Kutateladze Institute of Thermophysics

HTI HeatTransferInstitute

ENEA

Microgravity Research Centre ULB

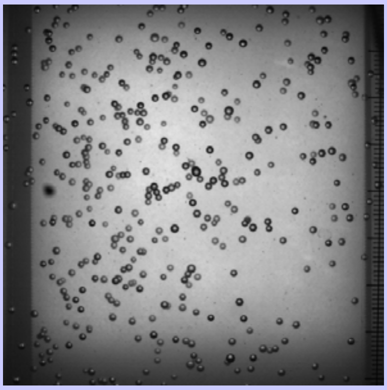
Universita' degli Studidi Napoli Federico II

esa

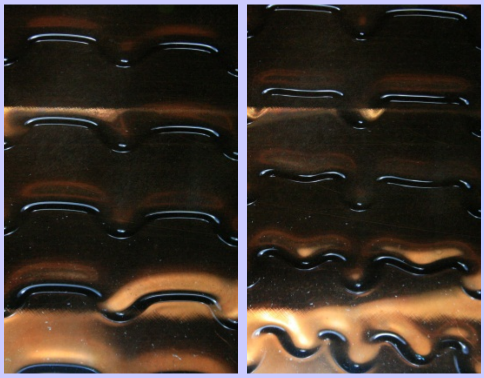
Sixth International Conference on

TWO-PHASE SYSTEMS FOR GROUND AND SPACE APPLICATIONS

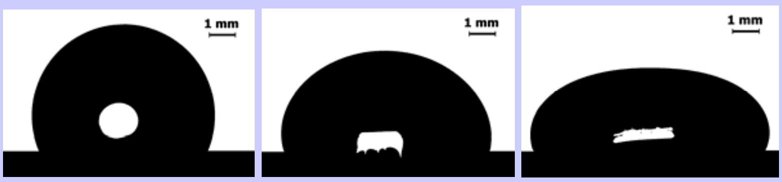
BOOK OF ABSTRACTS



Snapshot of a monodisperse bubble suspension of prescribed bubble size in microgravity conditions, Bitlloch et al.



Photographs of the liquid film's free surface showing the influence of the electric field, Rohlfs et al.



Gravity effect on spreading of 0.146 ml water drop, Zaitsev et al.

Editors: Oleg Kabov
Jean-Claude Legros
Gian Piero Celata
Raffaele Savino
Yulia Kabova

CAVA DE' TIRRENI (NAPOLI), ITALY
September 25-28, 2011

BOOK OF ABSTRACTS

Sixth International Conference on

TWO-PHASE SYSTEMS FOR GROUND AND SPACE APPLICATIONS

Conference Chairs

Oleg A. Kabov
Prof. and Co-Director of Heat Transfer International Research Institute of ULB and IT SB RAS, Belgium

Jean Claude Legros
Prof. Microgravity Research Center, Université Libre de Bruxelles, Belgium

Gian Piero Celata
Prof. Division of Advanced Technologies for Energy and Industry of ENEA, Rome, Italy

Raffaele Savino
Prof. Università degli Studi di Napoli ‘‘Federico II”, Dipartimento di Ingegneria Aerospaziale, Italy

Conference organizers

- Division of Advanced Technologies for Energy and Industry of ENEA, Rome, Italy
- Università degli Studi di Napoli ‘‘Federico II”, Dipartimento di Ingegneria Aerospaziale, Italy
- Heat Transfer Institute of ULB and IT SB RAS, Brussels
- Microgravity Research Center, Université Libre de Bruxelles, Brussels
- Institute of Thermophysics, SB RAS, Novosibirsk

Conference Scientific Secretary

Dr. Yulia Kabova
Technische Universität Darmstadt, Center of Smart Interfaces, Darmstadt, Germany
Heat Transfer Institute of ULB and IT SB RAS, Russia
e-mail: hti@ulb.ac.be

Scientific Institute and Astronomic Observatory A. Genoino,
CAVA DE’ TIRRENI (NAPOLI), ITALY
September 25-28, 2011

http://hti.ulb.ac.be/workshop-11/index.html

In memory of the 50th anniversary of the first space flight of the Russian cosmonaut Yuriy Gagarin on 12th April, 1961

Conference objective

The conference is intended to provide a platform for researchers to exchange information and identify research needs in this important area encompassing several engineering, mechanical and physical disciplines.

Topics for the presentations include:

- Experiments in Microgravity
- Boiling, Evaporation and Condensation
- Films, Layers and Interfaces
- Physics of Contact Line and Wetting
- Microchannels and Minichannels
- Thermocapillary flows
- Spray, Jets and Two-Phase Flows
- Bubbles, Drops and Foams
- Electronics Cooling
- Properties of Vapour - Liquid - Solid interfaces

Scientific Committee

- Dr. Yoshiyuki Abe, AIST, Japan
- Prof. Sergey V. Alekseenko, IT, Russia
- Prof. Avram Bar-Cohen, UM, USA
- Prof. Alberto Cavallini, UP, Italy
- Prof. Gian Piero Celata, ENEA, Italy
- Prof. Catherine Colin, IMFT, France
- Prof. Michael Dreyer, ZARM, Germany
- Prof. Alexander M. Frank, ICM, Russia
- Prof. Qiu-Sheng Liu, IM, China
- Prof. Dmitriy Markovich, IT, Russia
- Dr. Olivier Minster, ESA, The Netherlands
- Prof. Haruhiko Ohta, KU, Japan
- Prof. Vladislav V. Pukhnachev, LIH, Russia
- Prof. Peter Stephan, TUD, Germany
- Prof. Lounes Tadrist, IUSTI, France
- Prof. John R. Thome, EPFL, Switzerland
- Prof. Charles A. Ward, UT, Canada
- Prof. Jian-Fu Zhao, IM, China

Organizing committee

Dr. Carlo Saverio Iorio
Universite Libre de Bruxelles, Microgravity Research Center, Belgique
Dr. Elena F. Bykovskaya
Universite Libre de Bruxelles, Microgravity Research Center, Belgique
Dr. Loredana Marino
SOCIAL AND CULTURAL AGORA' ASSOCIATION
Dr. Brigida Stanziano
Università degli Studi di Napoli "Federico II", Dipartimento di Ingegneria Aerospaziale, Italy

Conference is being organized under the auspices of the European Space Agency.

INTRODUCTION

On behalf of the Organizing Committee of the Sixth International Conference on TWO-PHASE SYSTEMS FOR GROUND AND SPACE APPLICATIONS, taking place in the Scientific Institute A. Genoino, in Cava de Tirreni, Italy, on 25-28 September 2011 we would like to welcome you.

This scientific meeting is an initiative of the International Heat Transfer Institute (HTI-founded in 2005). It is a framework providing support to scientists to conduct research works. The collaboration between ULB (Université libre de Bruxelles) and IT (Institute of Thermophysics, Siberian Branch of Russian Academy of Sciences in Novosibirsk) has lasted more than 10 years and is at the origin of the HTI creation. HTI is dedicated to enabling long-term and sustainable cooperation between Russian, Belgian scientists and scientists from other countries, aiming to achieve significant results both in fundamental and applied science. The cooperation is based on joint exploitation of research equipment, joint research and training, participation in national and international collaborations, and on organization of workshops and conferences, such as the present one. The initiative of HTI to found this specific forum has been supported by the European Space Agency.

The first and third Workshop took place in Brussels, Belgium, in September 2006 and 2008; correspondingly, the second and fifth meetings were organized in Kyoto, Japan, in October 2007 and September 2010; the fourth conference took place in Novosibirsk, Russia in September 2009. The present sixth conference, organized in Italy, in collaboration with the Department of Aerospace Engineering of the University of Naples, shows an increased participation and a wider international representation. This demonstrates the developing importance of the addressed topics and the cutting edge level of the presented results. These Conferences provide platform for researchers to exchange information and identify research needs in this important area encompassing several engineering, mechanical and physical disciplines.

All facets of the heat transfer problem in two phase systems are addressed in a theoretical, numerical and experimental manner. Topics for the presentations include: Experiments in Microgravity, Boiling, Evaporation and Condensation, Films, Layers and Interfaces, Physics of Contact Line and Wetting, Microchannels and Minichannels, Thermocapillary flows, Spray, Jets and Two-Phase Flows, Bubbles, Foams and Drops, Electronics Cooling, Properties of Vapour - Liquid - Solid interfaces. Furthermore, numerous interactions between the different groups take place in-between Conferences, thereby accelerating the progress in the overall understanding required for the optimization of integrated systems. These International conferences are thus a valuable forum speeding up the exchange of information within the community.

The scientific program this year looks very promising; we have 13 keynote lectures, 27 oral presentations and 51 short oral presentations with posters during three days. Short oral presentations have been given to high level scientists with the debatable and interesting subjects. Therefore the organizers ask you to pay a special attention to the poster sessions. Three minutes short oral presentations give a possibility to introduce a team and to show some most important results. The immediately following poster session provides some additional time for detailed discussions, avoiding parallel sessions.

In a special issue of Microgravity Science and Technology journal, features articles reporting on research works presented at the conference, are published. The publication and diffusion of the scientific work are assured by Springer whose support is gratefully acknowledged by the Organizers. We would like to thank the Guest Editors and referees that helped to improve the quality of the papers to meet the high standards of Microgravity Science and Technology journal.

From the previous Conference in Kyoto an International Topical Team meeting on Two-phase Heat Transfer takes place as a one day follow-up event of the Conference. Organized by space agencies (mainly ESA and JAXA) ITT meeting this year will take place on September 29. Some of the main objectives of the meeting are: 1) To exchange detailed information about on-going space projects on two-phase heat transfer worldwide avoiding duplication of investments for future space programmes; 2) To emphasize the time scale and the broad scope of research on two-phase heat transfer and the necessity of coordination and eventually cooperation between the programmes managed by different space agencies worldwide; 3) To discuss optimisation of space experiments and propose opportunities for organising international teams to enhance the development in this field of research; 4) To pave the way for collaborative international experiments, missions, or space programmes.

We would like to thank the main institutions involved in the scientific and programmatic organization of this Conference: the Heat Transfer Institute, the Institute of Thermophysics of Russian Academy of Scienses, the Microgravity Research Centre of the Universite' Libre de Bruxelles, the Department of Aerospace Engineering of the University of Naples, with its long tradition in microgravity research, going back to the first Spacelab mission SL-1 in 1981, the Division for Energy and Industry of ENEA, the Administration of the City of Cava de' Tirreni and the Provincia di Salerno.

We are indebted to the European Space Agency and to the Russian Foundation for Basic Research for their financial support.

Finally we would particularly like to express our gratitude to all the people of the Scientific Committee for the contribution to the organization of the scientific programme, to the conference secretary Dr. Yulia Kabova and to the staff of the local organizing committee, Dr. Carlo Iorio, Dr. Loredana Marino, Dr Brigida Stanziano and Dr. Elena Bykovskaya whose supports and efforts were essential for the smooth preparation of the conference.

We hope that this conference will meet your expectations, facilitate and encourage communication of new ideas and approaches between scientists working on two phase systems.

Best wishes to all attendees for a successful conference.

On behalf of the Conference Scientific and Organizing Committees

Profs. Oleg Kabov, Jean Claude Legros, Raffaele Savino and Gian Piero Celata

For more detailed information about the conference and HTI please visit to the website http://hti.ulb.ac.be/.

FOREWORD

International Topical Teams co-sponsored by several space agencies play a prominent role in the incubation and definition of research projects utilising space, and the International Space Station in particular. They are fora where current questions prevailing in different research fields are discussed and strategies involving experimentation in space are developed to arrive at clear unambiguous answers.

Two-phase systems involving typically evaporation or boiling and condensation processes are growingly important in heat transfer but are characterised by a significantly higher level of complexity. This complexity makes their modelling and the development of predictive tools to help optimising their design one of the most difficult challenges for the scientific community that studies them and for the industry that develops and implements them.

Researchers from Europe, Canada and Japan as well as from Russia, the US and China consider the space environment to be not only one where heat transfer systems must perform reliably in spacecraft under rather unusual conditions, but also one where boundary conditions are seemingly simpler and provide for opportunities to learn about the fundamentals that drive these systems. The breadth of research that needs be done to arrive at the predictive tools that are required by system developers calls for formulating the different problems and tackling them in a coordinated fashion.

This is the purpose of a dedicated International Topical Team.
NASA, JAXA, ESA also CNSA in China are supporting this particular team and provide means for the implementation of experiments under strongly reduced gravity using drop-towers, aircraft flying parabolic trajectories, sounding rockets or unmanned orbiting capsules and also the International Space Station. Some of the first results are presented in this book of abstracts of the Sixth International Topical Team Workshop on Two Phase Systems for Ground and Space Applications; further advanced instruments currently under development will be operated in space in the coming years and deliver a wealth of new data that will help reach the objective of departing from classical empirical methods for designing the next generation of knowledge-based, energy efficient systems.

The joined efforts of many will indisputably lead to major advancements in this field.

ESA Noordwijk,
August 16th, 2011

Dr. Olivier Minster
Physical Sciences Unit Head
Directorate of Human Spaceflight and Operations
European Space Agency

CONTENTS

TWO-PHASE SYSTEMS

LIQUID FILMS

TWO-PHASE FLOWS

EXPERIMENTS IN MICROGRAVITY

TWO-PHASE SYSTEMS FOR GROUND AND SPACE APPLICATIONS-III

Subjects to be Clarified in ISS Experiment of Boiling Two-phase Flow

Haruhiko Ohta[1], Hitoshi Asano[2], Osamu Kawanami[3], Koichi Suzuki[4], Haruo Kawasaki[5] and Kiyosumi Fujii[5]

[1] Dept. Aeronautics and Astronautics, Kyushu University
744 Motooka, Nishi-ku, Fukuoka 819-0395, Japan
ohta@aero.kyushu-u.ac.jp
[2] Kobe University, [3] University of Hyogo, [4] Tokyo University of Science, [5] JAXA, Japan

Introduction

ISS experiment on flow boiling is prepared as an official project of JAXA. The design and operating conditions of experimental setup are checked in detail based on the ground test. Two different heated tubes of a transparent glass tube and a copper tube are located in parallel and connected to a test loop. On the inner wall of the transparent heated tube, a thin gold film is coated uniformly which is operated as a heater by the application of the electric current directly through it and simultaneously used as a resistance thermometer for the evaluation of averaged inner wall temperature. The structure gives the direct relation between the measured heat transfer data and the observed liquid-vapor behaviors (Ohta, 2003). Also a copper tube with conventional specifications is introduced for the measurement of local heat transfer coefficients distributed along the tube axis and of CHF values. At the individual downstream of these heated tubes, non-heated test sections are connected for the detailed observation and measurement of interfacial behaviors by a high-speed video camera.

One of the important features of the experimental setup is that the test loop can realize the extremely low mass velocity less than 50kg/m^{2}s. The inner diameter of both tubes is selected as 4mm, and the tubes are regarded as normal ones without indicating the characteristics of mini or micro tube for the test liquid FC72 circulated at near atmospheric pressure. Experiments in a wide range of vapor quality and heat flux become possible by using such thin tubes under the limited power supply. Additionally to the acquisition of heat transfer data, the clarification of dominant force map is an alternative important subject of the present research.

Heat transfer coefficient

Effect of gravity on boiling heat transfer is quite ambiguous despite of a lot of existed experiments on pool boiling. We only know the change of bubble size and shape in microgravity, and the expected increase of Marangoni effect. These trends might be true also in flow boiling at low subcooled or low quality under extremely low mass velocity conditions. Simulation of interfacial behaviors including those of thin liquid film underneath bubbles become a key point to know exactly the gravity effect, where both trends of heat transfer enhancement and deterioration are expected.

In moderate quality region, heat transfer due to two-phase forced convection is dominated in the presence of the flow in the annular liquid film. In the previous result by using the transparent heated tube, a distinct heat transfer deterioration was observed corresponding to the reduction in the turbulence in the annular liquid film, where the increase of annular liquid film thickness in microgarcity has a role of the secondary importance for the observed deterioration. Under high heat fluxes or extremely low mass velocities, generated bubbles by nucleate boiling are observed in the annular liquid film, and both of the interfacial behaviors and the resulting heat transfer coefficients become almost independent of gravity.

Critical heat flux

Measurement of CHF is quite difficult under the microgravity conditions of short period. There is almost no reliable systematic data on CHF conditions in microgravity. Utilizing the opportunity of ISS experiment, CHF data with high accuracy is to be obtained by gradual increase of heat flux to eliminate the effect of heat capacity of the heated tube assembly. CHF experiments are performed also at subcooled or low quality region where DNB becomes a dominant mechanism additionally to the CHF due to the dryout of annular liquid film at moderate or high quality. At the same time, temperature fluctuation resulted from the partial dryout between disturbance waves is recorded, and the mechanism of CHF as the starting of temperature excursion is to be clarified. To simulate the periodical cooling of dry patches by the liquid involved in the disturbance waves, the numerical data, e.g. the variation of annular liquid film thickness with time, is referred from the measurement in the non-heated test sections.

Dominant force regime map

It is clear that the gravity effect becomes smaller or disappears in the presence of strong inertia force of bulk flow or strong surface tension of interfaces. The definition of such dominant forces boundaries is important when we design the thermal devices for space application with high reliability. The establishment of the dominant force regime map is helped by the ground test by using mini-channels or micro-channels, where generated vapor occupied almost entire cross section as long vapor slugs often encountered also in microgravity. Such a situation is reflected by Bond numbers as a ratio of the body force to the surface tension force provided that the dimensionless parameter is appropriately defined. The definition of velocity involved in Froude and Weber numbers becomes a subject to be solved by the clarification of heat transfer mechanisms under different conditions of mass velocity and vapor quality. It is note worthy that there is no restriction for the change in the definition of dimensionless parameters and of the regime boundaries when flow pattern, pressure drop or CHF is concerned instead of heat transfer coefficient.

Concluding remarks

Subjects and problems concerning microgravity flow boiling are explained in brief. Because the opportunity of ISS experiment is very limited, the proposal of more detailed experimental conditions and planning of data analysis by the international collaboration is desired.

References

Ohta H, Microgravity Heat Transfer in Flow Boiling, Advances in Heat Transfer, Academic Press, Vol. 37, pp.1-76 (2003).

Two-Phase Thermal Transport in Microgap Channels – Theory, Experimental Results, and Predictive Relations

A. Bar-Cohen, J. Sheehan, E. Rahim
University of Maryland, Department of Mechanical Engineering, TherPES Laboratory
2106B Glenn Martin Hall, College Park, MD 20742, US
abc@umd.edu

The continued Moore's law progression in silicon technology and rapid developments in compound semiconductor materials are leading to ever higher chip power dissipation and heat fluxes in a wide range of computing, telecommunication, and photonic applications. Among the promising on-chip thermal management techniques, microgap cooler could provide unique advantages, eliminating the high and problematic thermal contact resistance and providing very high heat transfer coefficients by allowing heat flow directly into the dielectric liquid flowing and evaporating across the back surface of the chip or substrate.

A detailed analysis of microchannel/microgap heat transfer data for two-phase flow of refrigerants and dielectric liquids has shown that Annular flow is the dominant regime for this thermal transport configuration and grows in importance with decreasing channel diameter. A careful study of heat transfer rates in single miniature channels has identified the existence of a characteristic M-shaped heat transfer coefficient variation with quality (or superficial velocity), as shown in Figure 1. The second maximum, at moderate qualities in Annular flow, is postulated to reflect the onset of local dryout progressing to global dryout at near-unity qualities. Infrared

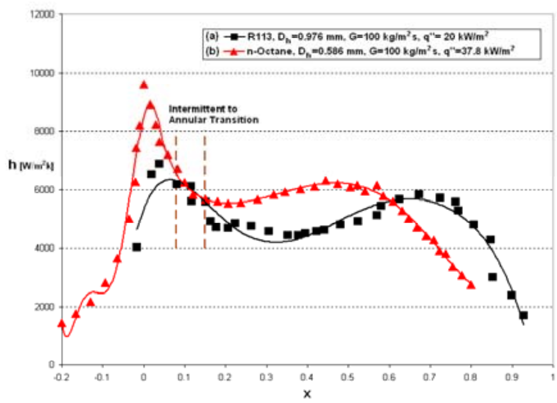


Figure 1: Two-phase flow characteristic M-shaped variation of the heat transfer coefficient

imaging of asymmetrically heated, microgap channels has revealed previously undetected, large amplitude, spatial and temporal surface temperature variations in Annular flow, as shown in Figure 2, which appear to support this hypothesis.

Integration of the results of IR thermography with a detailed, finite-element model of the laboratory microgap channel, made it possible to obtain inversely-determined, empirical heat transfer coefficients for the microgap channel and to confirm the existence of an M-shaped axial variation of the two-phase heat transfer coefficient. It has been found that for the higher quality conditions, in which Annular flow is attained, the venerable Chen correlation yields predictive agreement that is very often within 25% of the empirical heat

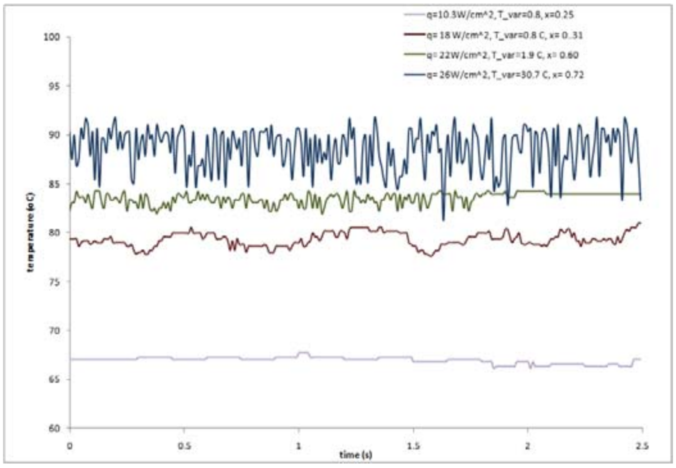


Figure 2: Heat Flux Effect on Wall Temperature Fluctuations - 210 micron channel, G=195.2 kg/m^2-s.

transfer coefficient data and provides a mechanistic context for the values attained at high qualities in microgap channels.

Biographies

Dr. Avram Bar-Cohen is an Honorary member of ASME, and Fellow of IEEE, as well as Distinguished University Professor of Mechanical Engineering at the University of Maryland. He is currently on assignment at DARPA in Virginia. Bar-Cohen's honors include the ICHMT's Luikov Medal (2008), ASME's Heat Transfer Memorial Award (1999), Edwin F. Church Medal (1994), and Worcester Reed Warner Medal (2000), and the IEEE CPMT Society's Outstanding Sustained Technical Contributions Award (2002). Bar-Cohen is the co-author of *Design and Analysis of Heat Sinks* (Wiley, 1995) and *Thermal Analysis and Control of Electronic Equipment* (McGraw-Hill, 1983).

Jessica Sheehan is nearing completion of her Ph.D. studies in Mechanical Engineering, focusing on two-phase cooling of electronics. She obtained her BS in Mechanical Engineering and BA in physics from the University of Virginia in 2006. She continued on to an MS in Mechanical Engineering at the University of Virginia in 2008, before joining the TherPES laboratory at the University of Maryland. Ms. Sheehan's PhD studies have been supported by an NSF Graduate Research Fellowship, A. James Clark Fellowship, L-3 Communications Fellowship, and a grant from ONR.

Emil Rahim is a PhD candidate at the University of Maryland studying two-phase flow in microgap coolers as applied to high heat flux electronics. Emil began his graduate studies at the Mechanical Engineering Department in September of 2005 and started his research at the PhD level in January 2007. Emil was granted an NSF travel award in 2006 to attend and participate in the 4th ICNMM International Conference in Limerick, Ireland. He worked at the Heat and Mass Transfer Laboratory (LTCM) at the Swiss Federal Institute of Technology (EPFL) in the summer of 2007.

Toward DNS-like of boiling flows

Richard Saurel

Aix Marseille University, Polytech, IUSTI UMR 6595
5 rue E. Fermi, 13453 Marseille Cedex 13, France
E-mail: Richard.Saurel@polytech.univ-mrs.fr

The Direct Numerical Simulation (DNS) of boiling flows poses serious theoretical and numerical challenges: liquid-gas interfaces, incompressible flow behavior in conjunction with compressible gas effects, capillary effects, phase transition at interfaces, heat conduction and viscous effects, contact angle modeling.

The approach promoted in this presentation deals with 'diffuse interfaces' modeling. In gas dynamics computations, since the pionner work of Von Neumann and Richtmyer (1950) and Godunov (1959) the discontinuities (shocks and contact discontinuities) are not tracked, nor fitted but *captured*. The diffuse interfaces approach considers interfaces capturing with a few computational cells with Godunov type schemes. The first method dealing with this approach is due to Saurel and Abgrall (1999) where the mixture cells present in the artificial smearing zone are solved with a two-phase averaged flow model in the presence of *stiff* mechanical relaxation, guarantying the interface conditions of equal normal velocities and pressures. Kapila et al. (2001) simplified this approach with the help of asymptotic analysis in the limit of stiff mechanical relaxation, by deriving a 5 PDEs model able to capture interfaces. These two types of models have nice features:
the same equations are solved everywhere with the same algorithm, merging and fragmentation is automatic, pressure waves are captured automatically as these models describe the mixture of compressible fluids.

Capillary effects have been inserted in the Kapila formulation by Perigaud and Saurel (2005). Later, Saurel et al. (2008) inserted phase transition effects at interfaces. The merging of these two extensions can be summarized in a unified formulation of diffuse interface with phase transition, surface tension and heat conduction:

$$\frac{\partial \alpha_1}{\partial t} + \vec{u}.\vec{\nabla}\alpha_1 = \frac{(\rho_2 c_2^2 - \rho_1 c_1^2)}{\frac{\rho_1 c_1^2}{\alpha_1} + \frac{\rho_2 c_2^2}{\alpha_2}} \mathrm{div}(\vec{u}) + \rho\nu(g_2 - g_1)\frac{\frac{c_1^2}{\alpha_1} + \frac{c_2^2}{\alpha_2}}{\frac{\rho_1 c_1^2}{\alpha_1} + \frac{\rho_2 c_2^2}{\alpha_2}}$$

$$+H(T_2 - T_1)\frac{\frac{\Gamma_1}{\alpha_1} + \frac{\Gamma_2}{\alpha_2}}{\frac{\rho_1 c_1^2}{\alpha_1} + \frac{\rho_2 c_2^2}{\alpha_2}}$$

$$\frac{\partial \rho Y_1}{\partial t} + \mathrm{div}(\rho Y_1 \vec{u}) = \rho\nu(g_2 - g_1)$$

$$\frac{\partial \rho}{\partial t} + \mathrm{div}(\rho \vec{u}) = 0$$

$$\frac{\partial \rho \vec{u}}{\partial t} + +\mathrm{div}\left(P\bar{\bar{I}} + \rho \vec{u} \otimes \vec{u} - \sigma\left(\left|\vec{\nabla} Y_1\right|\bar{\bar{I}} - \frac{\vec{\nabla} Y_1 \otimes \vec{\nabla} Y_1}{\left|\vec{\nabla} Y_1\right|}\right)\right) = 0$$

$$\frac{\partial \rho E}{\partial t} + \mathrm{div}\left((\rho E + P)\vec{u} - \sigma\left(\left|\vec{\nabla} Y_1\right|\bar{\bar{I}} - \frac{\vec{\nabla} Y_1 \otimes \vec{\nabla} Y_1}{\left|\vec{\nabla} Y_1\right|}\right).\vec{u} - \lambda_1 \vec{\nabla} T_1 - \lambda_2 \vec{\nabla} T_2\right) = 0$$

with the mixture total energy,

$$E = Y_1 e_1 + Y_2 e_2 + \frac{\sigma\left|\vec{\nabla} Y_1\right|}{\rho} + \frac{1}{2}\vec{u}^2 .$$

The first equation in this system expresses the gas volume fraction transport. This function is equal to nearly one in the gas and nearly zero in the liquid. It lies between these two extremes in the mixture zone, corresponding to the interface. The right hand sides of this equation express:
Pressure relaxation effects when velocity divergence is non zero, due to the eventual pressure waves presence.
Volume fractions variations due to mass transfers at the interface.
Volume fraction variations due to interfacial heat exchanges.
The second equation expresses the gas mass balance. The right hand side corresponds to the mass transfer, proportional to Gibbs free energies difference. The third equation corresponds to the mixture mass. The fourth one is the momentum equation involving a capillary tensor, where the surface tension is denoted by σ . Last the mixture energy equation contains the power of the capillary force and heat diffusion. Two temperatures are present as the model is in pressure equilibrium but out of thermal equilibrium.
With this approach, each phase has its own thermodynamics. The stiffened gas equation of state is used for each fluid (Le Metayer et al., 2004) and associated parameters are fitted on the saturation curves.

