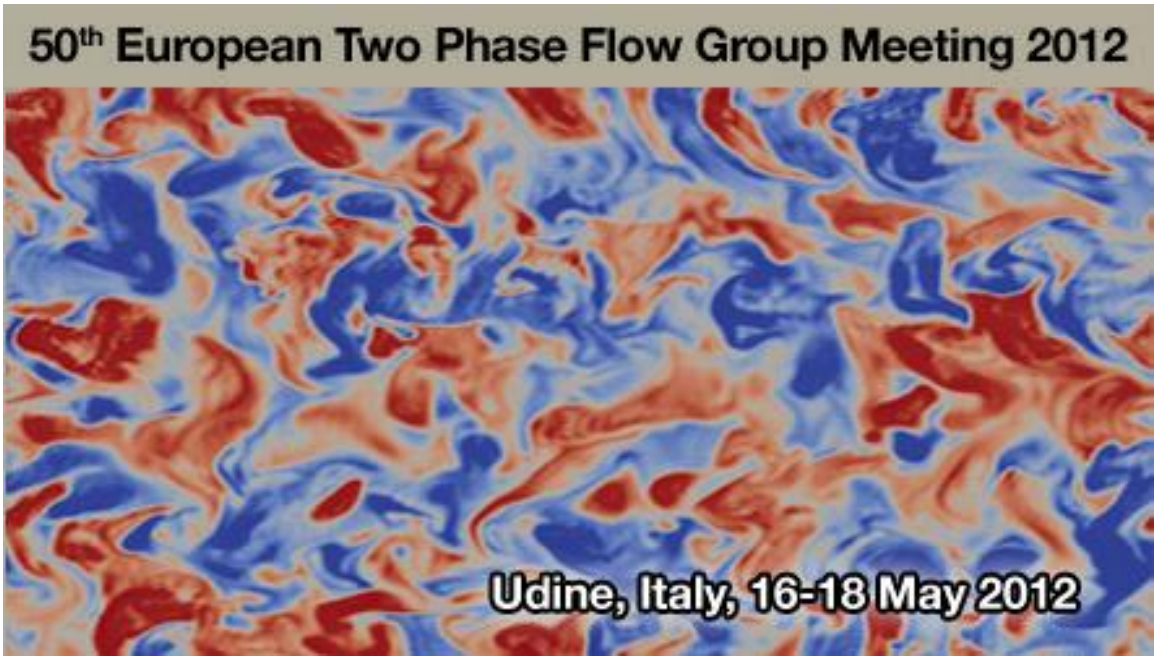


BOOK OF ABSTRACTS

Sponsored by:

CONFINDUSTRIA UDINE

CISM

Università degli Studi di Udine (Italy)

Comparative Analysis of 1D Relaxation Models Calculations Performed for Carbon Dioxide Two-Phase Transonic Flow Through Converging-Diverging Ejector Motive Nozzle

Wojciech Angielczyk[1], Jean-Marie Seynhaeve[1], Dariusz Butrymowicz[2], Yann Bartosiewicz[1]

[1]Université catholique de Louvain, Institute of Mechanics, Materials and Civil engineering
[2]The Szewalski Institute of Fluid-Flow Machinery, Polish Academy of Sciences

Abstract

One of the most important efficiency losses in a vapor-compression refrigeration system, especially in a transcritical CO_2 cycle, is caused by the throttling process [4]. The most promising solution is an application of a two-phase ejector as a compressor booster [1, 3, 4]. The schematic diagram of this solution is presented in Figure 1.

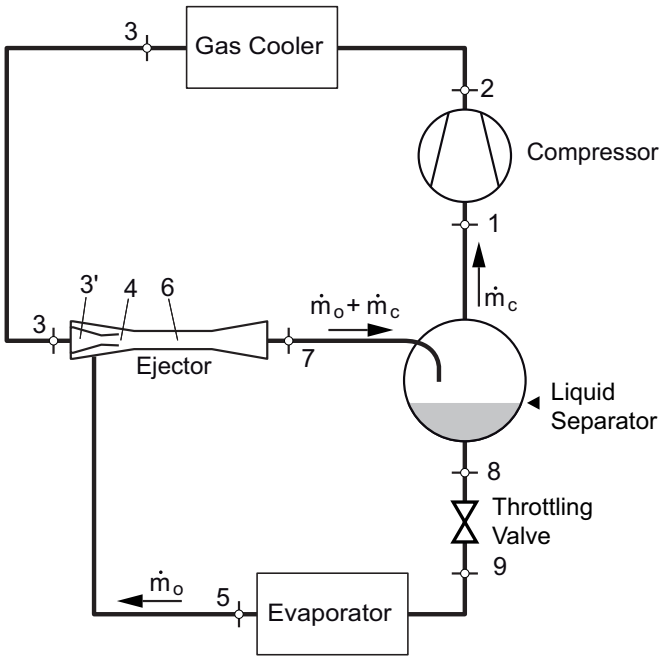


Figure 1: Ejector-compression refrigeration cycle.

Efficient ejector operation is mostly dependent on ejector geometry, where the converging-diverging motive nozzle plays an important role. In this nozzle, the pressure potential energy of the fluid is partly converted into kinetic energy. The lower pressure at the motive nozzle outlet, the higher motive stream velocity and the better potential of momentum exchange with the secondary stream farther downstream. For given operating conditions (inlet, outlet pressure) the maximum achievable velocity for a simple converging nozzle is fixed by the propagation velocity (the speed of sound). For a converging-diverging nozzle, the velocity at the nozzle outlet could be larger and depends on the divergent design. For two-phase flows this issue is additionally complicated by an occurrence of non-equilibrium effects: although the fluid pressure decreases below the saturation pressure, a phase change process does not occur and the flow remains single-phase (metastable liquid flow). After the pressure reaches an enough low value the phase change (flashing) starts but the formed two-phase flow is still far from equilibrium. In case of the CO_2 transcritical cycle, the described flow is even more complicated because of the fluid transition from supercritical to subcritical state. All those phenomena significantly influence the motive mass flow rate, which is a key parameter for an ejector performance. Therefore, an accurate prediction of a local speed of sound in non-equlibrium two-phase flows is a crucial issue for an ejector motive nozzle design.

Aforementioned circumstances were a motivation for research of a relatively simple and accurate way to describe the sound speed in two-phase flows. For this purpose, three approaches were analysed and compared, namely: homogenous equilibrium model, relaxation equilibrium model and delayed equilibrium model [1, 2]. Authors found that the absence of equilibrium can be relatively simply included in speed of sound predictions by usage of the relaxation models. Moreover, calculations performed on the base of an own implementation of mentioned models were compared to available experimental data [5].

References

[1] W. Angielczyk, Y. Bartosiewicz, J. M. Seynhaeve, and D. Butrymowicz. 1-d modeling of supersonic carbon dioxide two-phase flow through ejector motive nozzle. In *International Refrigeration and Air Conditioning Conference at Purdue*, Purdue, IN, July 12-15 2010.

[2] A. Attou and J. Seynhaeve. Steady-state critical two-phase flashing flow with possible multiple choking phenomenon part 1: Physical modelling and numerical procedure. *Journal of Loss Prevention in the Process Industries*, 12:347–359, September 1999.

[3] K. Banasiak and A. Hafner. 1d computational model of a two-phase r744 ejector for expansion work recovery. *International Journal of Thermal Sciences*, 50:2235–2247, November 2011.

[4] S. Elbel and P. Hrnjak. Experimental validation of a prototype ejector designed to reduce throttling losses encountered in transcritical r744 system operation. *International Journal of Refrigeration*, 31:411–422, May 2008.

[5] M. Nakagawa, M. S. Berana, and A. Kishine. Supersonic two-phase flow of co2 through converging-diverging nozzles for the ejector refrigeration cycle. *International Journal of Refrigeration*, 32:1195–1202, September 2009.

On the onset of stable dryout conditions in annular two-phase flows in heated channels

H. Anglart

Department of Physics, AlbaNova University Center, Royal Institute of Technology, 106 91 Stockholm, Sweden, henryk@kth.se

Extended Abstract

Dryout in annular two-phase flows occurs when the liquid film breaks up and a stable dry patch is established. Even though this mechanism is well known, there is still lack of well-accepted theory that elucidates the conditions for the stable dry patch formation and existence. In particular, one of the open questions is concerned with the minimum wetting rate, or film thickness, which is required to remove the dry patch.

In various phenomenological models of dryout, which are based on mass balance of the liquid film, it is assumed that the dryout condition corresponds to a complete liquid film disappearance. This is the basis to determine the dryout conditions in models developed by e.g. Hewitt and Govan (1990) and Adamsson and Anglart (2010). However, experimental data obtained by Ueda and Isayama (1981) with Freon indicate that dryout can occur even when the mass flow rate in the liquid film per unit perimeter is as high as 0.8 kg/s.m. This so called critical film flow rate is virtually equal to zero when the flow quality is higher than 50 %, but it rapidly increases when the quality decreases below that value.

The calculations performed for steam-water mixtures at various pressures indicate that the dominant forces which govern the dry patch stability are due to the stagnation pressure and the surface tension effects. In absence of other forces, the minimum wetting rate is governed by the balance between these two forces. With increasing heat flux, the importance of the thermo-capillary force grows and its neglect may lead to a significant error. The fourth important force is due to the skin friction. Its magnitude can be comparable to the stagnation force for small contact angles. This effect increases with increasing pressure of the two-phase mixture. Sensitivity study shows that one of the most important parameters affecting the over-all force balance is the contact angle appearing in the force balance equation. Application of the static contact angle may lead to significant errors and the dynamic contact angle should be used. The results of the present analysis can be used to support a formulation of a mechanistic model of dryout in various computational thermal-hydraulic codes.

1. G. F. Hewitt and A. H. Govan, “Phenomenological modeling of non-equilibrium flows with phase change”, *Int. J. Heat Mass Transfer,* Vol. 33, No. 2, 1990, pp. 229-242.
2. C. Adamsson and H. Anglart, “Influence of axial power distribution on dryout: Film flow models and experiments”, *Nuclear Engineering and Design,* Vol. 240, 2010, pp. 1495-1505.
3. T. Ueda and Y. Isayama, “Critical heat flux and exit film flow rate in a flow boiling system”, *Int. J. Heat Mass Transfer*, Vol. 24, No. 7, 1981. pp. 1267-1276.

50th. European Two-Phase Flow Group Meeting 2012.
16th-18th May 2012, by Universita degli Studi di Udine in Udine, Italy.

Authors: Gustavo Rabello dos Anjos gustavo.rabello@epfl.ch
Navid Borhani navid.borhani@epfl.ch
John R. Thome john.thome@epfl.ch

Heat and Mass Transfer Laboratory (LTCM)
Ecole Polytechnique Fédérale de Lausanne - EPFL
Lausanne, Switzerland - phone +41 21 6935442

Title: 3D ALE-FEM Simulation of Microscale Two-Phase Flows with Phase Change

Numerical simulation is employed to simulate diabatic two-phase flow phenomena using the continuum method for surface tension modeling. The set of equations are based on the 'one-fluid' Arbitrary Lagrangian-Eulerian (ALE) description of the Navier-Stokes, which includes the mass, momentum and energy conservation equations. These equations are discretized by the Finite Element method on a tetrahedral unstructured grid in which the phase boundary is represented by a triangular surfaces that are part of the volumetric computational mesh, thus a sharp and precise representation is successfully achieved. This geometrical procedure also ensures undesirable modes and spurious oscillations are damped out, thus leading to the convergence of the results. A Laplacian smoothing operator is applied to the volumetric and surface meshes to keep the elements homogeneously distributed, thereby avoiding large concentrations of nodes in one specific region due to the moving interface. Moreover, by varying a single parameter, the formulation can be set to a fixed or a complete moving mesh technique. The new methodology proposed here to simulate diabatic two-phase flows in the ALE context is the fist time this has been done for phase change. It is shown to provide an accurate description of the interfacial forces, bubble dynamics, the heat and mass transfer between phases. The employed formulation, the interface representation, the phase change model, and results will be presented. Furthermore, 3D microscale simulations of diabatic slug flow will be presented.

[1] J.U.Brackbill, D.B. Kothe and C. Zemach, A Continuum Method for Modeling Surface Tension, Journal of Computational Physics,100:335-354, 1992.

[2] J. Donea, A. Huerta, J.-Ph. Ponthot and A. RodrÌguez-Ferran, Arbitrary Lagrangian-Eulerian Methods, Encyclopedia of Computational Mechanics, Volume 1: Fudamentals, John Wiley and Sons, 2004.

[3] S. Negami, Diagonal flips in triangulation of surfaces, Discrete Mathematics, 135:225-232, 1994.

[4] A. Esmaeeli and G. Tryggvason, Computations of film boiling. Part I: numerical method, International Journal of Heat and Mass Transfer, 47: 5451-5461, 2004.

[5] G.R. Anjos, N. Borhani, N. Mangiavacchi and J.R. Thome, A 3D ALE-FEM Method for Two-Phase Flows, Journal of Computational Physics, submited, 2012.

Local and Instantaneous Temperature Field around a Taylor Bubble

Valery Babin, Dvora Barnea and Lev Shemer.
Tel Aviv University, Faculty of Engineering, Tel Aviv, 69978, Israel
babinv@eng.tau.ac.il

Heat transfer mechanism in two-phase flows and particularly in slug flow is of high interest both for basic hydrodynamic research and industrial applications. The flow field in the liquid around a Taylor bubble propagating in a vertical pipe can be subdivided into three distinct hydrodynamic regions: the thin liquid film surrounding the gas bubble, a highly turbulent liquid wake in the vicinity of the bubble bottom and a the liquid slug body away from the Taylor bubble. The hydrodynamic parameters in each region were investigated in numerous studies during the last decades. Due to the complexity and intermittent nature of slug flow, the available information on heat transfer characteristics in slug flow is quite limited.

The aim of the present study is to improve the understanding of the transient heat transfer mechanism in slug flow and to relate it to the local flow field parameters around the Taylor bubble. The experimental facility consists of air and water supply systems and a test section for heat transfer investigations. An infrared thermography technique was used to determinate the temperatures field on a thin metal foil heated by electrical current. The foil was installed in a window flush with the inner wall of the pipe so that the inner side is open to the flow (Fig. 1). Controlling the heat supply to the foil allows calculation of the local heat transfer coefficient at each of the three hydrodynamic regions of the flow.

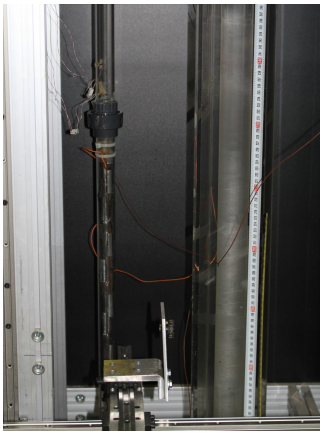


Figure 1. Experimental setup

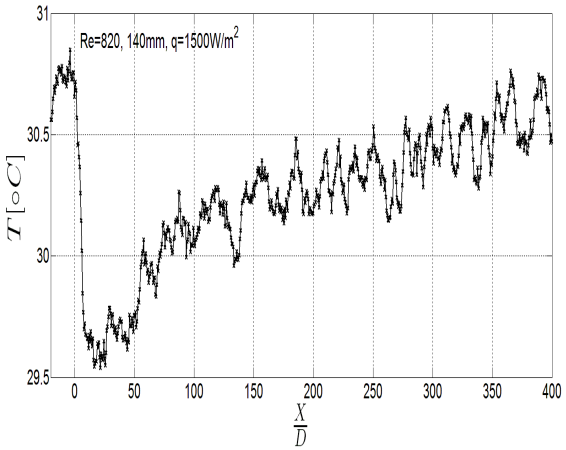


Figure 2. Instantaneous temperature variation in the course of the Taylor bubble passage

Data on the instantaneous local temperature variation in the process of passage of a Taylor bubble was collected for a variety of heating conditions: for two different pipe diameters, various liquid flow rates corresponding to laminar, turbulent and transitional background flow regimes, at various locations along the heated foil. The results were obtained relative to the Taylor bubble location. An example of the local temperature as a function variation with the distance from the Taylor bubble bottom is presented in Fig. 2.

50th European Two-Phase Group Meeting 2012
May 16-18, 2012, Udine, ITALY

RISE VELOCITY OF A TAYLOR BUBBLE IN DOWNWARD LIQUID FLOW

Valery Babin, Dvora Barnea and Yehuda Taitel
Tel-Aviv University, Faculty of Engineering, Tel-Aviv, 69978, Israel

Abstract

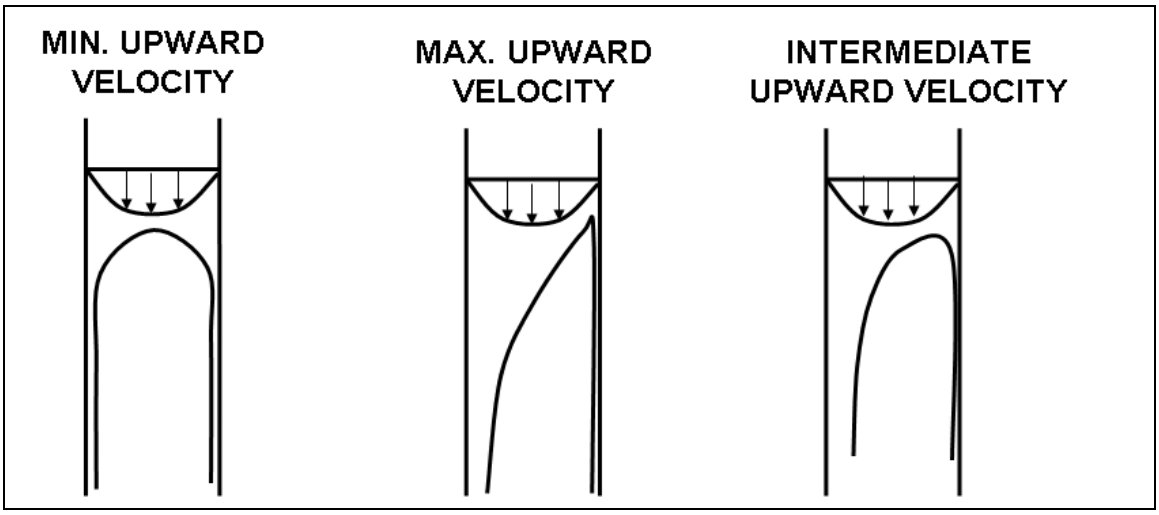
The translational velocity of a Taylor bubble in a vertical pipe in stagnant or upward liquid flow is well documented. In this case the bubble nose is symmetric and it has an approximate spherical shape. In the case of downward liquid flow the bubble shape can be asymmetric. The tip of the bubble is inclined to the pipe wall to avoid the fast moving liquid in the pipe center. If the liquid flow rate is high enough the bubble may remain stationary or even decent instead of rising (Polonsky et al. 1999, Espinoza & Fabre, 2011).

