Book Of Abstracts

Table of Contents

Effects of surface active agents on drop formation and emulsion properties

Dr Loïc Chagot, Dr Simona Migliozzi, Prof Panagiota Angeli,

ThAMeS Multiphase, Department of Chemical Engineering, UCL
p.angeli@ucl.ac.uk

Dispersions/emulsions find numerous applications in pharmaceutical and healthcare formulations, food and agrochemicals. In recent years, microchannels have been extensively used to produce emulsions with small, uniform drop sizes. Surfactants and colloidal particles are commonly added to vary the interfacial properties, control the drop size, stabilise the emulsions and influence the final product rheology. The drop size is determined by the droplet break up in the microfluidic channels which is linked to interfacial properties such as interfacial tension and rheology. These depend on the absorption and distribution of the surface active agents to the forming interfaces. During drop formation, surfactant concentration gradients can appear which give rise to Marangoni stresses at the interface, while presence of colloidal particles modifies the interfacial rheology; both these phenomena affect drop break up.
In the talk, I will discuss the phenomena occurring during drop formation and break up in microfluidic channels in the presence of surfactants and colloidal particles. Interfacial properties, such as interfacial tension and rheology, are characterised for different surfactants as well as hard and soft colloidal particles. The regimes of drop break up and the modifications in their boundaries due to the presence of surface active agents will be presented. Results will also be shown on the bulk rheological behaviour of emulsions stabilised by soft particles and the impact of particle properties on drug delivery applications.

Trinity College Dublin
Coláiste na Tríonóide, Baile Átha Cliath
The University of Dublin
ETPFGM 2024

54th European Two-Phase Flow Group Meeting
08-11 September 2024,
Dublin, Ireland
ABSTRACT

Investigation of the effect of thermocapillarity on deformation of evaporating films on structured substrates

R. Behle, P. Stephan and T. Gambaryan-Roisman

Institute for Technical Thermodynamics, Technical University of Darmstadt, Darmstadt, Germany
Contact E-mail : behle@ttd.tu-darmstadt.de

Thin liquid films play a crucial role in energy and chemical technology as they facilitate heat dissipation through evaporation. In particular, they are used in cooling systems in terrestrial and extraterrestrial environments that require high heat transfer rates. The behavior of thin films on heated surfaces is significantly influenced by the Marangoni effect, which arises from variations in surface tension along the liquid-gas interface. This phenomenon enhances convective heat transport, but also carries the risk of liquid film rupture, leading to the formation of dry patches and deterioration of heat transport. Theoretical and numerical studies have shown that using surfaces with topography induces Marangoni convection and allows to control the flow pattern, liquid topology and heat transfer. However, experimental data on Marangoni-induced interface deformation in evaporating liquids are very scarce.

In the present study, we experimentally investigated the thermocapillarity-induced deformation and rupture of a thin liquid film on a structured copper surface. The experiments were conducted in an open cell with a heated substrate. The structured copper substrate features sinusoidal grooves, where the wavelength of the topography is much larger than its amplitude. A thin liquid film of the fluorinated fluid HFE-7100 is deposited on the substrate in each experimental run, and its evolution and deformation are measured using a chromatic confocal line sensor. The data analysis focuses on evaluating the characteristics of long-wave Marangoni-driven film deformation and determining the evaporation rate.

The temperature difference between the substrate and the liquid-gas interface arises both due to the evaporative cooling of the interface and by the cooling of the surface by the ambient air. Controlled film deformation occurred due to thermocapillary stresses caused by the non-uniform temperature at the liquid-gas interface. The film flow resulted in localized thinning over the crests and thickening over the valleys of the grooves, consistent with the predictions of the long-wave theory [1]. Fig. 1 illustrates the deformation profiles of the liquid film at different temperature differences between the substrate and the ambient gas, demonstrating that surface deformation increases with rising temperature difference.

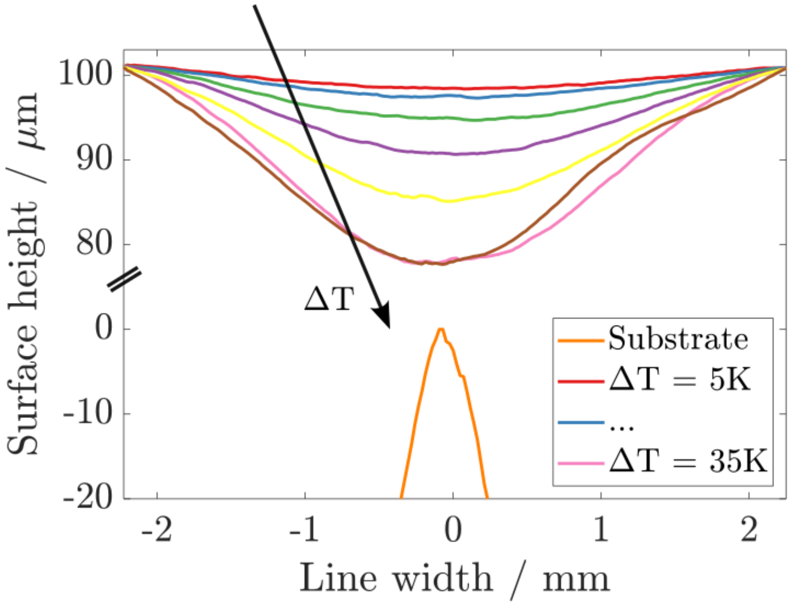


Figure 1. Liquid surface deformation over substrate crest. Illustrated deformation at temperature differences ΔT = [5:5:35] K between substrate and cell.

This work was supported by the German Aerospace Center (DLR) in the framework of the "Verständnis und Beeinflussung von Marangoni-Strömungen und Transportprozessen unter μ-g Bedingungen: "Marangoni in Films II"" project, grant no. 50WM2353.

[1] A. Bender, P. Stephan and T. Gambaryan-Roisman, Numerical investigation of the evolution and breakup of an evaporating liquid film on a structured wall, Int. J. Heat Fluid Flow 70, 104-113 (2018)

Bubble column fluid dynamics: a multi-scale perspective

Giorgio Besagni[1]

[1] Politecnico di Milano, Department of Energy, Via Lambruschini 4a, 20156 Milano, Italy
Contact E-mail : giorgio.besagni@polimi.it

When a gas phase is injected into a liquid phase, it generates a complex and intriguing fluid dynamic phenomenon. This lack of understanding, which poses challenges in the design and operation of multiphase reactors, is partly due to the absence of a definitive classification of flow regimes. Since pioneering research in the 1980s, it has been recognized that the "global scale" is influenced by the "local scale," with the interaction between these scales manifesting in various flow regimes. Specifically, the "bubble scale" (i.e., the motion of individual bubbles) affects the medium scale (i.e., turbulent eddies that transport the dispersed phase) and the large scale (i.e., circulation cells and central plume oscillations) characteristics, which define the "reactor scale." Unfortunately, a precise and analytical description of the connections between the "local scale" parameters and their scaling up to the "reactor scale" across different flow regimes remains elusive. Various studies have proposed different definitions of flow patterns and experimentally obtained some global and local flow properties, but a physically based description of these flow patterns is still lacking. Is there a theory that can determine, a priori, the boundaries of different flow regimes (and thus the flow regime for a given set of boundary conditions, phases, and system design)? In this lecture, drawing from a comprehensive dataset collected over recent years of research, a new theory is proposed. This theory redefines the current approach to bubble columns and is based on the following premise: the fluid dynamics in gas-liquid bubble columns can be interpreted through a general relationship, built upon five flow regime transitions, between two global fluid dynamic parameters (Figure 1).

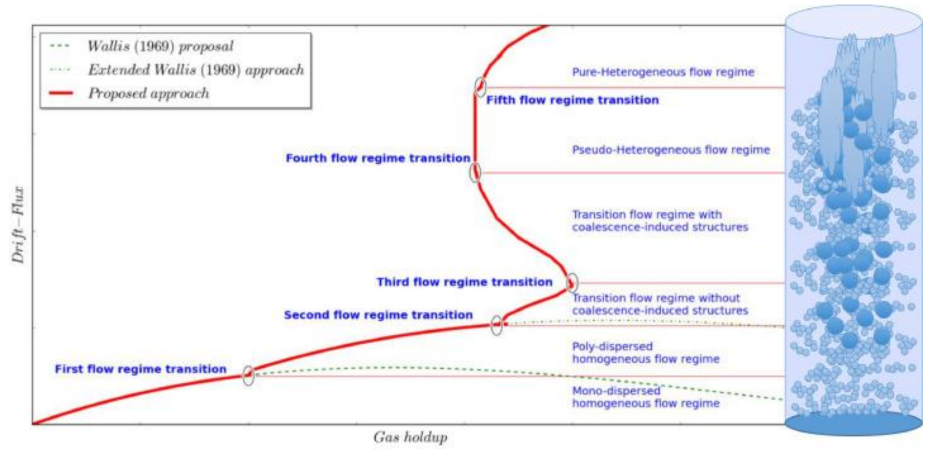


Figure 1. Anatomy of bubble column flow regime and flow regime transitions. In the graphical representation of bubble column, on the right, each vertical position represents a time-averaged condition for a fixed drift-flux value

Trinity College Dublin
Coláiste na Tríonóide, Baile Átha Cliath
The University of Dublin
ETPFGM 2024

54th European Two-Phase Flow Group Meeting
08-11 September 2024,
Dublin, Ireland
ABSTRACT

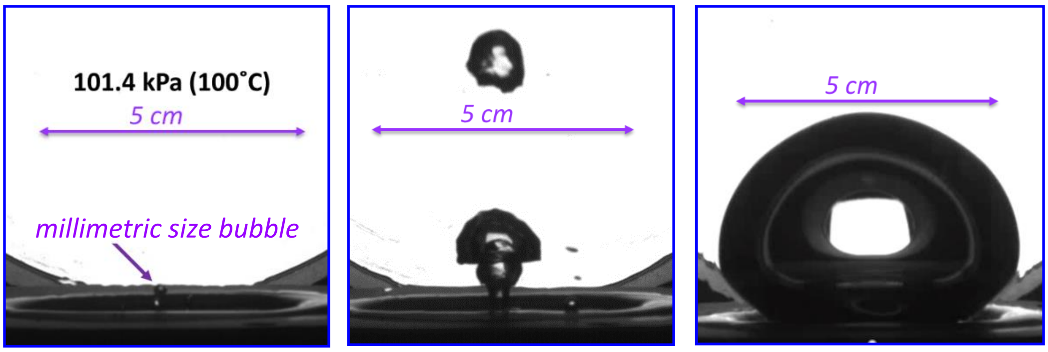
Toward thermally-generated sprays for efficient phase change heat transfer under low pressure

J. Bonjour, R. Rullière, A. Courouble

INSA Lyon, CNRS, CETHIL, UMR5008, 69621 Villeurbanne, France
Contact E-mail: jocelyn.bonjour@insa-lyon.fr

Sorption chillers emerge as a promising alternative to address new environmental constraints associated with air cooling systems. For air conditioning applications, the refrigerant conventionally used is water (absorption: water/lithium bromide, adsorption: water/silica gel). In these applications, the water temperature in the evaporator is approximately 10°C and the saturation pressure is around 10 mbar (1 kPa). The design and sizing of this heat exchanger still remain too empirical at present.

For several years, the CETHIL laboratory has been studying the fundamental phenomena of water vaporization at low pressure, often collaborating the Faculty of Mechanical and Power Engineering of Wroclaw University of Science and Technology. In a previous communication in the 8th EJTPFGM held in New York in 2018, it was shown that under low pressure (i.e. in the vicinity of the triple point), the bubble detachment diameter during pool boiling is quite high (up to 10 cm) while the bubbles can adopt unconventional shapes (cf. photo gallery, Fig. 1). In addition, particularly long waiting times were recorded between two successive bubbles. This results in a decrease in the average heat transfer coefficient. These aspects negatively affect the efficiency of low-pressure heat exchangers, leading to poor compactness and thereby hindering the development of these environmentally friendly air conditioning technologies.

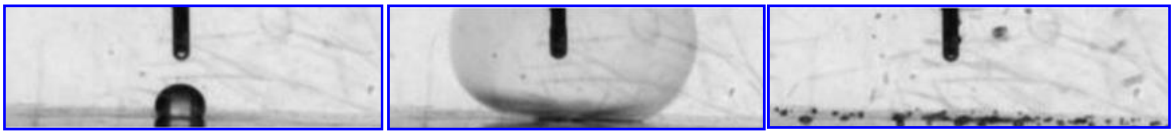


a. P = 101.4 kPa b. P = 47.4 kPa c. P = 4.7 kPa

Figure 1. Photo gallery of bubbles close to their detachment during boiling under (sub)atmospheric pressure

A solution to improve the phase change heat transfer in such conditions would reside in the generation of a spray. Nevertheless, generating a spray usually require high pressure nozzles, whose operational control would be difficult to manage as the outlet pressure must be compliant with the low pressure in the sorption chillers. The vision developed by the authors is that the bursting of a large bubble (typical of low pressure) that has grown inside a drop has the potential to generate a spray of daughter droplets. Owing to the burst, these droplets can be ejected all over the hot wall, where they evaporate with a high heat transfer rate and an increased effective area.

Preliminary tests demonstrated the validity of this vision: Fig. 2 shows the formation of a large bubble inside a drop (Fig. 2a) followed by its burst (Fig. 2b) and spreading of daughter droplets (Fig. 2c). To better assess the potential of the concept of thermally-induced spray under low pressure, a test bench was developed based on a temperature- and pressure-controlled vessel in which vaporization of saturated water under low pressure can be studied. It includes a heated wall (8 cm x 8 cm) on which drops (of millimeter size) can be deposited. The heated wall comprises 25 heat flux sensors (5 x 5 matrix) that allow to determine the thermal characteristics of the various phase change regimes that occur (e.g. thermal signatures like that displayed on Fig. 3), depending on if a bubble nucleates or not in the drop, and if it bursts or not. The conditions for the occurrence of the various regimes can therefore be examined so as to favor the regime that results in the most efficient heat transfer.



(a) (b) (c)

Figure 2. Growth of a large bubble inside a drop (a), burst (b), creation of a spray of droplets (c)

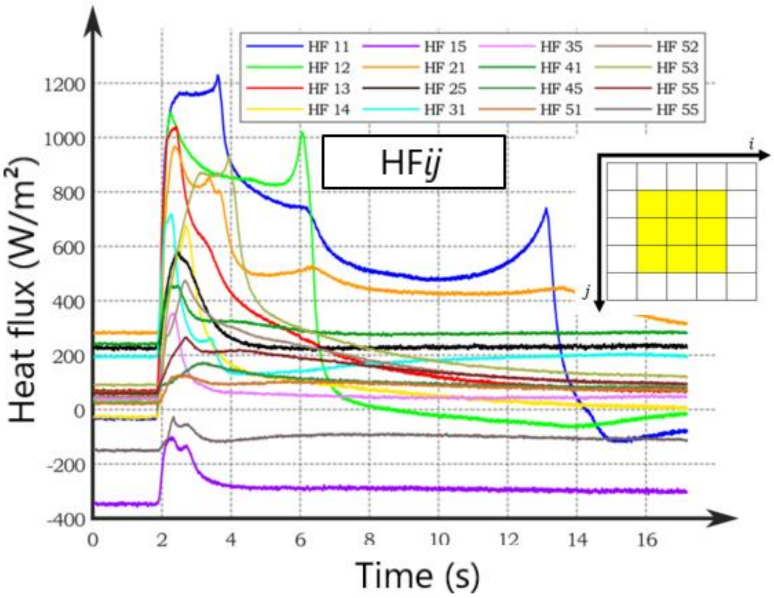


Figure 3. Time evolution of the local heat flux in the periphery of the heated wall resulting from the burst of the bubble generated inside the mother drop and spread of daughter droplets

54th European Two-Phase Flow Group Meeting
08-11 September 2024,
Dublin, Ireland
ABSTRACT

Dropwise and filmwise condensation on vertical hydrophilic surfaces: experimental investigation with saturated steam

M. Tancon, A. Abbatecola, S. Bortolin and D. Del Col

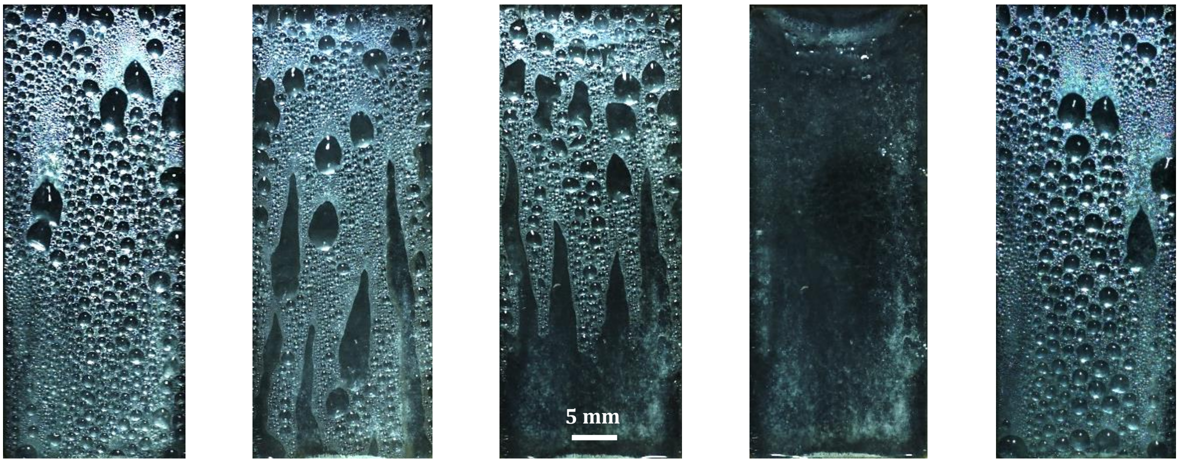
Department of Industrial Engineering, University of Padova,
Via Venezia 1, 35131, Padova, Italy
Contact E-mail: stefano.bortolin@unipd.it

Dropwise condensation (DWC) can significantly enhance the heat transfer with respect to classical filmwise condensation (FWC). The resulting condensation mode (FWC or DWC) depends on the operative conditions (e.g. heat flux, vapor velocity, saturation temperature,…) and on the interaction between the condensing fluid and the surface (wettability). Hydrophobic coatings (static contact angle ~ 90°) are usually employed to promote DWC on metallic surfaces. Promoting DWC on hydrophilic surfaces with low contact angle hysteresis can provide additional advantages such as reduced resistance associated with thermal conduction through the droplets, enhanced nucleation sites density and thus higher heat transfer coefficients (HTCs). Besides the challenging aspects in materials science concerning the fabrication of robust coatings, one aspect that must be further investigated is the ability of such hydrophilic surfaces to promote and sustain DWC without flooding, even at high values of heat flux.

In the present work, transition from dropwise to filmwise condensation is addressed. DWC of saturated steam (T_{sat} = 107.5 °C, vapor velocity 3.5 m s^{-1}) is investigated on two different aluminium sol-gel coated substrates named MTO-450 and MTO-300. The area of the samples where condensation takes place is 50 mm × 20 mm and the substrates were mirror-polished before the deposition of the coating. The aluminium substrate exhibits advancing contact angle θ_a = 60° and receding contact angle θ_r < 10°. After coating deposition, the wettability of the functionalized surfaces changed significantly, resulting in a marked reduction of the contact angle hysteresis: the MTO-450 coated surface displays θ_a = 57° and θ_r = 31°, while the MTO-300 coated surface presents θ_a = 71° and θ_r = 49°. The data obtained using the two samples (MTO-450 and MTO-300) are also compared against measurements previously obtained on a nearly hydrophobic surface named P7M3 (θ_a = 87°, θ_r = 64°).

The experimental apparatus used to study condensation is a two-phase thermosiphon loop. Steam is generated in a boiling chamber and flows into the test section where it is partially condensed over the vertically mounted aluminium sample, rejecting heat to the cooling water flowing on the back side of the specimen. Heat transfer measurements and video analysis were performed varying the saturation-to-wall temperature difference (ΔT_{sub}) and thus the heat flux q in the range 150-850 kW m^{-2}. As shown in Fig. 1, when varying the wall subcooling (ΔT_{sub}), different condensation modes were observed on the MTO-450 sample.

I) DWC	II) DWC with rivulets	III) DWC/FWC	IV) FWC	I) DWC



Time = 20 min	Time = 80 min	Time = 150 min	Time = 210 min	Time = 320 min
ΔT_{sub} = 1.2 K	ΔT_{sub} = 2.9 K	ΔT_{sub} = 11.5 K	ΔT_{sub} = 39 K	ΔT_{sub} = 1.3 K
q = 201 kW m^{-2}	q = 429 kW m^{-2}	q = 600 kW m^{-2}	q = 580 kW m^{-2}	q = 198 kW m^{-2}

Figure 1. Images taken during condensation on the MTO-450 coated surface when varying the wall subcooling ΔT_{sub} and thus the heat flux q. The corresponding condensation regime, time, wall subcooling and heat flux are reported.

