<td>Yang Xuegeng</td>
<td>48/2, 48/2</td>
</tr>
<tr>
<td>Yang Yan</td>
<td>4/2, 83/3, 4/2, 83/3</td>
</tr>
<tr>
<td>Yang Zhiquiang</td>
<td>122/4, 122/4</td>
</tr>
<tr>
<td>Yang Zonglin</td>
<td>20/2, 20/2</td>
</tr>
<tr>
<td>Yao Chao</td>
<td>25/5, 25/5</td>
</tr>
<tr>
<td>Yao Jun</td>
<td>26/6, 68/3, 26/6, 68/3</td>
</tr>
<tr>
<td>Yasukawa Yoshito</td>
<td>15/1</td>
</tr>
<tr>
<td>Yates Matthew</td>
<td>93/5, 93/5</td>
</tr>
<tr>
<td>Yazdani Md</td>
<td>56/5, 56/5</td>
</tr>
<tr>
<td>Ye Jiamin</td>
<td>132/6, 132/6</td>
</tr>
<tr>
<td>Yeo Kyongmin</td>
<td>132/2, 132/2</td>
</tr>
<tr>
<td>Yeung Ho</td>
<td>36/2, 50/6, 36/2, 50/6, 67/2</td>
</tr>
<tr>
<td>Ye Wenchao</td>
<td>116/5</td>
</tr>
<tr>
<td>Yu Yong</td>
<td>19/2, 19/2</td>
</tr>
<tr>
<td>Yu Zi-Tao</td>
<td>29/3, 112/6, 29/3, 112/6</td>
</tr>
<tr>
<td>Yuan Ghizhen</td>
<td>33/6, 33/6</td>
</tr>
<tr>
<td>Yuan Ye</td>
<td>81/5, 81/5</td>
</tr>
<tr>
<td>Yue Xiaocing</td>
<td>149/3, 149/3</td>
</tr>
<tr>
<td>Yun Byongjo</td>
<td>45/3, 45/3</td>
</tr>
<tr>
<td>Yuto Oi</td>
<td>42/1, 42/1</td>
</tr>
<tr>
<td>Zaaboub Abdelghafour</td>
<td>64/6, 64/4</td>
</tr>
<tr>
<td>Zabarankin Michael</td>
<td>34/6</td>
</tr>
<tr>
<td>Zabiego Magali</td>
<td>125/6, 125/6</td>
</tr>
<tr>
<td>Zabkova Katerina</td>
<td>63/1, 63/1</td>
</tr>
<tr>
<td>Zadravec Matej</td>
<td>149/1</td>
</tr>
<tr>
<td>Zaki Tamer</td>
<td>17/3, 17/3</td>
</tr>
<tr>
<td>Zaleski Stephane</td>
<td>10/6, 33/1, 10/6, 33/1</td>
</tr>
<tr>
<td>Zamansky Remi</td>
<td>129/4, 129/4</td>
</tr>
<tr>
<td>Zander Hans-Jorg</td>
<td>83/2, 83/2</td>
</tr>
<tr>
<td>Zangana Mohammed</td>
<td>27/5</td>
</tr>
<tr>
<td>Zanotta Marie-Line</td>
<td>121/5, 138/6, 121/5, 138/6, 121/5, 138/6, 121/5, 138/6</td>
</tr>
<tr>
<td>Zare Marjan</td>
<td>128/3, 149/4, 128/3, 149/4, 128/3, 149/4</td>
</tr>
<tr>
<td>Zarghami Ahad</td>
<td>86/1, 86/1</td>
</tr>
<tr>
<td>Zeglhol Ammar</td>
<td>49/5</td>
</tr>
<tr>
<td>Zegarnik Yurti</td>
<td>20/4, 20/4</td>
</tr>
<tr>
<td>Zeng Zhuxiong</td>
<td>114/6, 114/6</td>
</tr>
<tr>
<td>Zenit Roberto</td>
<td>55/2, 150/5, 55/2, 150/5</td>
</tr>
<tr>
<td>Zha Lu</td>
<td>124/6, 124/6</td>
</tr>
<tr>
<td>Zhekaevy Dauren</td>
<td>99/2, 99/2</td>
</tr>
<tr>
<td>Zhang Baill</td>
<td>30/4, 110/4, 30/4, 110/4, 30/4, 110/4</td>
</tr>
<tr>
<td>Zhang Gangfu</td>
<td>95/3</td>
</tr>
<tr>
<td>Zhang Huiqiang</td>
<td>103/5, 103/5</td>
</tr>
<tr>
<td>Zhang Jian</td>
<td>19/2, 19/2</td>
</tr>
<tr>
<td>Zhang Liang</td>
<td>29/3, 112/6, 29/3, 112/6</td>
</tr>
<tr>
<td>Zhang Y.</td>
<td>34/2, 34/2</td>
</tr>
<tr>
<td>Wenderlich Stephan</td>
<td>143/5, 143/5</td>
</tr>
<tr>
<td>Wess Windl</td>
<td>11/4, 11/4</td>
</tr>
<tr>
<td>Wielhui Hashi Wielhui</td>
<td>26/1, 26/1</td>
</tr>
<tr>
<td>Wei Anyang</td>
<td>126/2, 126/6</td>
</tr>
<tr>
<td>Wei Jinja</td>
<td>141/2, 141/2</td>
</tr>
<tr>
<td>Weickgenannt Christina</td>
<td>21/4, 21/4</td>
</tr>
<tr>
<td>Weigand Bernhard</td>
<td>60/5, 134/2, 137/2, 60/5, 134/2, 137/2</td>
</tr>
<tr>
<td>Welch Samuel</td>
<td>88/5, 88/5</td>
</tr>
<tr>
<td>Wen Chuanng</td>
<td>4/2, 83/3, 4/2, 83/3</td>
</tr>
<tr>
<td>Westendes Jos</td>
<td>35/5, 35/5</td>
</tr>
<tr>
<td>Westerweer Jerry</td>
<td>79/5, 79/5</td>
</tr>
<tr>
<td>Willen D.</td>
<td>135/5, 135/5</td>
</tr>
<tr>
<td>Wilson Robert</td>
<td>131/1, 131/1</td>
</tr>
<tr>
<td>Wiratini Wiratini</td>
<td>91/1, 91/1</td>
</tr>
<tr>
<td>Wittgens Bernd</td>
<td>73/1, 73/1</td>
</tr>
<tr>
<td>Woerner Martin</td>
<td>63/4, 68/4, 63/4, 68/4</td>
</tr>
<tr>
<td>Wollborn Tobias</td>
<td>57/5, 57/5</td>
</tr>
<tr>
<td>Wolters Jorg</td>
<td>39/2, 39/2</td>
</tr>
<tr>
<td>Wrobel Luiz</td>
<td>122/3, 122/3</td>
</tr>
<tr>
<td>Wu Dan</td>
<td>100/1, 100/1</td>
</tr>
<tr>
<td>Wu Fan</td>
<td>139/6, 139/6</td>
</tr>
<tr>
<td>Wu Fan</td>
<td>145/1, 145/1</td>
</tr>
<tr>
<td>Wu Haihao</td>
<td>123/3, 123/3</td>
</tr>
<tr>
<td>Wu Hao</td>
<td>67/3, 144/3, 67/3, 144/3</td>
</tr>
<tr>
<td>Wu Jiandong</td>
<td>17/4, 17/4</td>
</tr>
<tr>
<td>Wu Juliana</td>
<td>30/1, 47/6, 30/1, 47/6</td>
</tr>
<tr>
<td>Wu Ke</td>
<td>11/2, 11/2</td>
</tr>
<tr>
<td>Wu Wangxia</td>
<td>70/1, 70/1</td>
</tr>
<tr>
<td>Wu Wen</td>
<td>71/1, 71/1</td>
</tr>
<tr>
<td>Wu Wengjuan</td>
<td>1/2, 1/2</td>
</tr>
<tr>
<td>Wu Zan</td>
<td>47/2, 47/2</td>
</tr>
<tr>
<td>Wu Zhipeng</td>
<td>132/6, 132/6</td>
</tr>
<tr>
<td>Wucherplennig Thomas</td>
<td>75/6, 75/6</td>
</tr>
<tr>
<td>Xayaseth Arunvady</td>
<td>78/3, 78/3</td>
</tr>
<tr>
<td>Xia Guangong</td>
<td>144/1, 144/1</td>
</tr>
<tr>
<td>Xia Jun</td>
<td>122/3, 122/3</td>
</tr>
<tr>
<td>Xia Junjie</td>
<td>135/6, 135/6</td>
</tr>
<tr>
<td>Xia Kai</td>
<td>2/1, 2/1</td>
</tr>
<tr>
<td>Xiang Min</td>
<td>123/4, 123/4</td>
</tr>
<tr>
<td>Xiang Sujian</td>
<td>140/4, 140/4</td>
</tr>
<tr>
<td>Xie Zhihua</td>
<td>221/1, 113/5, 221/1, 113/5</td>
</tr>
<tr>
<td>Xiong Yan</td>
<td>44/4, 44/4</td>
</tr>
<tr>
<td>Xu Wei</td>
<td>61/6, 61/6</td>
</tr>
<tr>
<td>Xu Wenkai</td>
<td>40/1, 40/1</td>
</tr>
<tr>
<td>Xue Xinbei</td>
<td>23/2, 23/2</td>
</tr>
<tr>
<td>Xue Yuling</td>
<td>25/5, 25/5</td>
</tr>
<tr>
<td>Yadav Ashutosh</td>
<td>32/2, 32/2</td>
</tr>
<tr>
<td>Yadav Sumedh</td>
<td>141/5, 141/5</td>
</tr>
<tr>
<td>Yamada Sairi</td>
<td>80/2, 80/2</td>
</tr>
<tr>
<td>Yamaguchi Eri</td>
<td>146/3, 146/3</td>
</tr>
<tr>
<td>Yamaguchi Seichi</td>
<td>138/1, 138/1</td>
</tr>
<tr>
<td>Yamamoto Daisuke</td>
<td>80/6, 80/6</td>
</tr>
<tr>
<td>Yamamoto Keita</td>
<td>21/2, 21/2</td>
</tr>
</tbody>
</table>

ICMF 2016 International Conference on Multiphase Flow
Firenze, Italy, May 22 - 27, 2016
<table>
<thead>
<tr>
<th>Name</th>
<th>Pages</th>
</tr>
</thead>
<tbody>
<tr>
<td>Zhang Meng</td>
<td>99/1, 144/2, 146/5, 99/1, 144/2, 146/5</td>
</tr>
<tr>
<td>Zhang Mingyu</td>
<td>84/1, 84/1</td>
</tr>
<tr>
<td>Zhang Peiyu</td>
<td>142/5, 142/5</td>
</tr>
<tr>
<td>Zhang Qian</td>
<td>113/1, 113/1</td>
</tr>
<tr>
<td>Zhang Qiaoling</td>
<td>25/4, 25/4</td>
</tr>
<tr>
<td>Zhang Wenbin</td>
<td>70/1, 70/1</td>
</tr>
<tr>
<td>Zhang Wenhao</td>
<td>10/5, 10/5</td>
</tr>
<tr>
<td>Zhang Xiaoling</td>
<td>149/3, 149/3</td>
</tr>
<tr>
<td>Zhang Xiaopeng</td>
<td>29/3, 29/3</td>
</tr>
<tr>
<td>Zhang Yilin</td>
<td>29/3, 29/3</td>
</tr>
<tr>
<td>Zhang Yuning</td>
<td>20/6, 20/6</td>
</tr>
<tr>
<td>Zhang Ziwei</td>
<td>83/1, 83/1</td>
</tr>
<tr>
<td>Zhao Hui</td>
<td>126/6, 145/1, 126/6, 145/1</td>
</tr>
<tr>
<td>Zhao Jianfu</td>
<td>11/2, 11/2</td>
</tr>
<tr>
<td>Zhao Liang</td>
<td>88/4, 127/5, 149/6, 88/4, 127/5, 149/6</td>
</tr>
<tr>
<td>Zhao Lihao</td>
<td>8/4, 94/6, 102/4, 8/4, 94/6, 102/4</td>
</tr>
<tr>
<td>Zhao Quanbin</td>
<td>77/1, 77/1</td>
</tr>
<tr>
<td>Zhao Yanlin</td>
<td>26/6, 68/3, 26/6, 68/3</td>
</tr>
<tr>
<td>Zhao Yuyang</td>
<td>144/4, 144/4</td>
</tr>
<tr>
<td>Zhe Jiang</td>
<td>127/5, 149/6, 127/5, 149/6</td>
</tr>
<tr>
<td>Zhendong Yang</td>
<td>25/4, 25/4</td>
</tr>
<tr>
<td>Zheng Chuguang</td>
<td>44/4, 100/1, 124/6, 134/5, 44/4, 100/1, 124/6, 134/5</td>
</tr>
<tr>
<td>Zhou Chengsi</td>
<td>9/5, 9/5</td>
</tr>
<tr>
<td>Zhou Guanyu</td>
<td>121/5, 121/5</td>
</tr>
<tr>
<td>Zhou Kun</td>
<td>116/1</td>
</tr>
<tr>
<td>Zhou Lixing</td>
<td>114/6, 114/6</td>
</tr>
<tr>
<td>Zhou Xiangyang</td>
<td>24/2, 24/2</td>
</tr>
<tr>
<td>Zhou Yixian</td>
<td>82/5, 82/5</td>
</tr>
<tr>
<td>Zhou Zifu</td>
<td>53/2, 53/2</td>
</tr>
<tr>
<td>Zhu Ge</td>
<td>33/6, 33/6</td>
</tr>
<tr>
<td>Zhu Weiping</td>
<td>10/4, 24/5, 88/2, 10/4, 24/5, 88/2</td>
</tr>
<tr>
<td>Zhurov Andrei</td>
<td>145/2, 145/2</td>
</tr>
<tr>
<td>Ziegenhein Thomas</td>
<td>19/4, 72/2, 95/5</td>
</tr>
<tr>
<td>Zinatullin Rasul</td>
<td>74/2, 74/2</td>
</tr>
<tr>
<td>Ziskind Gennady</td>
<td>79/6, 79/6</td>
</tr>
<tr>
<td>Zonta Francesco</td>
<td>22/6, 71/1, 84/3, 111/3, 22/6, 71/1, 84/3, 111/3</td>
</tr>
<tr>
<td>Zoueshtlagh Farzam</td>
<td>98/4, 98/4</td>
</tr>
<tr>
<td>Zun Iztok</td>
<td>37/4, 55/1, 37/4, 55/1</td>
</tr>
<tr>
<td>Zuzio Davide</td>
<td>24/1, 118/4, 24/1, 118/4</td>
</tr>
</tbody>
</table>
NUMERICAL ANALYSIS OF THE FLOW AND PARTICLE PATTERN IN THE HUMAN NASAL CAVITY
All Fournou, University of Heidelberg, Germany; Xinghua 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; Kai Ithavong, RMIT University, Australia; Jiyyun Tu, RMIT University, Australia; Rui Chen, National Center for Nanoscience and Technology of China, China; Chunyong Chen, National Center for Nanoscience and Technology of China, China
This paper presents a comparative nanoparticle deposition study using a three-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. 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-Seth / Ghost Fluids method is developed. Previous work focused on the boiling of a static bubble immersed in an overheated liquid has allowed numerically estimating. Using a surface unwrapping technique, the deposition hot spots quantified by deposition flux were fully accessed over the whole nasal cavity. 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.

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 droplet 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.
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 droplets 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 density extended the ranges where the droplet 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.

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 microm 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-dimensional model coefficients. A standard cavitation model which provided the distribution of vapor and liquid phases 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 helps for future sophisticated modelling 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 Fuji, 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.**

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

Foivos C. Koukouvinis, City University London, UK; Ioannis Karathanasis, 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 cavitation flow, but 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 considered 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. Interface tracking models are used to account for proper pressure recovery in the micro channels of the injector. The described methodologies are compared and the results show that, with the increase in the initial vapor-to-surface temperature difference and with the decrease in the ethanol vapor concentration, the process time becomes shorter. The condensation mode transformed from droplets 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 density extended the ranges where the droplet 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 JSPS KAKENHI 26630043 and 25820038 are gratefully acknowledged.**

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 drops remain intact, 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 reflective and stretching sequences. 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
Roderick 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. An initial step 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 past have to be not valid. 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.


MULTI-SCALE MODELING OF BUBBLE COALESCEANCE 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 gas dispersion and maintenance of bubbles can play a central 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 scales model reliability concerning coalescence and breakup 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 IMMERSED BOUNDARY METHOD FOR SPHERICAL PARTICLES OF ARBITRARY DENSITY RATIO
Silvio Tschigrale, 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 be adapted which extends the 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
Gennady M. Grishkov, Eindhoven University of Technology, The Netherlands; Niels G. Deen, Eindhoven University of Technology, The Netherlands; Johan Padding, Eindhoven University of Technology, The Netherlands; Alfred Jongmans, 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 collisions for the modeling and development of spray dryer 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 modeling of the effect of viscosity on the droplet-droplet collision outcome. The role of viscosity is characterized by the capillary number. Together with other parameters, the capillary number is used for empirical correlation of the outcome (bouncing, coalescence, agglomeration or breakup). Consequently, the results of this investigation can be used for the closure of macro scale models for spray dryer operations.

GENERALIZED IMMERSED BOUNDARY METHOD FOR UNSTRUCTURED MESH
Mohd Hazmil Syahidy Abdul Azis, 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 modifications will be discussed. This contribution will also present a generalization of the IBM. This novel approach 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 of the Navier-Stokes equations is proposed. In addition, different types of interpolation methods and weighting functions for force spreading are discussed. Validation and application parameters 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 IMMERSED BOUNDARY METHOD FOR COMPLEX-SHAPED BUBBLES REPRESENTED BY SPHERICAL HARMONIC FUNCTIONS
Benjamin Krull, TU Dresden, Germany; Silvio Tschigrale, 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 bubble 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 AS. 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 recover 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 (0.15<n<1.2) and temperature-dependent viscosity (0.5<n<5.5) 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.

**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 winds and waves from the Adriatic Sea and is vulnerable to morphodynamic activity during extreme weather events (e.g., winter Bora). Sediment loads and transport patterns may be strongly influenced by the confinement of fine cohesive suspended sediment contained in the drainage from the river mixing with coastal sand and 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 near 0.4 m of deposition was observed 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 6 m water depth locations.

*Support provided by Office of Naval Research in Wash., DC and London, UK (N02909-13-1-N202) and Italian RITMARE (SP3-WP4).

**THE EFFECT OF BED GEOMETRY AND COHERENT VORTICES ON INCipient 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 would be induced by the pressure gradients (Steady 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 consecutive grains 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 hydrodynamics 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–Stead 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.

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

Joao Vasques, University of Nottingham, UK; Andrey Cherdantsev, Kuteletadze Institute of Thermophysics, Russian Federation; David Hains, University of Nottingham, UK; Buddhika 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 acquired 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 characterization. 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
Joachim Saraee Lee, Audi AG, Germany; Daniela Vincenzi, 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 1 to 3 kg/s into water flowing through a pipe with a 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 used. The current time-lag pressure fluctuations were simultaneously recorded with a laser-induced fluorescence (LIF) technique, brightness-based LIF (BBLIF) and planar LIF (PLIF), all of which were simultaneously shell and qualitatively investigated. The liquid film thickness was measured using a bright-field microscope technique and it was observed that the liquid film thickness is higher at the edges of the orifice than in the center. The flow regime is visually recognized as slug and plug flow. The slug and plug frequency lies between 0.1 and 5 Hz, and the liquid velocity rises with the increasing air mass flow rate. The power spectral density of the pressure fluctuations increases with a growth rate. It is found that the agreement is remarkable.

VISCOUS OIL-WATER HORIZONTAL WAVY STRATIFIED FLOW: CFD AND EXPERIMENTAL DATA COMPARISON
Ricardo Avila, University of São Paulo - USP, Brazil; Andre Arrojeitas, 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 friction. The data of interfacial wave characteristics, amplitude, length and wave velocity were compared with CFD results; the same was done for pressure drop and volumetric friction from experimental data base were compared with predictions from phenomenological models and CFD results. Although the volumetric friction 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 investigated flow focusing on continuous relaminarization.

Spatial- and Time-Resolved Measurements of Liquid Hold-up Distribution in a Packed Bed Using Electrical Tomography and Volume Fraction 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/m2s) and air (0.2–2 kg/m2s) were used as working fluids and alumina particles (d50 = 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 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.

SIMULTANEOUS BRIGHTNESS-BASED AND PLANAR LASER-INDUCED FLUORESCENCE MEASUREMENT OF DOWNWARDS ANNUAL FLOW
Jae Sik An, Imperial College London, UK; Andrey Cherdtantsev, 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 wavelengths from 0.1 to 3.2 [g/s] to water flowing in a pipe with a diameter of 25 mm, as 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 to measure the liquid 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 thickness in an annulus, current gas-liquid two-phase flow in a vertical pipe with a nominal bore of 32.4 ± 0.4 mm. These two non-intrusive high-resolution techniques were used for flows that covered a liquid Reynolds number range of ReL = 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 compared with film thickness data extracted from simultaneous high-speed visualisation and a twin-wire conductivity probe.

ACOUSTICAL CHARACTERISTICS OF TWO-PHASE HORIZONTAL INTERMITTENT FLOW THROUGH AN ORIFICE
Joachim Saraee Lee, Audi AG, Germany; Daniela Vincenzi, 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 0.1 to 3 kg/s into water flowing through a pipe with a 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 used. The current time-lag pressure fluctuations were simultaneously recorded with a laser-induced fluorescence (LIF) technique, brightness-based LIF (BBLIF) and planar LIF (PLIF), all of which were simultaneously shell and qualitatively investigated. The liquid film thickness was measured using a bright-field microscope technique and it was observed that the liquid film thickness is higher at the edges of the orifice than in the center. The flow regime is visually recognized as slug and plug flow. The slug and plug frequency lies between 0.1 and 5 Hz, and the liquid velocity rises with the increasing air mass flow rate. The power spectral density of the pressure fluctuations increases with a growth rate. It is found that the agreement is remarkable.

Spatial- and Time-Resolved Measurements of Liquid Hold-up Distribution in a Packed Bed Using Electrical Tomography and Volume Fraction 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/m2s) and air (0.2–2 kg/m2s) were used as working fluids and alumina particles (d50 = 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 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.
EFFECTS OF SURFACE ROUGHNESS AND INTERPARTICLE COLLISIONS ON ELBOW EROSION

Victorio Vargas Santos, Federal University of Uberlândia, Brazil; Francisco José 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 contact, chemistry, and flow velocity. 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 material 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 flows 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 surfactant kinetics. With increasing surfactant solubility, maximum stability is predicted to shift gradually from close pack 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≈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 Tukovíc 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 and evaporation-controlled (fast) and kinetically-controlled (slow) adsorption, which necessitate substantially different numerical treatments. To account for both cross-diffusion and non-idealties of surfactant mixtures, diffuse interface models are employed in both the microscopic 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 straightforward to implement for the fine 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.

A HYBRID NUMERICAL METHOD FOR TWO-PHASE FLOW WITH SOLUBLE SURFACANT

Michael Siegel, 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 the evolution of a drop on a surface, a study of the surfactant distribution and properties such as a Biot number and 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 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 Schranner, 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 formation of ascending bubbles is 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, incompressible 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 complexity of bubble shape and wake by surfactant parameters.

We acknowledge the Deutsche Forschungsgemeinschaft (DFG) for funding. F.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 Nikrytyk, 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 (St) for liquid and gas phases and the Peclet number (Pe). 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
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 Parnore, 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 only the cushioning of the gas beneath the droplet prior to impact. I will discuss in this talk the different mechanisms 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.

PHENOMENA OF DROP IMPACT ON HOT SURFACES
Jan Breitenbach, TU Darmstadt, Germany; Ilia V. Rothman, 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 droplets, 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).
SPALSH OF A LIQUID DROP IMPACTING ONTO A WALL FILM OF DIFFERENT LIQUID
Hannah Kittel, Technische Universität Darmstadt, Germany; Ila 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 layer 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. Randomly distributed particles are placed in homogeneous shear flow. The confinement 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 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 of fluid and particle suspensions in 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.

ROLE OF PREFERENTIAL CONCENTRATION IN THE DESTABILIZATION OF SHEARED PARTICLE-LADEN FLOWS
Mohamed Housssem Kasbaoui, Cornell University, USA; Donald Koch, 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 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 of fluid and particle suspensions in 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.

A SLUG CAPTURING METHOD APPLIED TO UNCONVENTIONAL SCENARIOS: HIGH VISCOS 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
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-ε 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 ε. 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 provides very accurate description of turbulent flows and particle transport and deposition at modest computational cost.
A MECHANISTIC APPROACH FOR HORIZONTAL SLOUGH FLOW FROM TWO-PHASE (GAS-LIQUID) TO THREE-PHASE WITH THE FORMATION OF GAS HYDRATES
Carlo 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 homogeneous 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 these 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.

Analitical 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 time, 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 bubbles rising in vertical ducts. 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 OF THE EVOLUTION OF HIGHLY VISCOUS OIL SLUG LENGTHS IN HORIZONTAL PIPES
Gianluca Lois, Università di Brescia, Italy; Pietro Poesto, Università degli Studi di Brescia, Italy
Slug flow is an unintermittent regime that often occurs in gas-liquid pipe flow and it is exhausted 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 for understanding 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 length distributions. The presence of slugs is observed during the experiments and the probability density functions of their lengths are built.

An experimental slug tracking model has been implemented, which calculates the increase or decrease in length for each individual slug from a random input distribution, managing also events like shorter slugs disappearance. These analyses provide insight into the mechanism of hydrodynamic wave instability and into the stability and reliability of prediction process.

EXPERIMENTAL INVESTIGATION OF THE FLOW STRUCTURE AROUND TAYLOR BUBBLES IN VERTICAL DUCTS IN THE PRESENCE OF DISPERSED BUBBLES
Emilio Palladino, SINMEC / Federal University of Santa Catarina, Brazil; Rafael Cerequeira, SINMEC / Federal University of Santa Catarina, Brazil; Rosa Brito, SINMEC / Federal University of Santa Catarina, Brazil; Cristian Maliska, SINMEC / Federal University of Santa Catarina, Brazil; Cristina 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 Agunjejika, 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 (pipelines, 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 tallied 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 conduit as occurs in Mount Etna, Sicily. 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

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 investigated 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 influence the first one. We 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

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. For surface tension dominant 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 at specific point of the cloud by normal fittings. 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.

BUBBLES IN NUCLEATE BOILING

Alexander Gulin, Massachusetts Institute of Technology (MIT), USA; Jacopo Buongiorno, Massachusetts Institute of Technology, USA; Stephanie Zaleski, Université Pierre et Marie Curie (Paris 6), France; Shahnaw 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 ($\mu m$) and the microbubble ($\mu m$) 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 evolution 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 high-order 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 flow properties. The microlayer subgrid model 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 models popularly 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 geometry characteristic different time period: hemi sphere at inertial, from hemi sphere to spherical in thermal diffusion controlled period and sphere plus bottleneck in the departure period. Variations 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 Tatekuri, Hokkaido University, Japan; Kazunori Kobayashi, Hokkaido University, Japan; Masao Watanabe, Hokkaido University, Japan; Hiroki Fuji, 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 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 physics-consistent set of 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 coefficients in 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 26030043 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 cavitating 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 cavitating bubble clouds as a function of their size, void fraction, and nuclei distribution. Cavitlation 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-DK04381 is gratefully acknowledged.

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

Helmholtz-Zentrum Geesthacht, Institute of Mechanics, Chinese Academy of Sciences, China; Jianhua Zhao, Institute of Mechanics, Chinese Academy of Sciences, China; Hui-Xiong Li, Xian Jiaotong University, China; Kai Li, Institute of Mechanics, Chinese Academy of Sciences, China

SOBER-SJ10 (Single bubble POOL boiling Experimert aboard SJ-10) is one of 27 programs of the experiment 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 temperature on the surface of the heater. 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 experiment will be performed in April of 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 XDA04020404 (Strategic Priority Research Program on Space Science/CAS) is gratefully acknowledged.

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


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 superposition of 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 cavitation fluid as a dispersion of spherical bubbles in compressible water, is tested of 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 fluid 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-condensible gas. These results also differ from the corresponding values obtained using the pressure cut-off. The final 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 Timosheskij, Novosibirsk State University, Russian Federation; Korstantin Pervunin, Kutateladze Institute of Thermophysics, SB RAS, Russian Federation; Dmitry 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 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-cavitation flows. A hydroacoustic pressure transducer was employed to perform local pressure measurements and to analyze frequency characteristics of gas-vapor cavities. Cavitlation 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 inclination, 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 (ReH=14000) laden with nanoparticles at a controlled concentration and particle size distribution (PSD). A high concentration (> 1012#/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 carbon-soot particles - GMD=8nm). Hydrodynamics, 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 strated in order 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.

* Financial support from Bev is gratefully acknowledged.

PERFORMANCE ASSESSMENT OF EULERIAN VERSUS EULERIAN-LAGRANGIAN LEVEL SET IMPLEMENTATIONS FOR THE SIMULATION OF TWO PHASE FLOWS
Gael Loriel, Université catholique de Louvain, Belgium; Philippe Chatelan, 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 avoids any convective stability constraints (CFL). We thus expect significantly lower computational costs together with higher accuracy for treatment waives classical stability constraints for convection (CFL). We thus 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 straffled bubble, - GMD=8nm). Hydrodynamics, 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 strated in order 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.

* Financial support from BelV is gratefully acknowledged.

HYDROPARTICLES 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 Lu, 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 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 avoids any convective stability constraints (CFL). We thus expect significantly lower computational costs together with higher accuracy for treatment waives classical stability constraints for convection (CFL). We thus 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 carbon-soot particles - GMD=8nm). Hydrodynamics, 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 strated in order 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.

* Financial support from Bev is gratefully acknowledged.

PREFERENTIAL CONCENTRATION IN COHESIVE 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 concentrations. 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.

BUBBLE DYNAMICS IN PLUG FLOW REGIME FOR HORIZONTAL GAS-LIQUID FLOW
Kamalnayan Benjesee, S V National Institute of Technology Surat, India; Jgnesh Thaker, S V National Institute of Technology Surat, 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 1500 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. When plug bubble tail encounters 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 the natural 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 coalesce 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.

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 FLOWS SIMULATIONS IN SOLID ROCKET MOTOR INSTABILITIES
Valentin Dupin, ONERA, France; Marc Massot, CentraleSupélec, EM2C, France; Joel Dupays, ONERA, France; Frederique Laurent, EM2C, CentraleSupélec, 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.