The translational velocity of a Taylor bubble is affected by two factors. The velocity of the liquid and the buoyancy induced velocity of the bubble. It is usually assumed that the propagation velocity of the Taylor bubble is related to the maximum liquid velocity ahead of the bubble tip. This assumption was confirmed by Polonsky et al. (1999) by direct measurements of the cross sectional velocity profiles using PIV in stagnant upward and downward liquid flow.

For upward liquid flow this maximum is at the pipe center. However for downward negative flow the maximum is at the pipe wall where the velocity is zero. As a result, the nose of the Taylor bubble tends to move towards the pipe wall.

In our experiments we identified 3 different translational velocities. (1) A negative translational velocity where the bubble nose is symmetric, (2) The fastest upward translation velocity where the nose is almost touching the pipe wall and (3) an intermediate velocity were the nose is close to the wall.

The 1st and the 2nd types are unstable and do not survive for long. The 3rd one is the final steady upward translational velocity. It moves faster than the symmetric one (1st type) and slower than the 2nd type.



Self-dispersion of high-Re bubbles rising within a thin gap.

E. Bouche[1,3], V. Roig[1,3], F. Risso[1,3], A. M. Billet[2,3]

1 Institut de Mécanique des Fluides de Toulouse, Université de Toulouse (INPT, UPS) and CNRS, Allée C. Soula, Toulouse, 31400, France.
2 Laboratoire de Génie Chimique, Université de Toulouse (INPT,UPS) and CNRS, 4 allée E. Monso, BP74233, Toulouse cedex 4, 31432, France.
3 Fédération de recherche FERMaT, CNRS, Toulouse, France

(Received 3 February 2012)

A lot of chemical, biological or environmental applications involve bubble column technology for heat and mass exchangers or reactor devices. Bubbles are used to maximize interfacial area and enhance mixing. Their displacements induce liquid agitation that in turn influences the bubble distribution and velocity. A general understanding of this two-way coupling is fundamental. It is however particularly difficult to achieve. Comprehensive investigations of well-defined elementary situations still appear to be necessary. The present work focuses on a swarm of bubbles rising at large Reynolds number in a Hele-Shaw cell (Bouche *et al.* (2012)). This configuration is attractive for the general understanding of bubbly flows since it makes possible to study the agitation generated by large-Reynolds-number rising bubbles with unstable wakes while turbulence production is prevented by the strong confinement. Comparisons with unconfined situations are thus expected to be meaningful. Moreover, interface detection is considerably facilitated compared to classic three-dimensional cases because bubbles cannot hide each others: a complete and accurate description of bubble interfaces can therefore be obtained by means of a single camera facing the cell. The dynamics of the gas phase has been investigated in a homogeneous swarm of bubbles rising within a Hele-Shaw cell for gas volume fractions α between 1 % and 14 % in a flow regime where inertia plays a major role ($Re \approx 500$). Each bubble is followed by unstable wake where a regular vortex shedding occurs, and rises along an oscillatory path while keeping an almost constant elliptical shape (Roig *et al.* (2011)). However, in contrast to what is observed in unconfined situations, the wake is strongly attenuated by the wall friction.

Analysis of the experiments reveals that two phenomena control the statistics of the bubble motions: the wake-induced oscillations and the strong velocity disturbances localized at the bubble rears. Depending on the direction that is considered, the relative importance of these two phenomena is changed and the physical mechanisms are totally different.

In the vertical direction, the major mechanism is the entrainment by the wakes. First, it induces an increase of the mean bubble rise velocity with the gas volume fraction (Fig. 1). Second, it causes an almost linear increase of the variance of the vertical bubble velocity as α increases (Fig. 1). This linearity comes from the fact that the bubble wake remains almost unchanged as α increases whereas the probability that a bubble is located within the wake of another bubble is proportional to α. Finally, wake entrainment generates a vertical dispersion of the bubble characterized by a dispersion coefficient with is almost proportional to α (Fig. 2).

In the horizontal direction, the bubble agitation is essentially caused by the wake-induced oscillations, the intensity of which is independent of the gas volume fraction. As a consequence, the variance of the bubble velocity is independent of α (Fig. 1). Sinusoidal oscillations are unable to cause dispersion. However, random interactions between bubbles, by causing the decay of the sinusoidal time correlation of the bubble velocity, generate a horizontal bubble dispersion, which increases as the rate at which a bubble is perturbed by an other bubble. The horizontal dispersion coefficient is

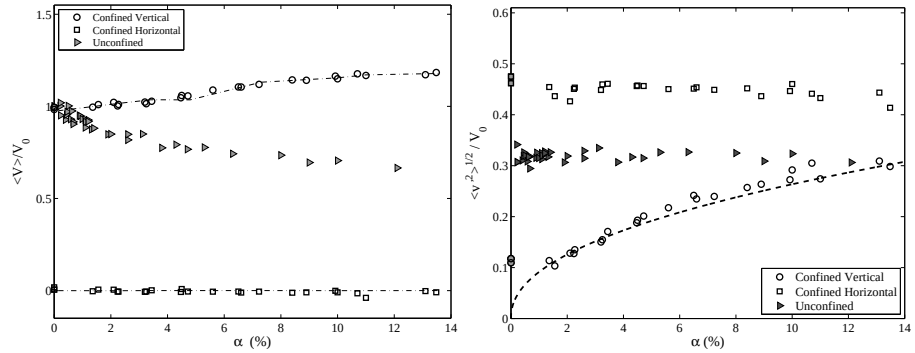


FIGURE 1. Mean bubble velocity and standard deviation of the bubble velocity normalized by the mean velocity of the single rising bubble. □, horizontal component , ○, vertical component, △, vertical component in an unconfined bubble swarm from Riboux *et al.* (2010)

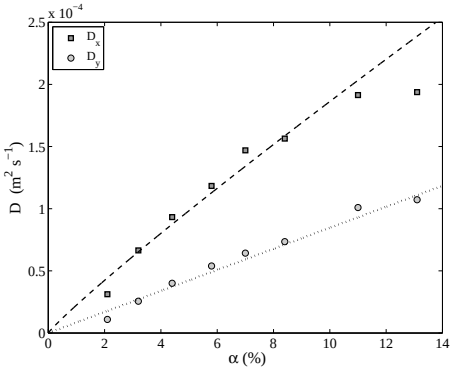


FIGURE 2. Evolution of the dispersion coefficients as a function of α.

also found to increases linearly with α but remains smaller than that in the vertical direction (Fig. 2).

Thanks to the advantages of a two-dimensional configuration for optical measurements, we have been able to draw a quite comprehensive picture of the mechanisms controlling the bubble dynamics.

REFERENCES

BOUCHE E., ROIG V., RISSO F., BILLET A. M. 2012 Homogeneous swarm of high-Reynolds-number bubbles rising within a thin gap. Part 1: Bubble dynamics. *J. Fluid Mech.* , submitted

RIBOUX G., RISSO F., LEGENDRE D. 2010 Experimental characterization of the agitation generated by bubbles rising at high Reynolds number. *J. Fluid Mech.* **643**, 509–559.

ROIG V., ROUDET M., RISSO F., BILLET A. M. 2011 Dynamics of a high-Reynolds-number bubble rising within a thin gap. *J. Fluid Mech.* , submitted

Why do miscible liquids take so long to mix?

E. Brunazzi, C. Galletti and R. Mauri

In the past 10 years there have been a series of works claiming that when two miscible fluids are brought in contact with one another, an effective interfacial tension keeps them well separated for a time that is much longer than one would expect. The experiments followed the spinning drop method for measuring the interfacial tension, while the theoretical model was based on the fact that surface tension is a coarse-grained manifestation of the Korteweg stresses, arising in systems far from equilibrium, i.e. in the presence of chemical potential gradients. [B. Zoltowski, Y. Chekanov, J. Masere, J. Pojman, V. Volpert, *Langmuir* 23*,* 5522 (2007); J. Pojman, Y. Chekanov, . V. Wyatt, N. Bessonov, V. Volpert, *Microgravity Sci. Technol* 21, 225 (2009)]

First of all, the explanation did not seem convincing, as the Korteweg stresses should accelerate, not retard, the drive of a system towards its thermodynamic equilibrium. Therefore, to understand the phenomenon, we conducted a simple experiment, where two miscible fluids, i.e. water and ethanol, were brought in contact within a T-shaped micromixer at low Reynolds number, so that no vortices were observed in the mixing region. When we compared the results with those obtained when the liquids at the inlets were both water, we saw that, at the confluence, the separation line between the two phases was much sharper, and the resulting mixing length was consequently much longer. However, we also observed that, as soon as the flow is stopped, the two miscible liquids mix very rapidly (i.e. faster than two water streams), indicating that the sharp separation between the two miscible liquids is not a thermodynamic feature (as the name *effective surface tension* would indicate), but it is due to the fluid motion.

The explanation of our results, we believe, is that, as the two fluids start to mix, the resulting mixture looses some volume (5% in the case of a 50-50 water-ethanol mixture) and this, in turn, induces a convection that is opposite to the diffusive fluxes. Our model was further clarified through a simple perturbation analysis, and then confirmed by numerical simulation.

Simulation of stratified gas-liquid flow in near-horizontal pipes

B.Calgaro, P.Andreussi,
University of Pisa, Italy

V. Faluomi, P. Ciandri
TEASistemi, Italy

M.Margarone, ENI E&P, Italy

Abstract

The simulation of gas-liquid flow in pipes can be based on the solution of steady state or transient 1-D conservation equations, which depend on a set of closure relations, in general of empirical nature. The closure relations adopted in commercial codes are similar to those proposed in the open literature, on the basis of data taken at low pressure, in small diameter pipes. These correlations have then been modified on the basis of experimental data taken at conditions which more closely resemble field conditions. Field data are used for the final validation of these codes. Notwithstanding the amount of work performed for code testing and validation, quite often the simulations of real pipelines are unsatisfactory, the reason probably being that from one hand the quality of available data is poor, from the other, these data are quite limited and do not cover the full range of flow parameters encountered in practical applications.

In the present work, we use different data sets relative to stratified gas-liquid flow in near horizontal pipes to test major commercial codes used for pipeline simulation. Among these data we include a new set of measurements of pressure drops and liquid hold-up taken at TEASistemi Laboratory in Pisa, Italy. These data are relative to water-nitrogen flow in a 4" horizontal pipe and at operating pressures in the range 5-25 Bar. Considering the values of pipe diameter and gas density, these data represent a bridge between published data and the data used for code development and validation, both of which are also examined in the present work. The results obtained allow to shed some light into the reliability of commercial codes adopted for the hydraulic design of multiphase pipelines and will be used for the development of advanced correlations to be introduced in a new flow simulator named MAST(1) which allows the user to choose among a library of closure relations or, eventually, to introduce his own closures.

(1) M.BONIZZI, P.ANDREUSSI, S.BANERJEE, "Flow Regime Independent, High Resolution Multi-Field Modelling of Near-Horizontal Gas-Liquid Flow in Pipelines", **Int. J. Multiphase Flow**, **35**, 34-46, 2009.

Minimal perfusion flow for cellular growthon lattice scaffolds

M. Campolo

Center for Fluid Mechanics and Hydraulics, University of Udine, 33100 Udine, Italy

Extended Abstract

In this work we propose a modelling approach to identify sets of culture conditions to promote homogeneous growth of cells in perfusion bioreactors equipped with regular shape scaffolds. We identify cases in which dynamic culturing is necessary using a zero dimensional mass tran sport and reaction model. Then, based on the three dimensional rendering of the flow field inside the bioreactor, we identify regions where cellular growth may become critical; finally, using a one dimensional mass transport and reaction model we calculate the minimal perfusion flow necessary to maintain the cellular growth rate above a target threshold.
The developed approach is used to analyse culturing conditions inside an indirect perfusion bioreactor equipped with a lattice scaffold. Regions where the perfusion flow is inadequate to foster cellular growth at the desired rate are identified. The perfusion flow required to maintain the target growth rate inside the bioreactor is calculated.

Lagrangian tracking of evaporating droplets in a homogeneous nearly isotropic turbulence using in-line digital holography

D. Chareyron, J. L. Marie, J. Gire, C. Fournier, N. Grosjean, L. Denis, M. Lance, and L. Mees

University of Lyon,
Lyon and St-Etienne, France

We present here an in-line digital holography technique to measure Lagrangian 3D trajectories and size evolution of droplets evaporating in a high Re number turbulence. The experiment has been performed in a homogeneous, nearly isotropic, turbulence created by the meeting of 6 synthetic jets. The holograms of droplets have been recorded with a single high-speed camera at framing rates of 1-3 kHz. The originality of the technique lies in the use of "an inverse method" to reconstruct the holograms time series, hence the tracks. It allows the detection of droplets out the field of view and increases the accuracy on the estimated droplet positions and size. It has been first validated in a non-evaporating case, with 60μm diameter water droplets. Lagrangian statistics on 1000 reconstructed tracks are presented. The PDFs of the Lagrangian accelerations show that the position uncertainty in the longitudinal direction remain higher, as expected with an in-line set-up, but can be decreased to that of the transverse directions by using a filtering algorithm. The accuracy on the diameter measurement is about 1.6%, and proves to be quite suitable for evaporation studies. The method has then been tested with R114 Freon droplets at the early stage of evaporation. The striking feature is the presence on each hologram of a thermal wake image, aligned with the relative velocity fluctuations seen by the droplets (crossing trajectory effect). Its orientation compares rather well with that calculated by using a dynamical equation for describing the droplet motion. A decrease of size due to evaporation is measured for the droplet having the highest residence time in the turbulence domain.

50th European Two-Phase Flow Group Meeting
CISM, May 16-18, 2012, Udine (Italy)

Recent advances in stochastic modelling of turbulent particle-laden flows

Sergio Chibbaro
Institut Jean Le Rond D'Alembert
Université Pierre et Marie Curie
case 162 4, place Jussieu
75252 Paris Cedex 05

We would like to present an approach which can be particularly suited for turbulent two-phase flows: the stochastic modelling or probability density function (PDF) approach. These models have been first developed for reactive flows [S. B. Pope, Prog. Energy Combust. Sci., 11:119, 1985] and afterwards extended to turbulent polydispersed two-phase flows [J.-P. Minier and E. Peirano, Physics Reports, 352:1, 2001]. The key advantage of this approach with respect to RANS approach (the two-fluid model in two-phase flows) is the possibility of treating turbulent transport and non-linear local terms (reaction as well as polydispersion) without approximation. While for simple flows RANS or even more heuristic models can work reasonably well, whenever some complex phenomenon are added (reaction, polydispersion, strong turbulence, etc.) Lagrangian models represent a viable and much more accurate alternative. Moreover, these models apper to be the natural choice for LES subgrid modelling.
From a practical point of view, the fluid phase is solved with an Eulerian method (typically RANS) and particles are tracked from a Lagrangian point of view. More specifically, a stochastic model is used to represent the particle dynamics (particle position, its velocity and the velocity of the fluid seen by particles). Some details about the method will be given together with some recent results, notably obtained for the deposition and resuspension of particles in a channel flow.

ETPFGM'12 - Udine

Condensation in minichannels: effect of channel shape and inclination

Davide DEL COL, Stefano BORTOLIN, Alberto CAVALLINI

University of Padova, Dipartimento di Ingegneria Industriale
Via Venezia1, I-35131 Padova, Italy
Email: davide.delcol@unipd.it

This work is aimed at studying the effect of cross sectional shape and channel inclination during condensation of R134a inside a single minichannel. This investigation is carried out by means of experimental tests and numerical simulations using the VOF method.
The experimental tests are performed during condensation inside two minichannels: the first one displays a circular cross section, with 0.96 mm diameter, while the second one presents a square cross section, having a 1.18 mm side length. Tests have been also performed with variable channel inclination, at 40°C saturation temperature and mass fluxes ranging between 100 and 800 kg $m^{-2}s^{-1}$.
Simulations of condensation of R134a are made in steady-state conditions at the same mass fluxes as in the experiments. The VOF (Volume Of Fluid) method is used to track the vapour-liquid interface and the effects of interfacial shear stress, gravity and surface tension are taken into account. Uniform wall and vapour-liquid interface temperatures are fixed as boundary conditions.
The simulations are compared to the experimental results and then used to understand more about the physical phenomenon and the effect of channel shape and channel inclination at variable operating conditions.

Break-up conditions of a flapping liquid jet

Antoine DELON, Jean-Philippe MATAS and Alain CARTELLIER
mel: first name.last name@legi.grenoble-inp.fr
LEGI CNRS/UJF/Grenoble-INP, BP53, 38041 Grenoble, France

In air assisted atomization, small droplets arise from the stripping of the liquid jet (or film) by the fast gas stream [1,2,3,4,5]. Yet, the incoming liquid is never fully atomized by the stripping process alone. Instead, the remaining jet experiences a flapping instability [2], similar to the instability observed on liquid sheet configurations [6]: the resulting large scale structures (see fig.1a) break into large liquid lumps at some distance downstream the injection. Little is known on the underlying mechanisms of this instability, and on the characteristics of the large drops it produces, though these large drops most probably control flame extent in combustion devices.

