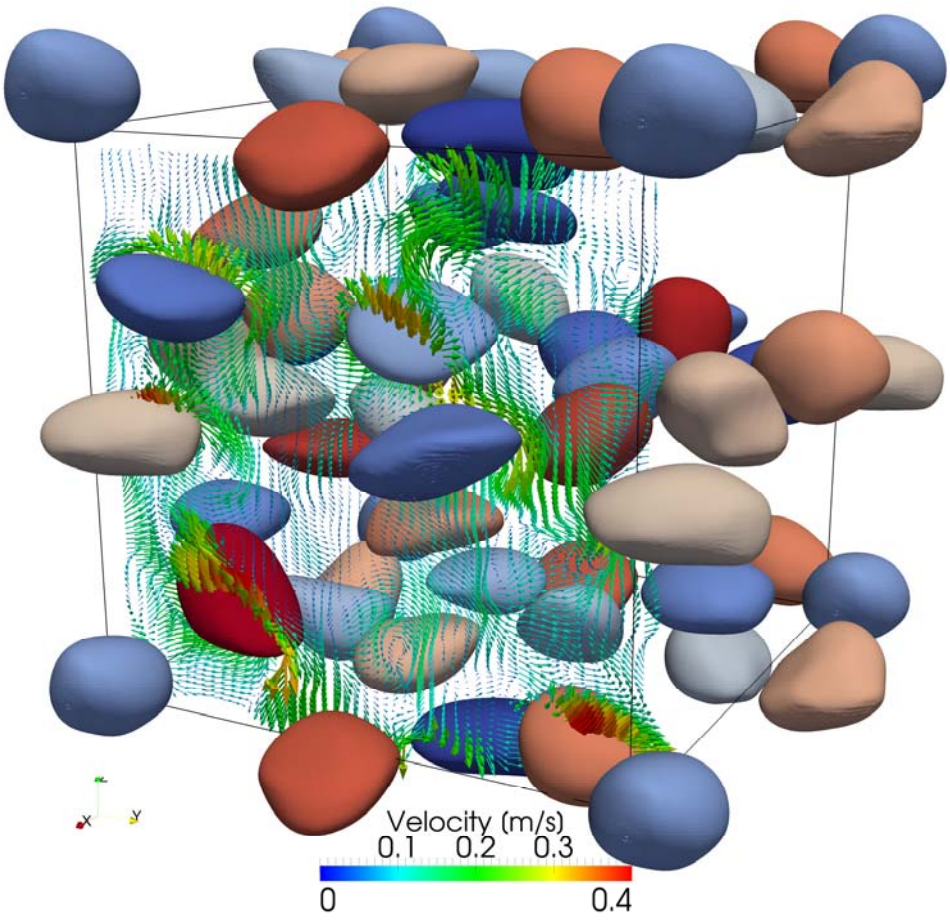
2ND FERMaT-IMPACT MEETING
13-16 OCTOBER 2009

# BOOK OF ABSTRACTS

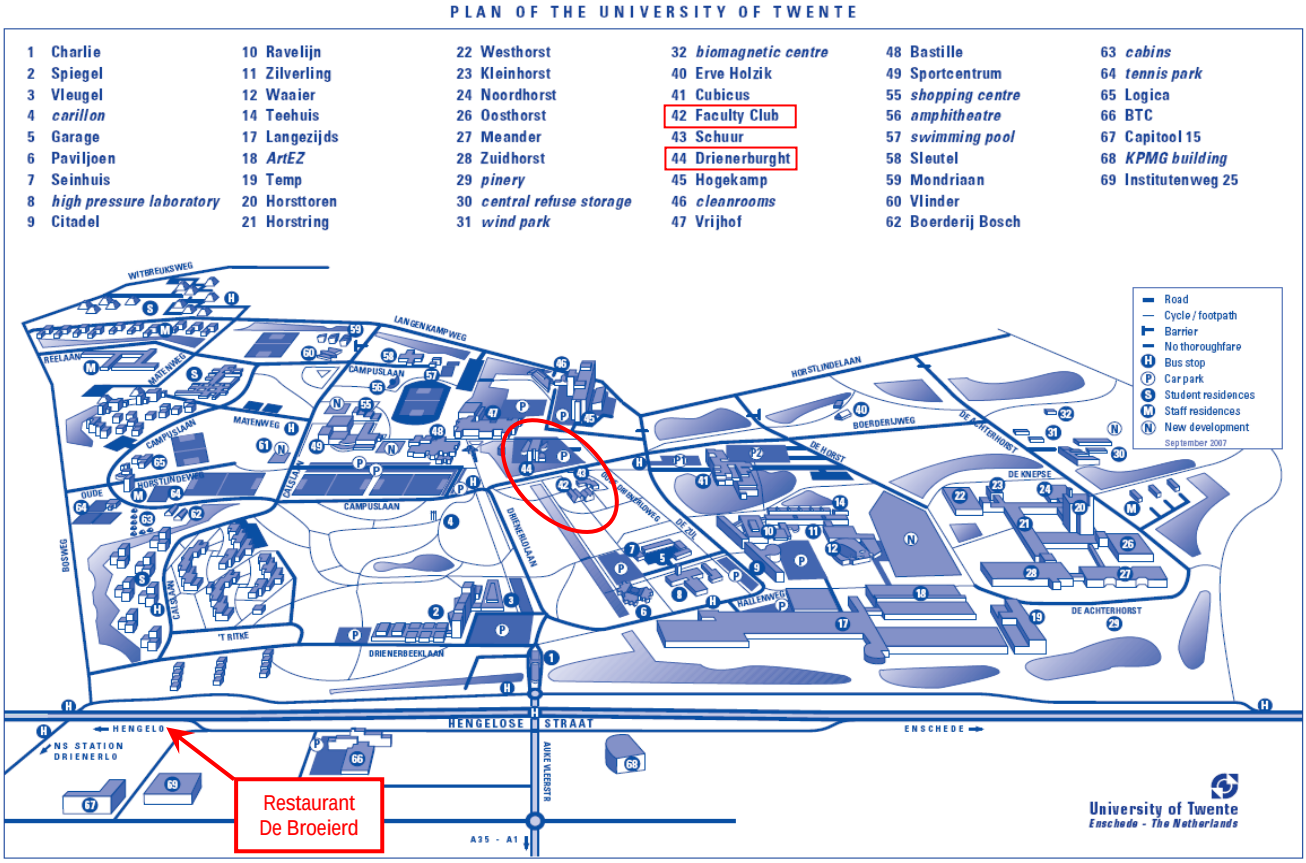
(Eds.: Onno Bokhove, Niels Deen, Rob Hagmeijer & Chao Sun)




UNIVERSITY OF TWENTE.
IMPACT
INSTITUTE OF MECHANICS,
PROCESSES AND CONTROL TWENTE

# Program - Tuesday October 13

Welcome at Enschede railway station and transfer to Conference hotel Drienerburght by bus.




## From Enschede railway station to the University

The University of Twente can be reached by bus from the railway station in Enschede; the services usually run every half hour and takes approximately 15 minutes. From Enschede railway station take line number 1 in the direction of 'Universiteit Twente'. The last bus departs at 23:33 hrs.

## Important phone numbers

| | |
|---|---|
| Conference hotel Drienerburght | +31 (0)53 433 1366 |
| The Faculty Club | +31 (0)53 489 2399 |
| Restaurant de Broeierd | +31 (0)53 850 6500 |
| Restaurant Tien | +31 (0)53 431 3144 |
| Chao Sun | +31 (0)53 489 5604 |
| Secretariat IMPACT | +31 (0)53 489 2489 |

## **Program** - Wednesday October 14

8:30 Welcome

9:00-9:40 Opening session (Hans Kuipers – Olivier Masbernat)

9:40-9:50 Practical information (Onno Bokhove - Rob Hagmeijer)

9:50-10:50 Session 1

Modelling of sub sea gas release (30 minutes)
Paal Skjetne (SINTEF)

Relative motion of coalescence of a pair of two-dimensional bubbles rising in tandem
Sander Huisman, Patricia Ern, Veronique Roig

Supersonic Air Flow due to Solid-Liquid Impact
Stephan Gekle, Ivo Peters, Jose Manuel Gordillo, Devaraj van der Meer and Detlef Lohse

10:50 – 11:10 Break

11:10 – 12:25 Session 2

Detailed modeling of gas-liquid-solid flows
Martin van Sint Annaland, Niels G. Deen, J.A.M. Kuipers

Direct numerical simulations and particle tracking to investigate the effect of mixing on the assimilation by microorganisms
Marco Martins Afonso, J. Morchain, P. Fede

An experimental study of the effect of collision properties on spout fluidized bed dynamics
Maureen S. van Buijtenen, Matthias Börner, Niels G. Deen, Stefan Heinrich, Sergiy Antonyuk and J.A.M. Kuipers

Some experiments on particle capture in flotation
Zhujuna Huang, Pascal Guiraud, Dominique Legendre

Discontinuous Galerkin Finite Element Methods for Fluid Flow
J.J.W. van der Vegt, O. Bokhove, V.R. Ambati, S. Rhebergen, W.E.H. Sollie

12:30 – 14:00 Lunch at the Faculty Club, University of Twente

14:00 – 15:00 Session 3

Space-Time Discontinuous Galerkin Finite Element Method for Two-Fluid Flows
W.E.H. Sollie, O. Bokhove , J.J.W. van der Vegt and V.R. Ambati

Dispersion of bubbles in a turbulent Taylor-Couette flow
A. Chouippe, E. Climent, D. Legendre

Eulerian Method for Ice Accretion on Multiple-Element Airfoil Sections
Jacco M. Hospers and Harry W.M. Hoeijmakers

Modelling and simulation of drop breakup downstream of a sudden section reduction
R. Maniero, O. Masbernat E. Climent, F. Risso

15:00 – 15:20 Break

15:20 – 16:00 Panel discussion

16:00 – 17:00 Lab. visit (PoF of Detlef Lohse, FCRE of Hans Kuipers)

19:00 Dinner at restaurant Tien, Volkspark Enschede

## Program - Thursday October 15

9:00 – 10:15 Session 4

Capillary breakup of stretched liquid jets (30 minutes)
J. M. Gordillo (University of Sevilla)

Formation of microdroplets from liquid jets
Wim van Hoeve, Arjan van der Bos, Jacco Snoeijer, Michel Versluis, and Detlef Lohse

Observation of pore fouling by particles and cells in a microfluidic device
A. Marty, C. Agbangla, P. Bacchin, M. Meireles, P. Duru, D. Bourrier, M. Dilhan

Scaling the drop size in coflow experiments
Elena de Castro and J. M. Gordillo (University of Sevilla)

10:15 – 10:45 Break

10:45 – 12:00 Session 5

Nanofluidic systems: characterization of capillary filling rates
P. Joseph, V.-Nguyen Phan, L. Djeghlaf, A. Allouch, J.-C. Millet, D. Bourrier, P. Dubreuil, D. Belharet, A.-M. Gué, P. Abgrall, N.-T. Nguyen

Microscopic mechanism of electrowetting driven transitions on superhydrophobic surfaces
G. Manukyan, J. M. Oh, D. van den Ende, R. G. H. Lammertink, M. Wessling, and F. Mugele

Numerical modelling of the flow of microparticles (channel and pore clogging)
Gbedo-Constant Agbangla, Patrice Bacchin, Éric Climent

Mixing and electrothermal effect in AC electrowetting
Pablo Garcia-Sanchez, Frieder Mugele

Drying of a square capillary tube
F. Chauvet, P. Duru, M. Prat

12:00 – 13:45 Lunch at the Faculty Club, University of Twente

13:45 – 14:45 Session 6

Effect of cluster resolution on drag and reaction kinetic interaction in riser reactors
Schalk Cloete, Shahriar Amini, Stein Tore Johansen (SINTEF)

Biofilms in porous media
Y. Davit, G. Debenest, M. Quintard

Carbon nanofibers (CNFs): A novel structured catalyst support layer for multi-phase microreactors
D. B. Thakur, R. M. Tiggelaar, J.G.E. Gardeniers, L. Lefferts, K. Seshan

Electrical properties of lanthanum and yttrium doped barium titanate ceramics sintered by Spark Plasma technique – P. Dufour, S. Guillemet-Fritsch, Z. Valdez-Nava, C. Tenailleau, J.Y. Chane-Ching

14:45 – 15:00 Break

15:00 – 16:00 Lab visit (PCF of Frieder Mugele, CPM of Leon Lefferts, EFD of Harry Hoeijmakers)

16:00 – 17:00 Guided tour on the university campus by Wim van Hoeve

18:00 Dinner at restaurant De Broeierd, on walking distance from campus

## Program - Friday October 16

8:30 – 9:30 Session 7

Drag and lift forces on a counter-rotating cylinder in rotating shear flow
Chao Sun, Tom Mullin, Leen van Wijngaarden and Detlef Lohse