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 interface 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.


NUMERICAL PREDICTION OF SEDIMENT DEPOSITION AT A RIVER CONFLUENCE USING AN EULER-LAGRANGE METHOD
Souria Hamdouche, Université de Lorraine - LEMTA, France; Boris Arcen, Université de Lorraine - LRGF, 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 barge 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.

A NUMERICAL INVESTIGATION OF PARTICLE-BED-TURBULENCE INTERACTIONS IN OSCILLATORY FLOWS
Chaitanya Ghodke, Oregon State University, USA; Sourabh Apte, Oregon State University, USA
Particle resolves direct numerical simulations are performed to investigate the effects of an oscillatory flow field over a rough-bed, corresponding to the experimental setup of Keller & Sleath (1976) for transitional and turbulent flows over a range of Reynolds numbers (6000-45000) 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.

COMPUTATIONAL MODELLING OF PROPPANT TRANSPORT IN FRACTURES
Goodarz Ahmad, Clarkson University, USA; Amir Abdollahi Mofakham, Clarkson University, USA; Dustin Crandall, 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.

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-Newtonian 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 μm 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 four phase-weakness probe arrays. 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.

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

Andrew Sederman, MIRC, UK; Wes Maru, Oil & Gas Measurement Limited, UK; Susithra Lakshmanan, MIRC, 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 well 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 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.

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

Honayoon 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 techniques. 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.

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

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 four phase-weakness probe arrays. 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.

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

Andrew Sederman, MIRC, UK; Wes Maru, Oil & Gas Measurement Limited, UK; Susithra Lakshmanan, MIRC, 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 well 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 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.

**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.

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

Wibeke 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, but still 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 deduce 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 before-mentioned purpose. For the numerical computation an in-house available Smoothed-Particle-Hydrodynamics (SPH) code developed by ANDRITZ Hydro AG equipped with 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.
The air/fuel mixture, especially the spray penetration. 15 explained in terms of transverse size of disturbance waves. Two scenarios of 3D in transverse coherence with increasing pipe diameter and gas velocity was resolution of 0.22 mm/pixel. It was shown that the disturbance waves are initially rectangular duct. Measurements were performed over an area of duct’s bottom surface of liquid film in presence of high-velocity gas stream. Knowledge of Disturbance waves are large lumps of liquid travelling with high speed over the channel wall and stabilize at the center of the channel while the drops/bubbles completely, i.e., the contaminated drops/bubbles drift away from the channel wall and 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.

**SURFACANT INDUCED LATERAL MIGRATION OF BUBBLES AND DROPS IN A SHEAR FLOW**

Melin 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 alter the dynamics of the drops/bubbles completely, i.e., the contaminated drops/bubbles drift away from the channel wall and 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.

**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 Merler, 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; Barry James Attzopardi, 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 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 examined. The coalescence increasing the transverse coherence of film wavy structure were observed.

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

Basant Kumar Rana, Indian Institute of Technology Kharagpur, India; Yagee Sharma, Indian Institute of Technology Roorkee, India; Arup Kumar Das, Indian Institute of Technology Roorkee, 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. We observed that the film attains maximum thickness at the base and then 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 film depends on the liquid properties (ρ1, p2, μ1, μ2 and d) and size of cylinder (D) respectively. Rotational speed (ω) 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.
This work presents a modelling study of gas-particle heat transfer on two distinct scales. 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 Peclet 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 models for the impact velocity on the splash formation. We also compared our results with the splash/non-splash boundary reported by Vandel Wal et al. (1999). We developed a high-speed droplet injector using a syringe to observe droplet impacts with velocities of \(10 \text{ m/s}\) on an acrylic plate. The Eulerian or Lagrangian approaches for computation of droplets velocities and its concentration field are compared. The unresolved Lagrangian approach also incorporates a 1D heat conduction model to directly simulate heat transfer inside the particles. This model is verified against DNS data in more wall effects and intraparticle 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° 604556.

**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 is investigated. A droplet model is used to numerically simulate the droplet evaporation processes. The numerical model is based on Eulerian by Zaichik (1999) and full Lagrangian by Osipovtsov (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 standard k-\(\epsilon\) model neglecting second moment closures. Particle In-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; Hiroki Fujii, Hokkaido University, Japan; Toshiyuki Sanada, Shizuoka University, Japan; Viktor Terekhov, Kutateladze Institute of Thermophysics, Russian Federation; Lakehal, ASCOMP AG, Switzerland; Viktor Terekhov, Kutateladze Institute of Thermophysics, Russian Federation; Viktor Terekhov, Kutateladze Institute of Thermophysics, Russia

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 23.6 m/s by using our high-speed droplet injection. The results show that the droplet dynamics after the impact is closely related to 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.

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

Toshihisa Isorno, Hokkaido University, Japan; Masao Watanabe, Hokkaido University, Japan; Kazumichi Kobayashi, Hokkaido University, Japan; Hiroki 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 is 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.
FREQUENCY AND NUCLEATION RATES OF SUPERCOOLED WATER DROPS IMPACTING ON A COLD SURFACE

Markus Schreiber, Technische Universität Darmstadt, Germany; Ilia V. Roisman, Technische Universität Darmstadt, Germany; Suad Jakirlic, Technische Universität Darmstadt, Germany; Cameron Tropea, Technische Universität 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 considerably 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, time snapshots, row by row, 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, the effective Stokes number then can be defined in terms of 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 the influence of the flow speed and the particle diameter on the deposition efficiency. 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.

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 particle characteristics lead to significant differences 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. An effective Stokes number can be defined in terms of 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 the influence of the flow speed and the particle diameter on the deposition efficiency. 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.

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

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

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 considerably 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, time snapshots, row by row, 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. An effective Stokes number can be defined in terms of 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 the influence of the flow speed and the particle diameter on the deposition efficiency. 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 IN Variant 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. On 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
Scheinetz Ibrahimi, Toulouse, France; Nathalie Le Saux, Toulouse university, France; Emmanuel Cid, Toulouse university, France; Olivier Masbernat, Tlemoine, France
Phase inversion phenomenon of a concentrated oil 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 phase drops in 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.

WIRE-MESH-BASED TOMOGRAPHY IN UPWARD-VERTICAL OIL-WATER-AIR THREE-PHASE FLOW
Hugo Fernando Velasco Penha, University of Sao Paulo, Brazil; Adriana Bonilla Riano, University of Sao Paulo, Brazil; Carlos Henrique Monteiro De Carvalho, Petrobras, Brazil; Valdir Estevam, Petrobras SA, Brazil; Oscar Mauricio Hernandez Rodriguez, University of Sao 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 oil, gas, nuclear, food and chemical industries. There are two kinds of wire-mesh sensors that can be applied in two-phase flows, the conductivity type sensor and the volumetric sensor. Therefore, 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 16x16 sensor.

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 Sao Paulo, Sao Carlos School of Engineering, Brazil; Carlos Henrique Monteiro De Carvalho, Petrobras, Brazil; Valdir Estevam, Petrobras SA, Brazil; Oscar Mauricio Hernandez Rodriguez, University of Sao 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.

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 Da Castro, State University of Campinas - UNICAMP, Brazil; Jorge Luiz Blazussi, State University of Campinas, Brazil
It is well known that different flow patterns lead 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 those for viscous and ultra-viscous oils. It already has 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.

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 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.

MODEL OF BUBBLE RISE AND COLLISION WITH COMPLEX DEFORMABLE INTERFACES
Derek Chan, University of Melbourne, Australia; Rogiero 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.

MODEL OF BUBBLE RISE AND COLLISION WITH COMPLEX DEFORMABLE INTERFACES
Derek Chan, University of Melbourne, Australia; Rogiero 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 Léonard, 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 shape at detachment 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 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.

HYDRODYNAMICS OF BUBBLE COLUMNS IN THE HETEROGENEOUS REGIME
Pedro Maximiano Raimundo, IFP Energies 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 Energies Nouvelles, France; Frederic Augier, IFP Energies 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 and simulations have been 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 available literature data.

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; Buddhiktha 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 0.6 and 1.27 m 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 the evolution of the void fraction with time as well as the 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.

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(0.25m/min-0.714m/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][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

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.

TWO-PHASE TURBULENCE IN LAMINAR BUBBLY PIPE FLOWS
Hyungmin 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 (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.
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 throat 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.

CFD SIMULATION OF CAVITATION UNDER EXPERIMENTALLY ESCALATING HEAT INPUTS
Guanyu Su, Massachusetts Institute of Technology, USA; Matteo Bucco, 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 experimentally 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 parameters. 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 2500 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.

LAGRANGIAN TRACKING OF DROPLETS EVAPORATING IN A HI TURBULENCE BY USING IN-LINE DIGITAL Holography
Jean-Louis Marie, CNRS, France; Thibaud Tironchon, Ecole Centrale de Lyon, France; Nathalie Grosjean, CNRS, France; Loic Mees, CNRS, France; Michel Lancel, 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 the interface and pressure by calculating 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 Charayon et al. New J Phys 14:43039 (2012) and Maré 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.

EXPERIMENTALLY STUDYING 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 Zegarnik, 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 patterned. The tests were at heat input of the range of 4-40 kW/m2 and mass flow rate up to 800 kg/(m2 s) water subcooling relative to saturation temperature 25-70°C, and heat flux density values from 1-2 MW/m2 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.).

VORTEX STRUCTURES OF LOW STEAM QUALITY BOILING TWO-PHASE FLOW IN BUBBLE-TYPE PLATE CHANNEL
Chen Jianyi, Xian Jiaotong University, China; Zonglin Yang, Xian 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, vortices and temperature of a boiling two-phase flow in bubble-type plate channel. The results showed that the flow boiling in channel has a remarkable vortex structure. The vortex structures can be divided into two parts: one is the initial vortex structure which is formed close to the wall and the other is the separated vortex which covers the entire flow section. The initial vortex structure is formed due to the secondary flow in the channel and the separated vortex is formed due to the forced vortex. The results showed that the vortex structures are closely related to the characteristic of motion and heat transfer.
SYNCHROTRON X-RAY PHASE CONTRAST IMAGING OF CAVITATION IN FUEL INJECTOR NOZZLES WITH VARIOUS SIZES
Abid Soi, Kobe University, Japan; Rubby Prasetya, Kobe University, Japan; Seokmoon No, Nat’l Inst of Advanced Industrial Science and Technology, Japan; Yoshihisa 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 exist for nozzle size, (2) cavitation tends to be distortions and, therefore, shorter in small nozzles due to the relatively large normalized radii of curvature 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.

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 Universität Darmstadt, Germany; Christina Weickgenannt, , Germany; Cameron Topoea, Technische Universität 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/s². 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 Bourgoin, CNRS, France; Alain Cartellier, Université Grenoble Alpes and CNRS, France
Particle-laden flows or flows with inertial particles are of a great interest of USA; Mickael Bourgoin, CNRS, France; Alain Cartellier, Université Grenoble Alpes and CNRS, France; Alberto Aliseda, University of Washington, 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.

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; Jincar 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 drop in the atmosphere is demineralized by clusters. 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 3rp (rp 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.
DIFFUSE INTERFACE SIMULATIONS OF PHASE SEPARATION OF SHEARED BINARY MIXTURES USING A FINITE VOLUME SOLVER

Riccardo Mosini, University of Pisa, Italy; Giuseppe Di Vitantonio, University of 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 phase boundaries. The diffuse interface model can be implemented in a wide variety of simulation codes, allowing to model a broad range of problems. This includes e.g. convection, thinning, coalescence, nucleation, thinning, break-up, and a variety of combinations of these processes. The numerical method is tested in various test cases such as the relaxation of an ellipse or a cross with and without convection. Second, the weak formulation of 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 (varational 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.
SURFACING AND CLUSTERING OF GYROTACTIC SWIMMERS IN FREE-SURFACE TURBULENCE
Salvatore Lombaiche, Institute of Mechanics of the Fluids of 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 microorganisms 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 CO2 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. Velocities and vortex can be so large that they even reorient themselves naturally rising motion. Knowing the rate at which swimmers reach water surface is important to estimate global CO2 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, reacting to the surface or 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; Jaiabakalya GuhaThakurta, University of Stuttgart, Germany; Youssef Baroud, University of Stuttgart, Germany; Sven Simon, University of Stuttgart, Germany; Gunter Rinke, Karlsruhe Institute of Technology (KIT), Germany; Roland Dittmeyer, 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 gas-liquid reactions, especially in bubble columns. A frequently used tool for concentration is optical absorption, but due to the curvature effect of each bubble interface into account. Finally the proposed measurement techniques can be described with three terms, a local average term due to the “characteristic” bubble has on the liquid turbulence surrounding it. Analogous to single-phase velocity vector in 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 Bubbles Flows”.

CRUDE OIL PLUMES IN CROSSFLOW: EFFECT OF TURBULENT FLOW STRUCTURE ON OIL RESIDENCE IN PLUME
David Melin-Marrero, Johns Hopkins University, USA; Xiaoli 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 Froude (Fr) 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 crossflow 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 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, trapping by the CVP and non-turbulent 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; Yasmin 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 used to determine the local phase velocities and gas hold-up, in this work, visualization techniques used to determine quantitatively the individual phases, 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 velocity vector in the presented Real-Time Raman Process Analysis System based on a dedicated high-speed camera with integrated real-time image processing.

“The role of the tracer particles in the presented Real-Time Raman Process Analysis System based on a dedicated high-speed camera with integrated real-time image processing.

THE ROLE OF INLET FLOW INSTABILITIES IN LIQUID-LIQUID FLOW
Rhys Morgan, Forsys Subsea, UK; Christos N. Marikides, 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 aequous 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 imposed 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.

DEVELOPMENT OF MEASUREMENT Technique FOR BUBBLE VELOCITY VECTOR IN GAS-LIQUID TWO-PHASE FLOW BY USING 4-SENSOR PROBE
Yasuhi Saito, Kyoto University Research Reactor Institute, Japan; Gen Aiyoshi, 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

**Methodology** has been tested on two atomization configurations, a planar atomization setup and another setup involving a chemical source term. The atomization model acts as a volume source term for the multi-fluid solver and is designed 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 takes into 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, when simulating unsteady processes like combustion instabilities. 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 solver. In addition, multi-fluid simulations of reactive combustion problems are also integrated in industrial simulations of reactive combustion phenomena.

**Keywords:**
- Flow modeling
- Multi-fluid solver
- Atomization
- Reactive combustion

**Acknowledgements:**

This work proposes a large scale numerical methodology (LSS, "Large Scale Simulation") to simulate the largest scales of the unsteady assisted atomization process, when simulating unsteady processes like combustion instabilities. 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 solver. In addition, multi-fluid simulations of reactive combustion problems are also integrated in industrial simulations of reactive combustion phenomena.

**References:**


**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

**Keywords:**
- Fire suppression
- CFD
- Particle tracking

**Abstract:** The focus of this work is on simulating the interaction of spray and surface film flow in order to optimize flow rate and pattern. It was found that droplet detachment from interface walls is an important phenomenon in two phase flow. The simulation of droplet detachment is critical for fire suppression applications. The simulation of droplet detachment is important for optimizing spray and surface film flow in order to achieve the desired fire suppression effect. The simulation of droplet detachment is critical for fire suppression applications. The simulation of droplet detachment is important for optimizing spray and surface film flow in order to achieve the desired fire suppression effect.
THE EFFECTS OF NANOPARTICLES IN SUBCOOLED FLOW BOILING IN THE DIFFERENT CHANNELS WITH SAME HYDRAULIC DIAMETER

Shams Mehrzadeh, Khage Nasir Toosi university of technology, Iran, Hasan Almoradi, 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-section 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 implemented 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; Chenyao 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. 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.

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

Hong Hyun Son, Hanyang University, Republic of Korea; Kyung Soo, Hanyang University, Republic of Korea; Uju Jeong, Hanyang University, Republic of Korea; Gyondong Jeun, Hanyang University, China; Sung Joong Kim, Hanyang University, Republic of Korea; Uiju Jeong, Hanyang University, Republic of Korea; Gwang Hyeok Seo, Hanyang University, Republic of Korea; Simon Lo, CD-ADAPCO, UK

Pool boiling CHF experiments were performed to examine the effect of flow obstruction, which has various shape (square, circular, elliptic) and size (10, 15, 20, 25 mm). Stainless steel 304 heaters (99×49×2 mm3) 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 obstruction, the CHF decreased slightly with an increase of obstacle size because lager 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 obstacle was remained as an 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.

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

Yang Zhejiang, xian University of Technology; china; Qiaoqing Zhang, xian University of Technology; china; Qincheng Bi, Xian 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 result 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 of changeable cross-section and constant cross section were 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 implemented 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

Mehdi 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 technology. 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 phenomena. 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. A new parameter is determined by the mechanistic analysis 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 avoids 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.
SURFACANT EFFECTS ON DROPLET COALESCEENCE WITH A LIQUID-LIQUID INTERFACE
Panthongset Angkoon, University College London, UK; Victor Voudouropoulos, University College London, UK; Wehlye Wehlye Hashi, University College London, UK; Maxime Chirnside, University College London, UK
In this study, droplet coalescence on an oil-water flat interface is investigated experimentally in a vertical H-type Hele Shaw cell with 1.5 mm width. In such a configuration, the flow field recorded perpendicularly to the planes 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 boundaries which are then integrated which is the effect 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 SURFACANT CONCENTRATION AT AN INTERFACE OF SINGLE SPHERICAL DROP
Kosuke Hayashi, Kobe University, Japan; Yuya Masukura, Kobe University, Japan; Shigeo Hosokawa, Kobe University, Japan; Akio Tsuchiya, Kobe University, Japan
The Frumkin-Levich model for the adsorption-desorption kinetics of surfactant is 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 interface velocity and the Marangoni force. The authors will present data of single contaminated glycerol-water drops in silicone oil at selected Triton X-100 concentrations in the coalescence (and to a lesser extent for the interface breaking points which are then advected to the bulk of the liquid). The computations were done 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 seen 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 boundaries which are then integrated which is the effect 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.

EXPERIMENTAL EVALUATION OF SURFACANT CONCENTRATION AT AN INTERFACE OF SINGLE SPHERICAL DROP
Shigeo Hosokawa, Kobe University, Japan; Yuya Masukura, Kobe University, Japan; Kosuke Hayashi, Kobe University, Japan; Akio Tsuchiya, 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 droplet/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 at an interface.

EFFECT OF INTERFACIAL PROPERTIES ON DROPLET-WATER 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 mechanisms and characteristic features 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 liquid-solid 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 liquid-solid interface.

NUMERICAL INVESTIGATION OF PARTICLE DEPOSITION IN TURBULENT DUCT FLOWS
Jun Yao, Xi an 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 across-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.

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 Tsvrda, TNO, The Netherlands; Raoul 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 physics 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 free-flight model. In this study, droplet coalescence on an oil-water flat interface is investigated experimentally in a vertical H-type Hele Shaw cell with 1.5 mm width. In such a configuration, the flow field recorded perpendicularly to the planes 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 boundaries which are then integrated which is the effect 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.

EXPERIMENTAL EVALUATION OF SURFACANT CONCENTRATION AT AN INTERFACE OF SINGLE SPHERICAL DROP
Shigeo Hosokawa, Kobe University, Japan; Yuya Masukura, Kobe University, Japan; Kosuke Hayashi, Kobe University, Japan; Akio Tsuchiya, 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 at an interface.

CLIENT LOGIN
ICMF 2016 International Conference on Multiphase Flow
Firenze, Italy, May 22 - 27, 2016
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 particle deposition and transport 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. 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 for the lift of a non-rotating particle reduces the deposition rate predicted by Milliken-Chen, but the change is 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 ($\tau = 20$) in the near wall region.

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

Roberto Ibarra, Imperial College London, UK; Ivan Zadrzal, 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 variety of industrial processes from reactors to oil export pipelines at offshore facilities. The behaviour of these flows is a complex phenomenon not found in which interfacial characteristics and droplet formation play an important role in the development of liquid-liquid segregated flow patterns and dispersed flow phenomena. The co-current flow of immiscible liquids such as oil/gas or water/oil, with a difference in density greater than 0.1, is a complex phenomenon due to the interaction between the dispersed phase and the continuous phase, which results in the formation of various flow patterns. The characterization 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 Exxon D140 in a horizontal 32 mm 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 Azizopardi, 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 is explored. These data are combined to infer the fraction of liquid entrained in the gas. The 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.

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 simulating 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 on the particle location invariably make worse predictions and do 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 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. This work is motivated by the fact that 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. Monodisperse 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

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

Shara Mohammed, University of Nottingham, UK; Abbas Hasan, University of Nottingham, UK; Barry James Azizopardi, University of Nottingham, UK; Georgios Dimitrakis, The university of Nottingham, UK

The majority of the literature on gas-liquid slug flow has been focused on low and moderate viscosity fluids. However, high viscosity liquids are widely used in a variety of industrial processes due to their unique properties. The injection of air into a high viscosity liquid can significantly affect the flow structure and dynamics. In this work, we investigate the effects of air injection positions on the flow structure of high viscosity oil in a large diameter pipe. Studies on the flow structure of high viscosity oil (300 Pa.s) in vitable columns of 240 and 290 mm in 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 frequency and velocity, 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 frequency 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 Azizopardi, University of Nottingham, UK; Buddhidhika Hewakandambay, University of Nottingham, UK; Josef Essig, Essig, 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 58 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 0.1 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 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 PLANEAR CO-FLOW CONFIGURATION
Javier Ruiz-Rus, Universidad de Jaén, Spain; Rocio Balanós Jimenez, Universidad de Jaén, Spain; Candido Gutierrez Montes, Universidade 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 \( \eta(n), \eta(b), \eta(t) \), and pressures \( \pi(n), \pi(b), \pi(t) \), respectively. Lengths are made dimensionless with the air thickness at the outlet, \( H_a \), and pressures with the air dynamic pressure, \( p_a \). In a reference frame moving with the water velocity, \( \omega_w \), 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, \( \omega = \omega_w/\omega_a \), and \( \Lambda = u_0/\omega_a \), where \( \omega \) 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 experimental 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 are 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 Nikookar, 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 hydrogels 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
David Hann, University of Nottingham, UK; Andrey Cherdantsev, Katedalske Institute of Thermophysics, Russian Federation; Buddhikha Hewakandambage, 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 additional bubbles are unstable and pop. The majority of bubbles created by the bubble 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 Prahl Wittberg, Royal Institute of Technology (KTH), Sweden; Lazio 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). In this paper, the development of arterial secondary 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
Nicolas Berg, KTH, Sweden; Laszlo Fuchs, Royal Institute of Technology (KTH), Sweden; Lisa Prath 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 Hus, 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 superhydrophilic surface on flow boiling enhancement in a natural circulation thermosiphon. The super-hydrophobic 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 50%. 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.

PERFORMANCE EVALUATION OF CAPILLARY-ASSISTED PLATE-TYPE EVAPORATOR WITH CU-CNT-TIO2 COMPOSITE COATING
Edward Joshua Plastago, Chonbuk National University, Republic of Korea; Hyun-Ho Ha, Chonbuk National University, Republic of Korea; Dong Ao Cha, Republic of Korea Institute of Industrial Technology, Republic of Korea; Kyoung Jin Bae, Korea Institute of Industrial Technology, Republic of Korea; Min Soo Kim, Chonbuk National University, Korea; Oh Kyoung Kwon, Republic of Korea Institute of Industrial Technology, Republic of Korea; Chan Woo Park, Chonbuk National University, Republic of Korea

Ternary Cu-CNT-TiO2 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 Cu. 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.
VOID FRACTION FLOW FIELD MEASUREMENTS OF CAVITATING FLOW IN THE WAKE OF A CIRCULAR CYLINDER
Harish Ganesh, University of Michigan, USA; Julian 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 cylinder are studied for a range of cavitation numbers, at a Reynolds number of 1x10-5, 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. K-han Kim.

RELATIONSHIP BETWEEN MOTION OF ACOUSTIC CAVITATION-ORIENTED BUBBLES AND FLOCULATION PATTERNS, DEPENDING ON COMPONENTS OF DISSOLVED GASES IN WATER
Koji Nomata, Shizuoka University, Japan; Sayuri Yanai, Shizuoka University, Japan; Hirona 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 μm-order due to the wavelength limitation. We used kHz-band ultrasound and have overcome the limitation. Floculation of sub-mm-order particles dispersing 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 floculation position by the acoustic radiation force acting on the bubbles. We discovered the floculation patterns were categorized into two types: spherical or chain-like, and the key factor was the dissolved gas components. The spherical floculation resulted from the dissolved air gas, and the chain-like one resulted from the dissolved CO2. However, the mechanism of the particle floculation 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 floculation, 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 floculation patterns and the bubble motions.

THE BREAKUP OF A VISCous LIQUID DROP IN A HIGH REYNOLDS NUMBER SHEAR FLOW
Chin Hei Nig, University of Washington, USA; Alberto Alliseda, 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) submersed round jet. Unlike in the H-K theory, applicable to the breakup 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

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 low 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 12μm/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 (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.

NUMERICAL SIMULATIONS OF THE UNSTABLE CAVITATING FLOWS AROUND A CIRCULAR DISK AND CONICAL CAVITATOR
Minje Kim, Agency for Defense Development, Republic of Korea; Kurrchul Lee, Agency for Defense Development, Republic of Korea
In the present study, the unsteady cavitation 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; Bali 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 accompanying 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.
PARTICLE GROWTH IN TURBULENT FLOW UNDER DYNAMICALLY CRITICAL STOKES STANCES
Michèle Gualia, 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 \( S_t = \frac{p}{f} \), due to increased collisional probability. We assume that during growth, \( S_t \) stays critical, with the particle response time \( p \) growing according to an evolving flow time scale (i.e., the vorostics around which progresses the least solid norm and is used to design 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 solid-liquid and solid-solid combinations are compared to study the effect of the material property and domain setup on the shock wave refraction, the bubble shape and the transient response of the solid.

* Support from NSF grant No.1U1530147 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; Byoung Jeon, 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-2014R1A2A2A01004879 is gratefully acknowledged.

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 least solid norm is used to design 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 solid-liquid and solid-solid combinations are compared to study the effect of the material property and domain setup on the shock wave refraction, the bubble shape and the transient response of the solid.

A THREE-PHASE NUMERICAL MODEL FOR LIQUID-GAS-GAS FLOWS WITH RELAXATION PROCESSES
Michele Palandri, 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 to the species to simulate phase transition. The homogeneous hyperbolic portion of the equations is solved numerically via finite volume wave propagation schemes. Relaxation terms are treated by source terms that exactly conserve the relaxed states. We present numerical results for, three-phase cavitation tube test, showing that the behavior of the predicted sound speed is consistent with 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 shock-ed compressible, evolutions, large heat sources for the relaxed states. We present numerical results for, three-phase cavitation tube test, showing that the behavior of the predicted sound speed is consistent with 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.

FLOW MEASUREMENT IN OPAQUE FLUIDS USING ULTRASOUND IMAGING VELOCIMETRY
Christian Poeltma, 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 (a more fundamental case for studying turbidity).

FLOW MEASUREMENT IN OPAQUE FLUIDS USING ULTRASOUND IMAGING VELOCIMETRY
Christian Poeltma, 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 (a more fundamental case for studying turbidity).
X-RAY RADIOSCOPY AS A POWERFUL TOOL FOR VISUALIZATION OF LIQUID METAL BUBBLE FLOWS
Olga Rosauchkina, Helmholtz-Zentrum Dresden-Rossendorf (HZDR), Germany; Natalia Shevchenko, Helmholtz-Zentrum Dresden-Rossendorf (HZDR), Germany; Erik Stumpf, Helmholtz-Zentrum Dresden-Rossendorf (HZDR), Germany; Dietrich Hoeppe, 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 bubble flow. The drop size measurements validated analytical 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 GainSn 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 Casstrillon-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 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 analysis. The analysis of the different tests has shown that the solidification conditions have a strong influence on the resulting droplet size distribution. 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.

EXPERIMENTAL AND COMPUTATIONAL INVESTIGATION OF FLOW INSIDE A MILLING CIRCUIT HYDROCYCLONE
John Furlan, GIW Industries, USA; Renjie Ke, Case Western Reserve University, USA; Jakirsdin 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. The present work focuses on the fragmentation of a single molten alloy droplet. 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 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.

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; Kazuhiro Suzumoto, 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.

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 (polymer). Mathematical models are used in the present study and the 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 permeativity-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 of plastic melt and temperature distribution. The present work was constructed 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 ERC grant Advanced, Open Source Technology Transfer Program through Target-driven R&D, Japan Science and Technology Agency (#AS251Z00116K) is gratefully acknowledged.

GAS AND LIQUID DYNAMICS IN BUBBLE COLUMNS: ADVANCED FLOW IMAGING WITH ULTRAFAST X-RAY AND RADIOACTIVE PARTICLE TRACKING
Salam Azziz, 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 summarized 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 reaction 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 (FP7-307360) is gratefully acknowledged.
A NUMERICAL CLOSE-UP ON THE SPRAY FORMATION FROM A GAS-LIQUID MIXING LAYER
Yuri G. D. Pashkarev, Marie Curie (Paris 6), France; Daniel Fuster, CNRS UMR 7190, France; Greta Tyagyvason, 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 gas-liquid mixing layers commonly seen in 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 were 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, the problem of break-up is less advanced than the theoretical and experimental understanding of the process. 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 co-flowing gas and liquid streams, in simulations of different mass fluxes (from 4 to 4 billion kg/s) 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. These fingers 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.

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 circumstances. In this work, we 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 cm cylindrical jet with air. Triangular angles and a 0.27m/s and gas flow velocity of 22.6m/s. The grid is 512x12x1024, the mesh is equal to 66µm and massively parallel computing is used (8192 cores). Boundary layers are taken into account in the initial profiles and proved to be of low 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 femtosecond laser).

EFFECT OF THE ALIGNED FLOW OBSTACLES ON SUBCOOLED FLOW BOILING IN AN INCLINED RECTANGULAR CHANNEL
Ulu Jeong, Hanyang University, Republic of Korea; Hong Hyun Son, Hanyang University, Republic of Korea; Kyeong Boom Sung, Hanyang University, Republic of Korea; Kwang Hyuk Seo, Hanyang University, Republic of Korea; Juon 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 core 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 of the heat transfer on a gas-liquid mixing layer containing two studs in a line. A test section scaled down by one-forth has been designed. The experimental conditions adopted are mass fluxes (50-300 kg/m²s) and subcooling of 5-20K at the channel with dimensions of 30x118x215mm using indirectly heated flat plate, made of SS316, with active heated area of 107.5x215 mm. With these experiments, effect of the aligned studs on 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 CHF is lower than the bare CHF. The resulting stagnant region of the vapor between the aligned studs is found to cause the significantly decreased CHF.