We therefore examine the break-up conditions as a function of two control parameters, namely the liquid and gas exit velocities. The break-up distance, given as the longitudinal extent of the liquid structure still connected to injection, happens to smoothly decrease with $M=\rho_G U_G^2/\rho_L U_L^2$ (fig.1c). We also investigate the break-up mode, by considering the time evolution of the number of daughter drops versus time, and we sought to relate the size of the very first drops with the wavelength. The flapping frequency is a constant for given control parameters [2] but the wavelength increases with the distance to the injector. This behavior arises from the acceleration experienced by the large scale structures as they are convected downstream (fig.1b). This acceleration is found to be larger than the one due to gravity, indicating that the co-flowing air stream plays an active role in the spatial evolution of these structures.

According to available observations, it happens that the size of the largest detached drop is directly controlled by the wavelength, and besides, that the number of daughter inclusions is a smooth function of the gas and liquid velocities. These findings will be discussed in more detail during the presentation.

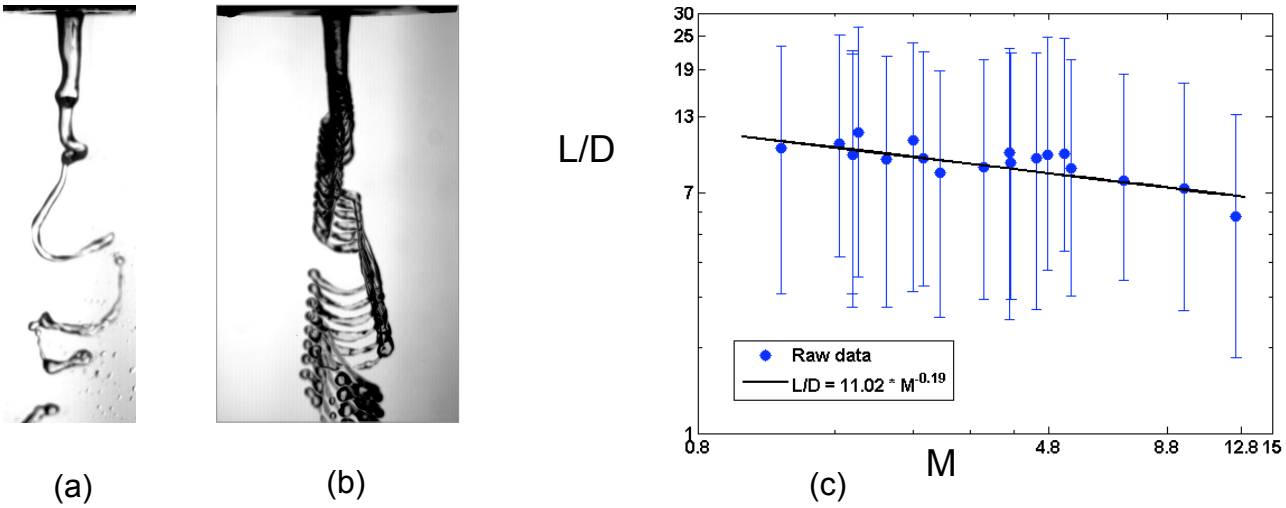


Fig.1*: (a): large-scale structures; (b): Superposition of time sequence; (c): Evolution of the break-up distance with M. Error bars show the minimum and maximum value of L/D observed for a given M.*

References:

[1] J.-P. Matas, S. Marty and A. Cartellier, "Experimental and analytical study of the shear instability of a gas-liquid mixing layer", *Phys. Fluids*, **23** 094112 (2011).
[2] J.-P. Matas and A. Cartellier, "Flapping instability of a liquid jet", Proc. 3rd INCA colloquium, ONERA, Toulouse, France, November 17-18, 2011.
[3] P. Marmottant and E. Villermaux, "On spray formation", *J. Fluid Mech.*, **498**, 73-111 (2004)
[4] M. Hong, A. Cartellier and E.J. Hopfinger, "Atomization and mixing in coaxial injection", Proc. 4th Int. Conf. on Launcher Technology, Liège Belgique (2002).
[5] F. Ben Rayana, A. Cartellier and E. Hopfinger, "Assisted atomization of a liquid layer: investigation of the parameters affecting the mean drop size prediction", CD Proc. ICLASS 2006, Kyoto Japan.
[6] A. Lozano, F. Barreras., C. Siegler and D. Löw, "The effects of sheet thickness on the oscillation of an air-blasted liquid sheet", *Experiments in Fluids*, **39**, 127-139 (2005).

50th European-Japanese Two-Phase Flow Group Meeting, Udine, Italy

NUMERICAL SIMULATION OF BUBBLE GROWTH UNDER THE ACTION OF ELECTRIC FORCES

Paolo Di Marco*, Ryo Kurimoto° Giacomo Saccone* and Akio Tomiyama°

* Department of Energy and Systems Engineering, University of Pisa, Pisa, Italy

° Graduate School of Engineering, Kobe University, Japan

p.dimarco@ing,unipi.it, tomiyama@mech.kobe-u.ac.jp

The classical problem of growth and detachment of a gas bubble from an orifice is revisited in this study to achieve a better comprehension of the role played by additional electric forces acting on the bubble. In particular, numerical simulation of the bubble growth was performed and compared with the bubble shape obtained by the experiments.
The experimental apparatus (Fig.1) consisted essentially in an orifice (0.30 mm in diameter) drilled in a flat stainless steel plate submerged in the test fluid (FC-72). A dedicated gas injection system, made up by a small-volume syringe operated by a micrometric screw, allowed to create slowly growing or even static bubbles of any desired volume, up to the detachment one. An electric field could be imposed by applying 20 kV to a washer-shaped electrode, centered on the orifice and laid parallel to the surface at 6 mm distance. Data were acquired via a high resolution video camera, equipped with a microscopic lens, achieving a resolution of 2.6 micrometer per pixel, and were digitized and processed via a dedicated software, implemented in Matlab.
Numerical simulations were carried out using an interface tracking method. The interface tracking is based on a level set method with a volume correction scheme. The ghost fluid method is used to evaluate surface tension force. The electric field is obtained by solving the equation of Gauss' law using a standard difference method. Two-dimensional cylindrical coordinate systems were used and 15 computational cells were assigned for the orifice radius to accurately predict air injection from the orifice. Figure 2 shows an example of comparison between measured and predicted shapes of the bubbles attached to the orifice. Excellent agreement was obtained for bubbles without the electric field. Though slight differences were observed for bubbles with the electric field, bubble elongation caused by the electric force was well predicted.

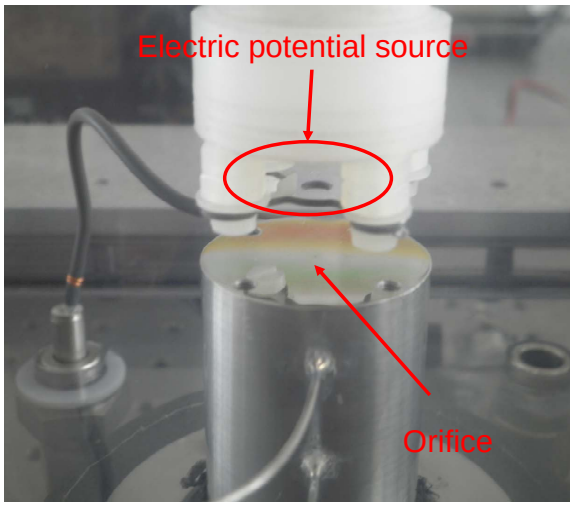


Fig.1 – Experimental apparatus

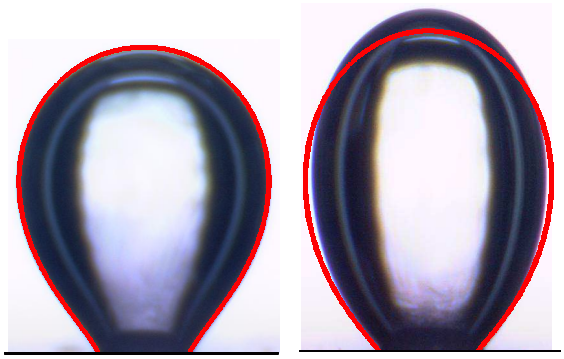


Fig.2 Comparison of bubble shapes (red line is numerical prediction)

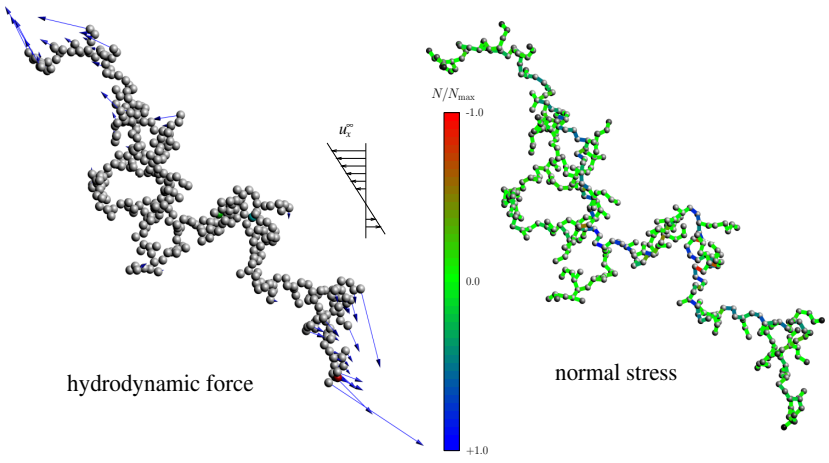
Breakage and restructuring of agglomerates in shear and elongational flows

Andrea Gastaldi, Lucas Sanchez Fellay, Marco Vanni
Dipartimento di Scienza Applicata e Tecnologia, Politecnico di Torino, Corso Duca degli Abruzzi 24, 10129, Torino, Italy

The hydrodynamic dispersion of colloidal aggregates is important in various applications, as it influences the rheological properties of suspensions and plays significant roles in food, polymer and pharmaceutical processing, nanoparticle synthesis, environmental chemistry. The outcome of such processes depends strongly on the ability of the fluid to fragment the aggregates into smaller entities. In fact the breakage of an aggregate in a viscous system occurs when the stresses induced in the aggregate by the flow of the external fluid prevail over the cohesive force in some point of the cluster.

The process is characterized by phenomena of different scales: inter-monomer adhesion forces operate at nanometer distances, while fluid dynamic interactions, including lubrication forces, internal permeation and external flow, range in scale from few nanometers to the size of the agglomerate. What links such phenomena is the redistribution of the stresses in the internal structure of the cluster and in fact it is the redistribution that determines the global effect, enhancing or damping the load on the inter-monomer bonds.

In this study the distribution of stresses in rigid colloidal aggregates under different types of linear fluid flow fields was investigated numerically for aggregates with fractal dimensions ranging from 1.7 to 2.3. Stokesian dynamics was used to calculate the hydrodynamic force on each monomer, while the internal inter-monomer interactions were calculated by applying the methods of structural mechanics. Although the hydrodynamic forces act mainly on the periphery of the clusters, their filamentous structure propagates and accumulates internal stresses towards the inner region of the aggregates, where consequently the most loaded intermonomer bonds are located. The spatial stress distribution, when scaled by the proper power of the radius of gyration, is nearly independent of aggregate size and fractal dimension. This feature has made it possible to identify the most probable locations of bond failure in the structure and to estimate the relationship between flow strength and particle size for the occurrence of restructuring and of breakage.



Inlet condition effects on gas-liquid flow in mini-tubes

J. Gregorc and I. Žun
Laboratory for Fluid Dynamics and Thermodynamics
Faculty of Mechanical Engineering
University of Ljubljana, Slovenia

Presentation summarizes the current status of our experimental and numerical research on inlet conditions of gas-liquid flow in mini-tubes. The results of our previous work on the separation of gas and liquid in mini manifold showed strong impact of inlet conditions on formation as well as behavior of gas-liquid interface further downstream the header. Insufficient knowledge of this mechanism can cause major problems in numerical simulation of relevant flow regimes.

A new experimental test loop with interchangeable glass mini-tubes has been constructed to assess the effects of inlet conditions on interfacial structures development in horizontal tube. A number of tubes with inner diameters D=1.2mm, 1.00mm, 0.8mm, 0.6mm, and 0.4mm and length of 210mm which corresponds to L/D from 175 to 525 have been tested. Air and water were mixed either in T-junction, cross-junction, or porous media mixer before entering the mini tube. The following flow patterns are going to be discussed in details for the 1.2 mm mini-tube: Slug flow, Bubbly flow, BTS flow, Churn flow and Semi-annular flow with gas and liquid superficial velocities ranging from 0.2m/s to 11m/s and 0.25m/s to 3m/s, respectively. Arrival and resident times of bubbly structures reveal, for example, that beyond 20D, the flow pattern gets stabilized and neither bubble coalescence nor interfacial brake up cannot be expected anymore further downstream. It can be argued that the flow patterns were predetermined by initial conditions.

The results of numerical simulations of Bubbly flow, BTS flow and Slug flow are presented. VOF method proved to be powerful enough to predict the occurrence and distribution of gas structures along the mini tube even when one would mimic the mixing process of two fluid (air and water) streams at the tube inlet. Mixing itself results in realistic two-phase flow patterns that can be validated by emergent parameters on macro- and meso-scales. However, instabilities at gas liquid interface are caused during the mixing process which has not been resolved yet. In search for better numerical method, VOF+LS was introduced and the results are promising. The penalty of using VOF+LS algorithm is in significant increase in computational time.

Measurement of hold-up and bubble size inside a porous structure by planar laser induced fluorescence using index matching

R.Häfeli, P. Rudolf von Rohr
ETH Zürich, Institute of Process Engineering, Sonneggstrasse 3, 8051 Zurich, Switzerland; tel. +41 44 632 7941, e-mail: haefeli@ipe.mavt.ethz.ch

Optical methods for the study of single-phase flow through porous media are enabled by using refractive index matching techniques. Wiederseiner et al. [1] give a comprehensive overview of index matching which is mostly used for the study of flow through porous structures like packed beds. However in industrial applications very often a flow of two phases through a porous structure is encountered. Optical investigation in this case requires the matching of the refractive indices of two liquids and a solid.

We apply planar laser induced fluorescence (PLIF) in a liquid-liquid two-phase flow through a porous structure. This system allows to measure hold-up and bubble size.

Figure : LIF image showing the cross-section of one bubble illuminated by NdYAG laser. Continuous phase: NaI and ZnI in water, bubble: anisol traced by RhodamineB.

By that the mechanisms of bubble break-up and coagulation can be investigated. Also the identification of flow regimes inside the porous structure is of interest for the design of chemical reactors where one goal is to achieve a narrow residence time distribution.

We yet performed particle image velocimetry (PIV) measurements in single-phase flow through porous media by using index matching. We compare measurement results to LES simulations. In a further step the system will be used for the measurement of the velocity field in the two phases inside the porous structure by PIV.

References

[1] Wiederseiner S., Andreini N., Epely-Chauvin G., Ancey C. (2011) “Refractive-index and density matching in concentrated particle suspensions: a review” Exp Fluids, **50**, 1183-1206

50th European Two-Phase Flow Group Meeting, Udine, 16th - 18th May 2012

Effect of Sorption Phenomena on the Pressure Build-Up in Twin-Screw Multiphase Pumps

F. Hatesuer[1], J. Lewerenz[2], M. Reichwage[2], A. Luke[1]

[1] Institute of Thermodynamics, University of Kassel, Germany
[2] Joh. Heinr. Bornemann GmbH, Obernkirchen, Germany

Twin-screw multiphase pumps (MPPs) are used in crude oil and natural gas production processes for long distance conveyance, especially in offshore and remote onshore applications. Multiphase mixtures of oil, accompanying fluids, associated gas and even solid particles are boosted in a single flowline from the wells to central processing facilities. Like this, separation units close to the widely scattered wellheads are no longer necessary. Flaring or even venting of large volumes of associated gas can be avoided and greenhouse gas emissions are reduced. This contributes to lower production costs and to increase the yield of resource. Hence, even conventionally depleted or difficult to access offshore reservoirs can become worth to be further exploited.

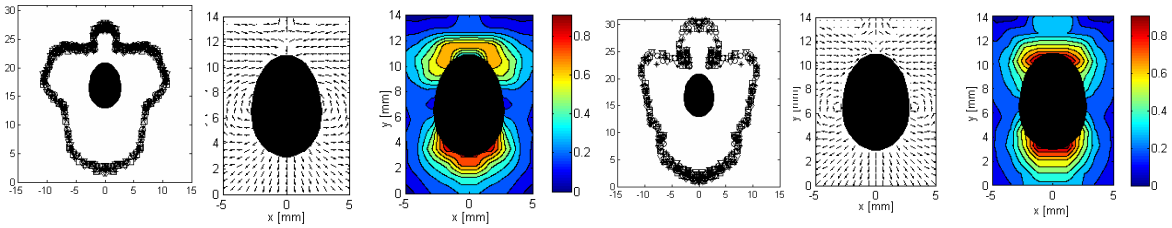
MPPs are known as continuous conveyors. Inside of them, two intermeshing screws are enclosed by a casing and form conveying chambers. By counter-rotation, they move the multiphase mixture towards the outlet. Adjacent chambers are connected by gaps enabling the fluid to flow back from one chamber to another and finally to appear in the inlet as a loss flow. These gap flows determine the pressure build-up along the twin-screws and the conveying characteristic of the MPP. It is known to be affected by the boosted pressure difference, the rotational speed of the screws and the multiphase gas fraction. However, recent experiments including oil and variable gases show, that the physical properties of the fluids gain in importance. Among others, the impact of the pressure-dependent solubility of gases in liquids has to be regarded.

Therefore, the delivered volumes and the pressure build-up along the twin-screws are measured for various fluids in a wide operating range. The resulting conveying characteristic is discussed and the impact of sorption phenomena is demonstrated. When conveying two-phase mixtures with high sorption capability an impact of gas solubility is clearly discernible. In particular, for low and medium gas volume flow fractions and high rotational speed of the screws the pressure build-up changes.