Path instability of deformable bubbles in a Hele-shaw cell
Frédéric Risso, Anne-Marie Billet, Veronique Roig, M. Roudet

Non-equilibrium condensation in swirling flow
Edwin van der Weide, Ryan Sidin, Rob Hagmeijer

Imaging method for interface dynamics characterization
Nicolas Abi Chebel, Frederic Risso, Olivier Masbernat, Pascal Guiraud

9:30 – 10:00 Break

10:00 – 10:45 Session 8

Energy and materials (EnMAT) sciences at the French FERMAT research Federation based in Toulouse
Christophe Tenailleau

Cu2MSnS4 (M=Co, Zn, Fe) Quantum dots
A. Gillorin, J.Y. Chane-Ching, P. Dufour, A. Baloochi, X. Marie

Nanostructured energetic materials based on Al/CuOx nanowires
M. Petrantoni, C. Rossi, V. Conedera, L. Salvagnac, D. Bourrier, C. Tenailleau, P. Alphonse

10:45 – 11:15 Closure (Hans Kuipers – Olivier Masbernat)

11:15 Handout of lunchbox

# Modelling of sub sea gas release

Paal Skjetne
SINTEF Materials and chemistry, Department of Process Technology, Sem Sælandsvei 2A, 7465 Trondheim, Norway. Paal.Skjetne@sintef.no

Since the advent of oil and gas exploration in the North Sea some 40 years ago several large scale and hundreds of small scale sub sea gas releases have occurred. Historically so called integral models [1-3] have been used to model the fate of such large scale bubble plumes. Such models still represent the state of the art in modelling of such events in connection with HSE risk assessments performed for the oil companies. In a 2005 seminar held by the Norwegian Petroleum Safety Authority (PSA) four consultancy companies delivering HSE risk assessments for offshore explosion hazards where invited to give their state of the art analyses of both a sub sea gas release from a broken gas pipe and the resulting atmospheric dispersion of methane. The sub sea gas release modelling was required to generate the boundary condition for the atmospheric dispersion modelling. The workshop revealed that the quality of and current understanding of sub sea gas release modelling varied strongly between the companies.
SINTEF has a long history in modelling the fate of sub sea gas and oil release [4], with a whole department dedicated to oceanic dispersion, fate and bioaccumulation of such spills. However, until recently this work was based on integral plume models (both Eulerian and Lagrangian). In the wake of the 2005 PSA workshop, PSA, Gassco and Statoil invited SINTEF to help assess current models and to explore the possibilities of using CFD for sub sea gas release modelling.
In this talk we will present background information on sub sea gas release, modelling and validation, the current status on using CFD tools and current challenges.


Figure: Left: Actinia blow out in Vietnam 1993, the height of the boiling ocean surface was estimated to be approximately 30m. Right: The Petromar V drillship sinking in the South China Sea in 1981.

[1] Bettelini, M.S.G and Fanneløp, F.K., Underwater plume from an instantaneously started source, *Applied Ocean Research* **15**, 195-206 (1993).
[2] Brevik I. and Killie R., Phenomenological description of the axisymmetric air-bubble plume, *Int. J. Multiphase Flow* **22**, 535-549 (1996).
[3] Milgram, J.H, Mean flow in round bubble plumes. *J. Fluid Mech.* **133**, 345–376 (1983).
[4] Johansen, Ø., H. Rye and C. Cooper, DeepSpill – Field Study of a Simulated Oil and Gas Blowout in Deep Water, *Spill Science & Technology Bulletin* 7 (2002).

# Relative motion and coalescence of a pair of two-dimensional bubbles rising in tandem

Sander Huisman, Patricia Ern and Veronique Roig
Institut de Mécanique des Fluides de Toulouse (IMFT)

We investigate experimentally the interaction of a pair of bubbles rising in tandem in a Hele-Shaw cell at high Reynolds and Bond numbers. We focus on the case of a cap bubble released in the wake of a larger bubble. The evolution of the bubbles is recorded with a high frequency camera and we measure through image processing their trajectories and shapes. The major steps in the interaction are then identified and analysed: first, the aft-bubble is accelerated by the wake of the fore-bubble; when the aft-bubble enters the attached wake both bubbles are strongly deformed until they coalesce.

# Supersonic air flow due to solid-liquid impact

Stephan Gekle[1], Ivo Peters[1], José Manuel Gordillo[2], Devaraj van der Meer[1] and Detlef Lohse[1]
1 Physics of Fluids, University of Twente, The Netherlands
2 Grupo de Mecánica de Fluidos, Universidad de Sevilla, Spain

As an object impacts a water surface a sizeable cavity is formed below the surface which subsequently collapses due to hydrostatic pressure. The air flow as it is squeezed out of the shrinking cavity is visualized using smoke particles, see Fig. 1. Numerical simulations combining a boundary-integral method with a fully compressible Euler solver illustrate the intricate structure of the gas dynamics. Two effects of the air flow close to pinch-off are identified consistently in experiment and numerics: (i) the cavity wall is not smoothly curved but exhibits a “kink” and (ii) the cavity neck is pushed upwards. Despite an impact velocity of merely 1 m/s the air reaches supersonic speeds (Fig. 1).

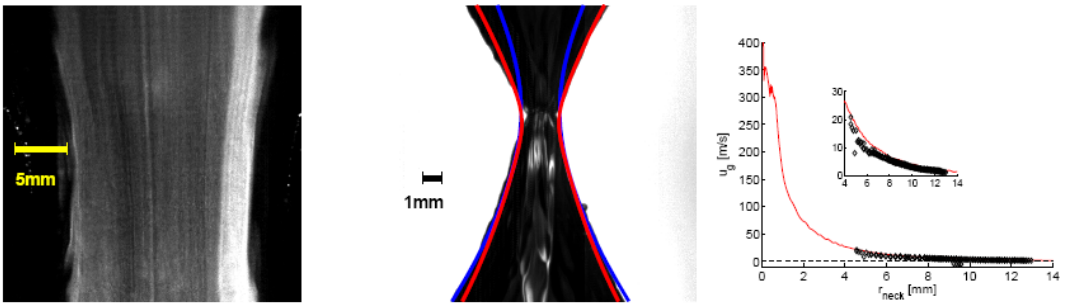



**Figure 1.** Left: The cavity is filled with smoke particles that allow measuring the air velocity. Dark portions represent water. Center: The experimental neck shape shows a pronounced increase in curvature at the neck, the smoothly rounded curved obtained by the purely one-phase simulation (blue line) is not able to capture the exact neck shape. Including air as a second fluid into the simulations (red line) improves the matching with experiment significantly. Right: The speed of the gas flowing through the neck obtained from the fully compressible numerical scheme (red line) agrees very well with the experimentally measured velocity (black diamonds).

# Detailed modeling of gas-liquid-solid flows

Martin van Sint Annaland, Niels G. Deen, J.A.M. Kuipers
Institute for Mechanics Processes and Control Twente, Faculty of Science and Technology, University of Twente, PO Box 217, Enschede, 7500 AE, The Netherlands

A simulation model is presented for the Direct Numerical Simulation (DNS) of complex multi-fluid flows in which simultaneously (moving) deformable (drops or bubbles) and non-deformable (moving) elements (particles) are present, possibly with the additional presence of free surfaces. Our model combines a Front Tracking model developed by van Sint Annaland et al. (2005) and an Immersed Boundary (IB) model developed by van der Hoef et al. (2008). The Front Tracking (FT) part circumvents the explicit computation of the interface curvature. The Immersed Boundary part incorporates both particle-fluid and particle-particle interaction via a Direct Forcing Method and a hard sphere Discrete Particle (DP) approach. In our model a fixed (Eulerian) grid is utilized to solve the Navier-Stokes equations for the entire computational domain. The no-slip condition at the surface of the moving particles is enforced via a momentum source term that only acts in the vicinity of the particle surface. For the enforcement of the no-slip condition Lagrangian force points are used, which are distributed evenly over the surface of the particle. Dissipative particle-particle and/or particle-wall collisions are accounted via a hard sphere DP approach using a three-parameter particle-particle interaction model accounting for normal and tangential restitution and tangential friction. The capabilities of the hybrid FT-IB model are demonstrated with a number of examples, including the effect of particle properties on the bubble induced drift (see Figure 1).

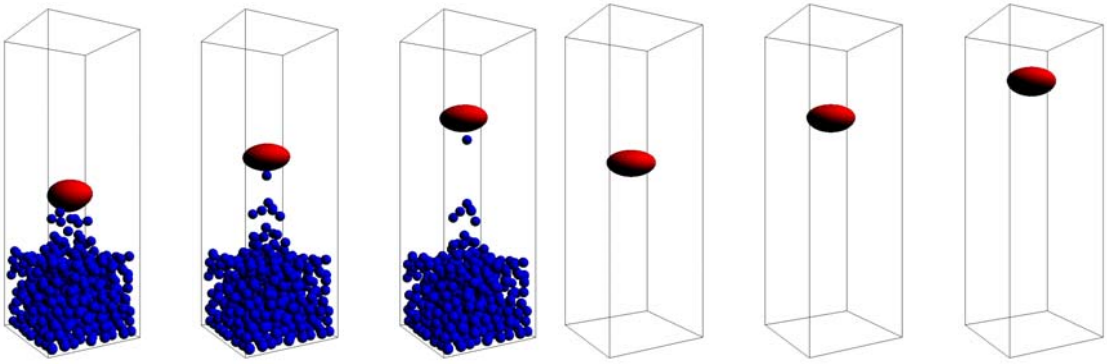


**Figure 1:** Snapshots at different times of the rise of a bubble with a diameter of 0.02 m through a suspension of 512 spherical particles (left) and liquid without particles (right).

van der Hoef, M.A., van Sint Annaland, M., Deen, N.G. and Kuipers, J.A.M., Numerical Simulation of Dense Gas-Solid Fluidized Beds: A Multiscale modeling Strategy, *Ann. Rev. Fluid Mech.* **40**, 47-70, 2008.
van Sint Annaland, M., Deen, N.G. and Kuipers, J.A.M., Numerical simulation of gas-liquid-solid flows using a combined front tracking and discrete particle method, *Chem. Eng. Sci.* **60**(1), 6188-6198, 2005.

# Direct numerical simulations and particle tracking to investigate the effect of mixing on the assimilation by microorganisms

Marco Martins Afonso[1,2], Jérôme Morchain[3] and Pascal Fede[1,2]
1. Université de Toulouse, INP/UPS/IMFT, allée du Professeur Camille Soula, 31400 Toulouse, France
2. CNRS, IMFT, allée du Professeur Camille Soula, 31400 Toulouse, France
3. Université de Toulouse, Laboratoire d'Ingénierie des Systèmes Biologiques et des Procédés, INSA/CNRS/INRA, 135 avenue de Rangueil, 31077 Toulouse cedex, France

When cultivated in large scale bioreactors, microorganisms are exposed to fluctuations of their local environment in terms of concentration and turbulence properties, especially in continuous and fed-batch cultures.[1] The typical size of microorganisms is usually small compared to the Kolmogorov scale, and thus the last step of mass transfer from the liquid to the biological phase can be considered as governed by diffusion. However, the fluctuations of concentration far from the cell are determined by the turbulent mixing. The substrate uptake rate is given by a stationary law based on the Monod formulation,[2] which links it to the average concentration in the medium. The coupling of hydrodynamics and biological reactions, using such a stationary law for the mass transfer, leads to unsatisfying results.[3] It can be thought that the local uptake rate does not obey the stationary law but is also controlled by the local mixing intensity. With the aim of understanding this point, we perform direct numerical simulations to describe the mixing of sugar (modelled as a passive scalar) in homogeneous isotropic turbulence with an imposed mean concentration gradient, and particle tracking of tracer microorganisms (at present without considering their feedback on the flow). We analyse the statistics of the substrate flux toward the microorganism, computed either through the classical Monod saturation law or via a piecewise (linear/constant) approximation. The latter is being investigated in parallel by means of a numerical code, which solves the diffusion equation in spherical coordinates, imposing a variable signal far from the cell and the Robin (i.e., mixed Dirichlet/Neumann) boundary condition at the cell interface.

[1] F. Bylund, E. Collet, S.-O. Enfors, G. Larsson, *Bioproc. Biosys. Eng.* **18**, 171 (1998).
[2] J. Monod, *Ann. Rev. Microbiology* **3**, 371 (1949).
[3] S. Schmalzriedt, M. Jenne, K. Mauch, M. Reuss, *Adv. Biochem. Eng./Biotechnol.* **80**, Springer (2003).