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; Hacal 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 are off-center in rectangular blocks, were carried out under high heat fluxes up to the ITR 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 inclination heat fluxes on heat transfer were all investigated. The main attention was 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 be 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 commercial Fluid Dynamics (CFD) method to obtain, which were consistent with the experimental results.

Keywords: heat transfer, swirl tape, nonuniform heat flux, ITER
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; Hongkai Chang, China University of Petroleum,Beijing, China; Jing Gong, China University of Petroleum Beijing, China

The deformation behavior between oil-in-water emulsion drops along a liquid flow 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 monolaurate, 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 effectively reflect the deformation phenomenon. The range of the lowest increase in force 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 coalescence 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 VISCOS COMPRESSIONAL FLOW: FLAT AND TOROIDAL DROPS

Avinoaam 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 with an overheating problem.

*Supported by Israel Science Foundation Grant 327/14

CONVEXTIVE LOCAL HEAT TRANSFER AROUND ELONGATED BUBBLES PROPAGATING IN DOWNWARD LIQUID PIPE FLOW

Adam Freytag, Tel-aviv University, Israel; Devora 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 study the effect of liquid property on 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) and an intermediate case with a slower upward bubble velocity and a nose that is distorted and shifted away from the pipe. The first two regimes are unstable; initially the 3rd 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 bubble shapes and corresponding bubbles are shown to have a considerable effect on the flow field in the bubble’s wake and thus on the convective heat transfer rate. Thermoanometry 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 (Ims, Kogakuin University, Japan; Hiyasu Chikato, 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 energy under grant number 1501473A on the basis of a decision by the German Bundestag.

ENERGY UNDER GRANT NUMBER 1501473A ON THE BASIS OF A DECISION BY THE GERMAN BUNDESTAG

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 with slots with high overhear pressure. The drops are stretched on their way through the orifice due to the extensional flow and depending on their size, they may be defomed to about prolate ellipsoid or even to threads. The local deformation depends on the flow 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 < Reg < 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 a range of 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 overpressure, 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.

HOMOGENIZATION: SIMPLIFIED DROP DEFORMATION ESTIMATE DURING TRANSITION THROUGH ORIFICES
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.

TWO WAY COUPLING IN PARTICLE-LADEN TURBULENT JET

Francesco Battista, Sapienza Università di Roma, Italy; Paolo Guatieri, Sapienza Università di Roma, Italy; Gior gia 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 dispersive phase eventually affects 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 trajectory at the solid-fluid interface 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-resolved, and profiles as a function of distance from the surface of solid particle-laden channel flow are analyzed at several levels: whole-field, phase-resolved, and profiles as a function of distance from the surface of solid

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 becomes 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.

UPWARD ANNULAR-MIST FLOW

Yuchen Wang, Tokyo University of Marine Science and Technology, Japan; Sanwel Andayi, Tokyo University of Marine Science and Technology, Japan; Tatsuya Hazuku, Tokyo University of Marine Science and Technology, Japan; Tomo Takamasa, Tokyo University of Marine Science and Technology, Japan

Accurate knowledge of flow characteristics of the entrained droplets in annular two-phase flow are required to confirm the validity of these simulations. In this study, measurements of droplet size and velocity in vertical upward annular-mist flow were performed using an imaging-processing based on a shadowgraph technique. The time-resolved 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 flow rates, varying from 20 to 40 m/s for gas phase and from 0.1 to 1.1 m/s for liquid phase. The measured droplet size distribution varied greatly with the 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.

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

MOLLEDING OF CHURN-ANNULAR FLOW

Jos Westende, TNO, The Netherlands; Pajman Shoebishomrani, 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 counter-current gas-liquid flow in maturing gas wells or to assist in removing downhole accumulated liquids. According to Ninnewegen (2015), who performed experiments with foam flows, foam accomplishes this, since the creation of foam postpones the transition from annular to churn flow 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 Ninnewegen. 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 solids concentration and compared with experimental data.

* Support from grant TKG01022 and participating partners is gratefully acknowledged.

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 becomes 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.

UPWARD ANNULAR-MIST FLOW

Yuchen Wang, Tokyo University of Marine Science and Technology, Japan; Sanwel Andayi, Tokyo University of Marine Science and Technology, Japan; Tatsuya Hazuku, Tokyo University of Marine Science and Technology, Japan; Tomo Takamasa, Tokyo University of Marine Science and Technology, Japan

Accurate knowledge of flow characteristics of the entrained droplets in annular two-phase flow are essential for proper thermal-hydraulic design and safety analysis of plants. While a majority of two-phase flow studies 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 becomes 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; Sanwel Andayi, Tokyo University of Marine Science and Technology, Japan; Tatsuya Hazuku, Tokyo University of Marine Science and Technology, Japan; Tomo Takamasa, Tokyo University of Marine Science and Technology, Japan

Accurate knowledge of flow characteristics of the entrained droplets in annular two-phase flow are essential for proper thermal-hydraulic design and safety analysis of plants. While a majority of two-phase flow studies 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 becomes 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; Sanwel Andayi, Tokyo University of Marine Science and Technology, Japan; Tatsuya Hazuku, Tokyo University of Marine Science and Technology, Japan; Tomo Takamasa, Tokyo University of Marine Science and Technology, Japan

Accurate knowledge of flow characteristics of the entrained droplets in annular two-phase flow are essential for proper thermal-hydraulic design and safety analysis of plants. While a majority of two-phase flow studies 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 becomes 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; Sanwel Andayi, Tokyo University of Marine Science and Technology, Japan; Tatsuya Hazuku, Tokyo University of Marine Science and Technology, Japan; Tomo Takamasa, Tokyo University of Marine Science and Technology, Japan

Accurate knowledge of flow characteristics of the entrained droplets in annular two-phase flow are essential for proper thermal-hydraulic design and safety analysis of plants. While a majority of two-phase flow studies 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 becomes 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.
IMPACT OF GAS VELOCITY PROFILE ON SIMULATED WAVE FREQUENCIES IN VERTICAL AIR-WATER CHURN FLOW
Matej Tekavc, Jožef Stefan Institute, Slovenia; Bocašan Koncar, Jožef Stefan Institute, Slovenia; Ivo Kliškan, Jožef Stefan Institute, Slovenia; Toshiyuki Sanada, Shizuoka University, Japan; Hiroaki Kusuno, Shizuoka University, Japan
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 lack of modeling of the gas flow, namely 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 interacting 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 Arrenti, 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 dynamic 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 open interfaces. For instance, high-fidelity simulations that successfully model the formation of droplets at the scale of the droplet and 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 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 forcing taken at the eigenfrequency 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 homogeneous turbulent flow by a forced oscillator. Our model requires the simulation of the dynamics time scales of the process, which are set by the 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.

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; Lyson 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 of liquid velocity at the bottom of the pipe. 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 by 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, Thémone, France; Frederic Risso, IMFT- CNRS/Université of 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 forcing taken at the eigenfrequency 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 homogeneous turbulent flow by a forced oscillator. Our model requires the simulation of the dynamics time scales of the process, which are set by the 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. 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 maintaining 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
Anatole Morente, EDF R&D / Institut de Mécanique des Fluides de Toulouse, France; Jerome Lavieille, EDF R-D, France; Dominique Legendre, Institut de Mécanique des Fluides de Toulouse, France
We present a L² 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 shaped in either one 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 against DNS approach.

SIMULATION OF BUBBLY FLOW IN AN INDUSTRIAL ELECTROCHEMICAL REACTOR
Mehdi Arjmand, AkzoNobel, Sweden; Bert Vreman, Akzo Nobel, The Netherlands; Albert Bakkers, AkzoNobel, The Netherlands; Johan Lij, AkzoNobel, Sweden; Kaile Pein, 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 tubes due to 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 μm), so the presence of strong electrolytes inhibits coalescence. It appears that for a realistic gas production, gas accumulates and no statistically stationary state is reached even if the bubble diameter is fixed. A proposed strategy could be to use a so-called airlift principle where the bubbles are added to the reactor at a constant rate, so that the basic state 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.

INFLUENCE OF CONVECTION IN TUBE BUNDLES DURING BOILING
Andrea Luke, University of Kassel, Germany; Joseph Addy, University of Kassel, Germany; Bjorn 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 convective effects, that are 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 in MIT’s Reactor Hydraulics Laboratory. The 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 in MIT’s Reactor Hydraulics Laboratory. The 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 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 in MIT’s Reactor Hydraulics Laboratory. The 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 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 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 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 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 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 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 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 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 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 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 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 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 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 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 in MI...
TRANSIENT MASS TRANSFER FROM A SPHERICAL DROPLET EVAPORATING IN GASEOUS ENVIRONMENT
Simona Toselli, Enrico Val clipsi, University of Bergamo, Italy; Gianpi eerto Elviro Cossali, University of studies of Bergamo, Italy
Analytic modelling of droplet 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 energetic derivation of problem of evaporation in a gaseous environment relaxing the above mentioned approximation. The spherical symmetric species conservation equation for vapour transport in a steady gas environment is developed in non-dimensional form. The solution allows to estimate 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.

SUPPRESSION MEASURES OF STEAM EXPLOSIONS WITH AND WITHOUT EXTERNAL PRESSURE PULSE
Masato 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 degree at the target pressure. The effective concentration became denser when large share stress and/or external force act on the vapor film.

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 Malli, 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 wettabiliy restrains condensation in hydrophilic regime for convex spherical substrate and in hydrophobic regime for concave spherical substrate.

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; Hiroshi 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 flow in the bearing for a rocket turbopump. Four types of 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.

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 waves 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 estimated.

BECKER-DÖRING MODEL FOR NON-EQUILIBRIUM STEAM CONDENSATION IN A SUPERSORPSON NOZZLE
Rob Hagemjeier, University Twente, The Netherlands; Edwin Van Der Weide, University of Twente, The Netherlands; Vitaly Kalkamov, 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. The largest droplets typically consist of ten million molecules and the evaporation rate during this time lapse is expected to be much higher than that under steady conditions. The present work is aimed to define the energetic derivation of problem of evaporation in a gaseous environment relaxing the above mentioned approximation. The spherical symmetric species conservation equation for vapour transport in a steady gas environment is developed in non-dimensional form. The solution allows to estimate 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.
NUMERICAL STUDY OF WATER DROPS BEHAVIOUR IN OIL UNDER ELECTRIC FIELD
Biao Chu, Xiyan Jiaodong University, China; Meng Huang, Xiyan Jiaodong 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 increase of the particle diameter is smaller than the viscous length scale. 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 Nemés, 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 small 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 classes 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 Lange, LMFA, France; Pascal Alix, IFF Energies nouvelles, France; Christine Dalimazzone, IFF Energies nouvelles, France; Yacine Haroun, IFF 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. Prior, 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 free-surface level-set) was 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 and experimental results of fingering instabilities.

NUMERICAL SIMULATIONS ON COPPER DROPLET COLLISIONS IN THE SLAG CLEANING PROCESS
Hsong-Kun Yang, Forschungszentrum Jülich, Germany; Jorg Wolters, Forschungszentrum Jülich, Germany; Philipp Plischke, 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 consists of 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 developed, particle-in-fluid (PIF) model for the copper drop and turbulent shear flow. The trajectories of the particles in two-phase flow were simulated and validated by visual experiments. The most commonly used stochastic algorithm. It provides better predictions of the collision probability among the incompletely distributed droplets. The outcome of collisions in the numerical simulations was firstly determined by the current 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 LMTECH 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 the 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 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. It is observed 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, width of the 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 disturb the control of the satellite. As accelerations are smaller compared to capillary forces, testonical 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 for this 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.
LEB-DEM SIMULATION OF GAS-SOLID TWO-PHASE IMPINGING STREAM
Wen and party. 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 current air stream, focusing on the large scales instabilities. Experiments were performed over a wide range of liquid (0.1-0.5m/s) and gas (10-150m/s) velocities and for various liquid jet diameters (5 to 20mm) 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. sinusoidal or vortice) 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; Roney Brito, SINMEC / Federal University of Santa Catarina, Brazil; Clovis Malta, 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 PVIV measurements be done under repeatable conditions and the deviation of the averaged field. The laser diode photodiode 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-Behaddi, University of Nottingham, UK; David Hargreaves, Faculty of Engineering, University of Nottingham, UK; Barry James Azopardi, University of Nottingham, UK; Barry James Azopardi, 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 now being performed 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 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 a Dirichlet condition 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 axis of the two static mixers, the foam-like porous structure is found to be well represented by the Sauter mean diameter. A correlation is proposed correlating the droplet size as a function of Weber 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 dispersive 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.

DROPLET SIZE MEASUREMENT IN STATIX MIXERS USING REFRACTIVE INDEX MATCHING AND LASER-INDUCED FLUORESCENCE
Philipp Rudolf von Rohr, ETH Zurich, Switzerland; Richard Hafeli, ETH Zurich, Switzerland; Oliver Ruegg, ETH Zurich, 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 of the internal structures using laser-induced fluorescent. 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 number 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 dispersive 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.

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 Azopardi, University of Nottingham, UK; Buddhika Hewakandamby, University of Nottingham, UK; Yousuf Abub, 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=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 thickness, wall 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 characterized as coherent rings, rates or small diameter pipes. Many studies are 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.

ICMF 2016 International Conference on Multiphase Flow
Firenze, Italy, May 22 - 27, 2016
VOID FRACTION CHARACTERIZATION THROUGH AN OPTICAL METHOD FOR AIR-WATER UPWARD FLOW ACROSS TUBE BUNDLES

Fábio Toshiro 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 was 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 Ranen, 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 progress towards 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 account for interfacial 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. 201510143) are gratefully acknowledged.

COMPUTATIONALLY-EFFICIENT SIMULATION OF INDUSTRIAL BUBBLE COLUMN BIOREACTORS: INVESTIGATION OF SCALE AND MIXING EFFECTS

David Fletcher, University of Sydney, Australia; Dale Moclure, 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 570 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

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 simulate the behavior of cryogenic liquids model fluids like silicone oil or methyl-pentane are used. 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 – 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.

* Supported by National Research Foundation of Korea Grant No. NRF-2014R1A2A1A11049599.
NUMERICAL SIMULATION OF A HIGHLY-VISCOS 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 expelling particle-phases agrees well with the predicted velocity. 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-ε type closure with a buoyancy modification term is used to account for the turbulence. A system of particles 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 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 expelling particle-phases using the drag coefficients. The inter-particle collision among the same particle sizes is considered as the additional viscous effect of the particle phase. 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 runout 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.

* Support from project TRUMA (Treated Urban Micropollutants and Pharmaceuticals in Wastewaters) being a part of ACQUEAU – the Eureka Cluster for Water is gratefully acknowledged.

ANALYSIS OF INTERMITTENT FLOW PATTERNS OF WATER-STEAM FLOW IN SOLAR THERMAL POWER PLANTS WITH DOUBLE-PHASE 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

Karl Swiderski, ETH Zurich, Switzerland; Bruno Heiner, 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. The model is a finite volume approach 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 TRUMA (Treated 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 BUBBLE JET FLOW

Aroa Aouadi, National Engineering School of Tunis, Tunisia; Ghazi Bellahal, 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 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 asymmetric 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 jet core. Interphase force on the vertical relative velocity.
THE EFFECT OF THE CHANNEL INCLINATION ON FLOW PATTERN CHARACTERISTICS DURING GAS-LIQUID FLOW IN A SMALL HORIZONTAL CHANNEL

Francisco Loyola, University of Magallanes, Chile; Gerhardt 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 x 6.5 mm². The experiments were run for inclinations channel from varying from +90° to −90°. 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 kg/m²s, corresponding to gas and liquid superficial velocities from 0.03 to 19.4 m/s and from 0.1 to 0.76 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.

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 the potential of using a high resolution camera system (time: >30k fps; space: macro lens). A low energy consumption, high efficiency system has been developed and tested 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 improved accuracy of the predicted correlation under two-phase flow region.

VISUALIZATION AND HEAT TRANSFER OF HIGH PRESSURE CARBON DIOXIDE IN A SMALL HORIZONTAL TUBE

Haical Lyu, Xi’an Jiaotong University, China; Qincheng Bi, 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 kg/m²s, and the pressure is 6MPa to 8.5MPa, which includes subcritical and supercritical conditions. The forced convection and two-phase flow 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 high pressure. To get a clearer understanding of two-phase flow, a high resolution 4k glass tube was selected. A high definition recorder of sony HXR-N70C was used as recording equipment. Two metal layers 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 mass loading of 1.6 to 5.6, yield 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 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 Bugliarello, University of Rome “La Sapienza”, Italy; Leonardo Bragelli, Guglielmo Marconi University, Italy; Luca Saraceno, ENEA, Italy; Giuseppe Zummo, ENEA, Italy

The present paper concerns an experimental investigation of the effect of channel inclination and rotation on the two-phase flow characteristics for air-water flow in a 4.0 mm tube at micro-gravity conditions. The experiments at low gravity were performed during the parabolic flight campaign of October-November 2013. The 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 PC-72, widely used in parabolic flights. The observed flow patterns at low gravity varied from mass loading of 1.6 to 5.6, yield efficiency as high as 93% for a first-stage compression cycle. Flow 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, noise 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.

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 superposition by hydrodynamic effects (kinetic energy, oscillation) have a linear trajectories, and absence of deformation in comparison to those observed 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.

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 Bugliarello, University of Rome “La Sapienza”, Italy; Leonardo Bragelli, Guglielmo Marconi University, Italy; Luca Saraceno, ENEA, Italy; Giuseppe Zummo, ENEA, Italy

The present paper concerns an experimental investigation of the effect of channel inclination and rotation on the two-phase flow characteristics for air-water flow in a 4.0 mm tube at micro-gravity conditions. The experiments at low gravity were performed during the parabolic flight campaign of October-November 2013. The 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 PC-72, widely used in parabolic flights. The observed flow patterns at low gravity varied from mass loading of 1.6 to 5.6, yield efficiency as high as 93% for a first-stage compression cycle. Flow 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, noise 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 COALESCEANCE – 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: >30k 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 (Cl-, NO3-, SO4- und 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 AND ARRAY
Ayumu Watanabe, University of Tsukuba, Japan; Soma Watahiki, University of Tsukuba, Japan; Motonori Nwa, 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 Capeceletto, 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 uncoiled 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 transported to the solid-phase by the particle inertia. We 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 Lauder, 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 (<5-100 μm) 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 (>10 μm) and temporal resolution (~200 μs), yielding high quality statistics of velocity and acceleration over O(107) 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 OSCILLATORY 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 (KIT), 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 γ=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(2x105) particles arranged in 13-15 layers. DNSs reproduced two of the experiments performed by Blondeaux et al., 1986. 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; Zhaoqiu Liu, Xian Jiaotong University, China; Chuguang Zheng, Huazhong University Of Science and Technology, China
The subgrid-scale motions relevant to the particle pairs dynamics represent a great challenge to the LES of turbulent particle-laden flows. In order to address this challenge, we explore the capability of DNS (filtered DNS) to predict the collision-related statistics such as the radial distribution function g(r), the radial relative velocity wr 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 the 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 Kawai, 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 fine 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-averaged 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; Emilia Mestas, SINTEF, 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.

AN EXPERIMENTAL INVESTIGATION OF INTERFACE SHAPE IN SEPARATED FLOW REGIMES INSIDE THE INCLINED PLATE
Taehwan Ahn, Pusan National University, Republic of Korea; Byongyu 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° to 30°. An atmospheric air-water was used at superficial velocities of 0.05~0.7 m/s for gas, and 0.01~1.0 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; Hiyoriaki 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. MgSO4 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, the rising velocity of the inclined flat inclined 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 (cm) bubble-bubble interaction quantitatively. The results shows that coagulations of two close bubble configuration frequently appear, the horizontally-oriented one being stable compared to the vertically-oriented one. A qualitative tendency of the bubble configuration is confirmed to be the same as the bubble-bubble interaction. However, quantitative differences occur due to the difference in the wake structure between these two conditions.

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 Pts 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 two 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. The tube wall was made of steel 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 wall 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.

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 nanomicrobubbles 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 solved using the fast multipole method. This model has been presented in previous ICMF papers. In this work, the influences of the characteristics of the tube wall 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.

INTERACTION OF TWO BUBBLES RISING IN A THIN-GAP CELL: EXPERIMENTS AND MODEL FOR THE VERTICAL ENTRAINMENT
Veronique Roig, INP Toulouse, France; Audrey Fillola, 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 HP IV, 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.

* Supports from JSPS KAKENHI Grant Number 15K13876 are gratefully acknowledged
ADHESION AND SLIDING DYNAMICS OF AIR BUBBLES ON SUPERAEROPHILIC SURFACES
Rebecca Spriestersbach, 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 silicon substrate and a micropillar 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 using a captive bubble method. Inclination to the substrate and a sliding motion is applied to 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 buoyancy force and pressure force. The results indicate that the sliding angle, contact angle hysteresis, adhesion force are much lower for the micropillared superaerophobic than for the aerophobic surface, which is mainly due to the entrapped water layer into the micropillars. It is observed that the former implies a water resistance for the bubble de-adhesion and the latter 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 DOWNSSTREAM GAS SPARGERS OF DIFFERENT ORIFICE PATTERN IN BUBBLE COLUMNS
Felix Moeller, Helmholtz-Zentrum Dresden-Rossendorf (HZDR), Germany; Andra Bieneke, 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; Felix Moeller, Helmholtz-Zentrum Dresden-Rossendorf (HZDR), 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 buoyancy and drag forces, in turn, depending on bubble size, control the bubble traveling velocity, and eventually the overall hydraulic behavior inside the bubble column and gas-liquid contact. 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 m/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 Rosnouchkina, 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 unusually difficult due to the lack of experimental feasibility, 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 a better understanding of the interaction of the bubbles, an experiment was performed in a cuboid vessel 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 m/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.

EXPERIMENTAL INVESTIGATION OF HEAT TRANSPORT IN BUBBLY FLOW
Biljana Govzic, 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
Bubbles 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, connected to the depressurized gas-liquid transport. 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 buoyancy and drag forces, in turn, depending on bubble size, control the bubble traveling velocity, and eventually the overall hydraulic behavior inside the bubble column and gas-liquid contact. 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 m/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.

ON THE FORMATION OF MESO-SCALE STRUCTURES IN DISPERSED GAS-LIQUID FLOWS
Henrik Strom, Chalmers University of Technology, Sweden; Klas Jareleg, Chalmers University of Technology, Sweden; Srijan Sastik, 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 overall bubble volume fraction fields. 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 is often interpreted 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 Dendi, LGI -Université de Grenoble, France; Jean-Philippe Matas, Université Claude Bernard Lyon and CNRS, France; Pierre Boivin, M2P, France; Alain Carteller, 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 Tsat, the vaporization rate and resulting drop sizes were independent of the reservoir pressure, and hence of the superheat (Desnouss 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 several 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 OF FLUID CO2

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 GWP and GWP of 1). Effort has been made to develop a better correlation for flow boiling heat transfer with CO2 as working fluid. As an initial step, comparative study has been carried out with 3425 experimental data collected from 12 research papers pertaining to CO2 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 CO2. 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 Mickel, 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 µm x 116 µm. 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.

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 periodic 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. K-han Kim.

EXPERIMENTAL AND NUMERICAL ANALYSIS OF FLASH ATOMIZATION WITH CYROGENIC FLUIDS

Mohamed Awad, Mansoura University, Egypt; Zen 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, Kandikar first number, Kandikar second number, Kandikar 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 in jet 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 effects of initial pressure and temperature (or superheat level) will be compared to the numerical results. 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 periodic 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. K-han Kim.

A LOOK ON RELEVANT NONDIMENSIONAL NUMBERS FOR FLOW BOILING IN MICROCHANNELS AND MINICHANNLES

Mohamed Awad, Mansoura University, Egypt; Zen 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, Kandikar first number, Kandikar second number, Kandikar 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 in jet 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 effects of initial pressure and temperature (or superheat level) will be compared to the numerical results. 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 periodic 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. K-han Kim.

ICMF 2016 International Conference on Multiphase Flow
Firenze, Italy, May 22 - 27, 2016
JATROPHA BIO DIESEL PRODUCTION USING HYDRODYNAMIC CAVITATION

Prateek Ghosh, 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 biomedial 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 are non-spherical, which increases the importance of lift forces and hydrodynamic interactions. The whole method aims at remaining with a penalization method in a similar way as immersed boundary methods. 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 transition from single particle simulations to analyzing a single highly elongated particle in fluid flow. A highly parallel lattice Boltzmann solver is used to perform direct numerical flow simulations. 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 mult-particle simulations will be performed to analyze the influence of solids volume fraction on the drag, lift and torque coefficients.

EXPERIMENTAL INVESTIGATION OF THE LORENTZ-FORCE-DRIVEN CONVECTION AROUND A GROWING HYDROGEN BUBBLE

Dominik Baczymski, Universitaet der Bundeswehr, München, Germany; Franziska Karinbach, IFW Dresden, Germany; Xuegeng Yang, Technische Universität Dresden, Germany; Gerd Mutschke, Technische Universität Dresden, Germany; Margitta Uhlmann, 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.

CHARACTERIZATION OF FLOC MORPHOLOGY IN A TURBULENT TAYLOR-COUETTE REACTOR

Lea Guerin, Chemical Engineering Laboratory, France; Carole Courdot Sauvajaud, 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 alternatively 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.

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.

NUMERICAL STUDY OF AGGREGATION DYNAMICS WITH A LATTICE BOLTZMANN METHOD COUPLED WITH DISCRETE ELEMENTS THROUGH FLOW PENALIZATION

Jeff Ngoma, Université de Lorraine, France; Jean-Sebastian 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 solid-liquid 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. Parallel 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.

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
Abdelwahid Azizi, University of Sciences and Technology Houari Boumedien, Algeria; Ammar Zeghioui, University of Sciences and Technology Houari Boumieden, Algeria; Faiza Saidi, University of Sciences and Technology Houari Boumieden, Algeria; Abdelkader Messlem, University of Sciences and Technology Houari Boumedi, Algeria; Barry James Azzopardi, University of Nottingham, 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 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 solver is used to simulate the fluid dynamics both in 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 fluid dynamics and is independent of the flow patterns upstream of the bend. (2) the method predicts well the capillary force effect on the linear translation. The method considers the effect of gravity on the droplet motion 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.

MULTI-MATERIAL MODELLING FOR COMPRESSIBLE FLOWS WITH THE CONSERVATIVE SHARP-INTERFACE METHOD
Universiti Pan, Technische Universiteit München, Germany; Xiangyu Hu, Technische Universität München, Germany; Nikolaus Adams, Technische Universität München, 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 including three or more materials, most of these methods have difficulties on handling interface singularities, where three or more materials meet. In this work, we propose a multi-material scheme 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 material interfaces. In order to force high-order finite-difference schemes, such as WENO schemes, near singularities, a simple, local and fast phase-reconstruction operator is developed. Furthermore, a new 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 presented approach, accurate and computationally significantly more efficient than previously proposed methods.

PRESSURE DROP THROUGH ORIFICES FOR SINGLE- AND TWO-PHASE VERTICALLY UPWARDS FLOW
Abdelwahid Azizi, University of Sciences and Technology Houari Boumieden, Algeria; Ammar Zeghioui, University of Sciences and Technology Houari Boumieden, Algeria; Faiza Saidi, University of Sciences and Technology Houari Boumieden, Algeria; Abdelkader Messlem, University of Sciences and Technology Houari Boumieden, 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 model. 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 orifice thickness. It increases with the superficial velocity of the phase. The correlations of Hidditch and Akhtar et al. correlate well the orifice pressure drop coefficient. For single-phase flow, an increase 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.

SOLID VOLUME FRACTION FLUCTUATIONS IN A LIQUID FLUIDIZED BED
Maedehe 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 with at different axial postions 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, UMF, 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 in the bed. 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, CHESti, 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), in which 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. Gidaspov, 1934).

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; Buddhika Hekawandramby, 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 second-order horizontal to vertical 90° bend and downstream. Air-Silicone oil (viscosity 5 cP, density 930 kg/m3) 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 flow rates: A (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 structure and controlling the phase distribution of the 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.
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.

NUMERICAL OPTIMIZATION OF THE PHASE INTERFACE IN HIGH-PRESSURE DIE CASTING
Markus Frings, RWTH Aachen University, Germany; Marek Behr, 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, which reproduces the leak flow in the containment vessel, 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. 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 investigations 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.

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 agglomeration of inclusions 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 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.

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; Shusicho Mwa, Hokkaido University, Japan
This study aims at accurately observing spreading behaviors of molten-core liquid that reproduces the leak flow in the containment vessel, which emerged 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 the 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-UT experiment and the present experiments using CFD code. The bottom crust 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.

FLOW USING GAS VOLUME FRACTION FLUCTUATION CHARACTERIZATION OF DYNAMICS OF THREE-PHASE SLURRY FLOW

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; Young-Bae Kang, POSTECH, Republic of Korea
Computational modeling of turbulent fluid flow is 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-ω 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.

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 Trnike, University of Kentucky, USA; Jose Granadeiro 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 hundreds 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 entrainment, 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 entrainment 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 entrainment 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.

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; Dmitry 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 regimes 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

Chaos 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. In this study we simulate surface measuring 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.

DYNAMICS OF VERTICAL FALLING FILMS – AN EXPERIMENTAL AND NUMERICAL STUDY
Anders Akesjo, Chalmers University of Technology, Sweden; Mathias Gourdon, Chalmers University of Technology, Sweden; Lennart Vanli, Chalmers University of Technology, Sweden; Srdjan Sasic, Chalmers University of Technology, Sweden

Vertical falling liquid films are found in a large number off 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 surface measuring Volume of Fluid (VoF) method. We 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.

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 Chaheed, 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 Chaheed 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 reduces the shear stress. On the basis 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 modification of the shear 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 cavities 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.

EULER-REACTION MODELING OF 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

Chaos simulations of dispersed bubbly flow on the scale of technical equipment are feasible within the Eulerian two-fluid framework of interpenetrating continua. 

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 welled 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, 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.

DNS ASSISTED MODELING OF BUBBLY FLOWS
Greter Tryggvason, The University of Notre Dame, USA; Jiacai 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 an 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**

Solenne Fleau, EDF R&D, France; Stephane Mounia, 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 multi-field 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 mass transfer, which is a key parameter for the 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.