A PIV study of sedimentation of drops in yield-stress Fluids

Yulia Holenberg[1)], Uri Shavit[2)], Alexander Liberzon[3)], Olga Lavrenteva[1)] and Avinoam Nir[1)]
1) Department of Chemical Engineering, Technion, Haifa 32000, Israel
2) Department of Civil and Environmental Engineering, Technion, Haifa 32000, Israel
3) School of Mechanical Engineering, Faculty of Engineering, Tel Aviv University, Tel Aviv 69978, Israel

Drops settle in visco-plastic material if the local stress near their interfaces exceeds a critical value. In such cases the drops are surrounded by a yielded region. The shape of this region and the dynamics of the flow in it during a steady sedimentation under gravity of drops are the subjects of this report. We use optical observations, Particle Tracking Velocimetry (PTV) and Particle Image Velocimetry (PIV) methods to establish settling speed, the boundary and shape of the yielded region and the flow direction and intensity within it, respectively (see Fig. 1). Low concentrated aqueous gel of Carbopol 940 (0.07% w/w) was used as the yield stress material. Newtonian drops (R ~ 2.8mm) of various densities having similar viscosity and interfacial tensions were used. Heavier and faster drops move with velocity of O(0.1 mm/s) when isolated, while lighter ones move an order of magnitude slower, with velocity of drops of O(0.01 mm/s). The reported velocity data are results averaged for 5 experimental runs, with at least 40 image pairs in each run.

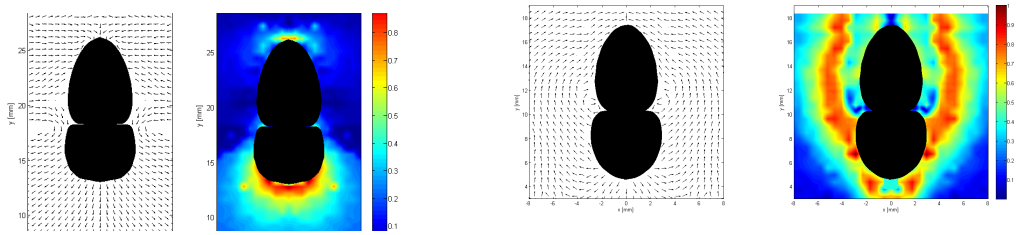


(a) (b)

Fig.1. The boundary of and flow direction and intensity within the yielded region around a settling drop (a) Slow drop (b) Fast drop.

For the motion in unbounded domain, it was demonstrated that the yielded region extends more vertically than horizontally. The vertical extent does not significantly change with the drop speed, but the horizontal dimension at the equator expands considerably for drops with higher velocity. When the drops settle in the proximity of a vertical solid wall their behavior is contrary to what is anticipated when they settle in a Newtonian fluid or a visco-elastic domain. Their settling speed is augmented rather than retarded, and they drift slowly toward the wall. The increase in settling speed can be attributed to the dynamic formation of a thin clear solvent layer providing an effective wall slip.

We have also used PIV to study the axisymmetric dynamics of two equal-size drops trailing each other and eventually coalescing. When the trailing drop reaches the leading one, the drop doublet moves ensemble a relatively long time passing before the coalescence begins, as is shown in Fig. 2(a). At the beginning of the coalescence process the flow field around the drop doublet changes drastically. The drops elongate and the flow field becomes more intensive (see Fig. 1(b)).



(a) (b)

Fig.2. Flow field (left subplots) and flow intensity (right subplots) around a coalescing doublet system during its quasi-static motion, when the trailing drop reached the leading one, but coalescence did not begin (a) and immediately after coalescence starts (b) The time interval between the two cases is 0.2 seconds.

The interaction and coalescence of two large bubbles rising in tandem in a thin-gap cell

Sander G. Huisman, Véronique Roig and Patricia Ern

Institut de Mécanique des Fluides de Toulouse,
Allée du Prof. Camille Soula, F-31400 Toulouse, France

We investigated the relative motion and deformation of two large bubbles released consecutively in a quiescent liquid confined in a thin-gap cell (gap size e). The following different stages of interaction of the bubbles were identified and characterized using high-speed imaging: *(i)* the trailing bubble is accelerated by the long-range wake of the primary bubble, *(ii)* the second bubble enters the recirculating wake of the leading bubble, undergoes horizontal contraction and decelerates, *(iii)* the bubbles adapt their shape to each other, in particular their curvatures are significantly modified, *(iv)* the shape of both bubbles together resembles a single bubble and the liquid between the bubbles is squeezed out, and *(v)* the bubbles merge: the liquid film breaks and is drained at a constant velocity. In all the cases investigated here, the velocity of the leading bubble is unaltered during the whole interaction process, whereas the trailing bubble, even if smaller, catches up with the leading bubble. We have shown that this acceleration is related to the wake of the leading bubble: the evolution of the velocity of the trailing bubble with the separation distance follows a unique master curve, provided the velocity is normalized with the velocity of the leading bubble V_1 and the separation distance with the viscous lengthscale associated to the leading bubble $l_\nu = V_1 e^2/\nu$, where ν is the liquid kinematic viscosity[1].

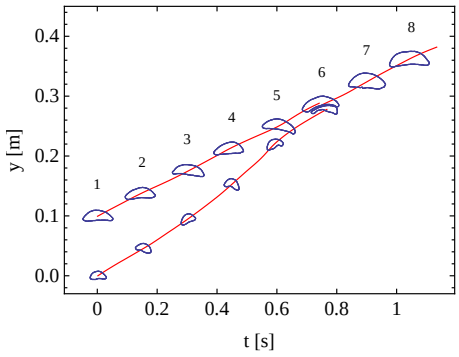


Figure 1: Overview of the trajectories of two large bubbles ending up in coalescence. The trailing bubble is entrained in the wake of the leading bubble (1-5), the bubbles adapt their shapes (6), and coalesce forming a single bubble (7, 8).

[1]S. Huisman, P. Ern and V. Roig (2012) *Phys. Rev. E*, in press

Particle accumulation structures (PAS) in thermocapillary liquid bridges

H. C. Kuhlmann and F. H. Muldoon

Vienna University of Technology, Resselgasse 3, 1040 Vienna, Austria

The surface-tension-driven flow in a liquid bridge is a widely-used model for the floating-zone crystal-growth process. It has evolved to become a paradigm for thermocapillary convection and has been studied quite extensively [1]. A volume of liquid, held in place by surface tension, is suspended between two coaxial cylindrical rods in vertical orientation. With both rods at different temperatures, variations of the surface tension $\sigma(T)$ result in thermocapillary convection. Under certain flow conditions, particles suspended in the liquid can de-mix on a very rapid time scale resulting in dynamic particle accumulation structures called PAS [2]. In PAS, the particles typically align along a closed spiral which rotates about the axis of the liquid bridge. Figure 1a shows a projection of such a closed particle string.

We present a theoretical framework to understand PAS. It is based on the topology of the thermocapillary flow in form of an azimuthally traveling hydrothermal wave (figure 1b). The motion of perfect tracers in this incompressible flow is governed by Hamiltonian dynamics. Under dissipative perturbations elliptic orbits of the Hamiltonian system transform to limit cycles which are interpreted as PAS. We demonstrate that particle–free-surface interactions can lead to limit cycles which are always stable [3]. This is in contrast to the slow inertial dynamics of small particles. Moreover, inertia can lead to either stable or unstable limit cycles.

Support from ESA under contract number 4000103003 is gratefully acknowledged.

[1] H. C. Kuhlmann. *Thermocapillary Convection in Models of Crystal Growth*, volume 152 of *Springer Tracts in Modern Physics*. Springer, Berlin, Heidelberg, 1999.

[2] D. Schwabe, A. I. Mizev, M. Udhayasankar, and S. Tanaka. Formation of dynamic particle accumulation structures in oscillatory thermocapillary flow in liquid bridges. *Phys. Fluids*, 19:072102–1–072102–18, 2007.

[3] Ernst Hofmann and Hendrik C. Kuhlmann. Particle accumulation on periodic orbits by repeated free surface collisions. *Phys. Fluids*, 23:0721106–1–0721106–14, 2011.

[4] H. C. Kuhlmann and F. H. Muldoon. Particle-accumulation structures in periodic free-surface flows: Inertia versus surface collisions. *Phys. Rev.* E, to appear, 2012.

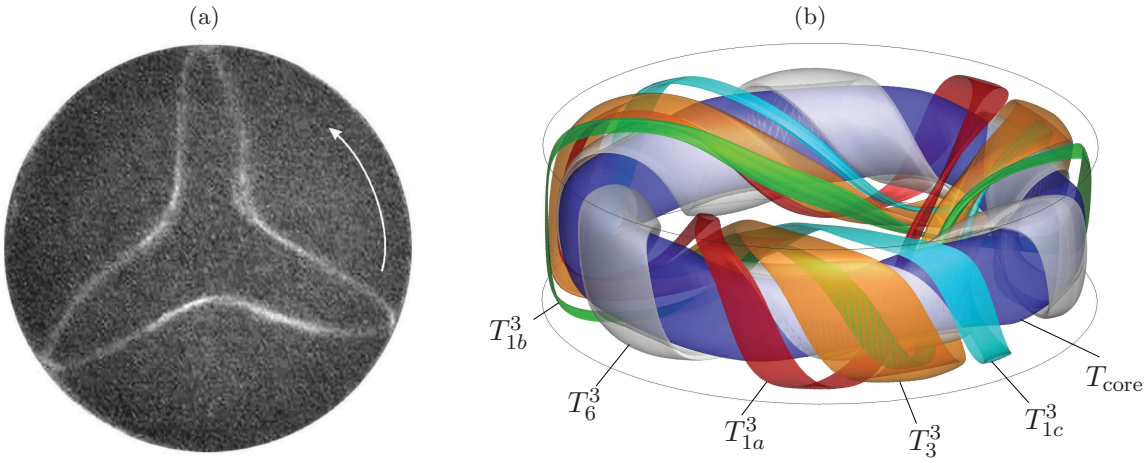


Figure 1: (a) An example for PAS with period $m = 3$ viewed axially through a transparent heating rod [2]. (b) Invariant stream tubes in a model flow in the frame of reference co-rotating with the hydrothermal wave [4].

50th European Two-Phase Flows Group Meeting 2012

Quantitative data from ultra-fast electron beam X-ray computer tomography measurements

D. Lucas, D. Hoppe, M. Banowski, U. Hampel, F. Fischer, U. Sprewitz, M. Beyer

Helmholtz-Zentrum Dresden-Rossendorf (HZDR)

Abstract

The ultra-fast electron beam X-ray computer tomography was developed during last years at HZDR and turned out to be a suitable measuring technique to get new insights into two-phase flow structures (Barthel et al., 2011, Lucas et al., 2011). The aim of new experiments done at a vertical pipe with an inner diameter of about 54 mm and a length of 4 m (usable for experiments) was to obtain quantitative data suitable for CFD-code development and validation. The pipe was built up using Titanium. This allows a wall thickness of only 1.6 mm which is enough for steam-water experiments which will be carried out at a pressure up to 6.5 MPa. To generate the X-rays an electron beam is focused on a circular two-step target. This electron beam is deflected very fast and moves over the target resulting in a fast moving X-ray source. The X-rays pass through the object to be investigated and are registered by an arc of detectors. A tomographic reconstruction algorithm provides images of the attenuation distribution in the cross section of the object which is interpreted as density distribution. This allows the determination of the gas- liquid interfacial structure inside the considered object. Measurements were done with a measuring frequency of 2500 images/s and measuring time was 10 s. Air-water and steam-water flows were investigated for different combinations of gas and liquid flow rates and different distances from the gas injection.
The tomographic reconstruction provides 2 3D matrices of gray-scale values since the two-step target allows measurement in two planes with an axial distance of about 11 mm. Cross-correlation algorithms are applied to obtain the information on local gas velocities. Due to noise and artifacts a binarisation of the data seems to be necessary. Using a simple threshold method may cause small bubble to go missing. For this reason another method basing on gradients was developed. The presentations discusses this method and the present status of this new measuring technology in respect to quality and uncertainties of the quantitative data obtained as e.g. local void fraction and bubbles sizes.

This work is carried out in the frame of a current research project funded by the German Federal Ministry of Economics and Technology, project number 150 1411.

Barthel et al. (2011), Ultra-fast electron beam X-ray CT for two phase flow phenomena, Proceedings of the 14th International Topical Meeting on Nuclear Reactor Thermalhydraulics (NURETH-14), CD-ROM

Lucas et al. (2011), Some new insights into gas-liquid upward pipe flows by ultra fast X-ray Computer Tomography, 49th European Two-Phase Flows Group Meeting, Tel-Aviv, Israel, May 29 – June 2, 2011

ORIENTATION OF NON-SPHERICAL PARTICLES WITH INERTIA IN SHEAR AND EXTENSIONAL FLOW

Fredrik Lundell*[a)] & Tomas Rosén**

*Linné FLOW Centre and Wallenberg Wood Science Center, KTH Mechanics, Royal Institute of Technology, 100 44 Stockholm, Sweden

**Linné FLOW Centre, KTH Mechanics, Royal Institute of Technology, 100 44 Stockholm, Sweden

The rotation of ellipsoids in canonical flows is a key ingredient for the understanding and modelling of multiphase flows with non-spherical particles. Here, the fundamental problem of the rotation of an ellipsoid in linear shear and extension flow will be studied. The dynamics of such an ellipsoid in flow are governed by the two non-dimensional groups

$$Re = \frac{\dot{\varepsilon} l^2}{\nu}, \qquad St = \frac{\rho_p \dot{\varepsilon} l^2}{\rho_f \nu}, \tag{1}$$

where $\dot{\varepsilon}$ is the relevant flow-velocity gradient, $2l$ the length of the major axis of the particle, ν the kinematic viscocity of the fluid, ρ_p and ρ_f the densities of the particle and fluid, respectively. The groups are called the Reynolds and Stokes number, respectively. They measure the effects of fluid (Re) and particle (St) inertia.

The flow around an ellipsoid for the case of $Re = 0$ (creeping flow) has been solved analytically[1], providing the torques on the particle in any linear flow, any orientation and for any particle rotation rate. These results were used to obtain the motion for light particles ($St = 0$), the so called Jeffery orbits. In spite of the fact that they have been known for almost 90 years, there are still new findings on particle dynamics being made based on these torques (e.g. [2,3]). For finite Re, one must rely to numerical simulations and this is also a very active research area[4].

In this work, both analytical[1] and numerical[4] methods are used to investigate the motion of ellipsoids in shear and extension. In particular, the effects of fluid inertia on the recently presented transition in rotation rate (for creeping flow) for high particle inertia[2] will be studied at finite Reynolds number and the effects of particle inertia in extensional flow will be presented.

An example of results for a heavy particle in creeping extensional flow (i.e. $Re = 0$ and different levels of St) will be presented in some detail. As St is increased from zero, a critical value is reached at which the system is critically damped. We denote this value St_d and make the observation that at St_d, φ will approach 0 as fast as possible. For $St > St_d$, the particle will exhibit a dampened oscillation and for $St < St_d$, a particle released from rest will approach $\varphi = 0$ monotonically, albeit slower than for $St = St_d$. Figure 1 (b) shows time traces of φ for an ellipsoid ($k_b = k_c = 0.2$) released from rest at $\varphi = \pi/3$ is shown. The findings above are verified. For $St = St_d$ (red), a fast approach to φ occurs, for lower St (black), the approach is finished somewhat later and for higher St (green and blue), the particle exhibits a damped oscillation with decreased damping and frequency as St is increased.

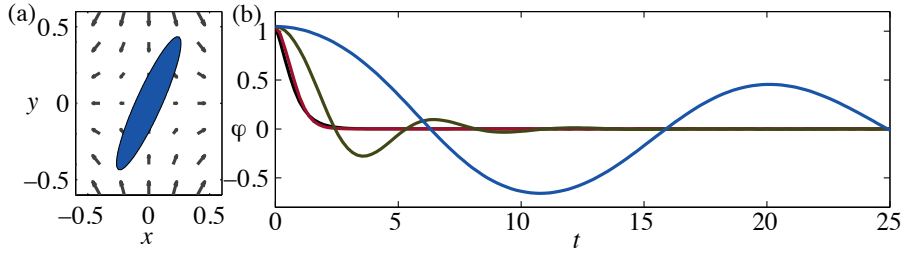


Figure 1. (a): Demonstration flow case: an ellipsoid ($k_b = k_c = 0.2$ in extensional flow. (b): Orientation φ as a function of time for $\varphi(0) = \pi/3$, the orientation shown in (a). $St/St_d = 0.1$ (black), 1 (red), 10 (green) and 100 (blue). Time is scaled with $\dot{\varepsilon}$.

References

[1] Jeffery, G.B.: The motion of ellipsoidal particels immersed in a viscous fluid. *Proc. Roy. Soc. A* ***102****(715):161–179*, 1922

[2] Lundell, F., Carlsson, A.: Heavy ellipsoids in creeping shear flow: Transitions of the particle rotation rate and orbit shape. *Phys. Rev. E* **81**:016323, 2010

[3] Lundell, F.: The effect of particle inertia on triaxial ellipsoids in creeping shear: From drift towards chaos to a single periodic solution. *Phys. Fluids* **23**:011704, 2011

[4] Wu, J., Aidun, C.K.: Simulating 3D deformable particle suspensions using lattice Boltzmann method with discrete external boundary force *Int. J. Num. Meth. Fluids* **62**(7):765–783

[a)] Corresponding author. E-mail: fredrik@mech.kth.se.

50th European Two Phase Flow Group Meeting 2012
Udine, Italy, 16-18 May 2012

NUMERICAL SIMULATION OF THE FLOW BOILING OF ELONGATED BUBBLES WITHIN MICROCHANNELS

M. Magnini*, B. Pulvirenti, J.R. Thome
* Author for correspondence
Department of Energy, Nuclear and Environmental Control Engineering
University of Bologna
Bologna, 40126, Italy
E-mail: mirco.magnini@unibo.it

ABSTRACT

We present our results on the CFD simulation of evaporating bubbles within horizontal circular microchannels. A vapor bubble at saturation conditions is initialized at the upstream of the channel and a saturated liquid inflow moves the bubble downstream. A constant heat flux applied at the channel's wall makes the adjacent liquid superheated such that the vapor bubble in contact with it evaporates sinking locally the sensible heat of the liquid.