Two relaxation parameters are present and H and ν, related to heat and mass exchanges. In order to fulfill the local thermodynamic equilibrium interface conditions, these parameters are set infinite. See Saurel et al. (2008) for details.

This model has been solved in many pratical applications involving fast transient situations and high velocity flows with the help of a Godunov type scheme (Saurel et al., 2009). To deal with boiling flows, an extra difficulty appears due to the quasi-incompressible liquid behavior and low speed flow conditions. A preconditioned implicit Godunov type scheme is under building in this aim.

References

A. Kapila, R. Menikoff, J. Bdzil, S. Son and D. Stewart, Two-phase modelling of DDT in granular materials: Reduced equations, Phys. Fluids 13, pp 3002-3024 (2001)

R. Saurel and R. Abgrall, A multiphase Godunov method for compressible multifluid and multiphase flows, J. Comp. Phys. 150, pp. 425-467 (1999)

R. Saurel, F. Petitpas and R.A. Berry, Simple and efficient relaxation methods for interfaces separating compressible fluids, cavitating flows and shocks in multiphase mixtures. J. Comp. Phys. 228, pp. 1678-1712 (2009)

R. Saurel and F. Petitpas and R. Abgrall, Modeling phase transition in metastable liquids. Application to flashing and cavitating flows. J. Fluid Mech. 607, pp. 313-350 (2008)

G. Périgaud and R. Saurel, A compressible flow model with capillary effects. J. Comp. Phys. 209, pp. 139-178 (2005)

O. Le Metayer, J. Massoni and R. Saurel, Elaboration des lois d'état d'un liquide et de sa vapeur pour les modèles d'écoulements diphasiques. Int. J. of Thermal Sciences, Vol 43, pp 265-276 (2004)

Control of pool boiling incipience in confined space: effect of the dynamic morphing of the wall

Laetitia Léal[1], Marc Miscevic[1], Pascal Lavieille[1], François Pigache[2], Lounès Tadrist[3]

[1]LAPLACE, GREPHE, Toulouse University, 118 route de Narbonne, 31062 Toulouse cedex 9, FR, laetitia.leal@laplace.univ-tlse.fr, marc.miscevic@laplace.univ-tlse.fr, pascal.lavieille@laplace.univ-tlse.fr
[2]LAPLACE, GREM3, Toulouse University, 2 rue Charles Camichel, 31071 Toulouse cedex 7, FR, françois.pigache@laplace.univ-tlse.fr
[3]IUSTI, Technopôle de Château Gombert, 5 rue Enrico Fermi, 13453 Marseille cedex 13, FR, lounes.tadrist@polytech.univ-mrs.fr

The intensification of heat transfer is an important societal challenge in terms of saving energy and materials, thermal control, compactness, etc. One major problem in thermal management systems using liquid-vapor phase-change phenomena is the great temperature that must be reach to obtain the incipience of boiling (Dupont et al., 2003). In confined boiling situation, a way to enhance and control the heat transfer consists in imposing a dynamic deformation of the wall. Indeed, as it is possible to control liquid pressure by the dynamic morphing of the wall, a mean to decrease the wall superheating necessary to the onset of boiling is to dynamically deform the confinement wall. Decreasing the pressure induces an increase of the liquid metastability so that the nucleation begins at lower wall temperature.

Experimental setup

An experimental device was designed and realized to study the effect of dynamic deformation of the confinement wall on pool-boiling incipience. In this experimental device, the liquid (n-pentane) was confined between a heated wall and a confinement wall. The distance between these two walls was adjustable from 50 µm to 1 mm with an accuracy of 10 µm. The confinement wall was circular (diameter=4cm). It was dynamically deformed at its center whereas its periphery was maintained fixed. Significant wall morphing amplitude was obtained by using a piezoelectric actuator. The deformation amplitude range was from 0 to 250 µm and the deformation frequency range was from 0 to 500 Hz. The main parameters imposed in the experiments were the distance between the heated wall and the confinement wall, the heat flux and the amplitude and frequency of the deformation.

Discussion

First, experiments were carried out without dynamic wall morphing by varying the distance between the heated wall and the containment wall. Thus, the effect of confinement on the boiling incipience was highlighted. Then, experiments were carried out in confined space with dynamic wall morphing. The experimental procedure consisted in imposing a constant heat flux, the confinement wall being motionless. The same experiment was then carried out with dynamic deformation of the wall.

Figure 1 reported the effect of the confinement and the effect of dynamic wall morphing on boiling incipience. Free boiling designated an experiment carried out without confinement effect. Confined boiling designated an experiment carried out with a distance between the heated wall and the confinement wall equal to 625 µm. Confined boiling with wall morphing designated an experiment carried out with a distance between the heated wall and the confinement wall equal to 625 µm, a deformation frequency equal to 100 Hz and a deformation amplitude equal to 210 µm. Experimental results reported relatively small differences between free and confined boiling incipience superheating: 22.3 K without confinement and 18.5 K with confinement. Contrariwise, experimental results highlighted large difference between confined boiling incipience superheating without and with wall morphing: with wall morphing, boiling incipience was obtained for wall superheating down to 0.8 K.

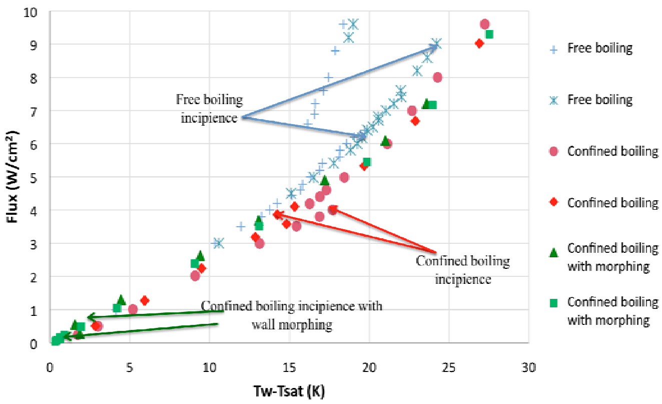


Figure 1: Pool boiling curves obtained without and with confinement effect, and boiling incipience superheats measured in the various experimental configurations.

	Free boiling	Confined boiling	Confined boiling with wall morphing
Model	25 K	25 K	22K
Experiment	22.3K	18.5K	0.8K

Table 1: Theoretical and experimental boiling incipience superheats

Simultaneously to the experimental campaign, theoretical considerations were developed both from hydrodynamic and thermodynamic point of view. Table 1 highlighted a good agreement between experimental and theoretical results when confinement wall was not deformed. Great differences were obtained when the wall morphing was imposed, indicating that some physical phenomena not considered in the model need to be taken into account (compressibility, convection, etc.).

References

Dupont V. et al., Boiling Incipience of Highly-Wetting Liquids in Horizontal Confined Space, Int. J. Heat Mass Transfer., vol. 46, Issue 22, pp. 4245-4256, 2003.

Investigation of wall temperature and heat flux distribution during nucleate boiling in the presence of an electric field

Nils Schweizer[1], Paolo Di Marco[2], Peter Stephan[1]

[1] Institute of Technical Thermodynamics, Technische Universität Darmstadt
[2] Department of Energy and Systems Engineering, University of Pisa
p.dimarco@ing.unipi.it

Heating wall temperature distribution and bubble shapes were measured during nucleate boiling of N-perfluorohexane (nPFH) in microgravity during the 50th ESA parabolic flight campaign. The overall results of these multi-scale investigations can be found in Schweizer (2010). The presented results here focus on the influence of an electric field and were achieved within a collaboration of the universities of Pisa and Darmstadt.

The high heat transfer performance of pool boiling processes makes them interesting for space application where cooling issues have to be treated with special attention due to the absence of natural convection in conjunction with weight and volume limitations. Straub (2001) stated that the establishment of a stable pool boiling process is possible in weightlessness. It relies on the mechanism of a large primary vapor bubble hovering above the heated surface, picking and gathering smaller bubbles emerging from the surface. However, the detachment and removal of this primary bubble has to be ensured to prevent dry-out of the surface. This may be accomplished with an electric field.

Boiling in the presence of an electric field was already subject of a number of investigations. The present study is complementary to the research of Di Marco and Grassi (2007) and it is focused on the local temperature and heat flux of the heated wall. Beside the scientific objectives, the performed experiments served also as a preparation of the RUBI (Reference mUlti-scale Boiling Investigation) experiment for the Fluid Science Laboratory on board the International Space Station.

A scheme of the experimental setup is shown in Figure 1. A 20 µm stainless steel foil served as heated wall for the boiling process. The temperature distribution of the foil was measured via infrared thermography at a frame rate of 1000 Hz and with a resolution of 30 µm/pixel. The bubbles were observed by a synchronized high speed camera. A washer-shaped electrode (outer diameter 15mm, hole diameter 5 mm, thickness 1 mm) was located parallel to the heated wall in a distance of 5 mm. The electrode was charged to up to +10kV while the heating foil served as ground potential.

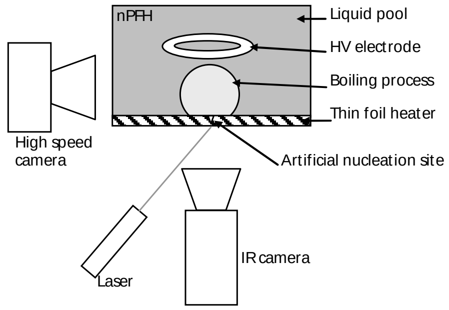


Figure 1: Scheme of the experimental setup.

Due to the lower electric permittivity of nPFH vapor compared to nPFH liquid the vapor is driven towards the zone of weaker electric field, as explained in detail by Di Marco and Grassi (2007). The electric field created by the electrode has a local minimum in the centre of its hole. Thus, the vapor bubbles are attracted towards this region (see Figure 2, left image) and furthermore they show a typical elongated shape due to the action of the electric stress. The characteristic ring-shaped imprints of the bubble in the local temperature and heat flux distribution are visible in the middle and left images of Figure 2. It can be seen that high heat fluxes occur in the vicinity of the contact line leading to a local temperature drop of the heating foil.

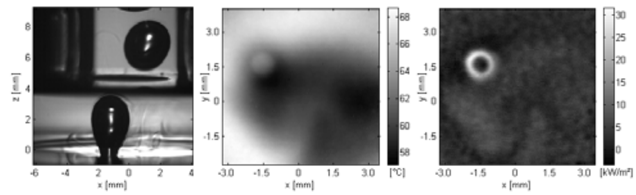


Figure 2: Elongated vapor bubble (left), characteristic local temperature (middle) and heat flux (right) imprint during boiling in microgravity with activated electric field.

Additional to the investigations at single bubbles, fully developed boiling was also studied. The global temperature and heat flux development as well as local imprints were compared for boiling in microgravity with and without the electric field and boiling in normal gravity conditions without the electric field. The results prove the existence of a reliable bubble detachment and re-wetting mechanism, promoted by the electric field, allowing the implementation of a nucleate boiling process in microgravity.

The authors acknowledge ESA for financial and organizational support and access to the parabolic flight campaigns.

References

Schweizer, N.: *Multi-Scale Investigation of Nucleate Boiling Phenomena in Microgravity*. Darmstadt, Techn. Univ., PhD thesis, 2010. (URL: http://tuprints.ulb.tu-darmstadt.de/2377/, URN: urn:nbn:de:tuda-tuprints-23770)

Straub, J.: Boiling Heat Transfer and Bubble Dynamics in Microgravity, *Advances in Heat Transfer*, Vol. 35, 2001, pp. 57-172.

Di Marco P., Grassi W.: Pool boiling in microgravity: old and recent results, *Multiphase Science and Technology*, 19, 2, 2007, pp.141-165.

A general theory of moving contact lines with phase change

Pierre Colinet and Alexey Rednikov

Université Libre de Bruxelles, Faculty of Applied Sciences, TIPs (Transfers, Interfaces and Processes) – Fluid Physics Unit
50 av. F.D. Roosevelt, C.P. 165/67, Brussels, B-1050 Belgium
pcolinet@ulb.ac.be, aredniko@ulb.ac.be

Far from claiming any ultimate resolution of the contact line paradoxes, we argue that a somewhat controversial paradigm, originally employed by de Gennes and collaborators (see e.g. [1, 2]), actually appears both to be quite reasonable at its foundations and to lead to physically consistent final results in a wide variety of situations. Curiously enough, while containing a singularity in itself, the approach nonetheless renders the classical contact-line singularities – both hydrodynamic and thermal – integrable, in particular as far as several quantities of interest are concerned (viscous dissipation, total heat flux, …). It is also readily applicable to quite a few situations: from equilibrium shapes and moving contact lines of a non-volatile liquid [1, 2], to cases with evaporation into either a pure-vapor [3] or an inert-gas [4] atmosphere. The paradigm actually consists in an approach involving both the (positive or negative) spreading coefficient and the disjoining pressure in the form of a positive inverse cubic law, a conceptual framework that most notably describes microstructures with *truncated* precursor films on a macroscopically bare solid surface (see left inset of Fig. 1).

In the same situation (positive disjoining pressure tail), there exists another well-known microstructure of the contact line, characterized by an *extended* precursor film covering all apparently "dry" parts of the solid surface (see right inset of Fig. 1). This solution was introduced by Potash and Wayner [5] and used in many studies since then (see also [6] for further references, an extensive parametric analysis and a discussion of the role of Marangoni and vapor recoil effects). As there is no truly bare zone in this case, this solution does not depend on the spreading coefficient. However, we argue here that the regime with a truncated microfilm is preferred if the spreading coefficient is below a positive (still perfect wetting) critical value dependent upon the superheat, in which case the extended-microfilm thickness is smaller than that of the "pancake" introduced by de Gennes and co-workers [2]. Conversely, for a given positive spreading coefficient, there is a critical superheat above which the microfilm gets truncated, whereas for a negative one (partial wetting) the truncated regime should be preferred at any superheat. A parametric study of the apparent contact angle (a nonlinear eigenvalue of the steady microstructure problem) versus the spreading coefficient is carried out, some results of which are shown in Fig. 1. The spreading "number" Σ is defined by

$$\Sigma = 2\frac{S^*}{\gamma^*}\varepsilon^{-2}, \text{ with } \varepsilon = \left(\frac{3^{3/2} a^* \rho_l^* L^* \Delta T^*}{\gamma^* T_0^*}\right)^{1/3}$$

where S^* is the spreading coefficient, γ^* the surface tension, a^* is a molecular length scale, ρ_l^* is the liquid density, L^* the latent heat, ΔT^* the superheat, T_0^* the saturation temperature (at a given vapor pressure), and ε must remain small for the lubrication approximation (applied throughout the present analysis) to be valid.

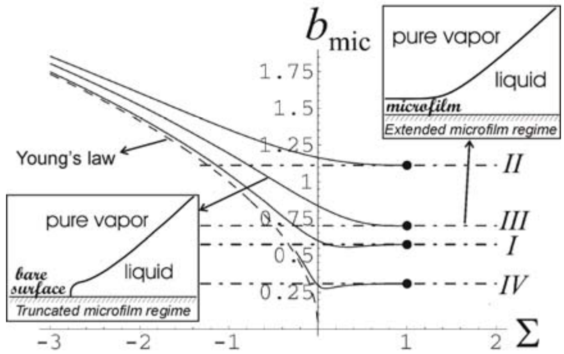


Figure 1: The (scaled) apparent contact angle b_{mic} of the microstructure (the true one being $\theta_{\mathrm{mic}} = \varepsilon\, b_{\mathrm{mic}}$), as a function of the spreading number Σ, for various cases (labeled I to IV). $\Sigma = 1$ corresponds to the threshold for the transition between extended and truncated microfilm regimes.

Note in particular from Fig.1 that for large negative spreading number Σ (e.g. for small superheat), Young's law is asymptotically recovered (dashed curve).

In addition to a discussion of these two forms of microstructures and their relative stability, this presentation will also discuss the behavior of the apparent contact angles as measured at large scales, where the effect of contact line motion becomes important. Somehow expectedly, a generalized form of the Tanner-Cox-Voinov law is obtained in this case, independently of the microstructure regime considered (except through some numerical coefficients).

References

[1] Hervet H. and de Gennes P.G., The dynamics of wetting: precursor films in the wetting of 'dry' solids, C.R. Acad. Sci. Paris, Vol. 299 II, pp. 499-503 (1984).
[2] de Gennes P.G., Brochard-Wyart F. and Quéré D., *Capillarity and wetting phenomena* (Springer, New York, 2004).
[3] Rednikov A.Ye. and Colinet P., Truncated versus extended microfilms at a vapor-liquid contact line on a heated substrate, Langmuir, Vol. 27, pp. 1758–1769 (2011).
[4] Colinet P. and Rednikov A.Ye., On integrable singularities and apparent contact angles within a classical paradigm. Partial and complete wetting regimes with or without phase change, in press, Eur. Phys. J. – Special Topics (2011).
[5] Potash M. and Wayner P.C., Evaporation from a two dimensional extended meniscus, Int. J. Heat Mass Transfer, Vol. 15, pp. 1851-1863 (1972).
[6] Rednikov A.Ye., Rossomme S. and Colinet P., Steady microstructure of a contact line for a liquid on a heated surface overlaid with its pure vapor: parametric study for a classical model, Multiphase Science and Technology, Vol. 21, pp. 213-248 (2009).

The effect of three-phase contact line speed on local evaporative heat transfer: Experimental and numerical investigations

Peter Stephan[1,2], Christian Kunkelmann[1], Khalid Ibrahem[1,2], Nils Schweizer[1], Tatiana Gambaryan-Roisman[1,2]

[1]Institute for Technical Thermodynamics, Technische Universität Darmstadt, Darmstadt 64287, Germany
[2]Center of Smart Interfaces, Technische Universität Darmstadt, Darmstadt 64287, Germany
pstephan@ttd.tu-darmstadt.de

In pool boiling or flow boiling devices or e.g. during meniscus evaporation within capillary structures the local heat flux and evaporation rate at the position where the liquid-vapor interface meets the solid wall can be extremely high. This three-phase contact line region is characterized by a thin liquid film with a very low heat resistance. Depending on the application the contact line can move with velocities of several meters per second, either in receding (dewetting) or in advancing (wetting) direction.

In this presentation, experimental and numerical results on the influence of three-phase contact line speed on the local heat transfer in the contact line region during pool boiling and during meniscus evaporation are presented and analyzed (Fig.1).

It is shown that the local heat flux can be one or more orders of magnitude higher than the mean heat flux supplied to the system. This local heat transfer peak is almost independent of the contact line speed in the case of a receding contact line while it significantly increases with increasing contact line speed for an advancing contact line. This behavior could be observed in different evaporation configurations and with different fluids. Experimental and numerical results agree well and allow a characterization of the transient heat transfer phenomena in the contact line region during evaporation (Fig.2). It is shown that:

- In case of an advancing meniscus the local maximum heat flux near the 3-phase contact line increases with rising contact line velocity.
- In case of a receding meniscus the local maximum heat flux near the 3-phase contact line is almost independent from the contact line velocity.
- The local maximum heat flux near the 3-phase contact line is typically more pronounced in case of an advancing meniscus than in case of a receding meniscus.
- The heat transfer next to the contact line is governed mainly by two mechanisms in superposition: micro layer evaporation and transient conduction.
- The contribution of micro layer evaporation to the overall heat transfer is nearly constant in both situations, advancing and receding contact line, and independent from the contact line speed.
- The contribution of transient conduction increases in case of the advancing contact line with rising contact line velocity, which leads to enhanced heat transfer under these circumstances.

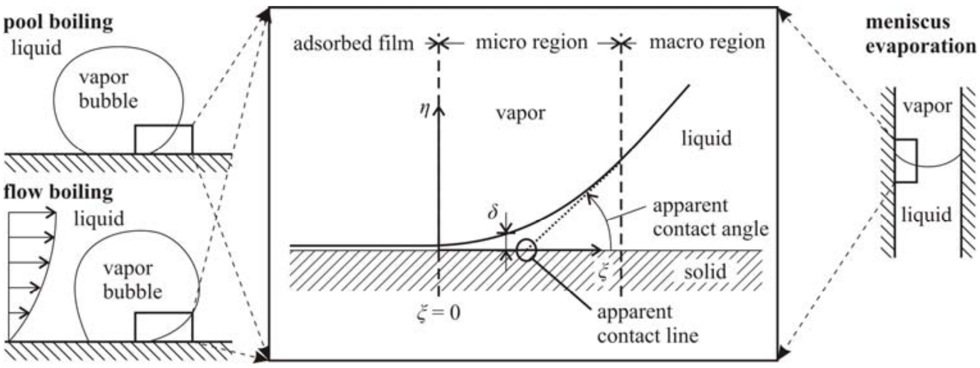


Figure 1: Definition of the micro region, the apparent contact line and the apparent contact angle.

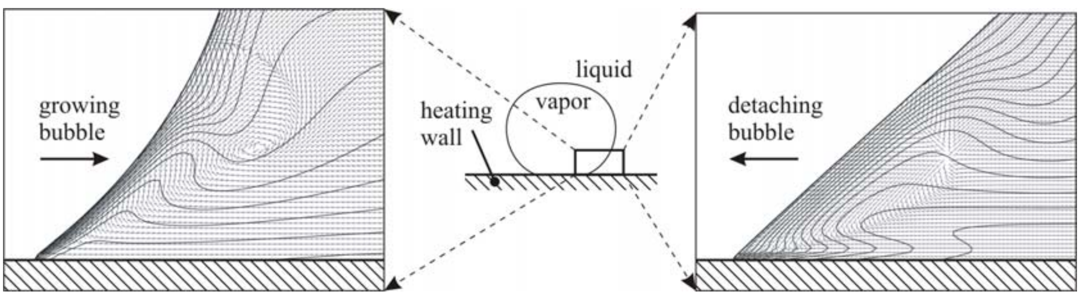


Figure 2: The temperature and velocity field in vicinity of the contact line for a growing and a detaching bubble.

Rotational vibration effect on the onset of morphological instability in the directional solidification of binary alloys

Tatyana Lyubimova, Yanina Parshakova

Institute of Continuous Media Mechanics UB RAS, Computational Fluid Dynamics Laboratory
1, Koroleva Str, Perm, 614013, Russia
lubimova@psu.ru

It is known that in the solidification process of binary alloys the crystallization temperature does not remain constant, it depends on the solute concentration at the solidification front. In the present paper the directional solidification of binary alloys with segregation coefficient less than unity is studied. In this case, during the motion of crystal-melt interface, an excess of solute is formed in front of him which does not have time to enter into the crystal. Since the diffusion coefficient is usually small, a large concentration gradient arises in front of interface. As a result, the crystallization temperature increases with the distance from the front. If the crystallization velocity, and hence the concentration gradient, are large enough, then the growth of crystallization temperature is faster than the increase of temperature in the melt. Due to that, at short distance from the front, the melt is supercooled and small perturbations of the front begin to grow. An instability, known as morphological instability [1] arises.

Radial inhomogeneity of temperature in the melt, arising due to the difference in thermal conductivities of liquid and solid phases and to the heat release of phase change, leads on one side to the curvature of the solidification front, and on the other side to the convective flow. The effect of this flow on the solidification front in binary melt can lead to the formation of pit at the crystallization front in the vicinity of the axis [2]. The key point for the formation of pit is the presence, in the vicinity of the front, of radial flow directed towards the axis. The suppression of this radial flow or at least its displacement from the front to a distance greater than the thickness of diffusion layer, could prevent the formation of pit or at least to reduce its depth. In the present paper we analyze the possibility to achieve these effects applying rotational vibration of ampoule around its axis.

The effect of rotational vibrations on the onset of morphological instability in the directional solidification of binary alloys was studied numerically in [3] for high frequency vibrations in the framework of average approach. The calculations based on steady approach have shown that the vibrations lead to the generation of average flow, localized near the solidification front, the direction of flow induced by vibrations is the opposite to the buoyancy convection. The interaction of this flow with the buoyancy convection in the terrestrial conditions leads to the displacement of the gravitational vortex from the solidification front. As a result, the rotational vibrations prevent the formation of pit, the radial segregation of dopant on the crystallization front decreases with the increase of vibration intensity. The greatest effect of vibrations is observed in the case of binary system with heavy dopant.

In the present paper, numerical simulation of the directional solidification in the presence of rotational vibrations of finite amplitude and frequency is carried out on the basis of full unsteady equations and boundary conditions. The calculations are made by finite volume method, in axisymmetric approach. The systems with low phase change temperature (succinonitrile doped with acetone or ethanol (light solute) and succinonitrile doped with salol (heavy solute)) are considered. Nonuniform heat transfer processes on the sidewall, the solidification front motion and curvature, the presence of a two-phase zone between the melt and the crystal and the temperature dependence of the solidus and liquidus of the dopant concentration are taken into account. The data on the temporal evolution of velocity, temperature and solute concentration in the melt during the solidification process, as well as data on the distribution of solute in the grown crystal are obtained for various vibration frequencies. It is demonstrated that vibrations may prevent the formation of pit. They also result in a significant lowering of the radial segregation of dopant (see, Fig.1) and in an increase of the homogeneity of dopant distribution in the grown crystal.

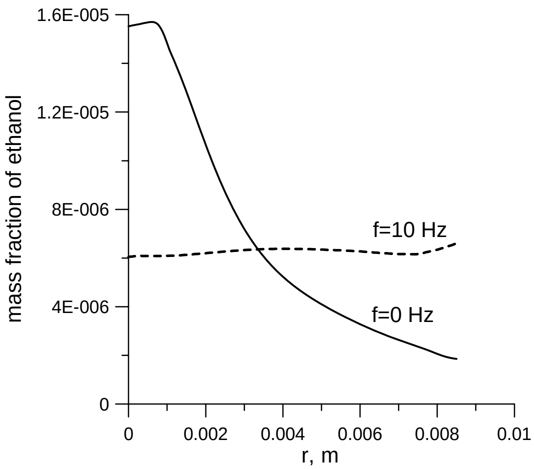


Fig.1

References

1. Mullin V.V., Sekerka R.F. Stability of a planar interface during solidification of a dilute binary alloy. J. Applied Physics. 1964. Vol. 35. Issue 2. P.444-451.

2. Lan C.W., Tu C.Y. Morphological instability due to double diffusive convection in directional solidification: pit formation. J. Cryst. Growth. 2000. Vol. 220. P.619-630.

3. Lan C.W., Lyubimov D.V., Lyubimova T.P., Ospennikov N.A., Parshakova Ya.N., Yu W.C. Influence of high-frequency vibration on the morphological instability in the directional crystallization of binary melts. Fluid Dynamics, 2008, Vol. 43, No. 4, pp. 514–523.

Dynamics of particle confined between two oscillating walls

Stanislav Mingalev[1], Dmitriy Lyubimov[1], Tatyana Lyubimova[2]

[1]Perm State University, Theoretical Physics Department
15, Bukireva Str., Perm, 614690, Russia

[2]Institute of Continuous Media Mechanics UB RAS, Computational Fluid Dynamics Laboratory
1, Koroleva Str, Perm, 614013, Russia
SMingaliov@gmail.com, lubimov@psu.ru, lubimova@psu.ru

The paper deals with the investigation of the dynamics of particle confined between two oscillating walls. The study is motivated by the preparation of space experiment on the behaviour of dissipative granular gas and cluster formation under vibrations. The interest to the dynamics of particle confined between two oscillating walls arises when Fermi [1] proposed the theory of cosmic radiation according to which the origin of the cosmic radiation is related to the accelerated motion of electrons in alternating magnetic fields. This process is similar to the elastic collisions of light and heavy bodies. Ulam analyzed a simplified model - dynamics of light particle confined between the fixed and oscillating walls assuming that the collisions of particle and walls is perfectly elastic. He found that the average velocity of particle remains constant while the instantaneous velocity changes in a random way.

Zaslavskij [2], Chirikov, Brahic [3] and Lieberman [4] have shown that the phase space can be divided into three domains: the domain of small particle velocities where the motion is stochastic, the intermediate velocity domain where the phase space it looks like stability islands in the stochastic sea. In the domain of large velocities all phase trajectories are regular stable.

In [4] chaos was found for both perfectly elastic and inelastic collisions. It was shown by Luck and Mehta in [5,6] that these results are related to the fact that the motion of the platform was ignored. They took into account the motion of the platform and proved the absence of chaos in these cases. They suggested the following explanation of the behaviour. When the particle lands in the absorbing region of the platform cycle (where it is kinematically impossible for it to be launched upwards), it "sticks" to the platform. It is then relaunched in the beginning of the transmitting region of the next cycle, with zero relative velocity. This periodic relaunching under identical kinematic conditions stops the evolution to chaos via a period-doubling route for the restitution coefficient $A<1$.

