51st European Two-Phase Flow Group Meeting

May 13, 14 and 15th 2013
Lyon, France

Centre de Thermique de Lyon
CETHIL
UMR 5008

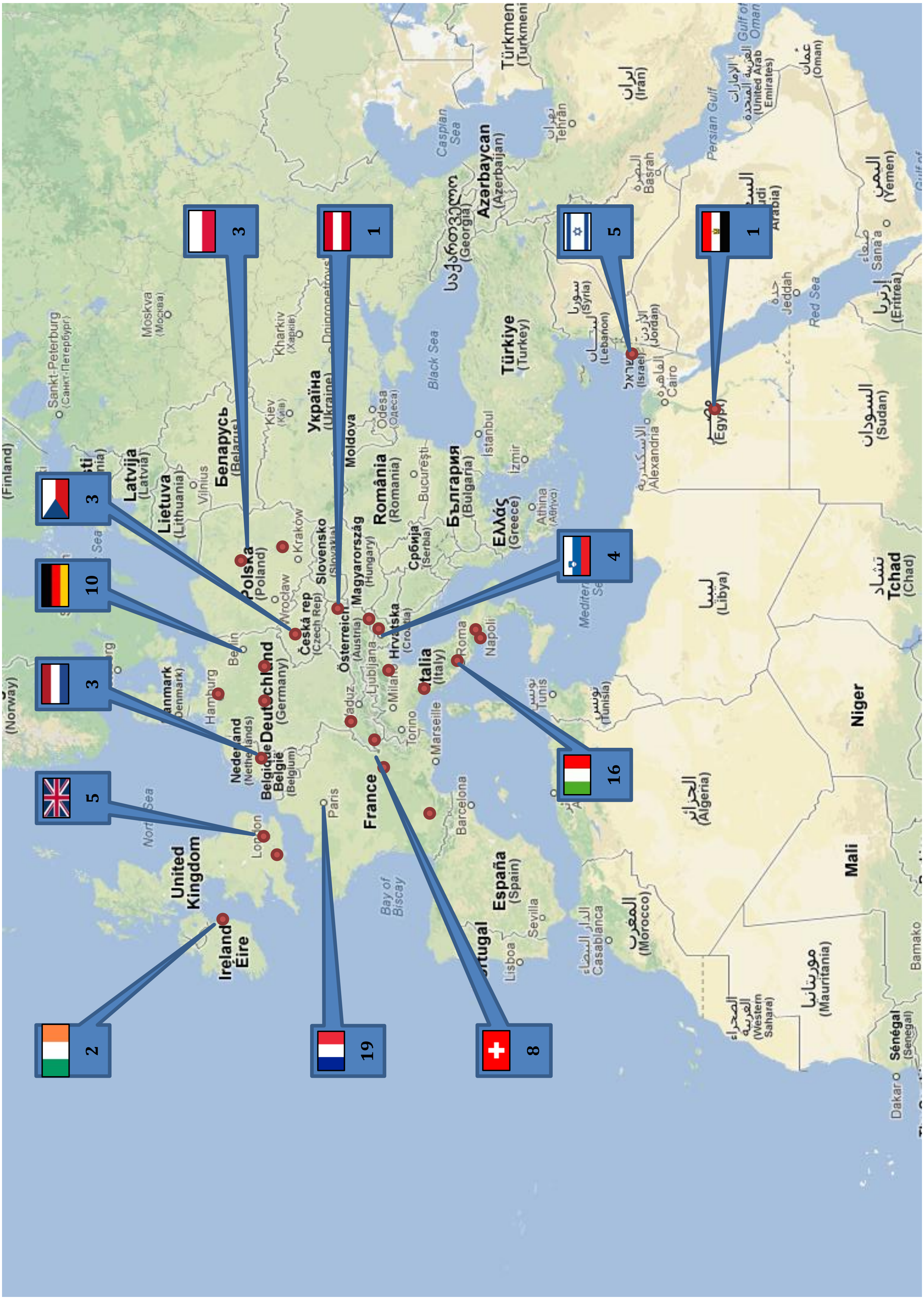
INSA LYON

Objectives and Scope

The European Two-Phase Flow Group is unique and it has a very special character unlike most other professional meetings. It is a prestigious meeting for invitees only that are expert in their field. Most of the participants are from Europe but it may include some distinguished scientists usually from the U.S. and Japan. The purpose is to establish informal get-together of first rate scientists dealing in multiphase flow with application to energy and environment and to present up-to-date research activity. Emphasis is placed on discussion and exchange of ideas. It is acceptable, and in fact encouraged, to present incomplete work in order to enhance fruitful discussions.

The ETPFG was established on October 4th, 1963, at the Royal Institute of Technology in Stockholm, after the EAES Symposium on Two- Phase Flow, Steady State Burnout and Hydrodynamics Instability. Professor Becker, who at the time held the chair of Reactor Technology at the Royal Institute of Technology, was the founder. Further information is available at the Enea site. The main objective of the Meetings has been to have informal discussions among researchers on multiphase flows. All subjects related to gas-liquid as well as fluid-solid flows can be discussed at the Group Meetings, boiling, flashing and condensation, dispersed flows (bubbly, droplet and particle laden flows), turbulence interactions, flow pattern regimes, singularities and choked flows, reacting flows, environmental and biological flows, experiments, theory and simulations.

Repartition of authors per country



Monday 13th - Program

Chairman : R. Revellin

8:00 - 9:30	Registration	
9:30 - 9:45	R. Revellin - *Introduction*	
9:45 - 10:30	**F. Lefevre - *Thermally induced oscillatory two-phase flow in a mini-channel: towards understanding transport mechanisms inside pulsating heat pipes***	p.9

.... break ....

Chairman : F. Lefevre

11:00 - 11:30	Y. Taitel - *Design of experimental steady state two chambers gasifier*	p.10
11:30 - 12:00	N. Lamaison - *Dynamic Modeling and Experimental Evaluation of Two-Phase On-Chip Cooling of Parallel Pseudo-CPUs*	p.11
12:00 - 12:30	A.W. Mauro - *Flow boiling of alternative refrigerants at high temperatures in a horizontal smooth tube: experimental investigation on the effect of reduced pressure on adiabatic frictional pressure gradients*	p.12

.... **Lunch** ....

Chairman : Y. Taitel

14:00 - 14:30	S. Lips - *Physical meaning of the pressure drop decomposition in two-phase flows*	p.13
14:30 - 15:00	Y. Kaplan - *Two Phase Liquid-Liquid Flow in Pipe Bends*	p.14
15:00 - 15:30	T. G. Karayiannis - *Flow boiling patterns, pressure drop and heat transfer rates of R134a and R245fa in micro tubes*	p.15

.... break ....

Chairman : S. Lips

16:00 - 16:30	R. Charnay - *Experimental investigation of R-245fa flow boiling in minichannels for different flow conditions*	p.16
16:30 - 17:00	S. Khodaparast - *Micro particle shadow velocimetry (µPSV) for air-water flow analysis in capillaries*	p.17
17:00 - 17:30	G. Zummo - *Visual study and heat transfer of flow boiling in a transparent tube*	p.18

....

19:30 Conference diner at the *Mercure Charpennes*

Tuesday 14th - Program

Chairman : *C. Rops*

8:00 - 8:30	M. E. Poniewski - *Numerical calculations of two – dimensional temperature field in a vertical minichannel*	p.19
8:30 - 9:00	S. Mosler - *Numerical simulation and experimental investigation of two-phase flows in Y-shaped mircochannels*	p.20
9:00 - 9:30	H. CV. Kuhlmann - *Flow Topology of Hydrothermal Waves in Liquid Bridges and its Relation to Dissipative Particulate Structures*	p.21
9:30 - 10:00	T. Saenen - *Dynamic modeling of a two-phase flow microchannel electronics cooling system*	p.22

.... break ....

Chairman : *J. R. Thome*

10:30 - 11:00	D. Del Col - *Experimental and numerical investigation of condensation inside minichannels*	p.23
11:00 - 11:30	A. Soldati - *Break-up of small inertial aggregates in turbulent channel flow*	p.24
11:30 - 12:00	C. Rops - *Pressure fluctuation reduction in once-trough micro evaporators*	p.25
12:00 - 12:30	J. Shrimpton - *The dispersion and two way coupling of small electrically charged particles in stationary isotropic turbulence*	p.26

.... **Lunch** ....

Chairman : *M. Sommerfeld*

14:00 - 14:45	**L. Fournaison – *The use of two-phase fluids for secondary refrigerants***	p.27
14:45 - 15:15	K. Zabkova - *Experimental Investigations of Influence of Solubility of Gas in Oil on Degassing Phenomena in Two-Phase Flow*	p.28
15:15 - 15:45	C. Besnaci - *Mixing at high Schmidt number in a random array of spheres*	p.29

.... break ....

Chairman : *D. Del Col*

16:15 - 15:45	M. Sommerfeld - *Multi-scale analysis of drug delivery through an inhaler device*	p.30
16:45 - 17:15	J. Urevc - *Viscosity model of time-dependent blood flow considering deformable RBCs clustering*	p.31
17:15 - 17:45	M. Lance - *Velocity measurements in a cavitating micro-channel flow*	p.32

....

19:30 **Gala diner at *La Cuvée* (departure from the conference venue at 19:00)**

Wednesday 15th - Program

Chairman : *P. Di Marco*

8:00 - 8:45	**A. J. Robinson** - ***My Experience with Bubbles***	p.33
8:45 - 9:15	A. Audouin - *Size and velocity distributions of the droplets produced by a single wave in a stratified air-water pipe flow*	p.34
9:15 - 9:45	M. Miscevic - *On the behaviour of a water droplet on a heterogeneous wettability surface*	p.35

.... break ....

Chairman : *A. J. Robinson*

10:00 - 10:30	Y. Wang - *An investigation of bubble formation and fluid dynamics in pool boiling of propane on horizontal tubes*	p.36
10:30 - 11:00	A. Filela - *Motion of a single bubble rising in a countercurrent flow in a Hele-Shaw cell*	p.37
11:00 - 11:30	J. Tihon - *Wall shear stress induced by rising Taylor bubbles*	p.38
11:30 - 12:00	P. Di Marco - *Detachment of gas bubbles from an inclined surface*	p.39
12:00 – 12:30	S. Siedel - *Quasi-static bubble shapes*	p.40

... Lunch ....

Lyon Subway Map

La Cuvée
coktail dinner
Tuesday evening

Mercure Charpennes
conference venue
and **Hôtel des congrès**

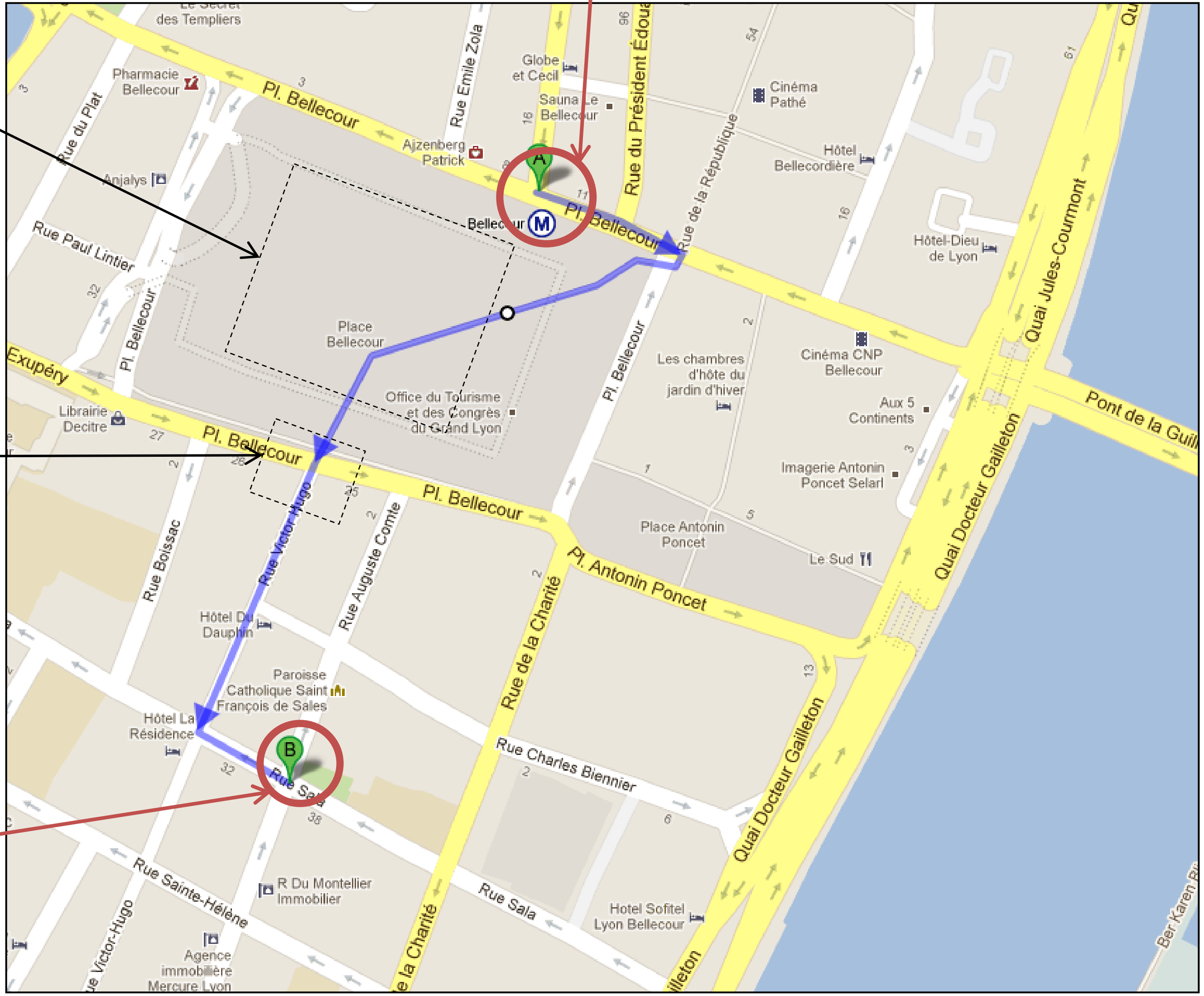
Hôtel Ariana

Tuesday evening at « La Cuvée »

Place Bellcour

Victor Hugo Street

Bellecour subway station



La Cuvée restaurant

angle of Sala street and Auguste-Compte street

Thermally induced oscillatory two-phase flow in a mini-channel: towards understanding transport mechanisms inside pulsating heat pipes

Frédéric Lefèvre[a]
Manoj Rao[a], Sameer Khandekar[b] , Jocelyn Bonjour[a]
[a] *Centre de Thermique de Lyon, UMR 5008 CNRS INSA Univ. Lyon, INSA-Lyon, F-69621 Villeurbanne, France*
[b] *Department of Mechanical Engineering, Indian Institute of Technology Kanpur, Kanpur 208016 India*
e-mail : frederic.lefevre@insa-lyon.fr

A Pulsating Heat Pipe (PHP) is an apparently simple looking heat transfer device, however with complex internal thermo-hydrodynamic transport processes, responsible for the self-sustained thermally driven oscillating two-phase Taylor bubble flow, which, in turn, leads to its unique heat transfer characteristics. Research on PHP has received substantial attention in the recent past, due to its unique operating characteristics and potential applications in many passive heat transport situations. A PHP consists of a simple capillary tube, with no wick structure, bent into many turns, and partially filled with a working fluid. When the temperature difference between the heat source and the heat sink exceeds a certain threshold, the vapor bubbles and liquid plugs present inside the capillary tube begin to auto-oscillate back and forth. Heat is thus passively transferred, not only by latent heat exchange like in conventional wicked heat pipes, but also by sensible heat transfer between the wall and the fluid. No external power is required to drive the device, internal oscillations being fully driven by thermal non-equilibrium existing which is sustained inside the system due to external heating and cooling.