The numerical framework employed is the commercial CFD solver Ansys Fluent version 12 and the Volume of Fluid method [1] is chosen for the mathematical and numerical treatment of the gas-liquid interface. In order to save computational time, an axisymmetrical formulation of the problem is adopted. Interfacial effects such as surface tension and evaporation govern the flow configuration simulated, therefore the accurate estimation of their magnitude is of main importance. We improved Ansys Fluent estimation of the surface tension force by replacing its default interface topology reconstruction algorithm with an Height Function algorithm [2]. The Height Function algorithm derives, from VOF-defined volume fractions, a smooth scalar field representing the local elevation of the interface above a reference axis. The high accuracy of the local interface norm vector and curvature computed by derivation of such scalar field, makes the Height Function method the most accurate algorithm available for multiphase CFD techniques based on interface capturing. No evaporation model is present as default in the software. We implemented an evaporation model which computes the local rates of mass and energy exchange at the interface according to the approach of Schrage [3]. Schrage assumed thermodynamic equilibrium within each phase but he supposed an interfacial jump in the temperature to exist. Thus, rather than fixing the interface at the saturation temperature, condition that may be untrue in the microscale, the model allows the interface temperature to deviate from the saturation condition and the local evaporation rate is computed proportional to such interface superheating. Both the models are introduced within the software as ad-hoc self-implemented user-defined subroutines.

The capability of the improved software to simulate the flow of elongated bubbles is assessed through several benchmarks and comparison with published correlations. The numerical code is able to reproduce accurately the thickening effect of inertial and viscous forces on the liquid film surrounding the bubble, as well as to capture the round-off effect of the surface tension on the bubble shape. The simulation of evaporating bubbles gives a detailed insight on the hydrodynamics and the wall heat transfer of the flow, allowing to investigate the local patterns which enhance the heat transfer with respect to a single phase flow. In particular, the thin film evaporation and the transient heat convection across the wall thermal boundary layer disturbed by the bubble transit are well captured by the numerical framework. The mutual influence of multiple bubbles evaporating in sequence in a slug flow is reproduced, allowing to establish the different flow conditions which leading and trailing bubbles are subject to.

REFERENCES

[1] C. W. Hirt and B. D. Nichols. Volume of fluid (VOF) method for the dynamics of free boundaries. *Journal of Computational Physics*, 39:201-225, 1981.
[2] S. J. Cummins, M. M. Francois and D. B. Kothe. Estimating curvature from volume fractions. *Computers and structures*, 83:425-434, 2005.
[3] R. W. Schrage. A theoretical study of interphase mass transfer. Columbia University Press, New York, 1953.

50^{th} European Two-Phase Flow Group Meeting,
Udine (Italy) May 16-18, 2012.

Turbulence modulation and microbubble dynamics in vertical channel flow

Cristian Marchioli†, Dafne Molin‡ and Alfredo Soldati†

† *Dipartimento di Fluidodinamica, CISM, 33100 Udine, Italy*

‡ *Dipartimento di Ingegneria Meccanica e Industriale, Università di Brescia, 25123 Brescia, Italy*

Abstract

In this paper we examine the mutual interactions between microbubbles and turbulence in vertical channel flow. An Eulerian-Lagrangian approach based on pseudo-spectral direct numerical simulation is used: bubbles are momentum coupled with the fluid and are treated as pointwise spheres subject to gravity, drag, added mass, pressure gradient, Basset and lift forces. Two different flow configurations (upward and downward channel flow of water at shear Reynolds number $Re_\tau = 150$) and four different bubble diameters (110, 165, 220, and 330 μm) are considered, assuming that bubbles are non-deformable (i.e. small Eotvos number) and contaminated by surfactants (i.e. no-slip condition applies at bubble surface). Confirming previous knowledge (Giusti et al., 2005), we find macroscopically different bubble distribution in the two flow configurations, with lift segregating bubbles at the wall in upflow and preventing bubbles from reaching the near-wall region in downflow. Due to local momentum exchange with the carrier fluid and to the differences in bubble distribution, we also observe significant increase (resp. decrease) of both wall shear and liquid flowrate in upflow (resp. downflow). We propose a novel force scaling for the wall shear stress (based on a simple one-dimensional model derived from an overall force balance on the channel) to examine results in vertical turbulent bubbly flows, which can help to judge differences in the turbulence features due to bubble presence. By examining two-phase flow energy spectra, we show that bubbles in upflow determine an enhancement (resp. attenuation) of energy at small (resp. large) flow scales in the channel centerline, a feature already observed in homogeneous isotropic turbulence (Mazzitelli et al., 2009). Energy is always increased by bubbles near the wall. In downflow, no energy cross-over from large to small scales is observed and bubbles always drain energy from the fluid. Bubble-induced flow field modifications, in turn, alter significantly the dynamics of the bubbles and lead to different trends in preferential concentration and wall deposition. In this picture, a crucial role is played by the lift force, which is a delicate issue when accurate models of shear flows with bubbles are sought (Mazzitelli et al., 2003).
To emphasize the impact that the lift force model has on the simulations, we examine the wall-normal behavior of all forces exherted on bubbles. Regardless of the flow configuration, in the near-wall region lift dominates over all other unsteady forces acting on the bubbles and is counter-balanced only by drag. Because of the extra contribution due to wall effects

(Takemura and Magnaudet, 2003), lift changes sign from negative (directed to the wall) to positive (directed away from the wall) in upflow when bubbles with diameter larger than 220 μm are considered. For these bubbles, reversal of the lift force leads to uniform wall-normal distribution and negligible preferential concentration. Wall effects also produce a lift force reduction in downflow when bubble size is increased.

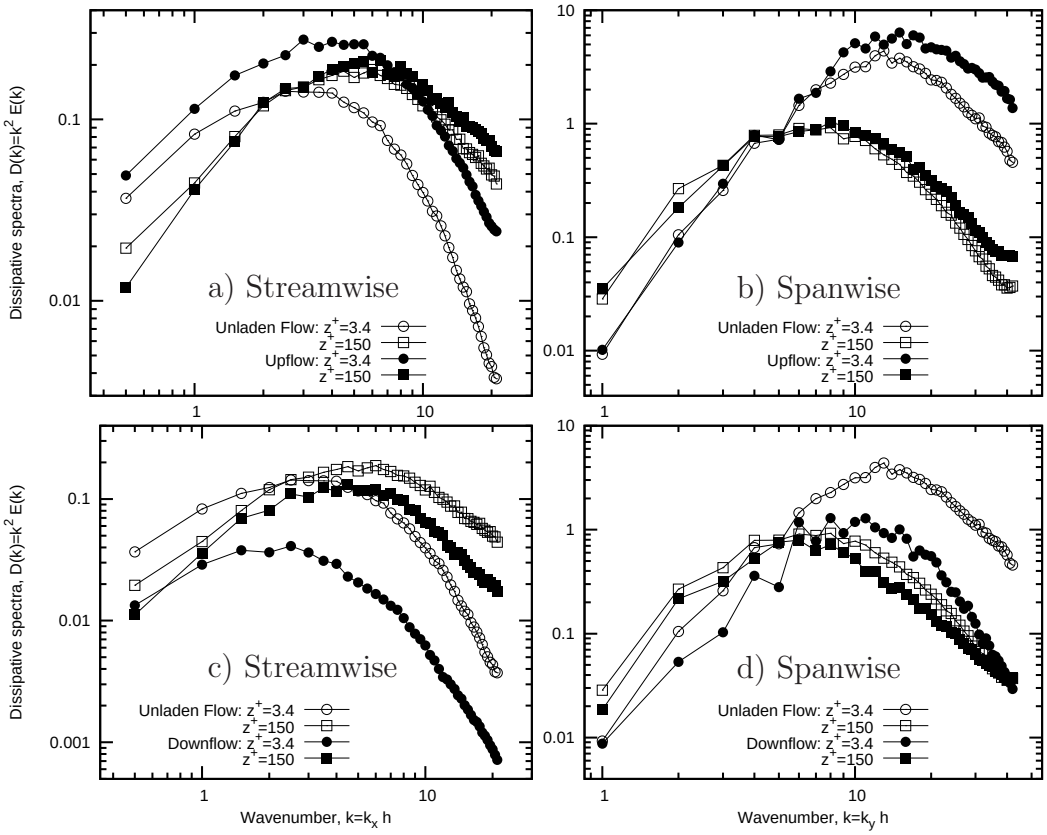


FIG. 1. 1D fluid dissipative spectra, $D(k)$, computed at $z^+ \simeq 3.4$ in the near-wall region (circles: ○, ●), and at $z^+ \simeq 150$ in the center of the channel (boxes: □, ■). Open symbols refer to unladen flow simulations, black symbols refer to two-way coupling simulations with the 220 μm bubbles. Panels: a-b) upflow; c-d) downflow; a), c) streamwise spectra; b), d) spanwise spectra.

References

1. A. Giusti, F. Lucci and A. Soldati, Influence of the lift force in direct numerical simulation of upward/downward turbulent channel flow laden with surfactant contaminated microbubbles. *Chem. Eng. Sci.*, **60** (2005), pp. 6176–6187.

2. F. Takemura and J. Magnaudet, The transverse force on clean and contaminated bubbles rising near a vertical wall at moderate Reynolds number. *J. Fluid Mech.*, **495** (2003), pp. 235–253.

3. I.M. Mazzitelli, D. Lohse and F. Toschi, On the relevance of the lift force in bubbly turbulence. *J. Fluid Mech.*, **488** (2003a), pp. 283–313.

4. I.M. Mazzitelli and D. Lohse, Evolution of energy in flow driven by rising bubbles. *Phys. Rev. E*, **79** (2009), pp. 1539–3755.

Small-scale dynamics and fluid-particle interaction around a cylinder placed over a porous seabed

M. Mattioli[1], M. Postacchini[2], S. Corvaro[2], A. Mancinelli[2], M. Brocchini[2]

1. Saipem s.p.a., Fano (PU), Italy (Matteo.Mattioli@saipem.com).
2. Università Politecnica delle Marche, Ancona, Italy (m.postacchini@univpm.it, s.corvaro@univpm.it, a.mancinelli@univpm.it, m.brocchini@univpm.it).

ABSTRACT

The interest in pipeline engineering has steadily increased over the last decades because of its importance in many fields of application, especially in the Oil & Gas industry. Hence, the scientific interest in the topic, e.g. studies referring to wave-induced forces on a cylinder near a plane boundary (Sarpkaya, 1977), to the scouring processes below an underwater pipeline (Sumer and Fredsøe, 2002), etc. The hydrodynamics concerning underwater pipelines laid on either rigid or erodible seabeds has been studied in depth. For example, some recent papers (e.g. Mattioli et al., 2012a,b) have given a detailed analysis of the velocity field, of the vorticity generation and of the turbulent structures that are generated close to a submerged cylinder.

In the present work, we describe some preliminary results of the experiments carried out in the wave flume of the Hydraulics Laboratory of the Università Politecnica delle Marche (Ancona). The aim is the study of the hydrodynamics induced by regular waves around a cylinder placed on a porous seabed. Two different seabed configurations have been tested, one made of gravel, the other made of plastic spheres. Both materials are arranged in the shape of several overlying layers (see fig. 1).

Fig. 1. Cylinder over the gravel seabed

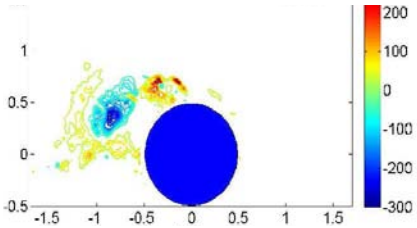


Fig. 2. Okubo-Weiss term close to the cylinder

The velocity field around the cylinder has been obtained using the Particle Tracking Velocimetry technique, an optical method that enables to follow the seeding particles (ground chili pepper, in the present case) that are transported by the fluid and whose motion is tracked. The particle motion enabled us to analyze in detail the main flow features like turbulent structures, vorticity and coherent vortices (see, for example, a map of the Okubo-Weiss parameter in fig.2), particle trajectories from/into the porous bed.

Previous studies of a cylinder-free similar configuration, undertaken by Corvaro et al. (2010), led to rather different vorticity and turbulence patterns. Comparison of such results with the new ones and inspection of the particle motion for the present ventilated boundary layer will be illustrated at the Conference.

REFERENCES

Corvaro S., Mancinelli A., Brocchini M., Seta E., Lorenzoni C. (2010). "On the wave damping due to a permeable seabed", *Coastal Engng*, 57(11-12), 1029-1041.

Mattioli M., Alsina J.M., Mancinelli A., Miozzi M., Brocchini M. (2012a). "Experimental investigation of the nearbed dynamics around a submarine pipeline laying on different types of seabed: the interaction between turbulent structures and particles", *Adv. Water Res.* (under peer review).

Mattioli M., Mancinelli A., Brocchini M. (2012b). "Experimental investigation of the wave-induced flow around a surface-touching cylinder", *J. Fluids Struct.* (under peer review).

Sarpkaya T. (1976). "Forces on cylinders near a plane boundary in a sinusoidally oscillating fluid", *J. Fluids Engng*, 98, 499-505.

Sumer B.M., Fredsøe J. (2002). *The mechanics of scour in the Marine Environment*, World Scientific, 535pp.

EHD Augmentation and Control of Heat Transfer Rates during Convective Boiling

Gerard McGranaghan, Anthony J. Robinson*
Department of Mechanical & Manufacturing Engineering, Trinity College Dublin, Ireland
*arobins@tcd.ie, +353 85 156 3366 (ph), +353 1 679 5554 (fax)

Summary
The purpose of this work is to investigate the influence of electrohydrodynamic (EHD) on the heat transfer rates associated with flow boiling of the dielectric fluid HFE7000. In particular, this work focuses on quantifying the level of overall enhancement which is achievable with EHD in the context of the additional heat extracted by the working fluid in the heat exchanger compared with the field-free case and the additional power penalties required to do so. For a fixed inlet refrigerant condition the field-free case provides the baseline overall heat transfer which increases with applied AC voltage up to the maximum safe voltage which can be applied in this test facility. An uncomplicated PID control system has been utilized in conjunction with a high voltage amplifier in order to accurately control the heat transfer rate of the heat exchanger. To our knowledge this is the first solid state control system of this type for a two phase heat exchanger.

Overview
This work investigates the influence of EHD forces on the flow and heat transfer during convective boiling. A unique tube-and-shell heat exchanger has been constructed with heated water flowing on the shell side and a saturated mixture of refrigerant flowing within the tube side. The heat exchanger is novel in that it allows full visual access to the inner tube whilst being both thermally and electrically conductive. In this work the influence of AC EHD on the flow regimes and subsequent heat transfer is investigated for fixed inlet refrigerant mass flux of G=100 kg/m^{2}s, low inlet quality of x~3% and constant inlet hot water inlet temperature. For these inlet conditions the applied voltage across a concentric inner electrode and the outer wall of the tube was varied between 0 kV and 10 kV at 60Hz.

Figure 1 illustrates how the application of EHD can cause unique flow patterns to occur in this heat exchanger. With regards to heat transfer, the unique flow regimes cause the overall heat transfer to the refrigerant to increase. For the highest voltage level tested the heat transfer increased by ~1.5 fold over the field free case. Even still, there is additional power required to increase the heat transfer due to the increased pressure drop and the electrical power required to establish the electric field. Figure 2 quantifies the actual enhancement as the ratio of the increased heat transfer over the field free case and the total additional hydraulic and electrical power. It is evident that the enhancement levels are considerable reaching a peak at 4 kV where the power gains are 35 higher than the power penalties.

Figure 3 illustrates a very novel aspect of EHD augmented heat transfer which is the control of the heat transfer rate in a heat exchanger. Here a PID controller is tuned to vary the applied voltage so as to control the overall heat transfer to a set point of 190W. It is clear that the set point is achieved with very good precision proving the concept that EHD augmented heat transfer is amenable to solid state control.

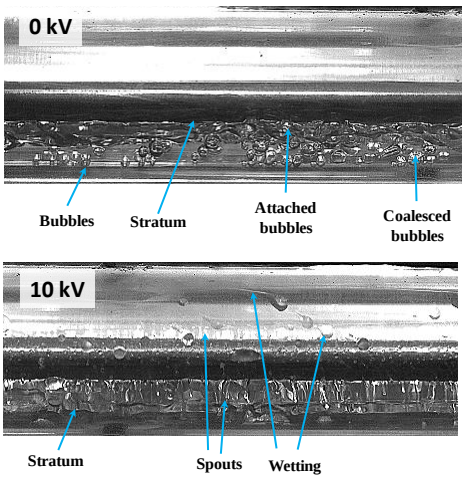


Figure 1: Flow regimes for 0 kV and 10 kV.

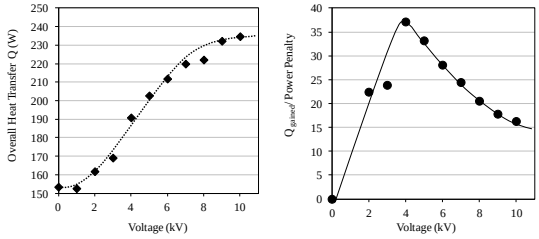


Figure 2: Heat transfer and enhancement.

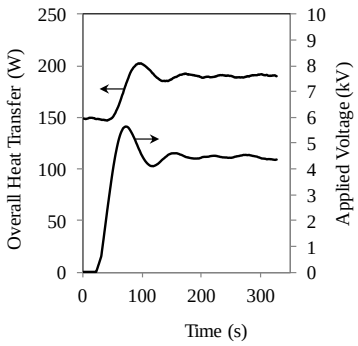


Figure 3: PID controlled heat transfer.