# An experimental study of the effect of collision properties on spout fluidized bed dynamics

Maureen S. van Buijtenen[1], Matthias Börner[2], Niels G. Deen[1], Stefan Heinrich[2,3], Sergiy Antonyuk[2,3] and J.A.M. Kuipers[1]
1 Institute for Mechanics Processes and Control Twente, Faculty of Science and Technology, University of Twente, PO Box 217, Enschede, 7500 AE, The Netherlands
2 Faculty of Process and Systems Engineering, Otto-von-Guericke-University Magdeburg, PO Box 4120, 39106 Magdeburg, Germany
3 Institute of Solids Process Engineering and Particle Technology, Hamburg University of Technology, 21071 Hamburg, Germany

Spout fluidized beds are frequently used for the production of granules or particles through granulation, which are widely applied for example in the production of detergents, pharmaceuticals, food and fertilizers. Spout fluidized beds have a number of advantageous properties, such as high mobility of the particles preventing undesired agglomeration and enabling excellent heat transfer control. During the granulation process, the particles contain different loadings of moisture which results in varying collision properties in time and location across the bed. However, little is known about the effect of the collision properties on the bed dynamics, and thus on the granulation process. This is mainly due to the fact that it is not visually accessible. In previous conducted numerical studies the effect on the collision properties, i.e. the restitution coefficient, are investigated on the bed dynamics. In Figure 1 (see next page) it is shown that if the restitution coefficient decreases, more bubbles are present causing more instability in the overall dynamics of the bed in more or less extent dependent on the flow regime. These conclusions show the great importance of the influence of the restitution coefficient on the dynamics of the bed. Therefore, an experimental study is of interest to examine different particle systems with different collision properties in a spout fluidized bed. In this work a non-intrusive measurement technique is used, viz. particle image velocimetry (PIV) to measure the particle flow field in a pseudo two-dimensional spout fluidized bed. Additionally, digital images are analyzed using a proposed digital image analysis algorithm (DIA) to evaluate the particle volume fraction. The experiments are conducted with different particle systems, such as glass beads, γ-alumina oxide and zeolite 4A particles. Each particle system is applied in three different flow regimes: intermediate/spout-fluidization regime (case B1), spouting-with-aeration regime (case B2) and jet-in-fluidized-bed regime (case B3). The experimental results of the different particle systems will be compared, taking the differences in density and particle size into account.

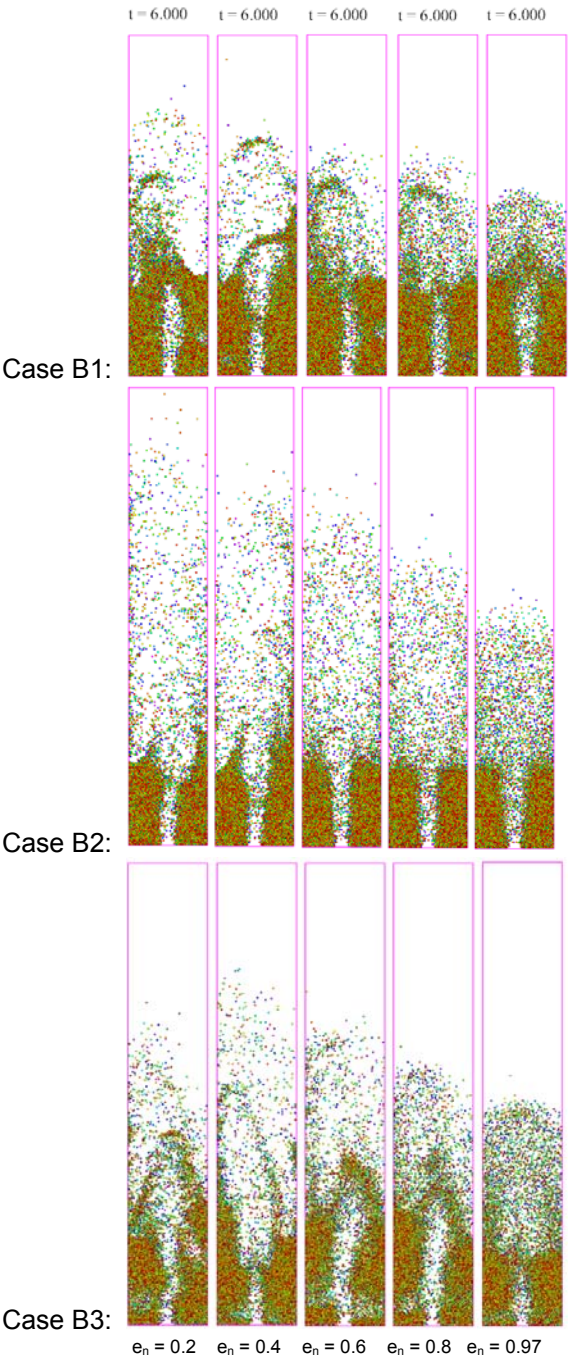



**Figure 1.** Snapshots of the simulated instantaneous particle positions for different restitution coefficients for case B1 (intermediate/spout-fluidization regime), B2 (spouting-with-aeration regime) and B3 (jet-in-fluidized-bed regime) at simulation time t = 6.0 s (after M.S. van Buijtenen et al., ICMF 2007, Leipzig, Germany).

# Some Experiments on Particle Capture in Flotation

Zhujuna Huang[1,2], Pascal Guiraud[1], Dominique Legendre[2]
1 Laboratoire d'Ingénierie des Systèmes Biologiques et des Procédés (LISBP)
2 Institut de Mécanique des Fluides de Toulouse (IMFT)

In a single bubble flotation column, the interactions between the small air bubble ($d_b$ = 0.35-1.30 mm) and spherical glass particles ($d_p$ = 15-56 µm) have been recorded by a CCD high speed video camera. The particles are shown to slip on the interface, then adhere to air bubbles individually or as aggregates and cover the free surface at the rear part of bubble.
These captured particles increase not only the bubble‘s effective density but also its surface contamination level (retardation of the interface). As a result, bubble‘s rising velocity decreases along the column‘s height and also its capture efficiency. A detailed analysis of the relationship between bubble coverage angle, rising velocity and capture efficiency at different vertical position in the flotation column is performed. This new approach gives an order of magnitude for the capture efficiency between 0.001 and 0.5 as well as that given by the counting methods.

# Discontinuous Galerkin finite element methods for fluid flow

J.J.W. van der Vegt, O. Bokhove, V.R. Ambati, S. Rhebergen, W.E.H. Sollie
Numerical Analysis and Computational Mechanics Group
Department of Applied Mathematics, University of Twente, Netherlands

During the past decade discontinuous Galerkin finite element methods have seen a very rapid development, both in their theoretical development and in various application domains. DG methods have been demonstrated to be very useful in many fluid mechanics problems. In particular, for flows which contain local features that need to be captured to obtain sufficient accuracy in numerical simulations. Examples are shocks, boundary layers, density interfaces and vortical structures.
The use of DG methods has been motivated by a number of important features of these numerical discretizations. In particular, DG methods result in a very local discretization suitable for hp-adaptation, which consists of local mesh refinement and adjustment of the polynomial order, and parallel computing. Also, the accuracy and stability of DG discretizations has been analyzed in great detail which provides important information for practical simulations.
In this presentation an overview of the research on discontinuous Galerkin methods in the Numerical Analysis and Computational Mechanics group will be given. Special attention will be given to space-time discretizations which are suitable for problems with time-dependent boundaries, such as in fluid-structure interaction, and efficient multigrid pseudo-time techniques for the solution of the resulting algebraic equations. Also, the development of the hpGEM finite element toolkit for the efficient implementation of DG algorithms will be discussed. Applications to compressible gas dynamics, two-phase flows, and dispersed multiphase flows will be highlighted.

1. S. Rhebergen, O. Bokhove, and J.J.W. van der Vegt, Discontinuous Galerkin finite element methods for shallow two-phase flows. *CMAME* **195** (2009).
2. C.M. Klaij, M.H. van Raalte, H. van der Ven, and J.J.W. van der Vegt, h-Multigrid for space-time discontinuous Galerkin discretizations of the compressible Navier-Stokes equations , *J. Comp. Phys* **227** (2007).
3. L. Pesch, A. Bell, W.E.H. Sollie, V.R. Ambati, O. Bokhove and J.J.W. van der Vegt, hpGEM- A software framework for Discontinuous Galerkin finite element methods, *ACM Transactions on Software* **33** (2007).
4. www.math.utwente.nl/~vegtjjw/ & www.math.utwente.nl/~bokhoveo/

# Space-time discontinuous Galerkin finite element method for two-fluid flows

W.E.H. Sollie, O. Bokhove, J.J.W. van der Vegt and V.R. Ambati
Numerical Analysis and Computational Mechanics Group,
Department of Applied Mathematics, University of Twente.
P.O.Box 217, 7500AE Enschede, The Netherlands.
w.e.h.sollie@math.utwente.nl

Moving interfaces are an important aspect in many fluid flow problems including free surface, multifluid and multiphase flows. Often the interface shape and dynamics are coupled to the fluid flow fields. Topological changes may occur due to breakup and coalescence processes.
A novel space-time discontinuous Galerkin finite element method (STDGFEM) is presented for solving two fluid flows which combines aspects of both front capturing (Level Set Method (LSM), [1]) and front tracking methods (front tracking, cut-cell, [2]). The interface is represented explicitly in both space and time to achieve high accuracy in the interface position and shape including the flow field approximation around it. The method uses a space-time Cartesian background mesh on which the interface is captured based on a cut-cell mesh refinement algorithm. The problem of small elements caused by the cut-cell mesh refinement is solved by applying a merging procedure. The interface evolution is computed using the LSM, which gives an implicit description of the interface in space-time. With the LSM, it is easy to handle mesh topological changes and to calculate the interface curvature. In the current numerical scheme, we use the global fluid flow velocity to advect the level set which was found to be sufficiently accurate. The STDGFEM [3] is well suited for two fluid flow calculations since it allows for discontinuities in the flow field, which tend to appear at the interface. The interface conditions are implemented in the STDGFEM by defining a suitable numerical interface flux. This keeps the numerical scheme local and thus suitable for hp-refinement and parallel computing. To stabilize the flow solution, we employ the HWENO slope limiter [4] in combination with a discontinuity detector [5]. The proposed STDGFEM method provides an accurate and versatile scheme for dealing with interfaces in two fluid flow problems which can alleviate some of the problems encountered with front tracking and front capturing methods.
The numerical scheme has been implemented in the hpGEM package [6], a software framework for DGFEM methods, which is currently under development in our Numerical Analysis and Computational Mechanics Group. The method has been applied to amongst others to a helium cylinder - shock wave interaction problem [7,8] for which results are shown in Figure 1 (see next page).

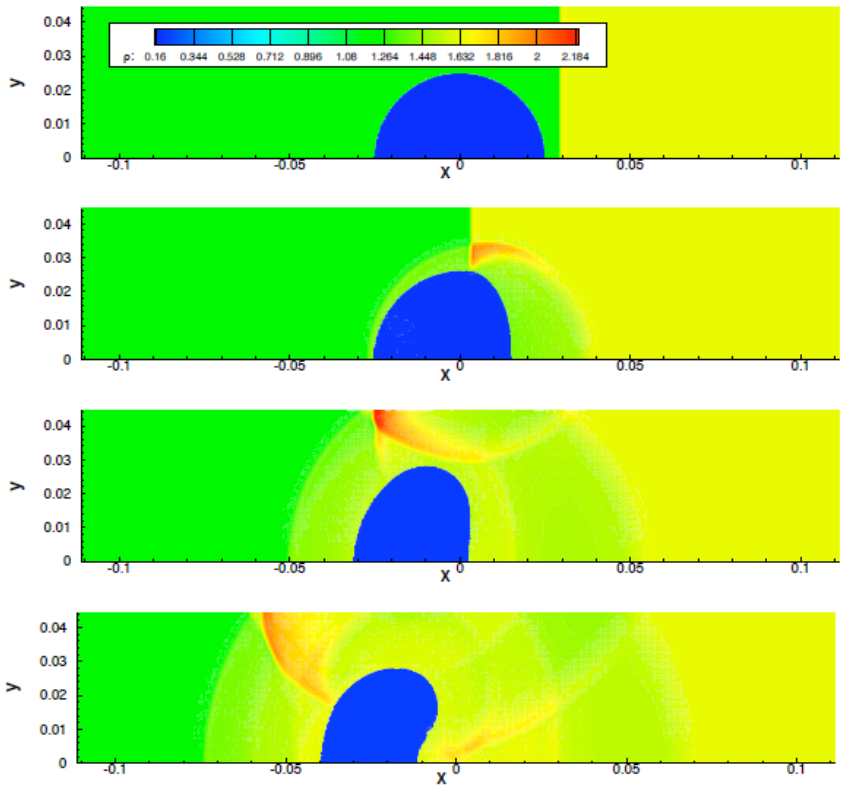



**Figure 1.** Density contours for helium bubble shock wave interaction test, using a mesh of 320 × 64 elements at times t = $0.625^{-4}$, $1.25^{-4}$, $1.875^{-4}$ and $2.5^{-4}$. Near the interface the mesh is composed of locally refined elements.