In the first part of the presentation, a review of the experimental and modeling methodologies to predict the behavior of a pulsating heat pipe will be presented. Compared to other cooling solutions, PHPs are simple and thus more reliable and cheap. However, there is no theoretical model or correlation that would predict the PHP behavior and heat exchange accurately at present. Moreover, the review clearly highlights the fact that the present understanding is rather insufficient for framing comprehensive models. This prevents the PHPs from being commonly used industrially. Indeed, reliable design tools can only be formulated if the nuances of its operating principles are well understood.

In the second part of the presentation, we will make an attempt to explain the self-sustained thermally-induced oscillations of a two-phase system consisting of an isolated confined liquid-vapor meniscus (a single liquid plug adjoining a vapor bubble) inside a circular capillary tube. The tube length being exposed to a net temperature gradient, a continuous cycle of evaporation and condensation is created, leading to thermally induced auto-oscillations of the meniscus. This system represents the simplest 'unit-cell' version of a pulsating heat pipe (PHP). Both theoretical and experimental explorations are presented and compared, leading to an enhanced insight into the PHP behavior.

Design of experimental steady state two chambers gasifier

Yehuda Taitel and Dvora Barnea
Tel-Aviv University, Israel
e-mail : taitel@eng.tau.ac.il

An attractive method for obtaining syngas (CO, H_2) from solid biomass (coal, wood, corn straw, sugarcanes, animal biosolids etc) was proposed by Judd et al. (1983,1986,1991) and it is refereed as Judd's two-component gasifier. In the 2 chamber biomass gasifier hot sand circulates between the right and left chambers as shown in the figure 1. Combustion takes place in the right chamber and production of Syngas is obtained in the left chamber. The requirement is that the right chamber will operate as a fluidized bed while the left chamber should move as a packed bed. Finally it is important that the flow of sand through the passages from the left chamber to the right chamber will ensure proper operation.

The present work provides a method of calculating the proper steam and air flow rate, the sand circulation rate as well as the variation of the biomass concentration and temperature along the chambers for a desired biomass flow rate and geometry of the chambers. Figure 2 present a typical result for the variation of the temperature and biomass concentration (Z starts at the top left chamber pointing downwards and ends at the top right chamber).

Fig. 1. Schematic geometry of two chambers gas gasifier

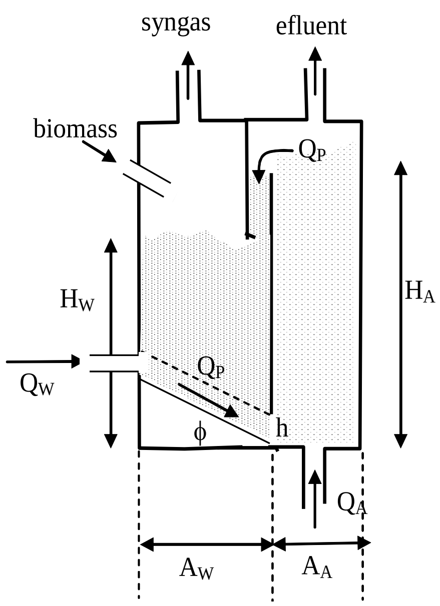
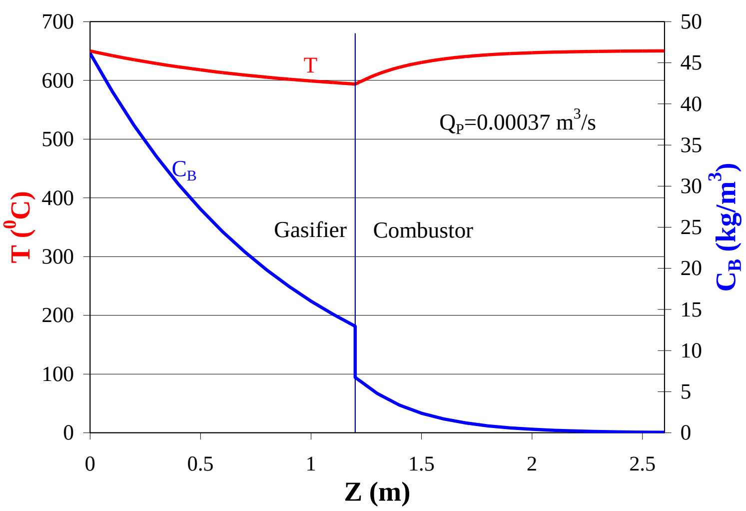


Fig. 2: Concentration of the biomass and the temperature along the 2 columns



References :

Judd, M.R., Masson, H., Meihack, W.F., 1983, Proc. of Fourth Int. Conf. on Fluidization, New York.

Judd, M.R., Rudolph, V., 1986, Proc. of Fifth Int. Conf. on Fluidization, New York.

Judd, M.R., and Pillay, M. 1991, Development of a horizontaly-configured circulating fluidized bed gasifier. J. of Energy R&D in South Africa, 2, 19.

Dynamic Modeling and Experimental Evaluation of Two-Phase On-Chip Cooling of Parallel Pseudo-CPUs

N. Lamaison, J. Marcinichen and J.R. Thome
EPFL, Lausanne, Switzerland
e-mail : John.Thome@epfl.ch

On-chip two-phase cooling of parallel pseudo-CPU's integrated into a liquid pumped cooling cycle (Figure 1) is modeled and experimentally verified. The system's dynamic operation is very relevant since the heat dissipated by microprocessors is continuously changing and critical heat flux conditions in the micro-evaporators (ME1 and ME2) must be avoided by flow control during heat load disturbances [1] or by designing for the worst case of ensuing flow maldistribution. The purpose here is to cool down multiple microprocessors in parallel and their auxiliary electronics (memories, DC/DC converters, etc.) to emulate servers with multiple CPUs. The dynamic simulation code was benchmarked using the test results obtained in the experimental facility, and was evaluated under steady state and transient conditions of balanced and unbalanced heat fluxes on the two pseudo-chips. The errors in the model's predictions of mean chip temperature and exit mixed vapor quality at steady state remained within ±10% [2]. Transient comparisons showed that the trends and the time constants were satisfactorily respected (Figure 2) [2]. A case study considering four microprocessors cooled in parallel flow was then simulated for different levels of heat flux (40, 30, 20, 10 W cm^{-2}), which showed the robustness of the predictive-corrective solver used. For a desired exit mixed vapor quality of 30%, an inlet pressure of 16 bar and an inlet subcooling of 3 K, the resulting mass fluxes in the microevaporators were 47, 53, 76 and 116 kg m^{-2} s^{-1} and yielded approximately uniform chip temperatures (maximum variation of 2.6, 2.0, 1.7 and 0.7 K). The predicted mass flow distribution was further investigated by comparing experimentally observed mass flow distribution with predicted ones with very satisfactory results.

Figure 1: Liquid Pumping On-Chip Cooling Cycle

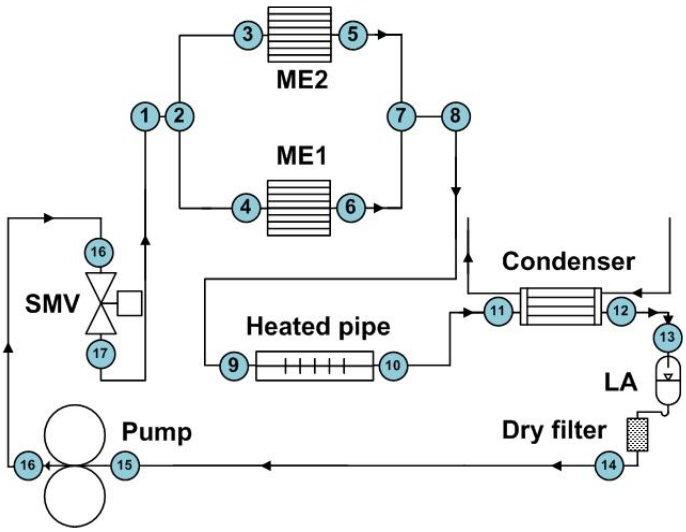
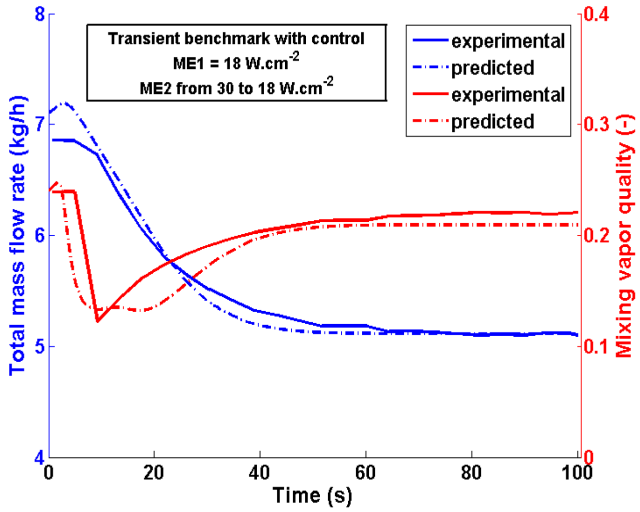


Figure 2: Transient benchmark of the dynamic modeling



References :
[1] Marcinichen, J. B., Olivier, J. A., Oliveira, V. and Thome, J. R., *A Review of On-Chip Micro-Evaporation: Experimental Evaluation of Liquid Pumping and Vapor Compression Driven Cooling Systems and Control*, International Journal of Applied Energy 92, 147-161, 2011
[2] Lamaison, N., Marcinichen, J.B. and Thome, J.R., *Two-Phase Flow Control of Electronics Cooling with Pseudo-CPUs in Parallel Flow Circuits: Transient Modeling and Experimental Evaluation*, Journal of Electronics Packaging, ASME, under review, 2013.

Flow boiling of alternative refrigerants at high temperatures in a horizontal smooth tube: experimental investigation on the effect of reduced pressure on adiabatic frictional pressure gradients

S. Grauso[1], R. Mastrullo[1], A. W. Mauro[1], G. P. Vanoli[2]
[1] Federico II University of Naples, Dept. of Industrial Engineering, p.le Tecchio, 80, 80125, Napoli, Italy
[2] University of Sannio, Engineering Dept., corso Garibaldi 107, Palazzo dell'Aquila Bosco Lucarelli, 82100, Benevento, Italy
e-mail : rita.mastrullo@unina.it

Flow boiling characteristics of alternative refrigerants are fundamental for the proper sizing of evaporators in both vapor compression systems and Organic Rankine Cycles (ORCs). The renewed interest for CO_2 as working fluid, in particular, for air-conditioning and heat pump systems, led to the deepening of the heat transfer characteristics at working temperatures typical for those applications, for which this fluid presents high values of reduced pressure due to its low critical temperature. Also, the diffusion of ORCs for the use of renewable energy sources at low-grade temperatures (100-200°C), lead to work with organic fluids (like refrigerants, as R410A, R134a, R236fa, R245fa) at medium-high reduced pressures during the evaporation process.

For those conditions, both CO_2 and those refrigerants at high reduced pressures have thermodynamic properties more favorable than refrigerants at low temperatures, as higher vapor density and liquid thermal conductivity, lower surface tension, liquid-to-vapor density ratio and liquid viscosity.

Several works can be found in literature on flow boiling characteristics in several ranges of reduced pressure: this parameter is found to affect highly, directly and indirectly, properly for the influence on thermodynamic properties, the heat transfer characteristics during flow boiling. In particular, in some recent works its effect was specifically investigated for flow regimes and their transitions [R. Mastrullo, A.W. Mauro, J.R. Thome, D. Toto, G.P. Vanoli, Flow pattern maps in convective boiling of CO_2 and R410A in a horizontal smooth tube: experiments and new correlations analyzing the effect of the reduced pressure, Int. J. Heat and Mass Transfer 55 (2012) 1519–1528] and local heat transfer coefficients [S. Grauso, R. Mastrullo, A.W. Mauro, G.P. Vanoli, Flow boiling of R410A and CO_2 from low to medium reduced pressures in macro channels: experiments and assessment of prediction methods, Int. J. Heat and Mass Transfer 56 (2013), pp. 107-118].

In this work the results of an experimental investigation on two-phase adiabatic frictional pressure gradients in a horizontal smooth tube of 6.00 mm of inner diameter are presented. The experimental investigation was conducted for CO_2 at saturation temperatures typical of the functioning of evaporators in air-conditioning and heat pumps systems, i.e. 7°C and 12°C, for which correspond reduced pressures of 0.57 and 0.64 respectively.
In order to extend the investigation of the effect of reduced pressure in a wider range and evaluate its effect for a different refrigerant up to high temperatures, the experiments were performed also for R410A from saturation temperatures from 5°C to 42°C, at which the reduced pressure varies from 0.19 to 0.52. For both fluids the mass flux was ranged between 150 to 500 kg $m^{-2}s^{-1}$, obtaining 1214 experimental data.

Physical meaning of the pressure drop decomposition in two-phase flows

Stéphane Lips[a,b,c], Remi Revellin[a,b,c]
[a]Université de Lyon, CNRS
[b]INSA-Lyon, CETHIL, UMR5008, F-69621, Villeurbanne, France
[c]Université Lyon 1, F-69622, France
e-mail : stephane.lips@insa-lyon.fr

It is commonly admitted in the literature that the total pressure drop, ΔP, is the sum of three different terms: the gravitational pressure drop, the momentum pressure drop, and the frictional pressure drop. The gravitational pressure drop is a consequence of the inclination angle of the tube. The frictional pressure drop is due to the shear stress between the fluids and the wall. The momentum pressure drop depends on the variation of kinetic energy of the fluids in the tube.

In single phase flow, the pressure drop decomposition can directly be linked to the definition of the total pressure $P_{tot} = P + P_{grav} + P_{dyn}$ where P is the thermodynamic pressure (also called static pressure), P_{grav} is the gravitational pressure and P_{dyn} is the dynamic pressure. An entropy analysis shows that the irreversible pressure drop, i.e. the frictional pressure drop, is equal to the variation of the total pressure.
For two-phase flows with phase change, the notion of total pressure is no more valid as the equivalent density of the fluid changes continuously. However, authors in literature often use the pressure drop decomposition, either to correct their experimental data from a kinetic or a gravitational energy variation or to develop predictive models that take into account different sources of pressure variation.