Segregation of particles in incompressible random flows: singularities, intermittency and random uncorrelated motion

E. Meneguz and M. W. Reeks
School of Mechanical and Systems Engineering, Newcastle University, Newcastle upon Tyne, NE1 7RU, UK
Keywords: segregation, inertial particles, random uncorrelated motion, singularities.
Presenting author email: Mike.Reeks@ncl.ac.uk

The transport of particles/droplets dispersed in turbulent flows is of crucial importance to a wide range of natural and engineering processes. In this theoretical and numerical study, we focus on the transport of heavy particles in an incompressible gas flow and exploit a Full Lagrangian method to measure the statistical properties of the particle segregation. While doing so, we are able to analyse some particular features of this ongoing process, and in particular to study the statistics of singularities in the particle concentration field and the recently observed Random Uncorrelated Motion (RUM): the velocity of particles with large inertia brought into close proximity may be strongly decorrelated not only with the flow but one with another.

In our recent work (IJzermans *et al*, 2009 and 2010), we have studied the segregation of heavy particles in turbulence by calculating the rate-of-compression of the particle phase in a kinematic simulation. Particles are advected by Stokes drag in a flow field composed of 200 random Fourier modes. The volume occupied by the particles centred around a position x at time t is denoted by $J = det(J_{ij})$, where $J_{ij} = \partial x_i(x_0)/\partial x_{0,j}$, where x_0 denotes the initial position of the particle. The particle-averaged compressibility, $\mathscr{C} = d < ln|J| > /dt$, gives a measure for the change of the total volume occupied by the particle phase. Numerical results showed that the particle-averaged rate-of-compression decreases continuously if the value of the Stokes number (the dimensionless particle relaxation time) is below a threshold value, St_{cr}, indicating that the segregation of these particles continues indefinitely. We find that the probability density function of $ln|J|$, the compression, tends to a Gaussian distribution for $St \sim 1$ when $t \to \infty$. We believe the explanation for Gaussianity is similar to that for the occurrence of a Gaussian distribution of displacement (Taylor, 1922), with $\mathscr{C}'(t)$, the fluctuating value of $\mathscr{C}(t)$ about its mean. However, we find that that such PDF shows a significant skewness towards negative compression (segregation), i.e. singularities in the flow are likely to play a significant role in determining the statistics of the segregation in these long term limits.

By counting events for which $|J(t)| = 0$, we can calculate the distribution of singularities over a fixed interval of time respectively for a set of St numbers. As shown in Figure 1 for $St = 1$, excluding the influence of an initial transient when no singularities are observed, the histogram that represents the discrete probability distribution is well approximated by a Poisson distribution that describes the probability of the occurrence of an event (singularity) in a specified time span $[0, \triangle t]$ as $\sim \lambda \triangle t = \Lambda$; λ is the rate constant for the occurrence of singularities. The Poisson process implies that starting from some initial fully mixed equilibrium distribution, the decay in the number of particles that have not experienced a singularity is $\sim exp(-\lambda t)$.

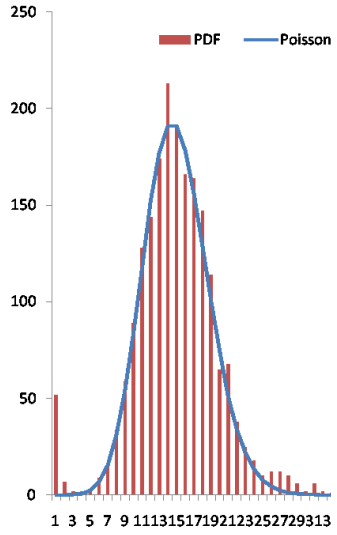


Figure 1: Comparison between theory and experimental data.

Finally, we discuss our work in relation to that of Falkovich & Pumir, 2007 and Wilkinson *et al*, 2007 and conclude that the occurrence of singularities is related to the formation of caustics and sling effect respectively, since it corresponds to the folding of the particle velocity field in phase space. We believe that RUM and singularities are intrinsically related and we are currently working to find a suitable way to demonstrate such theory from a mathematical and numerical point of view.

IJzermans, R. H. A., Reeks, M. W., Meneguz, E., Picciotto, M. and Soldati, A. (2009) Phys. Rev. Lett. **97**, 015302.

IJzermans, R. H. A., Meneguz, E. and Reeks, M. W. (2010) J. Fluid Mech. **653**, 99 – 136.

Taylor, G. I. (1922) in Proc. London Math. Soc. s2 **20**(1), 196 – 212.

Falkovich, G. and Pumir, A. (2007) Phys. Rev. Lett. **64**, M.

Wilkinson, M., Mehlig, B., Ostlund, S. and Duncan, K. P. (2007), Phys. Fluids **19**, 113303.

INCORPORATION OF NON-ADIABATIC EFFECTS TO PRESSURE DROP CALCULATIONS IN FLOW BOILING AND FLOW CONDENSATION

Dariusz Mikielewicz

1Faculty of Mechanical Engineering, ul. Narutowicza 11/12

Gdansk University of Technology, 80-233 Gdansk, Poland

Email: Dariusz.Mikielewicz@pg.gda.pl

Abstract

Flow boiling and flow condensation are often regarded as two opposite or symmetrical phenomena, however their description with a single correlation has yet to be suggested. Still separate approaches to modeling of pressure drop in boiling and condensation are developed. In case of conventional size tubes (diameters greater than 3mm) we are dealing with two major structures of two-phase flow, namely bubbly flow and annular flow. The task seems to be a little easier in the case of flow boiling/flow condensation in minichannels. In such case there is mostly encountered the annular flow structure, where the bubble generation/collapse is not present. In such case the heat transfer coefficient is primarily dependent on the convective mechanism, modeled in the majority of approaches in terms of the Martinelly parameter. The difficulty in devising a general method for pressure drop calculations, applicable to both flow condensation and flow boiling, lies in the fact that the non-adiabatic effects are not included into the present in literature models. Non-adiabatic effects alter the shear stress in flow boiling as well as in flow condensation and, in author's opinion, are the main explanation why up to date approaches to common modeling of flow boiling and flow condensation fail to devise a single robust model for that purpose. Additionally, in recent reports more are more conclusions are drawn that the pressure drop is a function of applied heat flux. There is however a different influence of heat flux on modification of shear stress in the bubbly flow structure, where it affects bubble nucleation. In case of annular flow it contributes to thickening and thinning of the liquid film, which corresponds to condensation and boiling respectively.

In the presentation a versatile method for calculation of pressure drop in flow condensation and flow boiling is presented. The key feature of the method is the approach to modeling of the modification of interface shear stresses in flow boiling and flow condensation due to mass flux and heat flux on interface. In case of annular flow structure incorporation of the so called "blowing parameter", which differentiates these two modes of heat transfer, is considered. That effect is devoted to a correct mass flux modelling on interface. The differences in shear stress between vapour phase and liquid phase is generally a function of non-adiabatic effect. Correct modelling of that heat flux enables to predict a thinner liquid film thickness in boiling and thicker in condensations at otherwise exactly the same flow conditions. That is a major reason why that up to date approaches, considering the issue of flow boiling and flow condensation as symmetric, are failing in successful predictions. In case of bubbly flow structure the effect of applied heat flux is considered. As a result a modified form of the two-phase flow multiplier is obtained, in which the non-adiabatic effect is clearly pronounced.

Some well established experimental data from literature for several fluids has been scrutinised in order to find the evidence for non-adiabatic effects. These data will be carefully

scrutinized to extract the effect of applied heat flux. Preliminary calculations show a satisfactory consistency of discussed model with experimental data.

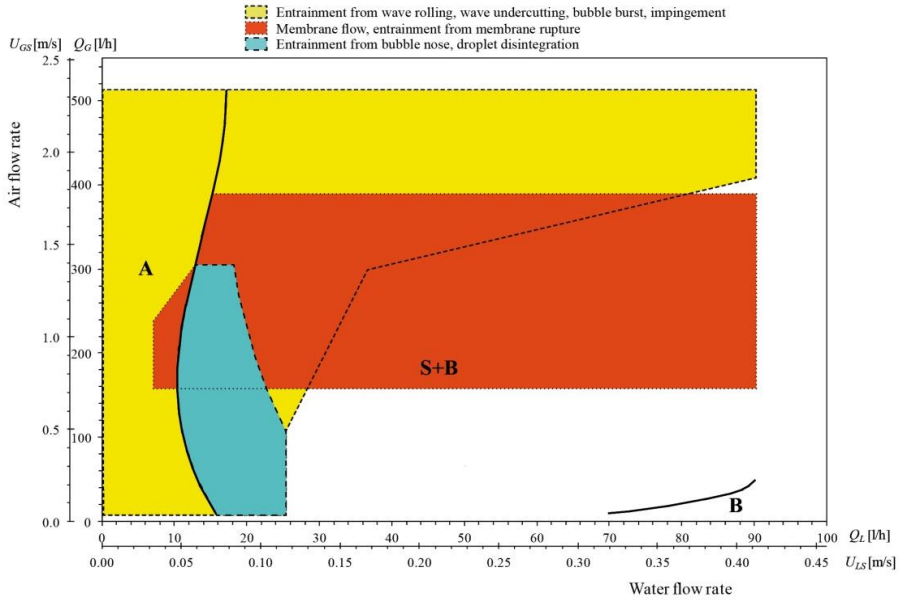
50th. European Two-Phase Flow Group Meeting 2012
Udine, Italy, 16 -18 May 2012

An Entrainment Mechanism Map for a Cocurrent Adiabatic Vertical Downward Air-Water Flow.

Marco MILAN, Navid BORHANI, John THOME.
Heat and Mass Transfer Laboratory LTCM, Swiss Federal Institute of technology EPFL, Lausanne, Switzerland.

The entrained fraction in a two-phase flow is of considerable practical importance for heat and mass transfer processes. Entrainment significantly alters the mechanisms of mass, momentum, and energy transfer between the liquid film and the gas core flow, as well as the transfer between the two-phase mixture and the wall. Depending from the flow regime, entrainment is characterized by different features while several entrainment structures form. Entrainment formation and deposition mechanisms which occur in two-phase flow are reviewed and investigated in this study. New entrainment mechanisms observed in an adiabatic concurrent downward air-water system at relatively low gas flow rates are presented. Entrainment from the bubble nose can occur in elongated bubble flow when high amplitude interfacial waves reach the bubble's downstream-end and create an increased pressure in the water slug. Membrane flow, in the form of a thin liquid bridge dividing the air core, was observed over a wide range of air and water flow rates. Although, this flow structure was found to be mainly inlet device dependent, it also formed due to air bubble coalescence and from liquid slug shrinkage. Finally, droplet deposition characteristics and the interaction between the entrainment process and the surrounding two-phase flow structure are described.

Our results gave evidence that the droplet entrainment size, inertia and distribution are strongly related to the formation mechanism and to the gas and liquid relative velocities. Furthermore, they confirm that the inlet device, the tube geometry, and the surface tension have an important influence on the entrainment formation and deposition rate. For the first time, an entrainment map is presented, which predicts, at a given combination of air and water superficial velocities, the entrainment structures which are expected to form, i.e. a liquid entrainment mechanism map. Such a map may play a valuable role in the development of a "smart" flow pattern map as recently proposed by Thome et al (2012), whose goal is to bring all mechanisms and models as an integral part of flow pattern maps, supplanting the current "GPS" type of map that only tells one in which regime one is in but gives no details of the actual flow.



Thome, J.R., Bar-Cohen, A., Revellin, R., Zun, I., 2012. "Smart" Mapping of Two-Phase Flow Patterns in Microchannels. In review.

Comparison of different methodologies for the simulation of coalescence, breakup and mass transfer in gas-liquid stirred tanks and bubble columns

M. Petitti, A. Buffo, M. Vanni, D. L. Marchisio
Politecnico di Torino, Dip. Scienza Applicata e Tecnologia, C.so Duca degli Abruzzi 24, 10129 Torino, Italy

The description of the fluid-dynamics and of mass transfer in gas-liquid systems requires the evaluation of bubble size and composition distributions, both varying in time and space, due to gas bubble breakage, coalescence and mass exchange with the liquid phase.

To this purpose a multivariate population balance model (PBM), coupled with an Eulerian multifluid approach, has been employed to describe the spatial and temporal evolution of bubbles size and composition distribution in a gas-liquid system. For the numerical solution of the multivariate PBM the Conditional Quadrature Methods of Moments (CQMOM) and the Direct Quadrature Method of Moments (DQMOM) have been implemented through user defined functions and scalars in the commercial computational fluid dynamics code Fluent (ANSYS). CQMOM and DQMOM are first compared and their performaces evaluated both on simplified flow systems as well as on realistic stirred-tank and bubble column geometries. Then model predictions are validated through comparison with experiments carried out in a 194 liter stirred-tank agitated by a Rushton turbine and operating with air and tap water. The gas-liquid mass transfer is investigated by considering the absorption of oxygen in the liquid phase, for which many experimental data are available, in many different operating conditions (stirring speeds and gassing rates). Validation is carried out by comparing first the model predictions with experimentally measured bubble size distributions and then with oxygen accumulation in the liquid phase.

Our simulations show that CQMOM and DQMOM are equivalent only for spatially homogeneous systems and under specific conditions. DQMOM suffers from the fact that it cannot conserve a large moment set, since it tracks the evolution of nodes and weights of the quadrature approximation, unless a complicated procedure is employed (labelled here as DQMOM fully conservative, DQMOM-FC). CQMOM is much more stable and conserves all the tracked moments, however it suffers from a numerical corruption problem, that can be overcome only by using spatial discretization schemes appositely formulated.

In addition our results show that, although coalescence and breakup tend to homogenize bubble composition, a multivariate PBM is necessary to properly describe mass transfer. In fact, due to the fact that smaller bubbles, generally located in regions characterized by high turbulent dissipation rates, exchange mass with a much faster rate than bigger bubbles, both bubble size and bubble composition should be included as internal coordinates.

Coalescence and breakup kernels, derived from the theory and already employed in our previous work, are here used together with different mass transfer rates (e.g., Higbie's or Lamont and Scott's laws) without any additional fitting, resulting in good agreement with experimental data in a wide range of operating conditions.

Eventually some guidelines for the use of CQMOM and DQMOM-FC in the simulation of gas-liquid systems are discussed.

Anisotropy in Pair Dispersion of Inertial Particles in Turbulent Channel Flow

E. Pitton[1], C. Marchioli[1], A. Soldati[1], V. Lavezzo[2], F. Toschi[2]

[1]Center for Fluid Mechanics and Hydraulics, University of Udine, 33100 Udine, Italy
[2]Dept. Applied Physics, Eindhoven Technical University, Eindhoven, The Netherlands

Abstract

The rate at which two particles separate in turbulent flows is of central importance to predict the inhomogeneities of particle spatial distribution and to characterize mixing. Pair separation is analyzed for the specific case of small, inertial particles in turbulent channel flow to examine the role of mean shear and small-scale turbulent velocity fluctuations. To this aim an Eulerian-Lagrangian approach based on pseudo-spectral direct numerical simulation of fully-developed gas-solid flow at shear Reynolds number $Re_\tau = 150$ is used. Pair separation statistics have been computed for particles with different inertia (and for inertialess tracers) released from different regions of the channel. Results confirm that shear-induced effects predominate when the pair separation distance becomes comparable to the largest scale of the flow.
Results also reveal the fundamental role played by particles-turbulence interaction at the small scales in triggering separation during the initial stages of pair dispersion. These findings are discussed examining Lagrangian observables, including the mean square separation, which provide *prima facie* evidence that pair dispersion in non-homogeneous anisotropic turbulence has a *super-diffusive* nature and may generate non-Gaussian number density distributions of both particles and tracers. These features appear to persist even when the effects of shear dispersion are filtered out, and exhibit strong dependency on particle inertia. Application of present results is discussed in the context of modelling approaches for particle dispersion in wall-bounded turbulent flows.

Heat transfer in spinodal decomposition: an experimental and numerical work

P.Poesio, D. Molin, S. Farisè, G.P. Beretta

With the trend of miniaturization in electronics, and the more widespread use of integrated systems, we need technologies that allow to exchange a large amount of heat in small devices and with the highest possible efficiency. Tuckerman and Pease (1981) introduced the concept of micro channel sink, and since then several technologies have been developed to exchange heat more effectively. In this framework using spinodal mixtures to generate an evenly distributed micro agitation which increases the effective diffusivity that therefore increase the heat exchange is a possibility .

When a binary mixture is brought from the single phase region of its phase diagram to the two-phase unstable region, it phase separates through a process that is referred to as spinodal decomposition. As shown in many experimental and numerical works, in low viscosity liquid mixtures this process is driven by the convection that is induced by phase transition, which is responsible for the experimentally observed enhanced coalescence among the drops. Accordingly, it becomes natural to assume that during phase separation an effective heat diffusivity could be defined, resulting from the fluctuations of the velocity and temperature fields, just like in turbulent flows.

This effect is well known in evaporation and liquefaction, where the strong density difference between the two phases drives a very strong, turbulent convection, which enhances heat transfer. On the other hand, in liquid-liquid phase transitions, density differences are much lower, and for isopycnic systems they can even be negligible. In that case, another effect becomes relevant, namely the motion induced by chemical potential gradients i.e., the so called Korteweg reversible body force, which is a typical non-equilibrium phenomenon that has been studied in numerous works. Although the enhancement to heat transfer caused by this effect is much smaller than that due to density differences, it is not negligible by any means and, as shown in recent experiments, it can reduce cooling times by a factor 2 or 4.

In this work we would like to present some results on the enhancement due to spinodal decomposition in microchannel. From a computational point of view the mixture is modeled through the diffuse interface model otherwise called model H, in the taxonomy of Hohenberg and Halperin which is based on the pioneering work of van der Waals, together with the Ginzburg-Landay theory of phase transition. This model was applied to spinodal decomposition of binary mixtures by Cahn and Hilliard and was later generalized by Kawasaki to include hydrodynamics. From an experimental point of view a mixture of acetone-hexadecane is quenched in a micro heat exchanger to induce spinodal decomposition. The heat transfer rate is enhanced by self-induced convective effects sustained by the free energy that becomes available during phase separation. The obtained results are compared with the pure substance heat transfer coefficients and to the computational data obtained with the Cahn-Hilliard model.