**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; DongHoo Kim, Republic of Korea University, Republic of Korea; Hansook Min, 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 channels. The design of the fluid model includes a channel cross-sectional area 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.

**MASS TRANSFER FROM SINGLE CARBON DIOXIDE BUBBLES IN WATER-ALCOHOL SOLUTIONS IN A VERTICAL PIPING**

Yohei Hori, Kobe University, Japan; Jiro Aoki, Kobe University, Japan; Kosuke Hayashi, Kobe University, Japan; Shigeo Hosokawa, Kobe University, Japan; Akio Taniyama, 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, kl, does not change at all even by further increase of surfactant concentration. The kl of contaminated ellipsoidal bubbles are smaller than those of clean ellipsoidal bubbles. The reduction of kl 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 kl 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 - LMTA, 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 numbers 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.

**EXPERIMENTAL INVESTIGATION OF TAYLOR BUBBLE DISSOLUTION IN MILLI-CHANNELS**

Mohammadreza Haghighatdar, 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 1 mm hydraulic diameter. The bubbles were held stationary in the downflowing 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 for comparison with other optical techniques. The results show that the long term dissolution of single CO2 bubbles show that the dissolution curve 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, kl in two curves. For square channel, kl increases with decrease of bubble sizes, while no obvious trend could be detected for the circular channel.

**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-2s-1, and qualities from 0 to 0.9. The apparatus and data reduction technique were validated by comparing results to volume heat face area and length of the bubble and were utilized to obtain the local 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 for comparison with other optical techniques. The results show that the long term dissolution of single CO2 bubbles show that the dissolution curve 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, kl in two curves. For square channel, kl increases with decrease of bubble sizes, while no obvious trend could be detected for the circular channel.
ANISOTROPIC TURBULENT CLUSTERING OF CLOUD DROPLETS SETTLING IN ISOTROPIC TURBULENCE
Kengo Matsuoka, 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 droplets on the anisotropic structure of turbulent clustering, 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.

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 on a very hot surface (3000 °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 and compared to the anisotropic structure of turbulent clustering. The data collected for the same droplet on the anisotropic structure of the solid-liquid interaction regime. The particle Image Velocimetry (PIV) and Particle Tracking Velocimetry (PTV) methods were used to perform the velocity measurements. The results showed 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.

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 turbulent, 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 × 10−4 to 2 × 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.

MACROSCOPIC CHARACTERISTICS OF HEAVY PARTICLE RESUSPENSION OBTAINED FROM DIRECT NUMERICAL SIMULATIONS OF PRESSURE DRIVEN CHANNEL FLOW
Stergios Yiantisios, Aristotle University of Thessaloniki, Greece; Panagiotis Saliakellas, 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 Discretized Lagrange Mupltiparticle/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 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 was a presentation of a new macroscopic equations derived from the suspension and particle phase momentum equations could describe the average equilibrium distribution and the flow of both phases.
MEASUREMENT OF LIQUID DROPLETS IN ANNULAR-MIST FLOW USING SHADOWGRAPH TECHNIQUE
Sanam Andayi, Tokyo University of Marine Science and Technology, Japan; Yuchen Wang, Tokyo University of Marine Science and Technology, Japan; Tatsuya Hazukus, 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.

TURBULENCE CONTRIBUTION IN MOMENTUM INTERFACIAL TRANSFER IN AN UPWARD LIQUID-LIQUID PIPE FLOW
Mariem 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 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.

THE EFFECT OF PHYSICAL PROPERTIES ON CHURN FLOW IN A LARGER DIAMETER RISER
Garaev Damir, University of Nottingham, UK; Josep Escrib 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 800 and a viscosity=5 times that of water. The experiments have been conducted at a liquid superficial velocity = 0.5 m/s and on conductivity 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 paper.
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 10 M = -6.0 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 the Eulerian domain moments from a to the equation of 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., 1995) 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 ~ 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 Wachen, Imperial College London, UK; Marco Pivello, Federal University of Uberlândia, Brazil

The Lagrangian front-tracking 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 is performed by domain moments from a to the equation of 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., 1995) 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 ~ 9, where Eo is the Eötvös number and E the aspect ratio.

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; Sebastien Cazin, Institut de Mécanique des Fluides de Toulouse, France; Catherine Colín, INP Toulouse, France

In the context of drag reduction by bubble injection applied to ships hulls, bubbles are generated from a vertical air hole on the top side of a hydrofoil. The trajectories and shapes of bubbles are obtained by using two high-speed video cameras, and the bubble images are analyzed by a software tool to evaluate the bubble generated by the device. The present study focuses on the effect of injection rate on the bubble dispersion and trajectories measured in a turbulent boundary layer, spatially evolving under a flat plate. The Reynolds number Re was varied from 780 to 3000. Bubbles, injected at the wall, were between 30 to 250 wall units in diameter (d). The Weber number (W) and the dimensionless frequency (f/d) were 1000 and 0.5 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 d/Re. Bubbles are investigated for different external velocities and streamwise locations.

FLOW PAST A BUBBLE-GENERATING HYDROFOIL FOR SHIP DRAG REDUCTION
Ichiro Kumagai, Meisei University, Japan; Takahiro Tsukahara, Meisei University, Japan; Yuichi Murali, 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 succeed 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. Thubble 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 open test section. 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.

THE INTERACTION OF A HIGH-RE MODERATE-WE BUBBLE WITH AN INCLINED WALL
Christopher 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 observe three different behaviors are observed. For all cases, the bubbles initially collide repeatedly with the wall. For small wall inclinations, after a brief period of oscillation 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.

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 inter-penetrating continua and an equation of 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., 1995) 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 ~ 9, where Eo is the Eötvös number and E the aspect ratio.

PLOW PAST A BUBBLE-GENERATING HYDROFOIL FOR SHIP DRAG REDUCTION
Ichiro Kumagai, Meisei University, Japan; Takahiro Tsukahara, Meisei University, Japan; Yuichi Murali, 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 succeed 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. Thubble 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 open test section. 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
Sajjad Valizadeh, 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 Surface for annular and intermittent flow evolution. The ensemble-averaged flow fields 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 breakup 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.

VARIATIONS IN GEOMETRIC VARIABLES OF WATER DROPLET ON LOW-SURFACE-ENERGY SOLIDS
Yukitiro Yonemitsu, 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 transfer is essential. The evaporation process of a 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 increases, 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 phases. In the regular solution model and Floer's 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 is used for combinatorial enthalpy 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 the Key National Science and Technology Specific Project is gratefully acknowledged.

STRATIFICATION OF INTERMITTENT AND ANNULAR R245-FA FLOW BOILING IN HORIZONTAL ORIENTATION
Thibaud Layssac, Centre d'énergie 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 position subsequently. The ensemble-averaged flow fields and momentum transport equations are coupled with population balance 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 breakup 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.

A MULTISCALE APPROACH TOWARDS IN-TUBE CONVECTIVE BOILING FLOW SIMULATION
Miau Yazdani, United Technologies Research Center, USA; Abbas Alahyari, United Technologies Research Center, USA; Thomas Radloff, United Technologies Research Center, USA; Marios Soteriou, United Technologies Research Center, USA

Conductive 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.

PREDICTION OF SOLID-LIQUID EQUILIBRIUM IN PARAFFINIC SYSTEMS WITH AN IMPROVED ACTIVITY COEFFICIENT METHOD FOR THE NON-IDEALITY OF SOLID PHASE
Juheong Yang, China University of Petroleum, Beijing, China; Wei Wang, China University of Petroleum Beijing, Beijing, China; Pan Song, China University of Petroleum, Beijing, China; Qianli Ma, China University of Petroleum, Beijing, China; Jing Gou, China University of Petroleum, Beijing, China; Xingyi Qu, 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 liquids, the regular solution model and Floer's 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 is used for combinatorial enthalpy 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.

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/liquid 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 different nose shapes of projectile, and evaluate its influence on the cavity evolution. The cavitation radius and drag coefficient are calculated and compared with experimental results. It is observed that the blunt 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 blunt nose is bent more obliquely and the cavity spreads along the fluid flow.
**INFLUENCE OF SORPTION EFFECTS ON THE EFFICIENCY OF MULTIPHASE CONVEYANCE**
Stefan Werner, 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 gas flows. The agitating phenomena within the first chamber is caused by the depression and the rotational and shear forces between the screws and the saturated liquid. The gas feeding 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, the model of the conveying chamber of the multiphase pump is designed. The screws are represented by rotating discs with similar surface microstructure. The gas flows as function of rotation mixed; thermal physical properties of the saturated fluid and the microstructure are 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 Chatarpampous, 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 770-280 microns and spherical particles 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 Vrman, AKZO Nobel, The Netherlands

Direct numerical simulation of particle-laden turbulent channel flow at relatively high Reynolds number (Re =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 flow problems, 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. Utilizing mono-dispersed solid particles of diameter dp=10, 20 and 40µm, the Stokess number based on the pipe diameter, Sk0, was varied across two orders of magnitude between 0.35Sk0=22.4. The resultant Reynolds number was in the range 10,000≤Re≤54,000. The degree of clustering was measured utilizing the widely-used Euclidean distance density 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 Sk0≤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. US0034) is gratefully acknowledged.

AN ACCURATE METHOD TO TAKE INTO ACCOUNT LUBRICATION FORCES IN NUMERICAL SIMULATION OF SUSPENSIONS.

Aline Lefebvre Lepot, CNRS, Ecole Polytechnique, université Paris-Saclay, France; Guillaume Chivot, Université Paris-Sud, France; Mathieu Grandin, École Polytechnique, CNRS, Université Paris Saclay, France; Loïc Guoarun, CNRS, France; Benoît Merlet, Université 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 tackled using the boundary element method (BEM). In the present contribution, we focus on a new method that is an extension of the single phase incompressible Navier-Stokes equations. It is based on an explicit expansion of the total lubrication forces exerted on pairs of particles and on a pairwise additivity hypothesis. In this contribution, we use a new pre-treated lubrication force, 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.

THE SPARSE CARDINAL SINE DECOMPOSITION APPLIED TO STOKES INTEGRAL EQUATIONS

François Mouges, Ecole Polytechnique, CNRS, université Paris-Saclay, France; Matthieu Aussal, Ecole Polytechnique, France; Aline Lefebvre Lepot, CNRS, Ecole Polytechnique, université Paris-Saclay, France; Franck Pigeonneau, UMR 125 CNRS/Saint-Gobain, France; Antoine Sellier, Ecole Polytechnique, CNRS, université 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 or 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 expansive 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 conditions with 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.

NUMERICAL SIMULATIONS OF VISCIOUS AND COMPRESSIBLE TWO-PHASE FLOWS USING HIGH-ORDER SCHEMES

Marion Capuano, Ecole Centrale de Lyon, France; Christophe Bogey, Ecole Centrale de Lyon, France; Peter Spell, Laboratoire de Mécanique 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 correction 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 mechanism 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 short discontinuities, whilst accounting for viscous effects.

NON-CONSERVATIVE PRESSURE-BASED COMPRESSIBLE FORMULATION FOR MULTIPHASE FLOWS WITH HEAT AND MASS TRANSFER

Mathieu Laboix, ASCOMP AG, Switzerland; Chidambaram Narayanam, 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 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; Buddhika 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.
THIN FILM THICKNESS MEASUREMENTS USING ULTRASONIC PULSE ECHO TECHNIQUE

Yousaf Alaufi, University of Nottingham, UK; Nicholas Watson, University of Nottingham, UK; Barbara James Azzopardi, University of Nottingham, UK; Buddhika 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 <8 mm in a vertical pipe. Initial benchtop experiments were performed and different techniques were developed. For film thickness <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 reservoir film thickness at incline 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 Charagiannis, Imperial College London, UK; Fabian Denner, Imperial College London, UK; Berend Van Wachem, Imperial College London, UK; Saraflim Karakoc, 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 = 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 was evaluated using velocity profiles. The time-averaged velocity fields 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 obtained from 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 grant EP/1006041/1, EP/K008595/1 and EP/M021556/1 is gratefully acknowledged.

REduced ORDER MODELLING OF LIQUid-GAS Channel FLOWS

Giuliana 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, as the absorption units for 

NUMERICAL AND EXPERIMENTAL INVESTIGATION OF LIQUID FILM FLOWS ON PACKINGS IN ABSORBERS FOR POST-COMBUSTION CO2 CAPTURE

Yoshiyuki Iso, IHI Corporation, Japan; Ryoaake Ikeeda, IHI Corporation, Japan; Shinya 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 CO2 Capture (PCC) is one of the economical and efficient technologies for reducing CO2 emissions. Gas-liquid interfacial flows, such as gas-liquid liquid solvent, are applied in CO2 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 pressure drop. Our advanced packing has approximately twice the absorption amount.

* Support from EPSRC Grant No. EP/M001482/1 is gratefully acknowledged.
MASSIVE PARALLEL DIRECT NUMERICAL SIMULATIONS OF THREE-DIMENSIONAL JET FLOWS
Lyes Kehouadi, Imperial College London, UK; Omar K. Matar, Imperial College London, UK; Jael Cherqui, LMSI-CNRS, France; Damir Juric, LMSI-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, 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 version of the Benard 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 trend. The results arising from the model are corroborated also by Navier-Stokes simulations of merging sheared two-current flows.

RATIONAL RELAXATIONS AND RELATED NUMERICAL METHODS FOR THE SIMULATION OF SEPARATED AND DISPERSED PHASES
Florence Driu, CEA, France; Marc Massot, CentraleSupélec, EM2C, France; Samuel Kohk, CEA, France; Adam Laret, 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.

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 Peschi, Technische Universität Darmstadt, Germany; Holger Marschall, Technische Universität Darmstadt, Germany; Kosuke Hayashi, Kobe University, Japan; Michael Schlueuter, 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 on 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. Towards this end, 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 CO2 and a special fluorescent dye to visualize the local dissolved CO2 concentration is applied as well as Triton X-100 as surfactant.

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, Universität 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.
HYDRODYNAMICS AND MASS TRANSFER PROPERTIES OF A ROTATING BUBBLE COLUMN
Ying Ma, Liu, Helmholtz-Zentrum Dresden-Rossendorf (HZDR), Germany; Markus Schubert, Helmholtz-Zentrum Dresden-Rossendorf (HZDR), Germany; Ary Mitomori Dashiborun, Laval University, Canada; Markus Schubert, Helmholtz-Zentrum Dresden-Rossendorf (HZDR), Germany

**Bubbles 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 impinging oscillations. Continuous rotation 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. It is shown that rotation schemes modify the gas-liquid dispersion pattern and the process of mass transfer.**

*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)*

THE SOLIDIFICATION OF WATER DROPLETS IMPINGING ON A COOLING HORIZONTAL PLATE
Ryota Kimura, Kyoto Institute of Technology, Japan; Yoshinichi 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 impinged droplet solidifies in the plate. 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 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)*

DEPPINNING 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 motion is considered, the results show that the linear relationship between depinning force and maximum 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 modes. 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. We also investigate the role of particle and fluid inertia, 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 particle 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; Luís 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 frictional 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 Kadianamariam, Karlsruhe Institute of Technology (KIT), Germany; Markus Uhlmann, Karlsruhe Institute of Technology (KIT), 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.

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; Luís 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 frictional 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.

EXPERIMENTS ON INCipient 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 carrying grains, in sewer systems, and in degrading 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.

FLoW OVER AN EVOLVING SEDIMENT BED
Erick De Moraes Franklin, UNICAMP - University of Campinas, Brazil; Iman Lashgari, KTH Royal Institute of Technology, Sweden; Aman Kadianamariam, Karlsruhe Institute of Technology (KIT), Germany; Markus Uhlmann, Karlsruhe Institute of Technology (KIT), Germany
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.

NUMERICAL 3-D SIMULATION OF DEVELOPING STRATIFIED GAS-LIQUID FLOW IN CURVED PIPES
Ali Haj, AmirKabiri University of Technology, Iran; Zohreh Manooori, AmirKabir University of Technology, Iran; Majid Saffar Avval, AmirKabir University of Technology, Iran; Goodarz Ahmad, 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
Katherine Molko, 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 on onshore and offshore regions is more susceptible to flow dynamic processes and instabilities. The up- and down stream flow stream leads within multiphase systems to various acoustic, also bubble, in 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 Bundesrat

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 onedimensional momentum and energy balance, the flow 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 bubble and liquid thicknesses \( n(t) \), \( l(t) \), and \( \eta \), respectively. Lengths are made dimensionless with the air thickness at the outlet, \( Ho \), and pressures with twice the air dynamic pressure, \( \rho_{\text{a}}ua^2 \). 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, \( \eta=\frac{\rho_{\text{a}}\Lambda^2}{\rho_w} \), and \( \Lambda=\frac{Ho}{ua} \), where \( \epsilon \) is the surface tension coefficient.

* Supported 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 IMPEIMENTG ON LIQUID SURFACE
Hisanobu Kawashima, Gunma University, Japan; Shun Fujishima, IHARA SCIENCE CORPORATION, Japan; Solchiro 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 impact 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 liquid of the same fluid. Furthermore, the high mobility 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 19K05782 is gratefully acknowledged.

MODELLING OF BUBBLE DYNAMICS AND EULER/LAGRANGE PREDICTIONS OF BUBBLE COLUMNS
Marcello Muzzi, 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 Simulation, 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, conical velocity 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

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. These phase fields methods treat the interface as a region of finite thickness and resolve this paradox by a pure diffusive mechanism for the MCL. Recently, we couple 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/coal.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 WING, 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 vortex 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 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.

* The funding by Helmholtz Energy Alliance “Energy-efficient chemical multiphase processes” (HA-E-0004) is gratefully acknowledged.
BREAKUP OF FINITE SIZE COLLOIDAL AGGREGATES IN TURBULENT FLOW INVESTIGATED BY 3D PARTICLE TRACKING VELOCIMETRY
Matthias 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; Petr 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 ± 0.4 mm which is large compared to the Kolmogorov length scale η = 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 breakup 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-COULETTE 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 most of drag. To further study the effect of various bubble flow patterns taking place into the flow system, we present an investigation of the drag reduction in bubbly Taylor-vortex flow systems. Ultrasonic 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) In the cases of different gas flow rates, the Taylor-vortex 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; Richard Rapberger, Johannes Kepler University, Austria; Konrad Dymowsky, Technical University of Lods, Poland; Yuichi Murai, Hokkaido 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 wettablility 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 PRESSURES

Sandra Michaile, 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/cm² 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; 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-air 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 different frequencies is applied and based on oscillation frequency; they assess their scope and limits 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. Bubbles 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.

A STOCHASTIC APPROACH TO COLLOIDAL PARTICLE COLLISION/AGGLOMERATION

Radu Eduard Maftei, INRIA, France; Mireille Bossy, INRIA, France; Jean-Pierre Minier, EDF, France; Christophe Profeta, Universite` 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. The collision and 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.

SPRAY IMPINGEMENT ON SPHERICAL TARGETS IN HOMOGENEOUS, ISOTROPIC TURBULENCE

Paul Andrade, Imperial College London, UK; Georgios Charalampous, Imperial College London, UK; Yannis Haralampous, Imperial College London, UK

Closures 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 without mean flow has been thus far mainly investigated for gravity driven falling films. Based on Langevin models for colloidal particle transport. More precisely, the stochastic model is used to represent colloidal particle transport. This model is based on the Langevin equation of Brownian motion, turbulence and wall-bounded flows, are met.

EXPERIMENTAL VALIDATION OF ANALYTICAL MODELS FOR LIQUID FILM BREAK-UP

Hakan Line, TU Graz, Austria; Heßfried Steiner, TU Graz, Austria; Guenter Brenn, Graz University of Technology, Austria; Philipp Engesser, Lam Research, Austria

In wet spinning the complete coverage of the processed surface with operating liquid is generally a prerequisite for achieving the desired results. Models for the description of break-up of liquid films have therefore been devoted to finding reliable criteria for the occurrence of the process. The underlying mechanisms for the occurrence of the breakdown processes are strongly influenced by a number of factors such as the film thickness, the liquid viscosity, the liquid surface tension, the liquid temperature, the liquid velocity, and the liquid temperature. In this work, we focus on the influence of the liquid velocity on the break-up of liquid films.

A STOCHASTIC APPROACH TO COLLOIDAL PARTICLE COLLISION/AGGLOMERATION

Radu Eduard Maftei, INRIA, France; Mireille Bossy, INRIA, France; Jean-Pierre Minier, EDF, France; Christophe Profeta, Universite` 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. The collision and 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. This definition needs to be generalized to more complex situations. In this work, we 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 speciation analysis, are addressed in the context of a two-phase system comprising colloidal particles and tracer.
A NEW MIXTURE MODEL FOR COMPRESSIBLE MULTIPHASE FLOWS
Marc Henry De Frahan, University of Michigan, USA; Shahab Alahyari Beig, University of Michigan, USA; Harish Ganesha, 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 discretized using the Discontinuous Galerkin method, 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, CentraleSupélec; EM2C, France; Stephane De Chaisemartin, IFPEN, France; Marc Massot, CentraleSupélec; EM2C, France; Frederique Laurent, EM2C, CentraleSupélec, France; Adam Larat, EM2C, CentraleSupélec, France; Stephanie 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. We have been obtained and shown to be capable of simulating the dynamics of polydispersely evaporating sprays and its coupling to a turbulent gaseous flow field. However, building up a global multi-scale model with the capability to resolve injection processes 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 latest library for adaptive mesh refinement and massively parallel computing. The key ingredient is a good choice of variables, which can describe both the polydispersity of a spray as well the topology of an injection process. After verification and validation on challenging two-phase simulations in terms of both physics and high performance computing will be shown.

LVV ANALYSIS OF GAS-LIQUID SLUG FLOW IN A HORIZONTAL PIPE
Jignesh Thaker, S V National Institute of Technology Surat, India; Jyotirmay Banerjee, S V National Institute of Technology Surat, India
Interfacial flow structure of liquid slug is responsible for several erosion-corrosion problems in the piping systems of industries including nuclear, petroleum, 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.

COMPUTING COMPLEX MULTIPHASE FLOWS WITH BLUE
Damir Junci, LIMSI-CNRS, France; Jafel Chergui, LIMSI-CNRS, France; Suphakorn Suwannaphan, Chulalongkorn University, Thailand; Song Ilhun, Hongik University, Republic of Korea; Lyes Kahouadi, Imperial College London, UK; Omar K. Matar, Imperial College London, UK
BLUE is software for simulations of multiphase flows on nearly unlimited processors. A dual Front Tracking/Level Set method is developed which 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 implementation in the p4est library for adaptive mesh refinement and parallel performance.

*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 (DQ MOM). 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 DQ MOM 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.

PARTICLE-WALL ADHESION MODEL FOR TURBULENT DISPERSE MULTIPHASE FLOWS WITHIN AN EULER-LAGRANGE LBE APPROACH
Michael Breuer, Helmut-Schmidt University Hamburg, Germany; Naser Almammad, Helmut-Schmidt University Hamburg, Germany
The present study is concerned with the particle-wall adhesion in turbulent particle-laden flows. The focus is on a four-way coupled Euler-Lagrange approach relying on the large-eddy simulation technique and a hard-soft sphere model with deterministic collision detection (inter-particle and particle-wall) a microscopic-based wall collision model is developed. Taking the various two-phase gas 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.

LDV ANALYSIS OF GAS-LIQUID SLUG FLOW IN A HORIZONTAL PIPE
Sanjay Alang, S V National Institute of Technology Surat, India; Jyotirmay Banerjee, S V National Institute of Technology Surat, India; Jignesh Thaker, S V National Institute of Technology Surat, India; Seungwon Shin, Hongik University, Republic of Korea; Lyes Kahouadi, Imperial College London, UK; Omar K. Matar, Imperial College London, UK
BLUE is software for simulations of multiphase flows on nearly unlimited processors. A dual Front Tracking/Level Set method is developed which 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 implementation in the p4est library for adaptive mesh refinement and parallel performance.

*This research was supported by the UK Engineering and Physical Sciences Research Council (EPSRC) Programme.

ICMF 2016 International Conference on Multiphase Flow
Firenze, Italy, May 22 - 27, 2016
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 in part to the presence of irregularly shaped particles. Therefore, to understand the hydrodynamics of the slug flow, it is paramount to study the design of crude oil production lines as well as in the project of equipment involved in oil and gas operations. An experimental rig in the NU14M365H was used to collect data. The goal of this work is to experimentally analyze slug parameters such as velocity and frequency 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 was used 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 composite capacitance plates was designed and validated for its measurement performance through dynamic experiments. The conductance sensor and the capacitance sensor work independently 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 signals. The conductance and capacitance parallel plate configuration 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 sensor technology has a wide range of application possibilities in industries.

CHARACTERIZATION OF NEARLY JAMMED GRANULAR FLOWS IN HIGH SHEAR SYSTEMS THROUGH MOLECULAR DYNAMICS SIMULATIONS
Mohammad Khaliliehrahm, 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 behavior and carry out Molecular Dynamics (MD) simulations (based on Hertz-Find sphere models) on a high shear disk. Air is injected bubbling inside the medium. A complete understanding of air motion requires understanding of the particle-wall collisional force and the rate of fluidization depends on the rate of air flow. Depending on the flow rate, exhibits different patterns. The shape and behavior of liquid layers depends on the inclination angle of the floor. We observe that the air stream meanders through the medium and is confined to sub-regions whereas a lower level of detail is sufficient for the rest of the 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 beads of size 400-600 μm. 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 fluidization. A complete understanding of air motion requires knowledge of fluid mechanics as well as gas-solid interaction mechanism.

DETERMINATION OF HYDRODYNAMIC PARAMETERS OF AIR-WATER SLUG FLOW USING ULTRASONIC TECHNIQUES VIA HILBERT–HUANG TRANSFORM
Balarama Moitra Abhagni, Cranfield University, UK; Hoi Yeung, Cranfield University, UK

Two-phase slug flow is the most common flow regime in horizontal flow pipe. It is intermitted, transient and its hydrodynamic parameters are difficult to predict. Determinations of the hydrodynamic parameters are required for many designs 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 0.168m 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 0.55 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.

COUPLING DIFFERENT LEVELS OF RESOLUTION IN DEM SIMULATIONS
Daniel Quetschiner, 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.
**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 Electromagnetic (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 due to the induced induced momentum on the particles. The resulting 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.

**Numerical Analysis of Erosion on Suspended Heat Transfer Surfaces in a 350MW Supercritical CFB Boiler**

Qiu-Yu Tu, Chinese Academy of Sciences, China; Hai-Gang 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.

---

**Influence of Angular Momentum on the Formation of Air-Core Intake Vortices in Cylindrical Water Tanks**

Nicolai Szegy, Hamburg University of Technology, Germany; Daniel Bezecny, Hamburg University of Technology, Germany; Steffen Richter, Hamburg University of Technology, Germany; Michael Schlueter, Hamburg University of Technology, Germany

The occurrence of air-introducing 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 (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 velocity vectors for 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.

---

**Influence of Partially Wetted Micro-Particles at a Contaminated Interface on Coalescence Process**

Erik De Malmezat, EDF, France; Frederic Riss, IMFT- CNRS/Université de Toulouse, France; Olivier Massenart, Tihonnie, 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, 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 becomes 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 rim reveals a sudden, localised snap-off occurring, 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**

Amineh Mohaghar, 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 is one of the critical steps in the formation of liquid drops near solid surfaces in many processes such as atomization in spray cooling and spray quenching. The generated droplets are then dispersed in the surrounding flow and participate in the transfer processes. The detailed understanding of the primary breakup processes is crucial for both computational and experimental studies in the field of multiphase flows.

**PARTICLE DISPERSION IN INHOMOGENEOUS TURBULENCE: AN ANALYSIS OF PDF-BASED MODELS**

Christopher Stafford, Newcastle University, UK; David Swales, Newcastle University, UK; Miljanar 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.

**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; Ryotaro 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 hydrophilic 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 diuret eugenol, which simulates an explosive substance, was collected by the rimming flow device. The experimental results showed that the collection efficiency increases when the concentration of the cylinder. This was because the particle concentration in the liquid decreases with increasing the concentration of eugenol gas near the inside wall by centrifugal force. The collection efficiency further increased by adding both OBPs compared with an inorganic salt and PBP1 expressing eugenol. Furthermore, the collection efficiency increased with the amount of OBPs.

**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 microstructures and 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 dislocations 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-Model is designed, in which the vapor film is built by a dense accumulation of vapor bubbles. The vapor phase is assumed to be a Dispersed System. For the numerical simulation, 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 bifacial 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
Wenbin Zhang, 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 phases. 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 numerical results are in good agreement with the analytical solutions to this problem. It is observed in all the numerical computations that the two principal data found in the literature as well as with the analytical solutions to this problem. The simulation numbers are in the range 0.1 to 100. In the case when the particle moves using a resolved discrete particle method (RDPM). The particle Reynolds numbers are allowed to appear far from these fronts. A number of numerical examples are provided and show this method has a reasonable agreement with both the exact solutions and the experimental data.

EFFECT OF THE VAPOR FLOW ON THE DROP SPREADING AT THE IMPACT IN THE LEIDENFROST BOILING
Guillaume Castelet, Université de Lorraine, CNRS, France; Ophelele 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, maximum spreading diameter were measured by high-speed photography. 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 rim/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.

DROP IN DROP DIAMETER FROM A VIBRATING CAPILLARY TUBE
Herve Duval, CentraleSupélec, Université Paris-Saclay, France; Aure 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) on the formation of microemulsions was not investigated. In this case, Aumaud (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 taken for the case of a drop 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 ratio and the Bond number. 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 exponentfrequency is given by Strani & Sabetta’s calculations and the damping term is modified to account for additional friction.

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 0.01 and was injected into an 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 effects were captured and altered droplet shapes appeared in confinement of the computational cells. 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 exponentfrequency 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; Efstathios 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 (RDPD). 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.

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 insulated 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 μm) 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 Marichio, University of Udine, Italy; Alfredo Solidà, 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.

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; Andrey Forniých, Ben-Gurion University of the Negev, Israel; Boris Krassovit, 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 the chemical potential. Despite the possibility to find different numerical simulations of turbulent pipe and channel flows, however, the ensuing results are scattered and 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 number density and inertia. For all modeled as a 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 flux. 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.

SEDIMENT-LADEN FRESH WATER ABOVE SALT WATER: STABILITY ANALYSIS AND NONLINEAR SIMULATIONS
Florian 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, two 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 modifies the traditional double-diffusive fingering by creating an unstable ‘nose region’ in the horizontally averaged profiles, transverse between the upward moving side 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 is a function of the particle volume fraction, 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 particle volume fractions, the dominant instability mode becomes Rayleigh-Taylor-like. A scaling analysis predicts that particle settling thickness is a function of the particle volume fraction and the density ratio between the layers.