Two different approaches can be found in the literature to define this decomposition. On the one hand, the momentum conservation principle is used. It leads to a pressure drop decomposition based on a two-phase equivalent density calculated by means of a weighted sum of the liquid and vapour densities, the weights being the void fraction and the liquid hold-up. On the other hand, the second method is based on the kinetic energy theorem. The decomposition depends on a two-phase specific volume calculated by means of a weighted sum of the liquid and vapour specific volumes, the weights being the vapour and liquid qualities.

In the present communication, the differences between the two approaches are highlighted in the case of a two-phase flow with a slip ratio different than one. It is shown than the first approach, which is the mostly used in the literature, leads to a pressure drop decomposition with a low physical meaning. On the contrary, the pressure drop decomposition based on the second approach is coherent with an entropy analysis: the frictional pressure drop is an image of the irreversibilites in the flow and the kinetic and gravitational pressure drops reflect reversible phenomena.

In the last part of the communication, the need of working with the correct pressure drop decomposition is emphasized by applying the two different approaches in the study of experimental pressure drops obtained for a condensing flow in an inclined tube. It is shown that a better understanding of the physical meaning of each pressure drop decomposition is necessary to achieve the development of predictive models for two-phase flows.

Two Phase Liquid-Liquid Flow in Pipe Bends

Y. Kaplan, A. Ullmann and, N. Brauner
School of Mechanical Engineering, Tel-Aviv University,
Tel-Aviv 69978, Israel
e-mail : ullmann@eng.tau.ac.il

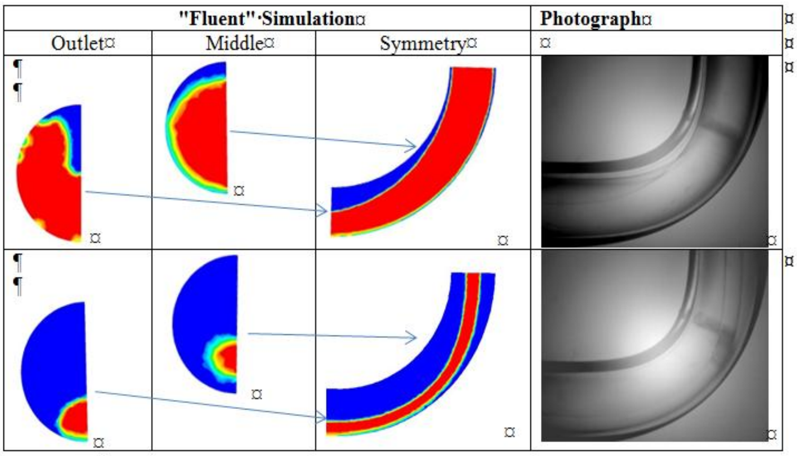
The centrifugal force acting on a fluid in a pipe bend is proportional to the density of the fluid. It is thus reasonable to assume that in two phase flow the heavy phase would be forced to the outer wall of the pipe at the bend. However, this is not always the case. Depending on the physical properties of the fluids, for certain ranges of liquid and gas flow rates, the light phase is forced to the outer wall. The magnitude of the slip velocity between the two phases is at times large enough that the greater centrifugal force acting on the faster light phase overcomes the density difference effect. This phenomenon was first experimentally observed in studies concerning gas-liquid flow in helical tubes and named 'film inversion' [1].

In the current study the phenomena was investigated in liquid-liquid flow with very low density difference (less than 1%). The experimental test section consists of 2 mm inside diameter glass pipe. The tested flow rates of the two liquid phases correspond to laminar flow. The flow upstream the bend is co-current down-flow, where in most cases the flow pattern was core- annular flow. The downstream section of the bend is horizontal. The flow pattern through bends was studied experimentally by photographing the flow through pipe bends at varies flow rates. In addition the flow was modeled using the 'Fluent' software. The simulations were also used to study the effect of density and viscosity on the phenomena.

A demonstration of a comparison between the results of the numerical simulation of the flow to the flow pattern observed in the test section is shown in Figure 1. While the simulation provides the flow pattern in the cross section, the photographs show the flow as viewed by an observer from the side of the bend. The comparison suggests that side view observation is insufficient for characterizing the flow pattern and the film inversion.

A mechanistic criterion for the occurrence of film inversion during the flow through the bend is suggested and compared with the numerical and experimental results.

Figure 1: Comparison between experimental results and numerical simulations (in which the red depicts the light phase).



References:
[1] Sanjoy Banergee, Edward Rhodes and D.S. Scott, " Film inversion on concurrent two-phase flow in helical coils." A.I.Ch.E. Journal, 13, 1 (1967)
[2] Amos Ullmann, Sharon Gat, Zvi Ludmar, Neima Brauner, "Phase separation of Partially Miscible Solvent Systems: Flow Phenomena and Heat and Mass Transfer Applications." Reviews in Chemical Engineering, 24 (4-5), 159-262 (2009)

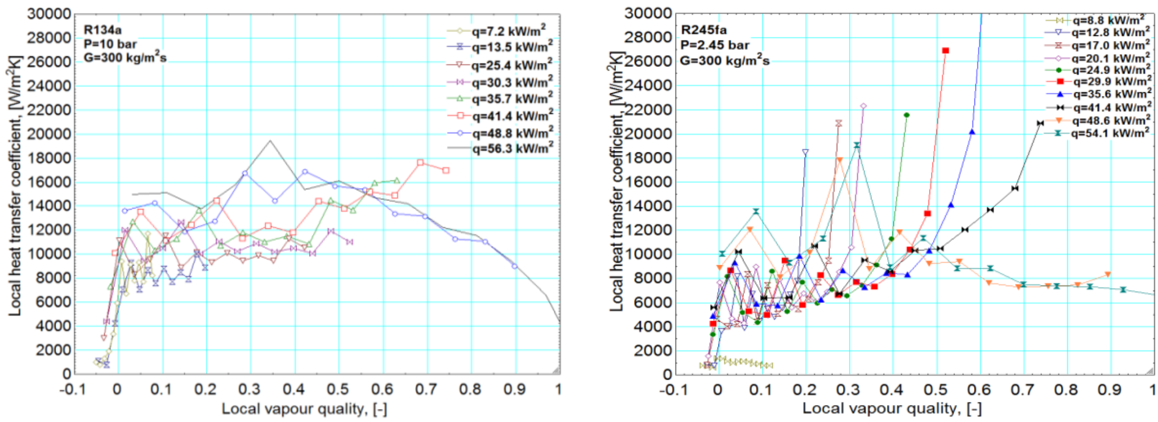
Flow boiling patterns, pressure drop and heat transfer rates of R134a and R245fa in micro tubes

E. A. Pike-Wilson*, M. M. Mahmoud** and T. G. Karayiannis*
** School of Engineering and Design, Brunel University London, Uxbridge, UB8 3PH*
*** Faculty of Engineering, Zagazig University, Egypt*
e-mail : tassos.karayiannis@brunel.ac.uk

An experimental investigation on two phase flow boiling patterns, pressure drop and heat transfer rates was carried out with R134a and R245fa in a vertical stainless steel test section of 1.1 mm internal diameter. R245fa is a low pressure refrigerant with different thermophysical properties to R134a. In particular, the surface tension, liquid dynamic viscosity and vapour density are different and these can affect the heat transfer rates and pressure drop significantly. The experimental conditions included a mass flux range 200-400 kg/m^{2}s, heat flux range 3-125 kW/m^2, inlet saturation temperatures of 31°C and 35°C which correspond to pressures of 8 and 10 bar and 1.8 and 2.4 bar for R134a and R245fa respectively and inlet subcooling of 5K. The flow boiling patterns were recorded using a high-speed high-resolution camera at the exit of the test section through a borosilicate glass tube.

The flow visualization results for R134a for increasing heat flux, revealed clearly defined flow patterns, i.e. bubbly, slug, confined bubble, churn and annular. These were not evident in similar experiments, again for increasing heat flux, with R245fa. For this refrigerant, there were not any clearly defined intermediate flow patterns between single phase flow and annular flow with increasing heat flux. However, when the experiments are conducted under decreasing heat flux, the same clearly defined flow patterns as R134a were seen for R245fa. The local heat transfer coefficient was plotted as a function of vapour quality and distance along the tube in order to compare the results for the two refrigerants, see fig. 1.

Figure 1. Heat transfer coefficients for (a) R134a and (b) R245fa at Tsat=39°C.



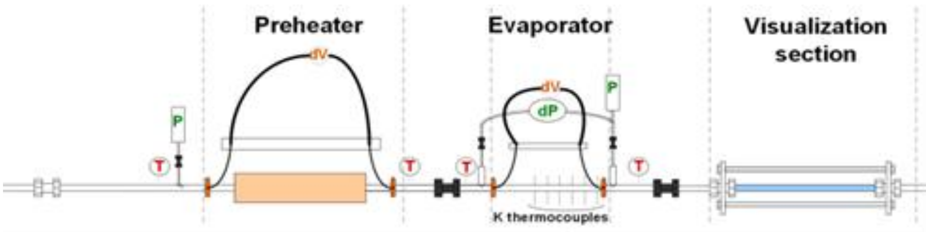
R245fa is shown to have a greater dependence on vapour quality, with the heat transfer coefficient increasing sharply beyond a certain heat flux while the heat transfer coefficient remains relatively flat with increasing vapour quality for R134a. Our results were also compared with existing heat transfer correlations and there was better agreement with R134a than with R245fa. This is mainly due to the different heat transfer rate trends of R245fa, i.e. the sharp increases in heat transfer coefficient as seen in fig. 1. The pressure drop showed the same trend for both refrigerants, increasing with heat flux, but with R245fa at an increased rate. The pressure drop for R245fa was up to 300% higher than R134a for the same inlet saturation temperature and mass flux. The past pressure drop correlations were also shown to predict the results for R134a better than those of R245fa.

Experimental investigation of R-245fa flow boiling in minichannels for different flow conditions

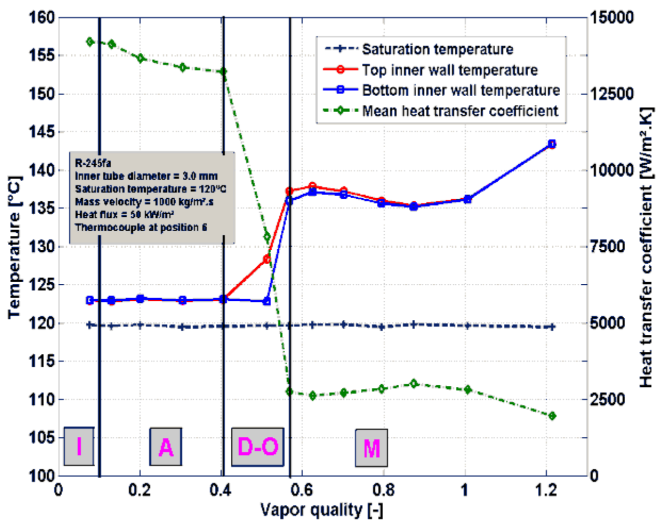
Romain Charnay*, Rémi Revellin* and Jocelyn Bonjour*
** INSA-Lyon, CETHIL, CNRS, UMR5008, F-69621, Villeurbanne, France*
Université Lyon 1, F-69622, France
e-mail : romain.charnay@insa-lyon.fr

Due to 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. Nevertheless, 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}).

A test rig was built to study the two-phase thermohydraulic behaviour of synthetic refrigerants in minichannels at high temperature, i.e. from 60 to 120°C. The tests were run in 3.0 mm inner tube diameter using R-245fa. The mass velocity ranges from 100 to 1500 $kg/m^2.s$, the heat flux from 10 to 90 kW/m^2 and the vapor quality from 0 to 1. The test section consisted of three parts: (i) a 2000 mm spirally stainless steel tube (called preheater), (ii) a 185 mm stainless steel tube (called evaporator) and (iii) a glass tube (called visualization section).



Twelve K-thermocouples were scotched at six positions along the channel. Six thermocouples are placed at the top of the evaporator and six at the bottom. The figure herebelow gathers measurements obtained for each operational condition, i.e. the wall temperature at the top and the bottom, the local heat transfer coefficient and the flow pattern. Four flow patterns have been characterized: Intermittent, Annular, Dry-out and Mist flow. The influence of the flow parameters on the heat transfer and the flow patterns has been investigated.



Keywords : Minichannel, Flow patterns, Two-phase flows, Heat transfer coefficient, Dry-out

Micro particle shadow velocimetry (μPSV) for air-water flow analysis in capillaries

S. Khodaparast, N. Borhani, J. R. Thome
Laboratory of Heat and Mass Transfer (LTCM), EPFL, Lausanne, Switzerland
E-mail : John.Thome@epfl.ch

We studied the motion of isolated air bubbles moving with constant velocity U_b in capillaries filled with water flowing at a mean velocity of U_l. Dynamics of such flows has always been of great interest in many different practical applications e.g., bubble flow meters, extraction of oil through porous media in oil industries, low-volume chemical reactors with high mixing characteristics and heat exchangers. Moreover, modeling the complicated physical phenomena involved in such flows has become a challenging task when the bubble shape and flow dynamics around it are to be calculated numerically. In this study, the new μPSV technique [1] is employed to investigate flow of the continuous water phase surrounding the air bubble in fluorinated ethylene propylene (FEP) capillaries of 0.5 and 1.0 mm diameters. The tube is submerged in a water bath with a flat surface facing the camera in order to eliminate the optical distortions caused at the curved walls, see Fig. 1. Accurate flow rate of the liquid phase is measured integrating the fully-developed downstream velocity measurements, which makes the flow loop independent of a flow meter. Confinement effects are studied by changing the bubble volume at each flow rate. Sharp interfaces in between the phases are created on the images containing particle shadows, thanks to the back-lit configuration of the optical system, see Fig. 2. These images are then used for measurement of bubble volume, liquid film thickness, bubble nose and tail curvature and bubble velocity using the recently introduced time strip method [2] while particle images are processed using the cross-correlation technique to obtain the velocity field around the bubble in the liquid phase [3]. For air-water flows in small tubes where effect of gravity forces and viscosity ratios can be neglected, the measured parameters are correlated with the flow Capillary number and the velocity ratio U_b/U_l. Finally, the measurements are compared to the available results obtained in previous experimental and numerical studies. It is believed that the precise quantitative results achieved in this study can be used for benchmarking numerical codes in the range of tested parameters. Moreover, compared to the complicated combined μPIV and shadowgraphy systems [4], the new optical technique used in this study is more efficient, less expensive and easier to implement.

Figure 1. Schematic of the experimental facility.