Pure DWC (I) with almost circular-shaped drops was observed at low values of subcooling, whereas FWC (IV) covered the entire surface at high subcooling. For intermediate ΔT_{sub}, a progressive transition from DWC to FWC was observed (II and III) with the formation of elongated droplets and rivulets firstly, and then the gradual flooding of the surface starting from the bottom. The transition from DWC to FWC seems to occur when the condensation rate exceeds the rate at which the formed droplets are removed by coalescence and sweeping events. Once flooding of the surface was achieved at the highest wall subcooling, recovery of pure DWC was observed when reducing the wall subcooling.

The results showed that only the MTO-450, which displays the highest wettability, exhibited a complete transition from DWC to FWC when increasing the surface subcooling. In particular, pure DWC with constant values of HTC (170 kW m^{-2} K^{-1}) was obtained when $\Delta T_{sub} < 2$. For ΔT_{sub} values greater than 25 K, FWC occurs on the surface. In the case of the MTO-300 sample, for ΔT_{sub} up to 3.5 K, the HTC remained almost constant to approximately 135 kW m^{-2} K^{-1} (pure DWC). For higher values of surface subcooling, the HTC decreases as the condensation mode varied from pure DWC to DWC with the presence of elongated drops and short rivulets. The P7M3 sample sustained pure DWC across the entire range of subcooling ($3 < \Delta T_{sub} < 11$), achieving constant HTCs of about 100 kW m^{-2} K^{-1}. For both the P7M3 and MTO-300 samples, the flooding of the surface was not observed for the range of heat fluxes here investigated. The different values of HTC measured with the three coated surfaces that present different wettability are discussed comparing the experimental data against predictions obtained from a DWC model.

Overall, the results indicate that the promotion of DWC on hydrophilic surfaces is possible if the droplet mobility is sufficiently high (i.e. reduced contact angle hysteresis) and the condensation rate is sufficiently low. Furthermore, the higher the surface wettability, the higher the HTC during dropwise condensation (DWC).

54th European Two-Phase Flow Group Meeting
08-11 September 2024,
Dublin, Ireland
ABSTRACT

Numerical simulations and mechanistic modeling of high-pressure gas-liquid flow in M-shaped jumper of subsea gas production systems

A. Yurishchev, R.B. Ravid, A. Ullmann, N. Brauner

School of Mechanical Engineering, Tel-Aviv University, Tel-Aviv 6997801, Israel
Contact E-mail : ullmann@tauex.tau.ac.il, brauner@tauex.tau.ac.il

Two-phase gas-liquid flow with low liquid loads is common in high-pressure natural gas offshore gathering and transmission pipelines. The liquid phase may originate from condensates, reservoir water, or chemicals added to prevent hydrate formation. During gas production slowdowns (or complete shutdowns), the liquid phase tends to accumulate in the low sections of subsea pipelines. This study focuses on gaining insights into the transient gas-liquid flow within an M-shaped jumper, a crucial component of subsea high-pressure gas production pipelines. The displacement of accumulated liquid during production ramp-up induces temporal variations in pressure drop across the jumper and forces on its elbows, resulting in flow-induced vibrations (FIV) that pose potential risks to the structural integrity of the jumper.

To bridge the gap between laboratory experiments and field conditions, transient 3D numerical simulations of compressible gas and liquid flow were conducted using the OpenFOAM software. The simulations illustrate the impacts of various parameters, such as pipe diameter, gas pressure level, initial amount of accumulated liquid, liquid properties, and gas mass flow rate on the temporal variations of pressure drop across the jumper and the forces acting on the riser's upper elbow during liquid displacement by the gas flow. Furthermore, the study determines the critical gas mass flow rate necessary for the complete removal of initially accumulated liquid in the jumper.

The results of the numerical simulations facilitated the development of scaling rules for determining the critical gas velocity in realistic field conditions based on data obtained in lab-scale atmospheric systems (e.g., see Figure 1). Additionally, a mechanistic model is proposed to elucidate the factors contributing to increased pressure and forces during the liquid purging process in high-pressure systems. The dominant frequencies of pressure and force fluctuations were identified, with low-pressure systems exhibiting frequencies associated with two-phase flow phenomena and high-pressure systems showing frequencies attributed to acoustic waves.

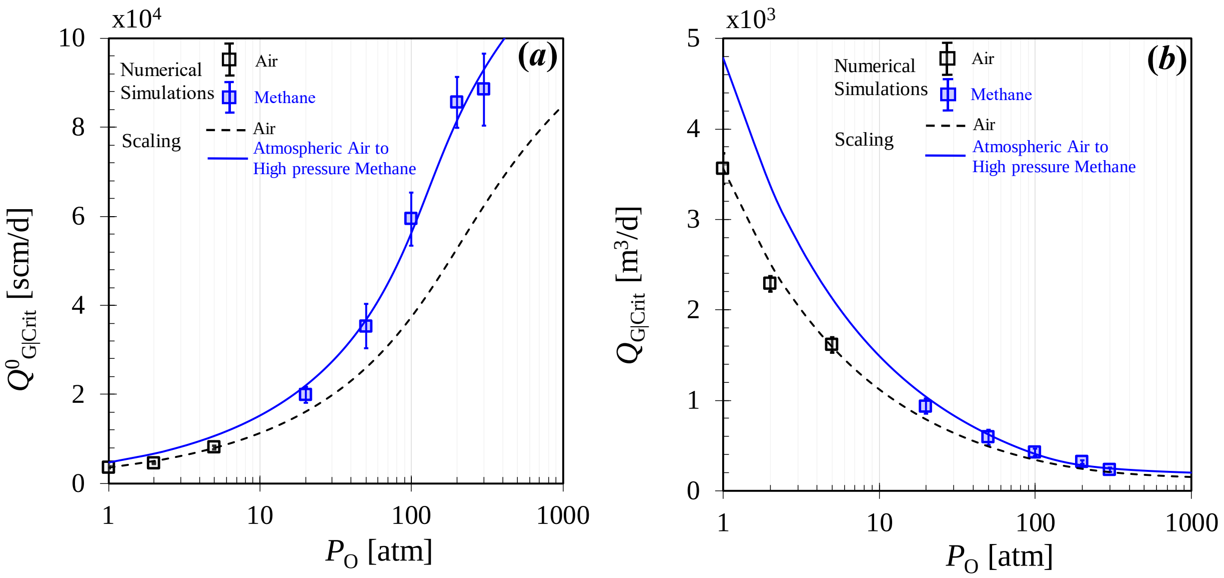


Figure 1: Results of numerical simulations (□) and scaling rules (--- and ━) regarding the critical gas volumetric flowrate as a function of gas pressure, P_O for air (dashed black) and methane (solid blue) with D = 50 mm at (*a*) at atmospheric pressure 20°C, $Q^0_{G|Crit}$ and (*b*) in-situ conditions, $Q_{G|Crit}$.

54th European Two-Phase Flow Group Meeting
08-11 September 2024,
Dublin, Ireland
ABSTRACT

Towards modelling mass transfer in poly-disperse bubble plumes

Y. Delauré[1], F. Dizaji[1], and T. Abadie[1]

[1]DCU Water Institute,
School of Mechanical and Manufacturing Engineering Dublin City University
Contact E-mail : yan.delaure@dcu.ie

Bio-chemical processes found in standard wastewater treatment plants often rely on aerobic suspended growth for biological treatment. So called activated sludge processes use aerobic micro-organisms in biological flocs to oxidise organic matter. They rely on a continuous supply of dissolved oxygen typically from the injection of air as bubble plumes. This may be achieved by injecting air supplied by blowers through bottom mounted disc diffusers or by injecting jets of water and air mixtures produced by submerged mechanical aerators. These aeration processes have large energy footprints and often account for the largest share of operational costs in activated sludge treatment. Previous research conducted at the DCU Water Institute considered a range of standard and innovative micro-bubble aeration technologies in both clean water and mixed liquor. Results confirmed and quantified the main benefits of micro-bubble aeration but also exposed open research questions, including on the sensitivity of smaller bubble plumes to external mixing [1].

The research presented aims to investigate whether optimal plume characteristics exist that will minimize energy consumption and maximize oxygen transfer. The research conducted in partnership with a hydraulic equipment supplier aimed to develop a simulation methodology to characterise air dissolution from the poly-dispersed bubble plumes and focussed on a mechanical aerator for testing at large scale. A dedicated simulation tool has been developed relying on the OpenFOAM libraries. The Eulerian-Lagrangian (EL) particle method has been coupled to LES and RANS turbulence models and transport equations for the dissolved gases as well as an actuator line model to simulate mixers. It has been shown to produce accurate predictions of the evolution of Dissolved Oxygen (DO) provided that the bubble plume can be characterised carefully, and that turbulence can be resolved within this EL framework. The bubble size distribution measured by a shadow sizing method and shown in Fig. 1 was found to overpredict the rate of DO increase when injected in an oxygen purged liquid volume. When bubble recirculation was considered in the estimation of the bubble size, much closer agreements between measurements and simulations were observed as shown in Fig.2 and detailed in [2]. Challenges remain with larger computational domains, when resolving turbulence scales becomes difficult. This is shown to have a significant impact on the dissolution of oxygen and nitrogen from the gas bubbles to the liquid phase.

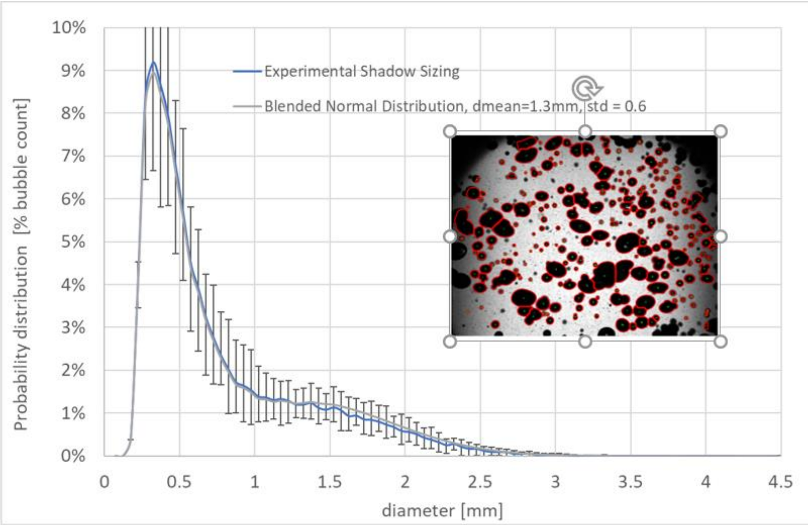


Figure 1. Measured size distribution of bubble plume

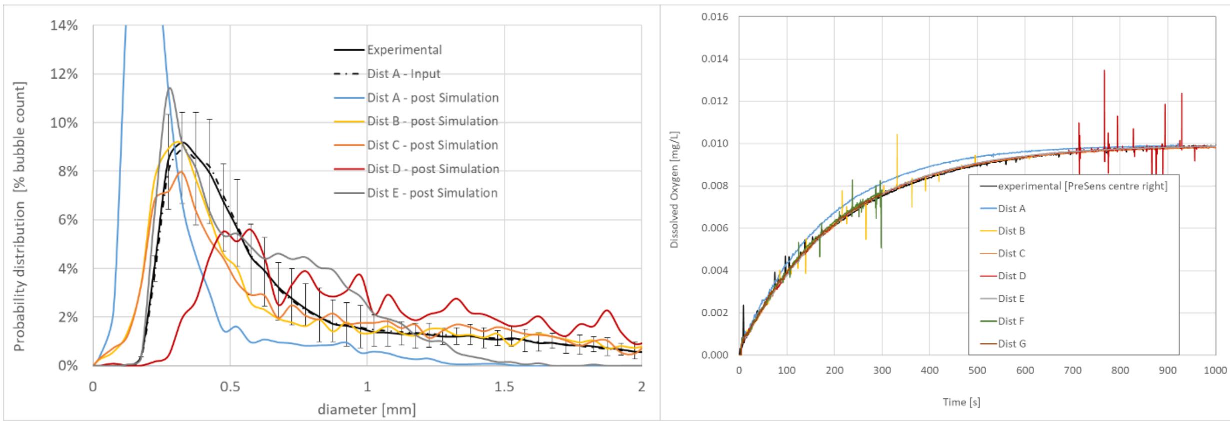


Figure 2. Sensitivity of the Dissolved Oxygen response curve with time [Right] to the bubble size distribution of the injected plume [Left]

References

1. Thomas Abadie, Sultan M. al Ma Awali, Brian Brennan, Ciprian Briciu-Burghina, Mohammad Tajparast, Thayse Marques Passos, John Durkan, Linda Holland, Jenny Lawler, Kieran Nolan, Brid Quilty, Lorna Fitzsimons, Fiona Regan, Yan Delauré, Oxygen transfer of microbubble clouds in aqueous solutions – Application to wastewater, Chemical Engineering Science, Volume 257, 2022, 117693, 10.1016/j.ces.2022.117693.
2. Farzad Faraji Dizaji, Micheál Cairns, Thomas Abadie, Abdulaleem Albadawi, Robert Connolly, Ben Breen, Stefan Berten, Ugo Bernabo, Yan Delauré, Significance of recirculating micro-bubbles in oxygenation from poly-dispersed plumes, International Journal of Multiphase Flow, Volume 165, 2023, 104408, 10.1016/j.ijmultiphaseflow.2023.104408.

Trinity College Dublin
Coláiste na Tríonóide, Baile Átha Cliath
The University of Dublin
ETPFGM 2024

54th European Two-Phase Flow Group Meeting
08-11 September 2024,
Dublin, Ireland
ABSTRACT

Flow boiling regimes in a horizontal-heated pipe

C.A. Dorao[1]

[1]Norwegian University of Science and Technology
Contact E-mail : carlos.dorao@ntnu.no

Flow boiling inside pipes has motivated vast research due to its relevance in industrial and everyday life applications. However, the physics of flow boiling remain under continuous scrutiny. In the case of flow boiling inside pipes, three regimes have been distinguished, namely convective flow boiling, nucleate flow boiling and a liquid deficiency boiling regime. At low heat fluxes and low working pressure, convective flow boiling is dominant, and the heat transfer coefficient depends on the mass flux and the thermodynamic quality. At high heat fluxes and high pressures, nucleate boiling is dominant, and bubbles produced at the wall are attributed the control of the heat transfer. In addition, experimental studies have shown that increasing the heat flux during flow boiling can led to a limiting value of the heat transfer coefficient before the boiling crisis. These three regimes have been the focus of diverse experimental studies. In this work, the focus has been to experimentally isolating working conditions where only one dominant mechanism play a dominant role. This work provides a new experimental framework for studying each heat transfer regime individually.

Trinity College Dublin
Coláiste na Tríonóide, Baile Átha Cliath
The University of Dublin
ETPFGM 2024

54th European Two-Phase Flow Group Meeting
08-11 September 2024,
Dublin, Ireland
ABSTRACT

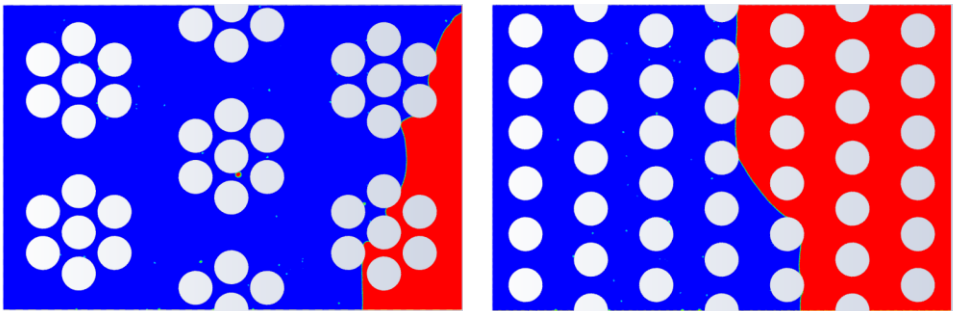
Capillary Flow Simulation in Biporous Media

M. E. Nimvari[1,2], T. Persoons[1] and M. J. Gibbons[1]

[1]Trinity College Dublin, University of Dublin, Ireland
[2]Amol University of Special Modern Technologies, Iran
Contact E-mail : eshaghnm@tcd.ie

Capillary flow in porous media is prevalent across numerous applications such as oil and gas recovery, ground water redistribution, heat pipe, fuel cell and drug delivery, to name a few, where the interplay of capillary forces and viscous losses governs fluid movement within pore structures [1]. The advent of biporous media has introduced a promising alternative to conventional monoporous media in capillary applications. Biporous structures (Figure 1a) are distinguished by their interconnected larger pores between clusters that facilitate permeability and smaller isolated pores within a cluster that enhance capillary pressure. Hence, this configuration allows for a simultaneous improvement in both capillary pressure and permeability [2].

Given the substantial complexities associated with conducting experimental studies in porous media, even for single-phase flow, there has been a growing emphasis on numerical simulations of heat and fluid flow in porous structures. Nevertheless, the challenges in accurately depicting the intricate geometry of biporous media have resulted in a limited number of numerical investigations to date [3].



(a) **(b)**

Figure 1 Water saturation at t = 4 ms for (a) proposed biporous and (b) monoporous geometries (The blue colour is water and the red one is air. Fluid flow direction is from left to right)

To address this research gap, the present study introduces a novel arrangement of biporous clusters in a staggered configuration that closely mimics a genuine biporous structure. Each cluster is composed of several particles arranged in a hexagonal grid (Figure 1a). Additionally, for monoporous media, a conventional staggered arrangement of solid particles is utilized in the simulation (Figure 1b). In addition, the study incorporates boundary conditions that replicate both passive and active capillary modes. While conventional research on capillary effects in porous media typically employs predetermined velocity or pressure (active capillary mode) as inlet conditions, the simulation of capillary pumping (passive capillary mode) offers a more accurate representation of capillary phenomena in porous media, albeit at a higher computational cost.

The volume of fluid method is utilized to simulate unsteady two-phase flow in this investigation. Initially, the new morphology for biporous media is validated by comparing simulation outcomes with experimental data, both in hydrodynamic [4] and evaporation heat transfer simulations [5]. The numerical results align with previous experimental findings regarding biporous media, indicating that the suggested biporous configuration effectively replicates two-phase flow in complex structures while keeping computational costs manageable.

The results reveal that capillary effects in biporous media can be as much as double those observed in monoporous structures. Furthermore, permeability in biporous media is improved by a factor of four under comparable conditions, with more than 95% of the mass flow rate occurring through the larger pores located between clusters. This simultaneous effect of enhanced capillary action and increased permeability results in superior wicking performance in biporous structures. This leads to greater invading velocity and saturation level in biporous media when compared to monoporous media, as illustrated in Figure 1.

While the presented simulation has the potential to revolutionise the modelling of two-phase flow in complex biporous media, additional evaporation heat transfer simulations are currently ongoing to explore biporous media performance as a heat pipe wick. Ultimately, the objective of this research is the design and optimization of biporous and hybrid porous media intended for phase change applications using additive manufacturing and surface engineering fabrication techniques.

Acknowledgements

This work was funded by a Royal Society-Science Foundation Ireland University Research Fellowship [Grant No. URF\R1\211776].

References

[1] S. Konangi, N. K. Palakurthi, N. K. Karadimitriou, K. Comer, U. Ghia, Comparison of pore-scale capillary pressure to macroscale capillary pressure using direct numerical simulations of drainage under dynamic and quasi-static conditions, Advances in Water Resources, 147 (2021) 103792.

[2] F. Zhou, Y. Liu, E. M. Dede, Design, fabrication, and performance evaluation of a hybrid wick vapor chamber, ASME - Journal of Heat Transfer, 141 (2019) 081802.

[3] L. Mottet, M. Prat, Numerical simulation of heat and mass transfer in bidispersed capillary structures: application to the evaporator of a loop heat pipe, Applied Thermal Engineering, 102 (2016) 770–784.

[4] T. Semenic, Y. Y. Lin, I. Catton, Thermophysical properties of biporous heat pipe evaporators, ASME - Journal of Heat Transfer, 130 (2008) 022602.

[5] F. C. Lin, B. H. Liu, C. C. Juan, Y. M. Chen, Effect of pore size distribution in bidisperse wick on heat transfer in a loop heat pipe, Heat and Mass Transfer, 47 (2011) 933–940.