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 size, fluid density, particle density, particle volume fraction, and Reynolds number. However, these approaches do not take into account the precise arrangement of all the particles in the neighborhood. Therefore, we use a multiphase approach to model the forces on a particle, which includes contributions from all other particles in the neighborhood. We find that the forces on a particle depend on the number of neighbors in the neighborhood, and the volume fraction. We also find that the forces on a particle depend on the distance between the particle and its neighbors. Finally, we find that the forces on a particle depend on the direction of the particle.

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≈400

Daniele Cavicchioli, 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 ≈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 Maniaca. Simulations were performed on 30 million grid cells, using about 2'000 parallel cores of the EPRE HPC clusters. Bubbles are generated by means of a novel boiling sub-grid model based on wall turbulence. Two different resolution levels, 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 are not covered by a thin layer, where they can be considered as an effective porous medium. One method which enables an efficient simulation of very light particles is DEM coupled with the continuum phase. The later is the subject of a Leidenfrost temperature impacting onto hot inclined surfaces.

STUDY OF BUBBLY FLOW THROUGH A PERFORATED PLATE USING A TWO-FLUID MODEL AND A DISCRETE ELEMENT METHOD

Carlos Pena Morfemen, Universitat Jaume I, Spain; Paul 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 are used, first at a resolution level of 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 are not covered by a thin layer, where they can be considered as an effective porous medium. One method which enables an efficient simulation of very light particles is DEM coupled with the continuum phase. The later is the subject of a Leidenfrost temperature impacting onto hot inclined surfaces.

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 1980'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. Experimental validation of the setup was performed by comparing gas holdup and bubble size distributions in the non-vibrating case with established data in the literature.
IMPLEMENTATION AND DEMONSTRATION OF A BOUNDARY CONDITION WALL FUNCTION FOR DIRECT PRECIPITATION FOUILLING
Sverre Gulikstad 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 foulling. 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 dominated.

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 gasses 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 SCORRE project is gratefully acknowledged.

MEASUREMENTS AND VISUALIZING DURING CONDENSATION INSIDE A 3 MM DIAMETER CHANNEL
Marco Azzolin, Università degli Studi di Padova, Italy; Stefano Bertolini, 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 the background, new techniques for the 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-phases 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; Fulling 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 and makes granular flow, giving rise to rear deposit. The critical location where l(x,t)~h(x,t) is the propagation front. The solutions reveal l(x,t)<h(x,t) in the front indicating 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 volume of water constant, or by imposing a constant positive fluid pressure.

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 effects 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.

3D SIMULATION OF NUCLEAR POOL BOILING USING DIFFUSE INTERFACE MODEL
Sungwon Tanimoto, Mie University, Japan; Koichi Tsujimoto, Mie University, Japan; Toshikiko Shakouchi, Mie University, Japan; Toshitake 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 effects 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 Belot, 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 allows the dust spreading to the long distance and the dust spreading inside the silo through the 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 Aussaloux, 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.

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 effects 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.
DIRECT AND INDIRECT MEASUREMENTS OF FRICTION COEFFICIENTS OF STEADY DRY GRANULAR FLOWS DOWN A ROUGH NARROW INCLINE OF ROUGH Beds
Fuling Yang, National Taiwan University, Taiwan; Chiu Tingyuen, 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, \( \mu \), varies with flow bulk inertial number IB estimated by volume flow rate and flow height. Over IB=0.2, the \( \mu \)-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 I-w related relation similar to \( \mu \)-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, \( u_l \), at the wall. When this internal \( \mu \) is examined with respect to local loc estimated via local slope of \( u_l \), it shows a similar linear trend with \( \mu_w \) but with greater dispersion, due to fluctuations in loc, and of a lower magnitude.

The difference between \( \mu_w \) 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 Blazuzzi, 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 Banwart, State University of Campinas, Brazil

The work aims to evaluate the efficiency 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 shear rate on the HOVR effect. A significant reduction in HOVR concentration of 1000 ppm, 1500 ppm and 2000 ppm at temperatures of 40 °C, 50 °C, 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.
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 and relative velocities in different bubble columns. Their relevance once evaluated, taking into account the relation between measured bubble diameters and its 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 of an increase of 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 of the critical Bond number and its counterpart 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 simulation approach to the problem of drop detachment on an increase of the critical Bond number. 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 Energies Nouvelles and Université Grenoble Alpes, France; Ann Forret, IFP Energies Nouvelles, France; Daniela Marchisio, Chemical Engineering, Politecnico di Torino, Italy; Frederic Augier, IFP Energies 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 accumulate 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 Almèras et al. (2015) is implemented in addition to the diffusion resulting from the shear induced turbulence.

TRIBO-CHARGING AND FLAME-SYNTHESIS TO CREATE SUPERHYDROPHOBIC 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; Francesca 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 realistic k-ε model. For gas-liquid and gas-solid-liquid systems, two-fluid model and multi-fluid model with the mixture k-ε 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 Universität 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 Schlüeter, 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 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 gel a deeper inside 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 Lauriniec, 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 bioaerobic fluids. In such bubble columns a periodic flow behavior which is characterized by quasi sinusoidal trajectory of bubbles, can be observed. Large bubbles lie in the middle of the column and descents 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 formation can be regarded for a large range of injection conditions by varying the injection type and flux. Different metrological conditions 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: THE EFFECT OF GAS-PARTICLE FLOW ANALYSIS

Takayuki Nagata, Tokai University, Japan; Taku Nonomura, Institute of Space and Astronautical Science, Japan; Shun Takahashi, Tokai University, Japan; Yusuke Azuma, 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 discrete simulations. In this work, we address a similar 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 considered. 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 based on a probabilistic and statistical framework, knowing 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 fluid 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/ST9/05677.

NUMERICAL SIMULATION OF A SETTLING ICE BALL WITH COUPLED HEAT AND MASS TRANSFER

Agathe Chouippe, Karlsruhe Institute of Technology, Germany; Michael Slumpf, Germany; Markus Uhlmann, Karlsruhe Institute of Technology (KIT), 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/spatial-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 model accounts for both temperature and water vapor fields, focusing on the influence of the wake structure upon the existence of supersaturation, as well as on the modification of the wake by both scalar fields.

References:
EXPERIMENTAL ANALYSIS ON THE CHARACTERISTIC AND MECHANISM OF THE SECOND DOMINANT FREQUENCY OF SUSPENDED SUSPENSIONS OF ATTRACTIVE MICROPARTICLES

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⁻²·s⁻¹ and water temperature ranges 20-70°C. The second dominant frequency is found to be generated by the separated steam bubbles based on the bubble oscillation theory. Firstly, the second dominant frequency of supersonic jets is investigated in detail and found to increase 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 ±15%. 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; Heles Baptiste, Imperial College London, UK; Richard Mathie, Imperial College London, UK; Alexandros Charogianis, Imperial College London, UK

We present an experimental technique that combines simultaneous planar laser-induced fluorescence imaging (PLIF) and infrared (IR) thermography, and a heat transfer model 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 temperature, interfacial intertemperatures, 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 HTCs 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.

EXPLORING THE RHEOLOGICAL BEHAVIOUR OF AETHERAL SUSPENSIONS OF ATTRACTIVE MICROPARTICLES

Eric Murphy, Iowa State University, USA; Gilson Lomboy, Iowa State University, USA; Srimur 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 the absence of rheological modifiers, cement is a dense suspension of attractive microparticles. At even small shear rates, e.g. 10 s⁻¹, thermal effects are small, producing aggregated systems and with Peclet numbers O(10⁴). 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-like behavior through analysis of extracted stresses and explain how the dynamics of fluid, attractive potentials, and friction conspire to form two-particle correlations

ENHANCED RECOVERY OF URANIUM(VI) AND EUROPIUM(III) USING IONIC LIQUIDS IN SMALL CHANNEL

Qi Li, University College London, UK; Dimitrios Tsolakidis, 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-DO). In this work Eu(III) and U(VI) in nitric acid solutions are extracted by an ionic liquid [(C4mim)(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⁻¹, and are significantly larger than in conventional extraction units. Computational fluid dynamics simulations of mass transfer were also performed and compared to experimental data.

VISCOCUSTIC FLOW MODELLING FOR POLYMER FLOODING

Shavuk De, Eindhoven University of Technology, The Netherlands; Frank Peters, Eindhoven University of Technology, The Netherlands; Johaan 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.

ATOMIZATION OF NON-NEWTONIAN FLUIDS USING AIRLESS SPRAYERS: EFFECT OF VISCOS 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 fluid and 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 sprays was qualitatively 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 oscillations in the liquid film. 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
Georgios Kapatos1, Benjamina Nour1, KTH, Sweden; Agathokli Pasios, Aristotle University of Thessaloniki, Greece; Alkaterini 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 μ-tubes (i.d. 110 μm), placed 170 & 710 μm 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 μ-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 μ-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 μ-tubes. For non-Newtonian and very viscous Newtonian liquids practically no coalescence was observed. This observation 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.

SIMULATION OF GASIFYING PARTICLE, INTERACTION WITH FREE INTERFACE FLUID FLOW
Zornitsa Malishev, 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 of-equilibrium state which induces the solid motion. However, it is numerically analyzed that the solid 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 solid motion is studied.

Support from grant Swedish Research Council (VR) is gratefully acknowledged.

A WALL-FUNCTION MODEL FOR NON-BROWNIAN HYDROSOL PARTICLES DEPOSITION
Magali Dupuy, CentraleSupélec, Université Paris-Saclay, France; Arunvady Xayaasen, 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 region 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 Ahmad (1995) whereas the particle behavior is obtained by the Eulerian tracking. Two particle dynamic equations 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 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.

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; Jacob Wans, 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 drops 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 and with other important factors on the 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.

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 the 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.

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 adapted 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 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**

Laura Griggi, De Brabandere, CORIA, France; Benjamin Robbins, CNRS, CentraleSupélec,Université Paris-Saclay, France; Frédéric Kmoch, CERFACS, France; Matthieu Boileau, CNRS - CentraleSupélec, France; Marc Massot, CentraleSupélec, EM2C, France, Eleonore Riber, CERFACS, France.

While combustion of gasoil 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 pulsed 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. The computational setup is consistent with the experimental conditions and was positioned at y=5.4, 10.3 and 20.0 cm, Re=891). The measurement volume was a horizontal slab of the inlet at a bulk flow velocity of $U_b=0.17$m/s ($Re_b=102\times10^3$, 0.99 = 5.0cm, Re = 891). The measurement volume was a horizontal slab (5.1x5.1x0.3, l x w x h) extending from the wall, 30cm above the bottom. The setup comprised four high-speed ImagerPro HS cameras (2016 x 2016 pixels), a high-speed laser (Nd:YLF, Darwin Duo 80M, Quantronix), optics/prisms and data acquisition/processing software (LaVision, DaVis 8.2). The data were acquired and without a stationary held sphere that had a diameter, D=6mm ($D=51$, 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 37.5mm ($y^+=46, 88$ and $319$) from the wall (measured from the sphere’s center). Finally, a semi-unstructured unstructured code is used to provide a detailed description of the pulsed jet dynamics and to evaluate a spray model that is currently used in the conference we will present the calculations is that the bubble shape and growth rate strongly depend on its position 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.

**TURBOPHORESIS; THE TURBULENT SORET EFFECT**

Nits Erland L. Haugen, Norwegian University of Science and Technology, Norway; Dhinubaditya 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 Soret effect in near-equilibrium thermodynamics, we can describe the flux of particles as a sum of two fluxes, described by two turbulent transport coefficients: turbulent diffusion of particles and the turbophoretic coefficient. The second (turbophoretic) flux is assumed to be proportional to the gradient of the second momen of turbulent kinetic energy. The ratio of these two coefficients would be analogous to the molecular Soret coefficient; hence we call this the turbulent Soret coefficient. Our numerical calculation show that such a description is a good * prediction of our data. Furthermore, we find that the turbulent Soret coefficient is a non-monotonic function of the particle inertia (described by the Stokes number); i.e. beyond a critical Stokes number the clustering of particles decreases, but in a smooth manner.
EFFECTS OF GRAVITY AND ITS ORIENTATION ON BUBBLE DEPARTURE
Paolo Di Mare, University of Pisa, Italy; Nicola Margonetti, 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 mm3/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), recorded and bubble shapes and local gas fractions are determined 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 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.

FLOW AND SHEAR INDUCED BY BUBBLES RISING THROUGH HOLLOW-FIBER MEMBRANES IN A PARALLEL ARRANGEMENT
Katsumi Iwatakiya, DOSHISHA University, Japan; Satoshi Okumura, 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 phenomena studied in two-phase flow 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 affects 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 is used in the 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 PIV analyses are conducted based on VOF evaluation and LDV—In the 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.

BOBBLY FREE AND IMPINGING FORCED JETS: EXPERIMENTAL STUDY BY PIV/PBFI/PTV
Konstantin Pervunin, Kutateladze Institute of Thermophysics, SB RAS, Russian Federation; Mikhail Timoshkov, Novosibirsk State University, Russian Federation; Dimitry Markovich, Kutateladze Institute of Thermophysics, SB RAS, Russian Federation
An experimental investigation of bubble-saturated free and impinging turbulent jets flow was conducted using planar fluorescence for bubbles tracking (PBFI) technique combined with PIV and PTV methods. PBFI 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 velocities in both phases, it is concluded on the impact of bubbles on characteristics of the flows were calculated, including mixing-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.

HYDRODYNAMIC STUDIES OF CUTTING BUBBLES IN MICRO-STRUCTURED SQUARE BUBBLE COLUMN USING ULTRAFAST X-RAY TOMOGRAPHY
Krishnarej Thiruvalavan Sujatha, Eindhoven University of Technology, The Netherlands; J. A. M. (hans) Kupers, Eindhoven University of Technology, The Netherlands; Yuki Man Lau, Helmholtz-Zentrum Dresden-Rossendorf (HZDR), Germany; Frank Barthel, Helmholtz-Zentrum Dresden-Rossendorf (HZDR), Germany; Uwe Hampel, TU Dresden, Germany; Markus Schulte, 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 means of superficial gas velocity to obtain bubble size distributions and velocities. We acknowledge the financial support from European Research Council (ERC) Grant SIG 259521 and SIG 307360.

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.

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; Yasushika 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-aqueous 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 future 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.
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; Peng 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, the dispersed oil-water flow velocity can be predicted. The experiments were performed in a horizontal pipe (mixing intensity) and a horizontal test section to verify the proposed correlation. Good agreement with the new experimental data. Correlation has been developed. The proposed correlation allows to achieve a good agreement with the new experimental data.

*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. 13TYSJC40200).

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 m inner diameter and 5.3 m height. The annular gap and the open tube (both with a pipe sparger and a spider sparger) configurations have been investigated 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 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 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.

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 Jorgen 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 (mP=25 mPa·s). Experiments were conducted in the Multiphase Flow Laboratory at the Norwegian University of Science and Technology. With an inner diameter of D=60 mm and a total length of 50 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.
ULTRASOUND REFLECTOR RECOGNITION AND TRACKING TECHNIQUE FOR GAS BUBBLES IN LIQUID
Antonin Povalny, Tokyo Institute of Technology, Japan; Hiroshige Kikura, Tokyo Institute of Technology, Japan; Tomonori Ibara, 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 method to measure gas bubbles in 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 URRRT, provide data for the development of this method and also to detect limits and measuring performance of this method. Results obtained by URRRT 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.

FLOW REGIME AND MASS TRANSFER STUDY IN STRUCTURED PIPE REACTORS USING ELECTROCHEMICAL MICRO-PROBES
Marco Altheimer, ETH Zurich, Switzerland; Dustin Becker, ETH Zurich, 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 the onset of the development of a turbulent flow 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.*

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

Inclined risers are used in 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 method to measure gas bubbles in 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 URRRT, provide data for the development of this method and also to detect limits and measuring performance of this method. Results obtained by URRRT 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.

DEVELOPMENT OF A DISCRETE GRAIN BREAKAGE MODEL TO PREDICT EROSION MECHANISM
Rahul Bharadwaj, ROCKY - DEM, USA; Karoline Ropetato, 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.

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; Pascale Aussillous, Aix-Marseille University, CNRS, 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.

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 μ(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 (ω:1:2=1:1:2:1:3) in a rotating tumbler under three rotating speeds (ω: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 of the local concentration function 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.

*S The support from the national natural science foundation of China (Grant Nos. 51576211 & 51106160) is gratefully acknowledged.

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 experiments. 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 μm, 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 0.6 m/s for 50 μm sand.

A TWO-DIMENSIONAL CFD MODEL FOR GAS-LIQUID FLOWS IN A COIL-WOUND HEAT EXCHANGER
Thomas Nühn, Linde AG, Germany; Manuel Knaup, 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 the twophase flow, are not always taken into account. Therefore, a two-dimensional CFD model is 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.

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 valve downstream of the separators is operated through controllers to ensure the desired interface level. The controller is dictated by the required quality of the separation.

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 Doolweerdt, 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 model of the electrolysis process is developed 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, evaporation 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; Gijsbert Ooms, Delft University of Technology, The Netherlands; Ruud Henkes, Delft University of Technology, The Netherlands; Mathieu Pourquie, Delft University of Technology, The Netherlands; Anton Kideiss, , 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 fairly 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 is not computed by satisfying the jump condition, it is 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 Turin, 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 Rayleigh number (Ra), 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. More details are obtained by varying the jump conditions, and initial stages of the wave generation process, the amplitude of the interface elevation grows in time as $t^{2/5}$. Wavenumber spectra, $E(kx)$, of the surface elevation in the capillary range are in good agreement with the prediction of the Wavenumber theory (Swift). The number of modes $k$ increases as $t^{1/3}$. We also extend our analysis to the simultaneous evolution of the temperature field.

CFD STUDY ON DROPLET FORMATION IN NON-NEWTONIAN FLUIDS FLOWING THROUGH T-JUNCTION MICROCHANNEL Arnab Atta, Indian Institute of Technology Kharagpur, India; Sumasekhara Goud Sorniti, 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 increasing 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 Iwasa, Nagoya University; Masahiro Yamashita, Nagoya University, Japan; Kazuyuki Nakagawa, Nagoya University, Japan; Yosihyuki 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 cooling, 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 film, 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 lip and the root of the finger.

EXPERIMENTAL INVESTIGATION OF DROPLET GENERATION IN A MICROFLUIDIC T-JUNCTION USING NEWTONIAN NANOFLOWS Katerina Lolouzou, University of Nottingham, UK; Edward Lester, University of Nottingham, UK; Buddhikatho 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 used 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-María 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’Alenbert, 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 force 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 additional 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 thickness of the film 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 disjoining pressure are significant, and interesting interfacial instabilities are observed.
* Support from grant ANR-13-B509-0011 is gratefully acknowledged.

EXPERIMENTAL STUDY OF DROPLET FORMATION IN AN OIL-WATER SYSTEM USING MICROCHANNEL GEOMETRY
Sontti, Indian Institute of Technology Kharagpur, India; Arnab Atta, Indian Institute of Technology Kharagpur, India; Somasekhara Goud Sorniti, Indian Institute of Technology Kharagpur, India; Alfredo Soldati, University of Udine, Italy; 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’Alenbert, France
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 increasing 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.

EFFECT OF THE LUBRICATION FILM DYNAMICS ON THE DROPLET MOTION IN A HELE-SHAW MICROCHANNEL
Jose-María 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’Alenbert, 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 force 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 additional 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 thickness of the film 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 disjoining pressure are significant, and interesting interfacial instabilities are observed.
* Support from grant ANR-13-B509-0011 is gratefully acknowledged.
BREAKUP OF DROPLETS IN BRANCHING MICRO-CHANNELS
Luis Portela, Delft University of Technology, The Netherlands; Cees Harling, 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; Michael 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.

PARTICLE DISTRIBUTION IN UNBAFFLED STIRRED VESSELS
Alessandro Tamburini, Università degli Studi di Palermo, Italy; Andrea Cipollina, Università degli Studi di Palermo, Italy; Francesca Scarsoglio, Università degli Studi di Palermo, Italy; Giorgio Micali, Università degli Studi di Palermo, Italy; Michele Ciolfi, 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” (typically vertically deployed along the vessel walls). There are some cases in which this 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 concentration effect increases as particle inertia increases. This singular behavior was also investigated by the CFD simulations and a good agreement with experiments was found.

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; Giuseppe Montanta, 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 information properties in complex 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, ERT constitutes one of the best tools for obtaining quantitative data for CFD models validation. In this paper, a practical method for assessing quantitative data from ERT systems is presented and validated through experimental data. This implies 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.

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 Heylman, 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 (IBE) method (Hassanizadeh and Renger, 2010) was used with the PDF to develop a new CQMOM methodology. 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 integrated into the OpenFOAM® CFD toolbox. While SPH gives accurate results for simulations of single bubbles/droplets, its performance in more complex cases is still object of research. For multiphase systems, 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 concentration effect increases as particle inertia increases. This singular behavior was also investigated by the CFD simulations and a good agreement with experiments was found.

COMPARISON BETWEEN DIFFERENT METHODS FOR THE SOLUTION OF THE BIVARIATE POPULATION BALANCE EQUATION
Antonio Buffo, Aalto University, Finland; Aloisius 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 Class (HMHMC) and Quadrature-based Moment Methods (QBMM). The comparison between different methods has been performed on a number of relevant test cases, considering coalescence and breakup problems, growth, nucleation, mass transfer and chemical reaction.

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 Heylman, 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 (IBE) method (Hassanizadeh and Renger, 2010) was used with the PDF to develop a new CQMOM methodology. 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 integrated into the OpenFOAM® CFD toolbox. 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 with possible remedies.

MODELLING OF THE FLOW REGIME TRANSITION WITH THE SMOOTHED PARTICLE HYDRODYNAMICS
Michał Olejnik, The Szwawskie Institute of Fluid-Flow Machinery, PASci, Poland; Kamil Szewc, The Szwawskie 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, thus offering the possibility of having a truly unstructured grid. 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 Nugeria-Plus project (grant agreement No. 604965).
A COMPARATIVE ASSESSMENT OF LATTICE BOLTZMANN AND VOLUME OF FLUID (VOF) APPROACHES FOR GENERIC MULTIPHASE FLOWS

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 Kenjereš, 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 parastatic currents (leading to a spurious Reynolds number in LBM) were seen to be rather 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 achieved: 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 NACA0012 airfoil.

*This project has received funding from the European Union’s Seventh Framework Programme for research, technological development and demonstration under grant agreement in ACP2-GA-2012-314314.

EASTERN GASEOUS ATMOSPHERE 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 Sao Paulo, Brazil

Air entrainment is a common phenomenon in industrial applications. For example, self-entrainment in spillways and open-channel, entrainment in bubble columns 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

Eliseo C. Berruezo, Uppsala University, Sweden; Gunilla Kreiss, Uppsala University, Sweden; Martin Kronbichler, Technische Universität München, Germany; Simon Stikov, 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 straightforward. 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 is based on an auxiliary 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 flow approximations 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.


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 gradient 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 a single solid ellipsoid in a simple shear flow. The case of a single spheroidal particle (an ellipsoidal particle with rotationally 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. It is 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 prevents 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.

INERTIA 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 gradient 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 a single solid ellipsoid in a simple shear flow. The case of a single spheroidal particle (an ellipsoidal particle with rotationally 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. It is 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 prevents 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 MULTISCALING MODELLING TOOL FOR THE SIMULATION OF POLYURETHANE FOAMS

Pavel Ferk, University of Chemistry and Technology Prague, Czech Republic; Mohsen Karimi, Chemical Engineering, Politecnico di Torino, Italy; Daniele Marchisio, Chemical Engineering, Politecnico di Torino, Italy; Jarun 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 pressure gradient and density evolution of the foam under different conditions. The agreement level observed between the numerical predictions and experimental data allowed to further improve current knowledge on bubble generation mechanisms in multiphase flows.

* 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 Kharchikin, Lomonosov Moscow State University, Russian Federation; Pavel Spesivtsev, Schumacher Moscow Reasearch 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.

VOLUME-AVERAGED EQUATIONS FOR DIRECT NUMERICAL SIMULATION OF PARTICLE-DISPERSED FLOW UNDER A GRID RESOLUTION COMPARISON TO PARTICLE DIAMETER AND KOLMOGOROV LENGTH SCALE

Toshiaki Fukuda, Osaka university, Japan; Shintaro Takeuchi, Osaka University, Japan; Takeo Kaijishima, 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 flow. 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.

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 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.

FULLY EULERIAN SIMULATION OF 3D TURBULENT PARTICLE LADEN FLOW BASED ON THE ANISOTROPIC GAUSSIAN CLOSURE

Macleo Sabat, Laboratoire EM2C, CNRS, CentraleSuplec, France; Aymeric Vie, Laboratoire EM2C, CNRS, CentraleSuplec, France; Adam Larat, EM2C, CentraleSuplec, France; Marc Massot, CentraleSuplec, 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 approach reproduces statistically the moments 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 from CORIA for the gas phase, and MUSE/3D solver from EM2C for the Eulerian dispersed 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 SCLAR 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 high-aspect-ratio 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), GERNI-CINES (Grant 2015-x201526893) 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 Bergeras, 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/F7/2007-2013 is gratefully acknowledged.

MULTIPHASE PHENOMENA IN UNDERWATER EXPLOSION NEAR THE FREE SURFACE
Bao Guo, 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 numerically. The numerical results are 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.

SPATIAL TRANSITION OF SYNTHETIC VOID WAVES IN A HORIZONTAL BUBBLE CHANNEL FLOW
Hyun Jin Park, Hokkaido University, Japan; Yui Tasaka, Hokkaido University, Japan; Yuichi Murai, Hokkaido University, Japan
There are relatively small number of studies on horizontal bubble 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, same transitions of bubble swarms, 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 distribution, and strong bubble-liquid or bubble-bubble interactions 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.

MEASUREMENT OF EJULLITION CYCLE CHARACTERISTICS IN SUBCOOLED NUCLATE BOILING OF WATER FLOWING IN A NARROW VERTICAL GAP
Henryk Anglart, KTH Royal Institute of Technology, Sweden; Helko Kramer, ABB AB Corporate Research, Sweden; Alexi Foresta Caro, KTH Royal Institute of Technology, Sweden; Tor Laneryd, ABB AB Corporate Research, Sweden; Rebel 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 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/cm²; water mass flux 58 kg/m²s 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 heat transfer model taking into account 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.
No. 51276140 is gratefully acknowledged. * Support from the National Natural Science Foundation of China under Grant No. 51276140 is gratefully acknowledged.

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 electric 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 collidet 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-Soon 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.

WET GAS MEASUREMENT BY V-CONE THROTTLE DEVICE

THE APPLICATION OF THE DEVELOPED IMAGE PROCESSING ALGORITHM TO STUDY THE BEHAVIOR OF WATER DROPLETS IMPINGING ON THE TILTED HEATED SURFACES
Hadyan Kuntoro, Gadjah Mada University, Indonesia; Akmal Irfan Majid, Gadjah Mada University, Indonesia; Desenderianto Desenderianto, 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 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 heating stainless steel surfaces (SUS304) with 3 variations of surface roughness (Ra = 0.4, 3 and 10) were employed. The surfaces with diameter 30 mm were tilted 15° to the normal plane and the surface temperatures were varied from 150 up to 500 °C. The droplet diameter and the impinging velocity were kept constant around 700 μm and 4 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 Lyotol 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 m/s. The pipe diameter was 0.0127 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-ω turbulent model has been used combined with multiphase solver. The CFD modeling has been used to provide more detailed information of the droplet size distribution 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.

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 operating conditions. Therefore, numerical calculations of the 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 characteristics. This is the result of the various phases and processes. Theoretical and numerical studies, the continuous phases and particles to turbulence, temperature history of the droplets, drying characteristics, as well as droplet coalescence and aggregation 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 powder characteristics. Now the model is implemented in the laboratory spray dryer. The 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 drop drying 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.
REFRACTIVE INDEX MATCHING METHOD TO STUDY GASOLINE SPRAY-WALL IMPINGEMENT
Nicolas Lamarque, Continental Automotive France, France; Giacomo Picozzi Leopardi, Continental Automotive France, France; Christoffer Schmitt, Continental Automotive France, France; Jerome Helle, 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

DYNAMIC BEHAVIOUR OF TWO-PHASE FLOW THROUGH ORIFICES USING PARTICLE IMAGE VELOCIMETRY
Olivier Eyrique Barnedele, 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 affects both safety and reliability of these systems and sometime lead to failures and huge economic loss. Although, two-phase flow through orifice piping components have been studied in the past, the dynamics behavior of two-phase flow un 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 flow 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.

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 Diemane et al. 2014 and to the Beveloet 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 Maidji, Gadjah Mada University, Indonesia; Deendarlianto Deendarlianto, Faculty of Engineering, Gadjah Mada University, Indonesia; Wiranthi Wirantri, Gadjah Mada University, Indonesia; Indarto Indarto, Gadjah Mada University, Indonesia; Enggar Dwianto, Gadjah Mada University, Indonesia; Baskoro Adiputra Punowo, 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 of using MBG as an aerator for the waste water treatment from laboratory to the industrial scale. Some basic studies on microbubbles 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 ENTRAINDED FLOW REACTORS

Giovanni Tretoli, Consiglio Nazionale delle Ricerche, Italy; Maurizio Troiano, Universita degli Studi di Napoli Federico II, Italy; Francesco Saverio Manna, Consiglio Nazionale delle Ricerche, Italy; Piero Salatino, Universita degli Studi di Napoli Federico II, Italy; Fabio Montagnar, Universita degli Studi di Napoli Federico II, Italy; Roberto Solimene, Consiglio Nazionale delle Ricerche, Italy

During slugging 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-D lab-scale cold entrained-flow reactor (length 0.1–2.5 m), cold slugging mode, was used with a wall pressure where the wall motion 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, where the particle–wall collisions are modeled with an Hertzian approach, including torque and cohesion effects. Results illustrate the different structures of near-wall 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 Moro, 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 last 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 Reynolds 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 homogeneous flow model with an effective viscosity accounting for non-Newtonian effects. The Lockhart-Martenselli model also predicted well the present data at an appropriate C-value in two-phase friction multiplier was adopted.
PROPERTIES OF FINE BUBBLES AND OF NANO PARTICLES IN PURE WATER
Yomi Tokuda, Institute for Sustainable Humanosphere, Japan; Yomi Tokuda, Institute of Chemical Research, Japan; Naoto Nihei, Graduate School of Agricultural and Life Science, Japan; Shoichiro Hamamoto, Graduate School of Agricultural and Life Science, Japan; Yuchii Ogawa, Graduate School of Agriculture, Japan; Akihumi 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 it is still difficult to conduct basic research because methods 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 number of plugs (from 1 to 4 plugs). For the same number of bottom bubbling plugs, the bubble 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 bubble bubbling plugs contributed to the flow velocity. The plugs located to the close side wall short 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 location of bottom bubbling plugs 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.