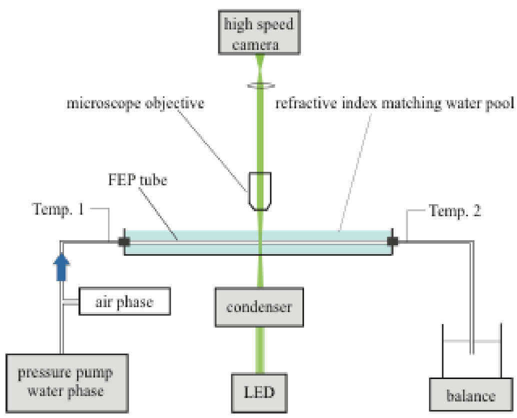
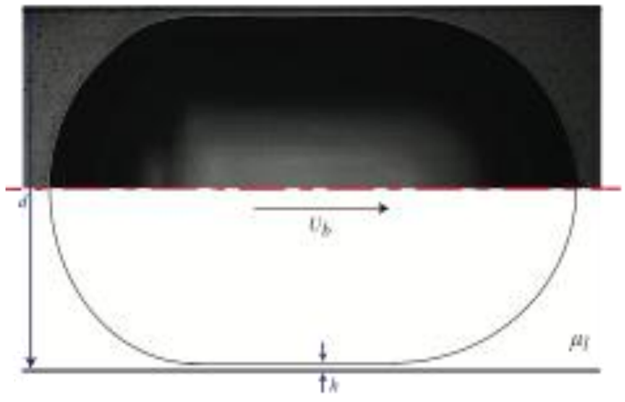


Figure 2. Image of a confined air bubble and its identified interface moving at Ca = 0.004 in the water filled 1 mm tube.



References :

[1] Khodaparast S, Borhani N, Tagliabue G, Thome JR (2013) A micro particle shadow velocimetry (μPSV) technique to measure flows in microchannels, *Exp. Fluids* 54: DOI 10.1007/s00348-013-1474-x

[2] Borhani N, Agostini B, Thome JR (2010) A novel time strip flow visualisation technique for investigation of intermittent dewetting and dryout in elongated bubble flow in a microchannel evaporator, *Int. J. Heat Mass Transfer* 53: 4809-4818

[3] Willert CE, Gharib M (1991) Digital particle image velocimetry. *Exp. Fluids* 10:181- 193

[4] Nogueira S, Sousa RG, Pinto AMFR (2003) Riethmuller ML, Campos JBLM, Simultaneous PIV and pulsed shadow technique in slug flow: a solution for optical problems, *Exp. Fluids* 35: 598-609

Visual study and heat transfer of flow boiling in a transparent tube

G.P. Celata, F. D'Annibale, L. Saraceno, C. M. Valencia-Castillo, G. Zummo
ENEA, Italy
e-mail : celata@casaccia.enea.it

An experimental campaign has been performed to investigate flow boiling heat transfer to a dielectric fluid in a vertical transparent tube. The tube is made of pyrex, 4.0 mm in diameter with a transparent heater coated on the outer surface. The heater is made of ITO (Indium-Tin Oxide) and the fluid, FC-72, flowed upward. High-speed visualizations up to 5,000 fps has been performed simultaneously with wall temperature measurements to analyse flow patterns, bubble characteristics, and heat transfer. The visualization method allows to obtain detailed observations of the vapour-liquid flow structures along the heated channel and to measure some parameters of two-phase flow phenomena (bubble formation, taylor bubble length and frequency, bubble velocity etc.). The experiments have been conducted for a mass flow rate 185 kg/m^2-s, inlet subcooling down to -20 K, and inlet pressure up to 1.8 bar.

The presentation will be focused on the detailed description of the vapour bubble parameters in the flow regime of bubbly and slug flow. The results of the quantitative analysis of images will be presented together with the heat transfer data. High-speed video of flow pattern transitions will be shown.

Numerical calculations of two – dimensional temperature field in a vertical minichannel

Sylwia Hożejowska[1], e-mail: ztpsf@tu.kielce.pl
Robert Kaniowski[1], e-mail: kaniowski@tu.kielce.pl
Mieczyslaw E. Poniewski[2], e-mail: meponiewski@pw.plock.pl
[1]Kielce University of Technology, Al.1000-lecia P.P.7, 25-314 Kielce, Poland
[2]Warsaw University of Technology, Plock Campus, ul. Lukasiewicza 17, 09-400 Plock, Poland

The paper presents the application of the Trefftz method to determining the approximated two-dimensional temperature field of a boiling refrigerant (FC-72) flowing in a vertical minichannel. The numerical calculations were based on the results of the experimental studies. The vertical measurement module with the minichannel of a rectangular cross-section is the main element of the tests stand. One of the minichannel walls, the heating foil made of Haynes-230 alloy is supplied with regulated DC power. A thin film of thermosensitive liquid crystals sprayed over the heating foil allows obtaining two-dimensional distribution of surface temperature. The heating foil is isolated from the external environment by a glass pane. Another glass pane, isolating the minichannel from the other side, makes it possible to observe the two-phase flow structures and to measure the void fraction. The refrigerant of temperature lower than the saturation temperature flows under elevated pressure into the minichannel. During the flow, the liquid is heated by the heating foil whose temperature exceeds the saturation temperature of the liquid. The temperature of the resulting mixture and its pressure are measured at the minichannel outlet. Other measurements include voltage and amperage in the foil and the flow rate in the minichannel.

Numerical calculations are conducted for the central part of the module (along its height) so that the phenomena occurring at the side edges of the minichannel do not affect the thermodynamic parameters in the investigated segment.

To determine the two-dimensional temperature distribution for the refrigerant, the Trefftz functions were derived for the equation of energy conservation at the parabolic profile of the liquid velocity. The unknown temperature of the liquid was approximated with a linear combination of the derived Trefftz functions. The combination coefficients were determined by minimizing the functional that adjusted by least squares the approximation to the adopted boundary conditions including the experimentally determined void fraction. The Trefftz method and harmonic functions were used to determine the two-dimensional temperature distribution of the heating foil and the glass pane. He known temperature of the liquid and the foil allowed calculate the heat transfer coefficient at the heating foil-boiling liquid contact.

The results were then compared with the results obtained for the simplified model, in which it was assumed that the heat flux generated in the foil is constant along its surface. The resultant simplified energy equation was solved with the Trefftz method combined with the inverse operations method. Both approaches gave analogous results.

Keywords**:** flow boiling, minichannel, Trefftz method, heat transfer coefficient

Numerical simulation and experimental investigation of two-phase flows in Y-shaped mircochannels

S. Mosler[a], M. Hoffmann[a], M. Schlüter[a]
[a]*Institute of Multiphase Flows, Hamburg University of Technology (TUHH), Germany*
e-mail : stefan.mosler@tu-harburg.de

The relevance of microchannels in process engineering has grown rapidly in the last decade. The reasons are the high ratio between surface and volume and the possibility to adjust exactly contact times and residence time distributions. Therefore chemical reactions can be controlled more exactly to achieve higher yields and selectivities. However the modeling, simulation and validation of hydrodynamics and mass transfer is still challenging due to the strong influence of interphase phenomena in this small dimensions e.g. 300 µm (diameter of the microchannel). Especially the contact angle affects strongly the hydrodynamics and mass transfer conditions and has rarely taken into account so far. In most cases numerical and experimental analysis of hydrodynamics and mass transfer are examined separately. Many publications are dealing with gas liquid systems e.g. raising bubbles [1].

One goal of this work is to get a deeper insight into the local hydrodynamic flow and mass transfer conditions in microchannels in dependency of the contact angle. Therefore numerical simulations have been performed with different contact angle, surface tension and volumetric flow rates. For the simulations a common single field model is used which is implemented in OpenFOAM [2]. As mass conservation is important to calculate exactly mass transfer a Volume of Fluid method is used to capture the interface [3]. With the help of such a model numerical simulations of the local hydrodynamic conditions have been done for different conditions and validated by experiments in microchannels. For the validation two different kinds of Y-Y-shaped micro channels were fabricated at the institute for microsystem technology at the Hamburg University of Technology. A high speed camera was used to measure slug velocity and slug length.

First a straight line reactor was examined. 2D and 3D simulations are compared to each other. It has been shown that 2D and 3D simulations differ in slug formation and slug length. 3D Simulations of the creation of a slug flow in microchannels will be shown and a break up mechanism will be presented. The slug length and slug velocity are used as a first validation criteria. It will be shown that the numerical and experimental results agree in a reasonable way. Furthermore the simulations are giving a deeper insight into different kinds of internal circulations inside a slug that are depending on the contact angle.

References :
[1] Onea, A., Wörner, M. & Cacuci, D. G., (2009). A qualitative computational study of mass transfer in upward bubble train flow through square and rectangular mini-channels. Chemical Engineering science, **64**, 1416-1435
[2] Rusche, H. (2003). Computational fluid dynamics of dispersed two-phase flows at high phase fraction. PhD thesis, Imperial College of Science, Technology & Medicine.
[3] Hirt, C. W. & Nichols, B. D. (1981). Volume of Fluid (VOF) Method for the Dynamics of Free Boundaries. Journal of computational physics, **39**, 201-225

Flow Topology of Hydrothermal Waves in Liquid Bridges and its Relation to Dissipative Particulate Structures

Hendrik C. Kuhlmann[1] and Roman Mukin[2]
[1] *Vienna University of Technology, Resselgasse 3, 1040 Vienna, Austria,* h.kuhlmann@tuwien.ac.at
[2] *Paul Scherrer Institute, 5232 Villingen PSI, Switzerland,*
e-mail : roman.mukin@tuwien.ac.at

The thermocapillary flow in a differentially heated cylindrical liquid bridge is investigated. For transparent liquids with Pr > 1 the flow typically arises in form of an azimuthally traveling hydrothermal wave if the driving force (axial temperature difference) is increased beyond a critical threshold. We analyze the three-dimensional flow structure of the wave in a frame of reference in which the wave is steady. High-resolution numerical calculations show that regular and chaotic streamlines co-exist. The KAM tori of the regular motion approach the free surface very closely. This feature has implications on the motion of suspended particles, in particular, on density-matched particles.

Hofmann and Kuhlmann (2011) and Muldoon and Kuhlmann (2013) have shown that density-matched particles can de-mix and form particulate structures due to a particle–free-surface interaction which restricts the motion of suspended particles near the free surface. Experiments indicate that the dissipation for the particle motion is almost entirely located in a thin sub-surface layer. When the dissipation is modeled by a hard-wall for the particle motion we find a rapid de-mixing of particles into so-called Particle Accumulations Structures (PAS) (Schwabe et al., 2007). The numerically obtained structures are very similar to the experimentally observed ones. Moreover, we predict a new form of period-doubled PAS.

Support from ESA under contract number 4000103003 is gratefully acknowledged.

References :

D. Schwabe, A. I. Mizev, M. Udhayasankar and S. Tanaka (2007) *Formation of dynamic particle accumulation structures in oscillatory thermocapillary flow in liquid bridges*, Phys. Fluids, 072102.
E. Hofmann and H. C. Kuhlmann (2011) *Particle accumulation on periodic orbits by repeated free surface collisions*, Phys. Fluids, 0721106.
F. H. Muldoon and H. C. Kuhlmann (2013) *Coherent particulate structures by boundary interaction of small particles in confined periodic flows*, Physica D (to appear).

Dynamic modeling of a two-phase flow microchannel electronics cooling system

T. Saenen, T. Baelmans and J.R. Thome
EPFL, Lausanne, Switzerland
e-mail : John.Thome@epfl.ch

Two-phase microchannel cooling is a promising high-performance cooling technique to cool next-generation electronic devices. The design of such a cooling system is nontrivial and its often complex dynamic behavior needs to be correctly modeled [1, 2]. To facilitate this design process, a dynamic system model of a two-phase microchannel cooling system was constructed [3]. The model numerically solved the transient 1D mixture mass, momentum and energy conservation equations using techniques borrowed from the CFD research community. Advanced non-uniform Total Variation Diminishing spatial discretization schemes was used to avoid spurious oscillations in the model's solutions associated with the strong gradients in the flow variables. Additionally, the accumulator and the conduction in the microchannel heat sink and heat exchanger were dynamically simulated as these components govern various important dynamic phenomena of the complete cooling system. The code of the model was rigorously verified using the well-suited method of manufactured solutions [3]. As a first application, the system model was used to investigate the performance of small, portable two-phase electronics cooling systems [4]. In particular, the influence of accumulator size on system performance was examined in detail (Figure 1). Analytical models of the size effects and design equations were obtained to incorporate these effects early in the design process. Finally, experimental validation of the numerical model and the size effects was performed with two separate test facilities focused on steady-state results. The two-phase microchannel test facility at Purdue University was used to validate the heat sink results of the system model and a purposefully built test facility at the KU Leuven was used to validate the size effects. Both the numerical system model and the analytical size effect models corresponded well with the experimental results (Figure 2).

Figure 1: Size effects portable cooling system

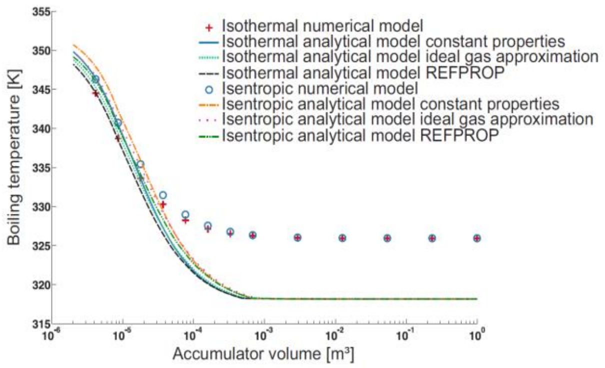
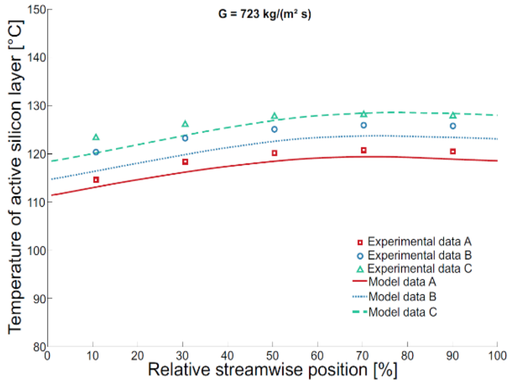


Figure 2: Two-phase heat sink validation



References :

[1] Marcinichen, J. B., Olivier, J. A., Oliveira, V. and Thome, J. R., *A Review of On-Chip Micro-Evaporation: Experimental Evaluation of Liquid Pumping and Vapor Compression Driven Cooling Systems and Control,* International Journal of Applied Energy 92, 147-161, 2011.

[2] Saenen, T., Baelmans, T. (sup.), *Modeling a Two-phase Microchannel Electronics Cooling System,* KU Leuven, Leuven 2013.

[3] Saenen, T., Baelmans, T., *Numerical model of a two-phase microchannel heat sink electronics cooling system,* International Journal of Thermal Sciences 59, 214-223, 2012.