[1] M. Sussman, P. Smereka and S. Osher "A level set approach for computing solutions to incompressible two-phase flow" *J. Comput. Phys.* **114**, 146-159 (1994).
[2] G. Yang et. al "A Cartesian cut element method for compressible flows. Part A: Static body problems" *Aeronaut. J.* Vol. **2**, 47-65 (1997).
[3] J.J.W. van der Vegt, H. van der Ven "Space-time discontinuous Galerkin finite element method with dynamic mesh motion for Inviscid Compressible Flows" *J. Comput. Phys.* **182**, 546-585 (2002).
[4] H. Luo, J.D. Baum, R. Lohner "A Hermite WENO-based limiter for discontinuous Galerkin method on unstructured grids" *J. Comp. Phys.* **225**, 686-713 (2007).
[5] L. Krivodanova, J. Xin, J.-F. Remacle, N. Chevaugeon, J.E. Flaherty "Shock detection and limiting with discontinuous Galerkin methods for hyperbolic conservation laws." *Appl. Num. Math.* **48**, 323-–338 (2004).
[6] L. Pesch, A. Bell, W.E.H. Sollie, V.R. Ambati, O. Bokhove, J.J.W. van der Vegt "hpGEM- A software framework for Discontinuous Galerkin finite element methods" *ACM Trans. Math. Softw.* **33**, Article 23 (2007).
[7] J.-F. Haas, B. Sturtevant "Interaction of weak shock waves with cylindrical and spherical gas inhomogeneities" *J. Fluid Mech.* **181**, 41-76 (1987).
[8] J.Wackers, B. Koren "A fully conservative model for compressible two-fluid flow" *Int. J. Numer. Meth. Fluids* **47**, 1337-1343 (2005).

# Dispersion of bubbles in a turbulent Taylor-Couette flow

A.Chouippe, E.Climent, D.Legendre
Institut de Mécanique des Fluides de Toulouse (IMFT)

We are aiming to study the effect of bubbles in a Taylor-Couette flow (flow within the gap between a rotating inner cylinder and a concentric fixed outer cylinder) under different turbulent flow regimes. The flow motion is obtained through direct numerical simulations of the Navier-Stokes equations coupled with an Euler-Lagrange approach for the bubble trajectories. We assume that bubbles are spherical and non-deformable. The force balance acting on each bubble takes into account buoyancy, added-mass, drag and lift effects.
The present communication is the first step of a more general study on the effect of bubbles on drag reduction in a wall bounded turbulent flow. Firstly, we are concerned with the dispersion of bubbles induced by the continuous phase flow (one-way coupling). The purpose is to highlight the mechanisms acting on bubble migration and possibly local accumulation. The description of the carrying fluid flow in the turbulent regime is achieved by the superposition of three contributions: the azimuthal flow driven by the rotation of the inner cylinder, large scale vortices (namely the Taylor Vortices) and small scale turbulent fluctuations. The level of velocity fluctuations can be split in the distinct contributions of the Taylor Vortex Flow and the turbulence. We are investigating their respective role on bubble dispersion and the mechanisms leading to preferential accumulation. Those phenomena are compared to the behavior of the dispersed phase already observed in the Taylor vortex flow regime at lower Reynolds numbers [1].
Further simulations will be focusing on two-way coupling interactions between the bubbles and the turbulent fluid flow leading to drag reduction. This feature has been observed experimentally but a comprehensive description of the basic mechanisms is still open.

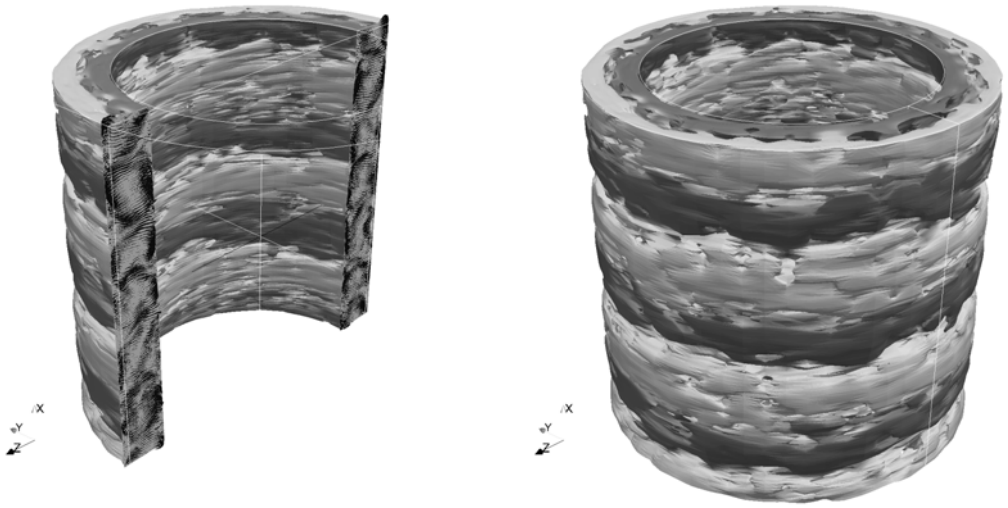


**Figure 1.** Flow visualization of the turbulent Taylor vortices at Re=6000 (the colours highlight the presence of large scale counter-rotating vortical structures).

References
[1] E.Climent M.Simonnet, J.Magnaudet. Preferential accumulation of bubbles in Couette-Taylor flow patterns. *Phys. Fluids* **19**, 083301 (2007).

# Eulerian method for ice accretion on multiple-element airfoil sections

Jacco M. Hospers and Harry W.M. Hoeijmakers
Faculty of Engineering Technology, group of Engineering Fluid Dynamics
University of Twente, PO Box 217, 7500 AE Enschede, the Netherlands

A computational method is presented that computes ice accretion on multiple-element airfoils in specified icing conditions given the flow solution. The numerical simulation method (Droplerian) uses an Eulerian method to determine the droplet trajectories and distribution of the Liquid Water Content (LWC). To solve the equations for the droplet trajectories and liquid water content distribution, Droplerian uses a Finite Volume Method for unstructured grids. Through the droplet velocities and Liquid Water Content at the surface of the airfoil configuration the droplet catching efficiency is calculated. The droplet catching efficiency and droplet velocities at the airfoil surface are input for the icing model, which is based on Messinger's model for ice accretion. The method includes a multi-disperse droplet distribution with an arbitrary number of droplet bins and a droplet splashing model. For a single-element airfoil a good agreement is found with experimental catching efficiencies and with the ice shapes predicted by other computational methods. Agreement with the experimental ice shapes is fair. For increasing droplet diameter the agreement with experimental results deteriorates. The application of the method to a three-element airfoil is described. The comparison of the catching efficiency predicted by both the Droplerian method and a Lagrangian method (2DFOIL-ICE) is good. The agreement of predicted ice accretions with available experimental data is reasonable.

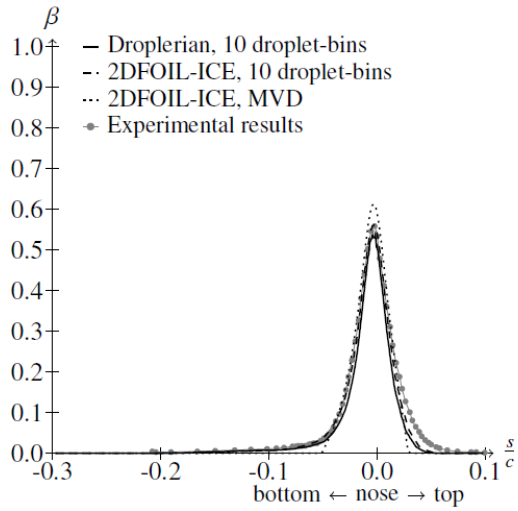



**Figure 1.** Calculated catching efficiencies, 20 μm MVD, NACA-23012, 2.5° AoA, $U_\infty$ = 78.23 m/s

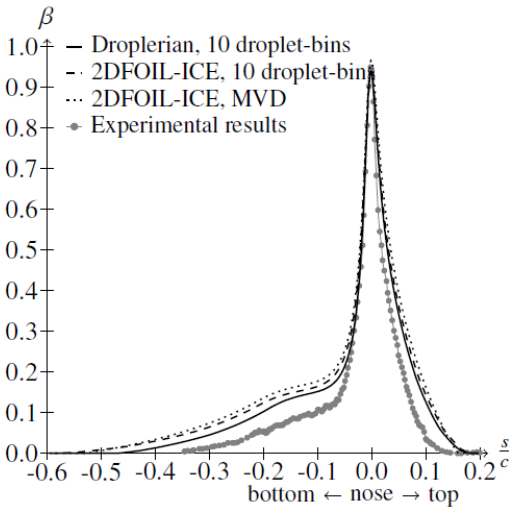



**Figure 2.** Calculated catching efficiencies, 236 μm MVD, NACA-23012, 2.5° AoA, $U_\infty$ = 78.23 m/s

# Modelling and simulation of drop breakup downstream of a sudden section reduction

R. Maniero[1], O. Masbernat[1], E. Climent[2], F. Risso[2]
1 Laboratoire de Génie Chimique, Toulouse
2 Institut de Mécanique des Fluides, Toulouse

This work aims at modelling numerically drop breakup in a turbulent inhomogeneous flow induced by a sudden reduction of section in a pipe. The turbulent flow field has been obtained through Direct Numerical Simulation (DNS) of the Navier-Stokes equations; the resulting velocity field has been coupled to the Lagrangian tracking of individual drops and to a deformation model. The drop deformation dynamics is modelled by a harmonic oscillator forced by the local and instantaneous fluctuations of the turbulent flow (local Weber number). The temporal evolution of the oscillator response is used to determine the breakup locations based on a critical threshold of the deformation.
The numerical simulations have been validated through comparisons with flow statistics (fig. 1) obtained from PIV. Detailed information on the external stresses experienced by the drop can be achieved by DNS. The contributions of the velocity fluctuations, pressure gradients and viscous shear stress to the instantaneous Weber number (fig.2) can be compared. Locations of breakup events have been compared to the experimental results. Different liquid/liquid systems have been investigated by changing the drop size and surface tension.

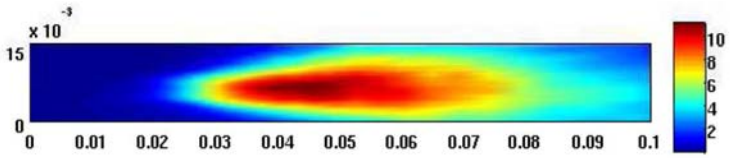



**Figure 1.** Spatial distribution of the average turbulent Weber number downstream of the orifice.

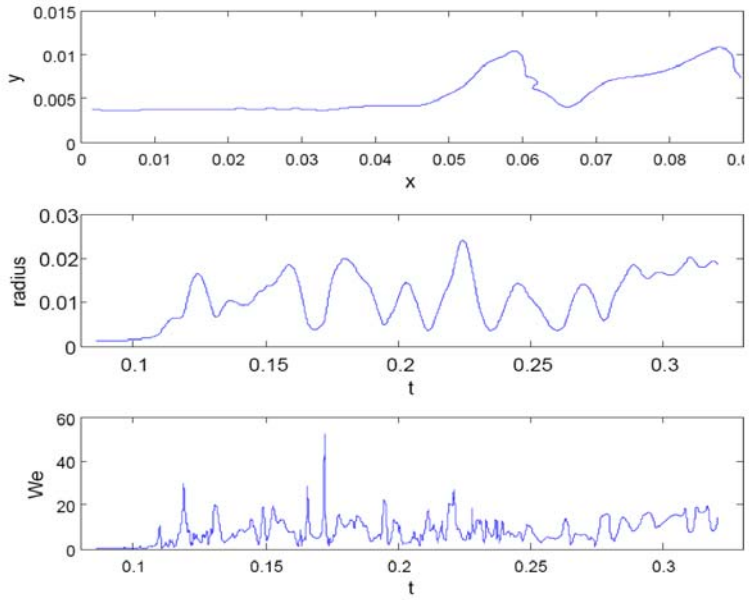



**Figure 2.** Trajectory of a drop and temporal evolution of its deformation and of the instantaneous Weber number.

# Capillary breakup of stretched liquid jets.

José Manuel Gordillo
Área de Mecánica de Fluidos, Departamento de Ingenería Energética y Mecánica de Fluidos, Universidad de Sevilla, Avda. de los Descubrimientos s/n 41092, Sevilla, Spain.