The goal of the present paper is to study the "sticking" of the particle in the framework of the Fermi-Ulam model taking into account the finite restitution and to analyze its influence on the stability of phase trajectory of particle. It is found that the particle can "stick" to the wall if in the moment of collision the wall acceleration is negative and its velocity is positive. Moreover, if in this moment the particle velocity equals to the wall velocity, the second collision with the same wall happens immediately after the first collision, and then this situation repeats again and again while the wall acceleration is negative. The particle is then relaunched with zero relative velocity. If the difference in the velocities of wall and particle is small, the next collision happens finite time after the first one however the time interval between the next collisions decreases and some time after the particle "sticks" to the wall. When the restitution coefficient A decreases, the range of the particle velocity, for which the particle "sticks" to the wall, increases.

The particle can "stick" to the wall in the case of perfectly elastic collisions too, however the moment of the relaunching strongly depends on the velocity at the moment of first collision with the wall. As the result, in the case of perfectly elastic collisions (A=1) the "sticking" leads to the chaotization of particle behavior. The ratio of distance between nearby phase trajectory at the moment after "sticking" to the distance at the moment before "sticking" quickly decreases, if the coefficient of restitution A becomes less than unity. The ratio is approximately 2 for A=0.999, while it is approximately 0.1 for A=0.99. The "sticking" in the case of elastic collision (A<1) leads to depression of chaos in the Fermi-Ulam model.

The dependence of average velocity of particle on the parameters was also investigated. It is found that in the case of small amplitude of wall coordinate variations and small A the average velocity is substantially greater than that in the case of large amplitude and restitution coefficient A close to unity. Using the approximation in which the wall coordinate is constant, while its velocity changes, attractive fixed point is found for small amplitude of wall coordinate variations. The decrease of average velocity is explained by the bifurcation of the fixed point.

The research described in this publication was made possible in part by award No. PE-009 of the U.S. Civilian Research & Development Foundation for the Independent States of the Former Soviet Union (CRDF).

References

1. Fermi E. On the Origin of the Cosmic Radiation. , Physical Review, 1949, Vol. 75, pp. 1169–1174.

2. Zaslavskii G. M., Sagdeev R. Z., Introduction to Nonlinear Physics [in Russian], 1978, Nauka, Moscow.

3. Brahic A., Numerical Study of a Simple Dynamical System, Astron. and Astrophys, 1971, Vol.12, pp. 98-110.

4. Lichtenberg A.J., Lieberman M.A. R. H. Fermi acceleration revisited, Physica D: Nonlinear Phenomena, 1980, Vol. 1, Issue 3, pp. 291-305.

5. Luck J.M., Mehta A. Bouncing ball with a finite restitution, Physical review E, 1993, Vol. 48, Issue 5, pp. 3988–3997.

6. Chirikov B.V. A universal instability of many-dimensional oscillator systems, Physics Reports, 1979, Vol. 52, Issue 5, pp. 263-379.

Generation of average flow in a pulsating flow near the curved gas-liquid interface

Dmitriy Lyubimov, Lyudmila Klimenko

Perm State University, Theoretical Physics Department, 15, Bukireva Str., 614990, Perm, Russia
lyubimov@psu.ru

We study generation of average flow near the curved gas-liquid interface under the effect of vibrations. As known, one of the interesting manifestations of the vibrations is the arising of average effects. Thus, in the dynamical boundary layer near the solid wall streamlined by nonuniform pulsating flow, the average vorticity is generated, moreover, at the outer edge of the boundary layer the average tangential velocity is nonzero and its value can be used as a boundary condition for finding an average flow in the bulk. For a flat interface between two liquids, additionally to the above mechanism of generation, there is also a mechanism of Dore (1973), which determines the effective shear stress.

For a free flat surface these mechanisms do not work, and only in the presence of surface deformations there arises another mechanism - the mechanism of Longuet-Higgins (1953), related to the average flow generation by propagating surface waves. However, none of these mechanisms works if the surface is free and non-deformable.

In the present paper a new mechanism for average flow generation near the curved free surface is found, that is associated exclusively with the average surface curvature. This mechanism is insensitive to the pulsating deformations, it is studied for the problem of gaseous bubble suspended in an unbounded liquid, neglecting the viscosity and density of the gas. It is found that around the bubble there is a steady flow, and surface deformations has no effect on the generation of this flow. Thus, the average flow is possible even when the surface undeformable, for example, in the case of very large surface tension. The formula is obtained, giving the value of the viscous stress at the outer boundary of boundary layer. These stresses, irrespective of the geometry, are proportional to the square of the curvature of the surface and the cube of the thickness of the Stokes boundary layer, and can play the role of effective boundary conditions for the flow in the outer region.

The generation of average flow near the curved interface between two liquids is also investigated. A liquid drop suspended in a vibrating liquid of other density is considered. Effective boundary conditions for the jumps of tangential velocity and shear stress, describing the above generation have been obtained. It is shown that at comparable density and viscosity of liquids the main mechanism of generation of average vorticity is similar to the Schlichting mechanism, whereas the effect of the interface curvature is less important factor. It should be expected that at low density medium inside the drop (and its small dynamic viscosity), all of the above mechanisms could contribute with the same order of magnitude.

In the present paper we study the interaction of these generation mechanism considering, as an example, a gaseous bubble suspended in a liquid.

To study the generation of average flow near the curved interface, we consider the behavior of a gaseous bubble suspended in an unbounded liquid. The flow of liquid and gas occurs under the influence of external vibrational field. Far from the bubble, the liquid moves progressively, according to sinusoidal law with amplitude b and frequency ω. The host liquid and the gas inside the bubble are considered as incompressible. The surface tension is assumed to be such large that the bubble surface can be considered as non-deformable.

The problem is characterized by three dimensionless parameters :

$\varepsilon = \dfrac{b}{a}$ – the ratio of the vibration amplitude to the bubble radius,

$\delta = \dfrac{1}{a}\sqrt{\dfrac{\nu_1+\nu_2}{\omega}}$ – the dimensionless boundary layer thickness and

$\gamma = \sqrt{\dfrac{\rho_2}{\rho_1}}$.

It is assumed that all these parameters are small. The uniformly valid expansion can be constructed only when all these parameters are of the same order.

Using the method of matched asymptotic expansions we obtain the effective boundary conditions describing the generation of average flow:

$$u^1{}_\vartheta - u^2{}_\vartheta = \frac{5}{4\omega}\upsilon\upsilon_\tau ,$$

$$[\sigma_{r\vartheta}]_{\delta\to 0} = -\frac{1}{4}\rho_2\upsilon\upsilon_\tau\sqrt{\frac{2\nu_2}{\omega}} - \frac{3}{2}\frac{\rho_1}{a^2}\upsilon\upsilon_\tau\left(\frac{2\nu_1}{\omega}\right)^{3/2}$$

where $\upsilon = 3b\omega\sin\vartheta$ is the amplitude of the tangential velocity of pulsating motion of the fluid outside the boundary layer, $\upsilon_\tau = \dfrac{1}{a}\dfrac{\partial\upsilon}{\partial\vartheta}$.

References

Dore D. On mass transport induced by interfacial oscillations at a single frequency, Proc. Camb. Phil. Soc. 1973. V. 74. P. 333-347.

Longuet-Higgins M.S. Mass transport in water waves, Philosophical Transactions A. 1953. V. 245. P. 353-581.

Gravitational Stability of Water Over Vapor in Geothermal Reservoirs

Georgy Tsypkin[1], Andrey Il'ichev[2]

[1] Institute for Problems in Mechanics RAS, Vernadskogo ave. 101, Moscow, 119526, Russia
[2] Steklov Mathematical Institute RAS, Gubkina str. 8, Moscow, 119991, Russia
E-mail: tsypkin@ipmnet.ru

Geological investigations and thermodynamic analysis show that in geothermal systems a water saturated layer may exist over a vapor saturated layer (White et al., 1971). In the framework of classical hydrodynamics the existence of heavy fluid over the light one is always unstable.

Schubert and Straus (1980) gave an example of the geothermal system, where a phase transition interface separates the water and vapor saturated regions. It was assumed that the lowest boundary of the geothermal system is a contact surface of impermeable rock and vapor saturated region and both phases are motionless at the unperturbed state. The authors presented linear stability analysis and numerical investigation of the dispersion equation. It was found that the critical value of the permeability coefficient, which separates the domains of stable and unstable states is about $k \sim 4 \cdot 10^{-17}$ m^2. If permeability is higher than the critical value than the system loses its stability. The authors noted that this critical value of permeability is less by the order of magnitude than the characteristic value of water over vapor geothermal systems. Thus, stability of the sufficiently high-permeability geothermal systems cannot be explained.

Tsypkin and Il'ichev (2004) considered the more complicated water over vapor geothermal system where motion of the both phases and phase change through the interface in basic unperturbed state were allowed (Fig. 1).

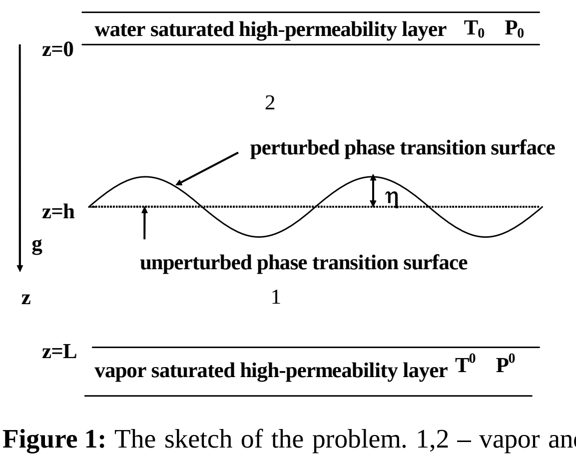


Figure 1: The sketch of the problem. 1,2 – vapor and water saturated regions.

For the basic stationary solution the unknown parameters are the location of the interface *h* and also the temperature and pressure on this surface. Analysis of linear stability of the basic solution showed that the stable regimes of fluid flows exist for permeability values exceeding the critical value obtained by Schubert and Straus by an order of magnitude. The new threshold of permeability explained the stability of a number of natural geothermal systems. In Tsypkin and Il'ichev (2004) a smallness was assumed of advective energy transfer as compared with the conductive transfer. Therefore the results can be applied for the geothermal systems, which characterized by small value of the permeability coefficient.

In the present work we generalized the last results for large values of the permeability when advective energy transport plays an important role (figure 2). The calculation showed that the stable regimes exist for an arbitrary value of the permeability.

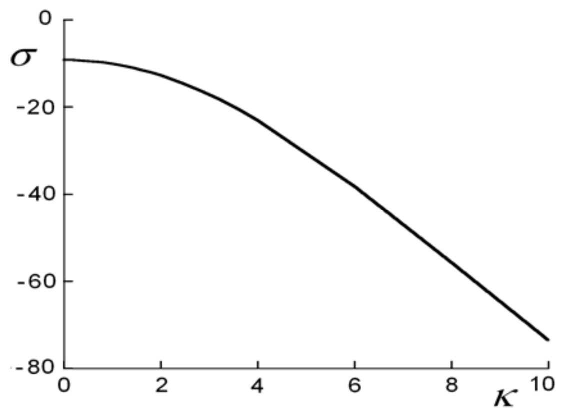


Figure 2: Decay rate σ versus wavenumber κ at $k \sim 10^{-15}$ m^2.

For high values of the permeability, in general case, the interface locates near the boundary of the vapor saturated region, and setting a constant pressure at the boundary does not reflect physics of the phenomenon. Then, in the case of large mass flux we need to treat the conjugate problem, taking into consideration interaction of flows in the region and in the high-permeability layer (fracture).

References

White D.E., Muffler L.J.P., and Truesdell A.H. Vapor-dominated hydrothermal systems compared with hot water systems, Econ. Geol., Vol. 66 pp. 75-97 (1971)

Schubert G.. and Straus J.M., Gravitational stability of water over steam in vapor-dominated geothermal systems, J. of Geophysical Research, Vol. 85 pp. 6505-6512 (1980)

Tsypkin G.G. and Il'ichev A.T. Gravitational stability of the interface in water over steam geothermal reservoir, Transport in porous media, Vol. 55 pp. 183-199 (2004)

Interaction between Flow Structure and Mass Transport of an Acoustically Levitated Droplet

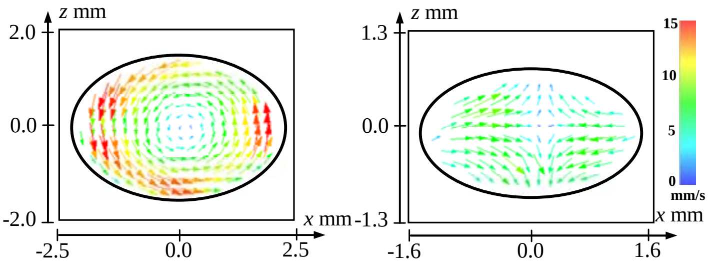
Hikaru Ishii[1 (a)], Yutaka Abe[1 (b)], Akiko Kaneko[1 (c)], and Koji Hasegawa

[1]Department of Engineering Mechanics and Energy, University of Tsukuba
1-1-1 Tennoudai, Tsukuba, Ibaraki, 305-8573, Japan
h-ishii@edu.esys.tsukuba.ac.jp[(a)], abe@kz.tsukuba.ac.jp[(b)], kaneko@kz.tsukuba.ac.jp[(c)]

It is crucial to save analysis time and cost in modern analytical chemistry. Analytical methods such as mass spectrometry, capillary gas chromatography, only require sample volumes in the microliter or submicronliter range. Handling of such small sample volumes is difficult in view of sample contamination and sorption processes on walls of the containers that are usually used to hold the samples during analytical procedures. Acoustic levitation is well suited for sample pre-treatment because surface tension and density are the only properties that determine the viability of the method, but suffers from evaporation and loss of solvents. R. Tuckermann, et al (2002) investigated evaporation rate of several kinds of liquid i.e. alkanes and alkanols. On the other hand, it is widely known an acoustic streaming is generated around levitated droplet and complex flow also appeared in a droplet. A. Yarin, et. al. (1999) indicated that flow field around a levitated droplet affects the rate of mass transfer. However, the effect of surrounding flow and mass transfer rate of an acoustically levitated droplet is not sufficiently investigated. The purpose of this study is to investigate the interaction between internal and external flow structure of an acoustically levitated droplet and the rate of mass transfer. Internal and external flow structure of levitated droplet is measured by Particle Image Velocimetry (PIV). The evaporation process of levitated droplets was recorded through the high speed camera with back light method and surface temperature of droplet is measured through the radiation thermometer simultaneously.

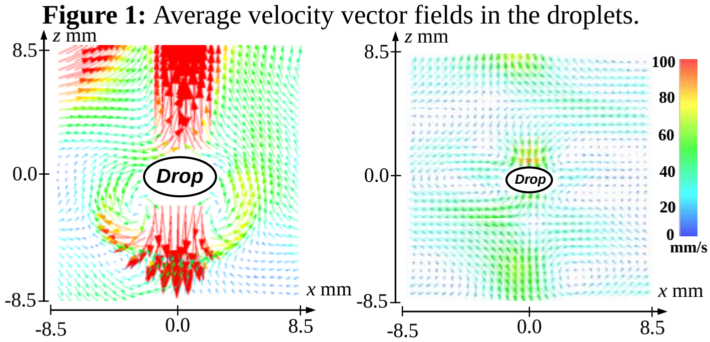
Fig. 1 shows the average vector fields in levitated droplets. Fig. 1(a) describes a water droplet at 4.4 [mm] of equivalent volume diameter, 162 [dB] of SPL had circular vortex in the droplet. In the case of ethanol droplet at 2.4 [mm], 160 [dB], two toroidal vortices appeared in the droplet, as shown Fig.1 (b). Fig. 2 shows the average velocity vector fields around the droplets. Fig. 2 (a) describes a water droplet had toroidal vortex below it. In the case of ethanol droplet, two toroidal vortices appeared below and above it, as shown in Fig. 2 (b). In comparison to flow structure of water and ethanol, the flow direction was opposite and it was found that the interaction between internal and external flow structure of ethanol droplet become strong than water droplet.

Fig. 3 shows evaporation process that evaluated by using size measurements of droplets. Evaporation process of droplet can be described with d^2-law i.e. a liner decrease of the squared diameter with time (Frohn and Roth, 2000).
As a result, water droplet volume decreased about 20 percent of initial volume for 15 minute. It was almost liner change of the squared diameter with time, hence it can be characterized by the d^2-law. The ethanol droplet volume evaporated about 90 percent of initial volume, and it has changed by two stages. The first drying stage can be characterized by the d^2-law. But the evaporation rate of the second stage became small than first stage. The effect of surrounding flow structure of droplets and mass transfer rate will be discussed.



(a) Water droplet. (b) Ethanol droplet.

Figure 1: Average velocity vector fields in the droplets.



(a) Water droplet. (b) Ethanol droplet.

Figure 2: Average velocity vector fields around droplets.

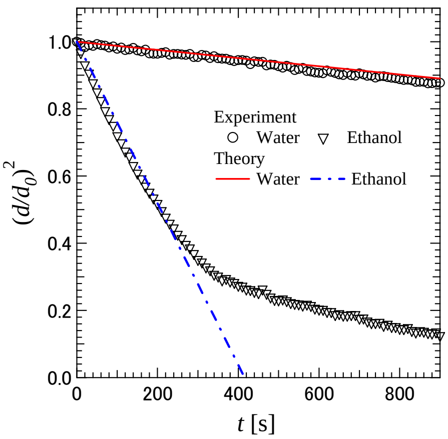


Figure 3: Evaporation process of levitated droplets.

References

A. Frohn and N. Roth, Dynamics of Droplets, pp.29-38, (2000).

R. Tuckermann, et al., Evaporation rates of alkanes and alkanols from acoustically levitated drops, Anal. Bioanal. Chem., Vol. 372, pp.122-127, (2002).

A. Yarin, et al., Evaporation of acoustically levitated droplets, J. Fluid Mech., Vol. 399, pp. 151-204, (1999).

A transient model of a capillary pumped loop for ground applications

Ahmed Kaled [1*], Sébastien Dutour [1], Jean-Louis Joly[1] and Vincent Platel [2]

[1] LAPLACE, Université Paul Sabatier, 118 Route de Narbonne, 31062 Toulouse Cedex 9, France
[2] LaTEP, Université de Pau et des Pays de l'Adour, Quartier Bastillac, 65000 Tarbes, France
[*]E-mail: ahmed.kaled@laplace.univ-tlse.fr

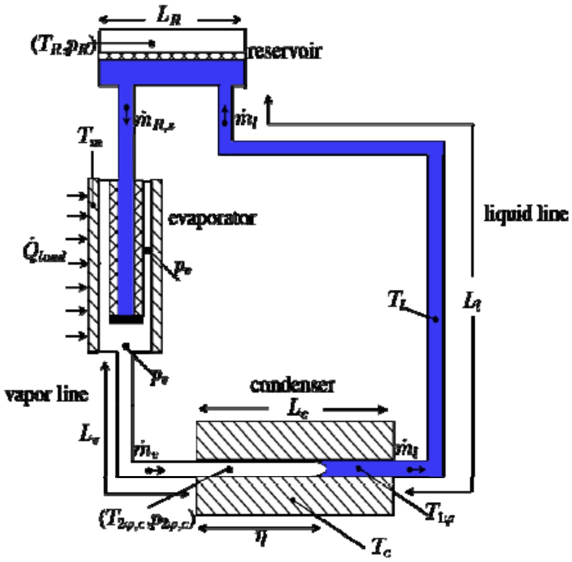
In the field of terrestrial transport (automotive, train, plane), R&D trend is an increasing use of the electric energy at the expense of mechanic, hydraulic and pneumatic energies. However, this development is leading to an increase of the on-board heat flux dissipation so that some more intensive thermal cooling solutions are simultaneously needed.

Used in several spacecraft in the past two decades, two-phase loops with capillary pumping (LHP and CPL) are currently among advanced thermal control technologies for aerospace and ground applications. Because of the efficiency of vaporization and condensation heat transfer and the flexibilty of the fluid transport processes (the heat can be transferred up to several meters at any orientation in the gravity field), passive two-phase closed systems are very attractive for transferring large heat fluxes with small temperature differences. However, these heat transfer devices will encounter specific constraints such as acceleration, vibration and different orientations vs the gravity which may significantly perturb their performances (Gottschlich and *al*.1996).

Figure 1 shows the schematic of a simplified CPL. A CPL is basically composed of the following components: a capillary evaporator (responsible for generating capillary forces that drive the working fluid), a condenser, a two-phase reservoir (to control the loop saturation pressure) and the liquid and vapor lines. The heat load applied to the evaporator is transported to the condenser where it is rejected to the heat sink.

We present in this study a simple transient model of a CPL (CPLIP developed by Euro Heat Pipe) to quantify the system's sensitivity to the inertial effect of the liquid motion in the loop following fast heat load steps. The modeling approach is based on conservation of mass, momentum and energy various subsystems (evaporator, condenser and fluid lines), which are thermally and/or hydrodynamically interconnected. This study is the first step of a general approach concerning the influence of the gravity force on the system performances. The theoretical development of this transient model is essentially based on the previously lab modeling investigations (Launay and *al*. 2007) and (Platel et *al*. 2004). and a recent experimental study (Lachassagne 2010) performed on this device.

The simulation results show that the accelerating or decelerating of the liquid mass in condenser and liquid line amplifies significantly the pressure drop between the two sides of the evaporator. We can even find a critical case in which a faster application of the heat load at the evaporator can theoretically lead the system to defuse.



References

Joseph M. Gottschlich Air Force Wright Lab. Gregg J. Baldassarre, Nelson J. Gernert Thermacore. Inc. Loop heat pipe for avionics thermal control. *SAE*, (961318), May 1996.

L.Lachassagne. Développement expérimental et modélisation numérique d'une boucle diphasique à pompage capillaire en environnement gravitaire : application au refroidissement de composants d'électronique de puissance en contexte automobile. PhD thesis, Ins- tituit P', Decembre 2010.

S.Launay, V.Platel. S.Dutour, Transient modeling of loop heat pipes for the oscillating. behavior study, *Journal of Thermophysics and Heat Transfer*, 21(3) :487–495, JUL-SEP 2007.

V.Platel, J.-Y.Grandpeix, C. Butto, E. Pouzet, J.-L. Joly. Dynamic response of a capillary pumped loop subjected to various heat load transients. *International Journal of Heat and Mass Transfer*, 47(2293–2316), 2004.

Isolated bubbles regime in capillary-cooling systems' condensers: about the thermal equilibrium between liquid and vapor

Georges El Achkar, Marc Miscevic, Pascal Lavieille

LAPLACE, Toulouse University, 118 route de Narbonne, 31062 Toulouse Cedex 09, France
georges.elachkar@laplace.univ-tlse.fr, marc.miscevic@laplace.univ-tlse.fr, pascal.lavieille@laplace.univ-tlse.fr

Most of the available studies in literature on in-tube convective condensation deal with the optimization of condensers for air-conditioning systems (El Hajal et al., 2003). In such applications, mass fluxes in the channel are several hundred of kg.m^{-2}.s^{-1}. Considering applications such as CPLs or LHPs, the order of magnitude of the mass flux is lower, typically of 10 kg.m^{-2}.s^{-1}. For such mass fluxes, capillary effects are preponderant and the flow pattern is very different to that in the condensers of air-conditioning systems. For instance, for mass fluxes between 1 and 20 kg.m^{-2}.s^{-1}, three main flow regimes were observed by Médéric et al. (2004) in a single-tube microcondenser: annular regime, intermittent regime, and bubbles regime.

In this study, heat transfer and flow distribution in an air-cooled multichannel microcondenser were investigated for mass fluxes ranging from 5 to 23 kg.m^{-2}.s^{-1} with n-pentane as the working fluid. This microcondenser consisted of 4 circular micro channels, each of 0.56 and 0.7 mm inner and outer diameters respectively, and of 100 mm length. These channels were placed horizontally. They were parallel to each other in the vertical plane, and were connected to a distributor and a collector. In order to determine accurately the boundary conditions (i.e., the external heat transfer coefficient between the channel wall and the air flow), a technique based on Laser Induced Fluorescence (LIF) was developed.

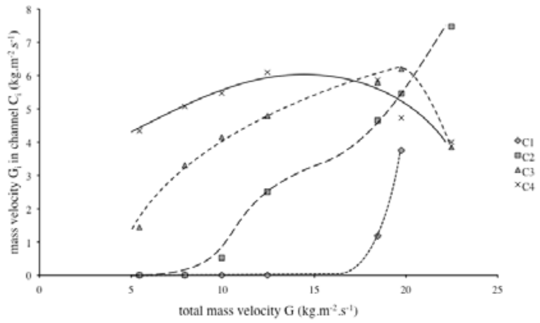


Figure 1: Mass velocity in each of the tubes versus the total mass velocity G imposed in the condenser.

The heat transfers in the annular zone in each channel were then quantified. Results showed that the quality at the end of the annular zone was nearly zero. So, condensation process mainly occurred in this zone despite the existence of a bubbles zone downstream of the annular zone. This result allowed the determination of the two-phase flow distribution in the four channels according to the total mass flux imposed in the condenser (fig. 1) in a non-intrusive manner. The behavior obtained was explained considering the amount of liquid in the collector: the flow distribution depended mainly on the vapor-liquid interface level in the collector, leading to differences in the pressure at the outlet of the channels.

In order to justify that the phase change rate in the bubbles zone is weak, a specific study was carried out on a single-channel microcondenser, consisted of a circular channel having the same dimensions than the channels of the first condenser. Hence, heat transfers were determined in the isolated bubbles zone for mass fluxes allowing the complete condensation of the vapor upstream the end of the channel to be obtained. The quality profile was determined in this zone, and showed a maximum value of 0.3% (fig. 2).

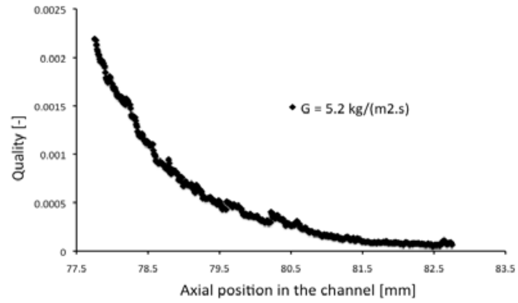


Figure 2: Average quality profile in the isolated bubbles zone for a mass flux G = 5.2 kg.m^{-2}.s^{-1}.

A major conclusion dedicated to designers of two-phase loops is that the fluid cannot be considered at saturation in the bubbles zone. The temperature of the liquid in this zone may reach non-negligible subcooling as shown in figure 3.a. Up to 80 to 90 % of the heat transfers may be exchanged in the form of sensible heat (fig. 3b) in the bubbles zone.

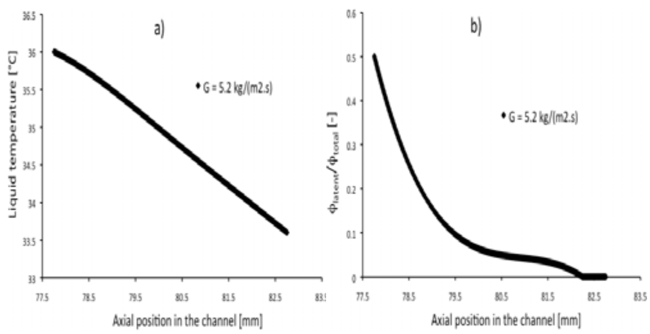


Figure 3: Temperature and (latent heat)/(total heat) profiles in the isolated bubbles zone for G = 5.2 kg.m^{-2}.s^{-1}.

References

J. El Hajal et al., Condensation in horizontal tubes, part 1: two-phase flow pattern map, International Journal of Heat and Mass Transfer vol. 46, pp. 3349-3363 (2003).
B. Médéric et al., Experimental study of flow characteristics during condensation in narrow channels: influence of diameter channel on structure patterns, Superlattices and Microstructures, vol. 35, pp. 573-586 (2004).

Linear-Stability Analysis of Thermocapillary Flow in an Annular Two-layer System with Upper Rigid Wall

Dong-Ming Mo[a], You-Rong Li[a,b], Wan-Yuan Shi[a,b] and Lan Peng[a,b]

[a] College of Power Engineering, Chongqing University, Chongqing 400044, China
[b] Key Laboratory of Low-grade Energy Utilization Technologies and Systems, Ministry of Education, Chongqing 400044, China
E-mail: modongming@126.com

The stability of the thermal convection in a two-layer fluid system is very important in many engineering applications, for example, crystal growth, metal casting, solar panel manufacturing and heat exchangers with air pockets et al. In the past, most theoretical and experimental investigations about Ryleigh-Bénard convection in two immiscible liquids are mainly devoted to vertical heating. Only few contributions are found dealing with a radial temperature gradient. In this paper, we report a set of linear-stability analysis results of thermocapillary flow in an annular two-layer system, which are formed with 5cSt silicone oil and HT-70.