[4] Saenen, T., Baelmans, T., *Size effects of a portable two-phase electronics cooling loop*, Applied Thermal Engineering, 50, 1174-1185, 2013.

Experimental and numerical investigation of condensation inside minichannels

Davide Del Col*, Stefano Bortolin
University of Padova, Dipartimento di Ingegneria Industriale
Via Venezia 1, 35131 Padova, Italy
e-mail : davide.delcol@unipd.it

An experimental and numerical investigation on condensation heat transfer inside a single square cross section minichannel is here presented. Local heat transfer coefficients during condensation of R134a, R32 and R1234ze(E) are measured in a 1.23 mm hydraulic diameter horizontal square channel at 40° saturation temperature. The effect of channel orientation has been studied with refrigerants R134a and R32 in 30°, 45°, 60°, 90° downflow and upflow configurations. The experimental technique adopted here is presented in Del Col et al. [1]. The mass velocity ranges between 100 and 400 kg m^{-2} s^{-1} for all tested fluids and the vapor quality varies between 0.9 and 0.2. These fluids present different thermodynamic and thermophysical properties, in terms of saturation pressure, vapor density, liquid viscosity, surface tension and thermal conductivity.

In the literature the number of local heat transfer coefficient values measured during condensation inside non-circular minichannels is rather limited and the effect of channel orientation during condensation is not much investigated. The relative importance of shear stress, gravity and surface tension may depend on operating conditions and orientation. In a square minichannel, due to the surface tension effect, the liquid is pulled towards the corners leading to a thinner liquid film on the flat sides and therefore to a lower thermal resistance on these parts of the channel. This effect may enhance the heat transfer in the presence of corners, as compared to the case of circular minichannels, at low mass velocity, when the relative importance of shear stress diminishes (Del Col et al. [1]).

From the experimental results, the effect of the channel inclination on the heat transfer coefficient at varying mass velocity and vapor quality is investigated. For R134a and R32, accounting for the experimental uncertainty, no difference has been noticed for the different configurations at mass flux above 200 kg m^{-2} s^{-1}, whereas the condensation heat transfer is controlled by the shear stress.

When experimental tests are performed at a mass velocity lower than 200 kg m^{-2} s^{-1} the influence of the channel inclination becomes evident in downflow configuration and its effect becomes more important as the vapor quality decreases. The value of the mass velocity at which the condensation heat transfer is affected by channel orientation is found to be dependent on the working fluid. A dimensionless parameter (Taitel and Dukler [2]) has been used to predict under what conditions inclination affects the condensation heat transfer coefficient.

For a better understanding of the forces governing condensation inside the present geometry, a number of steady-state simulations of condensation of R134a inside a square cross section minichannel (1 mm hydraulic diameter) are here proposed. The VOF (Volume Of Fluid) method is used to track the vapor-liquid interface and the effects of interfacial shear stress and surface tension are both taken into account. The minichannel wall is isothermal and a uniform temperature of 30°C is fixed as boundary condition, while the saturation temperature of the fluid is 40°C. The inlet thermodynamic vapor quality is equal to one. Depending on the mass velocity conditions, the liquid film is assumed laminar or turbulent while the vapor flow is turbulent for all the mass velocities considered in the present work. A detailed discussion on this topic, is reported in Da Riva et al. [3]. Condensation of R134a in the square minichannel has been simulated under horizontal and vertical downflow conditions at mass velocity G=400 kg m^{-2} s^{-1} and lower values.

References :

[1] Del Col, D., Bortolin, S., Cavallini, A., Matkovic, M., Effect of cross sectional shape during condensation in a single square minichannel, Int J. of Heat and Mass Transfer, 54, 3909-3920 (2011).

[2] Taitel, Y., Dukler, A.E., A model for prediction flow regime transitions in horizontal and near horizontal gas-liquid flow, AIChE Journal, 22 (1), 47-54 (1976).

[3] Da Riva, E., Del Col, D., Garimella, S.V., Cavallini, A., The importance of turbulence during condensation in a horizontal circular minichannel, Int. J. Heat Mass Transfer, 55, 3470-3481 (2012).

Break-up of small inertial aggregates in turbulent channel flow

Marco Svettini [1], Cristian Marchioli [1 2], and Alfredo Soldati [1 2]
1Universit`a di Udine, Udine, 33100, Italy (Email: soldati@uniud.it)
2Dept. Fluid Mechanics, CISM, Udine, 33100, Italy
e-mail : soldati@uniud.it

Break-up of small aggregates in turbulence is of high relevance to industrial applications such as processing of colloids and nanomaterials, design and operation of flocculation processes but also to environmental problems such as formation and sedimentation of marine snow. In spite of this practical importance, break-up phenomena are still poorly understood from a fundamental point of view and a basic understanding of break-up dynamics is still lacking. A reason for this is the complex role of turbulence and the way it generates fluctuating stresses to which an aggregate is exposed.

Current analyses in archival literature address the problem for the simplified case of brittle massless aggregates in homogeneous isotropic flow conditions. In particular, the main assumptions are that aggregates are smaller than the dissipative flow scale (pointwise assumption) and that an aggregate breaks instantaneously when the hydrodynamic stress generated by the surrounding fluid, σ, exceeds the critical value required to break that aggregate, σcr. In this work we keep the pointwise assumption but we extend the analysis of break-up phenomena to the case of (i) inertial aggregates in (ii) non-homogeneous anisotropic turbulence, considering both instantaneous break-up and (iii) "ductile" break-up. Ductile break-up is consequence of a non-instantaneous process activated when σ > σcr and occurs when the energy dissipated by the surrounding fluid, computed as E = R ϱ(τ |σ > σcr)dτ where ϱ is the fluid kinetic energy dissipation rate and τ is time, exceeds the critical value required to break that aggregate, Ecr.

We focus on the influence of turbulence on break-up dynamics by performing pseudo-spectral Direct Numerical Simulation (DNS) of fully-developed channel flow at bulk Reynolds number Reb = 2250, corresponding to a friction Reynolds number Re_ = 150, and modelling large swarms of aggregates with a standard Lagrangian Particle Tracking (LPT) approach. From this DNS+LPT database, post-processed statistics are extracted to characterize break-up events at varying aggregate inertia, parameterized here by the Stokes number, St (the dimensionless response time of the aggregate). Following our previous works on particleladen wall-bounded flows (see Soldati & Marchioli, 2009; and references therein), the following values are considered: St = 1, 5 and 25. For comparison purposes, tracer aggregates with St = 0 are also tracked.

In the final paper, we will discuss the behavior of the break-up rate, computed as inverse of the aggregates first exit-time (the time necessary for the energy dissipation along the aggregate trajectory to become large enough to cause break-up). We will also show PDFs of first exit location, given by the wall normal coordinate at which break-up occurs, and of the distance traveled streamwise by the aggregate before breaking. Finally, we will examine the possibility to apply currently-available models, developed for homogeneous turbulent flows and based on an exponential time decay law for the number of aggregates (Babler et al., 2011), to estimate the break-up rate in wall-bounded anisotropic flow.

References :
Babler, M., Biferale, L., and Lanotte A. (2012) Breakup of small aggregates driven by turbulent hydrody-
namic stress, Phys. Rev. E, vol. 85, pp. 025301.
Soldati A., and Marchioli C. (2009) hysics and modelling of turbulent particle deposition and entrainment:
Review of a systematic study, Int. J. Multiphase Flow, vol. 35, pp. 1-13.

Pressure fluctuation reduction in once-through micro evaporators

Cor Rops*[1], Giaco Oosterbaan[,2], Cees van der Geld[2]
[1]*TNO, De Rondom 1, 5612 AP Eindhoven, The Netherlands*
cor.rops@tno.nl, giaco.oosterbaan@tno.nl
[2]*Eindhoven University of Technology, PO Box 513, 5600 MB Eindhoven, The Netherlands*
e-mail : c.w.m.vd.geld@tue.nl

During the last decades fascinating progress has been made in the miniaturisation of micro evaporators. Decreasing the channel diameter leads to higher heat transfer coefficients and an increased surface-to-volume ratio. Additionally, miniaturisation leads to light weight and/or disposable equipment. On the other hand, moving towards small-diameter channels demands extra attention to reliability, fouling and thermo-mechanical management of the system. Apart from that, physical processes limit the maximum achievable heat flux. For instance, if the size is reduced, the pressure drop over a micro channel increases rapidly. Also, the reduced importance of body forces compared to surface forces causes an increased probability of channel blockage by the creation of vapour (Rops et al., 2007). In once-through micro evaporators, explosive vapour bubble growth is observed (Zhang et al., 2005) leading to large pressure fluctuations (Kennedy et al.,2000), flow instabilities (Cornwell and Kew, 1992), as well as possible backflow into the divider manifold (Kandlikar, 2002).

The present investigation explores possibilities to enhance the stability of the once-through micro evaporator by reducing its flow boiling induced pressure fluctuations. The pressure fluctuations are linked to the total pressure drop. The reduction of the momentum pressure drop is explored first by widening the channel, and thus creating space for expansion. Next, a possible reduction of the frictional pressure drop is examined. By a porous wall through which a low supply flow of Nitrogen is applied it is attempted to lift the boiling liquid from the wall creating an inverted annular flow. In case of the conical channel for both the momentum pressure drop, as well as the friction pressure drop a reduction factor is estimated theoretically. Experimental comparison using a five time diameter increase shows that the estimated reduction factor approaches the theoretical derived value for higher water supplies. Finally, the boiling pressure fluctuations are reduced by a factor of ten in the solid conical channel and by a factor of 15 in the porous conical channel. This presumably leads to less backflow and therefore to a better flow control.

References and citations :
Cornwell, K., Kew, P.A. (1992) Boiling in small parallel channels, *Proceedings of CEC Conference on Energy Efficiency in Process Technology*, Athens,Greece
Kandlikar, S. G. (2002) Fundamental issues related to flowboiling in minichannels and microchannels. *Experimental Thermal and Fluid Science*, **26**(2-4), pp. 389–407
Kennedy, J.E., Roach, G.M. Jr., Dowling, M.F., Abdel-Khalik, S.I.; Ghiaasiaan, S.M.; Jeter, S.M.; Quershi, Z.H. (2000) The onset of flow instability in uniformly heated horizontal microchannels. *Journal of Heat Transfer*, **122**(1), p.p. 118-125.
Rops, C.M., Lindken, R., Westerweel, J. (2007) Two phase heat transfer: Two phase flow and instabilities in micro channels. *6th International Conference on Enhanced, Compact and Ultra-Compact Heat Exchangers*, Potsdam, Germany.
Zhang, L., Wang, E.N., Goodson, K.E., and Kenny, T.W. (2005) Phase change phenomena in silicon microchannels. *International journal of heat and mass transfer*, 48, pp. 1572-1582
* Corresponding author

Keywords : Flow boiling, Operation stability, Microfluidics, Microchannels.

The dispersion and two way coupling of small electrically charged particles in stationary isotropic turbulence

John Shrimpton and Gabriel Amine-Eddine
University of Southampton
Southampton, UK
e-mail : john.shrimpton@soton.ac.uk

Particles with a certain range of Stokes numbers preferentially concentrate due to action of turbulent motion and body forces such as gravity are known to influence this process. The effect of electric charge, residing on particles, upon the phenomenon of preferential concentration is investigated. We use direct numerical simulations of one and two way coupled stationary isotropic turbulence over a range of particle Stokes numbers, fluid Taylor Reynolds numbers, and electrical and gravitational particle body force magnitudes, characterized by non-dimensional settling velocities.

In contrast to the gravitational body force, the electrical analogue, acting on an electrically charged particle, is generated by an electric field, which is in turn a function of the degree of preferential concentration. Thus, the electrical body force is created by, and mitigates, preferential concentration. It is seen that charging drastically reduces the radial distribution function values at Kolmogorov scale separations, which gravitational force does not. This implies that charging the particles is an efficient means to destroy small clusters of particles. On incorporating the gravitational force, the amount of charge required to homogenise the particle distribution is reduced. This estimation is corroborated by several different indicators of preferential concentration, and the results also agree reasonably well with corresponding experiments reported in literature.

The presentation will provide an overview of the above results and present new data concerning the kinetic and electrical energy budgets of the particle and fluid phases in one and two way coupled scenarios.

The use of two-phase fluids for secondary refrigerants

Laurence Fournaison
GPAN Irstea
1 rue Pierre Gilles de Gennes, CS10030, F- 92761 Antony
e-mail : Laurence.fournaison@irstea.fr

Environmental constraints related to global warming have led the refrigeration industry to seek and develop refrigerating systems with a reduced impact on the environment. The use of secondary refrigerants is an interesting solution: it makes it possible to contain the primary refrigerant, as it is the secondary fluid that is cooled and then transported towards the places of use. Two-phase (liquid-solid) secondary refrigerants have higher energy efficiency than single-phase fluids thanks to the use of the latent heat of fusion of the phase change material. The interest in using Phase Change Slurry (PCS) media as thermal storage and heat transfer fluids is increasing in air-conditioning and refrigeration applications, PCS systems represent a pure benefit resulting in the increase of thermal energy storage capacity, a high heat transfer characteristics and large possibilities of phase change temperatures that occur generally at atmospheric pressure. Hence, they allow the increase of energy efficiency and reduce the used quantity of thermal fluids. This paper presents an overview of phase change slurries, their physical properties, thermo-physical, rheological, heat transfer properties. It also gives interesting and valuable hints on the choice of the most suitable PCS media for laboratory and industrial applications. Finally, this paper will give a focus on two types of fluids: ice slurries and hydrates slurries...

Keywords : Phase change material PCM; phase change slurry PCS; hydrate slurry; ice slurry.

Experimental Investigations of Influence of Solubility of Gas in Oil on Degassing Phenomena in Two-Phase Flow

K. Zabkova[1*], F.Hatesuer[1], A.Luke[1]
[1] *Institute of Thermodynamics, University of Kassel*
e-mail : zabkova@uni-kassel.de

Physical phenomena occur during multiphase flow within pipelines and have to be considered. Various compositions of the mixture are possible in two-phase systems of gas and liquid phase. In mixtures, the compositions of the gas and the liquid phase are different due to their equations of state in equilibrium (EOS). The effects of the interaction between the phases as well as the pattern of the multiphase flow depend on the concentration within the phases and due to the transport phenomena during the flow. There are two pure separated phases and thus two-phase flow or one phase is completely saturated in the other one and therefore the transition from two-phase flow to one-phase flow is observed. The third alternative is characterised by completely solved gas in liquid and additional amount of gas which is not more possible to solve and thus two-phase flow occurs consists of saturated liquid and gas. This phenomena takes place under metastable conditions and cannot be described by the EOS. It is important to find the criteria of the creation of a nuclei, i.e. a new phase (= nucleation site). The nucleation sites are well known in boiling and condensations systems and they are also important in conveying of multiphase systems. Increasing of pressure supports the solubility (by e.g. a multiphase pump). On the other hand dissolving and degassing processes occur within the multiphase flow in pipelines because of pressure drops. These are caused by curves in the pipelines, the surface of the pipelines, valves, flanges and other similar disturbing subjects. Heterogeneous nucleation is observed. All mentioned impacts on the pressure drops have a strong influence on the amount of dissolved respectively degassed gas.