Trinity College Dublin
Coláiste na Tríonóide, Baile Átha Cliath
The University of Dublin
ETPFGM 2024

54th European Two-Phase Flow Group Meeting
08-11 September 2024,
Dublin, Ireland
ABSTRACT

Pool boiling enhancement with porous Ti-6Al-4V coatings obtained by cold spray manufacturing using water as working fluid

A. I. Garivalis[1], Y. Chen[2], E. Shatskiy[2], A. Robinson[2], R. Lupoi[2] and P. di Marco[1]

[1] Department of Energy, Systems, Territory and Constructions Engineering, University of Pisa, Largo Lucio Lazzarino 1, 56122, Pisa, Italy
[2] Department of Mechanical, Manufacturing, and Biomedical Engineering, Trinity College, University of Dublin, Ireland
Contact E-mail : alekos.garivalis@unipi.it

Porous surface coatings offer great potential for improving heat transfer in industrial applications. We are investigating the efficacy of the metal additive manufacturing technique of cold spray deposition for the realisation of enhanced boiling surfaces, with a particular focus on Ti-6Al-4V (Ti64) coatings on aluminium substrates. The cold spray process provides a rapid and low-cost manufacturing method to produce lightweight enhanced boiling surfaces. Usually, the cold spray method is employed to create dense metal deposits. However, we have tuned the process to produce highly inhomogeneous honeycomb porous Ti64 coatings. Critical cold spray deposition parameters such as particle velocity, preheat temperature and deposition rate were identified to produce repeatable porous coatings with thicknesses up to 3.0 mm. In one of our earlier works, several samples were subjected to systematic boiling heat transfer tests in a purpose-built pool boiling apparatus using an organic refrigerant (FC-72) as working fluid. Initial data analysis shows that some of the surfaces tested exhibit a significant increase in boiling heat transfer coefficient and critical heat flux (CHF). This improvement may be attributed to increased surface area, increased nucleation site density, capillary wicking and mitigation of lateral bubble coalescence. However, excessive coating thickness degrades heat transfer due to the additional conductive resistance of the titanium layer. In conclusion, the novel Ti64 structured surfaces, developed using the cold spray deposition technique, have great potential for industries requiring enhanced boiling heat transfer performance. Within this study, we intend to extend the investigation using water as working fluid.

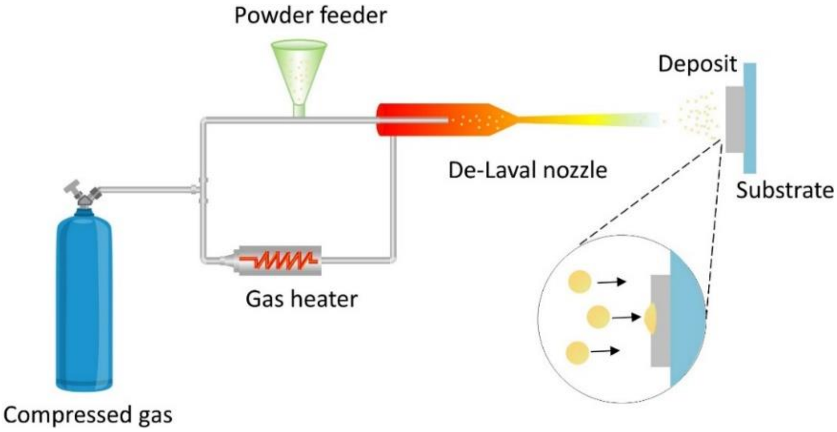


Figure 1. Cold spray additive manufacturing process.

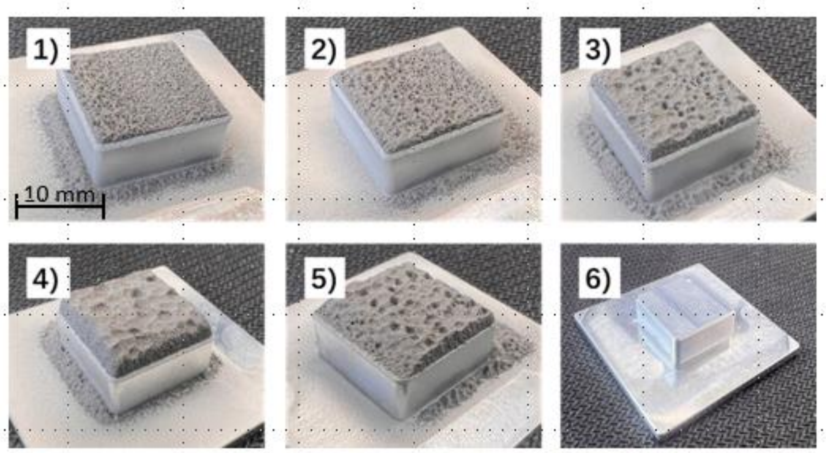


Figure 2. Coated samples (1-5) compared with the bare surface (6).

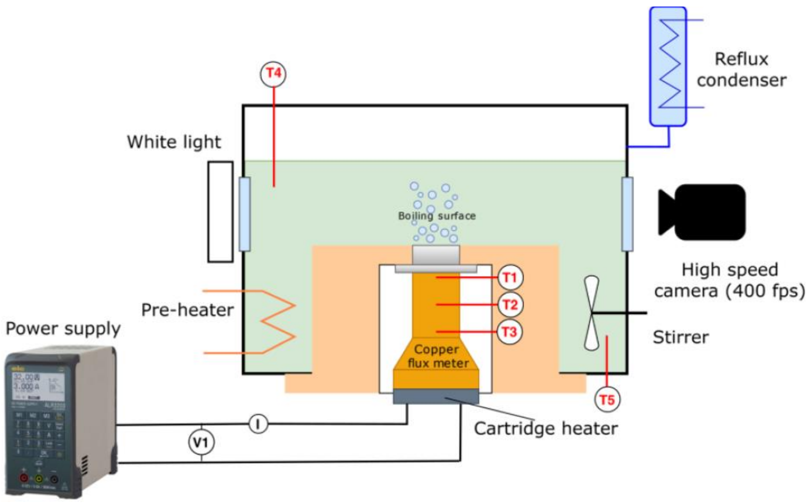


Figure 3. Experimental set-up for the boiling tests.

Trinity College Dublin
Coláiste na Tríonóide, Baile Átha Cliath
The University of Dublin
ETPFGM 2024

54th European Two-Phase Flow Group Meeting
08-11 September 2024,
Dublin, Ireland
ABSTRACT

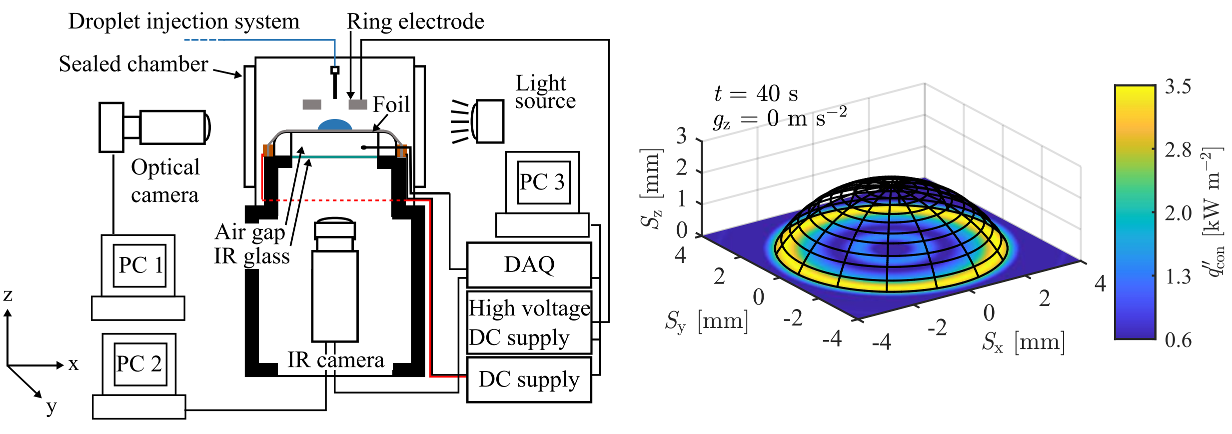
Droplet Contact Line Heat Transfer

M.J. Gibbons[1]

[1]Department of Mechanical, Manufacturing & Biomedical Engineering, Trinity College Dublin, the University of Dublin, Dublin 2, Ireland.

Contact E-mail: Michael.Gibbons@tcd.ie

The contact line is the region where the gas, liquid and solid phases intersect. A fundamental example of a contact line is the edge of a droplet on a solid surface. It is a ubiquitous natural phenomenon that surrounds our everyday life, from rain droplets on a window to condensation on a water bottle. Yet, we still do not fully understand it. Droplet evaporation is a complex interaction of diffusion within the substrate, buoyant convection in the gas and liquid phases, contact line evaporation, vapour diffusion, evaporative cooling at the liquid-gas interface and possible Marangoni effects. Albeit a proportionately small region compared with the overall droplet size, the heat and mass transfer at the contact line have been shown to be an important factor in droplet evaporation.



(a) (b)

Figure 1. (a) Schematic of the experimental apparatus, (b) 40 μL droplet 3D shape and heat flux distribution in microgravity on a heated foil (q''_{gen} = 590 W m^{-2}).

Sessile water droplet evaporation for varied surface wettability, electric field intensity and varied gravity conditions has been experimentally explored. Specifically, the influences of these parameters in the context of the heat flux distribution beneath the droplets using thin-foil thermography, as well as the droplet mechanics and resulting shapes. Terrestrial evaporation events were investigated for the entire droplet evaporation period, while varied gravity experiments (PFC 66) were limited to ~60 s periods corresponding to a parabolic flight parabola.

The peak heat transfer was located at the contact line for all cases. The research introduced the contact line density (CLD). The contact line density represents the ratio of the contact line perimeter to the total base area of the droplet. It was shown that the CLD largely dictates the relative proportion of the total heat transferred to the ambient via the low thermal resistance contact line region compared to the droplet contact angle or liquid-gas surface area. Across all experimental conditions, the average heat flux to the evaporating droplets was demonstrated to vary as a linear function of the contact line length density.

For varied droplet wettability and similar initial droplet volumes (~80 μL), the hydrophilic droplet dissipated a greater total power due to its larger contact line length and solid-liquid surface area, while the superhydrophobic droplet had a greater average heat flux due to its larger contact line length density for the majority of its evaporation time. The hydrophilic droplet evaporated 34% faster than its superhydrophobic counterpart due to its greater contact line length, liquid-gas interface temperature and solid-liquid surface area for the majority of its evaporation.

When an electric field is applied for both hydrophilic and superhydrophobic droplets, the net radial electric force is directed inward, resulting in a compressive force that influences the droplet shape in such a way that it appears elongated. Conversely, the net vertically directed electric force is determined to be downwardly directed for hydrophilic droplets, pressing the droplet to the surface, whereas it is upwardly directed for the superhydrophobic droplets, representing a lifting force. With regard to the heat transfer to the droplets, only a pronounced electric field effect was observed for the superhydrophobic droplet when the electric field acted to move the droplet contact line, suggesting that the heat transfer to the base of the droplet in the presence of an electric field is dominated by the electric fields influence, or lack thereof, on the contact line density.

For varied gravity conditions, the peak heat flux, average heat flux, and droplet evaporation rate were shown to vary strongly with gravity, with higher values noted for hypergravity conditions and lower values for microgravity conditions. The droplet thermal inertia was shown to play a significant role, with larger droplets taking more time to reach thermal equilibrium during the parabolic testing period. No significant impact of the electric field on the droplet evaporation was noted for these test conditions.

This work was a long-term collaboration with Dr Anthony Robinson, Dr Paolo Di Marco, Dr Seamus O’Shaughnessy, and Dr Alekos Garivalis, spanning several studies. It was facilitated by technical support from Trinity College Dublin and the University of Pisa.

ETPFGM 2024

54th European Two-Phase Flow Group Meeting
08-11 September 2024,
Dublin, Ireland
ABSTRACT

DC electric field impact on bubble dynamics

Tianle Gu[1] and Samuel Siedel[1]

[1]EPM Group, Laboratoire SIMaP, CNRS, Grenoble INP, Université Grenoble Alpes, France
Contact E-mail: tianle.gu@grenoble-inp.fr

Electrohydrodynamics (EHD) offers a promising avenue for enhancing and controlling boiling heat transfer by manipulating bubble growth dynamics and detachment sizes. This experimental study (see figure 1) investigates the impact of varying electric field strengths on bubble injection in HFE7100 at pressures of 300 mbar and 600 mbar. High-speed imaging was employed to capture and analyse successive bubble growth, focusing on parameters such as volume and aspect ratio under the assumption of axisymmetry. In the absence of an electric field (control group), volumetric growth curves demonstrated high consistency and linearity (see figure 2), indicative of a near-constant flow rate. When a DC electric field of 12 kV/cm is applied, the aspect ratio during growth exhibits significant elongation opposite to the direction of the electric field, suggesting the dominance of polarization forces. The examination of 14 successive bubbles reveals that the volumetric growth rate remains largely unaffected by EHD, except during the final stage if premature detachment occurs. Unexpected variations are observed concerning the growth duration and thus the detachment size of the bubbles. These variations seem to be linked to the waiting times prior to bubble growth and may also be related to EHD-enlarged contact angles at the triple line. These findings open new questions about the unsteady mechanisms of charge injection at the vicinity of the bubble base and the impact of the electrophoretic force on bubble dynamics. In parallel to the current study, a new experimental apparatus is under development to achieve greater control and precision in future investigations.

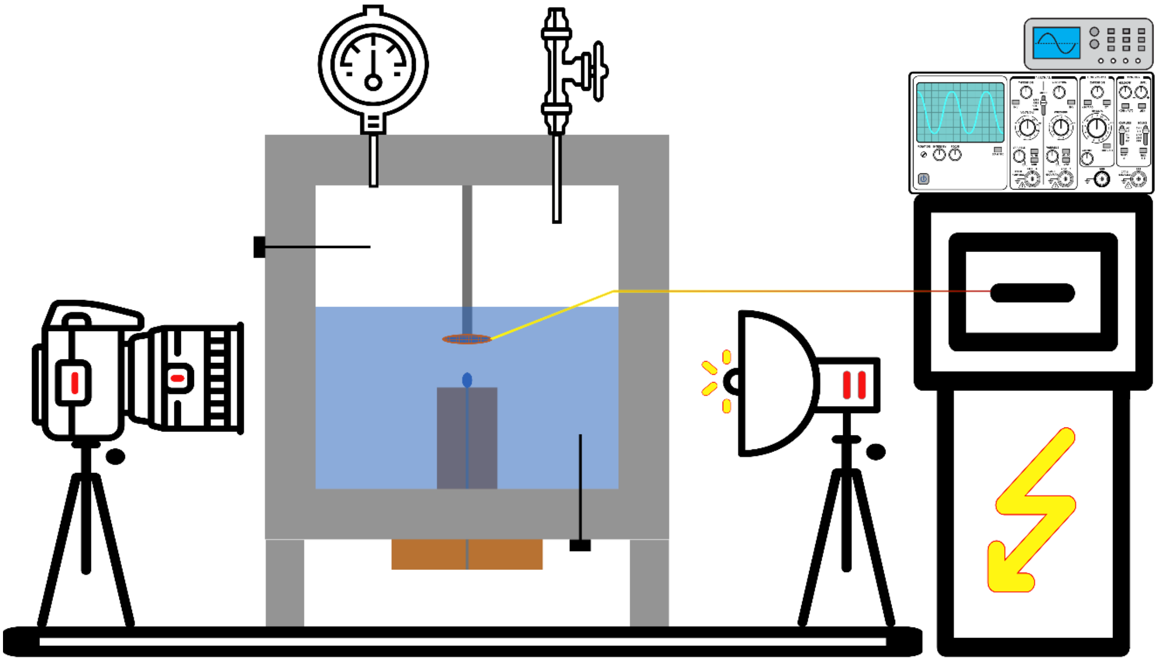


Figure 1 Schematic diagram of the experimental setup

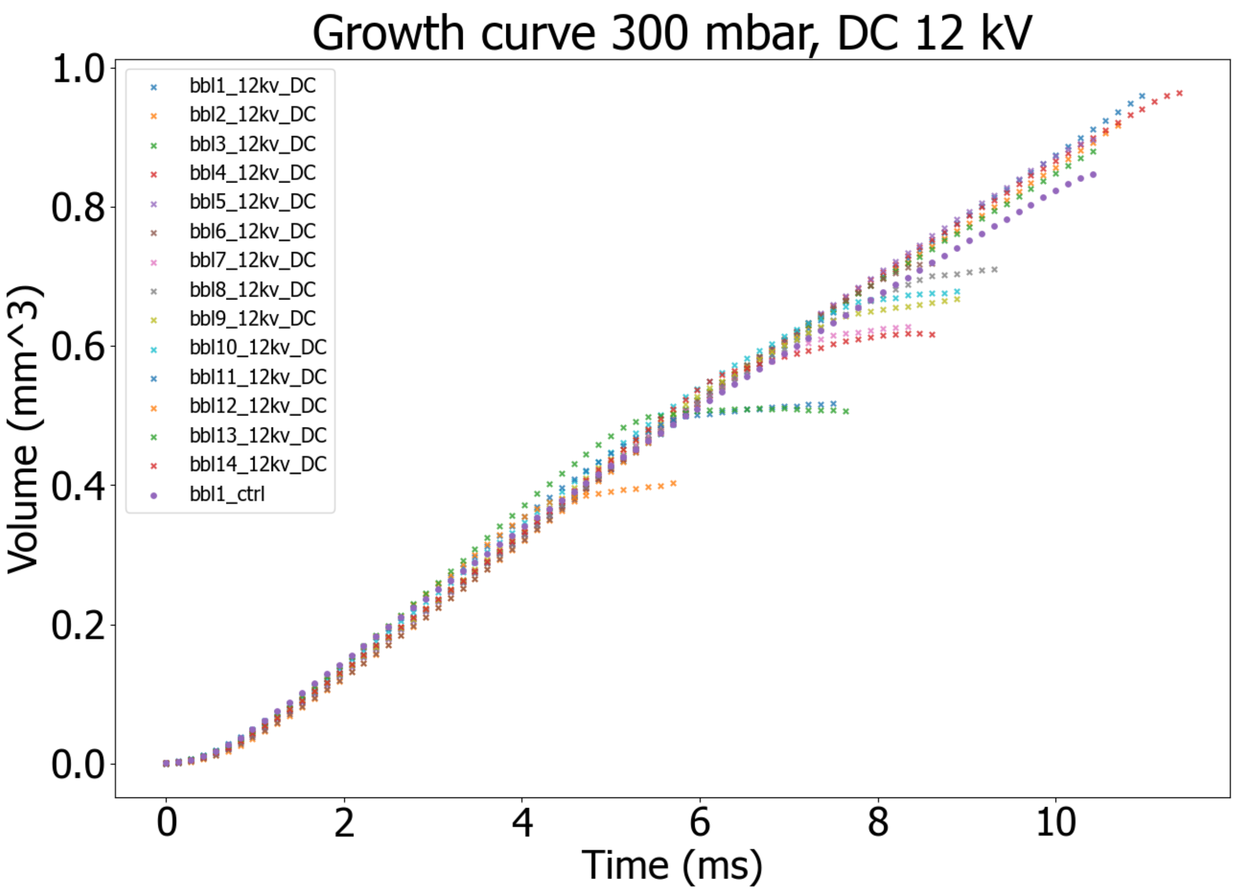


Figure 2 The volumetric growth curve of 14 successive bubbles under DC 12 kV

Trinity College Dublin
Coláiste na Tríonóide, Baile Átha Cliath
The University of Dublin
ETPFGM 2024

54th European Two-Phase Flow Group Meeting
08-11 September 2024,
Dublin, Ireland
ABSTRACT

From micro-droplets and bubbles to complex fluids

S. Fine, B. Uziel, H. Haustein

School of Mechanical Engineering, Tel-Aviv University, Tel-Aviv 6997801, Israel
Contact E-mail : hermanh@tauex.tau.ac.il

Two-phase gas-liquid flows are common to many engineering and medical applications. The present study reviews recent work at the MyFET lab at TAU, focused on the generation of microscale bubbles/droplets and their applications. For example, generation of mono-disperse micron-size bubbles of an oxidizer can improve water purification methods. Similarly, large quantities (>10^7) of micro-bubbles (<10µm) can be introduced into the body as a “contrast” agent for ultra-sound imaging, without the risk of an embolism. And finally, *complex fluids* such as a dispersion of boiling multiphase droplets can enhance micro-electronics cooling several-fold, both through phase-transfer and by enhancement of the host-liquid’s (“single-phase”) mixing. Such a fluid introduces additional design parameters – bubble/droplet size, inlet distribution and mass fraction/ phase’s flow rates. It also has significant *advantages* over traditional flow boiling: at low mass fractions it can be a “drop-in” solution for existing single-phase flow systems; while at intermediate fractions it can be designed to reduce the chance of dry-out, both through wettability aspects and by improved vapor evacuation (proper inlet distribution and total vapor limitation).