The annular two-layer system is heated from the outer cylindrical wall and cooled at the inner wall, the bottom and top surfaces are bounded by two rigid and heat-insulated walls. The influences of total depths h, depth ratio ε of the lower layer and the upper layer, and radius ratio Γ of the cold inner wall and the hot outer wall of the pool on stability are determined. In the region of $\varepsilon \leq 0.333$, the critical Marangoni number decreases with increase of ε, the most unstable region for ε is around 0.333-0.375. After ε exceeds 0.375, the critical Marangoni number increases. The critical Marangoni number, critical wave number and critical phase velocity decrease with the increase of radius ratio of the cold inner wall and the hot outer wall, as shown in Figs. 1 and 2. The characteristics of flow bifurcation and the surface temperature bifurcation have been found in the region of $0.625 \leq \varepsilon \leq 0.75$, and the typical hydrothermal waves are exhibited. Two different patterns of bifurcation are predicted, they are the first type hydrothermal wave (HTW1) characterized by the curved spokes and the second type hydrothermal wave (HTW2) characterized by the HTW near the inner part of the pool and pairs of counter-rotating longitudinal rolls near the hotter wall, as shown in Fig. 3.

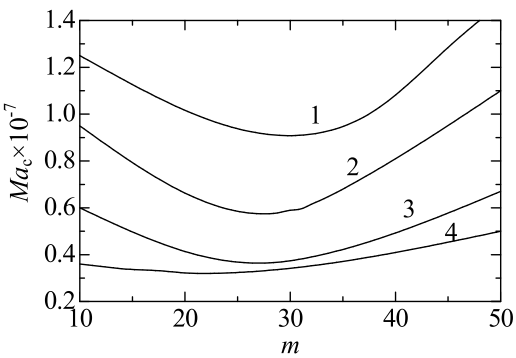


Figure 1: Variation of the Marangoni number at the marginal stability limit with the wave number m at ε=0.5 and h =4mm. 1: Γ=0.2; 2: Γ=0.4; 3: Γ=0.6; 4: Γ=0.8.

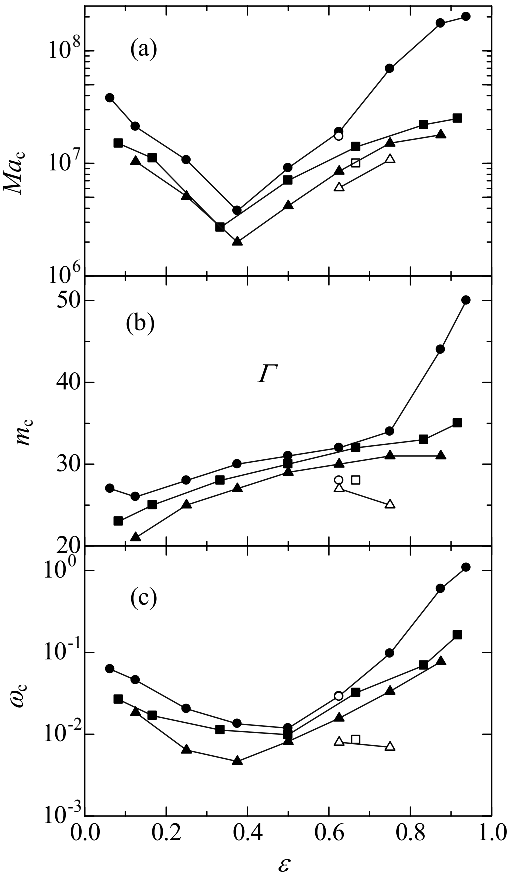


Figure 2: Critical Marangoni number (a), critical wave number (b) and Critical phase velocity (c) for the incipience of the flow instability at Γ=0.2. circle: h=4mm; square: h=6mm; triangle: h=8mm

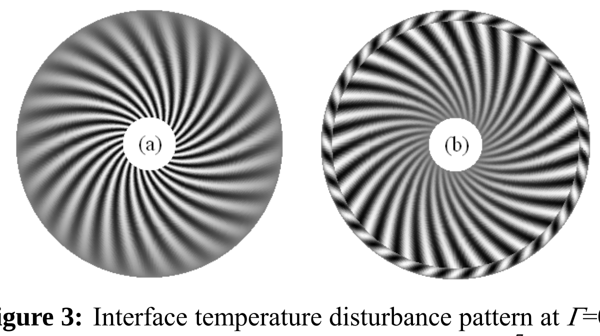


Figure 3: Interface temperature disturbance pattern at Γ=0.2, ε=0.625, h=4mm. (a) HTW1: Ma_1=1.73×10^7, m_1=28, ω_1=0.0288, ϕ_1=26.9°; (b) HTW2: Ma_2=1.90×10^7, m_2=32, ω_2=0.0293, ϕ_2=30.3°

Flow Structure and Surface Deformation of High Prandtl Number Fluid Under Reduced Gravity and Microgravity

Ruquan Liang[1], Guangdong Duan[1], Fusheng Yan[2], Junhong Ji[1], Masahiro Kawaji[3]

[1] Key Laboratory of National Education Ministry for Electromagnetic Processes of Materials, Northeastern University, [2] Research Institute, Northeastern University, No.3-11, Wenhua Road, Shenyang, 110004, China, [3] Department of Chemical Engineering and Applied Chemistry, University of Toronto, Toronto M5S 3E5, Canada
liang@epm.neu.edu.cn

The floating zone method is a widely adopted container-less crystal growth technique, which can avoid contamination of the crucible and is becoming a promising technique to produce high quality single crystals of semiconductors under microgravity from the melt. The half-floating zone liquid bridge model is simplified from the floating zone and is a typical model to numerically study Marangoni convection. In the present paper the numerical simulation has been conducted to investigate the flow structure and surface deformation in a liquid bridge of high Prandtl number fluid under reduced gravity and microgravity. The Navier-Stokes equations coupled with the energy conservation equation are solved on a staggered grid, and the mass conserving level set approach is used to capture the free surface deformation of the liquid bridge.

In order to visually show the deformation of the free surface, we investigate the effects of reduced gravity and thermocapillary convection on the surface deformation, and no horizontal acceleration is exerted on the liquid bridge. Fig. 1 presents the time evolution of the surface deformation for the case of $g_v = 0.6g_0$, where g_0 and g_v are the gravity and reduced gravity exerted on the liquid bridge, respectively. It can be seen that the surface of the liquid bridge deforms under the effects of reduced gravity and the thermocapillary convection induced by temperature difference between the hot and cold disks. The surface deformation is outward near the cold disk and inward near the hot disk, which is consistent with the increase in pressure near the cold disk and decrease in pressure near the hot disk due to the effect of reduced gravity. From the enlargement of the inward surface near the hot disk, the inward surface vibrates horizontally and the amplitude of the horizontal vibration decreases gradually, and the two surfaces at *t*=11.0 and *t*=11.5 are very close after several swings, which shows that the thermocapillary convection inside the liquid bridge starts to turn into a steady state after the initial period.

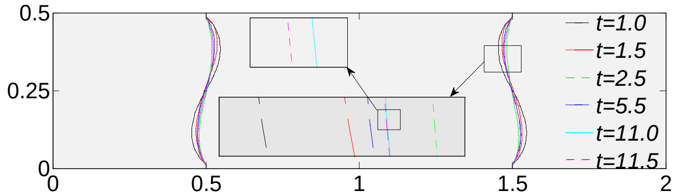


Fig. 1: Time evolution of the surface deformation and the equilibrium position ($\mathrm{Pr} = 48.7$, $\mathrm{Re} = 1072.3$, $We = 0.15$, $Ca = 0.05$, $g_v = 0.6g_0$, $\Delta T = 25°C$)

Time evolution of streamlines in the liquid bridge and ambient air has been studied as well. Under the effect of buoyancy coupled with surface tension gradient, the free surface deforms and a pair of vortices can be found inside the liquid bridge, indicating the existence of returning flow inside liquid bridge.

Surface deformation of the liquid bridge under different gravity levels ($g_v = 1.0g_0$, $g_v = 0.6g_0$, and $g_v = 0.2g_0$) is studied. The shape of free surface changes largely from 1-*g* condition to reduced-*g* condition, and the high gravity level obviously results in the large surface deformation. The cross point of the three shape curves locates nearby the middle height of the liquid bridge.

In addition, we also investigate the effects of acceleration level and frequency on the position of vortex center under horizontal external acceleration and zero gravity conditions. It is necessary to give vibrations to simulate the external accelerations referred to as g-jitter in the computation, and the external vibration patterns are given by the sinusoidal functions. The frequencies adopted are $f = 40Hz$, $f = 50Hz$, $f = 60Hz$, $f = 70Hz$ and $f = 80Hz$ with the same acceleration level of $20mg$ ($1mg = 10^{-3}g$, $g = 9.81\mathrm{m/s}^2$) and the acceleration levels adopted are $40mg$, $60mg$, $80mg$, $100mg$ and $150mg$ with the same frequency of $f = 20Hz$, respectively. Since the flow is driven in the hot corner initially, the center of the recirculating flow is situated near the hot wall. Meantime, another driving force for the flow inside the liquid bridge is the lateral external acceleration. As a result, the recirculating flow pattern generates a relatively large region where the flow moves radially. This radial convection tends to make the bulk fluid temperature distribution rather uniform near the free surface. Consequently, the hot corner becomes less active after initial period, and the vortex centers move toward the cold disk and reach an equilibrium position located about $y = 0.3$ after the initial period, where *y* is the non-dimensional distance away from the cold disk. After that, the vortex centers vibrate around the equilibrium position periodically. It can also be found that it takes less time for the vortex centers to reach equilibrium position in the initial period at the maximal frequency of $80Hz$ and the maximal acceleration level of $150mg$, respectively.

References

Liang, R.Q., Kawaji, M., Surface Oscillation of a Liquid Bridge Induced by Single and Multiple Vibrations, Microgravity Sci. Technol, 21, 31-37 (2009)

Investigation of Thermocapillary Convection of High Prandtl Number Fluid Under Zero Gravity

Ruquan Liang[1], Guangdong Duan[1], Fusheng Yan[2], Junhong Ji[1], Masahiro Kawaji[3]

[1] Key Laboratory of National Education Ministry for Electromagnetic Processes of Materials, Northeastern University, [2] Research Institute, Northeastern University, No.3-11, Wenhua Road, Shenyang, 110004, China, [3] Department of Chemical Engineering and Applied Chemistry, University of Toronto, Toronto M5S 3E5, Canada
liang@epm.neu.edu.cn

It is regarded as one of the most popular means to produce highly homogeneous and purified crystals using floating-zone technique. In this configuration, an intense heat source is passed along a polycrystalline rod so that the material melts and then resolidifies as the heat source passes. Such a process is modeled as a liquid bridge of crystals which is held by surface tension force between two solid disks. During the past years, great amount of numerical and experimental research has been carried out to investigate the mechanism of thermocapillary-driven convection within the liquid bridge column using the half-zone method. However, most of the existing numerical simulations had neglected the effect of the dynamic free surface movement, although the free surface is expected to deform dynamically due to the fluctuation of pressure field.

In the present paper, the flow and thermal fields in a 5cSt silicone oil bridge under zero gravity are investigated numerically considering the free surface deformation and the ambient air by using a mass conserving level set method. The present study aims at understanding the time dependent thermal fluid phenomena with dynamic free surface deformation in the half-zone liquid bridge by a direct nonlinear numerical simulation. The diameter of the top and bottom disks is $D = 5.0mm$, and the aspect ratio (H/R) is 1.0, and The temperature difference between two disks is 30K, and the schematic of the thermocapillary convection model is shown in Fig. 1.

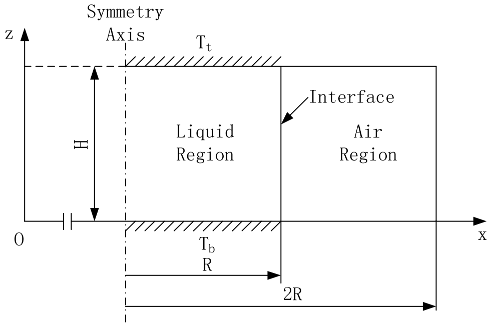


Fig. 1 Schematic of a thermocapillary convection model

The temperature distribution inside the liquid bridge is investigated. The results shows that the surface temperature level increases gradually from *t*=2.5 to *t*=150, showing that hot fluid in the hot corner flows toward the cold disk constantly due to the surface flow and return flow when the thermocapillary convection happens inside the liquid bridge. It can be found that surface temperature level is highest and the axis temperature level is lowest within different radius positions. Because the thermocapillary force mostly acts on the surface, the surface temperature rises fast. The heat transfer inside the liquid bridge is carried out from hot disk toward cold disk accompanying from free surface toward symmetry axis of the liquid bridge. The temperature gradient is relatively high nearby the hot disk. For the axis, the temperature gradient is high nearby the hot disk, while the temperature gradient is high nearby the cold disk for the free surface, which demonstrates that there are two higher temperature gradient regions inside the liquid bridge. In addition, four monitoring points on the surface are set to investigate the time evolution of temperature at different surface heights of the liquid bridge. Since the flow is driven in the hot corner, the vortex center is situated near the hot corner initially. The recirculating flow generates a radial convection, which tends to make the bulk fluid temperature distribution rather uniform near the free surface. Therefore, the temperature gradient in the hot corner decreases, while the surface temperature gradient in the cold corner increases. Consequently, the hot corner becomes less active and the cold corner becomes more active, which makes much of the surface flow originating from the hot corner go into the cold corner.

Finally, the velocity distributions on the free surface in axial and radial directions are studied as well. Fig. 2 presents the velocity profiles on the free surface in axial direction. The surface velocities in axial direction are almost negative, which indicates that the direction of surface flow is from the hot disk to the cold disk. The maximum surface velocities in axial direction are obtained at the position about z=0.6, where z is the non-dimensional distance away from the cold disk.

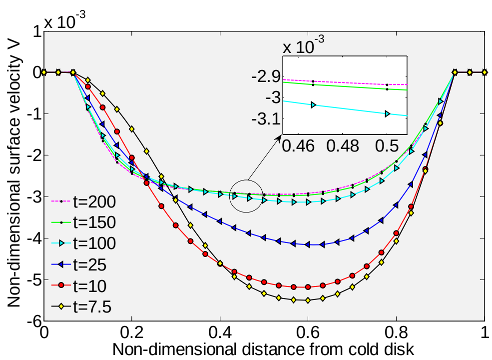


Fig. 2 Surface velocity distribution in axial direction ($\Pr = 48.7$, $Ma = 19700$, $We = 0.150$, $Ca = 0.0005$, $D = 5.0mm$, $H/R = 1.0$)

References

Liang, R.Q., Chen, Z., Dynamics for Droplets in Normal Gravity and Microgravity, Microgravity Sci. Technol., 21, 247-254 (2009)

Decoupling the thermocapillary and gravitational effects in evaporative convection: a numerical study

Carlo S. Iorio[1], Olga N. Goncharova[2,3], Oleg A. Kabov[1,3]

[1] Universite Libre de Bruxelles, Chimie-Physique EP-CP165/62, Microgravity Research Center, Av. Roosevelt 50, Bruxelles, Belgium
ciorio@ulb.ac.be, okabov@ulb.ac.be
[2] Altai State University, Faculty of Mathematics, Department of Differential Equations, Pr. Lenina 61, Barnaul, 656049, Russia
olga.n.goncharova@gmail.com
[3] Institute of Thermophysics, Russian Academy of Sciences, prosp. Lavrentyev 1, Novosibirsk, 630090 Russia

Many industrial processes involve a step in which a liquid is evaporated in presence of a current of inert gas. The effectiveness of such processes depends on the kind of thermo-fluid-dynamic instabilities that arise from the interaction of the evaporation process with the surface-tension-driven flow generated at the gas-liquid interface and with the buoyancy-driven effects. These latter are both induced by the thermal gradient generated at the interface by the endothermic character of the evaporation process. Although many aspects have been separately clarified a complete theoretical and numerical model for such kind of interactions is still lacking [1-5].

In this work, the numerical analysis of a model problem is presented. A pure, volatile liquid (ethanol) occupying a small liquid pool is evaporated in presence of an inert gas flow in a gravitational field. Aside from the coupled problem, an evaluation of the different kind of effects is given. At first, the effects of the shear stress of the gas current on the interface are calculated, while the evaporation, surface-tension driven and gravitational effects are neglected. Then, the evaporation is added in the model, while still neglecting the surface tension driven and gravitational effects. Later, the gravitation and surface tension effects are added separately and analyzed. Finally, the complete problem is studied for a fixed flow rate of inert gas and geometry of the evaporating pool and for a given volatile liquid.

Computations show that for the geometry studied - gas channels 5mm thick, liquid pool of 1.7mm depth - the contribution of the gravitational forces to the formation of the convection patterns is smaller respect to the one of the thermo-capillary forces. With reference to the Figure 1, in **a)** no convection patterns are distinguishable and the maximum temperature difference at the interface is of 7 K. When only gravity is added – Figure 1b – convection cells appear at the attack border of the pool respect to the direction of the flow. The temperature difference at the interface accounts for 6.8 K. In Figure 1c-d, the results for the coupled model evaporation-thermo-capillarity are shown. The convection patterns are clearly visible and the temperature difference at the interface is of 3.1 K and 2.8 K, respectively.

As expected from theoretical analysis, the evaporation rate as well as the heat fluxes at the interface is strongly dependent on the coupling effects of the evaporation and the surface-driven-induced flow. Buoyancy convection becomes an important factor only for thick liquid layers and in regions where the thermal gradient induced by evaporation is highly significant, such as the attack border of the evaporating liquid.

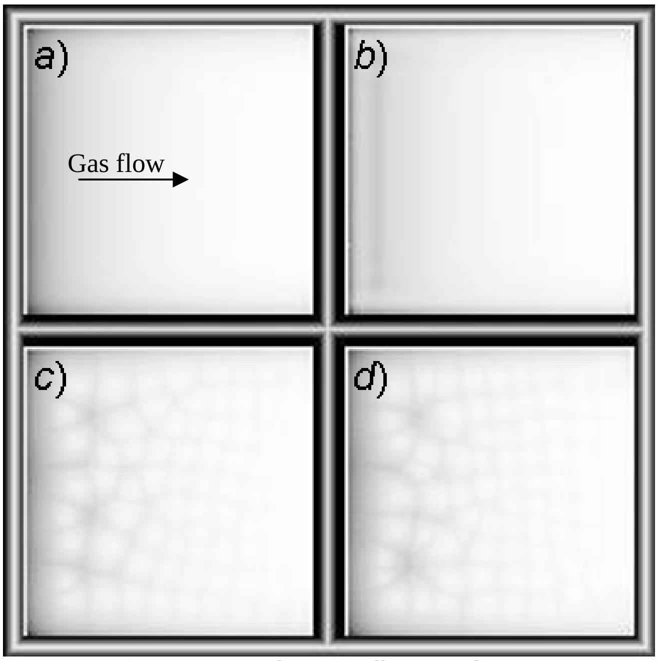


Figure 1 a) Gravity: No – Thermo-capillarity: No; **b)** Gravity: Yes – Thermo-capillarity: No; **c)** Gravity: No – Thermo-capillarity: Yes; **d)** Gravity: Yes – Thermo-capillarity: Yes

Acknowledgments: This study is performed in the frame of the ESA-sponsored program CIMEX-1, that is mainly targeted on the understanding of the convective instabilities arising during mass exchange processes.

References

[1]. Haut B., Colinet P., Surface-tension-driven instability of a liquid layer evaporating into an inert gas, *J. of Colloid and Interface Science*, 285, pp. 296-305 (2005)
[2]. Olga N. Goncharova, Oleg A. Kabov, "Numerical Modeling of the Tangential Stress Effects on Convective Fluid Flows in an Open Cavity", *Microgravity Sci. Technol* , 21, S119–S127 (2009).
[3]. Iorio C.S., Goncharova O.N., Kabov O.A.; Study of evaporative convection in an open cavity under shear stress flow, *Microgravity sci. technol.*, Vol. 21(1), P. 313-319 (2009).
[4]. Olga Goncharova, Oleg Kabov, "Mathematical and numerical modeling of convection in a horizontal layer under co-current gas flow", *International Journal of Heat and Mass Transfer*, Vol. 53, Iss. 13-14, Pages 2795-2807 (2010).
[5]. Iorio C.S., Goncharova O.N., Kabov O.A., Heat and mass transfer control by evaporative thermal pattering of thin liquid layers, *Submitted to Computational Thermal Sci.* (2011).

The Influence of the Horizontal Component of the Temperature Gradient on Nonlinear Oscillatory Convective Regimes in Multilayer System

Ilya Simanovskii (1), Antonio Viviani (2), Frank Dubois (3), Jean-Claude Legros (3)

(1) Department of Mathematics, Technion - Israel Institute of Technology, 32000 Haifa, Israel
yuri11@inter.net.il
(2) Seconda Universita di Napoli (SUN) Dipartimento di Ingegneria Aerospaziale e Meccanica (DIAM) via Roma 29, 81031 Aversa, Italy
(3) Universite Libre de Bruxelles, Service de Chimie Physique EP, CP165-62, 50 Av. F. D. Roosevelt 1050, Brussels, Belgium

Interfacial convection in multilayer fluid systems have been a subject of an extensive investigation at the past few decades (for a review, see Simanovskii and Nepomnyashchy (1993); Nepomnyashchy, Simanovskii and Legros (2006)).

There are two basic physical phenomena that produce convective instability in systems with an interface: buoyancy and thermocapillary effect. When heating is from below, the buoyancy instability generates the Rayleigh – Benard convection, while the thermocapillary effect is the origin of the Marangoni – Benard convection.

One of the interesting phenomena caused by the joint action of buoyancy and thermocapillary effect is the appearance of the oscillatory instability of the mechanical equilibrium by heating from below. This phenomenon was first discovered in the case of a two-layer system by Nepomnyashchy and Simanovskii (1984); Juel et al. (2000). A similar phenomenon under the joint action of both mechanisms of instability in the multilayer system, has been studied for free and rigid heat-insulated lateral walls (Nepomnyashchy et al. 2005; Simanovskii et al. 2011) as well as in the case of periodic boundary conditions on lateral boundaries (Simanovskii et al. 2009).

In reality, it is difficult to guarantee that the temperature gradient is directed strictly perpendicularly to the interfaces. Under experimental conditions, the temperature gradient is not perfectly vertical and the horizontal component of the temperature gradient appears. The appearance of this component changes the situation significantly: at any small values of the Marangoni number, the mechanical equilibrium becomes impossible, and a convective flow takes place in the system. Thus, it is reasonable to consider the influence of the horizontal component of the temperature gradient on convective regimes developed in the system.

In the present work, the influence of the horizontal component of the temperature gradient on nonlinear oscillatory convective regimes, developed under the joint action of buoyant and thermocapillary effects in system air - ethylene glycol - fluorinert FC75, is studied. This system is appropriate for Earth experiments because of the relatively low viscosity of the fluorinert. Scientific interest in this system is owing to the fact that it is subject to different kinds of instabilities. The nonlinear convective regimes are studied by the finite difference method. The calculations have been performed for two-dimensional flows. Transitions between different flow regimes have been investigated.

It is shown that under the action of the horizontal component of the temperature gradient, the asymmetric oscillatory flow takes place in the system. In comparison with the symmetric oscillatory flow, the vortices for the asymmetric oscillations have the tendency to become longer. At the definite interval of the paremeters, the phase trajectories of the asymmetric oscillations have the multi-loop character. With the increase of the Grashof number, the oscillatory flow becomes unstable and the steady asymmetric state is developed in the system. It is found that the region of nonlinear asymmetric oscillations is restricted by the Grashof number values both, from below and from above, by the regions of convecting steady states.

References

Juel A., Burgess J. M., McCormick W. D., Swift J. B. and Swinney H. L., Surface-tension-driven convection patterns in two liquid layers, Physica D, Vol. 143 pp. 169-186 (2000)

Nepomnyashchy A. A., Simanovskii I. B., Thermocapillary and thermogravitational convection in a two-layer system with a distorted interface, Fluid Dyn., Vol. 19 pp. 494-499 (1984)

Nepomnyashchy A. A., Simanovskii I. B. and Braverman L. M., Convective oscillations in multilayer systems under the combined action of buoyancy and thermocapillary effect, Phys. Fluids, Vol. 17 pp. 022103 (1-9) (2005)

Nepomnyashchy A. A., Simanovskii I. B. and Legros J. C., Interfacial Convection in Multilayer Systems, Springer, New York (2006)

Simanovskii I. B., Nepomnyashchy A. A., Convective Instabilities in Systems with Interface, Gordon and Breach, London (1993)

Simanovskii I. B., Viviani A., Dubois F. and Legros J.C., Nonlinear waves in a multilayer system, Comptes Rendus Mecanique , Vol. 337 pp. 75-83 (2009)

Simanovskii I. B., Viviani A., Dubois F., Legros J.C., Nonlinear development of oscillatory instability in a three-layer system under the joint action of buoyancy and thermocapillary effect, Phys. Fluids, Vol. 23 pp. 012103 (1-12) (2011)

Evaporation of the liquid from the moist particle

Vasily Salomatov, Vladimir Salomatov

Institute of Thermophysics, Siberian Branch of Russian Academy of Sciences,
Lavrentev ave. 1, 630090, Novosibirsk, Russia
E-mail: vvs@itp.nsc.ru

We consider moisture evaporation out of a single particle that is subject to simultaneous thermal influence through radiation and convection. The process is preceded with a particle heating stage that is at first approximation assumed to last until the fluid temperature becomes equal to the saturation temperature. Actually the saturation temperature is exceeded by 10-15 °C due to higher vapor pressure during evaporation of moisture within the particle. We are seeking approximate analytical relations describing the overall process to allow for detailed parametric analysis and implementation of rapid calculations. Procedures for constructing approximate solutions for heating and evaporation stages are demonstrated below with subsequent analysis of the results.

Heating stage. Warming up is assumed under the influence of a symmetric heat flux supplied to the surface of a spherical particle. We adopt the following key assumptions: the particle is a homogeneous two-phase system "liquid-solid" (dispersed phase dimensions are of micrometer range); effective thermal properties are constant; fluid motion is neglected; particle is non-isothermal; mass transfer is not considered. With the above assumptions mathematical formulation includes the heat equation with complex nonlinear boundary conditions of radiative-convective type. Due to the fact that heating stage solution serves as an input for the second stage of evaporation, the approximate solution for the first stage must satisfy certain requirements: like it cannot contain infinite series expressed in explicit form, etc. We construct asymptotic expansions for the temperature function at the limiting time values using Laplace operator method. We obtain the starting time for "liquid-vapor" phase transition and temperature distribution inside the particle at this moment. Calculated dependence of the dimensionless temperature Θ versus dimensionless time Fo in the case of neglected temperature drop in the particle (Biot number < 0.5, Stark number < 0.25) has the simplest form:

$$3SkFo = f[\bar{\Theta}(Fo)] - f[\Theta_0] \qquad (1)$$

where f is expressed analytically. Table. 1 shows the calculated heating time for particles of different diameter with an initial moisture of 60%. The data is in satisfactory agreement with experiments of Z. Huang (1986) [1] for particles of 1000 microns.

Table 1. Heating time for particles of different size with residual moisture at 900 °C.

d, мкм	200	500	1000	1500	2000
t^{heat}, sec; calculation	0.252	0.63	1.26	1.88	2.52
t^{heat}, sec; Huang (1986)	–	–	1.31	–	–

Evaporation stage. The evaporation stage is described by a nonlinear Stefan problem. To obtain approximate analytical solution we use a provision under which the dynamics of a phase transformation front is controlled by the temperature distribution at the limiting time. In our case that is a temperature distribution at large Fourier numbers. Particle evaporation process is represented as follows: After reaching saturation temperature at the surface, moving evaporation boundary divides the particle into two different zones. The outer zone contains no moisture and it is a stable dispersed phase conglomerate. Inner zone is a region with initial moisture level. Heat flux from the external environment is consumed by outer zone heating, inner zone warming and by phase transition on a moving "liquid-vapor" boundary. In addition to the basic assumptions of the heating stage we adopt the following assumptions for the evaporation stage: heat consumption for evaporation occurs at the phase transformation boundary; phase transition temperature and heat of vaporization is constant; particle keeps its shape after evaporation; the moisture diffusion is neglected. Using asymptotic procedures, which are again obtained with integral Laplace transformation, we derive evaporation boundary motion law. Calculated expression for evaporation dynamics when all the moisture in the second zone has a saturation temperature becomes

$$\tilde{m}Fo_1 = \frac{1}{2}\left(1 - R_{\grave{e}}^2\right) + \frac{1}{3}\left(1/Bi - 1\right)\left(1 - R_{\grave{e}}^3\right) + \frac{\tilde{m}}{10}\left[1 - 5R_{\grave{e}}^2\left(1 - \frac{6}{5}R_{\grave{e}}\frac{1 - R_{\grave{e}}^2}{1 - R_{\grave{e}}^3}\right)\right] \qquad (2)$$

Results. We have obtained approximate analytical formulas for the process of evaporation of the liquid from the moist particle with certain assumptions. At $\tilde{m} << 1$ formulae (2) exactly matches the one of L.S. Leybenzon (1955) [2] that was obtained ignoring the heat capacity in both zones considered. Dimensionless time of complete particle dewatering is linearly dependent on reverse Biot number (1/Bi) at different values of evaporation $\tilde{m}$.