To investigate and visualize various effects in multiphase flows a transparent model is prepared. The focus is up to causing of pressure drops in a two-phase system and investigating the solving and dissolving effects occurring. Especially the impact of rotational forces and the pressure drops caused by pushing the fluid through narrow gaps are investigated. The processes are recorded on a high-speed camera for further simulation of the multiphase effects.

Mixing at high Schmidt number in a random array of spheres

C. Besnaci, F. Risso, V. Roig
Institut de Mécanique des Fluides de Toulouse, Université de Toulouse and CNRS, Allée du Prof. C. Soula, 31400 Toulouse, France
e-mail : risso@imft.fr

We present experimental results on the mixing of a passive scalar in a flow crossing a random array of spheres. A fluorescent dye having a low molecular diffusivity is injected through a point source in the array (Fig. 1) and we observe by planar laser induced fluorescence the concentration field in a large section transverse to the flow located at different longitudinal positions z from the source. The flow configuration is characterised by a solid volume fraction of 2%, Reynolds numbers of the spheres Re in the range 100 - 1000, a Schmidt number equal to 2000 and a ratio between the injector diameter and that of the spheres equal to 0.1. This study allows us to analyse the fundamental mechanisms controling the mixing in the turbulence inside the array, which is also representative of the agitation in dispersed flows such as swarms of bubbles (Amoura, 2008). Some important results have been obtained about mixing (Besnaci, 2012). They show that a transition exists in the mixing around Re=400. For lower Re the main mechanism of mixing is the direct interaction of the filament of dye with the local deformation of the flow in the vicinity of the spheres. At higher Re, the flow being destabilized, turbulence is present in the array and is responsible for the mixing. Our investigations show that one can introduce an effective diffusivity to model the mixing at large distances as compared to the integral length scale (Fig. 2). Moreover, the spatial spectrum of the concentration reveals that the mixing is controlled by different mechanisms depending on the wavenumber range that is considered.

Fig. 1: View of the experiments.

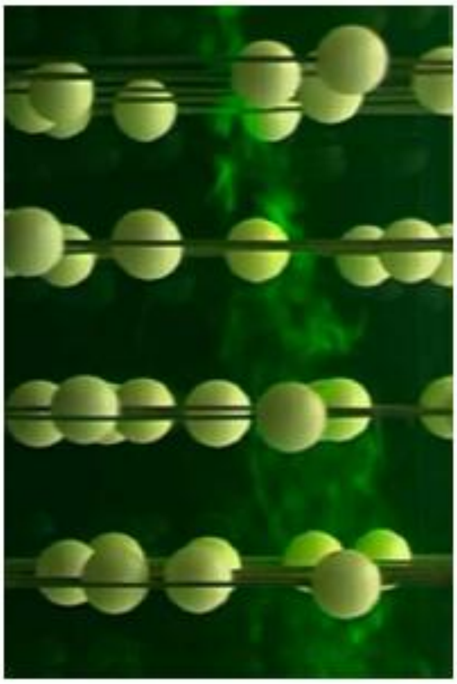
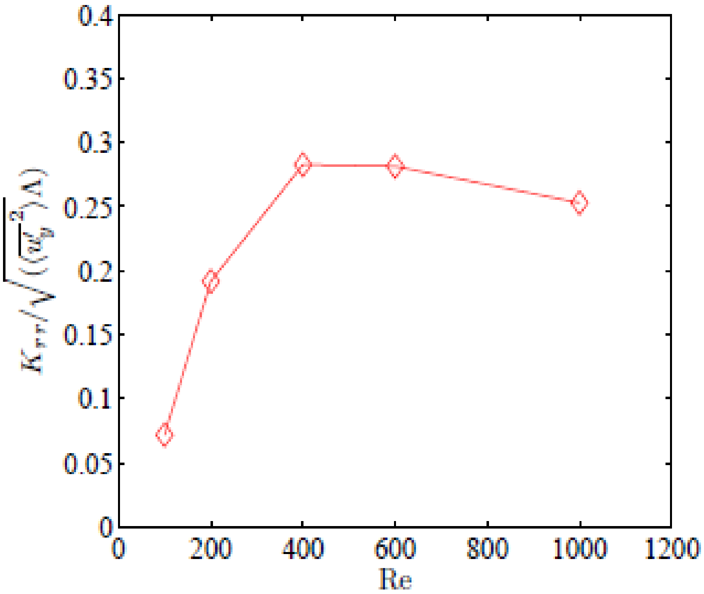


Fig. 2: Effective diffusivity in the random array.



Multi-scale analysis of drug delivery through an inhaler device

Y. Cui, S. Schmalfuß and M. Sommerfeld
Zentrum für Ingenieurwissenschaften, Martin-Luther-Universität Halle-Wittenberg
D-06099 Halle (Saale), Germany
e-mail : martin.sommerfeld@iw.uni-halle.de

Pulmonary drug delivery of dry powders is increasingly being used for medical treatment because of a number of benefits associated with an inhalation therapy. Essential for an efficient use of dry powder inhalers is the pressure drop and consequently the achievable breathing flow rate as well as the fine powder delivery to the patient. Therefore, numerous geometrical designs of inhalers are on the market. A major requirement for the application of dry powders is that their size should be smaller than 5 µm in order to reach the alveoli of the lungs. Such fine powders are very cohesive and therefore are hard to be dispersed in the inhaler. A solution to this problem is the blending of the fine drug particles with larger carrier particles (i.e. in the range between 50 and 100 µm). Consequently, the inhaler must insure the detachment of drug powder from the carrier. This detachment is brought about by fluid dynamic stresses and carrier particle wall impacts. The efficiency of drug powder delivery is typically between 30 and 60 % depending on the design of the inhaler.

In order to improve the efficiency of inhalers a multi-scale numerical analysis is adopted. First the fluid stresses acting on carrier particles are analysed for a typical device based on calculating the steady and unsteady flow field by solving the Reynolds-averaged conservation equations in connection with the κ-ω-SST turbulence model. Then a number of carrier particles are tracked through this flow field and instantaneous relative velocity, shear rates and turbulence are recorded along their path. From the resulting probability density functions the most probable values are extracted and then used to simulate the flow about a fully resolved carrier particle covered with drug particles using the Lattice-Boltzmann method. In this situation the cluster is centrally fixed in a cubic computational domain and exposed to laminar and turbulent plug flow as well as shear flow. From these simulations the fluid dynamic forces on the drug particles in dependence on their location on the carrier are extracted. Using experimentally obtained values for the van der Waals force and the friction coefficients it is possible to calculate a detachment probability as a function of relevant parameters, such as relative velocity (Reynolds number) and turbulence intensity.

In a later stage this detailed information will be used to develop detachment models which may be used in the Euler/Lagrange calculations of the entire inhaler. Consequently it should be possible to calculate the drug delivery efficiency in the future.

Viscosity model of time-dependent blood flow considering deformable RBCs clustering

Janez Urevc[a], Iztok Žun[b], Milan Brumen[c] and Boris Štok[a]
[a] *LNMS, Faculty of Mechanical Engineering, University of Ljubljana*
[b] *LFDT, Faculty of Mechanical Engineering, University of Ljubljana*
[c] *Chair of Biophysics, Faculty of Medicine, University of Maribor*
e-mail : iztok.zun@fs.uni-lj.si

Atherosclerosis is the leading cause of morbidity and mortality in the developed world. A significant factor affecting the formation and progression of the disease is blood viscosity. The latter mostly depends on plasma viscosity, amount of RBCs in blood (given with hematocrit *Ht*), aggregation of RBCs and their deformability. At low shear rates RBCs tend to aggregate, forming three-dimensional microstructures called rouleaux or clusters. An increase of shear rate decreases the average size of clusters until, at a critical value of shear rate, clusters are broken down into individual cells. Time-dependent properties of blood predominantly arise due to the finite time, needed for the flow-induced microstructure to respond to a change in flow.

We approach the problem of blood viscosity modeling by breaking down the whole blood relative viscosity Π_r into relative viscosity describing frozen pattern (steady state) blood's behavior Π_{fp} and relative viscosity due to blood's time dependent behavior Π_v. We believe that the flow-induced structure of particles/clusters has a major influence on Π_{fp}. Thus, we further decomposed Π_{fp} into: relative viscosity due to elastically deformed individual particles Π_{fp_e}, describing the frozen pattern behavior of fixed shaped particles; and relative viscosity due to the structural aggregation of elastically deformable particles Π_{fp_s}, taking into account the increase of relative viscosity due to the change of particle's/cluster's morphology. Each part of Π_r is determined based on dimensional analysis and experimental data (provided from literature).

Based on literature review, experimental data, and our experiences in two-phase flow, the following dimensionless terms are derived describing Π_{fp}: a term describing RBC diffusivity in a solution, weighted by a shape and diffusivity function; phase fraction of RBCs; and a term accounting the maximum packing fraction of RBCs. A parameter (χ) is formulated to account for the flow-induced morphology of a particle/cluster. χ is essentially defined to account for the drag of a particle/cluster in the direction of flow due to its size. In our current study, since it would have been difficult to experimentally trace the flow-induced geometry of particles/clusters, χ is modeled only phenomenologically. Dimensionless terms derived to describe Π_v are: a term describing the adaptability of a structure to a change of shear rate, weighted by a shape and diffusivity function; a term accounting for the size of a flow induced structure; a term representing phase fraction of RBCs; and a term describing the relaxation of a particle/cluster.

Velocity measurements in a cavitating micro-channel flow

C. Mauger[1], L. Méès[2], M. Michard[2], M. Lance[2]
[1]*Laboratoire de Physique de l'ENS Lyon (Phys-ENS), CNRS UMR5672 – École Normale Supérieure - Lyon*
[2]*Laboratoire de Mécanique des Fluides et d'Acoustique (LMFA), CNRS UMR 5509 – Ecole Centrale de Lyon – INSA de Lyon – Université Claude Bernard – Lyon 1, Ecully*
e-mail : : Michel.Lance@ec-lyon.fr

Cavitation is generally known for its drawbacks (noise, vibration, damage). However, it may play a beneficial role in the particular case of fuel injection, in the atomization process [1] or by reducing nozzle fouling. Studying cavitation in real injection configuration is therefore of great interest, yet tricky because of high pressure, high speed velocity, small dimensions and lack of optical access for instance. A simplified transparent 2D micro-channel (200-400 µm) continuously supplied with test oil at lower pressure (6 MPa) is adopted. A shadowgraph-like imaging system is set-up. It makes it possible to visualize vapor formations as well as density gradients (refractive index gradients) in the liquid phase [2], including scrambled structures connected to turbulence. Space and space-time correlation functions are used to characterize these structures' evolution. An integral length scale is defined. It tends to decrease as the Reynolds number increases. Once cavitation is reached, the integral length scale seems to slightly increase, suggesting a diminution in smaller (or an increase in larger) turbulent structures. Since these structures are correlated in time, an image correlation algorithm can be used to extract mean velocity field as well as fluctuations, without tracer. A good agreement is found between the flow rate measured with a flowmeter and that deduced from the image correlation algorithm. An increase in velocity fluctuations is observed at the channel outlet for a critical normalized length of vapor cavities equal to 40-50 %. A parametric study on channel height and oil temperature is performed: none of them impacts the critical normalized length but larger velocity fluctuations are observed in channel of larger height.

References :
[1] M. Birouk, N. Lekic, Liquid jet breakup in quiescent atmosphere: A review, Atomization and Sprays 19 (6): 501–528 (2009).
[2] C. Mauger, L. Méès, M. Michard, A. Azouzi, S. Valette, Shadowgraph, Schlieren and Interferometry in a 2D cavitating channel flow, Exp Fluid. 53: 1895-1913 (2012).

My Experience with Bubbles

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

The purpose of this talk is to tell the story of my experience with researching bubbles in the context of nucleate boiling. The story begins in 1994 during my MSc research where I first started investigating the effects of liquid subcooling on nucleate pool boiling. During this work I discovered a regime of thermocapillary heat transfer that occurs between the buoyant natural convection and subcooled boiling regimes [1]. Subsequent to this I began my doctoral research and decided to focus on mathematical modelling of nucleate pool boiling. This was partially motivated by new experimental data on microgravity boiling published by the University of Michigan and by numerical modelling techniques being developed by the University of Florida. The numerical model that I developed, albeit for a specific type of heterogeneous boiling, was possibly the first to accurately predict experimental bubble growth data without any empirical constants or adjustable parameters [2]. The numerical model was then used to gain a deeper understanding of the transitions between surface tension, inertial and heat diffusion controlled bubble growth dynamics for homogeneous boiling [3]. Although the numerical models worked and included the coupling between the Navier-Stokes and heat equations, they were very specific to extreme, and uncommon, bubble growth scenarios where the bubbles were spherical; these were not direct numerical simulations.

I subsequently joined the academic staff at Trinity College Dublin in 2005 where I began to think more deeply about the bubble shape during heterogeneous bubble growth. In particular I became interested in gravitational and electric field effects and how they act to influence the bubble behaviour. In an attempt to focus on bubble mechanics in various force fields I decided to uncomplicated the problem by performing numerical, analytical and experimental work on adiabatic bubble injection from submerged orifices. Fortuitously, it was during this stage of research that I met Professor Jocelyn Bonjour of CETHIL who was following the same line of reasoning by studying diabatic bubble growth of single bubbles nucleated from an artificial cavity, including electric field effects. Back in Ireland we developed a novel measurement technique whereby we used bubble images acquired by high speed videography to reconstruct the three dimensional bubble digitally. The 3D shape along with the instantaneous measurement of the internal gas pressure facilitated the calculation of the liquid pressure around the entire periphery of the bubble. This technique was then used to gain insight into the forces acting on bubbles of different Bond numbers [4] as well as being the first to indirectly measure the electrically induced pressures acting on bubbles under the influence of EHD [5]. Another key component to this investigation was the development of CFD models to predict adiabatic and quasi-static bubble growth and departure. In this context, different numerical techniques for interface capturing and reconstruction as well as different software options, including OPENFOAM, TranAT and FLUENT were investigated, using the benchmark experimental data to validate the CFD simulations [6,7,8]. In parallel to the work being performed in Ireland, separate investigations with Professor Bonjour of CETHIL and Professor James Cotton of McMaster University were also being performed regarding how bubble shape is related to its mechanical environment and ultimately influences its growth and departure behaviour [9,10,11].

As it may be apparent, my bubble research has a trajectory towards understanding bubble growth dynamics with an ultimate gaol of understanding the mechanisms of heat transfer during nucleate pool boiling. To progress towards this, my current research is coordinated to support the development of correct CFD predictions of heterogeneous bubble growth during boiling. This still requires development of triple contact line models, integration of the heat equation and inertial influences on the bubble growth dynamics.