Firstly, experimental and numerical studies relating to the lower limit for generation of micro-bubbles is presented. This system, tackles the classic problem of air in water, without the use of additional surfactants. It is shown that an inertial approach to the problem, using high liquid flow rates and a sub-micron gas injection needle very small bubbles (~5µm) can be generated in the pipe-inlet region. Therein, both the flow *accelerations* and *velocities* up to 10m/s create significant drag on micro-bubbles. These very small bubbles are not only quite spherical but very rigid as well, contributing to a skin-friction drag which is negligible at larger scales. The experimental findings were further supported by two-phase simulations and results from the literature, with trends forecasting the possibility of generating nano-bubbles (~100nm) in lower surface tension/higher viscosity fluids.

Secondly, the boiling of a volatile dispersion (Iso-hexane in DI water) was found to obtain an optimal heat transfer at intermediate heat fluxes and flow rates (~40W/cm^2, Re_{Dh}=800) – showing up to a three-fold increase over single-phase flow, even at low mass-fractions (~2%). Analysis showed that the primary enhancement was due to increased mixing of the host liquid by the droplets/bubbles. Whereas the reduction in heat transfer at higher heat fluxes, was associated with rapid boiling, excessive bubble coalescence and reduced mixing. It was concluded that higher flow rates and/or increased channel divergence should lead to better vapor evacuation and further enhancement, especially at higher mass fractions. Two-bubble coalescence was observed to occur at the timescale of bubble growth (<1ms), which was successfully predicted by the RCD model. Thus providing a single-droplet predictive tool for future design of the boiling multiphase droplets, generated through microfluidics, contributing to the optimization of heat transfer.

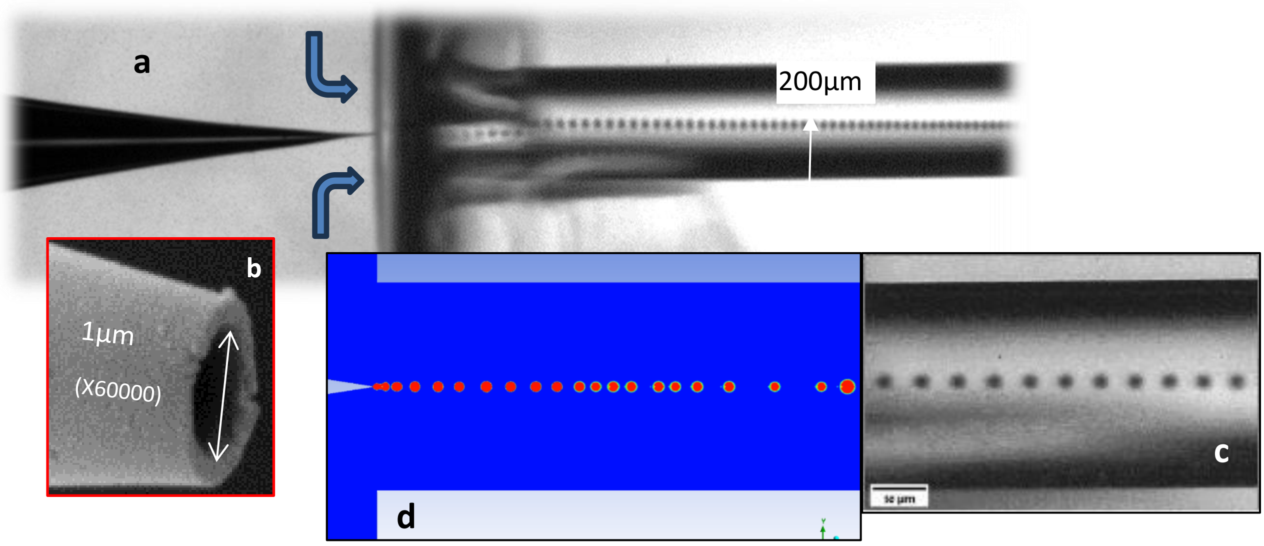


Figure 1: Micro-bubble generation: a) inertial micro-bubble generation system (100<Re<1,500); b) SEM image of 1micron gas injection needle; c) typical close-up of bubbles (~10µm, ~100kHz, Re=800); d) similar bubbles in numerical simulation (Δp_{lg}=4atm. ~U_l=4.3 m/s, Re=860)

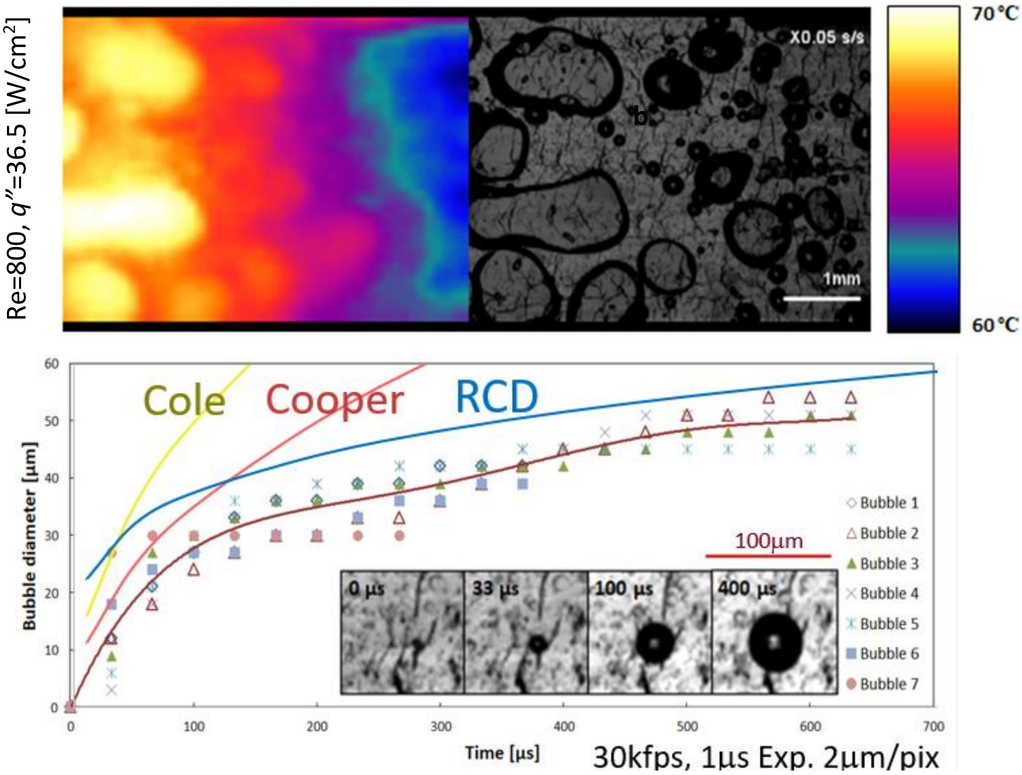


Figure 2: Micro-droplet boiling: a) dispersion boiling visualization (right) vs. corresponding wall temperature (Left) – High-speed IR imaging of transparent (ITO) heating surface; b) characteristic bubble growth curves and prediction by RCD models vs. other models, inset: visualization of typical bubble – reaching complete boiling at 50µm diameter after only 650µs.

54th European Two-Phase Flow Group Meeting
08-11 September 2024,
Dublin, Ireland
ABSTRACT

Advanced Ultrasound Techniques for the Characterization of Multiphase Flows

F. Hossein[1] and P. Angeli[1]

[1]ThAMeS Multiphase, Department of Chemical Engineering, University College London, Torrington Place, London WC1E 7JE, UK

Contact E-mail : f.hossein@ucl.ac.uk

This talk will present the development of non-intrusive ultrasound techniques for measuring particle or drop size, volume fraction distributions, and velocity profiles in multiphase flows. The study demonstrates the potential of ultrasound techniques as a diagnostic tool for characterizing complex flows. Ultrasound techniques have many advantages, such as their ability to penetrate any material, allowing their use with opaque equipment walls, as well as safety, robustness, and low cost compared to other current techniques such as imaging, laser-based diagnostics, X-ray, and MRI. We developed ultrasound techniques for measuring flow properties in solid-liquid and liquid-liquid flows. In these systems, the ultrasound attenuation coefficient, sound speed, and frequency shift were used to measure volume fraction, drop or particle size, and dispersed phase velocity, respectively. The experiments were carried out in different setups, including a fluidized bed and liquid-liquid contactors such as pipes, small channels and stirred vessels. An innovative approach that combines advanced ultrasound techniques with machine learning algorithms to characterize particle size distribution and volume fractions in solid-liquid flows will also be presented. The data from the ultrasound techniques were compared with those obtained from other techniques, including high-speed imaging and planar laser-induced fluorescence (PLIF), demonstrating very good agreement in all cases.

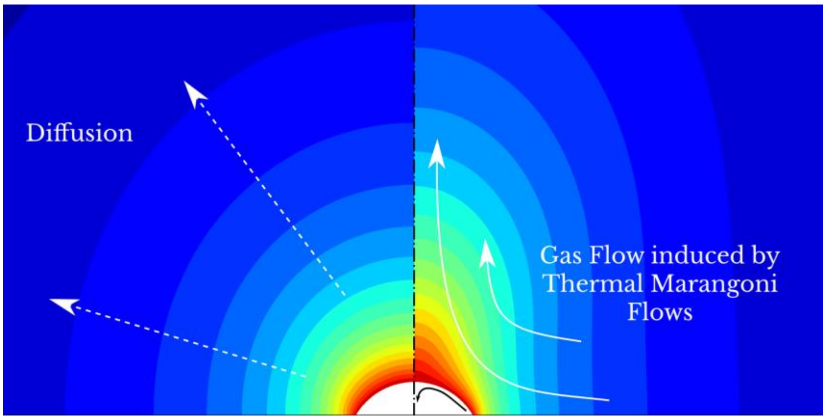
Trinity College Dublin
Coláiste na Tríonóide, Baile Átha Cliath
The University of Dublin
ETPFGM 2024

54th European Two-Phase Flow Group Meeting
08-11 September 2024,
Dublin, Ireland
ABSTRACT

Advanced Research on Liquid Evaporation in Space: A sounding rocket experiment

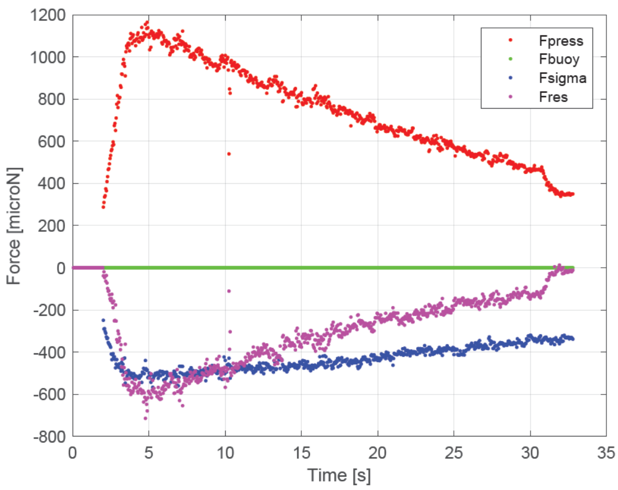
Carlo Saverio Iorio[1], Hatim Machrafi[1], Claire Perfetti[1]

1 Centre for Research and Engineering in Space Technologies – Université libre de Bruxelles - Belgium
Contact E-mail : carlo.iorio@ulb.be

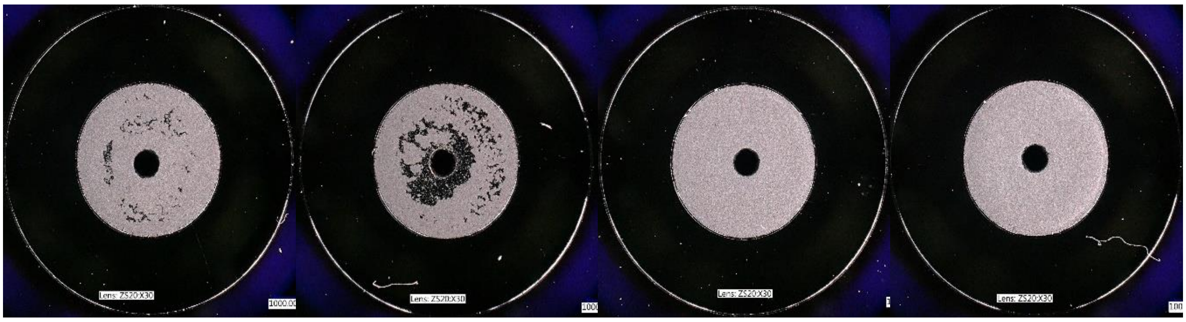


The ARLES experiment performed on the Sounding Rocket Maser 14 and Maser 15 intended to study evaporating drops of pure fluids, possibly containing a low concentration of nanoparticles. A specific module of the experimental rig was dedicated to the study of multidrop deposition, where droplets seeded with nanoparticles were injected subsequently to study the self-assembled patterns formed in micrograty. The study of pure fluid droplets' evaporation supported the development of the ISS experiment "DROP EVAPORATION".

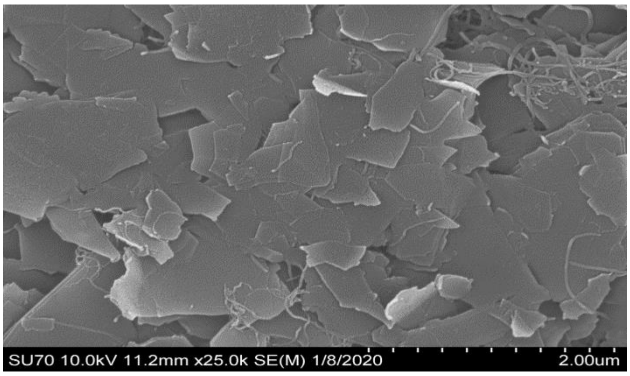
The experiment lead to important scientific findings: i) The evaporation rate measured during the microgravity phase was surprisingly smaller – up to 2.5 times smaller than the ones foreseen by the state-of-art models for such kind of configuration. This occurrence could be linked to the dynamic of the vapour cloud around the droplet during evaporation. The phenomenon is even more pronounced when nanoparticle dispersions are used. In that case, the vapour cloud evolves in a pure diffusive state, and the experimental evidence shows that the thermal Marangoni convection is completely stopped. The evaporation rate in this latter case is at least one order of magnitude smaller than what was theoretically calculated; ii) The application of the electric field enhanced the evaporation rate by more than 20%. The shape of the droplet appeared elongated due to circumferential electrical stress, and a preliminary force balance revealed that the resulting electric force is pressing the droplet against the surface. Interferometric analysis of the vapour could show a different vapour concentration distribution that seems to be induced by the dielectric force acting on the vapour phase; iii) For what concerns the multidrop experiment, it was possible to spot some original features of the deposition patterns. At first, it was not observed the characteristic coffee ring effects in any deposited droplets. This occurrence is even more surprising

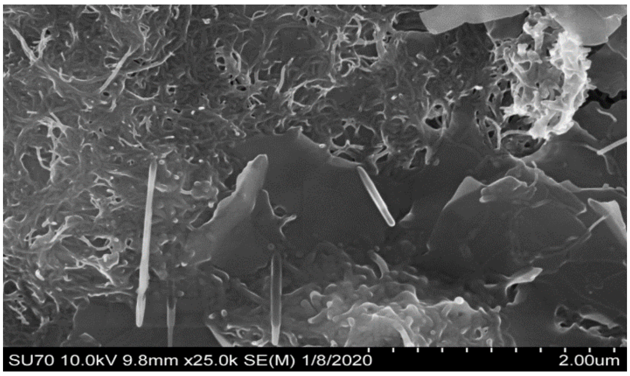


1 droplet **2 droplets** **3 droplets** **4 droplets**



by watching at the first droplet deposited on the substrates and shows complete different behavior respect to the one observed on the ground where not only the first droplet nanoparticles accumulated close to the pinning point, but subsequent droplets created a series of trough and peaks in the radial direction. In the picture below, it is clear the uniformity of the substrate coating starting from the third droplet





Trinity College Dublin
Coláiste na Tríonóide, Baile Átha Cliath
The University of Dublin
ETPFGM 2024

54th European Two-Phase Flow Group Meeting
08-11 September 2024,
Dublin, Ireland
ABSTRACT

Effect of bubble deformation and contamination on the stability of bubble chains

Dominique legendre[1], Omer Atasi[1], Mithun Ravisankar[2], Roberto Zenit[2]

[1]Institut de Mécanique des Fluides de Toulouse, Université de Toulouse, CNRS, 31013 Toulouse, France, [2]Center for Fluid Mechanics, School of Engineering, Brown University, Providence, Rhode Island 02912, USA
Contact E-mail : legendre@imft.fr

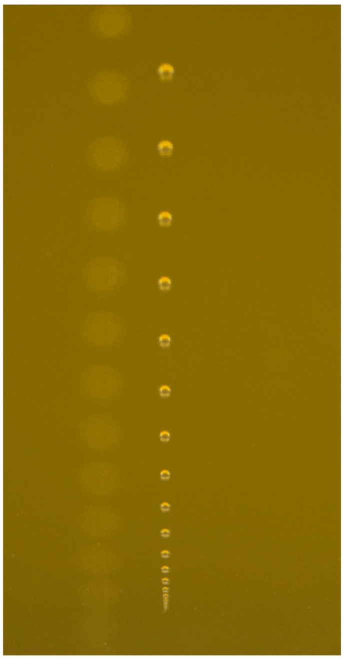


Figure 1. Bubble chain observed in carbonated drink [1]

Bubble interaction and clustering have impact in many two-phase problems of current importance from industrial application to everyday life, for example when a carbonated drink is poured in a glass [1]. Depending on the beverage different mode of interaction are observed. Very stable bubble chains are clearly observed in champagne or in beer (see figure), showing an almost straight line from microscopic nucleation sites from which they are continuously formed. In some other drinks such as soda, such chains are not straight (not stable) and bubbles are dispersed in the liquid. Considering pair interactions for spherical clean bubbles, bubble chains should not be stable [2], which contradicts these observations. The aim of this presentation is to explain the conditions for bubble chain stability and instability [3]. For this purpose, experiments and direct numerical simulation are reported. The bubble size as well as the level of contamination of the bubble interface are shown to affect the bubble chain stability. The transition from stable to unstable behavior results from the reversal of the lift force experienced in the wake of the leading bubble. The conditions of transition from stable to unstable bubble chains is discussed considering the production of interfacial vorticity at the bubble surface.

[1] R Zenit, Rodriguez, 2018, Physics Today
[2] Y Hallez, D Legendre, 2011 J. Fluid Mech., 673, 406-431.
[3] O Atasi, M Ravisankar, D Legendre, R Zenit, 2023, Physical Review Fluids 8 (5), 053601

Trinity College Dublin
Coláiste na Tríonóide, Baile Átha Cliath
The University of Dublin
ETPFGM 2024

54th European Two-Phase Flow Group Meeting
08-11 September 2024,
Dublin, Ireland
ABSTRACT

An Euler-Euler based CFD framework for multiphase flows combining resolved and unresolved structures

D. Lucas, F. Schlegel, R. Meller, B.Krull, R. Lehnigk

Helmholtz-Zentrum Dresden-Rossendorf
Bautzner Landstr. 400
01328 Dresden
Germany
Contact E-mail : d.lucas@hzdr.de

For simulations on multiphase flows in medium and large-scale industrial applications the Euler-Euler approach is most frequently used and often the only feasible one. In many flow situations, the involved interfaces cover a wide range of scales leading to different coexisting morphologies. Established simulation methods differ for the different interfacial scales. Large interfaces are represented in a resolved manner usually basing on the one fluid approach, e.g. Volume of Fluid (VOF) or Level Set. Unresolved (dispersed) flows are modelled using the two- or multi-fluid approach. A simulation method that requires less knowledge about the flow in advance would be desirable and should allow describing both interfacial structures – resolved and unresolved – in a single computational domain.

The morphology adaptive multifield two-fluid model *MultiMorph*, which is developed at HZDR, is able to handle unresolved and resolved interfacial structures coexisting in the computational domain with the same set of equations. An interfacial drag formulation for large interfacial structures is used to describe them in a VOF-like manner, while the usual closure models are applied for the unresolved phases. In addition, MultiMorph allows to simulate transitions between the morphologies. This concerns both physical transitions such as entrainment and detrainment as well as transitions resulting from a change in the size of the numerical mesh within the domain, if this changes the resolvability of a phase interface. Figure 1 shows a general scheme of the different representations and transitions.

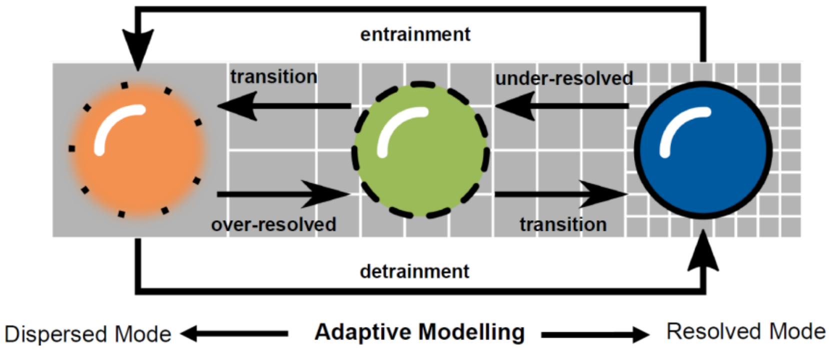


Figure 1. General Scheme of morphology transfers in MultiMorph

In the talk, the general approach and the handling of the transfer are presented and illustrated with simulation examples, including

- a single bubble rising in a stagnant bubble column represented by a numerical grid with changing size,
- bubbles rising through a stratified air-oil interface,
- disintegration and accumulation in an oil-water phase inversion
- complex flows at a column tray of a distillation column,
- gas core formation in a swirl separator
- air layer formation for drag reduction of a ship and
- gas entrainment as shown in Figure 2.