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 achieved 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.

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 Mychkovskiy, 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 12x12 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 4x10^4 up to 1x10^5. The inlet void fraction was varied from 1 to 20%. The gas was injected as nominally mono-size drops with variable diameter 0.3 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 at the gaps, inlets and outlets. The tracer was a phase of a tracer bubble, 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. The inlets 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.

MIXING INDUCED BY A HOMOGENOUS BUBBLE SWARM IN PRESENCE OF EXTERNAL TURBULENCE
Elise Almeras, University of Twente, The Netherlands; Varghes Mathai, University of Twente, The Netherlands; Defrel 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 homogenous bubble swarm in the presence of external turbulence. The experiments are performed in a two phase flow system 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 image processing, the horizontal (fluorescein sodium) is continuously injected from a lower part of the tunnel. Using a high-sensitive camera and image processing, the horizontal (fluorescein sodium) is continuously injected from a lower part of the tunnel. Using a high-sensitive camera and image processing, the horizontal (fluorescein sodium) is continuously injected from a lower part of the tunnel. Using a high-sensitive camera and image processing, the horizontal (fluorescein sodium) is continuously injected from a lower part of the tunnel.

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 bubbles with 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.

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 bubbles with 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
Eido Frederix, University of Twente, The Netherlands; Arkadiusz Kuczyk, 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 model, in an axisymmetric system. The urgency of radio-frequency electromagnetic field heating 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 the oil. The aim of our work is to provide solutions to these problems. In the present work the numerical model of radio-frequency electromagnetic field heating is validated against experimental data from literature. The rough walls are modeled by fixed layers of tiny spherical particles. 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.

NUMERICAL MODELING OF CONVECTIVE FLOW IN COMPLEX HYDROCARBON LIQUID UNDER RADIO-FREQUENCY HEATING
Liana Kovaleva, Bashkir State University, Russian Federation; Victor Kireev, Mavrlitov Institute of Mechanics, Russian Federation; Ayart Muisin, 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 manufactured (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 the oil. The urgency of radio-frequency electromagnetic field heating 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 the oil. The aim of our work is to provide solutions to these problems. In the present work the numerical model of radio-frequency electromagnetic field heating is validated against experimental data from literature. The rough walls are modeled by fixed layers of tiny spherical particles made from Polyoxyymethylen (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 7.4 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 affecting the experimental setup. 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. The effect of small unresolved scales in this case.

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. Solid spherical particles made from Polyoxyymethylen (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 7.4 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 affecting the experimental setup. 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 CFX-15.0. Initially, the methodology for particle-laden flow simulation is validated against experimental data from literature. Particle laden simulations are realized by means of Euler-Lagrange 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.

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; Matthew Yates, University of Nottingham, UK; Buddha Hewakandamby, University of Nottingham, UK; Matthew Yates, University of Nottingham, UK; Barry Duignan, Dublin Institute of Technology, Ireland
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 Reynolds numbers and for varying distances downstream of the injection point. Particle distribution data from literature 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.

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 nonuniformity 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 F. Ji, 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 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 directly 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

INTERACTION BETWEEN A FINITE-SIZE BUOYANT BUBBLE AND TURBULENCE
Aurore Le Ray, Université Claude Bernard Lyon 1, France; Aurore Naro, 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 in the thin gap scales present in the two-scale model. 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 water 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.

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 both the volume fraction and the surface tension 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 as those of the particle-bubble interaction. 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.

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 turbulence interactions. Results are compared to those obtained for solid particles.

* Supports from ANR (TEC2), GENCI-CINES (Grant 2015-x20152b6893) and FLMSN are gratefully acknowledged.

RESEARCH DIRECTIONS AND FUTURE CHALLENGES AT THE INTERFASE BETWEEN WALL-BOUND NEWTONIAN FLUID AND MICROañELETROANOMIC (MEMS) DEVICES
Debayan Jana, University of California, Berkeley, USA; Frank W. Schmidt, Lawrence Berkeley National Laboratory, Berkeley, CA

The interaction between wall-bounded Newtonian fluid and micro-/nano-electromechanical (MEMS) devices is becoming ever increasing important and challenging. The viscous fluid levitation and alignment are gaining world-wide interest. The interaction between flow and MEMS devices is seen in the passing of MEMS fluidic actuation devices, operation of microfluidic channels, and levitation of MEMS devices in fluidic environments. The modeling of these interactions is an active area of research. The interaction poses the following challenges:

1. How do we simulate the flow in presence of MEMS actuation?
2. What are the key properties of MEMS devices that affect the flow?
3. How can we model the fluid MEMS interactions in practice?

Our research focuses on these three key challenges. We apply flow field modeling and measurements to develop a systematic understanding of the flow-MEMS interactions. This systematic understanding will lead to the development of analytical models and design tools for the flow-MEMS interactions. This will be of wide interest to MEMS engineers.

* Support from DOE Grant DE-SC0012708 is gratefully acknowledged.

ROTATION OF NON-SPHERICAL PARTICLES IN TURBULENCE - EFFECT OF SIZE ON KINEMATIC STATISTICS
Evren 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 and the breakup of a viscous liquid ligament. Furthermore, results of the dual-scale model are compared to those obtained for 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 as those of the particle-bubble interaction. 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 Marchiol, 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 passingively following the surrounding fluid. An increase of fiber length, in addition, decouples the rotational dynamics of a fiber from its translational dynamics pointing 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.

RELATIVE ROTATIONAL MOTION BETWEEN RIGID FIBERS AND FLUID IN TURBULENT CHANNEL FLOW
Cristian Marchiol, 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/T R anslation IN NEAR HOMOGENEOUS ISOTROPIC TURBULENCE
Liachi Sabbian, Technion - Israel Institute of Technology, Israel; Assaf Cohen, Technion - Israel Institute of Technology, Israel; René 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 403cm3 cube by eight wipers 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.

Steep 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 (H=45m, 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 for the mainstream flow and cavity flows. These calculations yield 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. For the stepped structure, as well as on the longitudinal changes in air-water flow properties.

* Support from grant ARC DP12010481 is gratefully acknowledged.

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 fundamental problem concerning the translation and rotation of small, rigid, dense, and 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 oblate and prolate colonies. We have also studied the rotations of particles with chiral ends that exhibit 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; Yoshitiiro Arii, University of Tsukuba, Japan; Akiko Kaneko, University of Tsukuba, Japan; Fumio Takemura, National Institute of Advanced Industrial Science and Techno, Japan; Masatoshi Ike, Apexell, 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 allopahne. 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 allopahne 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
Danusz Mieklewiecz, Gdansk University of Technology, Poland; Jaroslaw Mieklewiecz, 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 fill 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 Schröder, 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. In general, although numerical analysis is a powerful tool, the eludications 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 based on the eludication 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 for the eludication 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 based on the eludication 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 for the eludication 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 based on the eludication 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 for the eludication 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 based on the eludication 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 for the eludication 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 based on the eludication 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 for the eludication 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 based on the eludication 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 for the eludication 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 based on the eludication 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 for the eludication 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 based on the eludication 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 for the eludication 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 based on the eludication 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 for the eludication 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 based on the eludication 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 for the eludication 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 based on the eludication 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 for the eludication 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 based on the eludication 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 for the eludication 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 based on the eludication 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

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 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 behavior 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

Klaas Verhoogd, The Swedish Institute of Fluid-Flow Machinery, Pinna, Poland
Problems involving interactions of fluid and granular media phases are active research fields in the geosciences and natural hazard management. Particular attention is paid to understanding the processes and how to predict the results of interaction between water waves and seabed (formation of tsunami, 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 decades, many so-called meshless methods have been developed for modeling of the most macro 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 computations (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 COALESCEANCE 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 drop-growth mechanisms at the contact line. 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 of the 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 checked against available experimental results. The coalescence process in the presence of rods is studied 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.

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

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-cavitaiton 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-Lagrangian 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-Lagrangian 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.

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, Università di Pisa, Italy; Chiara Galletti, Università di Pisa, Italy; Simone Camaru, 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 prior art use of the wave generator enables the coupling 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; Thiibaut 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 account for the 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, Pontifícia Universidade Católica do Rio de Janeiro, Brazil; Igor Braga De Paula, Pontifícia Universidade Católica do Rio de Janeiro, Brazil; Anis Awal Ayati, University of Oslo, Norway; Luis Fernando Alzuguir Azevedo, Pontifícia 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. The wave properties such as the wave frequency and amplitude are calculated from the image sequence. In this work, we use a continuum mechanics approach for the description 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 cylinder and the flow regime imposed to the materials.

* 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”

EXPERIMENTAL STUDY ON FRAGMENTATION OF A MOLTEN JET INTO WATER
Miloud Hadj-Achour, University of Lorraine, France; Nicolas Rimbert, University of Lorraine, France; Sebastian Castillón-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 (LS200) will be compared with DNS simulations using the open-source software Gerris. It’s expected that the turbulence and solidification play an important role on the final size of fragments.

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-CNRS, 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 blending behavior is influenced by the difference in shape and size 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 Naiyan Zhang, University of Petroleum Beijing; Jiangdong Kuang, National University of Petroleum Beijing, China; Yunta 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 emulsions 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 containing dispersed and stirred solution. 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, a new model was applied to LES formulation induced a subgrid scale. This work is supported by National Natural Science Foundation of China.

SIMULATION OF THE DYNAMICS COMPLEX INFLUENCE ON HYDROCARBON MIXTURES

Nakul Chaudhary, Interntional Islamic University, Kazakhstan; Kazakh-Turkish University, H.A.Yassawee, Kazakhstan; Dauren Zhakebayev, al-Farabi Kazakh National University, Kazakhstan; Ualikhan Abidbekov, Kazakh Naziona University named after al-Farabi, Kazakhstan.

This work deals with mechanical activation and acoustic impact on the hydrocarbon mixtures. Such action is organized by a special disk 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, forming the 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, 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 experimented, leading to change the concentration distribution inside the 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 hydrocarbonic mix, described above.

TOWARDS LARGE-EDDY SIMULATION OF LIQUID ATOZMIZATION: A-PRIORI SUBGRID ANALYSIS

Sebastian Kettler, Universität der Bundeswehr München, Germany; Markus Klein, Universität der Bundeswehr München, Germany; 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 single-phase Navier–Stokes equations, which were not present in single-phase flow. For 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 mixing layer in a homogeneous initial condition is presented, 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 Inside-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 CONSTRIC TED 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; Nobelito Mangiapacchi, 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 technological applications. Two-phase flow in 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 single-phase Navier–Stokes equations, which were not present in single-phase flow. In this study, numerical simulations of 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 in the direction of the flow. This allows studying 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.

* Support from the Beijing youth talent program funding (YETP0679) is gratefully acknowledged.
STUDY ON SEAWEED REMOVAL EFFICIENCY FROM THE WASTEWATER BY SWIRL FLOW SEPARATOR
Hiroaki Tsuhone, National Institute of Technology, Arika College, Japan; Masaya Shimomura, National Institute of Technology, Arika College, Japan; Syuichi Takesue, National Institute of Technology, Arika 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 seaweeds. In addition, the seaweeds settled in the canals 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 wastewater from the seaweed, 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.

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/ w/o cavitation were compared to examine the cavitation effects for acceleration. 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 the 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.

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; Mehrdigi Hemati, Laboratoire de Génie Chimique de Toulouse (LGCC), 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, 2005) reported that for bed temperatures lower than a critical value (< 850°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 couple 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 mass 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.
DIRECT NUMERICAL SIMULATION OF COMBUSTION NOISE OF A SPRAY FLAME
Abhishek Pillai, Kyoto University, Japan; Tomohiko Kitano, Kyoto University, Japan; Yoshiyuki 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 ethanolaer 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 Arhenius formation and the increase in the skin-friction drag. The streamline 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 attenuated become 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.

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; Jianwen Pan, 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 redistribution in the boundary layer integral parameters and the increase in the skin-friction drag. The streamline 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 attenuated increase 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.

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 ratios. In the numerical simulations, a three-dimensional model of the same geometry as the tested 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 those obtained 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.

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 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.
INVESTIGATION OF DRAG FORCE ON FIBRES OF BONDED SPHERICAL ELEMENTS USING A COUPLED CD-FDEM APPROACH
Anna Lyfne Jensen, Aalborg University, Denmark; Henrik Soerensen, Aalborg University, Denmark; Lars Rosen Dahl, Aalborg University, Denmark; Pål Adamsen, Grundfos, Denmark; Flemming Lykholdt Ustrup, Grundfos, Denmark
Clogging in waste water pumps leads to expensive downtime of pumping systems. The risk of pollution in the pipes 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, e.g. filament objects, in the waste water. Our simulation of clogging effects requires simulation of flexible objects and the interaction between these objects and fluid. This can be accomplished by a coupled CD-FDEM approach. Using this approach, the flexible objects are not resolved by the fluid solver but are modeled 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 solid sphere. 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 fibre is tested.

* Support from Grundfos is gratefully acknowledged.

DNS OF TURBULENT CHANNEL FLOW LADEN WITH A DENSE SUSPENSION OF NEUTRALY BUOYANT FINITE-SIZE PARTICLES
Mehdi Nazir Ardakani, 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
Suspended 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 theses 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 Reb = 5600. In this study the mean particle diameter Dp is about 20 viscous wall units with the volume fraction varying between Φ = 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.

UNCERTAINTY QUANTIFICATION OF INJECTED DROPLET SIZE IN MONO-DISPERSE EUCLERIAN SIMULATIONS
Thea Lancien, CNRS, CentraleSupélec, Université Paris-Saclay, France; Nicolas Dumont, CNRS, CentraleSupélec, Université Paris-Saclay, France; Olivier Giqcuell, CNRS, CentraleSupélec, Université Paris-Saclay, France; Ronan Vicqulin, 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 though uncertainty quantification. Using Polynomial Chaos Expansion and Clenshaw-Curtis cubature rule, several LES are performed for different injected droplet size in order to obtain a sensitive 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-Z60164 from GENCi (Grand Equipement National de Calcul Intensif) is gratefully acknowledged.

SUSPENSIONS OF INERTIAL SPHERICAL PARTICLES IN TURBULENT VERTICAL CHANNEL FLOW
Heige Andersson, Norwegian University of Science and Technology, Norway; Nirayan Reddy Challabotla, Norwegian University of Science and Technology, Norway; Ulla Zhao, Norwegian University of Science and Technology, Norway
The dynamic behavior of inertial spherical particles suspended in a turbulent vertical channel flow has been investigated by means of Eulerian-Lagrangian direct numerical simulations (DNS). 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 there 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 spheres’ trajectories. The results are compared to 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.

* Support from Grundfos is gratefully acknowledged.

TWO-SIZE MOMENT EUCLERIAN MULTI-FUID 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, 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 simulations of such flows. Recently, some hybrid Eulerian methods between that combine the multi-fluid method - one where the size is discretized - and the moment onemosme thodones 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 wernahas 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 crossing the fact that a particle is a size. The authors 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 presented. This new model is 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 Kruerten, Eindhoven University of Technology, The Netherlands; Cees Van Der Geel, Eindhoven University of Technology, The Netherlands; Giel Prieds, 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.

ISOADV: A VOF ALGORITHM FOR SOLVING THE SHARP INTERFACE ADEPTION PROBLEM ON ARBITRARY MESHES

Andreas Rendyby, DHI; Dennis Rensonck, 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 ε and 1 − ε, where typically ε ~ 10⁻⁶. For such cells we find the internal iso-interface, or isoface, cutting it into two parts containing each fluid, say, water and air, respectively. Using different isovolumes 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 δt by interpolating the velocity field to the isoface centre. This method is similar in terms of speed. 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 δτ, we analytically integrate the evolving "wetted" face area, to obtain an approximation of the water volume crossing the polygonal face during δτ. 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.

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 drops as oral 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 COMPRRESSIBILITY EFFECTS IN THE DROPLLET 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.

DYNAMICS OF DOUBLE EMULSION DROPLETS IN SHEAR AND CHANNEL FLOWS

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 drops as oral 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.

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; Prasanta Kumar Das, Indian Institute of Technology Kharagpur, India

Ferrofluid (FF) is a colloidal suspension of surfactant coated magnetic nanoparticles (MNPs, characteristic size ~ 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 ~ 1 mm) immersed in a ferrofluid inside a Hele-Shaw cell (wall gap ~ 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 CSR, India is gratefully acknowledged.
DEVELOPMENT OF CLOSURE MODELS FOR UNRESOLVED PARTICLE SIMULATIONS OF HEAT AND MASS TRANSFER IN DENSE PARTICLE FLOWS USING CPPPO: A LIBRARY FOR FAST PARALLEL DATA FILTERING

Federico Munichini, TU Graz, Austria; Stefan Radl, TU Graz, Austria; Christoph Gomiva, 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 open-source 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.

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 single particle or the volumetric ensemble, and dense (limit) and rarefied (limit) bubbly flow can be derived analytically. Explicit closures are instead not generally known for arbitrary particle density and particle shapes and we have to rely on approximations. 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.

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; Matte Hoffmann, University of Stuttgart, Germany; Philipp Offenhaeuser, University of Stuttgart, Germany; Uwe Ihms, 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 void fraction and temperature. The approach is two-phase and 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 Azopardi, 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 μm) has been fluidized using nitrogen in a 127 mm diameter bed. Both aerated and bubbly flows were measured. The void fraction and temperature 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 information about their properties and the knowledge of the cross-sectional distribution of solids fraction. An iterative reconstruction approach has been used to provide such information. The size 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.

STUDY ON THE COMPRESSION, FLOW AND PERMEATION OF FINE AND ULTRA-FINE, COHESIVE AND COMPRESSIBLE POWDERS

Abbas Karranani Manani, Chair of Mechanical Process Engineering, Germany; Andreas Bueck, Otto von Guericke University, Germany; Sergei Antonyuk, Kaiserslautern University of Technology, Germany; Dominique Thevenin, Otto von Guericke University, Germany; Jurgen Tomas, Chair of Mechan. The solids fraction and temperature 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 information about their properties and help to specify their flow behavior. The fluidization behavior of ultrafine cohesive powders is 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.

* Support from grant DFG-GRK 1554 “Micro-Macro Interactions in Structured Media and Particulate Systems” is gratefully acknowledged.

EVALUATION OF VOID FRACTION AROUND A TUBE IN TWO-PHASE FLOW ACROSS HORIZONTAL TUBE BUNDLE

Hideni 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 × 90 mm2 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 ~ 0.43 m/s, superficial liquid velocity, JL, of 0.2 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.

* Support from grant DFG-GRK 1554 “Micro-Macro Interactions in Structured Media and Particulate Systems” is gratefully acknowledged.

POWDERS

FINE AND ULTRA-FINE, COHESIVE AND COMPRESSIBLE

Barry James Azopardi, 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 μm) has been fluidized using nitrogen in a 127 mm diameter bed. Both aerated and bubbly flows were measured. The void fraction and temperature 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 information about their properties and the knowledge of the cross-sectional distribution of solids fraction. An iterative reconstruction approach has been used to provide such information. The size 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.

* Support from grant DFG-GRK 1554 “Micro-Macro Interactions in Structured Media and Particulate Systems” is gratefully acknowledged.
CDF-INFORMED UNIFIED CLOSURE RELATION FOR THE RISE VELOCITY OF TAYLOR BUBBLES IN PIPES WITH LIQUID FLOW

Eugene Garcia, MIT, USA; Jacobo Buongiorno, Massachusetts Institute of Technology, USA; Eissa Al Safran, Kuwait University, Kuwait; Djanel 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 for the Chebyshev collocation method is used to solve the system of ODEs. The results are compared with experimental data and with numerical solutions from the literature.

NUMERICAL ANALYSIS OF FLOW CHARACTERISTICS INSIDE HYDRAULIC COUPLING AT VARIOUS OPERATIONAL CONDITIONS

Minhee Kwak, Sogang University, Republic of Korea; Naèmeoun Hur, Sogang University, Republic of Korea; Choong-Soo Chang, VS ENG, Republic of Korea; Nam-Won Kang, VS ENG, Republic of Korea

Stochastic couplings are employed for the investigation of liquid flow in a pipe. A series of simulations is performed for different pipe diameters, Reynolds numbers, and liquid mass flow rates. The results are compared with experimental data and with numerical solutions from the literature.

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 spectrum of eigenvalues and associated eigenvectors. The most unstable mechanisms for such phenomenon.

INVESTIGATION OF PARALLEL CHANNEL INSTABILITIES WITH VAPOR BACKFLOW USING AN IMPROVED DYNAMIC MICROCHANNEL EVAPORATOR MODEL

Tom Saenen, ÉCOLE POLYTECHNIQUE FÉDÉRALE DE LAUSANNE, Switzerland; John Thorne, 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.

INVESTIGATION OF STRATIFIED TWO THERMOVISCOUS LIQUIDS FLOW IN A PLANE CHANNEL

Victor Koreev, Mavlutov Institute of Mechanics, Russian Federation; Adelina Nizanova, Mavlutov Institute of Mechanics, Russian Federation; Said Urmanceev, 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 flow parameters and temperature has been developed. In the simple case of one-layer plane Poiseuille isothermal flow, this system of ODE can be reduced to a non-linear 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 thin viscous 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.
CVD SIMULATION OF GROWING NANOPARTICLES IN A FSP FLAME USING DIRECT QUADRATURE METHOD OF MOMENTS
Pedro Bianchi, University of Blumenau, Brazil; Dircero Noriler, University of Blumenau (FURB), Brazil; Henry França Meier, Regional University of Blumenau (FURB), Brazil; Udo Fritsching, Foundation Institute of Matenral Science, Univ. Bremen, Germany; Florian Mehrer, 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 phase front of the combustion phenomenon has been studied to both CFD and 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.
VAPOORIZATION 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 hypothesis to 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 such multiphase systems 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 RAG 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 Vasilev, 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 their 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 subjected). 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 m) and high-inertial (50 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 System, 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, understanding 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 purpose, the behaviour of a crystal fluid 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 CFDDEMcoupling, connecting OpenFOAM with LIGGGHTS. After validating the simulation results with experimental measurements, CFD-DEM results are used for process understanding and improvement.

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; Joel Dupays, ONERA, France; Frederique Laurent, EM2C, CentraleSupélec, France; Pantelis Koulapis, University of Cyprus, Cyprus; Stavros Kassinos, University of Cyprus, Cyprus.

Polydispers 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 designed to a mathematically well-posed system of equations describing low inertia particles. In such cases, a gradient of 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 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 HECOMAX FP7/2007-2013 under grant agreement No 315760.
TRANSITION BETWEEN ROBUST EQUILIBRIUM DESIGN POINTS FOR STEADY STATE PROCESSES OF PULVERIZED COAL COMBUSTION

Tal Eluk, Ben Gurion University of the Negev, Israel; Avi Levy, Ben Gurion University of the Negev, Israel; Efim Korkyhi, Bergmann Campus, Ben-Gurion University of the Negev, Israel; Tal 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 data (Temperature and flue gas) the model was used for studying the various parameters affecting the combustion in downward oriented reactors and process optimization. Sponetic design found to act as strong stability of combustion structure. The peak temperature location along the reactor was observed after altering some operating parameters such as inlet temperature and mass flow rates. Using those parameters affected the reactor 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.

Optimising technology for carbon capture and sequestration (CCS) is oxygen-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-temperature reactions and process optimization. Sponetic design was found to act as strong stability of combustion structure. The peak temperature location along the reactor was observed after altering some operating parameters such as inlet temperature and mass flow rates. Using those parameters affected the reactor 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.

MEASUREMENTS OF VOID FRACTION, PRESSURE DROP AND HEAT TRANSFER IN HORIZONTAL AND DOWNWARD INCLINED GAS-LIQUID STRATIFIED FLOW

Swanand Bhagwat, Oklahoma State University, USA; Afsin 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 EQUIPMENT FOR SUSPENSION FIRING IN UTILITY BOILERS

Chungen Yin, Aalborg University, Denmark; Maryam Momeni, Aalborg University, Denmark; Soren Knudsen Kær, Aalborg University, Denmark.

This paper presents a comprehensive study of biomass particle combustion using highly diverse conditions as determined in suspension firing (SF) tests. Wood pellets are pulverized by hammer mills and roll mills, respectively, to prepare solid biomass feedstock. The actual particle density is measured by the Archimedes 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 then performed in a laboratory scale  combustor and compared to those in suspension-fired utility boilers as possible. A charge-coupled device camera is used to record the whole combustion process. The ignition, stabilization and burnout times of different biomass particles under different conditions are analyzed. A one-dimensional transient model, which not only properly accounts for the particle surface-ambient flow interaction but also 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 NANO PARTICLE SYNTHESIS

Florian Meierhofer, University of Bremen, Germany; Lizzo Buss, IWT Foundation Institute of Material Science, Germany; Dierk Raddi, University of Blumenau (FURB), Brazil; Henry Franz Meier, Regional University of Blumenau (FURB), Brazil; Udo Fritschiing, Foundation Institute of Material Science, University of 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 performing a single particle combustion reaction dispersed by a gas stream to 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 flame structure and ignition. This is achieved by controlling the ignition delay of the particle spray by changing the reactor structure (e.g. devolatilization, char oxidation) in an atmosphere that is different from air. Indeed, if even some experimental data are available on oxygen-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 heterogeneus kinetics in oxygen-fuel combustion from experiments performed in a pilot-scale drop tube reactor, which ensures high-temperature reactors and process optimization. Sponetic design was found to act as strong stability of combustion structure. The peak temperature location along the reactor was observed after altering some operating parameters such as inlet temperature and mass flow rates. Using those parameters affected the reactor 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.

STUDY OF A TURBULENCE DAMPING PROCEDURE NEAR THE INTERFACE OF GAS-LIQUID STRATIFIED FLOW

Vinsh 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. The turbulent Reynolds stress is ignored. The resulting source term 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 turbulence, and heat transfer 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.
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 Croce 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 dispersion phase for different values of the particle Stokes numbers and the persistence length. Particle dispersion and concentration, as well as 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.

### DNS of Evaporating Droplets in Decaying Isotropic Turbulence

We have performed direct numerical simulation (DNS) of decaying isotropic turbulence laden with thousands of evaporating droplets of Taylor length scale. 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, the mean Nusselt 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 Autónoma de Occidente, Colombia; Martín Sommerfeld, Martin Luther-University Halle-Wittenberg, Germany

The handling of powders though 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. The handling of powders though 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 many practical cases they are generally non-spherical. Unfortunately, the fluid forces acting on non-spherical particles for larger particle Reynolds numbers are only known. 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.

### 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 Vernel, 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 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.

### NUMERICAL SIMULATION AND VALIDATION OF ELONGATED FIBRES DISPERSION AND TRANSPORT

Santiago Lain, Universidad Autónoma de Occidente, Colombia; Brian Quintero, Universidad Autónoma de Occidente, Colombia; Martín Sommerfeld, Martin Luther-University Halle-Wittenberg, Germany

The handling of powders though 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 many practical cases they are generally non-spherical. Unfortunately, the fluid forces acting on non-spherical particles for larger particle Reynolds numbers are only known. 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.

### 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 Vernel, 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 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.

### Deviatoric Stresses in Suspensions of Flexible or Curved Fibers

Stefan Lindström, 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 embedded in 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.

* Financial support from AIC Androsh International Management Consulting GmbH 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 certain critical 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.

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

A CLSVOF method is coupled with an immersed boundary method by adopting a relevant work recently published. Via a fluid fracture 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-penetrating 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 Piccardi, Politecnico di Milano, Italy; Andrea Montorfano, Politecnico di Milano, Italy; Jerome Hellé, 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 liquid/gas interface and time-resolved turbulence modulating of dynamical 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 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 by performing pressure-correction equation to enforce continuity after topological changes in the pressure-velocity coupling algorithm of the solver and the solver is converged 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; Berhard 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 fluid 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 are 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. 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. 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.

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 1.1 and 1.1 μm/pixel, and by fluorescence microscopy at 0.2 μm/pixel. Results are performed shortly after breakup the DSDs are bimodal. The primary mode corresponding to 1-20 μm 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 μm, 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 μm) as the DOR increases from 0 to 0.5 and then to 1-100 and 1-2.5. * Support from NSF-CBET 1236462 and 1336232 grants is gratefully acknowledged.

HIGH FIDELITY SIMULATION OF IMPACT OF HIGH TEMPERATURE AND PRESSURE ON LIQUID JET IN CROSSFLOW ATOMIZATION
Marinos Sotiriou, 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 combuster 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 process. The resulting study is one of investigating 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 reheteroth in the development of 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.

RECRYCLATION, ENTRAINMENT AND CLUSTER FORMATION IN BOUNDED SPRAYS
Aljoscha Lampa, University of Bremen, Germany; Soren Sander, Foundation Institute of Material Science Bremen, Germany; Daniel Schwenck, University of Bremen, Germany; Udo Fritsche, 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 inhomogeneties are identified and quantified with a box-counting algorithm (Garmcarek’s method). The spray structures in- and outside the central spray cone flow like the recirculation zone 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 2P-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.
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; Jianwen Yan, 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. The drag force acting on a particle in a fluidized bed (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 filter provides insights into the behavior of numerical simulations. The average DEM-type force is about 20-30 % smaller than the DNS-type value. 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 monodispersive dynamic fluidized bed.

NUMERICAL SIMULATION OF PERIODIC CIRCULATING FLUIDIZED BED OF BINARY MIXTURE OF PARTICLES – BUDGET ANALYSIS
Solene Cheverier, 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 characteristic 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 soln. These contributions can be applied in order to measure such subgrid contributions appearing in the theoretical filtered approach. Such kind of numerical simulation is very expensive and is developed to very simple configurations. In the present study, the Euler-Euler 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.

INSTABILITY DUE TO INTERFACIAL TENSION IN STRATIFIED 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 is used to develop the mechanism of transition models based solely on data of pipe flow of low viscosity fluids.