References :

1. S. Petrovic, A.J. Robinson, R.L. Judd, (2004) "Marangoni Heat Transfer in Subcooled Nucleate Pool Boiling," International Journal of Heat and Mass Transfer, 47 (23), pp. 5115-5128.
2. A.J. Robinson, R.L. Judd, (2001) "Bubble Growth in a Uniform and Spatially Distributed Temperature Field," International Journal of Heat and Mass Transfer, 44, pp. 2699-2710.
3. A.J. Robinson, R.L. Judd, (2004) "The Dynamics of Spherical Bubble Growth," International Journal of Heat and Mass Transfer, 47 (23), pp. 5101-5113.
4. S. Di Bari, A.J. Robinson, (2013) "Experimental Study of Gas Injected Bubble Growth from Submerged Orifices," Experimental Thermal and Fluid Science, 44, pp. 124-13.
5. S. Di Bari, A.J. Robinson, (2013) "Adiabatic bubble growth in uniform DC electric fields", Experimental Thermal and Fluid Science, 44, pp. 114-123.
6. A. Albadawi, D. B. Donoghue, A.J. Robinson, D. B. Murray, Y. M. C. Delauré, (2013) "An assessment of Volume of Fluid and Level Set methods for low capillary bubble growth and detachment," Chemical Engineering Science, 90(7), pp. 77-91.
7. A. Albadawi, D. B. Donoghue, A.J. Robinson, D. B. Murray, Y. M. C. Delauré, (2013) "Influence of surface tension implementation in Volume of Fluid and coupled Volume of Fluid with Level Set methods for bubble growth and detachment," International Journal of Multiphase Flow, 53, pp. 11-28.
8. S. Di Bari, D. Lakehal, A.J. Robinson, (2013) "A Numerical Study of Quasi-Static Gas Injected Bubble Growth: Some Aspects of Gravity" International Journal of Heat & Mass Transfer (accepted for publication).
9. F. Lesage, J.S. Cotton, A.J. Robinson, (2013) "Analysis of quasi-static vapour bubble shape during growth and departure" Physics of Fluids (accepted for publication).
10. S. Siedel, S. Cioulachtjian, A.J. Robinson, J. Bonjour, (2013) "Experimental Investigation on the Local Curvature of the Bubble Interface during Boiling on a Single Nucleation site," Heat Transfer Engineering (accepted for publication).
11. S. Siedel, S. Cioulachtjian, A.J. Robinson, J. Bonjour, (2013) "Bubble distortion due to EHD stress", International Journal of Plasma, Environmental Science & Technology (in press).

Size and velocity distributions of the droplets produced by a single wave in a stratified air-water pipe flow

A. Audouin, P. Ern , and F. Charru
Institut de Mécanique des Fluides de Toulouse,
Université de Toulouse and CNRS, allée C. Soula, 31400 Toulouse, France
e-mail : patricia.ern@imft.fr

When a liquid layer is sheared by a faster gas stream in a pipe, large amplitude waves may appear at the interface and propagate along the liquid surface. If the gas velocity is large enough, droplets detach from the top of these waves. The distributions of size and velocity of the droplets observed in the dispersed phase are then large: some drops (in particular the larger ones) originate from waves near the measurement volume and present a low longitudinal velocity (close to the celerity of the waves), while others (mainly the smaller ones) originate from more distant waves and rapidly accelerated by the gas, are able to travel along great distances. A better understanding of the effect of acceleration and transport by the gas and of the effect of sedimentation by gravity on the droplets detached from a rollwave is therefore needed. The aim of this work is thus to focus on the contribution to the dispersed phase provided by a single wave and to determine the size and velocity distributions of the droplets according to the distance between the measurement volume and the isolated wave. The rollwave is induced by a pulse in liquid flow rate in a stratified air-water flow in a pipe. The characteristics of the droplets produced from this wave were investigated using a high-speed camera (5000 fps) coupled to a 20ns flash lamp to get motion-blur-free images. Digital image processing, made by a droplet detection program and a tracking routine, allowed us to measure the probability density functions of droplets sizes and velocities for two distances to the wave. In both cases, the size and velocity distributions of the droplets are quite large. However, we observe that larger drops are present in the vicinity of the isolated wave and disappear when increasing the distance between the measurement volume and the wave, as a consequence of redeposition. On the other hand, the amount of small drops is kept almost constant for both distances to the wave. Futhermore, it turns out that the droplets size distributions can be satisfactorily fitted by a lognormal law in both cases. Comparison of the distributions measured in this work with those obtained for a wavy-stratified regime in which atomization occurs naturally from various waves indicates that the relative weight in these distributions of smaller drops to that of larger drops is stronger in the latter regime, which supports the idea of an accumulation in this regime of smaller droplets originating from a larger range of rollwaves. Analysis of the distribution of horizontal velocities also revealed that the droplets have a lower speed for the smallest distance to the wave, confirming the acceleration effect of the gas on the droplets. Our goal is now to obtain a better description of the atomization and deposition distributions used for the closure models in the momentum conservation equations in the liquid and gas phases.

Acknowledgements :
This work was supported by TOTAL Exploration & Production and Scientific Development Division. We would like to thank in particular D. Larrey, F. Luck and J.M. Munoz for their involvement in the project. We also thank S. Cazin, G. Ehses, H. Ayroles and M. Marchal from IMFT for the technical support.

On the behaviour of a water droplet on a heterogeneous wettability surface

F. MANCIO REIS[a,b], M. MISCEVIC[a,b], P. LAVIEILLE[a,b], B. SCHLEGEL[a,b]

[a] *Université de Toulouse, UPS, INPT, LAPLACE (Laboratoire Plasma et Conversion d'Energie), 118 route de Narbonne, F-31062 Toulouse*

[b] *CNRS, LAPLACE, F-31062 Toulouse, France*

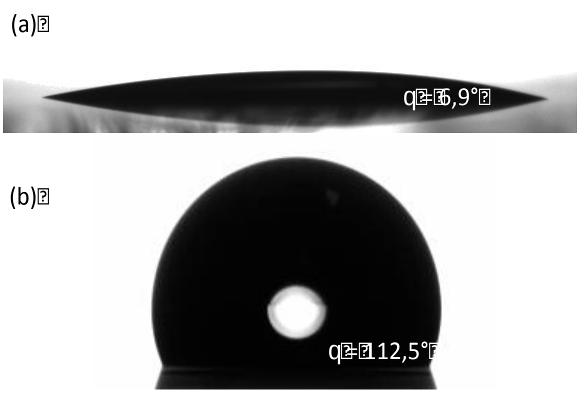
e-mail : marc.miscevic@laplace.univ-tlse.fr

It is commonly considered that two-phase systems are the most efficient to evacuate large heat fluxes. These systems are based on the latent heat associated to a change of state of matter, i.e. a phase transition such as the boiling of a liquid. Even if two-phase systems are known effective, their control under microgravity conditions is not yet completed: the gravity forces do not allow the evacuation of the disperse phase anymore. Moreover the devices' miniaturisation leads to the same problem of the evacuation of the dispersed phase. In the present work the used of surface tension forces induced by heterogeneous wettability is proposed to solve this problem. It is considered that a heterogeneous wettability property of a solid surface enables the mechanical non-equilibrium of the embryos forming on the wall.

Dynamic contact angle have been widely studied in the literature, from both experimental and theoretical point of views. The most common approach is based on hydrodynamic theory focusing on the viscous dissipation near the liquid drop wedge. This approach consists in considering the balance between the driving force, directly related to the wettability gradient, and the viscous force near the contact line. This approach is well suited for highly wetting liquid but is questionable for higher contact angles. Additionally, models available in the literature do not take into account the contact angle hysteresis, which however appears to be a very important parameter in the ability of the surface to evacuate the disperse phase. So, a hydrodynamic model that takes into account both contact angle hysteresis and wettability gradient has been developed in order to predict the motion of a droplet on such a surface.

Simultaneously with the theoretical approach, an experimental setup is developed. Two ways are explored to obtain heterogeneous wettability surfaces. The first one consists in a chemical deposition, i.e. silanization. This surface treatment allows a silicon wafer or a glass surface to react with vapours of a volatile R-$SiCl_3$ by using a diffusion-controlled process. In order to minimize hysteresis, the humidity of air in the cell test is removed by injecting nitrogen. A drop of silane is deposed near the surface to be treated. As the silane droplet evaporates the vapour diffuses in the nitrogen gas phase and chemical deposition occurs on the surface. The concentration gradient creates then the wettability gradient.

Figure Water droplet before (a) and after (b) chemical deposition on a glass plate.



The second way explored consists in a plasma deposition. The processing of the samples is based on a Plasma-Enhanced Chemical Vapour Deposition (PECVD) using the Matrix Distributed Electron Cyclotron Resonance (MDECR) reactor.

Once the sample has been realized, the first experiment consists in dropping a known volume of water on the chemical-grafted or plasma-coated surfaces and in visualizing the drop motion with a high-speed camera. It is therefore possible to determine the speed and acceleration of the water drop and then to deduce the resulting forces associated to the surface tensions and viscous effects. The next step will be to add some heat transfers by heating or cooling the sample.

Acknowledgement: this work is funded by the European Space Agency, Microgravity Application Program "MANBO" (Multisacle ANalyses of BOiling)

<u>References :</u>

F. Brochard, "Motions of droplets on solid surfaces induced by chemical or thermal gradients", Langmuir 1989, 5, 432-438

Manoj K. Chaudhury, John C. Chen, "Fast drop movements resulting from the phase change on a gradient surface", Science 2001, Vol. 291.

Manoj K. Chaudhury, George M. Whitesides, « How to make water run uphill », Science 1992, Vol. 256.

An investigation of bubble formation and fluid dynamics in pool boiling of propane on horizontal tubes

Yabai Wang, Andrea Luke
Institute of Technical Thermodynamics, University of Kassel, Germany
e-mail : luke@ift.uni-hannover.de

The aim of our research projects is to develop new equipment and plant concepts to raise unused potentials for heat integration in industrial process. Process-integrated energy efficiency is achieved in industrial plants by these methods. Two phase systems allow the transport of high heat fluxes by small temperature differences. The main transport phenomena has to be understood to use this main advantage of boiling and condensing systems. The heat transfer in boiling is measured on single tubes and later on small tube bundles, the roughness and wettability of the heated surface is analysed in advance. The activation of nucleation sites, the bubble growth and departure and the sliding in the superheated boundary layer around the heating surface of the test tube are observed by high speed video sequences.

The bubble formation and fluid dynamics are investigated in pool boiling heat transfer of propane. The characteristic behaviour of nucleation sites during nucleate boiling, for example the local and temporal distribution of the nucleation sites and frequency, also of the distance to the next active nucleation site, on a sandblasted copper tube have been investigated from heat transfer measurements over selected heat fluxes and pressures using template-based evaluation of high speed video sequences. The raw-data of the evaluation of the video sequences is sorted and converted for the statistical analysis. The bubbles are tracked along their way sliding in the superheated liquid to obtain the influence of fluid dynamics on the pool boiling heat transfer. The bubble departure diameter and frequency are compared to some existing (semi-)empirical correlations of literature and a new model for the heat transfer near growing, departing and sliding bubbles. The calculated results for the bubble departure diameter as function of the azimuthal position on the tube according to the new model show a good agreement with the new experimental data. The relative increase of the active nucleation sites is described quite well. The influence of the convective effects is extended to consider them in empirical correlations for the design of tube bundles. Furthermore, there is a short review on different calculation methods for the integral heat transfer coefficient in tube bundles and convective effects in boiling.

Keywords : bubble formation, fluid dynamics, high speed video sequences, horizontal tubes, pool boiling, heat transfer

Motion of a single bubble rising in a countercurrent flow in a Hele-Shaw cell

Audrey Filella, Patricia Ern, Véronique Roig
Institut de Mécanique des Fluides de Toulouse, allée du Professeur C. Soula, 31400 Toulouse
e-mail : patricia.ern@imft.fr

We investigate experimentally the motion of isolated bubbles rising in a vertical Hele-Shaw cell in the presence of a downward flow. The bubbles are strongly flattened in the plane of the cell, their equivalent diameter d being large compared to the gap of the cell e. Furthermore, their dynamics is strongly influenced by the confinement which imposes thin liquid films between the bubble and the walls and strongly attenuates the flow perturbation in the liquid due to wall friction. The control parameters of the problem are: the confinement ratio e/d, the Archimedes number $Ar = \sqrt{gd}.d/\nu$, the Bond number $Bo = \rho g d^2/\sigma$ and the channel Reynolds number based on the average velocity of the counterflow $Re_c = U_Q e/\nu$. The Archimedes numbers considered are larger than 100, so that inertia plays an major role in this configuration. The Bond numbers vary between 0.01 and 100 corresponding to regimes with or without deformation of the bubble. The countercurrent flow remains laminar ($70 < Re_c < 125$).
The bubble dynamics is controlled by the coupling between the wake (possibly unsteady) and the degrees of freedom of the bubble motion in translation, rotation and deformation. The aim of this study is to determine the effect of the counterflow on the motion and shape of inertial bubbles as compared to their behavior in liquid at rest ([1]). The motion and deformation of the bubbles have been characterized by a shadowgraph method. The flow perturbation induced in the liquid by the bubble motion will be measured by high-frequency PIV.
We observe that the bubble dynamics changes remarkably in the presence of a counterflow. Notably, the mean relative rise velocity of the bubble $(\overline{U} - \overline{U_Q})$ is higher in a counterflow than in a quiescent liquid. The bubble's velocity follows the relation : $\overline{U} = 0.5\overline{U_Q} + \overline{U_\infty}$ where $U_\infty = 0.64\sqrt{gd}$ (for $500 < Ar < 4000$) is the velocity of the bubble in liquid at rest (Figure 1). The explanation of this behavior might be found in the change in mean aspect ratio of the bubble, which is less stretched in the horizontal direction in the presence of a counterflow. The onset of an oscillatory path of the bubble is also modified, the corresponding Archimedes number being lower in the countercurrent flow configuration (Figure 2). The frequency ω and the horizontal and vertical velocities amplitudes $\widetilde{V_x}$ and $\widetilde{V_z}$ are modified by the flow in the cell, but the Strouhal number $St = \omega d/\sqrt{gd}$ and the velocities amplitudes normalized by the velocity scale $\sqrt{gd}$ are shown to be independent of Re_c (Figure 2).

Figure 1 : $Re_{rel} = (U + 0.5U_Q)d/\nu$ as a function of Ar.