Contrarily to the capillary breakup of liquid cylinders in vacuum, we have found that the breakup of stretched liquid jets at high values of both the Weber and Reynolds numbers is not necessarily induced by the growth of capillary disturbances convected from an upstream location. These types of stretched jets correspond, for instance, to those ejected after the impact of a solid or a drop against a free surface. We describe here that the origin of capillary disturbances leading to jet breakup is a self-induced mechanism that depends on the local flow field existing at the jet tip. The process of drop separation from the rest of the jet can be described as a function of a single dimensionless parameter, $We_S = \rho R_o^3 S_o^2/\sigma$ which expresses the ratio of the capillary time to the inverse of the local strain rate. Here $\rho$ and $\sigma$ indicate, respectively, the liquid density and the interfacial tension coefficient and $R_o$ and $S_o$ are the radius of the jet and the axial strain rate at the origin. We have found a physical argument that gives the size of the drops generated from non accelerated stretching jets as $d_{drop}/R_o \propto We_S^{-1/7}$, whereas the time to breakup is given as $TS_o/We_S^{2/7}$. These scalings are in excellent agreement with numerical simulations and experiments. The conclusions extracted from the study of non accelerated jets are extended to that of stretched jets accelerated by gravity. The expression for the breakup length, which is theoretically found as a function of both the Bond and Froude numbers, will be compared with experiments.

# Formation of microdroplets from liquid jets

Wim van Hoeve, Arjan van der Bos, Jacco Snoeijer, Michel Versluis, and Detlef Lohse
Physics of Fluids Group, University of Twente, P.O. Box 217, 7500 AE Enschede,
The Netherlands, w.vanhoeve@utwente.nl, +31 (0)53 – 489 4473

The formation of small droplets with a well-controlled and narrow size-distribution is an important issue in a variety of industrial applications, e.g. for ink jet printing, or for the formation of medical inhaler microsprays. We use a one-dimensional approximation of the Navier-Stokes equation to study the formation of microdroplets from liquid jets[1, 2]. Above a critical velocity a stable laminar jet emanates from a nozzle. The jet rises to a certain length, and then breaks up into droplets. Droplet formation in the Rayleigh breakup regime – where the breakup is not influenced by the surrounding air – is characterized by a linear relation between the breakup length and jetting velocity. Instabilities govern, and therefore strongly influence, the controllability and reproducibility of the formation process. Here, ultra high-speed imaging at 1,000,000 frames per second is used to visualize the breakup of a 10μm liquid jet and to validate the model. We provide experimental material that supports the validity of a universal scaling law for diminutive Rayleigh jets.

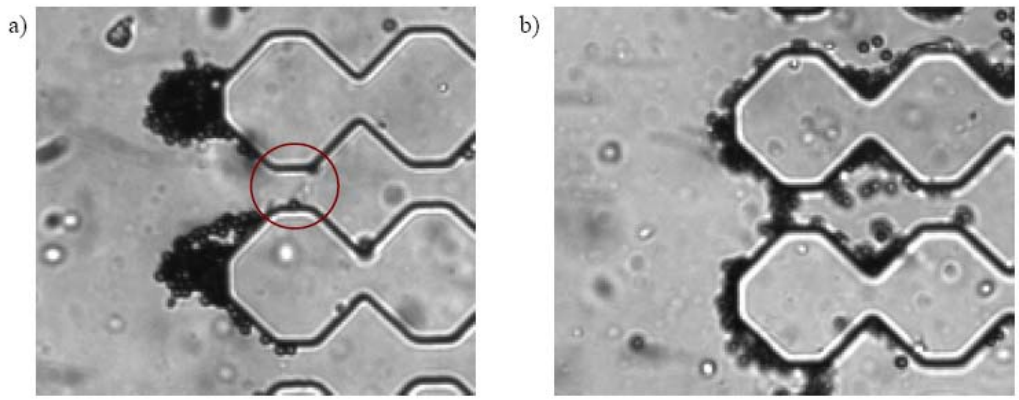
References

[1] X. D. Shi, M. P. Brenner, S. R. Nagel, A cascade of structure in a drop falling from a faucet, *Science* **265**, 219-222 (1994).
[2] J. Eggers, and T. F. Dupont, Drop formation in a one-dimensional approximation of the Navier-Stokes equation, *J. Fluid Mech.* **262**, 205-221 (1994).

# Observation of pore fouling by particles and cells in a microfluidic device

A. Marty[1], C. Agbangla[1], P. Bacchin[1], M. Meireles[1], P. Duru[2], D. Bourrier[3] and M. Dilhan[3]
1 Laboratoire de Génie Chimique
2 Institut de mécanique des fluides
3 Laboratoire d'Architecture et d'Analyse des Systèmes

The separation of micron-sized particle appears as an important operation in numerous industrial separations (environmental, chemical, pharmaceutical and biomedical applications). However, the mechanisms leading to the capture and the detachment of particles in a porous media are not well understood mainly because of the complexity of the interplay between hydrodynamic and particle interaction in complex porous geometry.
We develop a PDMS device with micro engineering techniques consisting in a single wide channel followed by an array of parallel micro-channels. The entrance of the micro-channels can be swept by a feeding flow with micron sized particles (similarly to a cross flow filtration mode) or feed by a flow in the axial direction of micro-channels (similarly to a dead end filtration mode). Such a device allows having a transverse view of the matter accumulation at the entrance or inside the micro-channels with different hydrodynamic conditions.
The experiments are performed with a dispersion of monodisperse latex particles (2 micrometers in diameter) in a series of micro-channels (twenty micrometers large). It will be presented conditions where the capture of particles is very different according to the conditioning step of the PDMS devices. In some conditions, the particles are captured on the surface between microchannels then leading to the growth of dentrites a). In other conditions, the particles form arches at the pore entrance b). These arches lead to the formation of a deposit which progressively blocks the pore entrance. Some results of cell filtration will also be presented.
The use of PDMS micro-separators allowed filtration experiments to be performed in a filter with a well-defined geometry and controlled hydrodynamic and physico-chemical conditions. Progress in understanding the clogging mechanism in such model filters could be an essential first step in controlling fouling of real porous materials.



# Scaling the drop size in coflow experiments.

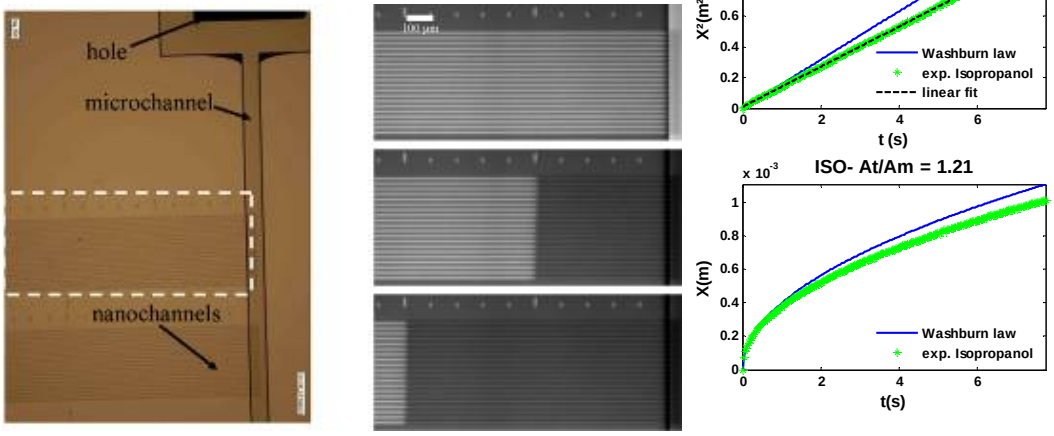
Elena de Castro and José Manuel Gordillo
Área de Mecánica de Fluidos, Departamento de Ingenería Energética y Mecánica de Fluidos, Universidad de Sevilla, Avda. de los Descubrimientos s/n 41092, Sevilla, Spain.

We perform extensive experiments with coflowing liquids in microfluidic devices. This configuration improves the drop or bubble generation process by making use of an outer coaxial flow. This provides several advantages with respect to using a quiescent bath: (i) Control of the drop size by appropriate tuning of the coflow properties; (ii) reduced coalescence between drops, in the absence of surfactants; and (iii) increased production frequency. In the absence of confinement effects, two different types of regimes have been identified, dripping and jetting. Dripping is characterized by the fact that no long jets of the dispersed phase are formed. By contrast, when jetting occurs, the dispersed phase forms long liquid jets and consequently, drops are emitted right at the tip of the liquid thread. In the jetting regime we could also identify two sub-regimes, widening and stretching. These names simply reject the shape adopted by the jet downstream of the injection tube in either regime. We provide a general expression, $d_d/Di = (1/Di)[(6Q_i\ d_j)/(k^*\ U_p)]^{1/3}$, to estimate the drop size ($d_d$) in either regime as a function of measurable parameters (jet diameter $d_j$, inner flow rate $Q_i$, wavenumber corresponding to the maximum growth rate of sinusoidal capillary perturbations k* and velocity of the jet at its most downstream position Up). This shows that despite the differences between these regimes, the drop size is governed by a unique scaling relationship. The only difference in either the narrowing or widening regimes is the way k*, dj and Up depend on the control parameters. Our experiments confirm this prediction, which can thus be used to design coflow experiments aimed to obtain droplets with a particular size distribution. This capability coupled to the possibility of multiplexing can contribute to the widespread use of this methodology.

# Nanofluidic systems: characterization of capillary filling rates

P. Joseph, V.-Nguyen Phan, L. Djeghlaf, A. Allouch, J.-C. Millet, D. Bourrier, P. Dubreuil, D. Belharet, A.-M. Gué, P. Abgrall, N.-T. Nguyen

Nanofluidics, which aims at controlling fluids in networks of channels with dimensions lower than 100 nm, is a fastly growing field. We report on technological developments for the fabrication of nanofluidic devices, and further test these devices experimentally, with the aim to characterize capillary invasion at the nanometer scale. Standard silicon microtechnologies based on slow plasma etching or on selective oxidation schemes have been revisited to obtain silicon/glass nanoslits. Using different solvents, we have conducted measurements of liquids filling kinetics inside these structures, and compared them to predictions and recent controversial experiments. With ethanol and isopropanol, our results are in quantitative agreements with macroscopic predictions for 80 nm nanoslits. For the thinnest nanochannels, deviations with filling kinetics 20% slower than expected are observed. In the case of water, complex filling processes are detected with trapping of bubbles which eventually dissolve, leading to a decreased flow-rate.



# Microscopic mechanism of electrowetting driven transitions on superhydrophobic surfaces

G. Manukyan[1,2], J. M. Oh[1], D. van den Ende[1], R. G. H. Lammertink[2], M. Wessling[2], and F. Mugele[1]
1 Physics of Complex Fluids, Faculty of Science and Technology, University of Twente
2 Membrane Technology Group, Faculty of Science and Technology, University of Twente, g.manukyan@utwente.nl

Superhydrophobicity [1] is commonly obtained by artificially adding small-scale roughness to hydrophobic surfaces. In many applications, ranging from “lab on a chip” devices to microelectronics cooling, it is desirable to adjust the degree of superhydrophobicity, i.e. by controlling how far the liquid penetrates the microscopic surface features. For a given surface topography a liquid droplet can be in two energetic states: 1) the Cassie Baxter state [2] when air remains trapped inside the microscopic crevasses below the droplet and 2) the Wenzel state [3] when the liquid completely fills those cavities. Previous studies suggest that a transition between the two states can be induced by electrowetting (EW), but the underlying mechanism is not clear, so far. We investigate, directly on the microscale, the morphology of the electrowetting induced transition between the Cassie Baxter and Wenzel state for a water droplet on a superhydrophobic surface.

In our experiments we use a microstructure with 8, 16 μm wide grooves (Fig.1). We analyze the light reflected from the micromenisci looking through the microstructure to the solid-liquid interaction area. Modulations in the intensity of the light reflected from individual micro menisci, indicates that the entrapped air below the droplet is gradually replaced. When applying an electric field the menisci bend, remaining pinned at the edges of the grooves. When the applied voltage excedes a critical value, the superhydrophobic state collapses, resulting in a transition to the Wenzel state (Fig. 1b). Numerical modeling of the system, based on a force balance between the Maxwell stress and the Laplace pressure, shows that above some critical voltage there is no equilibrium possible between these two forces, in accordance with our experimental observation.