References

[1] Huang Z., Theoretical analysis on CWM drop combustion history, USA, Orlando, (1986) – 358 pp.

[2] Leybenzon L.S., Collection of scientific works, vol. IV., USSR, Moscow, Nauka, (1955) – 360 pp.

Investigation of the impact and spreading of drops on solid surface for Ground and Space conditions

Alexey Fedyushkin, Alexey Rozhkov

Institute for Problems in Mechanics of Russian Academy of Sciences,
101/1, Prospect Vernadskogo, Moscow 117526, Russia
E-mails fai@ipmnet.ru , rozhkov@ipmnet.ru

The given work is devoted to numerical investigations of the impact and spreading of water drops on solid surface. Ground and weightless environment conditions are considered for different contact angles on solid surface. Mathematical simulation is performed on the numerical solutions of unsteady Navier-Stokes equations for flows of incompressible two-phase (air-water) systems and phase fraction equation. Dynamics of deformation of liquid volume was taken in account by using of VOF (Volume Of Fluid) taking into account of balance of superficial forces on interface with Euler method and iterations on each time step.

The modeling was carried out for various properties, contact angle diameters d_i, impact velocities v_i of drops for conditions of weightlessness and ground in a range of numbers Weber We =50-800 and Reynolds Re=70-10^3, accordingly.

.

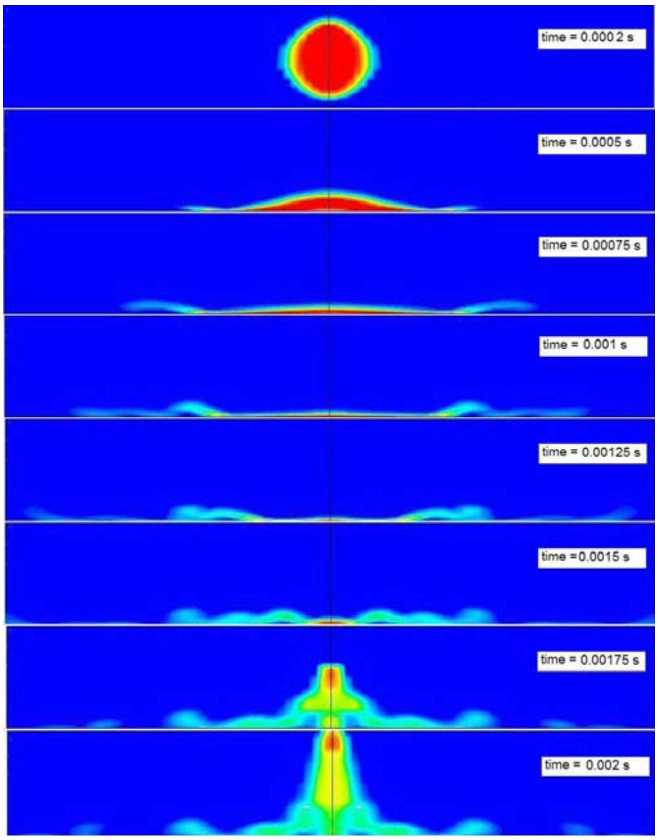


Figure 1: Distributions of water fraction during collision and spreading of water drop (d_i=1mm, v_i=3.83m/sec) on solid surface for different time moments (from t=2 10^{-4} to t=2 10^{-3} s).

The results of numerical simulation of formation of circular liquid lamella resulting from impact of water drop (diameter d_i=1mm and impact velocity v_i=3.83m/s) on solid surface (with contact angle 180 degree) are displayed in figure 1. These structure, speed, maximal lamella diameter d_m and the times of formation of liquid lamella were well compared to experimental data (Rozhkov et al. (2004), (2002)),

The comparison of results of numerical modeling with experimental data (Rozhkov et al. (2004)) for spread factor $\beta_m = d_m/d_i$ of drops on solid surface is given in figure 2. The comparison of experimental data, approximation by expression $(We/20)^{1/2}$ and the numerical results is showed a good agreement for range of Weber number from 50 to 700.

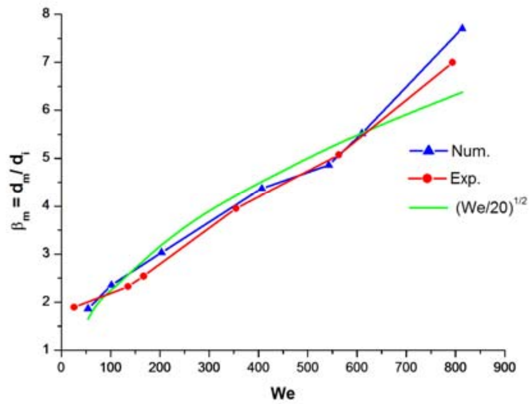


Figure 2: Weber number (We) dependences of maximum dimensionless diameter ($\beta_m = d_m/d_i$) of spreading of drops on solid surface (blue line is numerical simulation, red line is experiment by Rozhkov et al. (2004), green line is approximation by expression $(We/20)^{1/2}$)

References

Rozhkov A., Prunet-Foch B., Vignes-Adler M. Impact of water drops on small targets. Physics of Fluids, Vol. 14, N. 10 , pp. 3485-3504, (2002)

Rozhkov A., Prunet-Foch B., Vignes-Adler M. Dynamics of a liquid lamella resulting from the impact of awater drop on small target. Proc. Roy. Soc. London . A. Vol. 460. N. 2049, pp. 2681-2704, (2004)

VOF-based numerical simulation of thermocapillary flows using a 2-scalar approach for heat transfer

Chen Ma, Dieter Bothe

Technische Universität Darmstadt
Center of Smart Interfaces
Mathematical Modeling and Analysis
Petersenstrasse 32, 64287, Darmstadt, Germany
ma@csi.tu-darmstadt.de
bothe@csi.tu-darmstadt.de

This contribution concerns 3D Direct Numerical Simulation of two-phase flow driven by the thermal Marangoni effect. Such kind of flow plays a significant role in two-phase heat and mass transfer applications, e.g. in cooling technology, drying of polymer films etc. The mathematical model is based on continuum mechanics where the two-phase Navier-Stokes equations together with the energy balance in temperature form for incompressible, immiscible fluids are solved. For the numerical simulation we apply a finite volume discretization and capture the fluid interface using an extended VOF (volume of fluid) method with additional interface reconstruction (Piecewise Linear Interface Construction). Since the thermocapillarity is the driving force and depends highly sensitively on the interfacial heat transfer, the main challenge in this work is achieve high accuracy in the numerical calculation of the two-phase heat transfer with special respect to the interface temperature which plays a central role for thermocapillary flows. A novel 2-scalar approach is therefore developed to solve the two-phase energy equation. The heat transfer for the two phases is treated separately and, as a consequence, at the interface an accurate interface temperature can be calculated based on the energy transmission condition. Ghost values are used as extension of the phases beyond the interface, so that the temperature equation can be discretized in a consistent way for 3D flows with arbitrary position and orientation of the fluid interface. This new approach is validated within a 3D simulation of a 1D two-phase heat conduction and shows excellent agreement with analytical solution with respect to both transient behavior and the interface temperature.

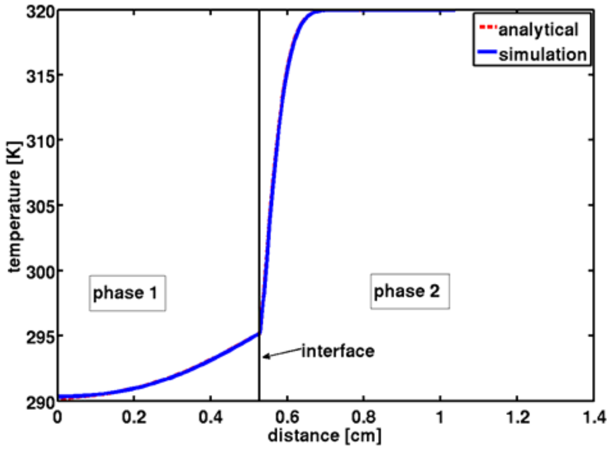


Figure 1. Temperature profile in two-phase heat conduction

Figure 1 shows a snapshot of the temperature profile during heat conduction in two phases with different heat conductivities. The initial temperature of phase 1 and phase 2 are 290 and 320 K, respectively. The jump of the temperature gradient normal to the interface is captured precisely using a sub-grid computation with second order polynomials. The two curves in Figure 1 completely coincide.

Thanks to the sub-grid model this method can be adopted into the VOF method and applied to complex 3D flows with dynamically deformable fluid interface. Also the incorporation of evaporation with accurate interfacial thermal condition is possible. Some complex thermocapillary flows like the Bénard-Marangoni instability etc. are studied using the 2-scalar method and the results are compared with experimental results.

Further relevant applications can be studied such as the heat transfer and rupture condition of a locally heated thin liquid film, or heat transfer enhancement in liquid film on structured substrates.

This work is supported by the "Excellence Initiative" of the German Federal and State Governments, the Graduate School of Computational Engineering and the Center of Smart Interfaces at Technische Universität Darmstadt.

Capillary Channel Flows in Microgravity

Peter Canfield, Michael Dreyer, Aleksander Grah and Joerg Klatte

University of Bremen, Production Engineering, Center for Applied Space Technology
Am Fallturm, Bremen, 28359, Germany
peter.canfield@zarm.uni-bremen.de

Mark Weislogel and Yongkang Chen

Portland State University, Maseeh College of Engineering and Computer Science
PO Box 751, Portland OR 97207, USA

In a reduced gravity environment such as Space, the decreased importance of gravity as a body force on liquid flows means that other forces must be utilized to efficiently transport and position fluids. Channels which rely on capillary forces in induce flow, ergo capillary channels, have been and are still being used in space technology. While they are most prominently used in propellant management devices, the possible areas of application include any systems that require fluids to be positioned or transported from one point to another. A capillary channel provides one or more free surfaces, which may become unstable and even collapse under certain conditions drawing gas into the liquid volume in the process. Generally, the generation of a two phase flow in this manner is undesired and in certain applications must be avoided as the inclusion of gas into the liquid flow may lead to technical disadvantages. The conditions that lead to the instability of the free surfaces must be determined so that capillary channels can be designed to avoid ingestion of gas into the flowing liquid.

A mathematical model has been developed to describe steady capillary channel flows with which the instability and, ultimately, the collapse of the free surfaces can be predicted. The model presents a Speed Index, which can be described as a capillary Mach number. The Speed Index helps to predict highest attainable flow rates in capillary channels before the free surfaces collapse. The validity of this model has been determined via numerous experiments in drop tower tests and sounding rocket flights. Additionally, extensive numerical studies have been performed to further the understanding of the flow instability. Most recently, the mathematical model has been refined to include unsteady flows and also the effect of the channel geometry has been examined. Currently, an extensive experimental study is being conducted on the ISS. The experiments are being operated from the control station at ZARM in Bremen, Germany. The experiments are planned to cover different channel geometries. The tests will include steady and unsteady flow regimes; the effects of oscillations of the free surface on their stability will also be investigated. In addition, bubbly flow will be studied. The gathered data promises to not only validate the current model, but also to extend it to cover additional flow regimes.

This talk focuses on open capillary channel flows in the regime of steady flow. The mathematical model, numerical studies, and past and current experiments will be presented.

References

Salim et al.: Experimental Investigation of a Bubbly Two-Phase Flow in an Open Capillary Channel under Microgravity Conditions, Microgravity Sci. Technol. 22, 87-96 (2010)

Grah, A. and Dreyer, M. E.: Dynamic stability analysis for capillary channel flow: One-dimensional and three-dimensional computations and the equivalent steady state technique, Phys. Fluids 22, 1-11, (2010)

Haake et al.: Flow Rate Limitation of Steady Convective Dominated Open Capillary Channel Flows Through a Groove, Microgravity Sci. Technol., 22, 129-138 (2010)

Klatte et al.: A fast numerical procedure for steady capillary flow in open Channels, Acta Mech. 201, 269-276, (2008)

Grah et al.: Stability limits of unsteady open capillary channel flow, J. Fluid Mech. 600, 271-289, (2008)

Rosendahl et al.: Choked flows in open capillary channels: theory, experiment and computations, J. Fluid Mech. 518, 187- 214 (2004)

Rupture of a liquid film on horizontal and inclined plates under local heating

Dmitry Zaitsev[1,2,3], Andrey Semenov[1,3], Yuriy Luilin[2,3], Oleg Kabov[1,2,3,4] ,
Yulia Kabova[3,4], Chen Ma[4,5] and Dieter Bothe[4]

[1] Institute of Thermophysics SB RAS (IT SB RAS), 1, Lavrentiev Ave, 630090, Novosibirsk, Russia. e-mail: zaitsev@itp.nsc.ru
[2] Microgravity Research Centre, Universite Libre de Bruxelles (ULB), CP 165/62, Avenue F.D. Roosevelt, 50, B-1050 Brussels, Belgium
[3] Heat Transfer International Research Institute of ULB and IT SB RAS, CP 165/62, Avenue F.D. Roosevelt, 50, B-1050 Brussels, Belgium
[4] Center of Smart Interfaces of Technische Universität Darmstadt, Petersenstr., 32, 64287 Darmstadt, Germany
[5] Graduate School Computational Engineering, Technische Universität Darmstadt, Dolivostr. 15, D-64293, Germany

The disruption of liquid films on a heated wall is of major technical importance since it determines the allowed heat transfer rate in many engineering applications. Motivated by potential application of liquid films in cooling of microelectronics and based on previous studies [1], we try in the present work to connect rupture of liquid films flowing over inclined plates with rupture of liquid layers resting on a horizontal surface.

The experimental setup is a closed-type flow loop. The main part of the test section is a thin stainless steel plate with a flush-mounted copper rod. At the working surface the rod has a 1×1 cm square head. The rod is heated by a heating spiral coiled around its bottom part. We use 3 configurations of the flow: 1) Liquid film flowing under the action of gravity over the plate inclined at the angle of 5.9 deg with respect to the horizon. The range of Reynolds number is Re=Q/ν=6.5–65 (Q is specific liquid volumetric flow rate, m^3/s; ν is liquid kinematics viscosity, m^2/s). 2) Liquid film flowing over the horizontal plate. In this case film flow occurs under the action of inertia and hydrostatic pressure. Re=2.8–8.5. 3) Liquid layer resting on the horizontal plate. The range of the initial film thickness is h=0.4–2.3 mm.

Distilled water with initial temperature of 22°C is used as the working liquid. The experiments are carried out at atmospheric pressure under quasi-stationary conditions. The experimental procedure is as follows: at given configuration of the flow and thickness of the film, the heat flux q is increased at increments of about 5% from 0 to the threshold heat flux, q_{idp}, at which a dry patch appears on the heater. For visualization of the film flow and rupture, the Phase Shift Schlieren technique was used.

Prior to film rupture appreciable thermocapillary deformations of the film surface appear, growing with the heat flux, Fig. 1a. Upon reaching a threshold heat flux the film rupture occurs, Fig. 1b. Figure 2 shows the dependence of the threshold heat flux, q_{idp}, on the initial film thickness, h, for 3 flow configurations. At a given h, q_{idp} for configuration 1 is much higher than that for configurations 2 and 3. Also, it is worth noting that curves for configurations 2 and 3 intersect. This is probably due to the fact that slow flow of liquid in configuration 2 prevents formation of convective cells responsible for high heat transfer rates in configuration 3. However, with the increase of the liquid flow rate the convective heat transfer into the film increases, and the threshold heat flux grows.

We study this phenomenon also theoretically by means of DNS employing continuum mechanical sharp interface models. The two-phase Navier-Stokes equations for incompressible flows together with the energy equation for

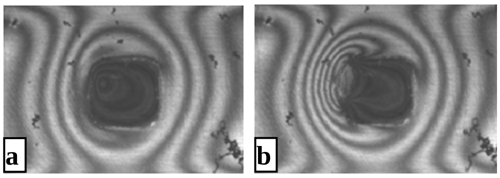


Figure 1: Film rupture in flow configuration 2, Re=2.8. a) q=4.8 W/cm^2; b) q=5.0 W/cm^2 (rupture). Flow directed from right to left.

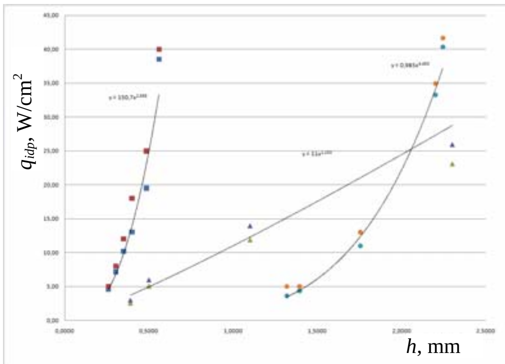


Figure 2: Dependence of the threshold heat flux on the film thickness for different flow configurations: squares - conf. 1; circles - conf. 2; triangles - conf. 3.

both liquid and gas phase are solved within a finite volume scheme. The dynamically deformable interface is captured using an extended Volume of Fluid (VOF) method with Piecewise Linear Interface Construction (PLIC). The thermocapillary force at the interface is calculated using an appropriately computed interface temperature field resulting from the interfacial energy transmission condition. Fig. 3 shows an amplified view of the resulting interface deformation of configuration 3.

Figure 3: surface deflection in the heater region of a water film resting on a horizontal plane

Acknowledgements: The authors gratefully acknowledge support of this work by the European Space Agency, HEAT TRANSFER PRODEX Project (Belgium), and Marie Curie Grant PIIF2-GA-2009-237869 as well as the 'Excellence Initiative' of the German Federal and State Governments and the Graduate School Computational Engineering at Technische Universität Darmstadt.

References

[1] Zaitsev D.V., Kabov O.A., "An Experimental Modeling of Gravity Effect on Rupture of a Locally Heated Liquid Film", Microgravity sci. technol.,XIX-3/4,pp.174-177, 2007.

The Effect of Gravity on the Dynamics of a Sessile Drop

D.Zaitsev[1,2,3], L.Valdarno[2,4], A.Semenov[1,2,3], I.Marchuk[1,3], S.Chikov[1,2,3],
P.Queeckers[2], E.Bykovskaya[2,3], L.Araneo[4], O.Kabov[1,2,3]

[1] Institute of Thermophysics SB RAS (IT SB RAS), 1, Lavrentiev Ave, 630090, Novosibirsk, Russia
[2] Microgravity Research Centre, Université Libre de Bruxelles (ULB), CP 165/62, Avenue F.D. Roosevelt, 50, B-1050 Brussels, Belgium
[3] Heat Transfer International Research Institute of ULB and IT SB RAS, CP 165/62, Avenue F.D. Roosevelt, 50, B-1050 Brussels, Belgium
[4] Politecnico di Milano, Dipartimento di Energia, via Lambruschini 4, 20156 Milano, Italy

Liquid drop is a very common phenomenon occurring in various natural and industrial processes, e.g. thermal management, combustion systems, etc. Although there is a lot of investigations on sessile drops in the literature, there is only a few performed under different gravity conditions.

In this work we present experimental results on sessile drops obtained under normal gravity (1g), microgravity (μg) and hypergravity (up to 20g). The microgravity experiment was conducted during the 52nd and 53rd ESA Parabolic Flight Campaigns in Bordeaux, France, Fig. 1a. The hypergravity experiment was carried out in the ESA Large Diameter Centrifuge in Noordwijk, Netherlands, Fig. 1b.

(a) (b)

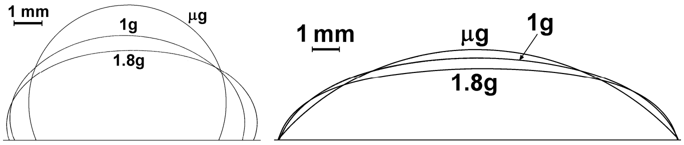
Fig. 1. (a) - setup for parabolic flight experiments (52-53 ESA Campaigns); (b) - ESA Large Diameter Centrifuge.

The goal of the experiment is to study the effect of the gravity 1) on the shape of a static sessile drop; and 2) on the dynamic advancing contact angle in a growing sessile drop.

Eleven different smooth and rough surfaces are used, with different contact angles (CA) and different contact angle hysteresis (CAH). Water is used as the working liquid. The main variable parameters are: gravity (μg–20g); drop volume (1μl–5ml); liquid flow rate (0.0625–16 ml/min); CA (30–130°). The drop shape is visualized from the top with the help of the Phase Schlieren System, and from the side with the help of the shadow technique with resolution of 6 μm/pix. In terrestrial conditions the DSA10 Contact Angle Measuring System by KRUSS is also used.

Numerical algorithms for a solution of the Young-Laplace equation describing the shape of sessile drops are developed and tested for different types of initial conditions. A set of direct and inverse problems is solved on determination of the capillary length and CA from the measured values of drop dimensions: height, diameter, diameter of the wetted spot, volume.

For the first time the spreading of a sessile drop under the effect of gravity has been experimentally observed on surfaces with low CAH, Fig. 2a. For surfaces with high CAH the contact line is pinned while CA adjusts for different gravity levels, Fig. 2b. Good agreement is obtained between the modeling and experiment, Fig. 3. The dynamic advancing CA is found to increase with the gravity, Fig. 4.



(a) (b)

Fig. 2. Gravity effect on drop shape (experimental data). (a) - 0.146 ml water drop on Teflon (CAH=12.6°); (b) - 0.313 ml water drop on Polyvinylacetate (CAH=34.5°).

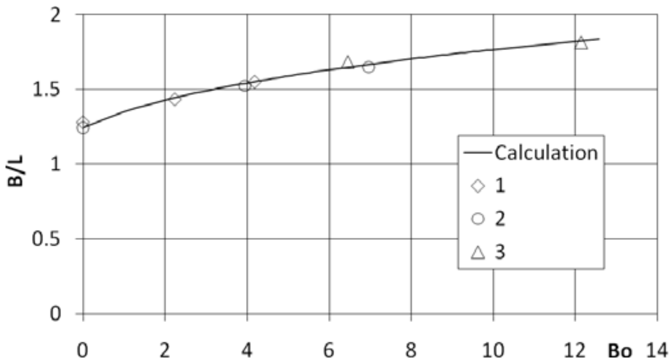


Fig. 3. Dependence of dimensionless diameter of wetted spot, B/L, on Bo number, $L^2 g\rho/\sigma$, for water drops on Teflon under different gravity (B- diameter of wetted spot, m; $L=V^{1/3}$, where V is drop volume, m^3; ρ- liquid density, kg/m^3; σ- surface tension, N/m). Calculation is for CA=113°. Experiment: 1- V=0.067 ml, 2- V=0.146 ml, 3- V=0.340 ml.

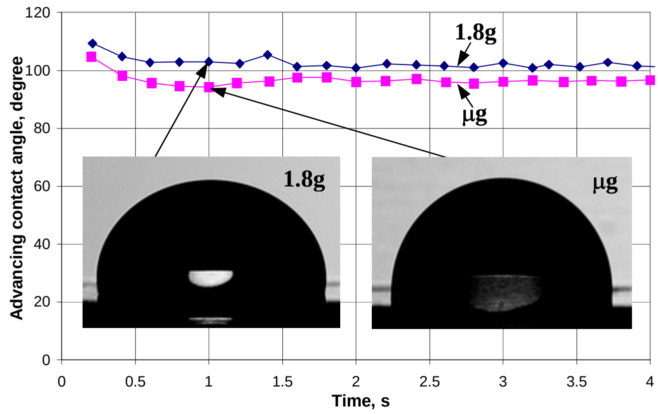


Fig. 4. Advancing CA vs. time in growing water drop on Copper under different gravity. Flowrate is 1 ml/min. Shown are 16 μl drops.

Acknowledgements:
The authors gratefully acknowledge support of this work by the European Space Agency (Spin your Thesis! program), HEAT TRANSFER PRODEX Project (Belgium) and Marie Curie Grant PIIF2-GA-2009-237869.

The influence of the gravity force on longwave Marangoni patterns in two-layer films

Alexander A. Nepomnyashchy, Ilya B. Simanovskii
Department of Mathematics, Technion - Israel Institute of Technology, 32000
Haifa, Israel
nepom@math.technion.ac.il ; yuri11@inter.net.il

In the past few decades, the development of microfluidics and nanotechnology led to a significant progress in the exploration of thin film flows. When the layers are sufficiently thin, the flows are strongly affected by interfacial phenomena, specifically by the Marangoni effect (for a review, see Simanovskii and Nepomnyashchy (1993); Nepomnyashchy, Simanovskii and Legros (2006)).

For the description of the ultra-thin film dynamics, it is necessary to take into account the long-range intermolecular forces (first of all, vander Waals forces) acting between molecules of the liquid and substrate (Israelachvili, 1992). It is essential that these forces act on distances large relative to interatomic distances. Hence, despite their microscopic origin, they can be incorporated into a macroscopic theory.

Dynamics of multilayer ultra-thin films is characterized by several Hamaker constants, which can be of different signs, therefore it can be much richer than that of the one-layer film. A theoretical description of two-layer ultra-thin films has been developed by Pototsky et al. (2004), (2005); Fisher and Golovin (2005).

As a rule, the influence of the gravity on the ultra-thin film dynamics is neglected. Indeed, the disjoining pressure depends on the film thickness h as 1/h^3, while the hydrostatic pressure is proportional to h. Therefore, the ratio of the hydrostatic pressure to the disjoining pressure decreases as h^4. Surprisingly, the effect of gravity on the film stability can be significant. As it has been shown for an isothermal two-layer film by Fisher and Golovin (2005), the hydrostatic pressure cannot be neglected in the case where the Hamaker constants are of different signs and the interaction between different media can ``cancel each other".

In the present work, we consider the influence of the gravity on the long-wave Marangoni patterns in two-layer films. The numerical analysis is carried out in the lubrication approximation (for details, see the review paper by Oron et al. 1997)).

We consider a system of two superposed layers of immiscible liquids with different physical properties. The bottom layer rests on a solid substrate, the top layer is in contact with the adjacent gas phase. The solid substrate and the gas phase are kept at constant different temperatures; the system is heated from above. The surface tension coefficients on the lower and upper deformable interfaces, are linear functions of temperature.

The investigation of the film evolution under the action of the thermocapillary stresses and gravity force, has been fulfilled by means of nonlinear simulations. Evolution equations have been discretized by central differences for spatial derivatives and solved using an explicit scheme. Periodic boundary conditions have been applied on the boundaries of the computational region. For any set of parameters, the evolution equations have been solved for two kinds of initial conditions: either (i) the initial conditions were chosen as their mean values plus small random deviations imposed using a code creating pseudo-random numbers, or (ii) they have been adopted from the simulations formerly carried out for another set of parameters. The computations have been performed in the region 240x240 using the grid 400x400.

Analytical expressions for thresholds of monotonic and oscillatory instability modes have been obtained. Except for the case of the Rayleigh - Taylor instability, the influence of gravity is stabilizing for both types of instability. A sequence of nonlinear wavy regimes, which develop by the decrease of the gravity parameter, has been studied. That sequence includes two-dimensional and three-dimensional traveling waves, as well as different kinds of three-dimensional standing waves. The flows observed in simulations in the presence of gravity are fully different from that found in the absence of gravity.

The work is partially supported by the Israel Ministry of Science through the grant No. 3-5799, and the grant PITN-GA-2008-214919 by the European Union.

References

Simanovskii I. B., Nepomnyashchy A. A., Convective Instabilities in Systems with Interface, Gordon and Breach, London (1993)

Nepomnyashchy A. A., Simanovskii I. B. and Legros J. C., Interfacial Convection in Multilayer Systems, Springer, New York (2006)

Israelachvili J. N., Intermolecular and Surface Forces, Academic Press, New York (1992)

Pototsky A., Bestehorn M., Merkt D. and Thiele U., Alternative pathways of dewetting for a thin liquid two-layer film, Phys. Rev. E, Vol. 70 pp. 025201 (2004)

Pototsky A., Bestehorn M., Merkt D. and Thiele U., Morphology changes in the evolution of liquid two-layer films, Chem. Phys., Vol. 122 pp. 224711 (2005)

Fisher L. S., Golovin, A. A., Nonlinear stability analysis of a two-layer thin liquid film: Dewetting and autophobic behavior, J. Coll. Interf. Sci., Vol. 291 pp. 515 (2005)

Oron A., Davis S. H. and Bankoff S. G., Long-scale evolution of thin liquid films, Rev. Mod. Phys., Vol. 69 pp. 931-980 (1997)

Temperature and concentration measurements in a flashing ethanol jet : Application to space propulsion.