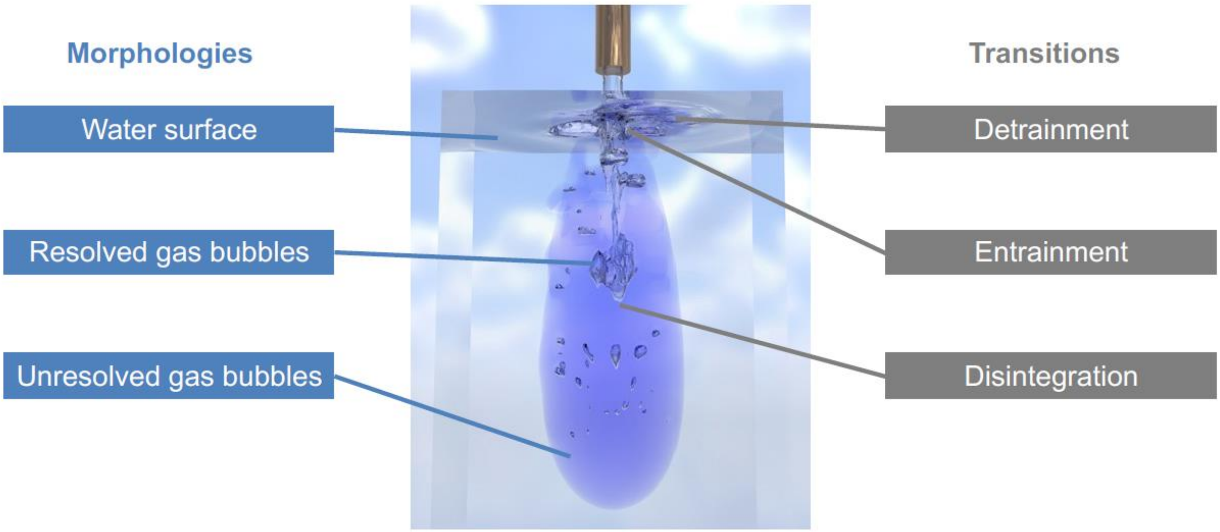


Figure 2. Simulation of a plunging jet with MultiMorph model with marked morphologies and phase transition regions.

There are other research groups that proposed similar modelling frameworks. The Dresden6 initiative was launched to encourage collaboration.

References

Meller, R., Schlegel, F., & Lucas, D. (2021). Basic verification of a numerical framework applied to a morphology adaptive multifield two-fluid model considering bubble motions. *International Journal for Numerical Methods in Fluids*, *93*(3), 748-773. 10.1002/fld.4907.

Meller, R., Tekavčič, M., Krull, B., & Schlegel, F. (2023). Momentum exchange modeling for coarsely resolved interfaces in a multifield two-fluid model. *International Journal for Numerical Methods in Fluids, 95(9),* 1521-1545. 10.1002/fld.5215.

Schlegel, F., Meller, R., Krull, B., Lehnigk, R., Tekavčič, M. (2022). OpenFOAM-Hybrid - A Morphology Adaptive Multifield Two-Fluid Model. *Nuclear Science and Engineering*, 1-14. 10.1080/00295639.2022.2120316.

Schlegel et al. Multiphase Code Repository by HZDR for OpenFOAM Foundation Software, Rodare. http://doi.org/10.14278/rodare.2321

Trinity College Dublin
Coláiste na Tríonóide, Baile Átha Cliath
The University of Dublin
ETPFGM 2024

54th European Two-Phase Flow Group Meeting
08-11 September 2024,
Dublin, Ireland
ABSTRACT

Laser-based thermographic measurements of nucleate flow boiling in a vertical miniaturised channel

Zengchao Chen[1], Surya Narayan[1,2], Aleksei S. Lobasov[1], Konstantin S. Pervunin[1] and Christos N. Markides[1,*]

[1]Clean Energy Processes (CEP) Laboratory, Department of Chemical Engineering, Imperial College London, SW7 2AZ London, United Kingdom
[2]Multiphase Thermal and Energy Research Laboratory (MTERL), Department of Mechanical Engineering, Indian Institute of Science, 560012 Bengaluru, Karnataka, India
[*]Contact E-mail: c.markides@imperial.ac.uk

The rapid increase in the thermal power dissipated in modern electronic devices along with a continued miniaturisation trend has promoted flow boiling in miniaturised and microchannels as one of the most promising solutions for effective heat dissipation in next-generation high-power-density electronic components, especially with low-boiling-point dielectric fluids. Despite promising recent advances in flow boiling research, a fundamental understanding of the thermal and hydrodynamic interactions between the continuous and dispersed phases taking place in confined boiling flows is still lacking, due to the challenge of obtaining reliable and detailed data in these flows. Experimentally, further progress can arise from the development and application of advanced non-intrusive optical measurement techniques, which can be used to provide insight into the complex thermal-fluid phenomena that underly these flows, along with reliable data for the development and validation of advanced numerical and theoretical predictive models.

In this work, an experimental investigation of flow boiling inside a vertical miniaturised square (5-mm) channel was performed, using dielectric HFE-7100. A new whole-field laser-based thermographic method referred to as single-dye multispectral planar laser-induced fluorescence (SDMS-PLIF) was employed to quantify the temperature fields in the bulk liquid phase. In its implementation, Nile Red, a thermosensitive lipophilic dye, was dissolved as the fluorophore in in HFE-7100, with ethanol used as an auxiliary solvent. SDMS-PLIF captures the fluorescence emission from two spectrally separate wavelength bands and, thus, allows spatiotemporally-resolved temperature-field measurements by evaluating the ratio between the intensities of these two thermally-dependent spectral regions. This method minimises the effects of non-negligible laser/excitation intensity variations due to optical reflection and/or refraction, and of non-uniform dye concentration gradients due to thermodiffusion on fluorescence thermography.

The test section comprises a base plate and a lid made of polycarbonate and was vapour polished for good optical clarity and accessibility, as shown in Fig. 1. At the downstream end of the vertically-oriented miniaturised channel, an indium tin oxide (ITO)-coated sapphire substrate is flush-mounted as a one-sided electrical heater, leaving a fully developed entrance length. The experiments were performed in an upward laminar flow at selected conditions (heat and mass flux and inlet subcooling degree) to achieve a nucleate flow boiling regime. The flow was illuminated with a thin light sheet generated by a high-speed Nd:YLF laser (527 nm, 20 kHz), equipped with a commercial sheet-optics arrangement. Two high-speed cameras were placed on the same side of the test section but perpendicular to each other to observe the flow through a beam splitter, at which two appropriate bandpass filters were mounted (Fig. 1a). An infrared camera was installed opposite at an angle to the heated wall to record the wall temperature distribution (Fig. 1b).

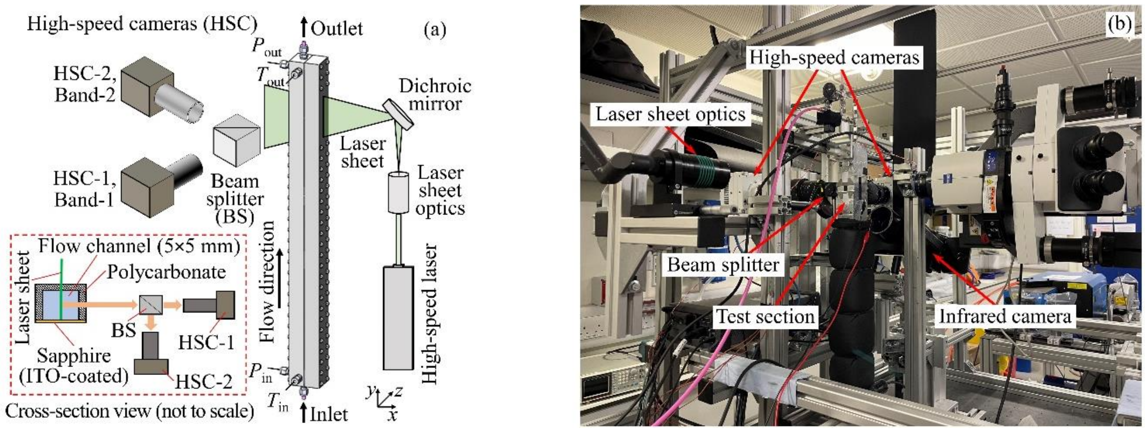


Figure 1. (a) Schematic, and (b) photograph of the flow apparatus and measurement arrangement.

Figure 2a shows that, with an increase in temperature, the emission spectrum of Nile Red shifts towards shorter wavelengths λ, resulting in an increase in the fluorescence intensity for $\lambda <$ 575 nm and a reduction in intensity for $\lambda >$ 575 nm. Thus, two wavelength bands with opposite temperature sensitivities, namely 550 < λ < 570 nm (Band 1) and λ > 650 nm (Band 2), were selected for SDMS-PLIF. As seen in Fig. 2b, while growing and sliding along the heated wall, nucleated vapour bubbles displace a portion of hot fluid out of the thermal boundary layer, affecting the local temperature field in the colder bulk phase in the flow core via the formation of thermal plumes. The individual vapour inclusions sometimes coalesce, resulting in the growth of thermal plumes around newly formed larger bubbles (Fig. 2b). This phenomenon can be described as bubble-induced mixing and is expected to improve the thermal transport rate which is one of the most important characteristics of nucleate boiling regime.

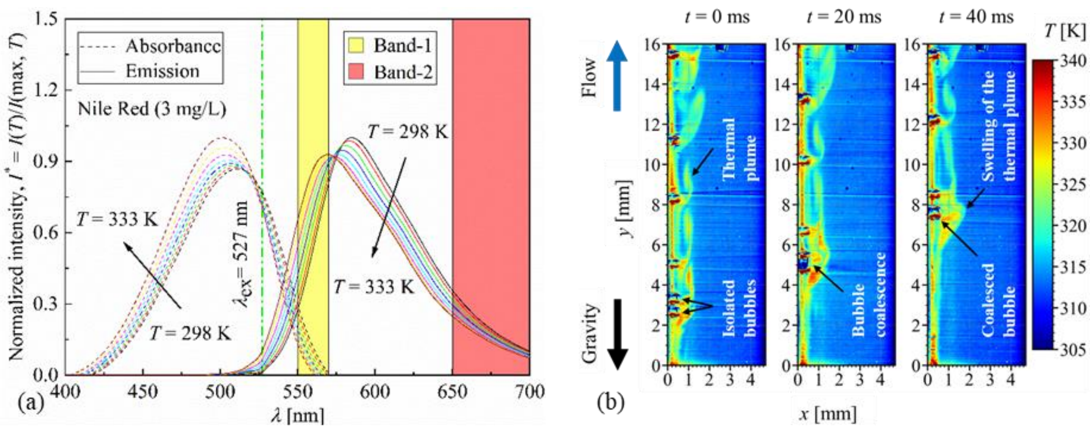


Figure 2. (a) Normalised absorbance and emission spectra of Nile Red (3 mg/L) in HFE-7100 with 1%-vol. admixture of ethanol at different temperatures. (b) Instantaneous temperature fields showing different phenomena typical of nucleate flow boiling.

In summary, a new measurement method (SDMS-PLIF) has been developed and is being applied to flow boiling in miniaturised channels. First-of-a-kind information is being made available on the interactions between vapour bubbles and the thermal fields in the nucleate flow boiling regime.

54th European Two-Phase Flow Group Meeting
08-11 September 2024,
Dublin, Ireland
ABSTRACT

Flow boiling and flow condensation at small, medium and high reduced pressures

M. Pysz, S. Głuch and D. Mikielewicz

Gdańsk University of Technology, Faculty of Mechanical Engineering and Ship Technology
Contact E-mail : dariusz.mikielewicz@pg.edu.pl

Progressing miniaturization in industrial and most importantly in electronic applications is one of the most researched subjects nowadays. The complexity of those small systems leads to several engineering challenges. One of them is the creation of lightweight heat exchangers capable of removal of high heat fluxes. The flow boiling and flow condensation are currently considered as one of the most efficient heat transfer method. However, the phenomena involved in that process are still not fully understood, especially on the fundamental side. There is a gap of knowledge in flow boiling and flow condensation of low boiling point and perspective refrigerants at high reduced pressures and high saturation temperatures. The increased interest in the knowledge of pressure drop and heat transfer close to thermodynamic critical point arose from the potential applications in the design of Organic Rankine cycle (ORC) evaporators or high temperature heat pumps evaporators.

Most of available data refer to refrigeration and air-conditioning applications, which means the lower values of reduced pressures, namely 0.2 to 0.4. The data bank for the moderate and higher reduced pressures is not as abundant. Recently, ORC systems and high-temperature heat pumps attracted the attention of researchers. Implementing such technologies requires basic knowledge of heat transfer and pressure drop at much higher saturation temperatures (80 ˚C and above). In the vicinity of the thermodynamic critical point, the surface tension forces are less pronounced because they disappear at the critical point and subside to gravity forces (decrease in Bond number value). Additionally, the density and viscosity of the vapour phase increase while the opposite happens for the liquid phase. As a result, the velocities of both phases are close to each other (liquid phase velocity rises while vapour phase velocity decreases). Most up-to-date models and correlations related to this topic were performed for water and carbon dioxide but only a few for new low boiling point refrigerants.

In the presentation the following issues are to be addressed:

1. Identification of the effects of various flow parameters (mass velocity, heat flux.) on heat transfer and pressure drop at moderate and high reduced pressures in minichannels. The study showed, especially in the case of heat transfer, that the obtained trends are strongly related to the reduced pressure value. Generally, the heat transfer coefficient increases with the reduced pressure.
2. Identification of the effects of flow structure on the heat transfer and pressure drop at moderate and high reduced pressures. The flow structures have a tremendous impact on heat transfer. In the bubbly flow region, the heat transfer is usually independent of mass flux and vapour quality and increases with heat flux. This starts to change in the

churn/wavy-annular flow where heat transfer increases with vapour quality and the effect of heat flux slowly diminishes.

3. Validation of the basic and improved in-house heat transfer models against the experimental database on flow boiling and flow condensation. The improvement of the model is based on the implementation of the reduced pressure effect into the term responsible for fine tuning of nucleate boiling effect and re-evaluation of the two-phase flow multiplier. The collected database was also compared against several selected well-known models found in the literature.
4. Verification of the model against the experimental database collected from the literature. The improved in-house model showed good accuracy, usually reaching MAPE below or close to 30%.

Finally some critical assessment of up to date progress in modelling of flow boiling and flow condensation at small, medium and high reduced pressures will be presented and the directions for future research indicated.

Trinity College Dublin
Coláiste na Tríonóide, Baile Átha Cliath
The University of Dublin
ETPFGM 2024

54th European Two-Phase Flow Group Meeting
08-11 September 2024,
Dublin, Ireland
ABSTRACT

A mathematical model and mesh-free numerical method for contact-line motion in lubrication theory

L. Ó Náraigh[1], K. E. Pang[1], R. Smith[1]

[1] School of Mathematics and Statistics, University College Dublin
Contact E-mail : onaraigh@maths.ucd.ie

We introduce a mathematical model with a mesh-free numerical method to describe contact-line motion. The model is based on lubrication theory and as such applies only to slender droplets. A regularization of the contact-line singularity is introduced involving a small parameter, similar to other models (e.g. slip-length, precursor film). The result is the *Geometric Thin-Film Equation*, so called because the equation of motion is derived using concepts from geometric mechanics [1].

Inherent in the model is a set of weak singular solutions, which can be used to construct an approximate solution to the thin-film equation. The singular solution consists of delta functions whose centres evolve over time according to a set of ordinary differential equations – so-called particles. Each particle is equipped with a positive weight. This approach provides a mesh-free solution of the Geometric Thin Film Equation. Crucially, the centres of the delta functions cluster in regions of high interfacial curvature, thereby mimicking the effect of adaptive mesh refinement, without the computational cost of that method. Hence, the approach bears some resemblance to Smooth Particle Hydrodynamic; another analogy would be with peakons in the Camassa-Holm equation for water waves [2] A key advantage of the current approach is the mathematical underpinning, which facilitates a rigorous convergence proof – namely that the approximate solution converges to a full solution of the model in the limit as the particle number tends to infinity.

We provide a proof-of-concept of the method by showing that the Geometric Thin-Film Equation reproduces Tanner's Law of droplet spreading (e.g. Figure 1, and Reference [2]). We describe our efforts to extend the equation to partial wetting. Finally, we will map out efforts to extend the concept beyond lubrication theory, for future applications in multiphase flow modelling.

[1] Holm, D.D., Schmah, T. and Stoica, C., 2009. *Geometric mechanics and symmetry: from finite to infinite dimensions* (Vol. 12). Oxford University Press.
[2] Camassa R, Holm DD. An integrable shallow water equation with peaked solitons. Physical review letters. 1993 Sep 13;71(11):1661.
[3] Pang, K.E. and Ó Náraigh, L., 2022. A mathematical model and mesh-free numerical method for contact-line motion in lubrication theory. *Environmental Fluid Mechanics*, *22*(2), pp.301-336.

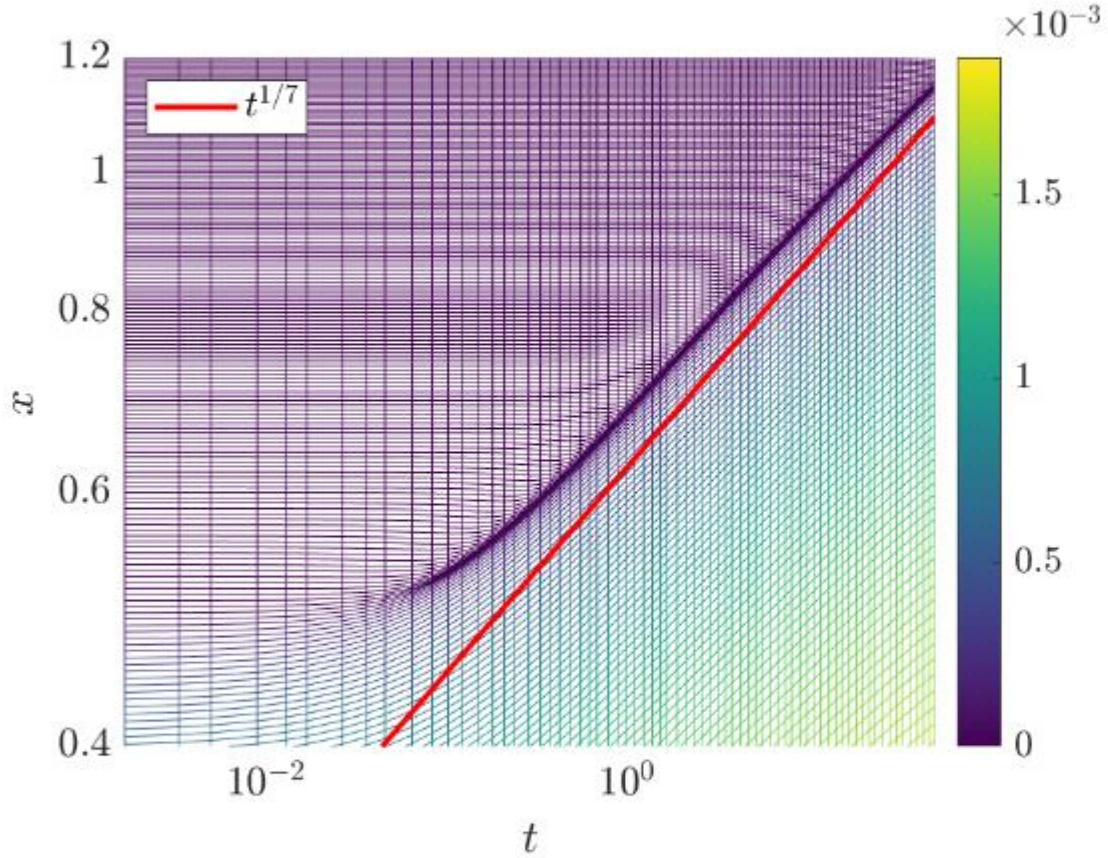


Figure 1. Evolution of the particle trajectories (logarithmic scales on both axes). The colours indicate the weight corresponding to each particle. The fact that the trajectories follow the power law $t^{1/7}$ (red line) shows that the contact line moves at the rate given by Tanner's Law (exponent 1/7 for the 2D case)

54th European Two-Phase Flow Group Meeting
08-11 September 2024,
Dublin, Ireland
ABSTRACT

Enhancing Condensation using Co-Blast Surfaces

E. Shatskiy, P.S. Kumavat and S.M. O'Shaughnessy

The University of Dublin, Trinity College
Contact E-mail: oshaugse@tcd.ie

Experimental research of dropwise condensation (DWC) has intensified over the past number of decades. Many studies have focussed on mechanisms to promote DWC, typically consisting of substrate/surface treatments in the form of coatings and/or surface structures, which encourage the development of droplets on said surface. Despite the growing attention on the subject, the application of DWC to industrial processes still poses significant challenges. To the authors' knowledge, stable and consistent DWC over a prolonged period has yet to be realised. Due to time-varying process parameters and degradation of surface treatments, DWC is very difficult to maintain.