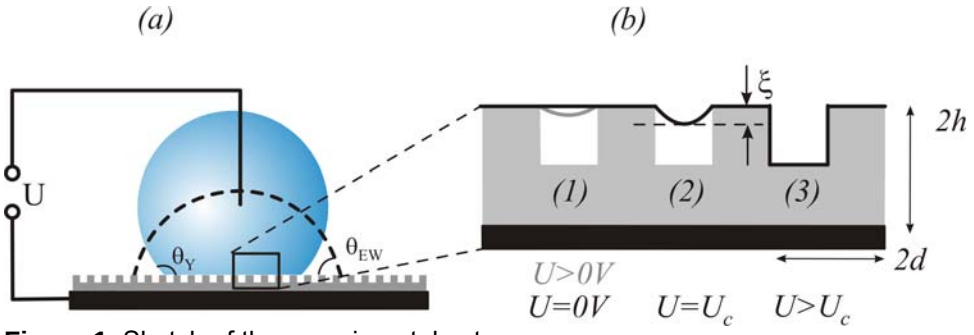


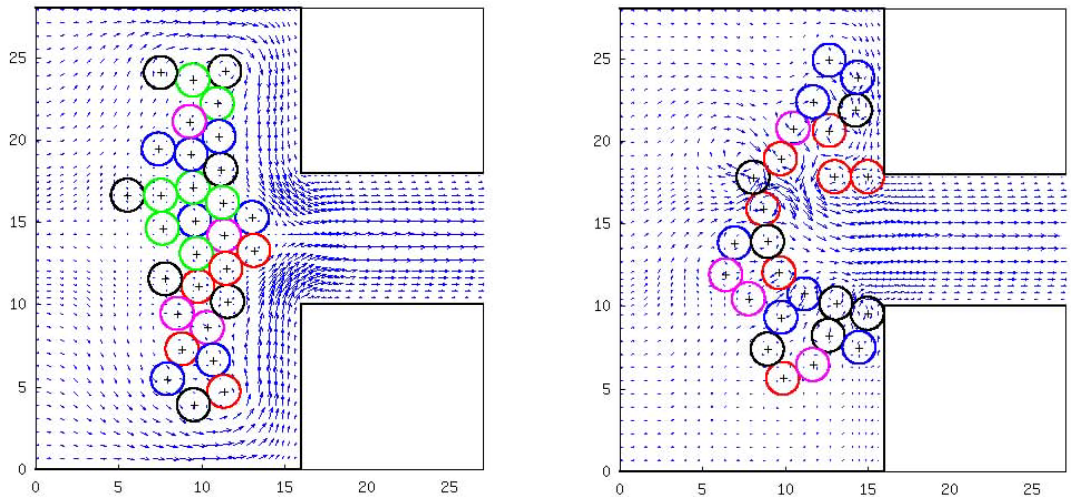

**Figure 1.** Sketch of the experimental setup.

[1] D. Quéré, *Reports on Progress in Physics* **68**, 2495 (2005).
[2] R. Wenzel, *Industrial & Engineering Chemistry* **28**, 988 (1936).
[3] A. Cassie and S. Baxter, *Trans. Faraday Soc.* **40**, 546 (1944).

# Numerical modelling of the flow of microparticles (channel and pore clogging)

Gbedo-Constant Agbangla[1], Patrice Bacchin[1] and Éric Climent[2]
1 Laboratoire de Génie Chimique, Toulouse
2 Institut de Méchanique des Fluides, Toulouse

In this study, the Force Coupling Method (FCM) is used to simulate the flow of colloidal particles in microchannels: filtration through a single pore and straight channels. The particles are experiencing simultaneously hydrodynamic stresses, particle/particle and particle/wall interactions.
The basic idea of the Force Coupling Method relies on the multipole decomposition of forcing terms accounting for the velocity perturbation induced by the presence of the particles. Those forcing terms are spread over spherical Gaussian wraps for spherical particles and are added to the continuous phase equations. The fluid flow is obtained by direct solution of the Navier-Stokes equations using the finite volume method. Each particle is represented by 6 to 8 grid nodes in the simulation domain.
This simulation model is used to simulate the effects of DLVO and hydrodynamic forces resulting in aggregation and clogging of pores occurring in a filtration process. We focus our study on simple geometries: straight microchannels and sudden section reduction similar to a pore entrance. Some preliminary results have been obtained in two-dimensional simulations. Figure 1(a) shows that particle-particle attractive forces together with particle-corner (or walls) repulsive forces lead to the formation of a stable aggregate at the pore entrance. The figure 1(b) shows the particle clogging effect caused mainly by particle-particle and particle-wall adhesion forces. These results are interpreted in terms of the temporal evolution of the permeability when the flow is driven either by a constant pressure drop or a constant flow rate. More realistic simulations are coming next using the fully parallel version of the simulation code.
Those simulation results will be validated by experiments in micro filtration channels with dispersions of spherical latex particles (radius: 5 μm) in controlled experimental conditions. This study should lead to a better understanding of the behavior of colloidal particles involved in many industrial (water waste treatment...) and physiological (aggregation of blood cells and platelets...) processes.




(a) Aggregation of particles at the pore entrance (b) Clogging of the pore by adhesion of particles

**Figure 1.** Aggregation of particles and clogging of a microchannel

# Mixing and electrothermal effect in AC electrowetting

Pablo Garcia-Sanchez, Frieder Mugele
Physics of Complex Fluids, Faculty of Science and Technology, IMPACT and MESA+ Institutes, University of Twente (The Netherlands)

*Electrowetting* refers to the reduction of the contact angle of sessile drops on partially wetting substrates by applying an external voltage [1]. The applied signals can be either DC or AC, and hydrodynamic flows within the drops have been reported for the case of AC voltages [2,3], fig. 1. Besides their fundamental interest, these flows are of relevant importance for the key problem of mixing in microfluidics.
The flows at low frequencies (fig. 1(a)) are generated by the oscillations of the drop interface. The amplitude of the oscillations vanishes as the frequency increases and so does the flow (fig. 1(b)). The fluid flow observed at higher frequencies (fig. 1(c)) lack of an explanation and this is the motivation for the theoretical study presented in this work. We have analysed the effect of Joule heating in AC *electrowetting* experiments and we have identified that these flows are generated by *electrothermal* effect [4,5], i.e. gradients in temperature give rise to gradients in conductivity and permittivity, the electric field acting on these inhomogeneities induces an electrical force that generates the flow.
We have solved numerically the equations for the electric, temperature and flow fields. The temperature is obtained from a convection-diffusion equation where the energy dissipation is introduced as a source term. From the solution of the electric field and the temperature, we compute the electrical force that acts as a body force in Stokes equations. The magnitude of the the velocity field depend on the conductivity of the liquid and on the frequency of the signal. The influence of these parameters was experimentally studied in [2] and their observations agree perfectly with our numerical predictions.

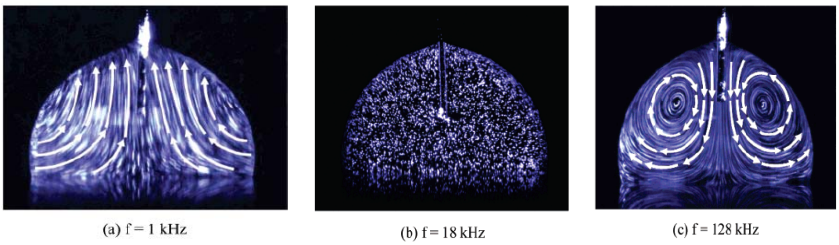



Fig.1. Hydrodynamic flows observed by Ko et al. [2]. Aqueous NaCl droplet of radius 1 mm (conductivity 13 mS/m). The amplitude of the AC signal is 80 Vrms.

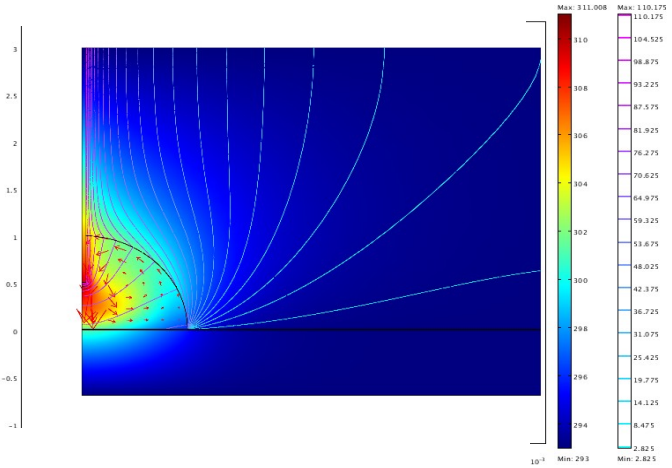


Fig.2. Flow pattern generated by electrothermal effect (125kHz,80Vrms). Typical velocities are around 1 mm/s. Contour lines correspond to the solution of the electric potential. The surface plot is the solution to the temperature field.

[1] F. Mugele and J.C. Baret. *J. Physics: Condensed Matter* 17, R705 (**2005**).
[2] S. H. Ko, H. Lee, and K. H. Kang. *Langmuir* 24, 1094 (**2008**).
[3] F Mugele, JC Baret, D Steinhauser. *Applied Physics Letters* 88, 204106 (**2006**).
[4] J. R. Melcher, M. S. Firebaugh. *Physics of Fluids 10 (6)*, (**1967**).
[5] A. Gonzalez, A. Ramos, H. Morgan, N. G. Green, A. Castellanos. *J. of Fluids Mechanics* 564 (**2006**).

# Drying of a square capillary tube

F. Chauvet, P. Duru, M. Prat
Institut de Mécanique des Fluides de Toulouse (IMFT)

Evaporation of volatile liquids in glass capillary tubes of square internal cross section is studied experimentally. The distinguishing feature of evaporation in a square tube is the presence of liquid films trapped by capillarity along the tube internal corners. These liquid films provide paths for the liquid between the receding bulk meniscus and the entrance of the tube. The liquid is transported within the films under the action of the pressure gradient induced by the meniscus curvature variation along the films. Situations similar to the one studied here, where a liquid confined in a corner or wedge flows under the action of capillary forces, are encountered in various research fields (e.g. microsystems like micro heat pipes) and have been discussed widely in the literature.
Here, we focus on the corner film dynamics, which is studied combining an infrared thermography technique and visualisations by ombroscopy. The experimental data are then analyzed using a simple model of flow within the corner films taking into account the effect of capillary, viscous and gravity forces. Among other things, this model leads to good predictions of film tip depinning from tube entrance and shows that the evaporation kinetic is quite sensitive to the corners internal roundness.

# Effect of cluster resolution on drag and reaction kinetic interaction in riser reactors

Schalk Cloete, Shahriar Amini, Stein Tore Johansen
SINTEF Materials and chemistry, Flow Technology research group, Department of Process Technology, NO-7465 Trondheim, Norway. Schalk.Cloete@sintef.no

Fine grid, reactive flow simulations have been completed on a 2D periodic section of a gas-solids riser reactor using the Eulerian KTGF approach. A fixed superficial gas velocity was maintained over the periodic section which contained a fixed volume fraction of solids. The flow development was then monitored for different spatial and temporal resolutions. The factors of grid width, grid aspect ratio and time step size were investigated in a central composite design. Model performance was reflected in four performance measures, solids flux, solids volume fraction, reaction time and simulation time.
The study revealed that full grid independence cannot be achieved even at a very small grid width as low as 5 particle diameters. It was encouraging, however, that the change in macroscopic system behavior, especially reaction kinetics, was not as sensitive to cluster resolution as expected. It is therefore possible that macroscopic system behavior can be adequately simulated using coarser grid resolutions. This possibility of attaining sufficiently accurate predictions with slightly poorer cluster resolution will be investigated in the future.

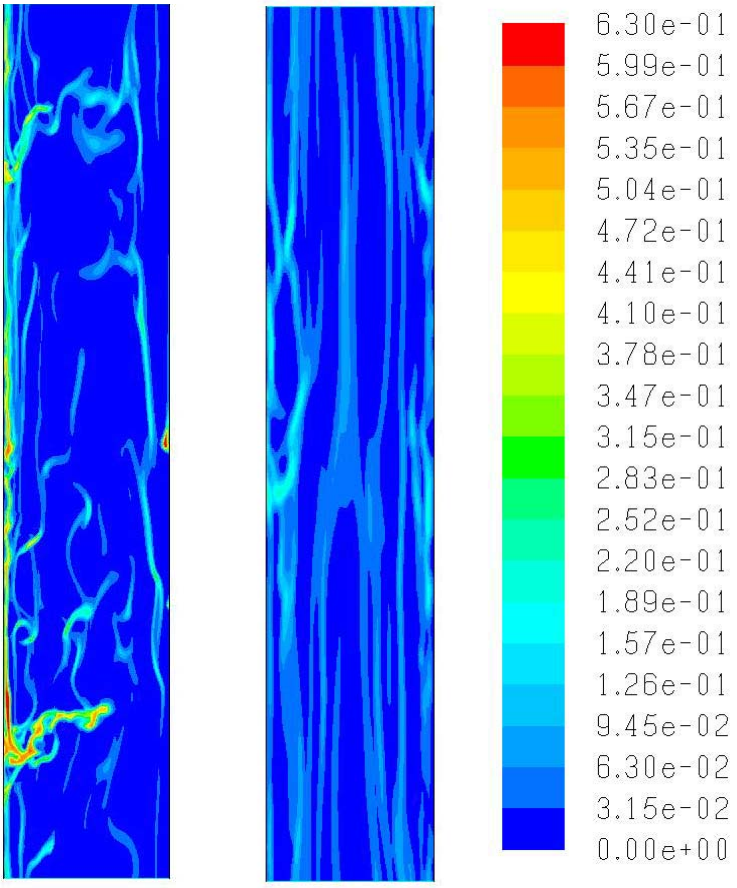



**Figure 1.** Solid volume fraction contours inside a riser with adequate (left) and inadequate (right) cluster resolution.