DNS of solid particle transport by hairpin vortices in a laminar boundary layer

M. Prevel, I. Vinkovic, D. Doppler and M. Buffat

Laboratoire de Mécanique des Fluides et d'Acoustique (LMFA) CNRS UMR 5509, France

Understanding and modelling the transport of inertial particles, such as sediments, is of practical interest for industrial engineering or environmental problems. In the near wall region, particles preferentially concentrate in low-speed streaks [1]. These regions move through the inclined loop of the hairpin vortices present in wall-bounded flows by induction from the legs and the head [2]. The current study focuses on determining how solid particles interact with artificially generated hairpin vortices in a laminar boundary layer.

Direct numerical simulations (DNS) [3] coupled with Lagrangian tracking of solid particles are used. The computational domain consists of two parallel walls in the xz plane. Discrete hairpin vortices are generated in a controlled way by a hemisphere protuberance placed on the lower wall in an initially stable laminar boundary layer. This hemisphere is introduced by the immersed boundary method. Particles are injected in the wake of the obstacle at low concentration. The simulations are performed for $Re_R = 750$ in order to match experiments [4]. Here Re_R is the Reynolds number based on R the radius of the hemisphere and u_{tip} the velocity within the impinging boundary layer that would exist at the same height as the upper tip of the hemisphere.

Figure 1 is a side view of transverse vorticity. The head, neck and legs of several hairpin vortices generated by the hemisphere can be seen. Far from the hemisphere, mean velocity profiles obtained by our simulations are in good agreement with the experiments [4]. From the instantaneous distribution of solid particles (Figure 1), it can be seen that particles cluster in regions of high strain, between the legs of the hairpin vortices and are ejected from the center of the vortex heads. In the full length paper, conditioned statistical analysis and acceleration autocorrelation will be shown in order to quantify the above described interaction mechanisms.

References

[1] C. Marchioli and A. Soldati. Mechanisms for particle transfer and segregation in a turbulent boundary layer. *J. Fluid Mech.*, 468:283–315, 2002.

[2] R.J. Adrian, C.D. Meinhart, and C.D. Tomkins. Vortex organization in the outer region of the turbulent boundary layer. *J. Fluid Mech.*, 422:1–54, 2000.

[3] M. Buffat, L. Le Penven, and A. Cadiou. An efficient spectral method based on an orthogonal decomposition of the velocity for transition analysis in wall bounded flow. *Comput. Fluids*, 42:1:62–72, 2011.

[4] M.S. Acarlar and C.R. Smith. A study of hairpin vortices in a laminar boundary layer. Part I. Hairpin vortices generated by a hemisphere protuberance. *J. Fluid Mech.*, 175:1–41, 1987.

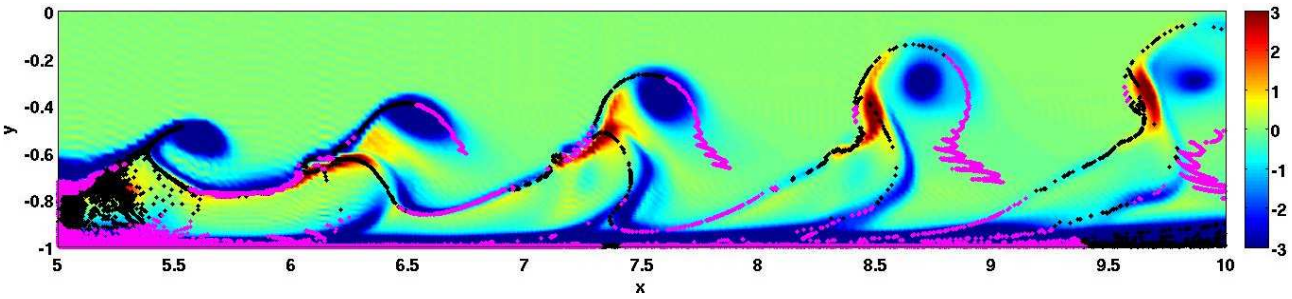


Figure 1: Isocontours of instantaneous transverse vorticity and particle distribution. Black dots - upward moving particles. Magenta dots - downward moving particles

Flow boiling of R-245fa at high reduced temperature

Revellin R.* and Charnay R. and Bonjour J.
*Author for correspondence
CETHIL UMR 5008
INSA Lyon
Bât Sadi Carnot
9 Rue de la Physique
69621 Villeurbanne cedex
France
E-mail: remi.revellin@insa-lyon.fr

Because of current environmental issues, some technologies are being developed to reduce the fuel consumption and to reduce the emissions of CO_2. Energy by means of Organic Rankine Cycles or Hirn Cycles recovery is one investigated track to answer these issues. Indeed, these cycles could represent an effective way to recover the waste heat energy of the internal-combustion engines. At present, some Organic Rankine Cycles are available in industry but advanced studies are needed to allow their application in the road transport industry. A better understanding of the two-phase fluid behaviour is necessary to optimize the design models of the components containing a two-phase refrigerant (evaporator and condenser). For the Organic Rankine Cycle, the thermodynamic conditions are different to standards relevant to refrigeration or air-conditioning systems. Indeed, a synthetic refrigerant is used at high reduced temperature (T / T_{crit}). The empirical models for boiling in such conditions are limited by the experimental data on which they are based, whereas analytical and theoretical approaches are needed to advanced knowledge on the behaviour of thermohydraulic two-phase refrigerant. Charnay et al. (2011) have highlighted some gaps in available experimental databases in literature for these thermodynamic conditions (high reduced temperature and high pressure). As well as the technological issue, this study offers a huge scientific interest with the comparison between the CO_2 and other synthetic refrigerants. This comparison will address the question of a presumed singular behaviour of CO_2 due to its use in reduced temperature ranges different from the usual range encountered with synthetic refrigerants.

A test rig was built to study the two-phase thermohydraulic behaviour of synthetic refrigerants in minichannels at high reduced temperature, i.e. from 0.2 to 0.9. Using R245fa as working fluid, a series of tests was performed to determine the heat transfer coefficients at high reduced temperature inside a 3.0 mm inner diameter smooth tube. The straight horizontal test section (length: 300 mm) is electrically heated by Joule effect. The mass velocity ranges from 100 to 1500 kg/m².s and the saturation temperature from 30°C to 130°C. The database obtained for the local heat transfer coefficient is compared with the prediction methods available in the literature. A visualization section made of a 3 mm inner diameter glass tube offers a better understanding of the flow patterns and allows developing flow pattern maps.

50^{th} European Two-Phase Flow Group Meeting,
Udine (Italy) May 16-18, 2012.

Turbulence modulation through the interface of a deformable drop

Luca Scarbolo†, Dafne Molin‡ and Alfredo Soldati†

† *Dip. Ingegneria Elettrica, Gestionale e Meccanica, Università di Udine, 33100 Udine, Italy*

‡ *Dip. Ingegneria Meccanica e Industriale, Università di Brescia, 25123 Brescia, Italy*

Abstract

Mass, momentum end energy exchange between two fluids components in turbulent systems are of huge importance in many industrial and environmental application. These phenomena are deeply correlated to the turbulence features in the proximity of the interface between the fluid components. Thus the prediction and modeling of turbulence behavior across the interface of large-size deformable droplets is crucial for the characterization of processes such as mixing in stirred chemical reactors, heat transfer in multiphase systems and fluid clusters dispersion in large scale oceanic currents.
In dispersed systems, like in oil emulsions, the evolution of the turbulent flow near the droplet interface is heavily affected by the interplay between turbulent forces and the droplet surface tension[1]. In this work we aim to analyze the effects due to surface tension on the turbulent flow across the interface of a single large deformable drop in a turbulent channel flow. For this purpose we developed a phase-field model (PFM) in which fluids properties, like density and viscosity varies in a continuous and smooth way from one fluid to the other; in this way the surface tension becomes a volume force distributed over a non-zero surface representing the interface between the two fluids. The fluids are considered incompressible, density-matched and viscosity-matched to avoid wake and interface dissipation effects.
The diffuse interface model is based on the free-energy functional already used by Badalassi et al[2]. and the system evolution is obtained solving the Cahn-Hilliard/Navier-Stokes coupled system in a channel geometry with a pseudo-spectral code.
We simulated a wide range of Weber numbers (ratio between inertial forces and surface tension) for three different droplet diameters and two different Reynolds number based on the friction velocity and on the channel half height (Re=100-150). The obtained results of velocity fluctuations, turbulent stresses and shear stresses across the interface are analyzed in respect to the droplet characteristic parameters such as the local curvature, the Gaussian curvature and the mean deformation.

1. Galinat S, Risso F, Masbernat O, Guiraud P, Dynamics of drop breakup in inhomogeneous turbulence at various volume fractions, *J. Fluid. Mech.*, **578** (2007), 85-94.

2. Badalassi VE, Cerniceros HD, Banerjee S, Computation of multiphase systems with phase field models, *J. Comp. Phys.*, **190** (2003), 371-397.

Experimental investigation of local oxygen mass transfer in reactive gas-liquid flows

Michael Schlüter[+], Ulf Daniel Kück[+], Norbert Räbiger*

[+]Institute of Multiphase Flows, Hamburg University of Technology,
Eissendorfer Straße 38, 21073 Hamburg, Germany

*Institute of Environmental Process Engineering, University of Bremen,
Leobener Straße, UFT, 28359 Bremen, Germany

In many fields of chemical engineering, biotechnology, life science and environmental process engineering the mass transfer from a gaseous disperse phase into the circumfluent liquid phase is of great importance. The accurate design of multiphase reactors is to date an almost unresolved problem. Mass transfer processes in particular with superimposed chemical reactions occurring under swarm conditions, cannot easily be determined by existing measurement techniques. Furthermore limitations in mass transfer are difficult to predict because the full reaction scheme, including kinetics, is often unknown and the timescales of mixing and reaction cannot be exactly calculated. Due to insufficient fundamental experimental data, the design of many types of mass transfer apparatus (e.g. bubble columns) is still based on empirical mass transfer correlations which are developed from integral measurements or models. In future enhanced calculation methods will be needed to allow a more profound characterization of multi-scale mass transfer in two-phase systems.

One step in this direction has been done within the cooperative Cluster (DFG PAK 119) of the German Research Foundation that focuses on reactive mass transfer from single bubbles and combines kinetic studies (Prof. Warnecke, University of Paderborn), experimental studies (Prof. Schlüter, TUHH) and direct numerical simulations (Prof. Bothe, TU Darmstadt). Mechanisms and principles have been investigated at different scales ranging from the molecular level to the macroscopic size, by using a combination of experimental and theoretical methods. Therefore interdisciplinary investigations of physico-chemical effects are performed experimentally and numerically by complementing each other. The mass transfer within the boundary layer of a free rising oxygen bubble into water has been studied by means of Laser Induced Fluorescence (LIF) and within the bubble wake by simultaneous LIF and Particle Image Velocimetry (PIV) measurements. Qualitative effects on oxygen concentration fields within the wake of single bubbles are discussed (Figure 1). The combined experimental and numerical data are used as basis for validation and the development of a verified model for local mass transfer in bubbly flows. Exemplary effects on the influence of local hydrodynamics on mass transfer at free rising gas-bubbles are shown. In addition, results with superimposed sulfite-sulfate reaction are presented.

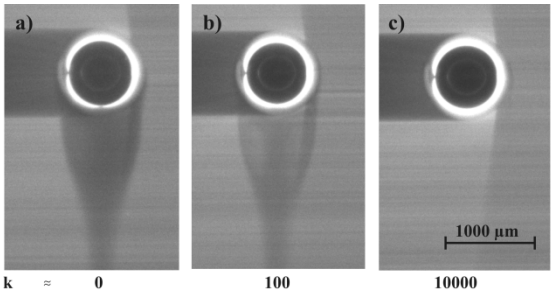


Figure 1: Qualitative effect of superimposed sodium sulfite-sulfate reaction on wake mass transfer

The Lattice-Boltzmann Method: A useful tool to study particle-scale phenomena

M. Sommerfeld, Y. Cui, M. Dietzel and M. Ernst
Zentrum für Ingenieurwissenschaften, Martin-Luther-Universität Halle-Wittenberg
D-06099 Halle (Saale), Germany

In recent years a Lattice-Boltzmann Method was developed in order to allow an efficient numerical simulation of flows about resolved single particles, agglomerates or particle clusters. Two features are essential for allowing such simulations with high spatial resolution. For resolving the complex curved boundaries of the considered particle, the standard step-like wall boundary conditions was replaced by a curved boundary condition which accurately accounts for the distance between wall and the nodes of the fluid domain. Moreover, for closely representing the particle contour with a sufficient number of nodes a local grid-refinement method was applied. This implies that regions around a complex particle are calculated on a fine mesh whereas regions in the outer flow region without strong gradients are calculated on a course mesh in order to save computational time. The number of grid refinement steps is depending on the considered problem.

The main objective in the application of the LBM is related to analysing flow phenomena occurring and the scale of the particle in order to develop correlations required in an Euler/Lagrange calculation, e.g. drag and lift coefficient of agglomerates. For demonstrating the capabilities of the developed LBM-code three problems are considered. The flow about fixed spherical fractal flocks is simulated in order to evaluate the resistance coefficients for different sizes and porosities of the agglomerate. As expected, the drag coefficient is decreasing with reduced fractal dimension and hence increased porosity. A larger particle covered with small particles (size ratio about 5/100) exposed to different flow situations (plug flow, shear flow and turbulence) is considered for evaluating the fluid dynamic detachment forces acting on the small particles. The dependence of the detachment force on coverage rate, the Reynolds number, the shear rate and the turbulence intensity is evaluated. Such a situation is found in an inhaler device where the drug particles have to be detached from the carrier particle. Finally the sedimentation of a particle cluster in a vertical channel with quiescent fluid is studied for analysing the interaction between the particles. During the sedimentation the primary particles will interact through the fluid and also collide to form agglomerates. The structure of the resulting agglomerates is analysed.

Experimental studies for allowing the modelling of droplet collisions with complex liquid properties

M. Sommerfeld and M. Kuschel
Zentrum für Ingenieurwissenschaften, Martin-Luther-Universität Halle-Wittenberg
D-06099 Halle (Saale), Germany

Sprays consisting of droplets with complex liquid properties are relevant for a number of technical and industrial processes. A typical example is spray drying where a solution or a suspension is atomised for producing a spray of fine droplets which are exposed to a heated gas stream in order to evaporate the solvent and produce a powder of defined size and morphology. Collisions between such droplets modify the size spectrum of the powder. Moreover, droplet collisions may be initiated on purpose to produce a defined size of the powder. Modelling droplet collisions in the classical Euler/Lagrange approach involves the following steps:

- detection of a collision by either a deterministic or a stochastic collision model
- identification of the type of droplet collision occurring based on a domain chart (i.e. impact parameter as a function of collision Weber number)
- determination of the collision outcome, namely, the resulting droplet sizes and velocities.

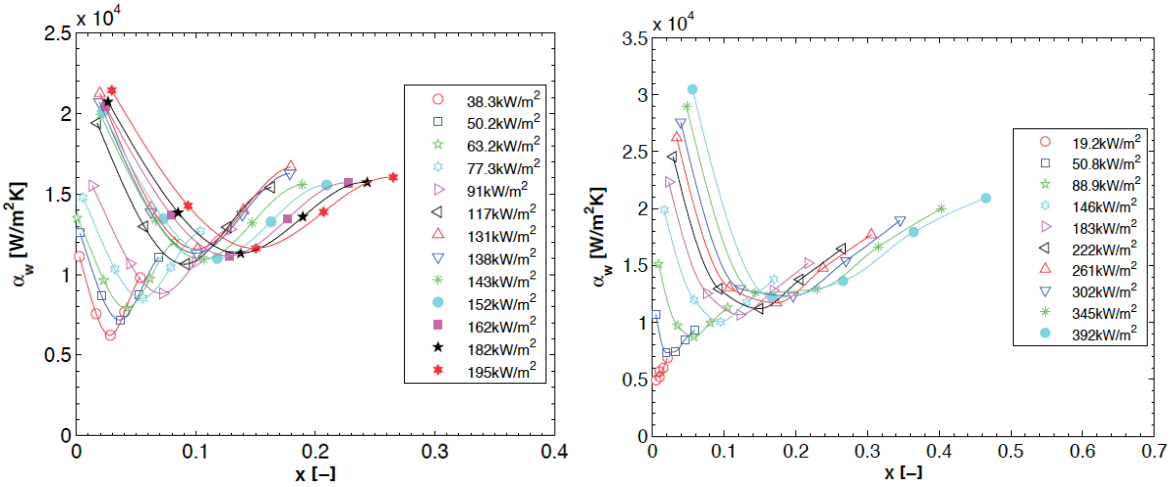
The types of droplet collisions observed in a spray are: bouncing, coalescence and stretching as well as reflexive separation. In order to distinguish between these collision types a domain chart is used where the distance between droplet centres upon collision (i.e. impact parameter) is plotted versus the collision Weber number, determined with the liquid properties and the relative velocity. Unfortunately, such domain charts are not universal and depend among others on the liquid properties. Domain charts are available for water and a number of model fuels, but not for solutions and suspensions. Therefore, a comprehensive experimental study was started for analysing the collision of solution and suspension droplets in order to provide the basis for a Lagrangian modelling. For that purpose two piezo-electric nozzles were used to produce droplet chains of defined size which were directed towards each other in order to initiate a collision of droplet pairs with defined conditions, i.e. collision angle and relative velocity. The collision was recorded by two perpendicular arranged high-speed cameras. The illumination was realised by two LED arrays.

In the studies different solutions with different solids content were used. The viscosities of the solutions were remarkably higher than those for water and drastically increase with growing solids content. Naturally the domain chats varied for different solutions and solids content. From these data it was tried to develop a generalised model for droplet collisions.