Maria Rosaria Vetrano[1], Alessia Simonini[1], Johan Steelant[2] and Patrick Rambaud[1]

[1]von Karman Institute for Fluid Dynamics
76 Chaussee de Waterloo, Rhode-Saint-Genèse, 1640, Belgium
[2]European Space Agency, Keplerlaan 1
2200AG Noordwijk, The Netherlands
Vetrano@vki.ac.be

Flash-atomization occurs when a liquid is injected into a low-pressure environment where the vapor pressure is also sufficiently lower than the saturation pressure. In this condition the liquid is in an unstable thermodynamic state and a rapid phase transition occurs through nucleation and bubble growth. Although flashing is considered to be detrimental in many technical applications it can have some potential benefits in propulsion systems as it is known to produce a fine spray with enhanced atomization, increase the effective spray angle, and decrease spray penetration. These significant changes in the spray characteristics have therefore an important impact on the fuel-oxidant mixing and hence on combustion efficiency. The process of flash-atomization and vaporization is clearly described as the subsequent occurrence of nucleation, bubble growth, breakup through bubble disruption, and droplet evaporation. Unfortunately the prediction and control of such phenomenon is still hampered by the unavailability of validated theoretical models for each sub-process. In this context, the creation of a reliable and consistent experimental database is crucial for assessing the accuracy of model predictions and their range of validity. The Planar Laser Induced Fluorescence (PLIF) technique (Lavieille et al. 2001) is capable to simultaneously measure the temperature and the concentration of a liquid undergoing evaporation. This technique is based on the property of a tracer to have different emission spectra in its liquid and gas phase. The liquid under investigation is seeded with a fluorescent dye (in the present work Rhodamine B) which is excited by a high energy pulsed laser (Nd:Yag at wavelength of about 532 nm) and re-emits light into the orange band of the visible spectrum. The fluorescence intensity emitted is simultaneously measured in two well separated 20nm width spectral bands. The resulting fluorescent intensity can be written as :

$$I_f(T,C,\lambda_i) = A^i C \cdot e^{\beta(i)/T}, \quad i = \{1,\ 2\} \qquad (1)$$

where λ_i is the center value of the i^{th} spectral band, β_i is the temperature sensitivity, T is the liquid temperature, C is the dye concentration and A^i is a constant depending on the optical configuration. Using the following two ratios :

$$R_f^{(i,j)} = \frac{I_f^i(T,C)}{I_f^j(T,C)}, \quad S_f^{(i,i)} = \frac{I_f^i(T_0,C_0)}{I_f^i(T,C_0)}, \qquad (2)$$

where T_0 and C_0 are two a priori known values of liquid temperature and concentration. The liquid temperature and the dye concentration can be written as follows:

$$T = \left(\frac{1}{T_0} - \ln\left(\frac{R_f^{12}(T_0)}{R_f^{12}(T)} \right) \Big/ \beta(1) - \beta(2) \right)^{-1} \qquad (3)$$

$$C = \frac{C_0}{S_f^{11}} e^{\beta(1)\cdot(1/T_0 - 1/T)}. \qquad (4)$$

In this work the PLIF technique is first used to measure the temperature variation of an ethanol jet undergoing rapid evaporation. The accuracy of the technique is investigated through the analysis of different error sources such as fluorescent light reabsorption and contribution of the scattered waves on the fluorescence phenomenon.
Then the PLIF technique is applied to measure the liquid phase temperature of an ethanol jet flash-atomized in a highly stable low pressure atmosphere filled with inert gas. The experimental conditions at which the measurements are performed range from 0.2-0.5 MPa for the liquid pressure injection, and at room temperature. An example of the temperature map of an ethanol jet at 311.5 K atomized in an environment at 293.15 K is presented in Figure 1. The temperature map has been obtained as an average on 100 pairs of images. The temperature variation along the axis of the atomization is in good agreement with the one obtained using a rack of thermocouples.

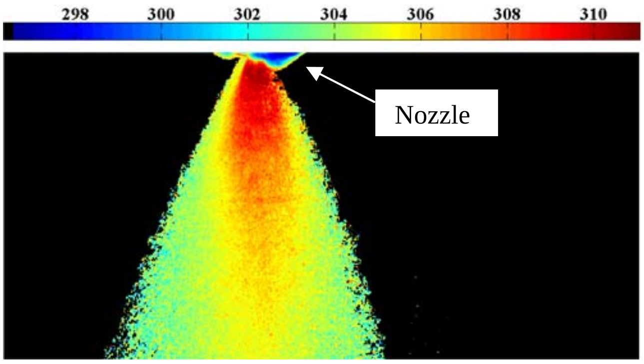


Figure 1: Temperature map of an ethanol jet undergoing evaporation. The nozzle temperature and the environmental temperature are respectively 311.15 K and 293.15 K.

References

Duwell I, Koban W, Zimmermann F. P., Dreier and Schulz C., Spectroscopic characterization of the fluorobenzene/ DEMA tracer system for laser-induced exciplex fluorescence for the quantitative study of evaporating fuel sprays, Applied Physics B, Vol. 97, pp. 909-918 (2009)

Lavieille F., Lemoine F., Lavergne G. and Lebouche M., Evaporating and combusting droplet temperature measurements using two color laser induced fluorescence, Experiments in Fluids, Vol. 31 pp. 45-55 (2001)

Experimental evidence of heat transfer enhancement by vibrations. Microgravity experiments.

Valentina Shevtsova, Aliaksandr Mialdun, Denis Melnikov, Yuri Gaponenko,
Jean Claude Legros

Microgravity Research Centre, Universite Libre de Bruxelles, CP 165/62, Avenue F.D. Roosevelt 50, B-1050, Brussels, Belgium
E-mail of corresponding person: vshev@ulb.ac.be

Vibrational convection refers to the specific flows that appear when a fluid with density gradient is subjected to external vibration. The density gradient may result from the inhomogeneity of temperature or composition. The case of small amplitude and high frequency vibration (when the period is much smaller than the characteristic viscous and heat (mass) diffusion times) is of special interest. In this case, the flow field can be represented as a superposition of 'quick' part, which oscillates with the frequency of vibration, and 'slow' time--average part (mean flow), which describes the non--linear response of the fluid to a periodic excitation. This effect is more pronounced in the absence of other external forces (in particular, static gravity).

The study of vibrational impact on fluids has fundamental and applied importance. Vibrational convection provides a mechanism of heat and mass transfer due to the existence of mean flows. In weightlessness, it is an additional way of transporting heat and matter similar to thermo- and solutocapillary convection. Mean flows show some similarity with gravity-induced convection and might serve as a way to control and operate fluids in space.

Heat transfer and convective pattern flows created by vibrations have been extensively studied in the experiment IVIDIL onboard of International Space Station (Mazzoni et al) and in Parabolic flights (Mialdun et al., 2008).

When a fluid is subjected to high--frequency vibration and density inhomogeneity is caused by thermal gradient, the vibrational convective mechanisms is characterized by the Gershuni number

$$Gs=(A\omega\ \beta_T\,\Delta T\ L)^2/2\nu\chi,$$

where A is the amplitude of vibration, $\omega=2\pi f$ is the angular frequency β_T is the thermal expansion coefficient, ν is the kinematic viscosity, χ is the thermal diffusivity, L is the characteristic size, and ΔT is the applied temperature difference.

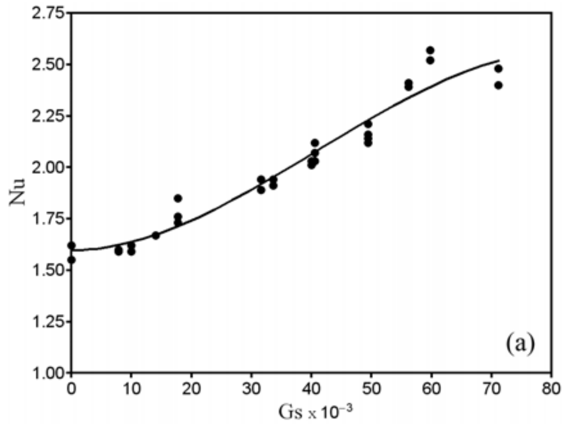


Fig 1. The experimental dependence of the Nusselt number on Gershuni number. Results from Parabolic flight.

The experimental techniques of IVIDIL and in parabolic flights were somewhat similar. The working liquid is placed in a cubic cell with transparent walls. The top and bottom walls are kept at constant and different temperatures. The experimental cell is attached to the linear motor, which performs translational harmonic oscillations in direction perpendicular to the temperature gradient, see detailed description in paper by Shevtsova et al., 2010. The thermo vibrational flows were monitored by measuring the variation of the refractive index inside the cell by optical digital interferometry, based on the concept of Mach-Zehnder.

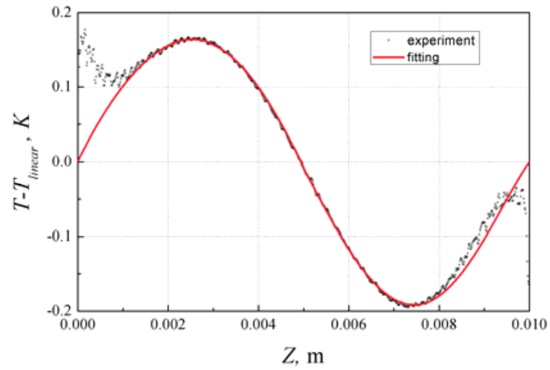


Fig.2. Deviation of the temperature from conductive profile. IVIDIL experiment on the ISS.

The experimental evidence of heat transfer enhancement by vibrations is presented in Fig.1 as dependence of Nusselt number on vibrational forcing. These results are coming from parabolic flight experiments where very large Gs numbers were achieved. On the ISS accessible Geshuni numbers were smaller but microgravity level was much better. The deviation of temperature profile from conductive one due to vibrational convection is shown in Fig.2 as results of ISS experiment. The obtained results from both microgravity platforms clearly demonstrate that vibrational convection strongly intensifies the heat transfer in the system.

References

S. Mazzoni, V. Shevtsova, A. Mialdun, D.E. Melnikov, Yu. Gaponenko, T. Lyubimova and M. Z. Saghir, Vibrating liquids in Space, Europhysics News, Vol. 41, p.14, 2010

A. Mialdun, I.I. Ryzhkov, D.E. Melnikov, V. Shevtsova, Experimental evidence of thermo-vibrational convection in low gravity, Phys. Rev. Lett., Vol.101, 084501 (2008)

V. Shevtsova, I.I. Ryzhkov , D.E.Melnikov , Yu. Gaponenko,, A. Mialdun, Experimental and theoretical study of vibration—induced thermal convection in low gravity, J. Fluid Mech., Vol. 648, pp.53-82, 2010

Precursor film in electrowetting and precursor chain in wetting of an interior corner for nano droplets

Ya-Pu Zhao

State Key laboratory of Nonlinear Mechanics (LNM), Institute of Mechanics
Chinese Academy of Sciences, Beijing 100190, China
E-mail: yzhao@imech.ac.cn

Molecular dynamics (MD) simulations and molecular kinetic theory (MKT) analysis are performed to explore the formation of the precursor film (PF) in electrowetting (Fig. 1) [1] and precursor chain (PC) in the wetting of an interior corner for nano droplets (Fig. 2), the latter is studied for the first time. A comparison is made among the PF, the PC and the single file chain in carbon nanotube [2]. It is found that the disjoining pressure converged at the interior corner is the driving force for the formation of the PC, and the PC could eliminate the pressure and energy singularities at the interior corner. The new findings in this talk would assist in the understanding of the multi-scale droplet dynamics under multi-physical-chemical fields [3, 4].

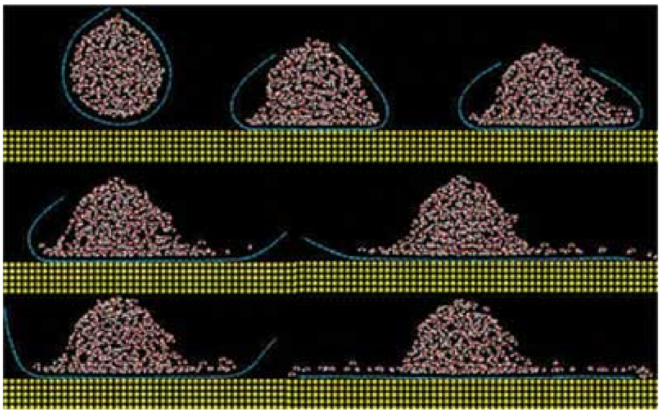


Fig. 1. The dynamic electro-elasto-capillary process of a droplet on gold substrate and the formation of the PF.

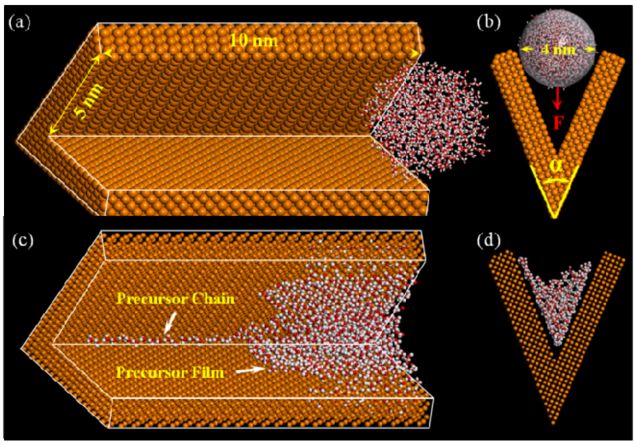


Fig. 2. Formation of precursor chain for the wetting of an interior corner under the action of disjoining pressure.

References

1. Yuan QZ and Zhao YP. Precursor film in dynamic wetting, electrowetting and electro-elasto-capillarity. *Physical Review Letters*, 104 (24): 246101 (2010). Cover paper.

2. Yuan QZ and Zhao YP. Hydroelectric voltage generation based on water-filled single-walled carbon nanotubes. *Journal of the American Chemical Society*, 131 (18): 6374-6376 (2009).

3. Feng JT, Wang FC and ZhaoYP. Electrowetting on a lotus leaf. *Biomicrofluidics*, 3: 022406 (2009).

4. Wang FC, Yang FQ and Zhao YP. Size effect on the coalescence-induced self-propelled droplet. *Applied Physics Letters*, 98: 053112 (2011).

Experimental investigation of 3-dimensional wavy liquid films under the influence of electrostatic forces

W. Rohlfs, G. F. Dietze, H. D. Haustein, V. V. Lel and R. Kneer

RWTH Aachen University, Institute of Heat and Mass Transfer, Eilfschornsteinstraße 18, Aachen, 52066, Germany
rohlfs@wsa.rwth-aachen.de

The wave topology of falling liquid films can be influenced by the presence of an electric field for the case of dielectric working liquids (see Tsvelodub, 2010 for a long-wave approximation of the problem). This provides the possibility to modify flow regimes with the goal of intensifying heat transfer. In the present study, liquid films with imposed regular 3-dimensional surface waves were investigated experimentally, showing that electrostatic surface forces destabilize the film flow.

Figure 1 illustrates the employed measurement section which enables the excitation of regular 3-dimensional surface waves on a falling liquid film. This is realized by way of an upstream loudspeaker (not pictured), imposing a constant wave frequency and thus a corresponding streamwise wavelength Λ_x, and equidistant needles, directly imposing a spanwise wavelength Λ_z.

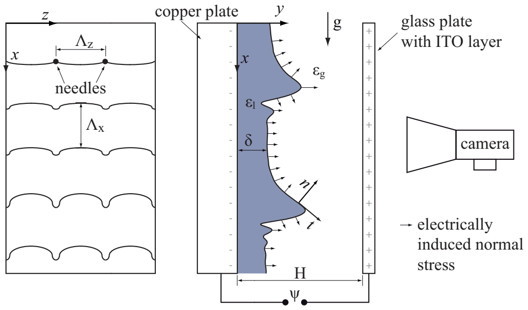


Figure 1: Schematic drawing of the experimental setup: front view (left) and enlarged side view (right).

Further, the resulting 3-dimensional falling liquid film is exposed to an electric field established between a copper plate and a glass plate coated with a thin electrically conductive Indium-Tin Oxide (ITO), thereby allowing optical access for different measurement techniques.

Using this setup, film flow regimes of different Reynolds number values (2-6) and streamwise frequencies (8-20 Hz) have been examined under the influence of an electric field. The strength of which was varied by applying different potentials (0.5-5.5 kV). Therein, qualitative measurements of the wave topology were performed using a digital single-lens reflex camera.

If an electric field is applied to a multiphase system with different dielectric permittivities (ε_l for the liquid side and ε_g for the gaseous side), an electric surface force normal to the interface (f_s) occurs according to

$$f_s = \frac{\varepsilon_0}{2}\left[\varepsilon_g\left(E_{g,n}^2 - E_{g,t}^2\right) - \varepsilon_l\left(E_{l,n}^2 - E_{l,t}^2\right)\right],$$

whereby ε_0 denotes the dielectric constant ($8.85 \cdot 10^{-12}$ C/Vm). The electric field strength $\vec{E}$ is given by the gradient of the potential $\vec{E} = -\nabla\psi$. The product of its normal component E_n and the respective permittivity ε is continuous across the interface and thus $E_{n,l}\varepsilon_l = E_{n,g}\varepsilon_g$, while E_t is continuous itself. In case that $\varepsilon_l > \varepsilon_g$, the electric force points from the liquid (in this case a silicon oil, DMS T12: $\varepsilon_l = 2.67$) to the gaseous phase. The magnitude of the force is mainly influenced by the local electric field, which is a function of the film thickness. Figure 2 illustrates the dependence of the resulting electrostatically induced normal interfacial stress on the film thickness.

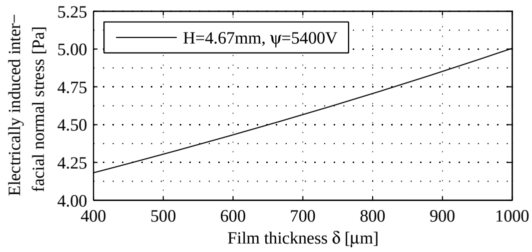


Figure 2: Electrostatically induced interfacial normal stress as a function of the film thickness δ for $\varepsilon_g = 1$ and $\varepsilon_l = 2.67$.

Figure 3 depicts photographs of the liquid film's free surface with and without an applied electric field. The additional electrostatic force is seen to destabilize the large wave humps in spanwise direction, leading to the occurrence of horseshoe-shaped waves of shorter wavelength.

ψ = 0 kV   ψ = 5.4 kV

Figure 3: Photographs of the liquid film's free surface showing the influence of the electric field: Re = 4.6, f = 12.2 Hz, H = 4.67 mm.

Acknowledgements

The authors thank Ms. Anne Mettner and Mr. Norman Lahann for their contribution to the development of the 3-dimensional excitation mechanism employed to obtain the experimental results in Figure 3. This work was financially supported by the Deutsche Forschungsgemeinschaft (grant number DFG KN 764/3-1).

References

Tsvelodub, O. Yu., Samatov, S.N., Effect of the electric field on the wave flow regimes of a thin film of a viscous dielectric fluid, J. Applied Mechanics and Technical Physics 51(3) (2010) 359-568.

Measurement of falling film thickness around a horizontal tube by a conductance probe technique

Hao HOU, Qincheng BI*

Xi'an Jiaotong University, State Key Laboratory of Multiphase Flow
No. 28 Xianning West Road, Xi'an, 710049, China
qcbi@mail.xjtu.edu.cn

Introduction

Falling film condensation and evaporation are important heat transfer processes in the horizontal-tube heat exchangers, which have been utilized in the refrigeration, chemical petroleum refining and desalination industries. For a fully developed laminar film flow, heat is transferred across the film by one-dimensional heat conduction, so that the local heat transfer coefficient is inversely proportional to the film thickness. Hence, falling film heat transfer, whether for condensing, evaporating, or subcooled films, is very sensitive to the actual thickness of the film.

In this article, the film thickness of a liquid flowing around a horizontal plain tube is measured using a conductance probe technique based on the conductivity characteristics of gas-liquid two-phase medium. The liquid film thickness distribution at different locations of the horizontal tube is investigated to reveal the flow behavior of falling film.

Experimental apparatus and method

A schematic of the test facility is shown in Figure 1. It is composed of three main parts: fluid circuit, distributor and test section. The fluid circuit is used to supply different kinds of flowing conditions to ensure a steady falling film around the horizontal tube. In the circuit, the fluid starts out in the tank that serves as a reservoir and then passes through a pump and a filter. The flow rate is measured with one of two rotameters, which are necessary to cover the desired ranges of mass flow. The thickness of a liquid film is measured by means of a conductance probe mounting at a micrometer, as illustrated in Figure 2. An electric circuit is used to indicate when the probe contacts with the film. Similarly, the micrometer is used to indicate the point at which the probe just touches the tube. The liquid film thickness is the distance between these two interfaces.

Results and discussion

Figure 3 shows the variations of the film thickness around the plain tube with the circumferential angle and inter-tube spacing at *Re*=574. It indicates the measured film thickness as a function of the circumferential angle β. With the increasing of the circumferential angle, the film thickness falls sharply and then increases gradually. The present measurements validate Nusselt's theory giving a reasonable prediction of film thickness around the upper perimeter of the tube but much poorer agreement on the lower perimeter. The minimal values of the film thickness tend to locate at different angular positions of 90º-115º under different conditions. Though the momentum acquired by the film at β=90º is not lost by the fluid when β>90º, this effect is not covered by Nusselt's theory, which assumes negligible momentum effect on the flow. In addition, the film thickness gets thinner with inter-tube spacing increasing and the flow becomes unstable.

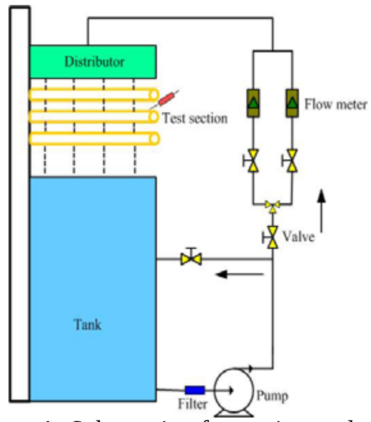


Figure 1: Schematic of experimental setup

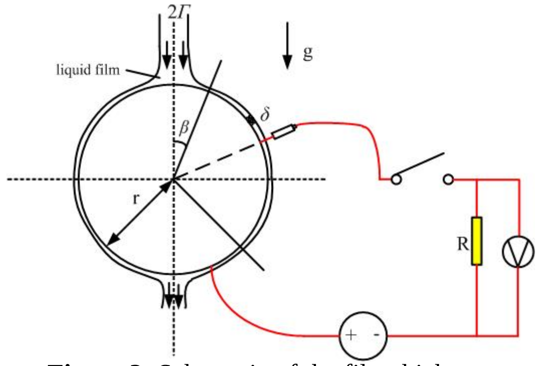


Figure 2: Schematic of the film thickness measurement system

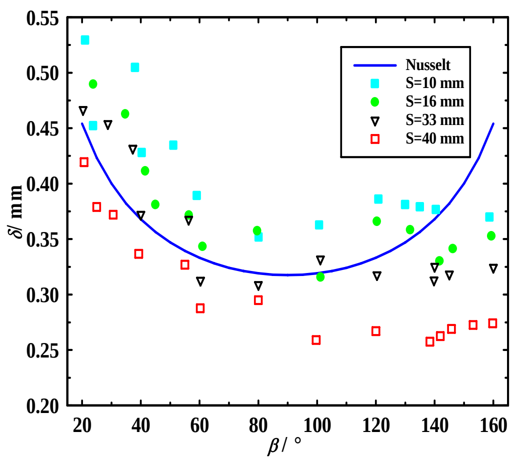


Figure 3: Effect of circumferential angle and inter-tube spacing on liquid film thickness (smooth tube, Φ25.4 mm, *Re*=574)

References

Ribatski G, Jacobi A M. Falling-film evaporation on horizontal tubes-a critical review, International Journal of Refrigeration, Vol. 28 pp. 635-653(2005).

Stability of thin liquid films on structured substrates

Vladimir Ajaev[1], Elizaveta Gatapova[2] and Oleg Kabov[2,3]

[1]Department of Mathematics, Southern Methodist University, Faculty, Dallas TX 75275, USA
[2]Institute of Thermophysics SB RAS, Lavrentyev Ave., 1, Novosibirsk, 630090, Russia
[3]Microgravity Research Center, ULB, CP 165/62, Ave. Roosevelt 50, B-1050 Brussels, Belgium
ajaev@mail.smu.edu, gatapova@itp.nsc.ru, okabov@ulb.ac.be

Structured surfaces with gas or vapor trapped between structure elements have a wide range of potential applications due to their property of effective slip reduction. The studies of the effects of substrate heterogeneity on film rupture (Kao et al. 2006) and the models of slip flow in thin films (Kargupta et al. 2004) represent two mostly independent directions of research in thin film dynamics.

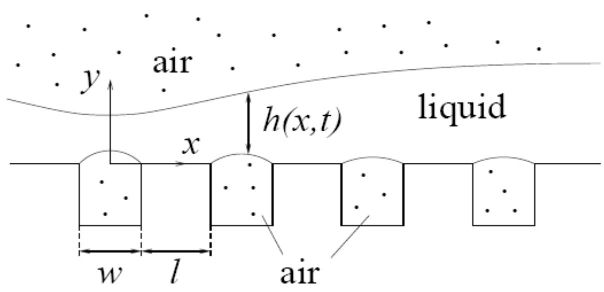


Figure 1: Sketch of a liquid film on a structured surface and Cartesian coordinates.

We investigate stability and break-up of a thin liquid film on a solid surface under the action of disjoining pressure. The solid surface is structured by parallel grooves. Air is trapped in the grooves under the liquid film (Fig. 1). Our mathematical model takes into account the effect of slip due to the presence of menisci separating the liquid film from the air inside the grooves, the deformation of these menisci due to local variations of pressure in the liquid film, and non-uniformities of the Hamaker constant which measures the strength of disjoining pressure. Both linear stability and strongly nonlinear evolution of the film are analyzed. Surface structuring results in decrease of the fastest growing instability wavelength and the rupture time. It is shown that simplified descriptions of film dynamics, e.g. using the standard formulas for effective slip, can lead to significant deviations from the behavior seen in our simulations. Self-similar decay over several orders of magnitude of the film thickness near the rupture point is observed. We also show that the presence of the grooves can lead to instability in otherwise stable films if the relative groove width is above a critical value, found as a function of disjoining pressure parameters. Simulations are also conducted for spatially non-uniform Hamaker constant. Rupture time is shown to decrease as the ratio of the maximum and minimum values of the Hamaker constant is increased (Fig. 2).

For stable films, we evaluate the potential efficiency of this configuration for cooling applications by solving a model which accounts for heat transfer in both the liquid film and the solid phase.

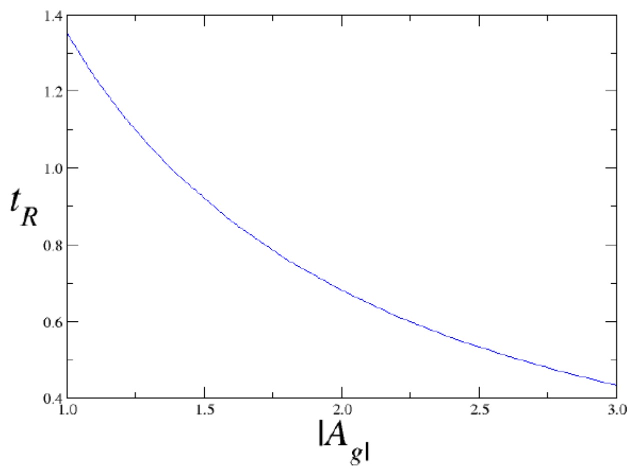


Figure 2: Rupture time of the film as a function of the ratio of Hamaker constants for β_0=10, N=10, w=l=0.02.

Our study is important for cooling technology in earth and space applications. It is well known that the structured surfaces are used in heat transfer devices for intensification of single-phase and two-phase heat transfer. Liquid film rupture in these devices has a significant influence on heat transfer rates. And the rupture conditions clearly depend on the parameters of the structuring. Our investigations are performed for negligible gravity situation when the thin liquid film on a solid surface breaks-up under the action of disjoining pressure. The present work is a part of the preparation of the SAFIR experiment of the European Space Agency onboard the International Space Station.