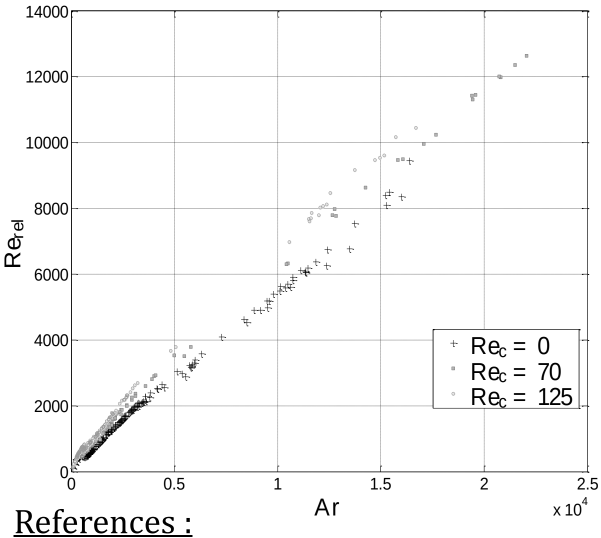
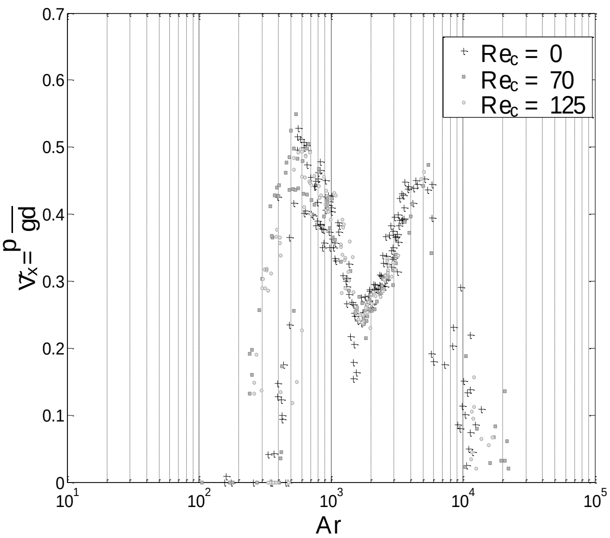


Figure 2 : Normalized amplitude of the horizontal velocity as a function of Ar.



References :
[1] V. Roig, M. Roudet, F. Risso, A. M. Billet (2012), J. Fluid Mech., vol. 707, pp. 444-466.

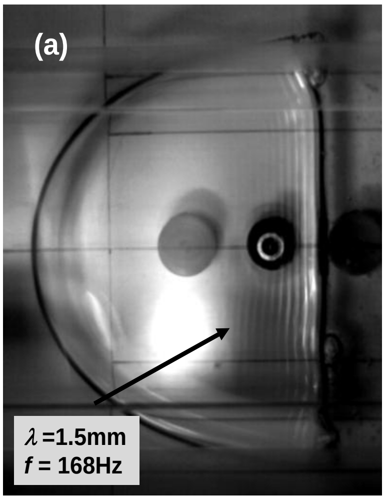
Wall shear stress induced by rising Taylor bubbles

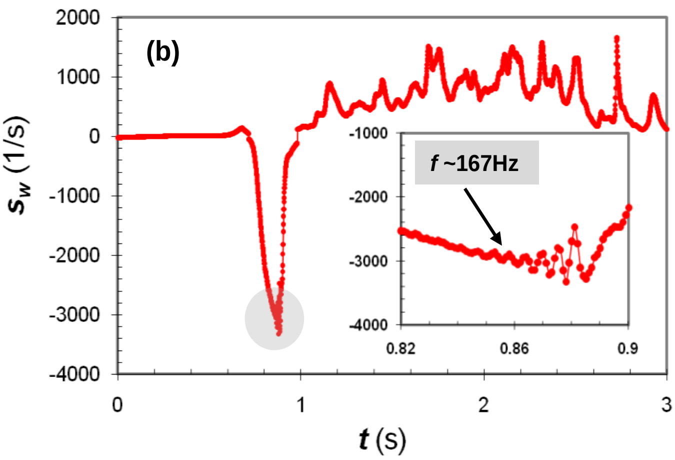
J.Tihon, V.Penkavova, J.Vejrazka
Institute of Chemical Process Fundamentals, Academy of Sciences of the Czech Republic, Rozvojová 135, 16502 Prague 6, Czech Republic
e-mail : tihon@icpf.cas.cz

The motion of single air bubbles in flat channels is experimentally investigated. The electrodiffusion technique of near-wall flow diagnostics is applied to measure the wall shear stress distribution under large rising bubbles. The measurements are synchronized with the visual observation of bubble movement by a high-speed camera. The analysis of video records provides information on the bubble shape and terminal velocity. The experiments are carried out for three different channel configuration (with heights of 1.5, 4, and 8 mm), cover a wide range of channel inclination angles (from horizontal to vertical position), and dealing with both the bubbles in stagnant and in co-flowing water.

The directionally sensitive, two-strip electrodiffusion probe is proved to be an effective tool to investigate the near-wall flow response to translating bubbles. It provides information not only on the wall shear rate distribution, but also detects the location of near-wall flow reversal, gives an estimate of the thickness of liquid film separating the large bubble from the wall, and provides also the characteristics of capillary waves appearing in the bubble tail region. The effect of channel inclination angle on the modification of wall shear stress distribution along the upper and bottom wall is especially discussed. Finally, the velocity scaling suggested for the rise of large bubbles in stagnant and co-flowing liquids is verified for inclined flat channels.

Figure 1: Large bubble rising in a vertical flat channel: (a) picture of the bubble at the sensor, (b) wall shear rate profile with capillary waves region shown in detail.





Detachment of gas bubbles from an inclined surface

Paolo Di Marco, Giacomo Saccone
DESTEC, University of Pisa, Pisa, Italy
e-mail : p.dimarco@ing.unipi.it

One of the main topics of the current activities in boiling heat transfer is the study of the influence of a shear flow on bubble detachment from a heated surface. In these conditions, the bubble is subjected to a shear force, which deforms the interface from its initial, axisymmetric shape. Models of bubble detachment in shear flow have been developed, among the others, by Duhar and Colin (2006), and account of the different value of contact angle on the trailing and leading side of the bubble.

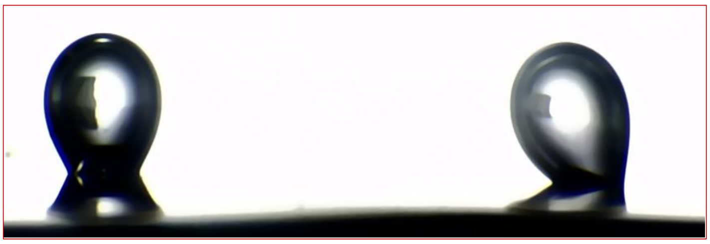
A similar situation can be reproduced by tilting the detachment surface, in order to have a shear force due to the gravity acting on the bubble. In these conditions, by injecting a controlled amount of gas from an orifice in the liquid, a static or slowly growing interface can be created, which can be more easily characterized and studied in controlled conditions. The results will help in clarifying the role of a shear force on bubble detachment and can be compared with existing models in order to check their validity.

The study has been carried out on ground, and has taken advantage of the experience developed by UNIPI in the digital acquisition of bubble interface and in the measurement of the contact angle (Di Marco, 2009). Detachment bubbles generated by gas injection from an orifice on a tilted plate has been considered, due to the simplified boundary condition when the three-phase line is "pinned" to the orifice rim.

The core of experimental setup is a series of bubble generators, each of them consisting in a flat plate with circular orifices of different diameter; the orifice is fed either by a micro-syringe (actuated by a stepping motor) or by a gas flowrate controller. In this way, continuous trains of detaching bubbles, individual bubbles growing very slowly or even static interfaces could be generated. The plate is hosted inside a fluid container (see Fig. 10) filled with FC-72 at atmospheric pressure, which can be tilted at any angle from 0 to 90 degrees, in order to make it possible to study bubble detachment from a horizontal to a vertical surface. Since the interface is asymmetrical, a dedicated mirror system has been developed in order to acquire in the same frame two images of the interface from different points of view at 90°, see Fig.1. The acquisition of images in the same frame avoided the necessity to synchronize frames acquired by different cameras, and made the data processing more simple.

The interface shape has been digitized with a dedicated image processing software, based on Matlab. Local interface curvature and apparent contact angles have been derived with appropriate interpolation methods of the profile. In particular, the trend of the incidence angle at three positions around the three-phase contact line has been determined as a function of the bubble volume, for different plate inclinations. It was also possible to check with acceptable results the overall force balance on the bubble. Finally, the shape of the bubble has been compared, with satisfactory agreement, with the one obtained by means of the numerical code Surface Evolver.

Figure 1 –Images of the bubble



References :

Di Marco P., 2009, Experimental measurements of the incidence angle during gas bubble growth and detachment, 7*th World Conference on Experimental Heat Transfer, Fluid Mechanics, and Thermodynamics,* Krakow, Poland, June 28-July 3, CD-ROM, pp 1-8.

Duhar G, Colin C., 2006, Dynamics of bubble growth and detachement in a viscous shear flow, *Physics of Fluids*, 18, 077101.

Quasi-static bubble shapes

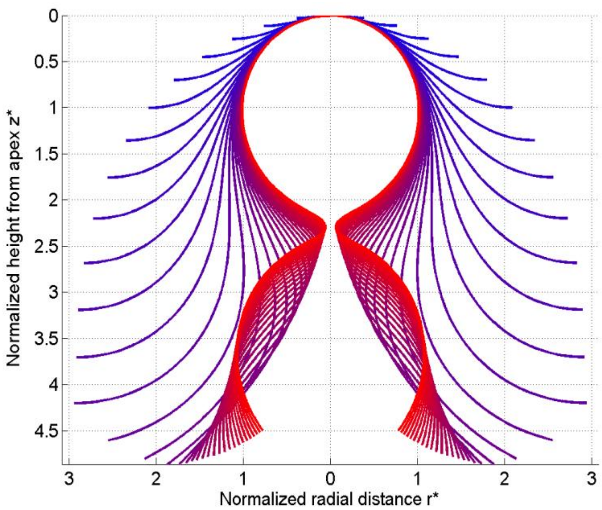
Samuel Siedel and Anthony J. Robinson
Department of Mechanical and Manufacturing Engineering, Trinity College Dublin, Dublin 2, Ireland
e-mail : arobins@tcd.ie

When investigating the physical mechanisms responsible for bubble departure from either a boiling surface or a gas injection orifice, different scenarios are encountered depending on the stresses and pressures acting on the bubble interface (surface tension, hydrodynamic pressure, viscous shear, etc.), on the geometry of the surface (flat surface, sharp-edged cavity, re-entrant cavity, etc.) and on the *fluid-surface* interactions (advancing and receding contact angles).
When the dynamic stresses and the momentum variations are negligible compared to the static stresses, a quasi-static mechanical equilibrium is reached at any location of the bubble interface for any instantaneous moment of its growth. This equilibrium is described by the Young-Laplace (YL) equation $\Delta P = \sigma C$ that states that the local external pressures ΔP acting on an interface are balanced by the internal stress, which is equal to the product of the surface tension σ and the local curvature C of the interface.
For a quasi-static bubble is a hydrostatic pressure field, the YL equation can be expressed as a function of the capillary length L_c, the curvature at the bubble apex C_0 and the height difference from the apex z :

$$zL_c^{-2} = C_0 - C(z)$$

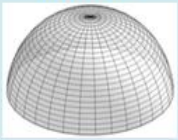
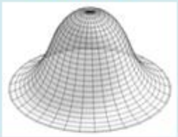
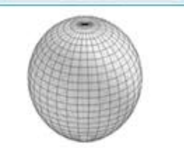
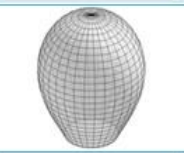
This equation can be normalized by any judicious reference length, for example the radius of curvature R_0 at the apex of the bubble. One family of solutions to the equation is found, and plotted in Fig. 1 as a function of the normalized capillary length $L_c^* = L_c/R_0$.

Figure 1: Young-Laplace profiles for normalized capillary length of 0.1 (blue) to 3.5 (red)



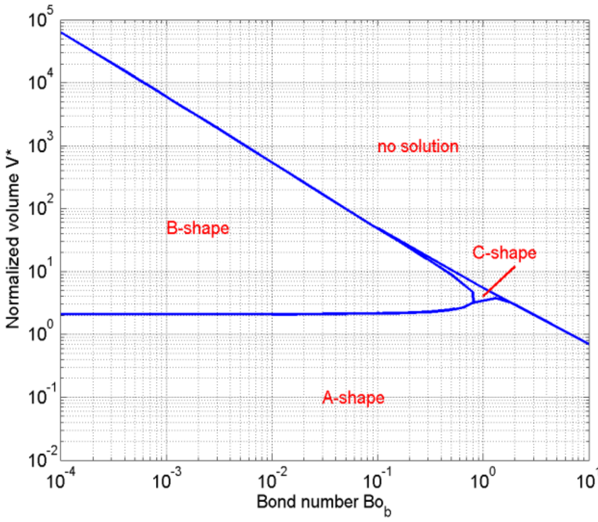
For a bubble attached to a horizontal surface, the limit of applicability of the YL equation is the triple line where the liquid-vapor (LV) interface reaches the solid surface. The LV interface is thus represented by corresponding YL profile on Fig. 1, truncated by the normalized bubble height. Any quasi-static bubble shape is thus fully defined by its capillary length, its height and a reference length.
However, although the capillary length is usually known for any practical case, the radius of curvature at the apex and the bubble height are not known *a priori*. A better set of parameter can be chosen, using the radius of the bubble base b as the reference length, and the bubble volume V instead of its height. The normalized volume $V^* = V/b^3$ and the Bond number $\mathrm{Bo}_b = b^2/L_c^2$ can be used to characterize a normalized bubble shape. Different families of solutions can be found to the YL equation, among which 3 families A, B and C, represented in Fig. 2, will be discussed. The family **A** is shaped as a cap, and has an apparent contact angle α higher than 90°. The family **B** is ball shaped without a neck, and with $\alpha < 90°$. The family **C** presents a neck with $\alpha > 90°$.

Figure 2: Shape families

Family¤	Low·Bo_b¤	High·Bo_b¤
A¤	¤ 	
B¤		
C¤		

For a given set of V^* and Bo_b, it is possible to find two solutions of families **A** and **C**, or **B** and **C**. The solutions of family **C** are discarded is this case as the resulting bubble would have a higher interface area and would thus be unstable.

Figure 3: Map of existence of shape families



The resolution of the normalized YL equation provides the limit of existence of the different shape families, as plotted in Fig. 3.
This map can be used in order to understand the growth and model the departure size of bubble. Indeed, the apparent contact angle is a function of the shape of the bubble. Depending on the size and geometry of the cavity or nucleation site on which the bubble initially forms, and knowing the advancing and receding contact angles, it is possible to evaluate if the bubble will remain attached to the rim of the orifice, and if it is possible to reach the maximum volume predicted from the YL equation.

Internet access at the conference venue

In order to connect your equipment to the wifi network of the conference venue please follow the instructions:

1. Choose the wireless network named ***orange***.
2. Launch your browser.
3. Click on ***identifiez vous***.