New Flow Pattern Based Model for Flow Boiling in Microchannels

Prof. John R. Thome and Dr. Etienne Costa-Patry
LTCM, EPFL, 1015 Lausanne, Switzerland

Abstract: This lecture describes our new flow pattern-based prediction method for flow boiling heat transfer coefficients in microchannels (1), based on new experimental results for R-134a, R-1234ze(E) and R-245fa for a copper test section with 52 parallel microchannels (1): 163µm wide and 560µm high with the channels 46µm wide fins and a silicon test section with 135 parallel microchannels (2): 85µm wide and 560µm high with 178µm wide fins. The pseudo-CPU heating element had 35 local heaters and temperature sensors arranged in a 5 by 7 array (2.5mm by 2.5mm each). The wall heat transfer coefficients were found to be a function of the heat flux, vapor quality and mass flux with a prominent minimum at the flow pattern transition (example in left diagram below for R-134a at 411 kg/m^{2}s). The new model was built by combining an updated version of our three-zone model for slug flow (3,4) together with our recent 1-d annular convective flow boiling model (5) extended to high aspect ratio rectangular channels while using an updated version of our diabatic microchannel flow pattern map (6) for the transition between these two regimes (simulation in right diagram at same conditions). The model also predicted data for four single microchannels accurately, capturing the trends versus vapor quality well.



(1) **Costa-Patry, E., Olivier, J. and Thome, J.R. (2012).** Heat Transfer Characteristics in a Copper Micro-Evaporator and Flow Pattern-Based Prediction Method for Flow Boiling in Microchannels, *Frontiers in Heat and Mass Transfer*, www.ThermaFluidsCentral.org, Vol. 3, paper 013002.

(2) **Costa-Patry, E., Nebuloni, S., Olivier, J. and Thome, J.R. (2012)**. On-chip two-phase cooling with refrigerant 85 µm-wide multi-microchannel evaporator under hot-spot conditions, *IEEE Transactions on Components and Packaging Technologies*, Vol. 35, in press.

(3) **Thome, J.R., Dupont, V. and Jacobi, A. (2004)**. Heat Transfer Model for Evaporation in Microchannels, Part I: Presentation of the Model, *Int. J. Heat Mass Transfer*, Vol. 47, pp. 3375-3385.

(4) **Dupont, V., Thome, J.R. and Jacobi, A. (2004)**. Heat Transfer Model for Evaporation in Microchannels, Part II: Comparison with the Database, *Int. J. Heat Mass Transfer*, Vol. 47, pp. 3387-3401.

(5) **Cioncolini, A. and Thome, J.R. (2011)**. Algebraic Turbulence Modeling in Adiabatic and Evaporating Annular Flows, *Int. J. Heat and Fluid Flow*, Vol. 32, pp. 805-817.

(6) **Ong, C.L. and Thome, John R. (2011)**. Macro-to-Microchannel Transition in Two-Phase Flow: Part 1 - Two-Phase Flow Patterns and Film Thickness Measurements, *Experimental Thermal and Fluid Science*, Vol. 35, pp. 37-47.

MICROCHANNEL EXTRACTIONS USING IONIC LIQUIDS FOR SPENT NUCLEAR FUEL REPROCESSING

Dimitris Tsaoulidis, Valentina Dore, Panagiota Angeli*

Department of Chemical Engineering, University College London, Torrington Place,

WC1E 7JE, UK

E-mail: p.angeli@ucl.ac.uk*

Nuclear energy from fission can provide substantial amounts of carbon free electricity and process heat. However, one of the main concerns is the management of the radioactive effluent from the nuclear reactor (spent fuel) which can remain toxic for thousands of years. Efficient spent fuel reprocessing through a series of operations can separate uranium U(VI) to be reused as reactor fuel and reduce the volume and toxicity of the rest of the spent fuel that needs to be stored or disposed. In most commercial applications U is separated from nitric acid solutions using tri-n-butylphosphate (TBP) as extractant in diluents of large aliphatic chain hydrocarbons (i.e. n-dodecane or kerosene). Recently ionic liquids (IL) have been suggested as alternatives to organic solvents because of their negligible volatility and flammability at common industrial conditions. Ionic liquids are salts that are liquid even at room temperature. Their high resistance to radiation makes them particularly suitable for use in the nuclear reprocessing cycle.

The extractions in the spent fuel reprocessing cycle are commonly carried out in mixer-settlers or pulsed columns. Such units have complex, non-uniform flow fields which bring severe limitations to their efficiency and ability to scale up. The pressing demands on sustainable, efficient, and safer flow processes make micro-fluidic devices an attractive option for chemical operations. The thin fluidic films intensify mass transfer while the increased surface to volume ratio favours interfacial transfer rates and allows the manipulation of interfaces to obtain favourable flow patterns.

In this work a detailed investigation of the extraction behaviour of U(VI) from a nitric acid solution (aqueous phase) by the extractant TBP dissolved in an ionic liquid in microchannels is reported. Capillaries made from materials that had different wetting characteristics and a nominal internal diameter of 0.2 mm were used as test channels. The two phases were separated at the outlet and the extracted U(VI) in the ionic liquid phase was detected with a spectrophotometric detector. Results will be reported on (a) flow patterns and pressure drop under different phase flow rates; (b) extraction efficiency particularly during plug flow which

is considered to enhance mass transfer because of the very thin films between the plugs and the wall and of the circulation patterns that develop within the slugs.

Enhancement of forced and free convection heat transfer rates during spinodal decomposition of equal-density binary liquid system

Amos Ullmann, Konstantin and Neima Brauner

School of Mechanical Engineering
Tel-Aviv University, Tel-Aviv 69978, Israel

Boiling heat transfer is common in various industrial applications, including mini and microelectronic systems, where high heat transfer rates are required. However, the heat transfer efficiency is deteriorated when the bubble size is of the order as the channel diameter. In such cases, earlier dry out occur, causing a significant drop in heat transfer rates, below the rates obtained in single phase liquid flow.

To overcome the above limitations, the possibility of using phase transition of liquid-liquid systems, instead of phase change in vapor-liquid systems was tested. The liquid system used is partially miscible solvent system, with a Critical Solution Temperature (CST). Such a system can alter from a state of a single liquid phase, to a state of two separated liquid phases, by a small temperature change. The heat transfer enhancement during phase separation is such systems has been recently investigated in our research group (Gat *et al.* 2006, 2009) and by Poesio *et al.* (2007). It was found that phase separation can enhance significantly the forced convection heat transfer in small diameter pipes (up to 200%). The heat transfer augmentation was attributed mainly to the self-propulsion of droplets and the resulting mixing, which are driven by Korteweg capillary forces during the non-equilibrium stages of phase separation. The free convection from the outer surface of the pipe was also studied and found to be enhanced. The augmentation in this case was attributed mainly to the larger density difference of the separating phases, as compared to single phase free convention for the same temperature difference.

In the present study we use a two-component system with almost identical densities, in order to confirm the findings that convective heat transfer rates in pipe flow are enhanced also in practically zero gravity systems. The obtained results basically confirm our previous finding that the lateral droplet motion, and the associated near wall mixing, is not due to buoyant forces. The dimensionless groups suggested in Gat et al. (2009) are used to derive an empirical correlation for the augmentation factor.

The equal density solvent system is also used to explore the free convection heat transfer phenomena from a vertical plate during phase separation. Unexpectedly, here too, enhanced heat transfer rates are found compared to single phase free convention for the same temperature difference. Visualization of the flow field during the phase separation enables to associate the heat transfer augmentation to the observed flow phenomena.

References

S. Gat, N. Brauner and A. Ullmann., Heat Transfer Enhancement via Liquid-Liquid Phase Separation", Proc. of the 13th International Heat Transfer Conf., Sydney Australia 13-18 August, paper No. HTE10, 2006.

P. Poesio, A.M. Lezzi, and G.P. Beretta. Convective heat transfer enhancement induced by spinodal decomposition". Physical Review E 75(6):066306, 2007.

S. Gat, N. Brauner, A. Ullmann. Heat Transfer Enhancement via Liquid-Liquid Phase Separation, Int. J. Heat and Mass Transfer, 52: 1385-1399 (2009)

EXPERIMENTAL INVESTIGATION ON HEAT TRANSFER IN TRANSIENT BOILING

Roberta Visentini[1,2]*, Catherine Colin[1] and Pierre Ruyer[2]
*Author for correspondence
1.Institut de Mécanique des Fluides de Toulouse, 31400, France
E-mail: rvisenti@imft.fr
colin@imft.fr
2.Institut de Radioprotection et de Sûreté Nucléaire, Cadarache, 13108, France
E-mail: pierre.ruyer@irsn.fr

ABSTRACT

During reactivity initiated accidents in the core of a nuclear reactor, a power excursion occurs on part of the fuel rods. The corresponding heat transfer toward the coolant can lead to the establishment of a vapour film surrounding the rod. This decreases the efficiency of the process of heat removal and leads to a rod clad temperature increase. The risk for rod clad failure is a matter of study for the nuclear power plants safety evaluation. Past research programs have shown that the transient clad to coolant heat transfer in these accidental conditions highly differs from classical heat transfer correlations. Our objective is to better understand the influence of power excursion on the boiling heat transfer and to characterise the phases of the rapid boiling phenomenon. An experimental set-up has been built at IMFT. The test section is a semi annulus. The inner half cylinder is made of a 50 microns thick stainless steel foil. Its diameter is 8.4mm. The outer part consists of a 34mm internal diameter glass half cylinder. The stainless steel foil is heated by Joule effect. Its temperature is measured by an Infra Red Camera filming the backside of the foil. The IR camera is synchronised with a high-speed video camera filming the boiling regimes at 90° from the IR camera. Measurements of liquid temperature, instantaneous current and voltage delivered by the electrical power supply are recorded at the same time. The electrical resistance of the foil is known, so the Joule power is calculated and the wall heat flux from the heated foil to the liquid is estimated. The foil Biot number is lower than 0.01, the foil temperature can thus be considered homogeneous. The time evolution of the averaged wall temperature is deduced from the IR images.

Two types of experiments are conducted. In the first ones, a squared current signal is applied to the foil. Its duration varied from 9s, for the lowest powers, to 4-5s, for the highest power. The two cases are compared on the boiling onset temperature. They show different behaviours. A power threshold (300W), that corresponds to a 50A applied current, can be found. Experiments having a lower power present an augmentation of the boiling onset temperature as the temperature growing rate increases. An asymptotic value of the boiling onset temperature is reached when the applied power is higher than the threshold.
In the second type of test, the applied current grows, following a triangular signal. In this case, the boiling onset seems to be less affected by the initial temperature growth rate, which is also less significant than in the case of squared signals. Moreover, the mean power increase rate during nucleate boiling appears steeper than in the case of square powers. The evolution of the wall temperature during nucleate boiling is also plotted, versus the mean power for steady and transient nucleate boiling regimes. The heat transfer is larger in transient nucleate boiling regime, than in steady nucleate boiling regime. A first qualitative comparison with literature data was made, but a deeper analysis of the heat loss during the experiments, is planned in the near future.

50th European Two-Phase Flow Group Meeting, Udine, 16th – 18th May 2012

Pool boiling heat transfer on low narrow finned mild steel tubes

Yabai Wang, Patrick Bujok, Andrea Luke

Institute of Technical Thermodynamics, University of Kassel, Germany

Process-integrated energy efficiency in industrial plants may be only achieved with new, innovative equipment concepts. These are not applied despite of large energetic benefits because of unavailability of adequate references and data for the equipment design and deficient documents for operating reliability claims. Therefore, industrial companies (e.g. plant manufacturers und operators, engineering offices) and five universities will develop new equipment and plant concepts to raise unused potentials for heat integration within a joint research project. The part of our institute shows methods to transfer heat effectively and reliably with a very small temperature difference by multiphase systems, by evaporating and by condensing. The apparatus will be even more effective by using finned or enhanced surfaces. Never than less, this is known since several decades to be the most effective way to transfer heat, especially in chemical industry smooth surfaces are still the standard.

The aim of our research project is to obtain microstructures that are particularly efficient in terms of heat transfer and production costs. Furthermore, a reliable data base for the design of evaporators is necessary to obtain such microstructures. Therefore, in the laboratory the boiling heat transfer on single tubes and in small tube bundles with different microstructures, which are produced industrially by the project partner (Wieland, Ulm), is investigated. Selected structures are then used on a pilot plant at our project partner (Linde, Pullach) to validate the ability of a transfer from laboratory scale - single tubes, mini-bundle - to technical standards.

The influence of low finned mild steel tubes with 30 trapezoidal fins per inch (fpi) on the pool boiling heat transfer in a wide pressure range from incipient to fully developed nucleate boiling is discussed. The test fluids are hydrocarbons and their mixtures, used natural refrigerants and representative fluids often encountered in industrial processes. The results on an electrical heated, horizontal test tube of mild steel are compared with the correlations in the literature. It is discussed, which area is representative for comparison of different finned tubes in boiling heat transfer, since the correlations in literature are formulated very unclear in this regard or even impractical for the technical applications.

Keynotes: heat transfer, pool boiling, hydrocarbons, finned tubes, tube bundle

Deformation of a Viscous Drop in Compressional Stokes Flow

Michael Zabarankin[1)], Irina Smagin[2)], Olga Lavrenteva[2)], Avinoam Nir[2)]
1) Department of Mathematical Sciences, Stevens Institute of Technology, Castle Point on Hudson, Hoboken, NJ 07030, USA; mzabaran@stevens.ed
2) Department of Chemical Engineering, Technion, Haifa, IL-32000, Israel, avinir@tx.technion.ac.i

In this communication we report the dynamics and deformation of a drop subject to a slow viscous compressional flow (extending the work on bi-axial extension by Stone & Leal, 1989). We address two related problems: the dynamics of a drop of a given shape and the steady shape of a drop under the deforming flow. The problems are addressed analytically employing generalized axisymmetric potential and numerically using a variation of the BIE formulation for slow viscous flow (Toose et al., 1996; Smagin et al., 2011). The parameters investigated are the inner to outer viscosity ratio, λ, and the capillary number, Ca. For the case $\lambda = 1$ we present exact shapes stemming from the analytic solution of the integral equation of Zabarankin and Nir (2011), expressed in terms of Chebyshev polynomials. For the general case of λ, in particular in the range $\lambda < 1$, we obtain numerical solutions with a high degree of accuracy.

It is demonstrated that while at relatively low capillary numbers the drop shape resembles that of an oblate spheroid (see solid shape in Fig.1), for capillary numbers close to the critical one the deformed drop assumes the shape of a flat disk with rounded edge (see dashed shape in Fig.1). Some interesting results are that, although the critical capillary number increases dramatically with the diminishing drop viscosity, the critical shape is very similar at all λ, having a form of a flat disc with similar dimension and with a rounded end. When λ is not very low ($\lambda \geq (1)$) and when Ca is close to the critical one, the drop acquires minor dimples at the axis of symmetry. Thus, contrary to the case of positive extensions where low viscosity drops can deforms to highly slender shapes, in compressional flow even the bubble would not experience indefinite steady deformation as the capillary number increases. For practical use we present a highly accurate two-parameter approximation of the steady deformed drop for all λ and Ca.

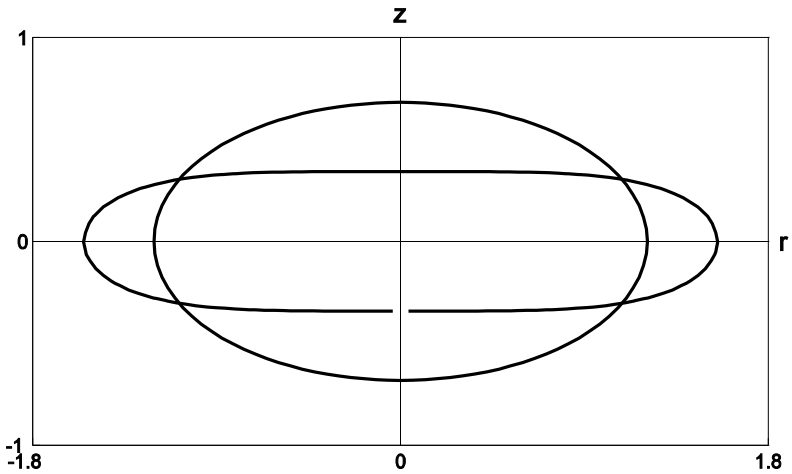


Fig. 1.The cross-section of the steady-state shapes of the drop
for equal viscosity, $\lambda = 1$. Ca = 0.15 (solid line) and Ca =0.193 (dashed line).

[1] Stone, H A and Leal, L G, *J Coll Interface Sci* **133**(2), pp 340-347, 1989.
[2] Zabarankin, M and Nir, A, *SIAM J. Appl. Math.* **4**, pp 925-951, 2011.
[3]Toose E, van den Ende, D, Geurts, B, Kuerten. J and Zandbergen P, *J Eng Math,* 30, pp. 131-15, 1996.
[4] Smagin, I, Pathak, M, Lavrenteva, O M and Nir, A, *Rheologica Acta,* 50 (4), pp. 361-374, 2011.

Air-Water Two Phase Flow Patterns in small Tubes Diameters Under Different Inlet Conditions

S. Zeguai [a], S. Chikh [b], L. Tadrist [c]

[a] UYFM, Faculté des Sciences et de Technologie, LMP2M, Ain d'heb, Médéa 26000, Algeria

[b] USTHB, Faculty of Mechanical and Process Engineering, LTPMP, Bab Ezzouar, Alger 16111, Algeria

[c] Aix-Marseille Université, Polytech'Marseille, Laboratoire IUSTI, CNRS UMR 7343, Marseille, France

Abstract

A tremendous work on two phase flow in mini or micro channels has been carried out during the last decades. However, an exhaustive analysis of published studies showed that there is no consensus on conditions yielding different flow patterns. Although the main two phase flow patterns were reported in the literature, namely bubbly flow, slug flow and annular flow, some other transition patterns were also visualized but not by all authors. The present study deals with air-water two phase flow in a circular tube of small diameter size and it focuses on the effect of inlet conditions on the flow patterns and the transition boundaries. A glass tube of 3 mm inner diameter is used as a test tube in the experiments and the two phases are injected in a co-current configuration. Air is injected axially through a nozzle of 0.26 mm and 0.11 mm inner diameter. In the investigated ranges of apparent velocity of the liquid and gas phases, the published works reported only a bubbly flow. A carefully controlled inlet conditions showed that several two phase flow patterns could be visualized in these ranges, namely bubbly, slug, annular, wavy annular flow and other flow patterns were seen for the first time.