Despite many studies' stated objectives of achieving DWC for improved heat transfer, many do not accurately measure or report heat transfer coefficients, or any thermal data. Moreover, a significant number of published works unknowingly used pressure sensors that were incompatible with the working fluid, likely leading to unreported large errors in the results. Often, studies report little or no quantification of uncertainty which can be very significant, making reported heat transfer performance data unreliable.

It is well established that surface topography influences the development of dropwise condensation on a surface as it affects droplet sliding angle and contact angle hysteresis. Recent research suggests that high contact angle with low hysteresis promotes droplet shedding and higher heat transfer rates. Surface roughness has been shown to promote the formation of droplets by providing nucleation sites which act as triggers for droplet formation. Nucleation can be triggered by impurities, ions, foreign particles, or cavities, to which individual molecules of a vapour can attach. The existence of nucleation sites on the condensation surface is integral to dropwise condensation promotion. The highly disordered phenomenon of condensation can be controlled on a chemically homogeneous surface by creating physical heterogeneity (surface topography). The creation of various surface features can facilitate the removal of droplets at smaller sizes. Various approaches exist and can be categorized as micro, nano, and micro-nano binary roughness, porous structures and coatings, hybrid surfaces, micro-grooved surfaces, and lubricant infused surfaces.

Low surface energy chemistry combined with a micro and nano scale roughness, can be developed in several ways: by coating low surface energy material over a rough substrate, by creating roughness over a low surface energy substrate, or by single step deposition of low surface energy rough coatings over smooth substrates. Various coating types are prevalent in literature such as polymer coatings, noble metals, rare earth oxides, self-assembled monolayers (SAMs), ion implanted surfaces, switchable wettability surfaces (SWS), graphene, and carbon nanotubes. Regarding coatings, there are several key considerations for industrial use:

- Low coating thickness is typically required to reduce the thermal resistance.
- Strong adhesion with the substrate is required for durability.
- Coatings must be resistant to fouling and be economic to deposit on the substrate.

This study aims to explore if Co-Blast, an established industrial fluoropolymer surface coating technique produced by the company ENBIO Ltd., can enhance condensation heat transfer on a vertically inclined surface, and if so, by how much. Three condensation surfaces are investigated, each with a surface area measuring 50mm x 50mm. All samples are manufactured from aluminium. One smooth surface is used as a baseline case. Two other condensation surfaces are coated using the Co-Blast method with perfluoroalkoxy alkane (PFA) and fluorinated ethylene propylene (FEP), respectively.

Experiments use distilled water heated in a 316 stainless-steel boiler by four 1kW-rated cartridge heaters. Vapour is transported through stainless-steel tubing into the condensation chamber, where temperature and pressure probes monitor the saturation conditions. A viewing window, made from clear polycarbonate, forms part of the front surface of the chamber. A cutout in the back wall of the chamber allows for the insertion and replacement of the condenser test surfaces, which are in contact with an aluminium meter bar, itself cooled by an external recirculating chiller. 4-wire precision resistance temperature detectors (RTDs) enable calculation of the meter bar heat flux and extrapolation to the condensation surface temperature, necessary to calculate the subcooling and heat transfer coefficient.

The performance of the experimental apparatus is first verified by comparing results using the baseline aluminium surface with established correlations for filmwise condensation (FWC). Then a series of tests is performed using the coated surfaces. The results for heat transfer coefficient, shown in Figure 1, show significant improvement using the coated surfaces, which was sustained over a period of days. DWC was observed for both surfaces, and no transition to FWC was observed over the narrow range of subcooling investigated.

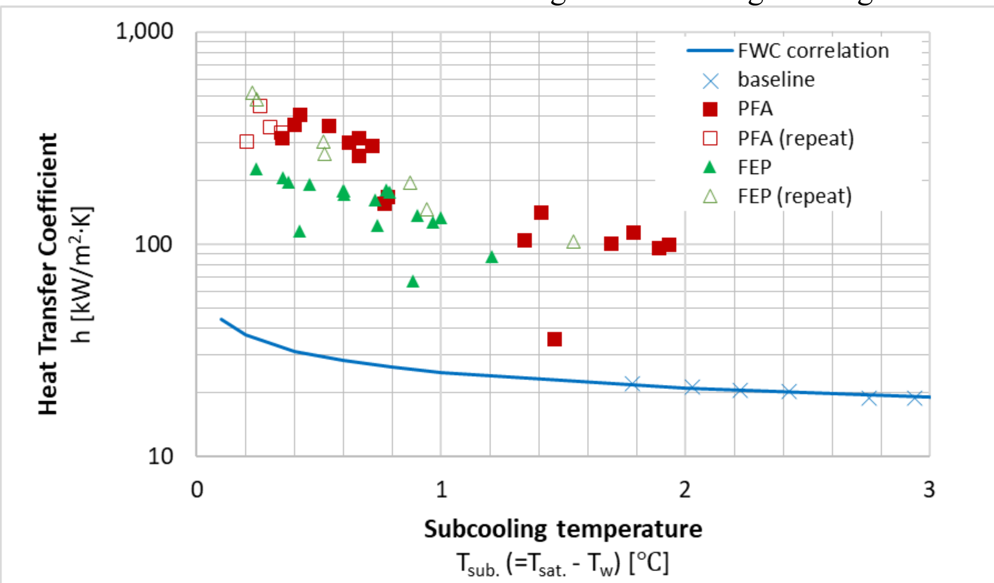


Figure 1. Heat transfer coefficient versus subcooling for the baseline surface, PFA coated surface, FEP coated surface, and comparison with established correlations for FWC.

Although the results are quite promising and repeatable, they are preliminary. Further investigations are required, in collaboration with our industrial partner, to properly characterise the surfaces and coatings, to assess their robustness and durability, and to explore a wider range of subcooling. Challenges remain regarding parameter uncertainty at very low subcooling. Future work should include other working fluids, particularly low surface tension fluids.

Acknowledgements The authors wish to gratefully acknowledge the support of industrial collaborator ENBIO Ltd. who provided the surface coatings for this study.

Flow boiling heat transfer coefficient in enhanced tubes: benchmark correlations and ANN comparison

A. F. Passarelli[1], R. Mastrullo[1], R. Revellin[2], L. Viscito[1] and A. W. Mauro[1]

[1]Department of Industrial Engineering, Federico II University of Naples, P.le Tecchio 80, 80125 Naples, Italy
[2]INSA Lyon, CNRS, CETHIL, UMR5008, 69621 Villeurbanne, France
Contact E-mail: wmauro@unina.it

Nowadays, refrigeration and air-conditioning industries are involved in a huge increase of demand, thus the heat transfer enhancement of heat exchangers is crucial to reduce both systems environmental impact and total costs. Fin and tube or shell and tube heat exchangers are widely employed in these sectors and since several enhancement have been reached for the one phase flow heat transfer (related mostly to air or water), having an approach between the internal and external heat transfer resistances, the two-phase heat transfer enhancement then has become a topic of interest. In particular, internal micro-fin tubes are commonly employed because of their enhancement effects such as promoting flow turbulence, delaying dry-out occurrence, mitigating boiling suppression due to forced convection and enlarging heat transfer surface. During the last 30 years, several experimental studies have been conducted to investigate the effect of different micro-fin geometries on flow boiling physics with the development of some predictive methods for heat transfer coefficient prediction. These methods have often been developed using small datasets in terms of operative conditions and fluids using frequently only one type of micro-fin geometry. Therefore, general accuracy is still an open question that drives the research towards the development of new predictive methods. Despite reduced parameters physical modelling has been for years the only way for predictive methods development, recently machine learning techniques have started to be accounted as a new predictive method. Among them, artificial neural networks seem to show a good potential due to their ability to create links between inputs and outputs replicating the non-linear relationships between variables involved in the physical phenomena. However, dataset needed to develop accurate and general neural networks should be large and accurate because this tool can be unreliable beyond development training ranges. For this reason, recent studies have started to analyse how to join machine learning techniques with physical knowledge, having the development of physics informed neural networks based on transfer learning techniques. According to the actual state of the art about predictive methods, a general assessment and comparison between reduced parameters physical modelling and artificial neural networks is something not very present in literature. In this regard, starting from a large experimental database collected from literature, the aim of the work is to present a comparison between benchmark flow boiling correlations and artificial neural networks, evaluating also correlations results as a parameter on which to build the network. The database has been collected from about 30 papers, with a total of 3179 points. It includes mass flow rates from 50 to 1000 $kgm^{-2}s$, vapour qualities from the onset of boiling to the dry-out occurrence and beyond, reduced pressure from 0.05 to 0.80 and fin tip diameters from 0.7 to 11.9 mm. Regarding micro-fin geometries, most studies focused on helical configuration, with trapezoidal micro-fin shape, having about 70% of the database with these features. Afterwards a literature review, six quoted correlations have been chosen for this work. In particular, starting from two

of the first correlations for micro-fin tubes such as Thome et al. and Cavallini et al. ones, using the Chen assumption as a starting point of correlation development, more recent ones can be considered as evolution of the previous two such as Wu et al., Diani et al. and Tang and Li methods. Instead, considering also correlations developed using only regression analysis, the one by Rollman and Spindler is often cited. On the other hand, several artificial neural networks have been trained varying their structures in terms of layers number (1 to 4), neurons per layer (10 to 80) with uniform and not uniform distributions on neurons among layers. Input sets have been defined selecting the most influencing variables for output prediction, having dimensional, non-dimensional and mixed sets. According to the main idea of the transfer learning, also correlation results have been considered as possible inputs. Training processes based on back-propagation algorithm involving 70% of the database, while the remaining 30% was used for validation and test phases. Table 1 show the statistical comparison in terms of mean absolute percentage error (MAPE) and number of points within ±30% error band ($\delta_{\pm 30\%}$). Despite almost all correlation show an intermediate accuracy with the one by Wu at el. showing a $\delta_{\pm 30\%}$ of 57%, some common outliers can affect significantly their performance as shown in brackets. Also, Rollman and Spindler one is not usable on the whole dataset. Instead, artificial neural networks present a very good accuracy, outperforming correlations for both the one with mixed input and the one using Cavallini correlation as input.

Predictive method	**N° of points**	**MAPE**	$\delta_{\pm 30\%}$
Thome et al. (1997)	2947 (3146)	50 (92)	44
Cavallini et al. (2006)	2947 (3146)	114 (162)	30
Diani et al. (2014)	2947 (3146)	51 (94)	54
Rollman and Spindler (2016)	2408	54	40
Tang and Li (2018)	2947 (3146)	43 (114)	44
Wu et al. (2013)	2947 (3146)	49 (195)	57
ANN (Mixed, 3 layers, 80-64-51 neurons)	3146	9	96
ANN (Cavallini, 2 layers, 50-25)	3146	30	79

Table 1. Assessment of the most quoted correlations and best artificial neural networks

However, in the local analysis shown in figure 1 it is clear that correlated informed neural network results in a more reliable predictive approach instead of standard artificial neural network that can bring to some unacceptable results. Therefore, artificial neural networks can be a useful tool in particular when predictions needed do not fall beyond training ranges, but using a physical approach could be a promising path to extend their reliability, resulting in a trade-off between correlations and pure machine learning.

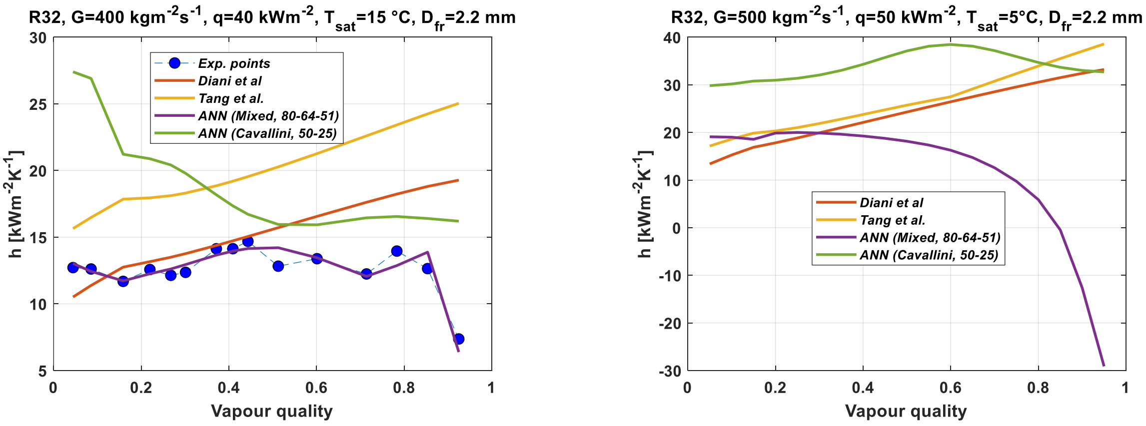


Figure 1. Local analysis of the heat transfer coefficient with respect of vapour quality within (left) and beyond (right) artificial neural networks training ranges

Hydrodynamics and extraction abilities of novel solvents in continuous small channels

Phakoukaki Yiota- Victoria, Panagiota Angeli

ThAMeS Multiphase, Department of Chemical Engineering, University College London, Torrington Place, London WC1E 7JE, UK

Contact E-mail : yiota.phakoukaki.18@ucl.ac.uk

This work focuses on the development of advanced solvent extraction processes using novel solvents such as ionic liquids (ILs) and aqueous biphasic systems (ABS) within small channels. These solvents present significant advantages over traditional organic solvents, which are often volatile, toxic, and environmentally hazardous. Aqueous biphasic systems (ABS), known for their high water content (up to 70%) and economic viability, offer improved extraction efficiencies while being environmentally friendly. Ionic liquids (ILs), have good solubilities and very low volatility, low flammability, and high thermal and chemical stability, compared to organic solvents, which are advantageous for extraction separations.

Continuous flow processing in small channels intensifies heat and mass transfer compared to batch systems, while the increased importance of interfacial forces results in well-defined and stable flow patterns. To better predict processes such as liquid-liquid extractions in small channels it is important to understand the hydrodynamics of two phase flows.

The talk will present our extensive studies on the hydrodynamics of two-phase flows of these novel solvents in small channels. The different flow patterns that form will be presented, including parallel, annular plug and drop flows. The effects on the flow patterns of the unique physical properties of ILs and ABS, the channel sizes and the phase flow rates, will be discussed. Some results on the solvent extraction in the small channels for applications relevant to pharmaceuticals purifications and to metal extractions will also be presented. The results demonstrate high extraction efficiencies exceeding 90%, achieved within residence times of just a few seconds compared to several minutes needed in conventional equipment used in industry such as mixer-setters or pulsed columns.

Trinity College Dublin
Coláiste na Tríonóide, Baile Átha Cliath
The University of Dublin
ETPFGM 2024

54th European Two-Phase Flow Group Meeting
08-11 September 2024,
Dublin, Ireland
ABSTRACT

Characteristics of Taylor bubbles moving in a shear-thinning fluid

D. Picchi[1] , A. Aquino[1], and P. Poesio[1]

[1]Department of Mechanical and Industrial Engineering, University of Brescia, Brescia 25123, Italy,
Contact E-mail : davide.picchi@unibs.it

The motion of elongated Taylor bubbles in micro-channels is frequently encountered in many industrial processes, such as small-scale reactors, coating processes, and microfluidic devices. For example, in medicine, it is crucial for the design of modern targeted microbubbles for drug delivery or understanding lung activity and air embolism. Although the study of confined Taylor bubbles is a canonical problem in fluid mechanics, a fundamental understanding of the problem is still an open issue when the fluids exhibit non-Newtonian behavior. Examples are biological solutions, emulsions, and polymers that behave like shear-thinning fluids, and their effective viscosity is a function of the imposed shear rate.

In this talk, we investigate the dynamics of Taylor bubbles that move in an inelastic shear-thinning fluid that obeys the Carreau-Yasuda viscosity model by means of numerical simulations. We focus on regimes where inertia and buoyancy are negligible to assess the effect of the fluid rheology on bubble characteristics up to finite capillary numbers. First, we compare the results with the scaling laws obtained by lubrication theory by analyzing the trends of the film thickness and bubble speed. Then, we show the existence of a general scaling law for the effective viscosity that embeds both the zero-shear rate and shear-thinning effects and holds up to finite capillary numbers. Interestingly, the shape of the Taylor bubble is strongly influenced by the fluid rheology, which competes with the capillary number. Finally, the analysis of the viscosity fields shows an interplay between the zero-shear rate and shear thinning effects in different regions of the bubble, including the presence of recirculating vortexes that form ahead and behind the bubble.

These results clarify the effect of fluid rheology on bubble characteristics, and although the motivation of our work is oriented toward understanding the dynamics of a single Taylor bubble, the scaling laws obtained may serve as a base for constructing more sophisticated models for trains of bubbles. We conclude by showing that the existence of a master curve for effective viscosity appears to be typical of a more general class of problems, including capillary imbibition (when a shear-thinning fluid invades a single pore) and flow in ducts with complex geometry.

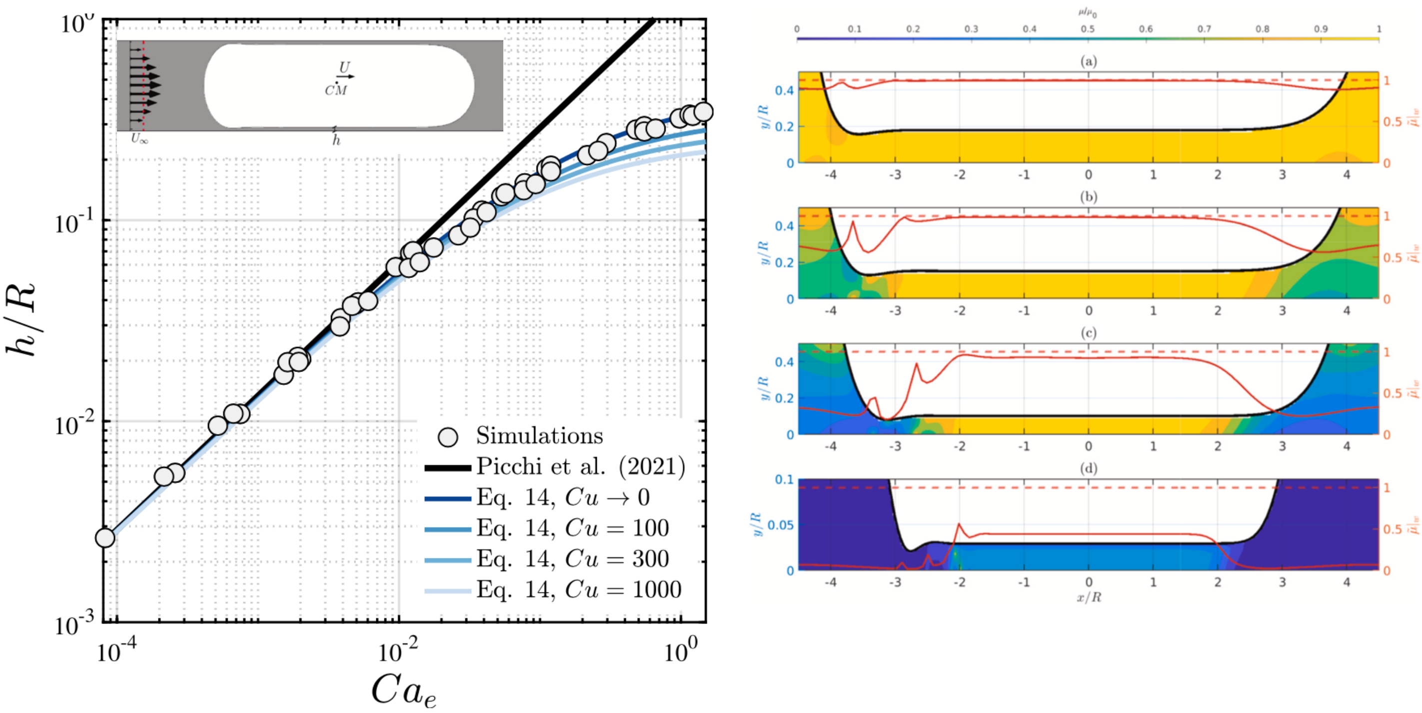


Figure 1. Left: Scaling of the film thickness as a function of the effective capillary number; Right: The viscosity field in the shear-thinning liquid surrounding the bubble.