NUMERICAL ANALYSIS OF DENSE FLUIDIZATION: A BENCHMARK FOR PARTICLE RESOLVED SIMULATIONS OF LIQUID- LIQUID FLUIDIZED SOLID-BEDS
Amir Esteghamatian, Ecole Centrale de Lyon, France; Manuel Bernard, IFP Energies nouvelles (IPFN), France; Abdelkader Hamoudi, IFP Energies nouvelles (IPFN), France; Anthony Wachs, University of British Columbia, Canada; Markus Uhlmann, Karlsruhe Institute of Technology (KIT), 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 multi-phase 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-contact surface 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 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.

DRAG FORCE MODELLING IN DILUTE TO DENSE PARTICLE-LADEN FLOWS WITH MONO-DISPERSIVE OR BINARY MIXTURE OF SOLID PARTICLES
Olivier Simonin, INP Toulouse, France; Solene Cheverier, 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 beds: dilute and dense regime, low and high particle Reynolds numbers, 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).

* Support from the National Natural Science Foundations of China (Grant Nos. 51390491, 51390493) is gratefully acknowledged.

* Support from the National Natural Science Foundation of China (NSFC) under Grant No. 51490617, 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 STRATIFIED 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; Yaming Li, Xi'an Jiaotong University, China; Qiang Zhang, Xi'an Jiaotong-Liverpool 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 the 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 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 pressure.
* Support from Grant No. 2015CB251502 is gratefully acknowledged.

VISCOSOUS VORTEX RECONNECTION IN THE CAVITATING SWIRL FLOW
Alekseenko, Institute of Thermophysics SB RAS, Russian Federation; Sergey Shlik, Institute of Thermophysics SB RAS, Russian Federation; Sergey Alekseenko, Institute of Thermophysics SB RAS, Russian Federation; Pavel Kulinich, 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, consistent with 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.

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 gradient in temperature and concentration across a large range of scales. Droplets 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 steam in a cooler, less dense fluid. The inhomogeneous thermodynamic and hydrodynamic conditions 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.

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 obtained and discussed regarding an approach of 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.

A PRESSURE DISCONTINUOUS CONTROL VOLUME VOLUME 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; Zhuhua 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-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) 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.

STATISTICAL MODELLING OF TURBULENCE INTERFACE INTERACTIONS WITH THE CONSERVATIVE LEVEL-SET METHOD
Tomasz Wacławczyk, Warsaw University of Technology, Poland
Based on studies in Wacławczyk M. & Wacławczyk T., Int. J. Heat Fluid Flow 52 (2015) in Wacławczyk 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 L3 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 $\epsilon_1 = CL$ ($L$ is the length scale) and the relaxation velocity $\epsilon_2$. For $\epsilon_2 = 0$ and $C = \text{const}$, sharp representation of the interface is recovered, when $\epsilon_2 \neq 0$ the new formulation is similar to the standard L3 method introduced by Sussman et al., 114 (1994). In the present work we address extension of this new method to the case when $\epsilon_2$ 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

Norooz Ul Haq Khatri, University of Tokyo, Japan; Shu Takagi, University of Tokyo, Japan; Xiaobo Gong, Shanghai Jiao Tong University, China; Yuichiro Matsumoto, RIKEN, Japan

For bubbly flow with biochemical reactions, all the analyses including overall fluid flow, bubble motion, bubble dissolution at local level and local bubble reactions/consumption of substrates are important. The developed system is provided by mixed Eulerian-Lagrangian formulation in which liquid media is represented by Eulerian elements, whereas bubbles are tracked individually. 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 alternative kinetic gas dissolution and mixing phenomena. 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.


MODELLING OF SLURRY STAGED BUBBLE COLUMN FOR CO2 CAPTURE IN CARBONATES SOLUTION WITH ENZYME CATALYSIS

Antonio Marzocchella, Università degli Studi di Napoli Federico II, Italy; Maria Eleonora Rezza, Consiglio Nazionale delle Ricerche, Italy; Piero Barcinski, Università degli Studi del Sannio, Italy; Giuseppe Oliveri, Università degli Studi di Napoli Federico II, Italy; Riccardo Chironi, Consiglio Nazionale delle Ricerche, Italy

In biological wastewater purification involving reactions using biocatalyst, the CO2 capture rate poorly increased when dissolved CA was used within the slurry biocatalyst was modelled as pseudo-homogeneous fine dispersed particles. The reversible Michaelis and Menten kinetics described CO2 conversion by the slurry biocatalyst. The role of mass transfer rate and heterogeneous enzyme catalysis on the CO2 capture rate and on the CO2 concentration profile in the liquid boundary layer was investigated. Simulation showed that the CO2 capture rate poorly increased when dissolved CA was used within the enzyme solution limit, and that in the presence of CA immobilized on fine particles the CO2 absorption rate was enhanced about three fold with respect to the slurry biocatalyst.

*Support from grant PON03PE_00157_1 ‘Smart generation’ is gratefully acknowledged.

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

In bubbly flow with biochemical reactions, all the analyses including overall fluid flow, bubble motion, bubble dissolution at local level and local bubble reactions/consumption of substrates are important. The developed system is provided by mixed Eulerian-Lagrangian formulation in which liquid media is represented by Eulerian elements, whereas bubbles are tracked individually. 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 alternative kinetic gas dissolution and mixing phenomena. 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.


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

In 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 real-time via 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®.

The developed solver has been tested for the absorption CO2 in water and the reaction CO2 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-671/1-1).

THE WAKE EFFECT OF SOLID PARTICLES AND COMBUSTING DROPLETS ON TURBULENCE

Liuxing Zhou, Tsinghua University, China; Ke Li, Innermogolia University of Science and Technology, China; Zhuxiong 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 combustion 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 combusting 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 combusting droplet is reducing the gas turbulence, and the drag force of an evaporating/combusting droplet is much smaller that 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 Arca Sprintan. 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-axially independently rotating cylinders) is simulated using a two-way coupled Euler-Lagrange approach. While the flow exhibits several characteristic phenomena that report development of Eulerian CFD models capable of predicting over clear pressure drop and liquid hold-up, there are only a few papers that investigated the ability of these models to predict the local and dispersed phase phenomena. 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 (0.023 m/sec), gas velocity (0.28 m/sec) and particle size (10 mm). The results provide insights into the interfacial anti-diffusion that may lead to inaccurate results.

FREE SURFACE FLOW IMPROVEMENTS FOR POOR MESH

Vineet Kumar Gupta, ANSYS Inc., India; Hernant Pandekar, ANSYS Inc., India; Srikhand Kvs, 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 which 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.

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 is 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 model for the deposition of particles on the 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

Danilusz Asendych, Institute of Thermal Machinery, Poland; Pawel Niegadowski, Czestochowa University of Technology, Poland

The paper presents 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 appropriate 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 distribution action of the packing, turned out to be independent of liquid flow confirming the opinions available in the source literature.
SIMULATION OF FIBER SUSPENSION IN A TURBULENT MIXING LAYER WITH AN EUCLER-LAGRANGIAN METHOD
Kai-Dong Wang, Zhaohui Liu, 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 infinitely thin 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 shape 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 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 characteristic of the multiphase flow is 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-ε turbulence model, Chang-Hsieh-Chen, is applied to study the turbulent pulp flow. The standard damping function $f_{\theta}$ 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 position, the vortex of the suction region gives rise to a two-phase flow structure. 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-particle interaction. The results clearly agreed with the pointwise model and other experimental and numerical studies on this topic.

TURBULENT DISPERSION OF ELLIPSOIDAL PARTICLES IN A WALL-BOUND TURBULENT FLOW
Rafik Ouachene, Université LUMA - LEMTA, France; Souira Hamidouche, Université de Lorraine - LEMTA, France; Boris Arcen, Université de Lorraine - LRGP, France; Mohammed Khajji, 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 this paper a new methodology 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, simulations of pointwise particles have been carried out. The aim of the present work is to explore if this is still the case when size effects are taken into account in the DNS. In the drag-reducing regime, we present the distribution of the viscosity stress, and the variance and co-variance components of the Reynolds stress tensor. We show the reliability of DNS for measuring time averaged turbulent fluctuations. We show that DNS/LPT 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 being enhanced 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 flow (particle size, concentration and inertia) and the 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 found to be close 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 Simonin, 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 Reynolds 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 rarefied 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 Giazzelli, Au-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 particles 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 chain dynamics in colloidal 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 steady Couette flow. 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 rarefied granular flow with interstitial fluid effect (Parmentier and Simonin, JFM 2012; Abbas et al. AIChE 2010)

ANALYZING THE RELATION BETWEEN FINITE-SIZE RESOLVED PARTICLES AND COHERENT STRUCTURES IN FORCED HOMOGENEOUS-ISOtropic TURBULENCE
Markus Uhlmann, Karlsruhe Institute of Technology (KIT), Germany; Agathe Choulippe, 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 Tobacco 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 eddies and 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.

EVIDENCE OF SHEAR-INDUCED MIGRATION IN A PIPE FLOW OF A DENSE EMMULSION
Micheline Abbas, LGC-CNRS/University of Toulouse, France; Ameule Pouplin, LGC-CNRS/University of Toulouse, France; Olivier Masbemart, Timmoine, 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 parabolic behavior, irregularly, from the high to the low shear rate regions of the flow. This phenomenon has been identified in Couette viscometers, but 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 chain dynamics in colloidal 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 steady Couette flow. 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 rarefied granular flow with interstitial fluid effect (Parmentier and Simonin, JFM 2012; Abbas et al. AIChE 2010)

SHEAR FLOW-INDUCED ENHANCEMENT IN EFFECTIVE THERMAL CONDUCTIVITY OF NANOPARTICLE SUSPENSIONS
Chengzhun 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 high viscosity 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 H2O+SiO2 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 shear rate. At lower shear rates, ETC reached a plateau value. We attributed this phenomenon to the dispersion of initially aggregated nanoparticles at high shear rates. We also found that the dependence of ETC with the shear flow 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 temperature, 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 Frunieri, 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; Frederik 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 presented in this talk is the GENTOP concept [1], which is based on the multi-scale 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, Euler-ψ 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.


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 gas fluid method is utilized to deal with the jump conditions. The inflow boundary condition is carefully conducted as 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 study, and 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 one. The breakup in the liquid phase can reduce 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 LR12ED001, 51222602, 51276163 is gratefully acknowledged.

DIRECT NUMERICAL SIMULATION OF FLUIDIZATION OF SOLID PARTICLES DUE TO LIFT FORCES

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 first 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 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
Thanh Mai Ta, CNRS and Grenoble university, France; Federico 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 2D and the 3D test cases. The convergence in 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 preliminaries 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 Estelvazieres, 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 (in 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 Haghgoo, 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 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 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.

ICMF 2016 International Conference on Multiphase Flow
Firenze, Italy, May 22 - 27, 2016
LAGRANGIAN MEASUREMENT OF FORCE ON COARSE OBJECTS IN GAS-SOLID FLUIDIZED BED

Wataru Yoshimoto, Hokkaido University, Japan; Tomoki Ikegaki, Hokkaido University, Japan; Kyotets Higashida, Osaka University, Japan; Kenta Rai, Osaka University, Japan; Takuya Tsujii, 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 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 system can be used in the interaction between the two different computational grids and a fine Cartesian mesh are used for the FSI multilevel algorithm coupled to an efficient compression-expansion technique is considered for the reconstruction and advection of the interface. A PLIC monolithic approach for the FSI problem while a Volume Of Fluid method (VOF) is discussed. In the latter case, the translational motion of the body and its rotation about its center of mass are described by the Newton-Euler equations. Schlieren 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.

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 due to the cohesion of the particles. Therefore, the evaluation of flowability of powder in the air is 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)2 A and B of the different particle diameter were used. As the experimental conditions, the initial void fraction and excitation force pressures on the orifice 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)2 B of the small particle diameter and the high initial void fraction and excitation force pressures on the orifice increased the powder discharge rate. It is shown that the flowability of Ca(OH)2 A was higher than that of Ca(OH)2 B of large particle diameter. This result indicate that the influence of the cohesion force to Ca(OH)2 B was strong. Furthermore, the mass flow rate of Ca(OH)2 B through an orifice was decreased. We can infer from this result that flowability of Ca(OH)2 B is low.

COMPARATIVE STUDIES OF SHOCK INTERACTION WITH STATIC AND MOVING CYLINDERS THROUGH IMMERSED BOUNDARY METHOD

Yijuan 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 immersed 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 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 the fixed origin of the coordinate as reference by comparing the positions of triple points of static and moving cylinder. 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 provided.

INVESTIGATION OF GAS-LIQUID TWO-PHASE FLOW INDUCED VIBRATION CHARACTERISTICS

Shuichiho Miwa, Hokkaido University, Japan; Yang Lu, Virginia Tech., USA; Hikoi Takashii, 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 fluctuations. The characteristics of force fluctuations on the elbow were discussed with experimental investigations and numerical simulations. The 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 results indicate that the ratio of separation force and gravity force of Ca(OH)2 B was higher than that of Ca(OH)2 A of large particle diameter. This result indicate that the influence of the cohesion force to Ca(OH)2 B was strong. Furthermore, the mass flow rate of Ca(OH)2 B through an orifice was decreased. We can infer from this result that flowability of Ca(OH)2 B is low.

A FLUID-STRUCTURE INTERACTION SOLVER COUPLED TO A VOLUME OF FLUID METHOD

Daniele Cerroni, University of Bologna, Italy; Luca Fancellu, 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 MEDMEN libraries included in the Salome platform. The FSI problem is solved with a parallel multigrid C++ Finite Element code. Several test cases are presented.

MULTIPHASE FORCES ON BEND STRUCTURES

Erik Nennis, TNO, The Netherlands; Stefan Bellford, 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 flexibilities 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 is 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, Statol, Total, Lundin, Akers Solutions, FMC Technologies.
LAYOUT SIMULATION OF VISCOELASTIC FLUIDS USING THE HYBRID IMMERSED-BOUNDARY METHOD

Andreas Meier, Päivi Salmi, Koichi Miyayoshi, Kobe University, Japan; 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 IB-Flow, the in-house multi-phase flow solver at the Fraunhofer-Chalmers Centre. In the solver two phases are modeled by the pseudo-plastic model and the visco-plastic reduction by a general CaReau 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 octave 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 of the slotted nozzle, which are installed on the flat bottom surface and a sufficient supply of air is required to ensure the formation of stable air layer by the coalescence of the bubbles. The air layer drag reduction becomes effective when the air layer thickness and the air flow rate exceeds the pumping power consumption. In this study, a model ship of 50 m medium range tanker is employed to investigate air lubrication method. The experiments were conducted in the 100 m long towing tank facility at the Pusan National University. To conduct the experiments, air injection system, 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 Kulbin, Institute of Thermophysics SB RAS, Russian Federation; Sergey Shitok, 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 of 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

Puyo Saloo, Koichi Miyayoshi, Kobe University, Japan; Kosuke Hayashi, Kobe University, Japan; Akio Taniyama, 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 2×10⁻⁵ to 7×10⁻⁴ m³/s. The gas flow rate from each aeration structure is measured, and through capturing generated bubbles. The number of bubbles outside the pipe are observed using a high-speed camera. Stratified or wavy two-phase flows are formed inside the pipe. At azimuthal angle e and low total gas flow rates, the aeration is likely to be non-uniform due to the transition of the interface 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, van Karman Institute for Fluid Dynamics, Belgium; Philippe Planquart, van Karman Institute for Fluid Dynamics, Belgium; Jean-Marie Buchlin, van 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 structures 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 at the Hands-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; Oulu Tammisola, 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 out by O Naragh et al. (IJFM 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 (TFLS) 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 level 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. To simulate this natural motility, we study a simple model of a flexible hoop that can provide fundamental understanding and mimicking of jumping on water. The 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, the hoop is observed. Using the jumping of hoops with different materials, dimensions, and wettability 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 adopting such dynamic characteristics.

**SOLVENT FOAM ON SEGMENTATION FOR CO₂**
Rajat Arora, 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 an 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-NA0002376.

**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 Cramondu, 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 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⁻¹, 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 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 of the impact of the electric field strength upon flow pattern and emulsion structure.

**EFFECT OF SOLID FOAM ON SEGMENTATION FOR CO₂**
Serge Simoens, CNRS, France; Guanry 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 Valorgeux, 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 CO2 in a pH-basis aqueous solution with or without foam. We used a channel of 2 mm x 2 mm square section and 70 mm 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 fluorescent dye for « μ-pH-H2F » 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 epifluorescence microscope with an HBO 100 lamp, 2,5X lens 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 kLa mass transfer coefficient.

**SIMULATIONS OF A SHOCK WAVE - PARTICLE CLOUD INTERACTION IN A SHOCK TUBE**
Bernard Molin, Embry-Riddle Aeronautical University, USA; Subramanian Annamalai, University of Florida, USA; Frederick Ouellet, 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 an 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-NA0002376.

**OIL-WATER PLUG FLOW FORMATION IN SMALL CHANNELS**
Maxime Chinaud, University College London, UK; Evangelia Roumpea, University College London, UK; Lyes Kahouadj, Imperial College London, UK; Claire-Hélène Miara, Imperial 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 phase flow rate ratio at the forming at the forming plug under otherwise constant conditions. The interfacial 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 breakdown. 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, used capable of handling parallel simulations of the two-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 Mahotra, IIT roorkee, India; Sumana Ghoosh, 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 2 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 formed plug 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 CHEMOSORPTION IN A QUIESCENT LIQUID

Daniel Bezenecy, 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 Schleider, 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-bubble 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 chemosorption 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; Zhuhua 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 flame ignited by a premixed coal-oxygen/hydrogen mixture. The sodium release and the formation of sodium-rich ash are captured by the simulations. The sodium release in a pulverized-coal jet flame is found to be limited by the char combustion. A comparison between the LES predictions and the experimental data is performed for a pulverized-coal jet flame, before which the release of sodium is found to be independent of Da and the turbulence is found to slow down the overall consumption rate due to the clustering of particles.
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. 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 trajectories of droplets and bubble sizes on the closure relations the agreement of the simulations with the experimental data, we find both good agreement as well as discrepancies. The large number of bubbles creates a void fraction of 10%. The Reynolds friction number is 180.

Eulerian multi-fluid regime, the flow is simulated over 4 crossing of the channel (of length 2\(r_h\)). 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 evaluate the capability of the proposed model, model predictions were validated against experimental data of Simo etc. at Michigan University. The large-scale cavity profile and the entrainment rate was correctly predicted. The bubbly flow with particles 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.

DNS OF A TURBULENT STEAM/WATER BUBBLY FLOW IN A VERTICAL CHANNEL
Guillaume Bois, CEA Saclay, France; Benoît Mathieu, CEA Saclay, France; Gauthier Fauchet, CEA Saclay, France; Adrien Toutant, Université de Pergipin Via Domitila (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 TriCoFD 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.

Eulerian multi-fluid regime, the flow is simulated over 4 crossing of the channel (of length 2\(r_h\)). 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 NURESSAFE project is gratefully acknowledged.
This work was granted access to the HPC resources of TGCC under the allocation 20XX- t201427239 made by GENCI.
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.

FLOW RHEOLOGY OF FIBER-LADEN AQUEOUS FOAMS
Anit 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 found that the velocity distribution of the foams varied between 60% and 75%. Addition of hardwood fibers at consistency 20 g/kg to plain aqueous foam increases 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%) at the pipe wall. While observed, the normal stresses on 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.

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 Maffei, 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, Tlemoïne, France; Eric Climent, Université de Toulouse, IMPT, CNRS, France

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 and shape of the particles. In complex systems, such as multi species in a suspension, it presents intensely anisotropic and causes particles to cluster easily, which results a non-uniform flow.

BROWNIAN DIFFUSION OF NANO-FIBERS
Lin Tian, RMIT University, Australia; Goodarz Ahmadi, Clarkson University, USA; Jyyuan 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 rotational 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, Xian Jiaotong University, China; Chuguang 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 work, 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 and shape of the particles. In complex systems, such as multi species 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 Bergougoux, Aix-Marseille University, CNRS, France; Gilles Bouchet, Aix-Marseille University, CNRS, France; Elisabeth Guazzelli, Aix-Marseille University, CNRS, France

We present a jointed 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 vertical 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 Doinneau, Sandia National Laboratories, USA; Marco Arlenti, 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 Botte, 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 Alkhami, 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 squilane 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 loosing its validity.

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 multiphase 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.

IN THE FRAME OF THE SAFETY ANALYSIS OF THE FUTURE ASTRID FAST NEUTRON REACTOR, CEA HAS DEVELOPED A MULTIPHASE FLOW SIMULATION PLATFORM, DUNE.

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 SCON software for the caesium-nitric acid 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 several 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 computations are performed between ICE and DUNE, both codes running on the same parallel 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.
**THE DENSE DISCRETE PHASE MODEL FOR SIMULATION OF BUBBLING FLUIDIZED BEDS: VALIDATION AND VERIFICATION**

Schalk, Goede, SINTEF Materials and Chemistry, Norway; Shahrar Ali Amiri, 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 collision balance equations for 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.

**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 frictional effect of the straight fiber to the flow which is 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. The result has shown that the permeability increases 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. The 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.

**MACROSCOPIC MODELLING OF PARTICLE FLUCTUANT KINETIC ENERGY IN A LIQUID FLUIDIZED BED**

Ali Ozel, Princeton University, USA; Jorge Cesar Brandle De Motta, CORIA, France; Olivier Masbernat, Themoine, 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.
LIQUID-GAS DYNAMIC SYSTEM UNDER VIBRATION
Timothy O. Hern, Sandia National Laboratories, USA; John Torczynski, Sandia National Laboratories, USA; Jonathan Claussen, 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 pressure difference, forming two three-phase contact lines, which 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 bifurcations, are revealed by the pseudo-bubbles from the gap geometry or the 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 wettabilities 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 wettabilities determines the volume of the air encapsulated in the trench and the extent of free surface deformation.

THE INFLUENCE OF FLUID VISCOITY 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; XiaoJuan Niu, Xi’an Jiaotong University, China; Jiang Zhe, University of Akron, USA; Liang Zhao, University Of Patras, Greece; YanYi Dimakopoulos, University Of Patras, Greece

Liquid film flow along an inclined, solid substrate flaking 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 Wenzel state. For this reason, for two-phase flow conditions within a trench, one encounters 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 bifurcations, are revealed by the pseudo-bubbles from the gap geometry or the 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 wettabilities 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 wettabilities determines the volume of the air encapsulated in the trench and the extent of free surface deformation.

STABLE FILM FLOW OVER 2D TOPOGRAPHY WITH AIR INCLUSION
Stylianos Varchanis, University of Patras, Greece; Dionisios Peletas, University of Patras, Greece; Yiannis Dimakopoulos, University Of Patras, Greece; John Tsamopoulos, University Of Patras, Greece

When a small amount of gas is added, the Bjerknes forces cause some of the gas to migrate below the piston. The resulting gas regions above and below the piston form a pressure difference, forming two three-phase contact lines, which 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 bifurcations, are revealed by the pseudo-bubbles from the gap geometry or the 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 wettabilities 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 wettabilities determines the volume of the air encapsulated in the trench and the extent of free surface deformation.

THE SWIRLING ANNULAR TWO-PHASE FLOW INSIDE A CIRCULAR PIPE INTERFACIAL PHENOMENA, FLOW CHANNELED AND MODELING
BoFeng Bai, Xian 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

The formation of swirling annular flow is a successful modulation. The interface phenomena and flow mechanism of swirling annular flow in a vertical pipe are well characterized by various 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 streamline direction, the liquid surrounding the gas column bridges, leading to the 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 HARTLACH 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 instability based on the relative magnitude of dynamic pressure of gas and liquid phases.

* From National Natural Science Foundation of China under the contract No. 51276140 is appreciated.

THE MECHANISM AND MODELLING OF SWIRLING ANNULAR FLOW IN RECTANGULAR MILLI-CHANNELS OF MODERATE ASPECT RATIO
Dominique Legendre, Institut de Mécanique des Fluides de Toulouse, France; Thomas Andrieu, 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 Aussillous and Quéret (2000) more recently, less attention has been paid 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 HARTLACH 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 instability based on the relative magnitude of dynamic pressure of gas and liquid phases.

* From the French "Agence Nationale de la Recherche", grant ANRT-09-BLAN-0381-01 (MIGALI) is gratefully acknowledged.

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 Andrieu, Instituto de Mécanica de Fluidos 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 Aussillous and Quéret (2000) more recently, less attention has been paid 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 HARTLACH 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 instability based on the relative magnitude of dynamic pressure of gas and liquid phases.

* From the French "Agence Nationale de la Recherche", grant ANRT-09-BLAN-0381-01 (MIGALI) is gratefully acknowledged.

DYNAMICS OF NEUTRALLY BOUYANT SPHERES IN OBLIQUE MICRO-BIFURCATION CHANNELS
Blaskar Medhi, Indian Institute of Technology, India; Virup Agrawal, Indian Institute of Technology, India; Anugrah Singh, Indian Institute of Technology, India

The formation of swirling annular flow is a successful modulation. The interface phenomena and flow mechanism of swirling annular flow in a vertical pipe are well characterized by various 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 streamline direction, the liquid surrounding the gas column bridges, leading to the 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 HARTLACH 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 instability based on the relative magnitude of dynamic pressure of gas and liquid phases.

* From the French "Agence Nationale de la Recherche", grant ANRT-09-BLAN-0381-01 (MIGALI) is gratefully acknowledged.

FLOW OF NEUTRALLY BUOYANT SUSPENSION IN OBELIQUE MICRO-BIFURCATION CHANNELS
Blaskar Medhi, Indian Institute of Technology, India; Virup Agrawal, Indian Institute of Technology, India; Anugrah Singh, Indian Institute of Technology, India

Flow of knowledge 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 p-PIV and p-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 toward 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 away from the center of the channel. We have observed that the 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.
YIELING THE YIELD-STRESS ANALYSIS: A STUDY FOCUS 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 Tsampopoulos, University Of Patras, Greece

For several decades, Carbopol has been assumed to be the ideal viscoelastic 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 of 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 properties values from the rheological characterization of Carbopol by Holenberg et al. (2012) via the LAOS method. Our study is based on the axisymmetric sedimentation of a particle in materials which exhibit elastoviscoelastic 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; Jeeo 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 porous media [1],[2],[3] and the Method of Moments (MoM) [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.


DIRECT NUMERICAL SIMULATION OF SUPERSONIC TURBULENT SPRAY JET FLAME IN HEATED COFLOW

Tai Jin, Zhejiang University, China; Kun Luo, Zhejiang University, China; Jienen 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 essentially non-oscillatory (WENO) scheme of spatial discretization and total Eulerian-Lagrangian approach. High-resolution bandwidth-optimized weighted Direct numerical simulations of three-dimensional spatially-developing

DEVELOPMENT OF A NEW OPENFOAM VOBF-BASED SOLVER COUPLED WITH POPULATION BALANCES FOR THE SIMULATION OF EXPANDING POLYURETHANE FOAMS

Mohsen Karimi, Chemical Engineering, Politecnico di Torino, Italy; Daniela Marchioli, 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 base-free 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 Modelling Method of Moments (MoM) 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.


DIRECT NUMERICAL SIMULATION OF N-HEPTANE DROPLETS INGINTING IN A TURBULENT FLOW

Abouelmagd Abdelsalami, 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 DINOASIRS. 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 & Sigirnino, 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 $U_c$. 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 CF-DVO 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 Mazzoli, University College London, UK; Paola Lettieri, University College of London, UK
The design and optimization of photobioreactors for intensive microalgae 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 microalgae, light, photosynthesis kinetics and gas-liquid hydrodynamics. A lumped kinetic model of photosynthetic unit (PSU) has been adopted for photosynthetic reactions. The dynamics of a microalg 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 microalg 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 Ripagoudas, 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. The particulate phase is described by terms of particle 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 adopt a novel framework 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 transformation the PBE is reduced 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 BaSO4 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 Timmers, 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 x 0.72R x 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 by 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 x 200 x 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*), u* = 0.68R 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 vortices pairs oriented in the streamwise direction. Detailed statistics of the present flow are compared with previous Reynolds number.
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 exposes that 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.

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 dewatering 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 doing so, 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.

ENERGY EFFICIENT HIGH SOLIDS PROCESSING OF FIBER SUSPENSIONS USING MRI
Maria Cardona, University of California, USA; Emilio Tozzi, Proctor & Gamble, USA; Nandapree Karuna, University of California, USA; Tina Jech, 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 cellulotic 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-Newtontian 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.

FIBERS SETTLING IN CELLULAR FLOW FIELD
Diego Lopez, LMI-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 and angular velocity is determined. Several image filters are used to distinguish 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.

EXPERIMENTAL STUDY ON DISPERSION OF SPHERICAL PARTICLES AND FIBRES IN A JET PENETRATING A CROSSFLOW
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 = 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 flashligh for shadow imaging. The jet is seeded with different types of fibres and spherical particles at low concentration and in addition with 40 µm PMMA tracer for measuring the jet fluid velocity. The arithmetic fibre aspect ratio has values between 6 and 10 for the different particle 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 d10 = 90 µm and d10 = 226 µm (material density of 1.19 g/cm³ and the characteristic turbulent diameter 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-SPERICAL PARTICLES
Bernd W. Haschke, Imperial College London, UK; Thomas Curran, Imperial College London, UK; Fabien Euvrad, 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 Quatamion 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. Constructions are then used to estimate the average gas-solid heat transfer. A budget analysis of the mean fluid contributions to the total drag for each case, to gather data on the drag, torque, and lift on the particle under various Re numbers and angles of attack. All these simulations are coarsegrained into drag, lift and torque correlations. Finally, the resulting relations and collision model are used in a two-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, Universite Paris-Est, France; Jorge Cesar Brandle De Motta, CORIA, France; Eric Climent, Universite de Toulouse, IMFT, CNRS, France; Jean-Luc Estivaleze, ONERA, France
We present in this paper resolved particle direct numerical simulation suspended in homogeneous sustained turbulence. Three particle-fluid density ratios have 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 accommodation time as well as velocity-angle between particles or 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 collision model is used. The influence of the inter-collision time as well as the pdf of orientation from a 2D projection of a specific pattern drawn onto the particles are analyzed. The detection method employs a minimization algorithm to obtain the orientation from a 2D projection of a specific pattern drawn onto the particles. This algorithm has been chosen to determine the margin of rotation 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 (≤ 30) the particles respond to the carrier turbulent flow, while at higher Ga (≥ 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.

MODELLING 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; Sudtheer 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 (PURelBM). 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 evolution 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.

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; Defrel Lohse, University of Twente, The Netherlands; Jean-Luc Estivalezes, ONERA, France
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 particles. This algorithm has been chosen to determine the margin of rotation 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 (≤ 30) the particles respond to the carrier turbulent flow, while at higher Ga (≥ 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 Formari, 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 between 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 0.001 to 200. The particle diameter is chosen to be approximately 12 Kolmogorov length scales 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.