References

J. C.-T. Kao, A.A. Golovin, S.H. Davis, Rupture of thin films with resonant substrate patterning, J. Colloid Interface Sci., 303, pp. 532-545 (2006)

K. Kargupta, A. Sharma, R. Khanna, Instability, Dynamics, and Morphology of Thin Slipping Films, Langmuir, 20, pp. 244-253 (2004)

On three-dimensional structure and interaction of primary and secondary waves in annular gas-liquid flow at low liquid flow rates

S.V. Alekseenko, A.V. Cherdantsev and D.M. Markovich

Kutateladze Institute of Thermophysics
1, Lavrentiev ave., Novosibirsk 630090, Russian Federation
Novosibirsk State University
2, Pirogov str., Novosibirsk 630090, Russian Federation
cherdantsev@itp.nsc.ru

Flow of thin liquid film along the inner wall of tube, sheared by intensive gas stream is called annular gas-liquid flow. Film surface is unstable to small disturbances, evolving into nonlinear waves, interacting with each other. Gas shear enhances generation of waves, increasing their velocity and passing frequency, steepening their slopes and decreasing their amplitude. Recent investigation with application of high-speed LIF technique has shown (Alekseenko et al 2009) that waves of two types coexist in such flow even at very low liquid Reynolds numbers. Fast long-living 'primary' waves generate slow short-living 'secondary' waves on their back slopes. After inception, secondary waves travel along residual layer between primary waves and finish by being absorbed by the following primary wave.

The picture of wavy motion, described above, was obtained in experiments with so-called '2D-approach'. This means that evolution of instantaneous distribution of local film thickness along only one longitudinal section of the channel was studied. Detailed analysis of space-time diagrams has shown that in rather rare (up to 6%) cases, deviations of the described scenario were observed. In 2D-approach these deviations looked either like transition of a secondary wave into primary one (secondary wave accelerated, growing in amplitude, and began to generate secondary waves itself), or like transition of primary wave into secondary one (vice versa, primary wave decelerated, loosing amplitude and was absorbed by the following primary wave).

These unusual events became an object of an additional special investigation, since they could shed light on nature of primary and secondary waves. Besides measuring the frequency characteristics of unusual events in 2D-approach, study of these phenomena in so-called '3D-approach' was performed. The latter means study of evolution of instantaneous distribution of local film thickness in the area of tube's inner surface of 12 mm width (1/4 of wetting perimeter) and 100 mm length. High-speed LIF technique, using high-power (3 Wt) continuous laser and high-speed camera with rectangular matrix allowed us to realize this approach.

First, 3D-data were presented as 2D-data along the central longitudinal section of working area ('centerline'). Then search for non-standard events like mentioned above transitions was performed in 2D-representation and 3D-evolution of film thickness in spatio-temporal neighborhood of event was investigated.

Such analysis showed that no transition of one type of waves into another take place. Both phenomena, interpreted as transitions in 2D-approach, represent 2D-projections of complex interaction of primary and secondary waves at the edges. In the first case, secondary wave, visible at the centerline, is being absorbed by the edge of primary wave, invisible at the centerline. In Figure 1 example of instantaneous picture of such phenomenon is given: wide primary wave on the left (x=50 mm, moving RTL) generates circumferentially localized secondary wave behind it (x=60 mm). This secondary wave will be absorbed soon by the edge of another primary wave, invisible at the centerline (x=65mm, y=10-12 mm). After absorption, secondary wave becomes part of a primary one, growing in amplitude and velocity, and beginning to generate new secondary waves.

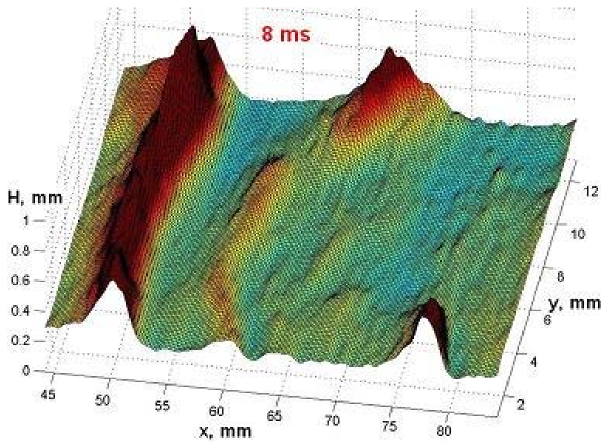


Figure 1: Surface of liquid film just before absorption of a secondary wave by the edge of primary one. Liquid Reynolds number 18, gas velocity 18 m/s, liquid viscosity 3 cSt.

In similar way, transition of primary wave into secondary one can be explained in terms of interaction of edges of two primary waves, when one of them becomes invisible at the centerline.

References

Alekseenko S.V., Antipin V.A., Cherdantsev A.V., Kharlamov S.M., Markovich D.M. Two-wave structure of liquid film and waves interrelation in annular gas-liquid flow with and without entrainment, Physics of Fluids, Vol. 21, pp. 061701-061704 (2009)

Effect of Gravity during Condensation of R134a in a Circular Minichannel

Enrico Da Riva and Davide Del Col

University of Padova, Dipartimento di Fisica Tecnica
Via Venezia 1, I-35131, Italy
davide.delcol@unipd.it

Numerical simulations of R134a condensation are performed at representative operating conditions (i.e. 1 mm i.d., 40 °C saturation temperature, 100 kg m^{-2} s^{-1} and 800 kg m^{-2} s^{-1} mass fluxes). The distribution of local condensate film thickness and local heat transfer coefficients is reported and discussed. Simulations have been performed assuming horizontal channel orientation, vertical downflow orientation and zero-gravity operating conditions in order to assess the relative influence of gravity, surface tension and interfacial shear stress.

The inlet thermodynamic quality is x = 1 and the refrigerant is condensed till reaching around x = 0.5. Since according to experimental visualization studies available in the literature (Garimella et al., 2002) the flow pattern is expected to be annular, the problem is treated as steady-state. The minichannel wall is held isothermal at 30 °C uniform temperature, while the saturation temperature of the fluid is 40 °C. The simulations employ the VOF method implemented in a commercial package.

The efficacy of this simulation approach has been assessed by comparing the numerical results with experimental data by Matkovic et al. (2009) for the horizontal minichannel case, displaying a good agreement both at G = 100 kg m^{-2} s^{-1} and G = 800 kg m^{-2} s^{-1}.

In the horizontal minichannel at G = 100 kg m^{-2} s^{-1} the condensation process is found to be gravity-dominated: the liquid condensed in the upper part of the tube is drained to the bottom by gravity, while the film thickness keeps pretty uniform in the upper half of the tube. Because of the stratification, much higher heat fluxes are obtained in the upper half of the channel as compared to the bottom.

A completely different behavior is achieved when the minichannel is vertically oriented: as one can see in Fig. 1, much higher heat transfer coefficients are obtained in the horizontal minichannel as compared to the vertical one. The stratification due to gravity is shown to enhance the heat transfer since this keeps the condensate thickness low in the upper region of the channel (i.e. 15~20 μm). The total channel length required to condensate the refrigerant till 0.5 vapour quality is 66 mm in the horizontal minichannel and 90 mm in the vertical one. Besides, at G = 100 kg m^{-2} s^{-1}, the total heat flow rate is 5% higher in the vertical downflow configuration as compared to the zero-gravity simulation: the gravity force acts together with the shear stress and a slightly thinner condensate film is achieved.

At G = 800 kg m^{-2} s^{-1} the interfacial shear stress is found to be the prevailing force since the distribution of local condensate thickness and local heat transfer coefficient is almost completely axisymmetrical. The normal-gravity horizontal, normal-gravity vertical and zero-gravity simulations cases display almost identical results. Besides, much higher heat fluxes are obtained as compared to the G = 100 kg m^{-2} s^{-1} case, mainly because of the influence of turbulence in the liquid phase. It is worth noting that at G = 100 kg m^{-2} s^{-1} the liquid flow is laminar and the effective thermal conductivity is equal to the molecular conductivity, while at G = 800 kg m^{-2} s^{-1}, because of the effect of turbulence, the effective thermal conductivity is more than ten times higher.

The global effect of surface tension is found to be negligible during annular flow condensation of R134a in a 1 mm i.d. circular channel. Some slight influence (heat flow rate difference by 2%) is seen only when condensate stratification occurs in the horizontal configuration at G = 100 kg m^{-2} s^{-1}; in this case, because of the effect of surface tension, the minimum liquid film thickness (corresponding to the maximum heat flux) is achieved at around half channel height, not at the top of the channel.

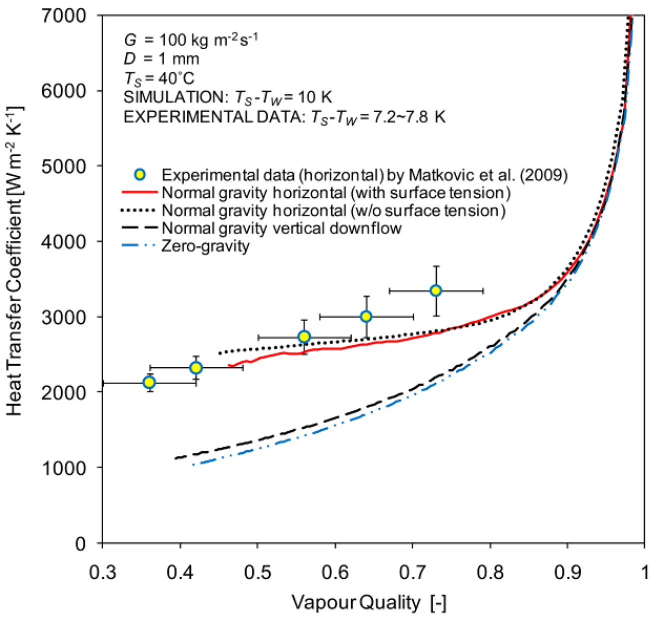


Figure 1: Cross sectional average heat transfer coefficient versus vapour quality at 100 kg m^{-2} s^{-1} mass flux.

References

Matkovic, M., Cavallini, A., Del Col, D., Rossetto, L., Experimental study on condensation heat transfer inside a single circular minichannel. International Journal of Heat and Mass Transfer **52**, 2311–2323 (2009)

Garimella, S., Killion J.D., Colemann, J.W.: An experimentally validated model for two-phase pressure drop in the intermittent flow regime for circular microchannels. ASME J. Fluids Eng. 124, 205–214 (2002).

Instability of the joint flow of liquid film and co-current gas flow: theory and experiment

Viktoriya Bekezhanova[1] and Oleg Kabov[2,3]

1-Institute of Computational Modeling SB RAS, Akademgorodok, 660036, Krasnoyarsk, Russia, bekezhanova@mail.ru
2-Universite Libre de Bruxelles, Microgravity Research Center, Av. Roosevelt 50, CP 165/62, B-1000 Brussels, Belgium
3-Institute of Thermophysics SB RAS, Prosp. Lavrentyev 1, 630090, Novosibirsk, Russia, okabov@ulb.ac.be

Multiphase flows including liquid non-isothermal media with deformable fluid-gas interface are widely applied in engineering. The simulation of these flows is a complex of problems arising in the process of prediction of technological process dynamics (intensification of heat transfer, cooling of devices, setup of life support systems etc.). In the framework of scientific projects supported by the European Space Agency [Celata at al.], technologies using thin films of dielectric liquids driven by gas flow are developed. Surface forces in this flow may dominate over volumetric ones or may be comparable to them. In experiments it was found out that a spatial self-organizing structure is formed in liquid film [Kabov et al., 2007]. Convective flows with interfaces are characterized by the presence of instabilities of different nature [Bekezhanova and Kabov, 2010]. With a view of optimization of the coming experiments, a further theoretical investigation of the influence of gravity, thermocapillary forces and controlling parameters (gas and fluid flow rates, properties of phases) on the characteristics of the arising structures has been performed.

A joint flow of thin liquid film and co-current gas flow in minichannel is considered. The motion of the liquid and gas is described by the Navier-Stokes equations. The Poiseuille flow is observed in the gas and fluid. The assumption is justified so far as the velocities and the sizes of the system are sufficiently small (the height of the channel is 1.4 mm). The fulfillment of the following common conditions is required on the interface: continuity of temperature, velocity, normal and tangential stresses, kinematic condition and full energy condition including the energy which is expended on the interface deformation.

The development of instabilities is studied using the linear stability theory. The influence of gravity and gas and liquid flow rates on the stability of the basic flow in the FC-72-nitrogen two-phase system is investigated. The arising instabilities lead to generation of different flow regimes. The solution for the problem on stability on $\mathrm{Re}_g, \mathrm{Re}_l$-plane is shown on Fig. 1, where $\mathrm{Re}_g, \mathrm{Re}_l$ are Reynold's numbers of gas and liquid, respectively. In zone *I* the film flow is steady. In region *II* the thermocapillary instability appears in the form of longitudinal shafts. The wavelength of disturbances generating this type of instability is less than 6 mm. In zone *III* the flow crisis is induced by running thermal waves, which are distributed in the basic stream's opposite direction. The velocity of these hydrothermal waves is more than the longitudinal perturbation velocity. The instability type is generated by disturbances whose order of wavelength is 3 mm. Domain *IV* is characterized by the irregular structure of the disturbed flow. It should be noted that under microgravity in the considered range of parameters $\mathrm{Re}_g, \mathrm{Re}_l$ only the first two instability types are observed (see Fig.1b). And transversal waves appear due to disturbances with a greater wave number.

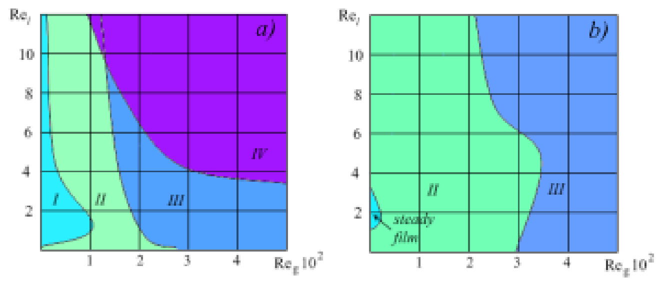


Figure 1: Neutral curve on $\mathrm{Re}_g, \mathrm{Re}_l$-plane: a) $g = g_0$, b) $g = 0$.

Comparison with the flow regimes map obtained in experiments [Cheverda and Kabov] and [Kabov et al., 2010] has been performed. The results were shown to coincide qualitatively. The obtained solution of the problem on stability predicts the appearance of experimentally observed structures.

Acknowledgments: The authors gratefully acknowledge support of this work by the European Space Agency in the framework of SAFIR PRODEX project (Belgium), by Siberian Branch of Russian Academy of Sciences on the integrated project # 64 and ONR NICOP Grant N00014-08-1-0263. V.B. gratefully acknowledges support by Siberian Branch of Russian Academy of Sciences, project # 116.

References

Celata G. P., Colin C., Colinet P., Di Marco P., Gambaryan-Roisman T., Kabov O., Kyriopoulos O., Stephan P., Tadrist L., Tropea C., Bubbles, drops, films: transferring heat in space, *Europhysics News*, Vol. 39, No. 4, p. 23-25, 2008.

Kabov O., Glushchuk A. and Legros J-C., Gravity Effect on Dynamics of Liquid Film with Two Contact Lines: Results of Parabolic Flights, Third International Symposium on Physical Sciences in Space, Keynote lecture, Oct.22-26, Nara, Japan, Book of abstracts, pp. 321-322, 2007.

Bekezhanova V., Kabov O.A, Change in the instability forms of stationary flow of liquid film driven by gas flow, Fifth Int. Topical Team Workshop on Two-phase systems for ground and space applications, Sept.26-29, Kyoto, Japan, Book of abstracts, pp.19, 2010.

Cheverda V. and Kabov O.A., Flow regimes map for FC-72 liquid film in the rectangular mini channel, Third Int. Topical Team Workshop on Two-Phase Systems for Ground and Space Applications, Brussels, September 10-12, Book of Abstracts, pp. 97, 2008.

Kabov O.A., Zaitsev D.V., Cheverda V.V. and Bar-Cohen A., Evaporation and flow dynamics of thin, shear-driven liquid films, *Experimental Thermal and Fluid Science*, 2010 http://dx.doi.org/10.1016/j.expthermflusci.2010.08.001.

Performance of Active-pumped Two-phase CO_2 Loops for the AMS-02 Tracker Thermal Control in Thermal Vacuum Tests

Z.Zhang[1], X.-H.Sun[1], G.-N. Tong[1], Z.-C.Huang[1], Z.-H.He[1,✉], A. Pauw[2], J. van Es[2], E. Laudy[3], R.Battiston[3], and B. Verlaat[4]

1. Center for Space Technology, Sun Yat-sen University, 519082 Zhuhai, People's Republic of China
2. National Aerospace Laboratory NLR, Voorsterweg 31, 8316 PR Marknesse, the Netherlands
3. Universitö and Sezione INFN di Perugia, Laboratory Study of Radiation Effect on Materials for Space application (SERMS), Via Pentima bassa 21, 05100 Terni, Italy
4. National Institute for Subatomic Physics Mechanical Engineering, Science Park 105, 1098 XG Amsterdam, the Netherlands

✉E-mail: stshzh@mail.sysu.edu.cn

A mechanically pumped CO_2 two-phase loop cooling system is developed for the temperature control of the tracker of AMS-02, a cosmological particle detector to work in the international space station. The cooling system (called TTCS), consists of two evaporator rings in parallel to collected heat from the tracker's front-end electronics, two radiators in parallel also to emit the heat into space, and a pump that drives the CO_2 who carry the heat to the radiators, and an accumulator that control the pressure of, and thus the temperature of the evaporators. Thermal vacuum tests were performed to check and to qualify the system operation in simulated space thermal environment.

Together with AMS-02, the TTCS is placed in the LSS Chamber in ESTEC cin a vertical configuration for the evaporator ring plane. The Chamber shroud was liquid nitrogen cooled down to a temperature of −90°C to simulate the background temperature for the AMS-02 detector. To achieve the unequal external heat flux condition, two scenarios were adopted: one employs 195W heaters glued on both of the heat pipe radiators; the other uses 375W infrared lamps facing the two radiators respectively. A schematic layout ot the TTCS is drawn in Figure1.

The mechanically pumped CO_2 two-phase loop cooling system for the AMS-02 tracker passed the thermal vacuum tests. Meeting the design requirement, it exhibits the following advantages: 1) temperature homogeneity and stability along the evaporator tube in orbital thermal environment, the temperature variation of the evaporators can be controlled within ±0.5°C (see Figure2); 2) the operation temperature of the cooling system is controllable, i.g., the accumulator temperature can be changed as programmed, which can adapt the space thermal environment change; yet the response of the evaporator temperatures delays; 3) in the case of imbalanced external heat flux, the system exhibits self-adaptive performance by adjusting the mass flow rates of CO_2 through the two condensers in parallel, which thermally contacted to the two radiators respectively; 4) as observed in most of the two-phase flow systems, especially in that with evaporators in parallel, dry-out would occurs at the evaporator outlet at low mass flow rate, which can be effectively suppressed simply by increasing the mass flow rate.

Acknowledgments

This work was supported by The Prophase research of National Basic Research Program of China under Grant No. 2006CB708613, and The Science and Technology Program of People's Government of Guangdong Province.

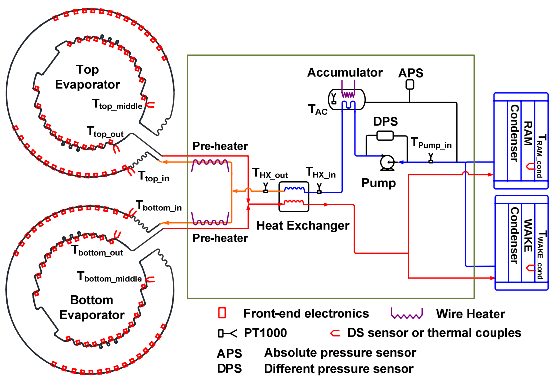


Figure 1: Schematic drawing of the TTCS under test.

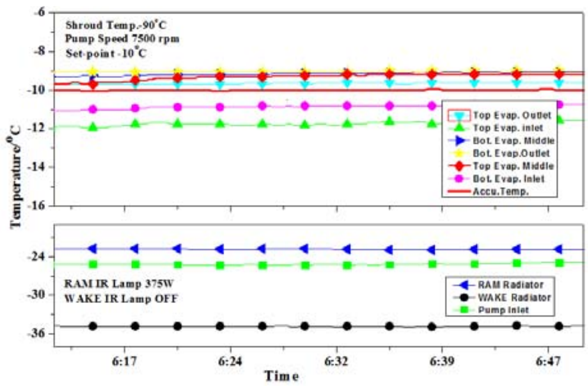


Figure 2: Schematic drawing of the TTCS under test.

References

Delil AAM, Woering AA, Verlaat B. Development status of the mechanically pumped two-phase CO_2 cooling loop for the AMS-2 TTCS, Space Technology and Applications International Forum - Staif 2003; pp.654-88(2003).

Sun XH, He ZH, Huang ZC, Xiao WJ, Qi XM, Pauw A, van Es J. Analysis of Temperature Oscillations in Parallel Evaporators of a Carbon Dioxide Two-Phase Loop, Microgravity Science and Technology; 21, pp.299-304(2009).

Parabolic Flight Experiment of Flow Boiling in Transparent Heated Tube

Osamu Kawanami[1,2], Tatsuya Hamada[1], Kenji Saruwatari[1], Soumei Baba[3], Yuto Wada[3], Hitoshi Asano[4], Yuuki Kanai[4], Yasuhisa Shinmoto[3] and Haruhiko Ohta[3]

[1] Dep. Mechanical and System Eng., University of Hyogo, 2167 Shosha, Himeji, Hyogo 671-2280 Japan
E-mail: kawanami@eng.u-hyogo.ac.jp
[2] Institut für Technische Thermodynamik Technische Universität Darmstadt, Petersenstrasse 30, Darmstadt 64287, Germany
[3] Dep. Aeronautics and Astronautics, Kyushu University, 744 Motooka, Nishi-ku, Fukuoka 819-0395 Japan
[4] Dep. Mechanical Eng., Kobe University, 1-1 Rokkodai-cho, Nada, Kobe, Hyogo, 657-8501 Japan

Recently, heat flux in the electro-devices of space satellite becomes higher and higher. For thermal management of space platforms, a compact boiling heat exchanger has a great potential because large amount of heat from electro-devices can be removed by latent heat. On the other hand, even though the effects of gravity on boiling phenomena are significant importance, the knowledge including critical heat flux in microgravity are not clear until now. Therefore, Japan Aerospace Exploration Agency (JAXA) plans the flow boiling experiment onboard KIBO of ISS. Many researchers including authors join to this project. During 1st to 10th February 2011, we conducted parabolic flight experiment of flow boiling using major components developed for ISS experiment.

A schematic diagram of the experimental apparatus used in this study is shown in Fig. 1. The apparatus mainly consists of pump, flow meter, pre-heater, two different heated sections made by a metallic heated tube and transparent heated tubes, unheated observation part made by acrylic tube, condenser, and tank with liquid separator. Deaerated FC-72 is used as a test fluid. Here, we describe the experiments of flow boiling conducted by transparent heated tube. The transparent heated tube made by authors is employed for detailed investigation not only heat transfer but also flow behavior in the tube. The tube is made of Pyrex glass, and it has an inner diameter and outer diameter of 4 and 6 mm, respectively. The heated lengths of the tube are 50 mm.

The inner wall of the tube is uniformly coated with a transparent gold film with a thickness of the order of 10 nm by electroless plating method. The film is used as a resistance thermometer to directly evaluate the inner wall temperature averaged over the entire heated length. In addition, the transparency of the film allows the visual observation of the flow behavior through the wall. That is, the flow behavior visualization, temperature measurement of the inner wall, and direct heating of the fluid can be simultaneously performed by using a transparent heated tube. We use three transparent heated tubes for the test section, i.e., the total heated length is 50 mm $\times$ 3 segments = 150 mm. Each tube is insulated electronically. The image of the test section is shown in Fig. 2.

The experimental conditions are vertical upward flow, mass velocity G = 30-600 kg/m^{2}s, heat flux q = 6-70 kW/m^2, inlet pressure P_{in} = 0.085-0.18 MPa. Subcooled temperature is set to be 15 K by using a preheater. The inlet and outlet liquid temperatures are measured by K-type thermocouples. The outer wall temperature of the tube is measured by a K-type thermocouple at the center of the heated tube. All data is recorded at 10 Hz by a data logger. Three CCD cameras are used for flow visualization at each segment.

Figure 3 shows an example of flow behavior in 1G and μG under the conditions of G = 35 kg/m^{2}s and q = 14 kW/m^2 at segment 2. The flow pattern was bubbly-slug flow in 1G. Under μG conditions, the flow pattern was translated to annular flow and we found that the boiling bubble was generated from the heated wall surrounded by a thin liquid film. Analyses of heat transfer characteristics are ongoing.

References

H. Ohta, O. Kawanami, H. Asano, H. Kawasaki, R. Imai, Y. Shinmoto, K. Suzuki, H. Ishizuka, K. Fujii and S. Yoda Research Subjects and Proposed Experimental Setup for Flow Boiling in Microgravity, *Proc. The Int. Topical Team Workshop on Two-phase Systems for Ground and Space Applications*, pp.146-147 (2009).

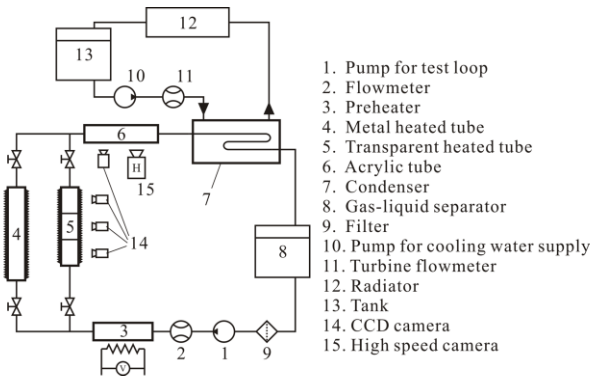


Fig. 1 Experimental set-up

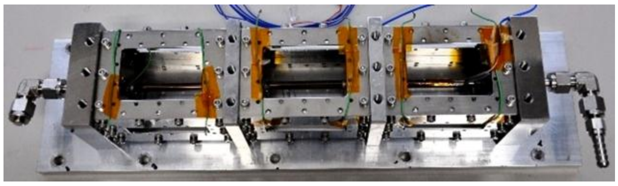


Fig. 2 Test section consisting of three segments including transparent heated tube.

Fig. 3 Flow behavior in 1g (top) and micro-g (bottom) under G = 35 kg/m^{2}s, q = 14 kW/m^2 at segment 2.

Experimented Two-Phase loop system for thermal control system in JAXA

Haruo Kawasaki
JAXA Thermal System Group, 2-1-1 Sengen, Tukuba-shi, Ibaraki JAPAN
E-mail : kawasaki.haruo@jaxa.jp

Abstract

Two-phase loop systems are appropriate for the thermal control system and the thermal management system which can treat a large amount of west heat. The two-phase loops for ground use are general, while it is not general for space. As one of the reasons, gravity effects on the two-phase flow phenomena and on heat transfer characteristics have not been clarified in detail. The two-phase loop systems are classified into the two-phase mechanical pumped loop, loop heat pipe and capillary pumped loop.

In this paper, the developed and the developing the experimental two-phase loop system in the orbit for the thermal control systems were introduced.

1. Two-phase fluid experiment (TPFLEX)

The apparatus of TPFLEX [1,2] was shown in Figure 1. TPFLEX was lunched on August 7th in 1997, experimented in the space shuttle (STS-85) and returned on August 19th in 1997. The apparatus of TPFLEX was consisted with the gear pump, thermal accumulator, evaporator which contain stainless steel porous wick. The working fluid was distilled water. The maximum heat load on the evaporator was 150 W. An experiment with 10 runs was done on the space shuttle. The experiment has effects the non-condensed gas in the loop, however, the operation of the pumped loop could be confirmed.

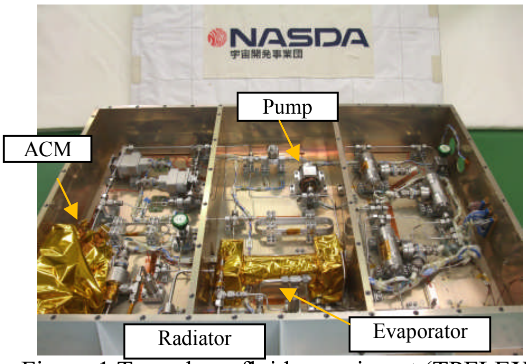


Figure 1 Two-phase fluid experiment (TPFLEX)

2. Reservoir Embedded Loop Heat Pipe (RELHP)

RELHP has a liquid reservoir, located in the evaporator, to ease start-up of the LHP. The RELHP is used to transfer heat from the experimental heat source to a deployable radiator (DPR) on the Engineering Test Satellite-VIII (Kiku-8) which was launched to a geo stationary orbit by the H-IIA rocket on 18 Dec.2006 [3]. The working fluid of the RELHP was NH_3.