References:

[1] A. Aquino, D. Picchi, P. Poesio, Dynamics of a Taylor bubble through a shear-thinning fluid up to finite capillary numbers, **Journal of Non-Newtonian Fluid Mechanics**, Volume 314, 2023. https://doi.org/10.1016/j.jnnfm.2023.105003

[2] Picchi D, Ullmann A, Brauner N, Poesio P. Motion of a confined bubble in a shear-thinning liquid. **Journal of Fluid Mechanics**. 2021;918:A7. doi:10.1017/jfm.2021.321

[3] C. Steinik, D. Picchi, G. Lavalle, and P. Poesio, **Phys. Rev. Fluids** 9, 023305 https://link.aps.org/doi/10.1103/PhysRevFluids.9.023305

54th European Two-Phase Flow Group Meeting
08-11 September 2024,
Dublin, Ireland
ABSTRACT

A novel one-dimensional homogeneous equilibrium model of two-phase ejector using CO_2 as refrigerant

Cong You, Benoit Michel, Rémi Revellin

CETHIL UMR5008, INSA Lyon 69621, Villeurbanne, France,
Contact E-mail : cong.you@insa-lyon.fr

The introduction of two-phase ejectors as expansion devices offers a promising solution to the efficiency challenges associated with CO_2 cycles in heat pump and refrigeration systems. Research has demonstrated that ejector technology significantly enhances system performance by reducing throttling losses and expansion irreversibility. However, ejectors are complex devices, involving the interaction of two distinct flows: a high-pressure motive flow and a low-pressure suction flow. The flow patterns within the ejector are further complicated by the presence of two-phase flow and transonic phenomena. To gain a more detailed understanding of these intricate flow patterns, simulations of the ejector are essential.

Given that CFD modeling is computationally expensive and 0-D modeling offers limited insight, 1-D modeling with the Finite Volume Method (FVM) provides a viable compromise between precision and computational efficiency. This work presents a novel 1-D homogeneous equilibrium model (HEM) based on FVM. The model is divided into various sections according to the geometric configuration, each employing distinct assumptions and modeling methods. The 1-D ejector model was experimentally validated using data from the literature, showing an absolute relative deviation within 10 % for outlet pressure and 15 % for motive mass flow rate.

Furthermore, the model was compared with existing 1-D homogeneous equilibrium models, revealing that the novel 1-D HEM offers improved efficiency and stability without sacrificing accuracy. This 1-D model proves to be a valuable tool in the design process, facilitating parametric analysis of ejector geometry. Five key parameters—the length and diameter of the constant-area mixing section, the length and angle of the ejector's diffuser, and wall roughness—were examined for their effects on ejector performance. The results indicate that the diameter of the constant-area mixing section and the diffuser angle have the most significant impact on performance.

`

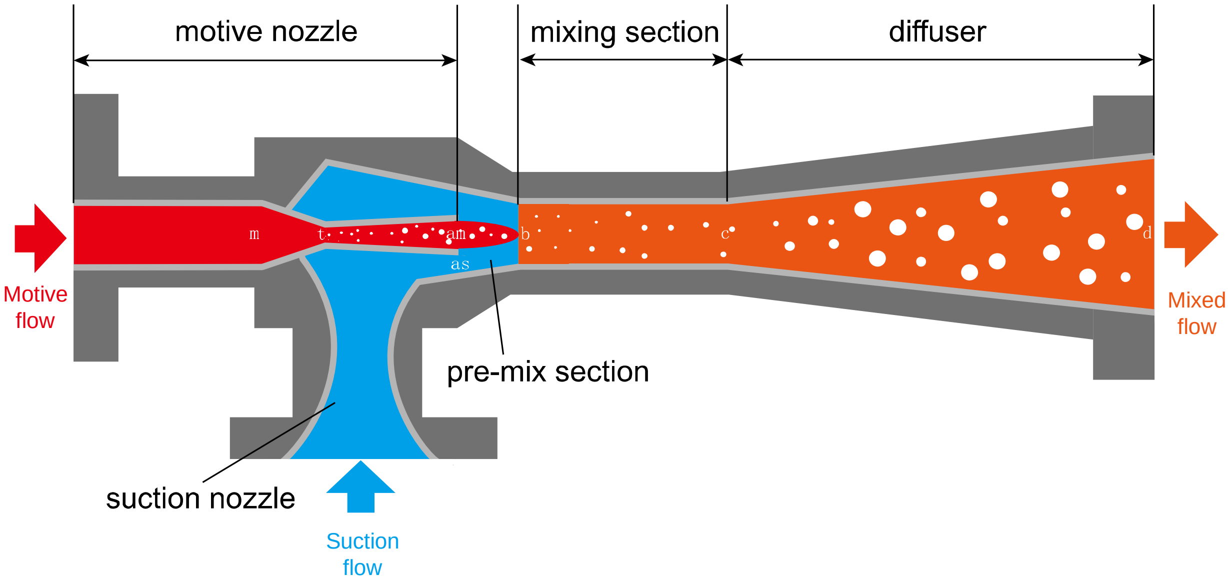


Figure 1. Schematic of a two-phase ejector

Trinity College Dublin
Coláiste na Tríonóide, Baile Átha Cliath
The University of Dublin
ETPFGM 2024

54th European Two-Phase Flow Group Meeting
08-11 September 2024,
Dublin, Ireland
ABSTRACT

Fizzics of Guinness

A.J. Robinson

Department of Mechanical, Manufacturing & Biomedical Engineering, Trinity College Dublin, Dublin 2, Ireland
Contact E-mail : arobins@tcd.ie

In carbonated beverages, the behavior of bubbles and the resulting two-phase flow are well-documented. The diffusion of dissolved carbon dioxide initiates nucleation at surface imperfections and cellulose fibers on the vessel's wall, leading to bubble growth and eventual detachment. These CO_2 bubbles, once large enough, ascend through the liquid, expanding further due to ongoing diffusion, and forming bubble trains as new bubbles continue to nucleate, grow, and detach [1]. However, Guinness draught defies this typical behavior, exhibiting puzzling and counterintuitive two-phase flow dynamics when poured. Notably, a swarm of bubbles can be seen descending along the glass's side—despite the upward force of buoyancy acting on each individual bubble [2]. Moreover, this downward flow is highly dynamic, with unsteady, ripple-like waves that create a unique textured motion distinct to Guinness among beers [3, 4]. This presentation will explore recent research on the two-phase flow phenomena observed in Guinness draught, specifically focusing on the mechanics behind the sinking bubble phenomenon and the associated ripple-wave flow texture.

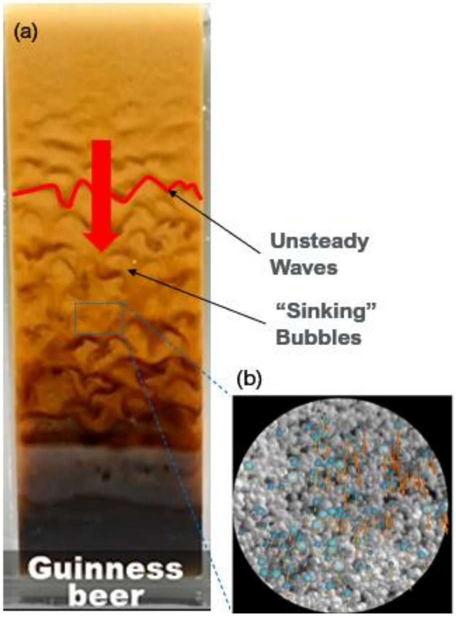


Figure 1. (a) Textured-flow [3], and (b) measured downward bubble motion [2] in Guinness draught.

[1] G. Liger-Belair, G. Polidori, and P. Jeandet, Recent advances in the science of champagne bubbles, Chem. Soc. Rev. 37, (2008), 2490.
[2] Benilov, S. Kaar, S.B.G. O'Brien, Why do bubbles in Guinness sink?, Am. J. Phys., 81(2), (2013).
[3] T. Watamura, F. Iwatsubo, K. Sugiyama, K. Yamamoto, Y. Yotsumoto, T. Shiono, Bubble cascade in Guinness beer is caused by gravity current instability, Scientific Reports, 9(1), (2019), 5718.
[4] T. Watamura, S. Sugiyama, Bubble cascade may form not only in stout beers, Physical Review E, 103, (2021), 063103.

Trinity College Dublin
Coláiste na Tríonóide, Baile Átha Cliath
The University of Dublin
ETPFGM 2024

54th European Two-Phase Flow Group Meeting
08-11 September 2024,
Dublin, Ireland
ABSTRACT

Influence of bubble formation history on rising velocity

M. Schlüter, F. Kexel[1],
A. Tomiyama, R. Koto[2]

[1]Institute of Multiphase Flows, Hamburg University of Technology,
Eißendorfer Strasse 38, 21073 Hamburg, Germany

[2]Department of Mechanical Engineering, Graduate School of Engineering, Kobe University
1-1 Rokkodai, Nada, Kobe, 657-8501, Japan

Contact E-mail: michael.schlueter@tuhh.de

The prediction of the rising velocity of a gas bubble in water is of significant importance for the design, and sustainable operation of gas-liquid reactors like bubble columns. Because of the importance a lot of research has been done on this topic for decades by many research groups (e.g. Brauer (1971), Clift et al. (1978), Tomiyama et al. (2002). Despite the large number of experiments that have been carried out, it is still difficult to reliably predict the rising velocity of bubbles, as some phenomena occur that are difficult to explain and are therefore not taken into account in today's models. Tomiyama et al. (2002) have already pointed out, that the initial deformation of bubbles during the formation process influence the rising behaviour and terminal velocity. Indeed, by using a hypodermic needle with a very precise bubble separation point, a high reproducibility of the bubble rising path is achievable (Sone et al., 2008). Later, Peters and Els (2012) reported, that bubbles are rising with different velocities in dependency of their residence time in the continuous phase before detaching from the used orfice. Krzan et a. (2023) showed, that bubbles are rising with different terminal velocities if the water is contaminated with different N-Alkanols. To sum up, the bubble rising velocity seems to be inherently influenced by interphase phenomena and bubble detachment mechanisms that might be difficult to predict and control in experiments of technical relevance.

In this presentation, latest results will be shown that have been achivied during a research stay of Akio Tomiyama and his PhD student Ryoya Koto from Kobe University, Japan at the Institute of Multiphase Flows at Hamburg University of Technology, Germany. The results, showing very strong deviation of bubble rising velocitites in water for obviously the same set-up by varying the bubble formation mechanism and residence time before lift-off (figure 1). In this presentation, the results will be shown and discussed in detail.

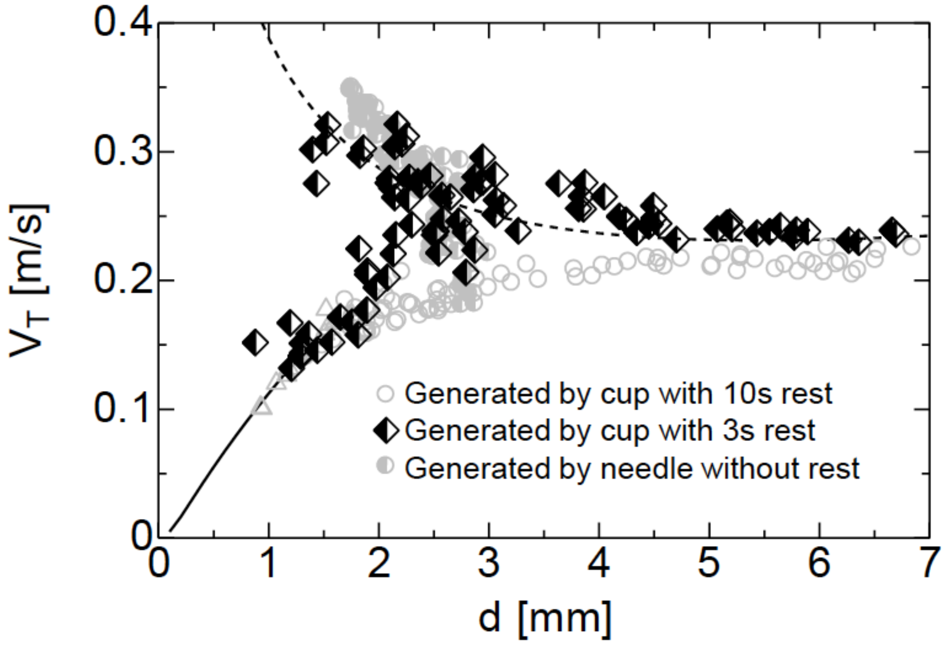


Figure 1. Different rising velocities achieved in the same liquid phase system (water) with varying orfice geometries and residence time before bubble release

Literature:

Brauer, H. Grundlagen der Einphasen-und Mehrphasenstroemungen; Sauerlaender AG: Aarau, Switzerland, 1971.

Clift, R., Grace, J.R. and Weber, M.E. (1978) Bubbles, Drops and Particles. Academic Press, New York. ISBN 0-12-176950-X.

Tomiyama, A.; Celata, G.P.; Hosokawa, S.; Yoshida, S.; Terminal velocity of single bubbles in surface tension force dominant regime. International Journal of Multiphase Flow, 2002, 28, 9, 1497-1519, https://doi.org/10.1016/S0301-9322(02)00032-0.

Sone, D.; Sakakibara, K.; Yamada, M.; Sanada, T.; Saito, T.; Bubble Motion and its Surrounding Liquid Motion through the Collision of a pair of Bubbles. Int. J. Energy Syst. 2008, 2 (1), 306–317. https://doi.org/10.1299/jpes.2.306

Peters, F.; Els, C.; An experimental study on slow and fast bubbles in tap water, Chemical Engineering Science, 2012, 82, 194-199, https://doi.org/10.1016/j.ces.2012.06.061.

Krzan, M.; Chattopadhyay, P.; Orvalho, S.; Zednikova, M. Effects of N-Alkanol Adsorption on Bubble Acceleration and Local Velocities in Solutions of the Homologous Series from Ethanol to N-Decanol. Materials 2023, 16, 2125. https://doi.org/10.3390/ma16052125.

AC electric field impact on bubble dynamics

Tianle Gu[1] and Samuel Siedel[1]

[1]EPM Group, Laboratoire SIMaP, CNRS, Grenoble INP, Université Grenoble Alpes, France
Contact E-mail: samuel.siedel@grenoble-inp.fr

The growth of bubbles in a dielectric liquid from an upward-facing flat surface in a quasi-homogeneous electric field has been the subject of numerous studies during the past 30 years. A DC electric field was usually employed, exhibiting the normal interfacial stress due to dielectrophoretic and electrostrictive forces. No significant electrophoretic effect is reported in such conditions, and consistent and repeatable bubble dynamics are observed.

In the current investigation, AC electric field of various wave functions and frequencies are imposed on air bubbles injected in HFE 7100. New behaviours of bubble dynamics are observed, such as the oscillation of bubbles when the AC frequencies are of the same order of magnitude as the natural oscillation frequencies of the bubbles (figure 1). Also, when a wave function alternating a positive and a negative polarity is used, some actuation frequencies tend to separate the growth dynamics of successive bubbles into two repeatable patterns (figure 2), with a change of pattern almost between all successive bubbles. These two patterns seem to be linked to a bubble initiation triggered by the electric field at two specific instants of the wave function.

As the growth pattern depends on the polarity of the electric field during the early stages of bubble growth, and as the polarization forces are only function of the square of the electric field, these observations clearly indicate an impact of the electrophoretic forces. Another striking effect of polarity can be clearly observed when imposing a symmetric square wave of -12 to 12 kV at a frequency of 10 Hz. A totally different behaviour is observed during the 50 ms of negative polarity and the 50 ms of positive polarity. A polarity exhibits usual bubble cycles of elongated bubbles, and the other shows a strong attraction of the bubble towards the bottom electrode that sits there and grows without detaching. Figure 3 illustrates a switch of polarity around frame 3. A bubble that had just detached is suddenly so attracted to the bottom electrode that it moves downwards until sitting again onto the surface.

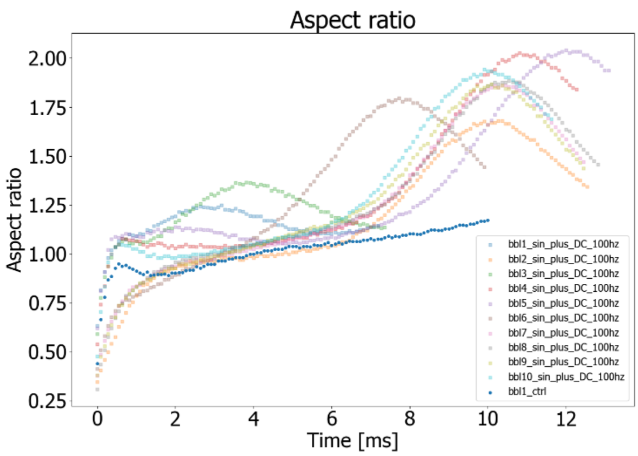


Figure 1: Height/width aspect ratio evolution of 10 successive bubbles under an asymmetric sine wave of 0 to 24 kV at 100 Hz.

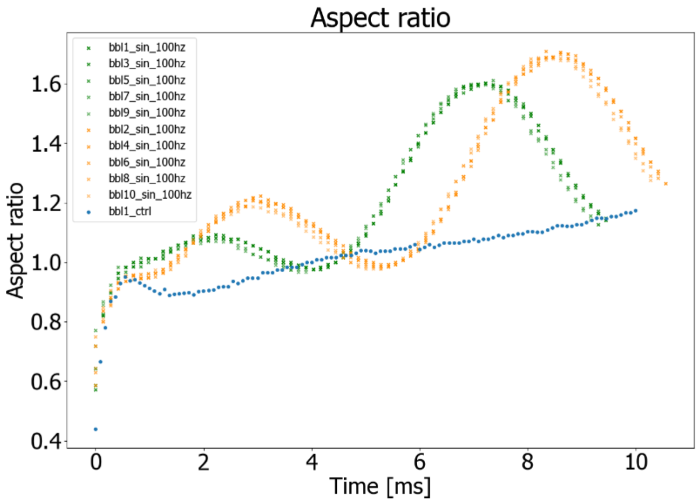


Figure 2: Height/width aspect ratio evolution of 10 successive bubbles under a symmetric sine wave of -12 to 12 kV at 100 Hz.

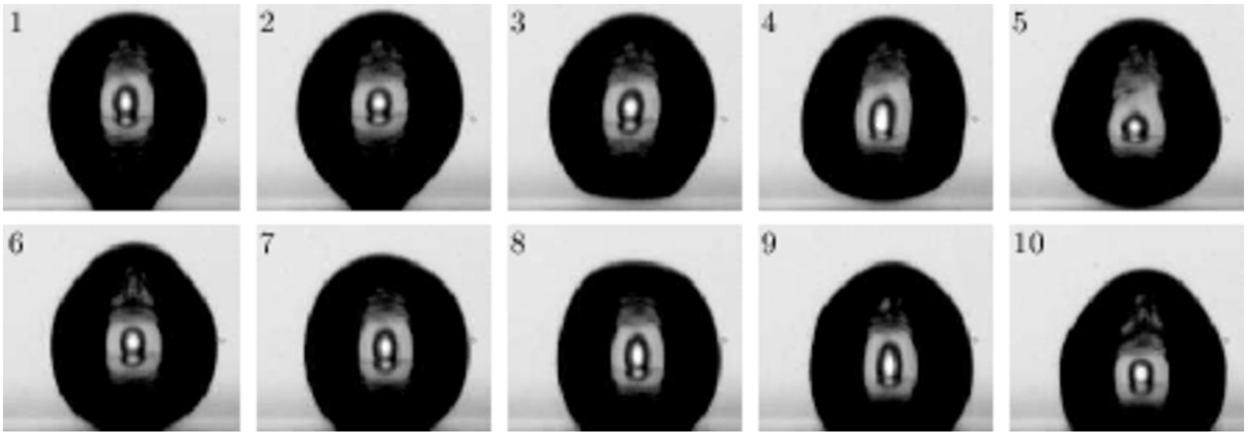


Figure 3: Successive images of a bubble detaching and returning to the surface under a square wave of -12 to 12 kV at 10 Hz. Time between frames is 0.55 ms.

Two phase flow regimes at low Reynolds number

Case of bubbly flows

L. Boubendir[1], S. Zeguai[2] , S. Chikh[1] and L. Tadrist[3]

[1] Yahia Farès University, Faculty of Technology, LMP2M Laboratory, Ain d'heb, Médéa 26000, Algeria
[2] USTHB, Faculty of Mechanical and Process Engineering, LTPMP, Alger 16111, Algeria
[3] Aix-Marseille Université, CNRS, Laboratoire IUSTI, UMR 7343, Marseille 13453, France
lounes.tadrist@univ-amu.fr

So far and despite the numerous investigations focusing on two-phase flow in mini/micro channels, considerable discrepancies among the data still exist mainly due to the difficulty in setting up the experiments and/or conducting the measurements. General flow pattern characteristics are largely decided in the inlet region and the complex mechanism of various flow behaviors remains far from being fully understood. Thus, further investigations on the bubble formation and detachment processes particularly at the injection nozzle are still needed.