## Biofilms in porous media

Y. Davit, G. Debenest, M. Quintard
Institut de Mécanique des Fluides de Toulouse (IMFT)

Mass and momentum transport in biologically reactive porous media have been particularly examined within the context of bioremediation of the hyporheic compartment and especially of rivers subsurface. The reactive transport is mediated by micro-organisms (primarily bacteria, fungi, and algae) aggregated and coated within an extracellular polymeric matrix; together these are called biofilms. Using the method of volume averaging, various upscaling models have been previously developed but mainly local mass equilibrium models, that is, often excessively restrictive. In this context, we developed a model which does not require the assumption of local mass equilibrium and which is consequently less limitative. Additionally, we have elaborated a technique to image biofilms in porous media using X-ray computed tomography. This method finds its essence in the utilization of a combination of two contrast agents which allows to capture a 3-phase contrast (biofilms, water and grains). Using this 3D geometrical information at the small scale, it is possible to compute the effective parameters of the porous medium and to study the effect of biofilms on the transport properties.

# Carbon nanofibers (CNFs): a novel structured catalyst support layer for multi-phase microreactors

D. B. Thakur[1], R. M. Tiggelaar[2], J.G.E. Gardeniers[2], L. Lefferts[1], K. Seshan[1]
1 Catalytic Processes and Materials, 2Mesoscale Chemical Systems
University of Twente, P. O. Box 217, 7500 AE Enschede, The Netherlands
Corresponding author's email: K.Seshan@utwente.nl

Currently, there is tremendous interest in the field of micro-structured gas-liquid contactor (multi-phase microreactor [1]) containing modified catalytic coatings on its microstructured internals (microchannels). Catalytic support layers based on carbon nanofibers (CNFs) offer a novel option to facilitate this task [2, 3]. With their inherent high surface area-to-volume ratio CNFs provide increased catalytic surface area [2], obtaining sufficient activity per unit of volume of catalyst. Additionally the bulk density, diameter and length of the fibers can be manipulated to achieve high porosity (comprising mainly meso or macro-pores) with minimized tortuosity in order to optimize the accessibility of the active phase deposited on the CNFs. This helps to diminish the internal diffusion limitations by preventing any concentration gradients inside the CNF layer.

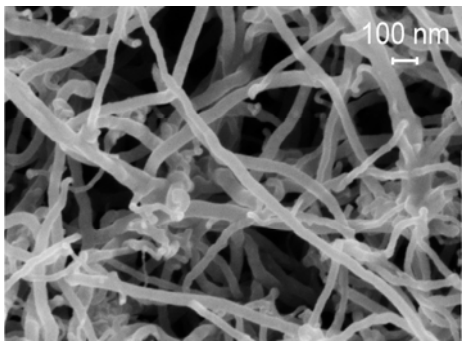



**Fig.1:** *SEM image of entangled CNFs*

In this work, synthesis of well-attached CNF layers on flat substrates (e.g. fused silica and silicon substrates representing microchannel wall surfaces), is attempted using a catalytic thermal chemical vapor deposition (C-TCVD) process of a hydrocarbon gas (e.g. ethylene) and metal-based (e.g. nickel) thin films. These well-attached layers of CNFs were further decorated with catalytic active phase such as ruthenium via typical solution based chemical methods and / or physical vapor deposition methods for testing them for appropriate multi-phase reactions.
It is expected that this novel approach of constructing microreactors comprising stable and well-defined layers of carbon nanofibers as a structured catalyst support will allow us to enhance activity and selectivity of the multiphase reactions being carried out and hence obtain reduced loss of waste products, achieving profitability for the fine chemicals and pharmaceutical industry along with significant environmental benefits.

[1] J. Kobayashi, et al., *Science* **304**, 1305 (2004).
[2] P. Tribolet and L. Kiwi-Minsker, *Catal. Today* **102-103**, 15 (2005).
[3] J.K. Chinthaginjala, D.B. Thakur, K. Seshan, L. Lefferts, *Carbon* **46**, 1638 (2008).

# Electrical properties of lanthanum and yttrium doped barium titanate ceramics sintered by Spark Plasma technique

P. Dufour[1], S. Guillemet-Fritsch[1], Z. Valdez-Nava[2], C. Tenailleau[1], J.Y. Chane-Ching[1]
1 CIRIMAT, LCMIE, Université Paul Sabatier, 31062 Toulouse, Cedex 09
2 LAPLACE, Université Paul Sabatier, 31062 Toulouse, Cedex 09
Email: dufour@chimie.ups-tlse.fr

In pursuit of high permittivity materials for electronic application, there has been a considerable interest recently in the dielectric properties of various perovskite oxides like barium titanates (BT) doped with various elements. When processed in a particular way, pure BT present at ambient temperature and at f = 1 kHz unusual interesting dielectric properties, a so called “colossal” *permittivity* value up to several $10^6$ [1]. Moreover and contrary to what is classically expected and evidenced for this type of materials, no temperature dependence of the permittivity is observed. This behavior is observed in nanopowders based ceramics. Nanometric powders of doped barium titanate are obtained by the co-precipitation synthesis. Two different n-type dopants are used and compared, lanthanum and yttrium. They are then sintered using the Spark Plasma Sintering technique. A sintering dwell of a few minutes at 1050 °C leads to high-densified ceramics. SEM FEG observations show that the grain size of the ceramics is less than 100 nm. This confirms that grain growth is avoided during SPS sintering. The electrical characterisation of these ceramics is performed in a wide temperature and frequency range. The obtained results are significantly different from those of conventionally sintered ones. In the temperature range -100 and +150°C, no Curie transition is observed anymore and the value of the relative permittivity is colossal. In this paper, the results concerning the synthesis, the sintering and the extraordinary electrical properties of the as-processed lanthanum and yttrium doped barium titanate will be presented. These results have important technological applications, since these nanoceramics open a new route to the fabrication of very thin dielectric films

[1] S. Guillemet-Fritsch, Z. Valdez-Nava, C. Tenailleau, T. Lebey, B. Durand, J.Y. Chane-Ching, Colossal permittivity in ultrafine grain size $BaTiO_3$-x and $La_{0.95}La_{0.05}TiO_{3-x}$ materials, *Advanced Materials* **20**, 551-555 (2008).

# Drag and lift forces on a counter-rotating cylinder in rotating shear flow

Chao Sun[1], Tom Mullin[2], Leen van Wijngaarden[1] and Detlef Lohse[1]
1. Physics of Fluids Group, Faculty of Science and Technology, J. M. Burgers Centre for Fluid Dynamics, MESA+ and Impact Institutes, University of Twente, The Netherlands.
2 Manchester Centre for Nonlinear Dynamics, University of Manchester, Oxford Road, Manchester M13 9PL, United Kingdom.

Flow around a rotating cylinder is of great fundamental interest but also of great importance in many industrial applications, such as flow control. We experimentally investigated the motion of a heavy cylinder in a drum filled with water, and rotating about a horizontal axis. The cylinder either co-rotates or counter-rotates with the rotating drum as shown in Fig. 1. The flow field around the cylinder, both for co-rotation and counter-rotation situations, was measured with Particle Image Velocimetry in order to investigate the different flow mechanism. For the counter‐rotation situation, the cylinder freely rotates without contact with the wall of the drum, due to the lift force acting on it. The drag and lift coefficients, on the freely counter‐rotating cylinder, were measured in a wide range of Reynolds numbers $2{,}500 < Re < 25{,}000$ and dimensionless rotation rates $0.0 < \alpha < 1.2$. We found that the drag coefficient is consistent with previous measurements on a cylinder in a uniform flow. However, a significant enhancement of the lift coefficient is observed in the present measurements. We expect the enhancement of the lift force is caused by the combined effects of rotation of the cylinder and the vicinity of a wall.

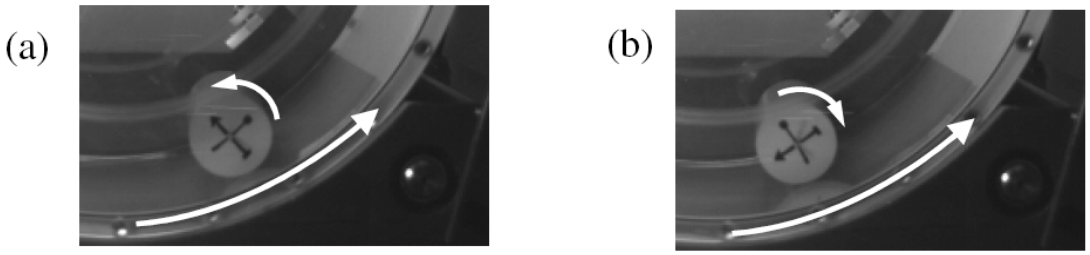



**Figure 1.** Representative images for the (a) co-rotating and (b) counter-rotating cylinder; the radius of the cylinder is 30 mm.

# Path instability of deformable bubbles in a Hele-shaw cell

Frédéric Risso[1], Anne-Marie Billet[2], Véronique Roig[1], Matthieu Roudet[1,2]
1 Institut de Mécanique des Fluides de Toulouse (IMFT)
2 Laboratoire de Génie Chimique de Toulouse (LGC)

The dynamics of isolated two-dimensional bubbles is experimentally investigated. The bubbles are said two-dimensional because they rise in a cell confined between two plates, and their equivalent diameter is greater than the distance between the plates. We study bubbles at high-Reynolds numbers, so that the inertial effects due to the hydrodynamic perturbation induced in the liquid by the bubble motion is not negligible compared to the viscous shear stresses. A shadowgraph method was used to study the bubbles shapes and paths for a large range of Archimedes numbers (100<Ar<12000). Increasing the Archimedes number, we observed several contrasted regimes of shapes and paths oscillations that may coexist or not, and that may be successively amplified or attenuated… Using PIV technique, the velocity field was investigated around the rising bubbles. We discuss the evolution with AR of the mean shape and of the structure of the wake, as well as the strong attenuation of the wake by the shear stress at the walls; these phenomena are essential to understand these regimes.

# Non-equilibrium condensation in swirling flow

Edwin van der Weide, Ryan Sidin and Rob Hagmeijer
Engineering Technology, University of Twente, P.O. Box 217, 7500 AE Enschede, The Netherlands

Selective condensation and extreme acceleration are the two key ingredients required to separate vapor components out of a gas/vapor mixture. In previous studies [1, 2] the Engineering Fluid Dynamics group at the University of Twente has implemented various models for non-equilibrium condensation, including the General Dynamic Equation (GDE) and its moment equations. The separation process was studied separately [3, 4].
Recently, these models have been replaced by the so-called Kinetic Equation (KE) [5, 6, 7, 8] and its moment equations [9]. This is considered a major step forward since the KE is a much more complete physical model than its first order approximation, the GDE. In all of these studies the quasi-one dimensional approach has been employed to compute the flow field and the droplet size distribution or its moments.
A second major step is to extend the quasi-one dimensional approach towards an axisymmetric approach, which is undertaken in the present study. First the mathematical-physical model for nonequilibrium condensation is presented, based on the Kinetic Equation and the Euler equations in a cylindrical coordinate system. Although the flow is assumed to be axisymmetric, it is still necessary to carry three momentum equations to account for the swirl. Secondly, the discretization technique is described. The major complication is the large number of governing equations, which results in a two order of magnitude increase in computational requirements. This is resolved by employing parallel computers in combination with domain decomposition as parallelization strategy [10]. The obtained method is applied to condensation in an air/water-vapor mixture leading to a detailed description of the droplet size distribution. Although the realization of such a simulation is a challenge in itself, one of the key questions is how the swirl will influence the droplet size distribution.

[1] R. Hagmeijer, *Phys. Fluids* **16**, 176-183, (2004).
[2] R. Hagmeijer, R.H.A. IJzermans, F. Put, *Phys. Fluids* **17**, 056101 (2005).
[3] R.H.A. IJzermans, R. Hagmeijer, *Phys. Fluids* **18**, 063601 (2006).
[4] R.H.A. IJzermans, R. Hagmeijer, P.J. van Langen, *Phys. Fluids* **19**, 107102 (2007).
[5] R.S.R. Sidin, R. Hagmeijer, U. Sachs, *Phys. Fluids*, revision phase (2009).
[6] J.H. Seinfeld, Atmospheric chemistry and physics of air pollution, (Wiley, New York, 1986).
[7] M.M.R. Williams and S.K. Loyalka, Aerosol science theory and practise, (Pergamon Press, Oxford, 1991).
[8] D. Kaschiev, Nucleation: basic theory with applications, (Butterworth-Heineman, Oxford, 2000).
[9] R.S.R. Sidin, PhD Thesis, University of Twente, in progress (2009).
[10] E. van der Weide, G. Kalitzin, J. Schluter, and J.J. Alonso, *AIAA* Paper 2006-421.