* 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; Andrea Kuhnmann, 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 particle 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 flow field is solved using the particle tracking Lagrangian method ERCOFTAC P-IBM. The computational domain is a particle-laden square cavity and the lid is moving at constant velocity. The particles are assumed to be spheres. The two-way coupling approach is compared with other methods such as the Maxey-Riley equation without a particle-wall interaction which is modelled as an inelastic collision.
VALIDATION OF THE TWOPHASEEULERFOAM SOLVER FOR JET INLET FLUIDIZED BEDS
Giovanni Tretioli, 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 one of the 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 average mesh size on the concentration 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.

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 (DFM, 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 CPFDF SIMULATION AND ECT MEASUREMENT
Haigang Wang, Chinese Academy of Sciences, China; Guizhi Qi, Institute of Engineering Thermophysics, China; Jianbin Ye, Institute of Engineering Thermophysics, China; Zhipeng Wu, School of Electrical and Electric Engineering, UK; Wuxiang 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, especially for a large scale CFB system. In this research, an Eulerian-Lagrangian model called CPFDF 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 CPFDF 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 CPFDF simulation, the effect of drag model on the prediction errors is given. The research presented a new strategy combined CPFDF simulation and ECT plus Doppler measurement, which could provide the whole-loop gas-solids phase flow hydrodynamics characteristics in a CFB system. The results provide valuable information for the optimum design of a circulating fluidized bed.

* CITATION: From grant NSFC No.51320106004 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; Ladlani 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 compensated 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-5um. 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 microstructured 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–COUETTE 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 polymers 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 UNDERSTANDING OF FABRICATING PROCESS OF CERAMIC MATRIX COMPOSITES (CMCs).

Audrey Pons, SAFRAN Herakles / IMS, France; Eric Arquis, CNRS, France; Marc Valat, CNRS, France; Cedric Le Bol, CNRS, France; Cedric Descamps, SAFRAN Herakles, France; Alexandre Alric, SAFRAN Herakles, 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 dead walls can increase the surface area 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 Thetis (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 Bösanquet’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 thermanl field.

* This work is financially supported by SAFRAN Herakles and the ANRT.

DYNAMICS OF A RISING BUBBLE IN A VISCOELASTIC LIQUID

Matthias Niedhammer, Technische Universität Darmstadt, Germany; Holger Marshall, 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-constitutive equations, the elastic and viscous contributions of which are formulated to obtain numerically more robust representations for high Feinstein number (HFN) 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 equations and applying volume of fluid (VOF) 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 tailing 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 Weise 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
Mitsuhito Ohta, Tokushima University, Japan; Mao Ogawa, Tokushima University, Japan; Shulchi Iwata, Nagoya Institute of Technology, Japan
It is acknowledged to the collaborative research by Soto et al. (2008) 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 PrimalTT TT-935 and PrimalTM TT-615 are quite different. Second, and surprisingly, it is observed that the unique bubble dynamics/bubble-shapes observed previously 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.

EXPERIMENTAL STUDY ON STRUCTURE OF A DILUTE GAS-LIQUID TWO-PHASE FLOW IN A VERTICAL PIPE USING AN ALGEBRAIC SECOND MOMENT CLOSURE
N.A. Adoo, University of Saskatchewan, Canada; A. S. M. Atiqu 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 flow conditions in the gas phase, 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 (EARS). 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.

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 Misha, 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 nozzle 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 vicinity 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 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.

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 (FSS3D), 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 polycrylamide Prinston into a computational domain with over 700 million cells. We simulate the injection velocity profiles of different nozzle 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.

* Support from grant Nos. 51300605, 51309494, 51276076 and FKSLCBB1409 are gratefully acknowledged.

PREDICTION OF GAS VOLUME FRACTION FOR POLY-DISPERSE GAS-LIQUID TURBULENT FLOW IN A VERTICAL PIPE
A. S. M. Atiqu 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 (RANS) 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.

MEASURED BY USING INTRUSIVE PROBE METHODS
Turbulent 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. For the German Research Foundation (DFG) under the Priority Programme SPP 1423 is gratefully acknowledged.

Support for the German Research Foundation (DFG) under the Priority Programme SPP 1423 is gratefully acknowledged.
CFD MODELLING OF POLYDISPERSE BUBBLE FLOWS
Alexander Vichnevsky, CAE-adapco, UK
Accurately simulating the individual liquid/liquid/particle dispersions 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 polydispersity nature of the flow. In the present work we use the A-MuSIG method implemented in STAR-CCM+ CFD code of CAE-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 later four are non-linear and being Reynolds averaged produce unclosed terms, which are treated 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.

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 capillitabilities - 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 wall effects 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-size, spherical bodies, and 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 the polydisperse nature of the flow. In the present work we use the A-MuSIG method implemented in STAR-CCM+ CFD code of CAE-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 later four are non-linear and being Reynolds averaged produce unclosed terms, which are treated 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.

FULLY RESOLVED NUMERICAL SIMULATION OF INTER-PHASE HEAT TRANSFER IN GAS-SOLID TURBULENT FLOW
Junjie Xia, Institute for Thermal Power Engineering Zhejiang University, China; Kuni Luo, Zhejiang University, China; Jannen Fan, Zhejiang University, China
Here begins the body of the abstract.

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 spatially and temporally varying 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 at the end of the process. To model the dispersion, a group of particles with different size and velocities are created. The simulation is carried out 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).

FULLY RESOLVED NUMERICAL SIMULATION OF INTER-PHASE HEAT TRANSFER IN GAS-SOLID TURBULENT FLOW
Junjie Xia, Institute for Thermal Power Engineering Zhejiang University, China; Kuni Luo, Zhejiang University, China; Jannen Fan, Zhejiang University, China
Here begins the body of the abstract.

HIGH-ORDER MOMENT-CONSERVING METHOD OF CLASSES IN CFD CODE
Mohammad Ali Juma, Aalto University, Finland; Antonio Buffo, 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 treatment of 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 HMMC coupling with CFD code is practical when employed. 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.


CFD STUDY ON AGGLOMERATION IN A COUNTER-CURRENT SPRAY DRYING TOWER DUE TO PARTICLE-WALL INTERACTIONS
Muzzammil 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 Ghandi, Institute of Particle Science and Engineering, UK; Andrew Bayly, Institute of Particle Science and Engineering, UK; Luis Martinez-Guedes, Polytechnic University of Madrid, Spain
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 atomisation or slurry by 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-plant sprays 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.
DNS ANALYSIS OF WEAKLY-COMPRESSIBLE VAPORIZING TURBULENT TWO-PHASE FLOWS
Benjamin Duret, Université de Rouen, France; Thibaut Menard, Normandie Université, France; Julien Revelillon, 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 flow. The average and scalar dissipation rate statistics in the gas phase. These quantities are of great interest to evaluate the quality 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 the numerical methods dedicated to phase change are designed with a constant gas density in mind, specific numerical methods have been developed to consider the shock-wave induced 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 fragmentations in lithotripsy treatment and cavitation erosion in fluid machines. Using a Lagrangian scheme, we simulate the interaction of a stable 40μm-radius air bubble in water with a 60MPa-lithotripter shock whereby the bubble is attached to an aluminium model (2785kg/m3 density, 26.5GPa shear modulus and 0.3GPa yield strength). Our model involves definition of a reduced parameter state space using dimensional analysis and Design of Experiment (DOE) and generation of a response surface. The method involves a wide range of uncertainty from different sources. An accuracy and repeatability.

A COMPUTATIONAL MODELLING APPROACH TO MINIMIZE PARAMETER SPACE AND GEOMETRY ADAPTABILITY WHEN INVESTIGATING CLEANSROOM 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 parameters that influence cleanliness level in cuboid-type grade C cleansrooms. 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 both dimensional analysis and DOE and generation of a space. The method involves a wide range of uncertainty from different sources. An accuracy and repeatability.

DIRECT NUMERICAL SIMULATION WITH HEAT TRANSFER OF GAS-PARTICLE-MULTIPHASE FLOW CONTAINING SHOCK WAVES BY IMMERSED BOUNDARY METHOD
Yusuke Mizuno, Tokai University, Japan; Shin 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. The numerical boundary 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 82, 023028 (2010)]. The numerical scheme is 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 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 Politek, 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 Gurbaran and Sujith (AIAA 2008-1046), is then a consequence of the NDD wave formation. In the frequency range of the acoustic excitation, the acoustic field is destabilizing, and the spray is oscillating. The oscillation of the spray is 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-dispersion. 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 Corvoro, Università Politecnica delle Marche, Italy; Maurizio Brocchini, Università Politecnica delle Marche, Italy; Massimo Mozzi, CNR – INSEAN, Italy; Francisco Alves Pereira, CNR - INSEAN, Italy; Carlo Lorenzoni, Università Politecnica delle Marche, Italy; Alessandro Mancinelli, Università Politecnica delle Marche, Italy; Cristina Tordi, 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 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 Distance Meter. The interface of the different phases is defined by the solid-liquid and solid-sediment separation, as well as by the sediment deposition around the cylinder. The cylinder's geometry and its position in the flow are obtained from high-resolution images. 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 particle path are extracted at different wave characteristics and results are compared for different wave forcing parameters. *Support from the Stuttgart Research Centre for Simulation Technology is gratefully acknowledged.

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-entrainment 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 Schönhöfer, University of Stuttgart, Germany; Christian Meister, University of Stuttgart, Germany; Georgzor Karch, University of Stuttgart, Germany; Filippo Sadiol, Heidelberg University, Germany; Thomas Ertl, University of Stuttgart, Germany; Bernhard Weigand, Universität Stuttgart, Germany

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 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 Distance Meter. The interface of the different phases is defined by the solid-liquid and solid-sediment separation, as well as by the sediment deposition around the cylinder. The cylinder's geometry and its position in the flow are obtained from high-resolution images. 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 particle path are extracted at different wave characteristics and results are compared for different wave forcing parameters. *Support from COLCIENCIAS (Republic of Colombia) is gratefully acknowledged.

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 excitations. 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 and 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 MINICHANNELS DRY-OUT CONDITION
Niccolo Gianellis, Waseda University, Japan; Kyosho 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 known, the two-phase flow situation can be evaluated. In particular, 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 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 commercial CFD framework of ANSYS CFX using the Euler-Euler model.

TOWARDS A 3D GLOBAL LINEAR STABILITY ANALYSIS FOR BUOYANT FLUID PARTICLES
Johnathan K. Kramer, 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 process like enhanced oil recovery, but also in industrial chemical applications. The present work relaxes the uniform viscosity assumption and aims at studying 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.

MODELING 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 Kær, Aalborg University, Denmark

In PEM water electrolysis, a gaseous oxygen phase inherently develops and 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 known, the two-phase flow situation can be evaluated. In particular, 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 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 commercial CFD framework of ANSYS CFX using the Euler-Euler model.

NUMERICAL STUDY ON EFFECTS OF ORIENTATION AND GRAVITY ON TWO-PHASE FLOW OF MINICHANNEL OF PEM FUEL CELLS
Ashraf Moosa, K.N. Toosi University of Technology, Iran; Shame 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 catalyst layer thickness of 25 μm. A 1 mm width and 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 up to down. 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. Both horizontal and vertical positions, the water coverage ratio of channels was increased by changing the inlet position from up to down. Unsteady flow path 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-LyonUniversité de Lyon, France; Marie-Line Zanotta, LGPC/CNRS-CPE Lyon-Université de Lyon, France; Frederic Bornet, 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

In PEM water electrolysis, a gaseous oxygen phase inherently develops and 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 known, the two-phase flow situation can be evaluated. In particular, 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 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 commercial CFD framework of ANSYS CFX using the Euler-Euler model.
Les of a reactive, multi-phase flow in the injector near-field of an entrained-flow gasifier

**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, and the 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 reaction interfaces 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 a droplet-motion tracking method and a 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 reaction 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.

**Interaction of Fluid Dynamics and Chemical Kinetics in a Three-Phase Dehydrogenation Reaction**
Norbert Hirth, Technische Universität München, Germany; Patrick Le Clercq, German Aerospace Center (DLR), Germany; Manfred Aigner, German Aerospace Center (DLR), 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. If 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-storages 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.

**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 10-3. 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 interaction described with 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.

**Flow Topology and Attractors for the Motion of Finite-Size Particles in a Three-Dimensional Steady Cavity Flow**
Hendrik Kuehnmann, 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 existence 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 intercollision 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.

**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; Win- 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=20 and 30 where “h” is half channel height and “d” is the particle diameter. We perform DNS at bulk Reynolds number ReB = 5600 where the box size and number of grid points in wallnormal, spanwise and streamwise directions are 2h=36h 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 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.

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 (DLMFD) 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 embed our results in our numerical 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
Shaiful Islam, Eindhoven University of Technology, The Netherlands; J. A. M (Jarno) 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 is used in our simulations. We also study the role of 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 is used in our simulations. We also study the role of the momentum boundary layer.

EFFECT OF GAS PROPERTIES ON THE CHARACTERISTICS OF A BUBBLE COLUMN EQUIPPED WITH FINE POROUS SPARGER.
Ariadni Chatzidaki, Aristotle University of Thessaloniki, Greece; Agathoklis Passos, Aristotle University of Thessaloniki, Greece; Sprios Paras, Aristotle University of Thessaloniki, Greece; Alkaterini 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. CO2, helium) covering a wide range of physical properties are used, while the liquid phase is water and deionized 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.

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-focused device, by changing the device geometry and control parameters. Ultrasound, as well as microwaves, 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 frequencies and phase position of the ultrasonic wave pair. This technology may provide new potential opportunities to targeted therapeutics.

* Supported by the National Natural Science Foundation of China (grant number 11372298) is gratefully acknowledged.
Here ends the body of the abstract.

EFFECT OF A CONCENTRATION GAS BUBBLES IN THE NEAR-WALL LIQUID FLOW ON THE DRAG REDUCTION
Aleksandr Dvosev, Kutateladze's Institute of Thermophysics, Russian Federation; Leonid Mal'tzev, 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 sensors. 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 a 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 measurement 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.

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; Leonid Maltzev, Kutateladze's Institute of Thermophysics, Russian Federation

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/cm2 to 100 mA/cm2, corresponding to voltages of approximately 1 V to 3.5 V. Measurements are conducted using a Telecentric Direct Image Method (TDM); 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
Shumeet 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 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 are discussed.

BENCHMARKING CFD AND NEW MECHANISTIC APPROACH FOR ENABLING SCALE-UP OF MULTIPHASE REACTORS VIA SOPHISTICATED MEASUREMENT AND COMPUTING TECHNIQUES
Muhammad Al-Dahhan, Missouri Univ of Science & Technology, USA

In our laboratory advanced measurement techniques, facilities and scale up methodologies have been developed, verified and implement 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.

NUMERICAL INVESTIGATION ON MICROBUBBLE DYNAMIC EVOLUTION AND TURBULENCE MODULATION IN A HORIZONTAL CHANNEL BUBBLE FLOW
Mingjun Pang, Changzhou University, China; Jinji Wei, Xian 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 data of pressure walls so as to understand how microbubbles influence the evolution of the spanwise vortices when they move towards the corresponding wall. The present study 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.

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.

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; Hyeseong 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 - 5.0 μm) 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 (WAF)) were obtained as function of heat flux and subcooling. The results show that the WAF 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 Korean 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. 2012R1A1A2A02024227).
MEASUREMENTS OF 3-PENTANONE LASER INDUCED FLUORESCENCE FOR IMAGING CONCENTRATION OF AN EVAPORATING NE-PENTANE LIQUID FROM A POROUS MEDIA
Quentin Mouret, CETHIL-INSIA de Lyon-UCBL-CNRS, France; Manuel Kuhni, CETHIL-INSIA de Lyon-UCBL-CNRS, France; Patrizio Vena, CETHIL-INSIA de Lyon-UCBL-CNRS, France; Cedric Galizia, CETHIL-INSIA de Lyon-UCBL-CNRS, France; Cedric Galizia, CETHIL-INSIA 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 of 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 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 Fujikoa, Keio University, Japan
Ultrasound bubbles (UFB) whose diameters are smaller than 1 μm hardly escape from water because of negligible buoyancy. Therefore, water suspending ultrasound deposits water, but when UFB is 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 microbubbles 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 collision, is important for 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.

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 cavitating bubbles generates noise and can cause erosion on solid surfaces. 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, 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 one-dimensional shock-tube, a bubble collapsing near a solid wall and a turbulent cavitating flow around a hydrofoil.

NUMERICAL SIMULATION OF IMPINGING JETS ATMOSIZATION
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 impinging 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 impinging 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.

NUMERICAL STUDY ON HEAD-ON COLLISION OF WATER DROPLETS AT HIGH WEBER NUMBERS
Muyuan Liu, Technische Universität Darmstadt, Germany; Dieter Bothe, Technische Universität 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 a thin lamella arising at high Weber collisions, we stabilize the lamella by correcting the surface tension computation and the surface reconstruction in the lamella region. A domain adjunction 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.

A NEW TWO-PHASE FRICTIONAL PRESSURE DROP FOR ADIABATIC AND CONDENSING MINI/MICRO-CHANNEL FLOWS
Mayvan Moshayekhpour, 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 kg/m2 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.

NUMERICAL STUDY ON HEAD-ON COLLISION OF WATER DROPLETS AT HIGH WEBER NUMBERS
Muyuan Liu, Technische Universität Darmstadt, Germany; Dieter Bothe, Technische Universität 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 a thin lamella arising at high Weber collisions, we stabilize the lamella by correcting the surface tension computation and the surface reconstruction in the lamella region. A domain adjunction 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.

* 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

Jarko Solvac, NTNU, Norway; Hans-A. Jakobsen, NTNU, Norway

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 making compromises 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 inert. Agglomeration phenomenon leads to larger particles (aggglomerates), which would be easier collected, improving the collection efficiency for smaller particles. This effect is captured in the frame of the stochastic interparticle 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 the aforementioned order agglomeration. 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, Sao Paulo, Brazil; Hugo Fernando Velasco Pena, University of Sao Paulo, Brazil; Oscar Mauricio Hernandez Rodriguez, University of Sao 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 of the fluids used in the experiments were tap water, deionized water, mineral oil, isopropyl alcohol and hexane. Known volumetric fraction 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.

MEASUREMENT IN THREE-PHASE MIXTURES

*Support from grant PETROBRAS is gratefully acknowledged.

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 Tukovic 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 introduced. Several models of surfactant adsorption are combined for the methods for the two separate mass transfer. 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 effect due to surfactants 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

Jarko Solvac, NTNU, Norway; Hans-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 and might even disappear. For this reason, the conventional numerical simulation of the flow 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 approximation 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.

A CONSERVATIVE LEVEL SET METHOD FOR TWO-PHASE FLOWS

Amin Mahmoudi Moghaddam, Tarbiat Modares University, Iran; Mehdi Shafeeifar, Tarbiat Modares University, Iran; Roobeh 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 long time simulation on a coarse mesh.

A CONSERVATIVE LEVEL SET METHOD FOR TWO-PHASE FLOWS

Amin Mahmoudi Moghaddam, Tarbiat Modares University, Iran; Mehdi Shafeeifar, Tarbiat Modares University, Iran; Roobeh 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 long time simulation on a coarse mesh.

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 Uberlandia., 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.

ICMF 2016 International Conference on Multiphase Flow
Firenze, Italy, May 22 - 27, 2016
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; Qingting 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 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 instable at the angle of 45 degree, but in angle of 50 and 135 degree the flow is very smooth and steady.

Acknowledgement: The work is financed by the National Basic Research Program of China (2012CB717074) 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; Peng 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-modal 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, the Kalman fusion is used to combine data from the conductance sensor and capacitance sensor to estimate water phase fraction ε. In the posterior layer, ε is regarded as a parameter. Cross-Correlation method is employed to analyze data generated by conductance ring to identify flow velocity uc. The uc and Doppler speed ud 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 μm 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

Jicheng 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 detect the instantaneous velocity at different points in a real surfactant solution and different modality was used to control the extent of dispersion. Turbulence parameters including turbulent kinetic energy and energy dissipation rates were evaluated based on velocities to analyze effect of the level of dispersion on turbulence flow filed. 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.
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 combination of 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 is obtained by digital image processed. In detail, the 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.

THE PROPAGATION OF SHOCK WAVES IN GAS-WATER-SATURATION BULK MEDIA
Andrew Zvonkov, Mech USC RAS, Russian Federation; Alfr 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 and glass beads, saturated with water and medium. After vibratory shock tube equipped with a section of bulk media (SBM). Piezoelectric sensors with charge amplifier on the base L-card coupled with a computer were used for the registration of pressure. 3 sensors were used for a long time 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 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.

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 vertical pipes in order to determine 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 drift-flux model. The drift-flux model is applied to both small-dust and the 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 flow appear.

FOUR-WAY COUPLED SIMULATIONS OF THE GAS-SOLID FLOW IN A CYCLONE SEPARATOR
Diego Martins, Federal University of Uberlandia, Brazil; Francisco Jose De Souza, Federal University of Uberlandia, Brazil; Ricardo Salvo, Federal Technologic University of Parana, 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/Lagrangian 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 for 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.

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 Kindsbeekten Kjeldby, StatOil, Norway; Peter W. J. Derks, StatOil, Brazil; Oscar Mauricio Hernandez Rodriguez, University of São Paulo, Brazil
Gas-liquid two-phase flow is 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 these is the drift with water-liquid interface, and the other is the drift-flux model. The drift-flux model is applied to both small-dust and the 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 flow appear.

ANALYSIS OF TWO-PHASE FLOW IN AN CENTRIFUGAL PUMP IMPELLER
Renzo Sabino, Federal University of Technology - Parana, UTFPR, Brazil; Henrique Stel, Federal University of Technology - Parana, UTFPR, Brazil; Dalton Bertoldi, Federal University of Technology - Paraná, UTFPR, Brazil; Rigoberdo 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 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. 

The PROPAGATION OF SHOCK WAVES IN GAS-WATER SATURATION BULK MEDIA
Andrew Zvonkov, Mech USC RAS, Russian Federation; Alfr 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 and glass beads, saturated with water and medium. After vibratory shock tube equipped with a section of bulk media (SBM). Piezoelectric sensors with charge amplifier on the base L-card coupled with a computer were used for the registration of pressure. 3 sensors were used for a long time 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 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.

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 vertical pipes in order to determine 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 drift-flux model. The drift-flux model is applied to both small-dust and the 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 flow appear.

FOUR-WAY COUPLED SIMULATIONS OF THE GAS-SOLID FLOW IN A CYCLONE SEPARATOR
Diego Martins, Federal University of Uberlandia, Brazil; Francisco Jose De Souza, Federal University of Uberlandia, Brazil; Ricardo Salvo, Federal Technologic University of Parana, 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/Lagrangian 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 for 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.
A STUDY OF TWO-PHASE FLOW MALDISTRIBUTION BETWEEN CELLS OF A HIGH PRESSURE PEM WATER ELECTROLYSIS STACK

EXPERIMENTAL MEASUREMENTS OF THE COLLECTOR PERFORMANCE UNDER TWO-PHASE FLOW CONDITION WITH THE MARTIN TECHNIQUE AND CFD SIMULATION

LIQUID MALDISTRIBUTION IN STRUCTURED PACKING IN AN INCLINED COLUMN: CFD SIMULATION AND VALIDATION

NUMERICAL STUDY ON MIXING CHARACTERISTICS IN SPHERICAL FLASK WITH MAGNETIC STIR BAR

LIQUID INFILTRATION CHARACTERISTICS INTO CLOSED END STRUCTURES UNDER THE EXTERNAL PRESSURE

ADSORPTION OF ALCOHOLIC ADDITIVES IN WATER/LIBR AQUEOUS SOLUTION AT LOW PRESSURE

EXPERIMENTAL STUDY OF THE INSITU DROP SIZE DISTRIBUTIONS AT DIFFERENT SPATIAL REGIONS IN STIRRED TANK

SPHERICAL FLASK WITH MAGNETIC STIR BAR

NUMERICAL STUDY ON MIXING CHARACTERISTICS

* Support from the Henry Fok Foundation (142021) is gratefully acknowledged.

* Support from ITN Marie Curie Network “SHINE” is gratefully acknowledged.

* Support from the Innovation Foundation Denmark under the project e-STORE is gratefully acknowledged.
INFLUENCE OF PARTICLE FORCES ON PARTICLES IN FLUID PHASE RESONANCE MIXERS
Silvio Sommet, 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 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 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.

MODELING OF SCATTERING INTENSITY OF MINERAL AEROSOLS WITH A GAUSSIAN BEAM
Hong Tang, China Jiliang University, China; Jianzhong Lin, China Jiliang University, China; Antonio Buffo, Aalto University, Finland; Marco Vanni, Politecnico di Torino, Italy; Silvio Schmalfuss, Martin-Luther-University Halle-Wittenberg, Germany

* The work was supported by the Russian Foundation of Basic Research, Grant No. 15-08-06262.

SIMULATION OF TURBULENT LIQUID-LIQUID DISPERSIONS: VALIDATION OF DIFFERENT BREAKAGE AND COALESCENCE KERNELS AND COMPARISON BETWEEN DIFFERENT MODELLING APPROACHES
Jeremiah 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 (QMM). 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.

DETERMINISTIC AND STATISTICAL MODELLING OF PARTICLE-PARTICLE COLLISIONS IN SUPERSORNE GAS-SOLID FLOWS PAST BODIES
Dmitry Revizinov, Moscow Aviation Institute (National Research University), Russian Federation; Andrey Sposobin, Moscow Aviation Institute (National Research University), Russian Federation; Aleksandr 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 problem of interaction of gas-solid flows with bodies depends on the inertia and concentration of particles. As to the non-spherical particles is proposed to represent the scattered intensity of the 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 particles. The problem is also used for calculating the scattering intensity of the spherical particles in order to reproduce the scattering intensity of the real randomly distributed mineral aerosols illuminated by the Gaussian beam. This work was supported by the Russian Foundation of Basic Research, Grant No. 15-08-06262.

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 gas cushions are created above the liquid inside and outside the pipe. 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 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.
ASSESSMENT OF THE EFFECT OF DYNAMIC PRESSURE CLOSURE ON SIMULATIONS OF VERTICAL ANNULAR GAS-LIQUID FLOW WITH THE 1D TWO FLUID MODEL

Eric Gonzalez, PUC-Rio, Brazil; João Carneiro, Instituto Sintef do Brasil, Brazil; Angela Neickele, 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 introducing 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.

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 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.

NUMERICAL SIMULATION OF ANNULAR FLOW IN HORIZONTAL PIPE USING THE TWO FLUID MODEL

Izabel Souto, PUC-Rio, Brazil; Angela Neickele, 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 e 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

Maltez Hristescu, University of Maribor, Slovenia; Gregor Hagadin, Silkeim d.o.o., Slovenia; Matej Zadrevec, 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, while 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, a well mixed 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; Xiaodong Zhang, China University of Petroleum, China; Xiaogang Yue, China University of Petroleum, China; Zhongwei 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-battery or water-deck 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 (ZR2011EM029) are gratefully acknowledged.

A NOVEL VISCO-PLASTICALLY LUBRICATION METHOD FOR CORE-ANNUAL 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.


THERMOPHORETIC DEPOSITION OF NANOPARTICLES IN A 90° SQUARE BEND TUBE

Zhiqian Yin, China Jiliang University; China; Zhongqing 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.

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 harden and develop gel strength. 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/100ft2 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 primary 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

Xuken He, X’ian Jiaotong University, China; Liang-Liang Fan, X’ian Jiaotong University, China; Jiang Zhe, University of Akron, USA; Liang Zhao, X’ian 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 (λ) and the particle deformability (shell stretching and bending coefficient). The results showed the deformation of the particle increases with the decrease of Re, λ. 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 COALESCEENCE BEHAVIOIRS UNDER VARIOUS SURFACANT CONCENTRATIONS BETWEEN DIFFERENT DRIFT VOLUMES
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

The 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 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 comparing the flow filed 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.

NUMERICAL INVESTIGATION OF THE GROUPING BEHAVIOUR WHEN THE PARTICLES ARE COAXIALLY SETTLING
Deming Xie, China Jiangli University, China; Rongqian Chen, China Jiangli 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 grouping behavior on characteristics of particles, n, the initial inter-particle separation, h0, 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. 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).

NUMERICAL INVESTIGATION OF HEAT TRANSFER AND FLUID FLOW BEHAVIOR AROUND AN OBLATE SPHEROID
Intissar Razzaghallah, Université de Lorraine, LEMTA, UMR 7563, France; Raffik 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 considered to linearly correspond 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

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 Tecnologico Metropolitano-ITM, Colombia; Anne Taniere, Université de Lorraine - LEMTA, France; Mohammed Khaliq, Université de Lorraine - LEMTA, France; Diego Hincapié, Instituto Tecnologico Metropolitano-ITM, Colombia; Sebastian Velez, Instituto Tecnologico Metropolitano-ITM, Colombia

This paper presents 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 Hößler & 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 Sugihara, 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.

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 Ruíz 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 an interface and also, it can handle topological changes of the bubble and the liquid stability. 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.
A PRELIMINARY STUDY ON THE STABILITY OF PARTICLE LADEN JETS THROUGH A FULLY COUPLED CFD-DEM SOLVER

Bernard 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 possibilities 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 XDGM code in order to treat the dispersed phase in a Lagrangian way.

NANOPARTICLE COAGULATION AND DISPERSION IN A TURBULENT PLANAR JET WITH CONSTRAINTS

Chenguang Tu, China Jiliang University, China; Songxi Lu, 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 suspended diameter distribution 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 flow field movement, and decreases along stream direction. 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 which 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.2012010110121) and Special major science and technology of Zhejiang Province (Grant No. 2012C21015-3)

ON THE SEGREGATION OF PARTICLE SIZE CLASSES IN BIDISPERSE GAS-SOLID FLOW: DEPENDENCE ON GAS-SOLID DRAG AND INFLUENCE ON INTERPHASE ENERGY TRANSFER DYNAMICS

Mohammed Mehboudi, 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

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.

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 Sprott Junior, Martin-Luther-University Halle-Wittenberg, Germany; Santiago Lain, Universidad Autónoma de Occidentale, 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 many others. 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 for 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 Eulerian approach and the particle phase is modelled by means of the k-omega-SST model and 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

Philip Mall, 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 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 the results in terms of 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 rise velocities and drag coefficients for moderate Eötvös and Morton numbers and shear rates.