Many analytical models based on the different forces acting on the bubble at the moment of detachment have been presented; however, no universal model has been retained yet. Different simplification done, reduced the accuracy of the predictions, such the assumption of spherical bubble shape during the evolution. On the other hand, the determination of the capillary force acting on the contact line solid-liquid-gas maintaining the attached bubble requires the determination of the dynamic contact angles, which is very delicate. Also, in the modeling, the neck formed between the bubble and the nozzle is faintly taken into account.

The aim of the present study is to investigate the distribution of bubbles and the influence of operating condition on bubble growth and detachment from bubble generation frequency at the tip of injection nozzle in a co-flowing configuration without heat transfer. Very low superficial gas and liquid velocities, respectively ranging from 0.042 to 0.28 m/s and from 0.0079 to 0.059 m/s are explored in order to avoid perturbation of the liquid-gas interfaces in focusing flow configurations. Horizontal and vertical tube orientations are considered to compare and thus understand the mechanism of bubbles' generation in small tube (figure 1).

To better understand the role of the surface tension in bubble growth and detachment in a micro-tube, a numerical investigation for a horizontal axisymmetric flow with the assumption of zero gravity and an upward flow accounting for gravity contribution is investigated. The continuous liquid phase is flowing in a tube of 500 μm inner diameter and the gas phase is axially injected through a nozzle of 110 μm inner diameter and 210 μm outer diameter. A single-fluid model is used to determine the flow field, solving the continuity and momentum equations associated with the volume of fluid method for interface tracking. An open source software, Open FOAM, is utilized for solving numerically this problem. The prediction results show that the

surface tension plays a double role. it keeps the bubble attached to the injection nozzle during bubble growth and neck formation. Then, it destabilizes the interface by pinching off the neck in the immediate vicinity of the nozzle at about a distance of 0.5 the nozzle diameter rather than right at the nozzle exit (figure 2).

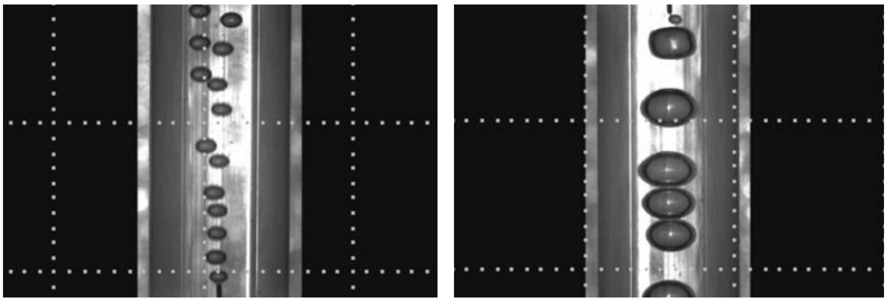


Figure 1. Visualisation the two-phase flow patterns in a tube diameter (D_{in}=3mm) at the nozzle exit (d_n=110 µm)

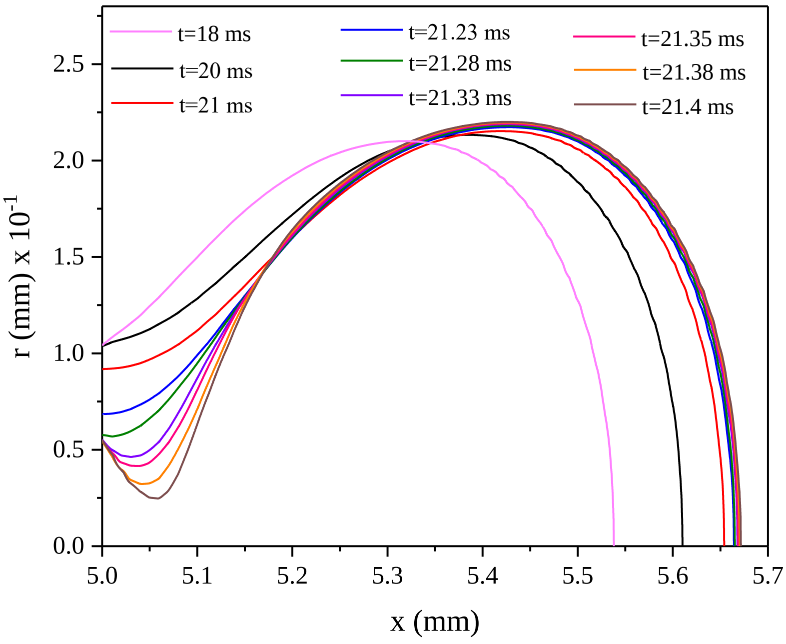


Figure 2. Bubble shape evolution and contact line motion : Contact line radius decreases from the external till to the inlet nozzle diameter and neck formation with its evolution over time. (U_{GS}=0.014 m/s, U_{LS}=0.076 m/s).

54th European Two-Phase Flow Group Meeting
08-11 September 2024,
Dublin, Ireland
ABSTRACT

Open Questions on Two-Phase Flow and Boiling in Microchannels

J.R. Thome[1]

[1]JJ Cooling Innovation Sàrl, Lausanne, Switzerland
Contact E-mail : john.thome@jjcooling.com

The last few decades have seen very extensive research and growing industrial interest on microchannel two-phase flows and heat transfer as well as electronics cooling applications. Without going into detail, there are a huge number of prediction methods for many of the pertinent aspects: boiling heat transfer coefficients, critical heat flux, pressure drops, void fractions, flow pattern effects, etc. Still there are quite a few unresolved issues and items, such as:

- What is the best, most reliable method to use for the above mentioned aspects of thermal design?
- How does one insure flow stability and good flow distribution?
- Is pumped two-phase cooling or passive two-phase cooling most appropriate?
- How does one handle two-phase flow distribution in micro-evaporators configured in series?
- What are the channel shape effects on heat transfer and pressure drop?
- What surface effects are important and which are not (roughness, wettability, etc.)?
- What are the new fluids we need to work with to address the F-fluid issue?
- Do we really need nano-structured surfaces or is this another last chance for nano-fluids?

The present talk does not intent to provide answers to these questions (and others) but to hopefully be a starting point for discussion and thus what might be topics for near future research.

Trinity College Dublin
Coláiste na Tríonóide, Baile Átha Cliath
The University of Dublin
ETPFGM 2024

54th European Two-Phase Flow Group Meeting
08-11 September 2024,
Dublin, Ireland
ABSTRACT

On Dynamics of Single Bubble and Bubbly Flow in Bubble Column

A. Tomiyama

Emeritus Professor, Kobe University
Contact E-mail : akiotomiyama@gmail.com

Some new findings on the terminal rising velocities of single bubbles and breakup of a planar bubble will be introduced. Effects of the bubble column geometry, i.e. the height and diameter of the column, on the total gas holdup will be also discussed based on void distribution in the column.

As demonstrated in Tomiyama, et al. [IJMF, Vol.28, No.9 (2002)], the terminal rising velocity V_T of a single spheroidal bubble at a high bubble Reynolds number Re is determined by a potential flow formed in the vicinity of the bubble nose. More strictly speaking, V_T of a spheroidal bubble at high Re is given by the potential flow about the ellipsoid circumscribed at the bubble nose. If the aspect ratio and the maximum horizontal axis of the ellipsoid circumscribed at the bubble nose are E_f and a, V_T based on the potential flow is given by

$$2D\ Planar\ Bubble: V_T = \sqrt{\frac{3\sigma E_f}{\rho_L a}\frac{1-E_f}{1+E_f} + \frac{(\rho_L-\rho_G)ga}{\rho_L}\frac{E_f}{(1+E_f)^2}} \quad (E_f > 0)$$

$$3D\ Oblate\ Bubble: V_T = F(E_f)\sqrt{\frac{4\sigma E_f}{\rho_L a} + \frac{(\rho_L-\rho_G)ga}{\rho_L}\frac{E_f}{1-E_f^2}} \quad (0 < E_f < 1)$$

$$3D\ Prolate\ Bubble: V_T = G(E_f)\sqrt{-\frac{4\sigma E_f}{\rho_L a} + \frac{(\rho_L-\rho_G)ga}{\rho_L}\frac{E_f}{E_f^2-1}} \quad (E_f > 1)$$

where

$$F(E_f) = \frac{\sin^{-1}\sqrt{1-E_f^2} - E_f\sqrt{1-E_f^2}}{1-E_f^2}, \qquad G(E_f) = \frac{E_f\sqrt{E_f^2-1} - \tanh^{-1}\frac{\sqrt{E_f^2-1}}{E_f}}{E_f^2-1}$$

Here σ is the surface tension, ρ_G and ρ_L are the gas and liquid densities and g is the gravitational acceleration. The bubble shape itself would be governed by many factors such as fluid properties, sorts and amount of surfactant adsorbed in the stagnant-cap region, and the surface internal energy loaded during bubble generation. In particular, V_T of a high Re bubble is apt to be widely scattered and unpredictable due to difference in the nose shape (E, a) caused by an uncontrollable difference in the initial surface internal energy loaded at bubble generation. The bubble shape at high Re is thus a sort of a stochastic variable. Hence, it would be important from an engineering point of view to acquire the knowledge on the mean bubble shape and the width of its distribution. As an example, shapes of air bubbles ranging from 2 to 7 mm in sphere-volume equivalent diameter d in clean distilled water at atmospheric pressure and room temperature were measured and the following tentative correlations of the mean, maximum and minimum aspect ratios $E_m, E_{\max}, E_{min}$ of spheroidal bubbles were obtained:

$$E_m = \max\left[\frac{1}{1+1.97Eo^{1.3}}, \frac{1}{1+0.357Eo^{0.389}}\right], \quad Eo = \frac{(\rho_L - \rho_G)gd^2}{\sigma}$$
$$E_{max} = \frac{1}{1+0.081Eo^{0.960}}, \quad E_{min} = \max\left[\frac{1}{1+1.97Eo^{1.3}}, \frac{1}{1+0.749Eo^{0.250}}\right]$$

It will be demonstrated that the mean, maximum and minimum values of V_T are reasonably evaluated by substituting the above shape correlations into V_T models expressed in terms of E.

Most of bubble breakup models proposed so far has dealt with disintegration caused by the impact of turbulent eddy on the bubble surface. However, quasi-two-dimensional planar bubbles can break up even in a stagnant liquid. Hence the interface motion and breakup of planar air bubbles in stagnant liquids were measured using a high-speed video camera. The observation confirmed two types of triggers in bubble breakup, i.e. the dynamic pressure and the capillary wave. In both cases, the wavelength of the interfacial wave just before the breakup corresponded to the unstable wavelength of the Rayleigh-Taylor instability. Experimental data on breakup obtained for several Morton number systems indicated that the breakup due to the dynamic pressure occurs when the Weber number We is larger than about 10, whereas the one caused by the capillary wave takes place when the Eötvös number Eo is larger than about 20. The breakup probability decreases with increasing the Morton number M and becomes 0 for $\log M > -6$.

Although many empirical correlations for predicting the total gas holdup of heterogeneous bubbly flow in a bubble column have been proposed, they unfortunately do not agree well with each other. One of the reasons for this disagreement is the difference in the column geometry from which the correlations were devloped. The effect of the column diameter and height on the total gas holdup should therefore be made clear to develop a gas holdup correlation applicable to various bubble columns and to facilitate the column scaleup design. Total gas holdups and void distributions in air-water bubble columns were measured for various column diameters and various initial water heights in the column. As far as the air-water cylindrical bubble columns are concerned, the experimental data showed that the column diameter D does not have any effect when $D \geq 200\ mm$, and as shown in Fig. 1, the total gas holdup $\bar{\alpha}_G$ at a constant gas volume flux J_G decreases as the initial liquid height H_0 increases and becomes independent of H_0 when H_0 exceeds a certain critical value (ca. 2.0 m) depending on J_G. This tendency will be clearly explained by the presence of three distinct regions in the vertical direction, each of which has its intrinsic void fraction which does not depend on H_0 and D.

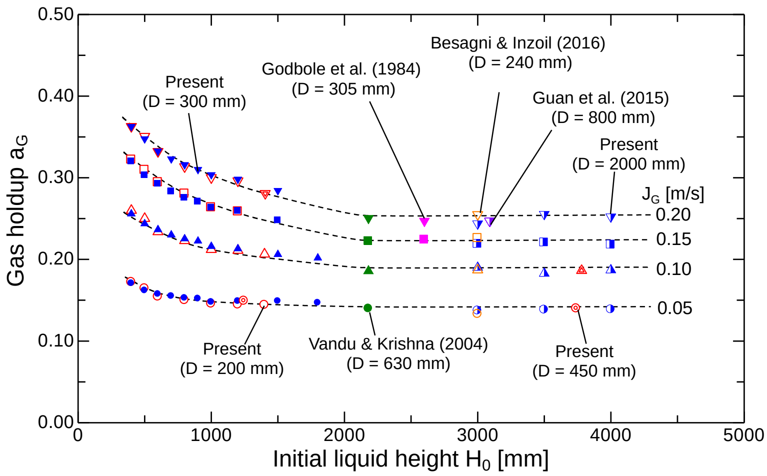


Figure 1. Effects of H_0 and D on $\bar{\alpha}_G$

*The author acknowledges the very kind support of Prof. Anthony Robinson.

Bubble dynamics during pool boiling on tube bundles under low pressure

B. Zajączkowski, T. Hałon, W. Lada, D. Kaczmarek

Department of Thermal Sciences, Faculty of Mechanical and Power Engineering., Wroclaw Uni. of Science and Technology, Wyb. Wyspiańskiego 27, 50-370 Wroclaw, Poland,
Contact E-mail: bartosz.zajaczkowski@pwr.edu.pl

Modern cooling needs, such as food preservation and thermal comfort, contribute significantly to greenhouse gas emissions. To maintain living standards, the development of efficient heat transfer technologies with environmentally safe substances such as water is crucial. Sub-atmospheric sorption chillers offer a promising solution. Our current research investigates bubble formation in low-pressure shell-and-tube exchangers. We hypothesize that in bundled configurations bubbles on lower tubes may slide along upper tube surfaces, activating cavities and promoting evaporation of superheated thin films at their bases. However, large bubbles at low pressures may also reduce heat transfer efficiency by preventing subcooled liquid from reaching neighbouring tube surfaces. The effects of these configurations under low pressure conditions have not yet been observed or fully understood.

The Department of Thermal Sciences at the Faculty of Mechanical and Power Engineering, Wroclaw University of Science and Technology, in collaboration with the CETHIL laboratory at INSA, Lyon, has extensively studied various aspects of the fundamental phenomena of water vaporization at low pressure. This collaboration has produced a series of papers that reveal the unique characteristics of this low-pressure process. The present study builds on these findings, expanding the investigation into the intricate structures of bundled tubes. The pioneering aspect of this work lies in its emphasis on the bubbles themselves, specifically, their shapes, interactions, and overall behavior under sub-atmospheric conditions. Preliminary tests have substantiated this conceptual framework, demonstrating that bubbles grow to substantial sizes and form complex configurations. Figure 1 illustrates bubble nucleation on the heated top row tube, with the liquid level LH = 30 cm.

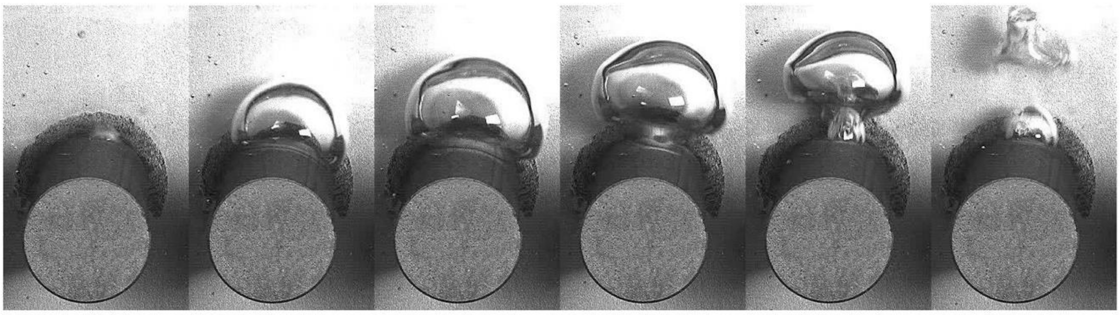


Figure 1. A single bubble nucleating at the top of the tube (25.4 mm OD) and then develops into a mushroom shape, characteristic of low-pressure conditions; p = 7.15 kPa, LH = 30 cm.

The test setup acknowledges that the performance of the tube bundle configuration is influenced by factors such as tube spacing, tube diameter, tube arrangement (staggered or in-line), heat flux, and the number of tube rows. It was designed to observe this configuration commonly found in practical applications and provides the capability to adjust these aspects as needed. Tubes are standardized inch size (25.4 mm OD), often used in cooling applications. Heat fluxes are regulated using independent cartridge heaters, and each tube is equipped with four Pt1000 temperature sensors, enabling independent observation of temperature and heat flux fluctuations for each tube in the set.

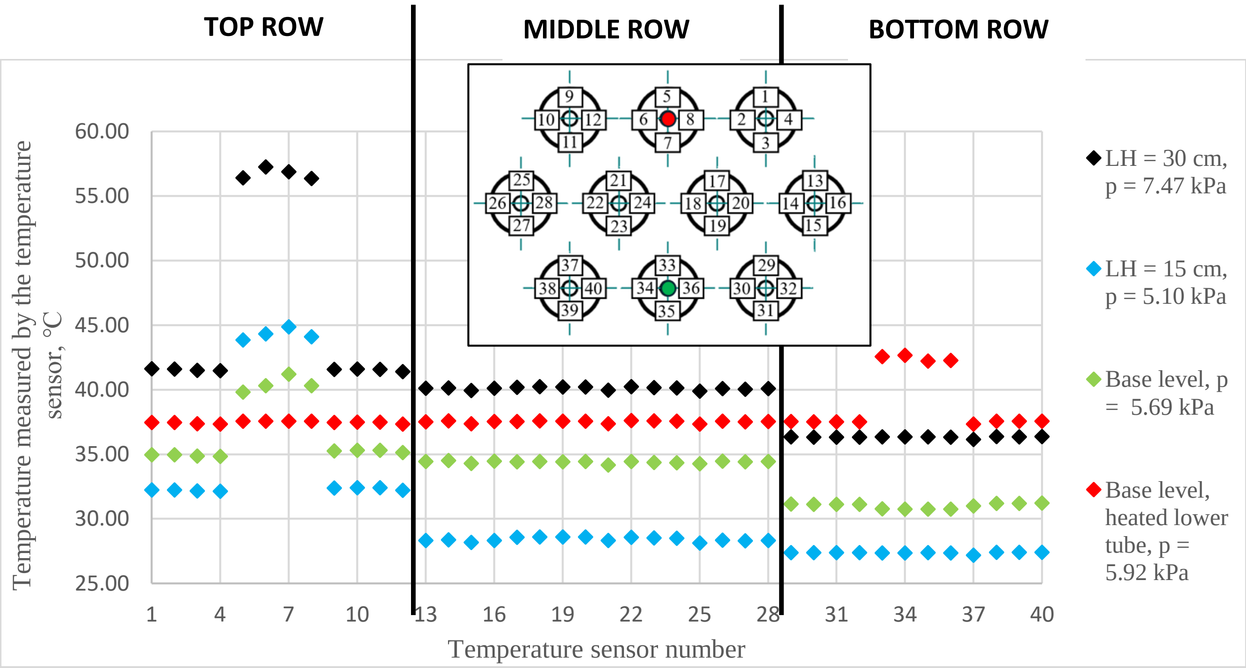


Figure 2. Four datasets, each encompassing readings from 40 temperature sensors. In the black, blue, and green datasets, the top-middle tube is heated, while in the red dataset, the bottom-middle tube is heated.

Our primary objective is to experimentally confirm how the growth of large bubbles within confined bundled spaces influences the overall operation of the heat exchanger. Figure 2 illustrates a sample dataset demonstrating how the setup will visualize fluctuations in temperature (and consequently heat flux and heat transfer coefficient). Coupled with high-speed camera recordings, this will allow for insight into the complex interactions between bubbles and tubes during boiling.

Initial analysis has revealed temperature gradients due to stratification: heating only the top tube results in lower temperatures in the bulk of the water below, whereas heating the bottom tube evens out temperatures throughout the vessel, thereby altering boiling conditions for the adjacent tubes. Additionally, at pressures down to 5 kPa, bubble sizes are too small to significantly affect neighbouring tubes. Therefore, further research is necessary to explore even lower pressure levels and establish the relationships between geometry and thermodynamic conditions to determine the optimal boiling conditions for these configurations.