# Imaging method for interface dynamics characterization

Nicolas Abi Chebel[1], Frédéric Risso[2], O. Masbernat[1], P. Guiraud[3]
1 Laboratoire de Génie Chimique de Toulouse (LGC)
2 Institut de Mécanique des Fluides de Toulouse (IMFT)
3 Laboratoire d'Ingénierie des Systèmes Biologiques et des Procédés (LISBP)

Interface dynamics play a major role in drop break-up and coalescence mechanisms. We developed a new characterization method for drops interface dynamics that identifies the oscillation modes and measures their resonance frequencies and damping rates. This method, based on high speed imaging technique, was applied for both clean and surfactant-covered liquid interfaces.

# Energy and materials (EnMAT) sciences at the French FERMAT research Federation based in Toulouse

Christophe Tenailleau
tenailleau@chimie.ups-tlse.fr

A brief and general overview of the main research domains studied by the Toulouse based laboratories involved in the Energy and Materials (EnMAT) thematic of the French FERMAT research Federation will be firstly presented.
Then, the specific methods of preparation and structural characterisation of nanoparticles, powders, micro/nanostructures and thin films used at the CIRIMAT laboratory will be shown. The CIRIMAT is one of the few units of the French Carnot Institute, a quality label in terms of research and industrial partnerships. It develops multidisciplinary research domains in all the major families of materials (metals and alloys, ceramics, polymers, composites) from their conception to their use in real conditions. The synthesis and/or processing of new materials with original properties, the comprehension of the role of interfaces during synthesis and phase transformations, the study of correlations between structure and properties (magnetic, thermal, mechanical, biological, reactivity...) and the improvement of materials performances (durability, influence of surrounding environment, mechanical-chemical coupling, bioactivity, resorption behaviour in-vivo...), are among the main objectives of the CIRIMAT. Research projects are usually related to the development and characterisation of materials with useful applications in microelectronic systems, transports, metallurgy, information storage, biomaterials, etc… and more generally to the Energy. One of the 9 teams of the CIRIMAT Institute, the OVM team (for Oxides with Valencies that are Mixed), is composed of scientists specialised in the preparation and structural characterisation of materials, in relationships with physical property studies. Colloidal suspensions, co-precipitation and hydrothermal methods are the privileged methods of sample preparation with controlled morphology and microstructure. Usual solid state routes of synthesis are also parts of our specialities. Team members possess the experience with preparing thin layers of materials (by PVD and CVD techniques) and the most powerful Spark plasma Sintering platform in Europe is situated just a few steps away from our research centre, which is mainly used for preparing very dense ceramics at low temperatures (T<900°C) in very short times. We have a strong experience with structural characterisation: X-ray and neutron diffraction, as well as SEM/TEM/HRTEM microscopy are the main techniques used at our laboratory. Thermal analyses (TGA/DTA and DSC) and spectroscopy techniques are also available on site, and physical (electric and magnetic) measurements can be easily accessed.
Finally, the speaker's main thematic of research developed at the CIRIMAT and based on the preparation and study of new light-absorber oxide materials for their integration on solar cells will be described. Indeed, the use of renewable energy sources for producing electrical current is one of the main challenges of this century. Research has considerably been intensified during the last years on the development of materials for solar cells, devices that can convert solar energy into electricity by the photovoltaic effect. Solar cells of polycrystalline chalcopyrite type semiconductors and derivatives are presently expected to make promising contributions in the production of cost-efficient and long-term stable solar cells. But other types of materials based on the $Cu_2O$, $CuFeO_2$, $CuMn_2O_4$ or PLZT structures present very promising physical characteristics for their future use in solar cells.

# $Cu_2MSnS_4$ (M=Co, Zn, Fe) quantum dots

A.Gillorin[1], J.Y. Chane-Ching[1], P. Dufour[1], A. Baloochi[2], X. Marie[2]
1 CIRIMAT, Université de Toulouse, UPS, INPT, CNRS, 118 Route de Narbonne, F-31062, Toulouse, France.
2 Université de Toulouse, INSA, UPS, CNRS, LPCNO,135 Avenue de Rangueil, F-31077, Toulouse, France.

$CuInS_2$ quantum dots were synthesized and recently proposed for third generation solar cells, creating an intermediate band allowing the harvesting of a much larger portion of the available solar spectrum. Alternative sustainable absorber material in terms of cost and availability is the quaternary compound Cu2ZnSnS4 which is an analog of $CuInS_2$. This promising candidate has the suitable optical band gap energy of 1.4-1.5 eV and large optical absorption coefficient of $10^4$ $cm^{-1}$. All constituents of CZTS films are abundant in the crust of the Earth and non toxic. We have recently prepared quantum dots of $Cu_2CoSnS_4$, $Cu_2ZnSnS_4$ and $Cu_2FeSnS_4$. The TEM images show monodisperse size distribution centered around 4 nm. Absorption and photoluminescence characteristics of the quantum dots will be discussed and compared with nanosized $CuInS_2$ characteristics recently reported in the literature. Confinement effect and optical transitions of $Cu_2MSnS_4$ will be compared to those of $CuInS_2$ quantum dots.

# Nanostructured energetic materials based on Al/$CuO_x$ nanowires

M. Petrantoni[1,2], C. Rossi[1,2], V. Conédéra[1,2], L. Salvagnac[1,2], D. Bourrier[1,2], C. Tenailleau[2,3] and P. Alphonse[3]
1 CNRS ; LAAS ; 7 avenue du colonel Roche, F-31077 Toulouse, France
2 Université de Toulouse ; UPS, INSA, INP, ISAE ; LAAS ; F-31077 Toulouse, France
3 CIRIMAT, 118 route de Narbonne, 31062 Toulouse Cedex 04, France

During the last decades, scientific technologies have been adapted and reviewed to realise the transition between micro-and nano-systems. New materials have been developed in particular among energetic materials (EM), which represent an interesting source of onboard energy. These materials could be integrated into micro and nano-systems to produce gas or heat release. However, the use of conventional energetic materials is limited by the rate of reaction, which is relatively slow, the formulation (usually in powder or liquid), the structure and stability. The integration of such materials into microelectronic systems has involved new types of demand, especially for the development and structuring methods. In this context, thermites are very interesting composite energetic materials due to their high potential energy density (10-25 $kJ/cm^3$). Thermites are made of an oxidizer (metal oxide or non-metallic) and a fuel (usually aluminium). To increase the rate of reaction, the reactants intimacy must be improved and the specific surface area must be increased. Consequently, energetic composites are being nano-structured to reduce the diffusion distance of reactants, increase the reaction rate and reduce the ignition energy of the exothermic reaction.
Our team is currently working on the Al/CuO nanowires thermites because of the significant heat of reaction and the common use of aluminum and copper in the microelectronic fields. Al/CuO nanowires come from the thermal oxidation of copper thin film on silicon substrate at 450°C for 5 hours and aluminum deposition by thermal oxidation. Thermal evaporation method is limited by the thickness of the deposition. So, for a Cu layer thickness greater than 2 μm, only electrodeposition can be used. Both deposition methods can give CuO nanowires. For Cu with thickness less than 300 nm, no nanowire was grown. The copper oxide nanowire diameters are usually between 40 and 90 nm and those of Al/CuO nanowires are 150-250 nm. Al/CuO nanowires composites were characterised by scanning electron microscopy (SEM), X-ray diffraction (XRD), differential scanning calorimetry (DSC), as well as differential thermal analysis (DTA). Two distinct exothermic reactions occurred at 515°C and 667°C and total energy release of such thermite was $10kJ/cm^3$.
This material is necessary to ignite a reaction in a short pyrotechnic chain. Therefore, a micro-detonator integrating this nanocomposite was fabricated using a Au/Pt/Cr resistive layer on Pyrex substrate. The material ability to initiate another energetic material placed in contact was checked. This new nanosized energetic layer opens the door to many pyrotechnical applications like micro-propulsion, pneumatic activation and pyrotechnic initiation.

## List of participants

| First name | Institute | E-mail address |
|---|---|---|
| Micheline Abbas | INPT/ENSIACET | micheline.abbas@ensiacet.fr |
| Vijaya Raghav Ambati | IMPACT | v.r.ambati@math.utwente.nl |
| Patrice Bacchin | INPT/ENSIACET | bacchin@chimie.ups-tlse.fr |
| Onno Bokhove | IMPACT | o.bokhove@math.utwente.nl |
| Schalk Chloete | Sintef | Schalk.Cloete@sintef.no |
| Agathe Chouippe | IMFT | chouippe@imft.fr |
| Eric Climent | INPT/ENSEEIHT | climent@imft.fr |
| Yohan Davit | IMFT | davit@imft.fr |
| Elena de Castro Hernandez | Univ. of Seville | elenacastro@us.es |
| Niels Deen | IMPACT | N.G.Deen@utwente.nl |
| Pascal Dufour | UPS/CIRIMAT | dufour@chimie.ups-tlse.fr |
| Paul Duru | INPT/ENSEEIHT | duru@imft.fr |
| Pablo Garcia-Sanchez | IMPACT | p.garciasanchez@utwente.nl |
| Constant Gbedo Agbangla | INPT/ENSEEIHT | gbedoconstant@yahoo.fr |
| Stephan Gekle | IMPACT | s.gekle@tnw.utwente.nl |
| Jose Manuel Gordillo Arias de Saavedra | Univ. of Seville | jgordill@us.es |
| Pascal Guiraud | INSA | pascal.guiraud@insa-toulouse.fr |
| Sophie Guillemet | UPS/CIRIMAT | guillem@chimie.ups-tlse.fr |
| Rob Hagmeijer | IMPACT | r.hagmeijer.ut@gmail.com |
| Johan Hoorn | DSM-ACES | johan.hoorn@dsm.com |
| Jacco Hospers | IMPACT | j.m.hospers@ctw.utwente.nl |
| Sander Huisman | IMPACT | s.g.huisman@gmail.com |
| Jo Janssen | Unilever R&D | jo.janssen@unilever.com |
| Pierre Joseph | CR-CNRS, LAAS | pjoseph@laas.fr |
| Bart Lammers | Twister BV | bart.lammers@twisterbv.com |
| Leon Lefferts | IMPACT | l.lefferts@utwente.nl |
| Dominique Legendre | INPT/ENSEEIHT | legendre@imft.fr |
| Detlef Lohse | IMPACT | d.lohse@tnw.utwente.nl |
| Jacques Magnaudet | DR-CNRS | magnaudet@imft.fr |
| Ricardo Maniero | INPT/ENSIACET | riccardo.maniero@ensiacet.fr |
| Gor Manukyan | IMPACT | g.manukyan@utwente.nl |
| Alvaro Marin | IMPACT | a.g.marin@utwente.nl |
| Marco Martins Afonso | IMFT | martins@imft.fr |
| Aurélie Marty | UPS/CIRIMAT | amarty@chimie.ups-tlse.fr |
| Olivier Masbernat | DR-CNRS | olivier.masbernat@ensiacet.fr |
| Martine Meireles | DR-CNRS | meireles@chimie.ups-tlse.fr |
| Frieder Mugele | IMPACT | f.mugele@utwente.nl |
| Jung Min Oh | IMPACT | j.m.oh@utwente.nl |
| Marine Petrantoni | LAAS | marine.petrantoni@laas.fr |
| Frédéric Risso | DR-CNRS | risso@imft.fr |
| Véronique Roig | INPT/ENSEEIHT | roig@imft.fr |
| Paal Skjetne | Sintef | paal.skjetne@sintef.no |
| Jesse Slot | IMPACT | j.j.slot@ctw.utwente.nl |
| Henk Sollie | IMPACT | w.e.h.sollie@math.utwente.nl |
| Chao Sun | IMPACT | c.sun@utwente.nl |
| Christophe Tenailleau | UPS/CIRIMAT | tenailleau@chimie.ups-tlse.fr |
| Digvijay Thakur | IMPACT | D.B.Thakur@tnw.utwente.nl |
| Anthony Thornton | IMPACT | a.r.thornton@utwente.nl |
| Maureen van Buijtenen | IMPACT | m.s.vanbuijtenen@tnw.utwente.nl |
| Dirk van den Ende | IMPACT | h.t.m.vandenende@tnw.utwente.nl |
| Dirk van der Plas | Corus RD&T | dirk.van-der-plas@corusgroup.com |
| Jaap van der Vegt | IMPACT | j.j.w.vandervegt@math.utwente.nl |
| Edwin van der Weide | IMPACT | e.t.a.vanderweide@ctw.utwente.nl |
| Dirk van Eijkeren | IMPACT | d.f.vanEijkeren@utwente.nl |
| Wim van Hoeve | IMPACT | w.vanhoeve@utwente.nl |
| Dennis van Putten | Twister BV | dennis.vanputten@twisterbv.com |
| Martin van Sint Annaland | IMPACT | m.vansintannaland@tnw.utwente.nl |
| Peter Veenstra | Shell | Peter.Veenstra@shell.com |
| Bert Vreman | AkzoNobel | Bert.Vreman@akzonobel.com |