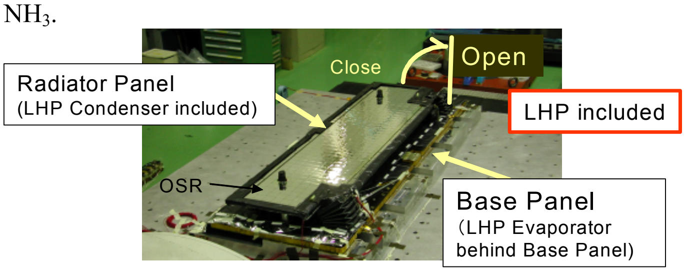


Figure 2 Schematic of the deployable radiator (DPR) on ETS-VIII

The experiment of the DPR and RELHP with various heater inputs (100 W - 400 W) was conducted without any trouble. The RELHP has been estimated through the thermal characteristics test from March 2007 [4]. At present, for practical use, The RELHP has been examining a long term stability on the orbit.

3. Interfacial behaviors and heat transfer characteristics in boiling two-phase flow (TPF)

A project for the experiment on ISS which is named as "Interfacial behaviors and heat transfer characteristics in boiling two phase flow (TPF) [5] has become an official project in JAXA. This experiment uses a two-phase mechanical pumped loop. The transparent heated tube and metal heated tube were used as the evaporator. The heated tubes are smooth tube without the groove. The heated tubes and condenser will be used for the investigation of the gravity effects on the two-phase flow phenomena and on heat transfer characteristics. As the working fluid, FC-72 will be used due to the safety and due to the lower latent heat and lower saturated pressure than one of water. The ground test of TPF and the design are proceeded with to launch in 2014.

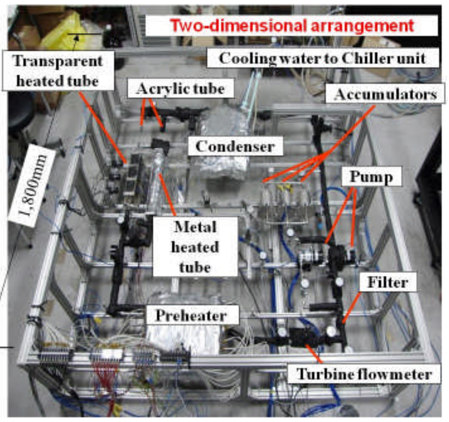


Figure 3 Schematic of ground test loop model (GM) for TPF

References

[1] M. Furukawa, K. Mimura, and M. Komori, " Demonstrative In-Orbit Operations of a Pressure-Regulated Two-Phase Fluid Loop Model", 7 th AIAA/ASME Joint Thermo physics and Heat Transfer Conference, AIAA 98 - 2451. (1998)

[2] M. Furukawa, Y. Miyazaki, and T. Yamazaki, "Thermally Driven Flow Experiments Aboard Space Shuttle", 31th AIAA Conference, ,AIAA95-2069 (1995).

[3] H. Kawasaki, H. Noda, T. Yab, H. Ishikawa, T. Nomura, Y. Saito, "Characteristics of Reservoir Embedded Loop Heat Pipe in Deployable Radiator on ETS-VIII at Beginning of the Experiment under Orbital Environment" AIAA Paper 08-3926, 2008

[4] H. Kawasaki, A. Okamoto, R. Hatakenaka, H. Ishikawa, T. Nomura, Y. Saito and T. Ogushi "Characteristics Variation of Reservoir Embedded Loop Heat Pipe under the Orbital Environment in Three Years" International Conference on Environmental System, AIAA-6059-2010 (2010)

[5] K. Fujii, H. Kawasaki, H. Ohta," Recent Progress of International Space Station KIBO Project - TPF", Fifth International Topical Team Workshop on TWO-PHASE SYSTEMS FOR GROUND AND SPACE APPLICATIONS (2010)

Nucleate Pool Boiling Experiments (NPBX) on the International Space Station*

V. K. Dhir, G. Warrier, E. Aktinol, D. Chao**, J. Eggers†, W. Sheredy**, and W. Booth†

Henry Samueli School of Engineering and Applied Science, UCLA
Los Angeles, California USA
vdhir@seas.ucla.edu

During the period of March – May 2011, a series of boiling experiments was carried out in the Boiling Experimental Facility (BXF) located in the Microgravity Science Glovebox (MSG) of the International Space Station (ISS). The BXF Facility was carried to ISS on Space Shuttle Mission STS – 133 on February 24, 2011. Nucleate Pool Boiling Experiment (NPBX) was one of the two experiments housed in the BXF. Results of experiments on single bubble dynamics (e.g., inception and growth), multiple bubble dynamics (lateral merger and departure if any), nucleate pool boiling heat transfer, and critical heat flux are described. In the experiments Perfluoro-n-hexane was used as the test liquid. The system pressure was varied from 0.6 – 2.4 atm, pool temperature was varied from 30º to 50ºC, and wall temperature was varied from 40º to 70ºC. The test surface was a polished aluminum disc 1mm in thickness, 85mm in diameter and heated from below with strain gage heaters. Five cylindrical cavities were formed on the surface with four cavities located at the corners of a square and one in the middle.

During experiments the magnitude of mean gravity level normal to the heater surface varied from 1.2 x 10-7ge to 6 x 10-7ge. The results of the experiments show that a single bubble (Fig. 1) can grow to occupy the size of the chamber without departing the heater. During lateral merger of bubbles, at high superheats a large bubble may lift off from the surface but continue to hover near the surface. Neighboring bubbles are continuously pulled into the large bubble. At low superheats bubbles at neighboring sites simply merge to yield a larger bubble. The larger bubble mostly locates in the middle of the heated surface and serves as a vapor reservoir. The latter mode continues to persist when boiling is occurring all over the heater surface. Steady state nucleate boiling and critical heat fluxes are found to be much lower than those obtained under earth normal gravity conditions. The data are useful for calibration of results of numerical simulations. Any correlations that are developed for nucleate boiling heat transfer under microgravity condition must account for the existence of vapor escape path (reservoir) from the heater, size of the heater, and the size and geometry of the chamber.

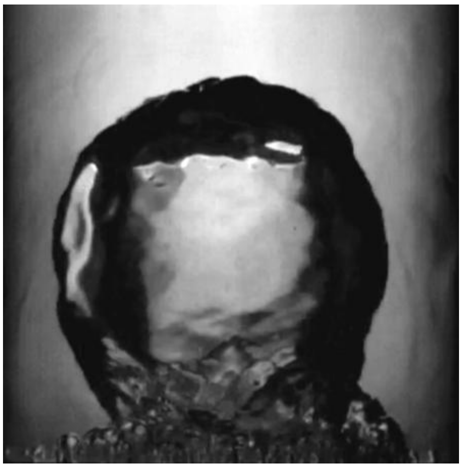


Figure 1: Single Bubble Growth on Space Station.

Reference

Dhir, V.K. "A Mechanistic Study of Nucleate Boiling Heat Transfer Under Microgravity Conditions", NASA Science Requirement Document, Sixth Revision (Jan. 2007)

* This work was supported under the NASA Microgravity Fluid Physics Program.
** NASA Glenn Research Center
† Zinn Technologies, Cleveland, Ohio

IR Based Flow Boiling Heat Transfer Measurements in Earth Gravity and Low-G

Valentin Solotych, Eric Kommer, Channing Tsai, Tae Hoon Kim, Serguei Dessiatoun, and Jungho Kim*

University of Maryland, Dept. of Mechanical Engineering
College Park, MD 20742 USA
*Corresponding author, kimjh@umd.edu

Two NASA workshops (McQuillen-2003, Chiaramonte-2004) highlighted thermal management (and phase change heat transfer in particular) for advanced life support and propulsion as one of the technologies critical for successful deployment of long duration missions. The primary objective of the research is to obtain the data needed to predict the behavior of two-phase flows in geometries relevant to advanced heat exchangers in variable gravity environments.

Unlike previous work where only time- or space-averaged heat transfer data were measured, *local* measurements of the wall heat transfer coefficients are measured. An IR camera is used to visualize fluid flow within a 6 mm ID, 10 cm long silicon tube and to obtain temperature distributions along the inside and outside walls of the tube so local heat transfer coefficients can be computed. The data is used to develop and validate sub-models of the flow within each flow regime. Understanding the heat transfer mechanisms will enable space-based heat exchangers to be designed with much more confidence. Although this work is primarily aimed at thermal management subsystems and development of lightweight evaporators, it also impacts other systems for novel power and transmission, advanced chemical propulsion, and energy storage.

The test rig will be used to obtain time resolved measurements of local heat transfer with the bulk fluid flowing upward, downward, and in low-g. Because the flow is nominally axisymmetric in all three of these configurations (unlike if the flow were flowing horizontally) the only difference between the three flows is the magnitude of the gravity vector, allowing the influence of gravity alone to be isolated. The parameters to be varied are the inlet flow rate, inlet liquid subcooling, and wall heat flux, and will produce flow regimes varying from bubbly flow, slug flow, churn flow, annular flow, inverse annular flow, and ultimately dryout. The time varying, local wall temperature will be measured in each of these flow regimes using an IR camera, from which the local heat flux will be obtained during postprocessing. The measured heat flux is used to validate models of flow boiling. The ultimate goal of the proposed research is to develop the capability to predict what the heat transfer capability of a particular flow configuration would be.

The main test loop includes the test chamber, preheater, flow meter, accumulator, gear pump and a heat exchanger. The silicon tube (Figure 1) was electrically heated and the power determined using a 4-wire technique. Preliminary data in high-g and low-g conditions were obtained in September, 2010 on a 727 aircraft through the NASA FAST program. These tests were used to obtain information regarding the test loop–measurements of tube temperature were not obtained during these runs.

Figure 1: Photograph of silicon tube (left) and IR image of bubble flow in high-g (middle) and inverted annular flow in low-g (right).

In earth gravity or high gravity, the liquid experiences a body force that can overcome the vapor pressure within the film, enabling the liquid to wet the wall for a longer distance from the inlet (Figure 1 middle). In low-g, this body force does not exist so the liquid can move off the surface more easily and inverted annular flow can occur at lower qualities and heat fluxes than in high-g (Figure 1, right). In the lower end of the tube, bubbly flow is visible, but the flow is observed to detach about 25% up from the inlet of the tube. In addition to inverted annular flow, bubbly flow and dryout were observed. Since the maximum flow rate that was performed at both days (0.35m/s) was relatively low, changes in bubble size and bubble behavior were clearly noticeable between the transitions from high to zero-g.

References

McQuillen, J., Rame, E., Kassemi, M., Singh, B., and Motil, B,"Results of the Workshop on Two-Phase Flow, Fluid Stability and Dynamics: Issues in Power, Propulsion, and Advanced Life Support Systems", NASA/TM-2003-212598.

Chiaramonte, F.P. and Joshi, J.A., "Workshop on Critical Issues in Microgravity Fluids, Transport, and Reaction Processes in Advanced Human Support Technology", NASA/TM—2004-212940.

Analytical model for liquid-vapour distribution in a Loop-Heat-Pipe evaporator

Hai-Trieu Phan [a,b,*], Sébastien Dutour [a], Vincent Platel [a] and Amaury Larue De Tournemine [b]

[a] LAPLACE, Université Paul Sabatier, 118 Route de Narbonne, 31062 Toulouse Cedex 9, France
[b] CNES, 18 avenue Edouard Belin, 31 401 TOULOUSE CEDEX 9, France
[*] E-mail: htmimo@gmail.com

A Loop Heat Pipe (LHP) is an efficient two-phase heat transfer device, which can operate under counter gravity conditions. In these systems, heat is mainly removed by liquid-vapour phase change processes such as evaporation, boiling and condensation, with a fluid flow generated by capillary actions. The LHPs were firstly developed for space applications, e.g. in space aboard satellites. Today, they are also employed in ground-based fields, especially for microelectronic cooling.

The LHP essentially starts by applying heat load at the evaporator where vapour is produced at the interface of the so-called primary wick. This will lead to capillary actions at the wick pores, allowing the fluid circulation in the loop. Indeed, the produced vapour will move out of the evaporator and thus circulates along a vapour line up to the condenser (cf. Figure 1), where it is condensed to liquid by ejecting heat to a cold source. If we look in details the fluid flow in the evaporator (the key element of the loop) as shown in Figure 2, the liquid circulates inside a bayonet before entering the evaporator core. Hence, it will be heated by parasitic heat fluxes from the primary wick and thereby can be evaporated. The feedback between heat and mass transfer through the capillary structure and the phase distribution in the evaporator core leads to a high sensitivity of the loop performance to the phase structure in the core and the evaporator cavity. As a consequence, LHP becomes sensitive to all the parameters and processes likely to affect the phase distribution in the core such as: microgravity, orientation vs. gravity for the ground applications, accelerations, etc.

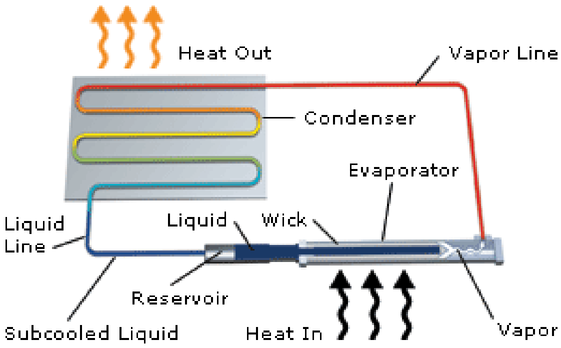


Figure 1: Illustration of different components of a Loop Heat Pipe.

Over the past decade, some experimental study have attempted to highlight the phase distribution in the evaporator core by direct visualisation (Bai et al., 2010), (D'entremont et al., 2008), but only a few theoretical approaches have been made to predict the heat and mass distributions inside the evaporator (Hoang and Ku, 2003), (Soler, 2009), (Dutour et al. 2010).

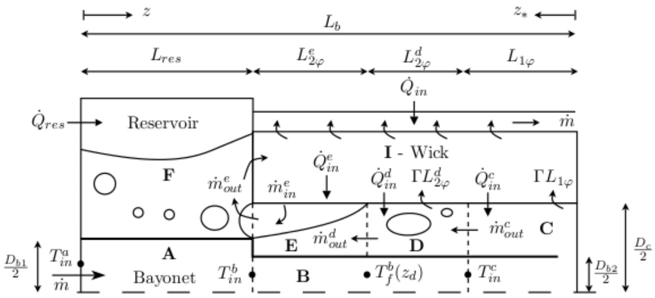


Figure 2: Schematic view of the fluid circulation and the heat distribution inside the evaporator.

In this study, an analytical model is suggested to simulate the evaporator at steady-state conditions. In particular, this model deals with different physical behaviors which could occur in the evaporator core such as: evaporation and condensation of fluid in the core; as well as invasion of subcooled liquid into the reservoir. The main physical mechanisms which impact the LHP operating temperature and the phase distribution in the evaporator core will be identified; and the parasitic heat fluxes will be determined. The effects of additional heat input applied at the reservoir will be also discussed.

References

Bai L., Lin G., and Wen D., Modeling and analysis of startup of a loop heat pipe, Applied Thermal Engineering, vol. 30, no. 17-18, pp. 2778 – 2787 (2010)

D'entremont B.P. and Ochterbeck J.M.,Investigation pf LHP startup using a liquid core visualisation, ASME Summer Heat Trasnfer Conference, Jacksonville, Florida USA, August 10-14 (2008)

Dutour S., Soler P., Platel V., Joly J.L., Tadrist L., Larue de Tournemine A., A theoretical investigation of the influence of phase distribution in the evaporator cavity on the performance of a LHP, 15th International Heat Pipe Conference (15th IHPC), Clemson (USA), April 25-30 (2010)

Hoang T. and Ku J., Heat and mass transfer in LHP, ASME Summer Heat Transfer Conference, Las Vegas, July (2003)

Soler P., Modélisation thermo-hydraulique des boucles à pompage capillaire (de type LHP) : influence de la répartition du fluide sur les transferts de chaleur et de masse au sein de l'ensemble cœur/réservoir. PhD thesis, Université Paul Sabatier-Toulouse (2009)

Thermal perfomances of flexible low bond number loop two phase thermosyphons

S. Filippeschi, A. Franco

Department of Energy and Systems Engneering (DESE)
University of Pisa
Largo L Lazzarino, 2, Pisa, Italy
Tel/Fax +39.050.2217153; E-mail: s.filippeschi@ing.unipi

If for high power applications the two-phase thermal loop has been highly investigated in the literature, thermal loops for power less than 1 kW has been scarcely studied. They could be divided into two main groups: high Bond number loops, where the capillary force can be neglected, and low Bond number loops, where the capillary forces cannot be neglected at all (Filippeschi & Franco, 2010). This paper deals with the effect of the geometry of the sections connecting the evaporator and the condenser, on the heat transfer rate in a low Bond number two-phase thermosyphon made of flexible pipes.

A low Bond number loop two-phase thermosyphon is also referred in the literature as single turned Pulsating Heat Pipe. These loops are characterized by a natural circulation of mass flow inside. They consist of: an evaporator, a condenser and two adiabatic sections (the riser and the downcomer) which connect the two exchangers. In many applications like as PV solar cells cooling the condenser and the evaporator can be out of alignment, so that the use of the flexible pipes seems to be necessary.

Several papers in the literature have experimentally shown that the heat transfer rate, which can be transferred from the evaporator to the condenser section, mainly depends on few parameters like as: the diameter of the tube, the working fluid, the adiabatic section length, the number of the turns and the operative orientation with respect to the gravity. However the thermal and hydraulic oscillations in these devices can not be accurately predicted and the heat and mass transfer phenomena is still unsolved problem. The use of flexible pipes for the adiabatic section introduce additional problems because the evaporator and the condenser sections cab be placed in any configuration, even out of alignment. In this configuration additional forces and additional pressure drops can decelerate or accelerate the motion. In addition the use of unconstraint pipes could develop damping pressure oscillations with different frequencies.

In particular in this paper the authors wish to understand the effect of the geometry of these sections on the slugs and plugs motion inside the loop, and in particular of the bends and of the extension of the horizontal section.

The experimental facility consists of a single closed loop of capillary dimensions. In particular the evaporator section is copper plate where a long circular hole has been realised (Internal Diameter: 2.0 mm, OD: 4.0 mm). The condenser section is a pipe made of brass (ID: 2.1 mm,OD: 3.0 mm). It is designed to facilitate parametric investigations as well as simultaneous flow-visualization. The condenser and the evaporator are connected with flexible pipes made of PTFE (ID: 2.0 mm,OD: 3.0 mm). The evaporator section is heated by two flat electrical heaters. The condenser section, made of a hollow copper chamber, is supplied with cooling water from a constant temperature bath. Adequate thermal insulation is provided on all sections. The working fluid is water. The motion of the plugs and the slugs is observed with a high-speed acquisition video-camera. The unstable two-phase flow of a single turned PHP has been qualitatively described in (Khandekhar et al, 2009) (Di Marco & Filippeschi, (2010). This paper shows the effect of the alignment distance on the thermal resistance in a upward heat transfer mode. Considerations on the increase of the instability and the reduction of the mass flow rate circulating with the flexible pipes have been drawn.

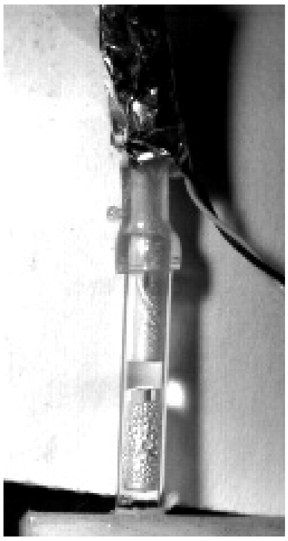


Figure 1: Vapour plug in the water PHP with flexible pipes.

References

Sauro Filippeschi, Alessandro Franco, Experimental analisys of heat and mass transfer in small dimension, two phase loop thermosyphons, Heat Pipe Science and Technology, Vol. 2,pp. 163-182, (2010).

Paolo Di Marco, Sauro Filippeschi, Pulsating heat pipes with flexible adiabatic section: first experimental results, Proceedings of 15th International Heat Pipe Conference (15th IHPC), Clemson USA, (2010).

Sameer Khandekar, A. P. Gautam, P. K Sharma., Multiple quasi-steady states in a closed loop pulsating heat pipe, International Journal of Thermal Sciences, Vol. 48, pp. 535–546, (2009).

Numerical simulation of heat transfer across fluidic interfaces in liquid bridge

Yuri Gaponenko and Valentina Shevtsova

Microgravity Research Center, Service de Chimie Physique E.P. , Universite Libre de Bruxelles
Av. F. D. Roosevelt, 50 CP 165/62, Brussels, 1050, Belgium
ygaponen@ulb.ac.be, vshev@ulb.ac.be

Heat/mass transfer on the moving gas-liquid interface is an important subject directly related to many industrial applications from crystal growth to cooling of electronic devices. There exist a large number of studies of two-phase flow through the circular pipe. Another well studied problem is two-phase flow in annuli when liquid films covered internal rigid wall of a cylinder and gas stream is moving along the center of the channel. Such systems have a large field of applications in the production of highest quality semi-conductors, fibers, micro-jets, etc.

Studies on the mechanisms governing the flow dynamics and heat transfer are particularly helpful in controlling the crystal growth process and thereby the quality of its products. The best quality semi-conductor crystals are grown by floating zone technique. The scientific model, which is used to mimic industrial floating zone technique, is called a liquid bridge: the fluid volume is held by surface tension between two differentially heated horizontal flat coaxial disks. Flow sets in the liquid due to variations of the surface tension along the free surface, and evolves under the combined action of buoyancy and thermocapillarity. The two-dimensional steady flow becomes unstable when the temperature difference exceeds a certain threshold, and the first instability of the basic state is through Hopf bifurcation (oscillatory) in the case of high-Prandtl-number liquids. As a rule, in numerical studies the flow is considered inside liquid, and the boundary conditions on the free interface capture the impact of surrounding gas through the Biot number.

The topic of our study is dynamics of the liquid bridge induced by interfacial phenomena via thermocapillary (Marangoni) and gas shear stresses. The aim of this investigation is concerned to space experiment JEREMI (Japanese European Space Research Experiment on Marangoni Instabilities) which is devoted to the study of the threshold of hydrothermal instabilities in two-phase systems in cylindrical geometry. The experiments is planned to be performed in 2013 in the Japanese module on ISS using the dedicated FPEF (Fluid Physics Experiment Facility).

We consider the system when the gas enters into the annular duct and entrains initially quiescent liquid. The problem is solved numerically in complex geometry, which corresponds to a liquid bridge axially placed into an outer cylinder with solid walls. The internal core consists of solid rods at the bottom and top, while the central part is a relatively short liquid zone filled with viscous liquid and kept in its position by surface tension. The solid rods have a complex geometry; they contain grooves and protrusions at the bottom and top parts closest to the liquid.

Our recent achievements deal with isothermal case of this problem. Gas-liquid flows in annulus were analyzed for fluids in large range of viscosity ratios in (Gaponenko et al. 2010). The favorable comparison between numerical and experimental results for different gas flow rates is recently presented by Gaponenko et al. 2011.

In the present study the gas-liquid flows induced by gas stream in annulus and Marangoni convection on free surface are analyzed for fluids in large range of gas flow rate **Q**, temperature difference **ΔT**, geometry of the system and volume ratio of the liquid bridge **VR**. The behavior of the liquid flow under mechanical stresses produced by co- or counter flow of the ambient gas is one of the points which were studied (see Fig. 1). Depending on gas flow rate and temperature difference between rods four different flow regimes were classified including oscillatory. For each flow regime the heat fluxes through the surface have been analyzed. The flow pattern as well as heat transfer through the interface is strongly depends on the shape of the interface. Our previous experimental studies and numerical analysis have shown that dynamic deformation is negligibly small, Gaponenko et al, 2011b. The static deformation of the free surface is taken into account in the numerical simulations.

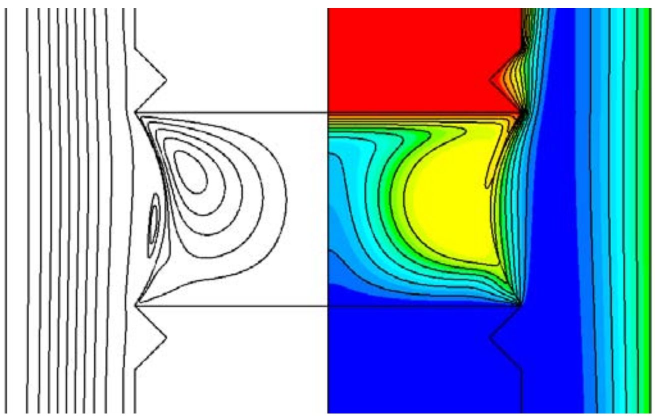


Figure 1: Level lines for stream function (left side) and temperature (right side) for **Q** = 2.057 ml/s, **ΔT** = 20 K**, VR** = 0.8.

References

Gaponenko Yu., Ryzhkov I.I., Shevtsova V., On flows driven by mechanical stresses in a two-phase system. FDMP, vol.6, no.1, pp.75-97, (2010)

Gaponenko Yu., Mialdun A., Shevtsova V., Liquid entrainment by gas flow along the interface of a liquid bridge. Accepted to European Physical Journal Special Topics (EPJ ST), vol. 192, (2011a)

Gaponenko Yu., Glockner S., Mialdun A. and Shevtsova V., Study of a liquid bridge subjected to interface shear stress, submitted to Acta Astronautica, (2011b)

Numerical Simulation of Nucleate Boiling Coupled with the Thermal Response of the Solid

Eduardo Aktinol and Vijay Dhir

Department of Mechanical and Aerospace Engineering
University of California, Los Angeles, CA 90095
vdhir@seas.ucla.edu

Nucleate boiling is the regime in boiling heat transfer where bubbles nucleate, grow, and detach from a heated solid surface. Due in part to its efficacy, nucleate boiling has been investigated extensively over the course of decades. Nevertheless, due to its complexity, a mechanistic prediction of nucleate boiling heat transfer remains elusive.

Numerical simulations and experiments were performed by Son et al (1999) to investigate the dynamics and heat transfer associated with a single bubble during nucleate boiling on a horizontal surface. Although the scope of the experiments was more limited than that of the simulations, there was good agreement between the experimental and numerical results. However, the numerical model employed assumed constant and uniform surface temperature, thereby decoupling the thermal response of the solid surface and imposing a constant waiting period between successive bubble nucleations that was independent of other variables such as wall superheat and contact angle. Williams and Mesler (1967) showed experimentally that waiting periods can vary significantly by simply changing the orientation of the boiling surface.

The present work describes a complete numerical simulation of single bubble dynamics coupled with the thermal response of a horizontal surface of a solid of finite thickness. Two dimensional, axis-symmetric, finite difference schemes are used to solve the governing equations in the solid, liquid, and vapor phases. The interface between the liquid and vapor phases is tracked by a level set method. An iterative procedure is used at the interface between the solid and fluid phases in order to match temperatures and heat fluxes. Parametric studies are carried out to determine the dependence of waiting period on other variables. A constant heat flux is applied as boundary condition at the solid substrate's base, and calculations are conducted for varying contact angles and gravity conditions.

As can be concluded from Figure 1, varying contact angles is found to affect temperature fluctuations, growth and waiting periods significantly. The minimum waiting periods are obtained for different contact angles while keeping other parameters such as applied heat flux and material properties unchanged. The results show that the growth period increases monotonically with increasing contact angle. The behavior of the waiting period can be related directly to the amount of heat transfer contributed by the microlayer and the overall improvement in heat transfer coefficient caused by the growth and departure of bubbles of different sizes.

Although they take longer to grow, more liquid must be evaporated in order to form larger bubbles and thus more energy must be supplied by the wall and boundary layer. Additionally, larger bubbles cause more agitation of the liquid during growth and after departure, resulting in better heat transfer. The tradeoff between departure diameter and growth rate is quantified to show that at the end of a growth cycle, the waiting time can be related to the total energy removed from the wall during bubble growth, which can be in turn related to the varying contact angle.

Reducing gravity is found to have a large effect on growth period but a small effect on waiting period. Gravity is the main force tending to dislocate the bubble from the surface. Therefore, it has a direct effect on bubble departure diameter and growth period. However, the solid substrate's temperature recovery mechanism remains unchanged and the only effect from reducing gravity is indirect through the different velocity field generated by the departing bubble.

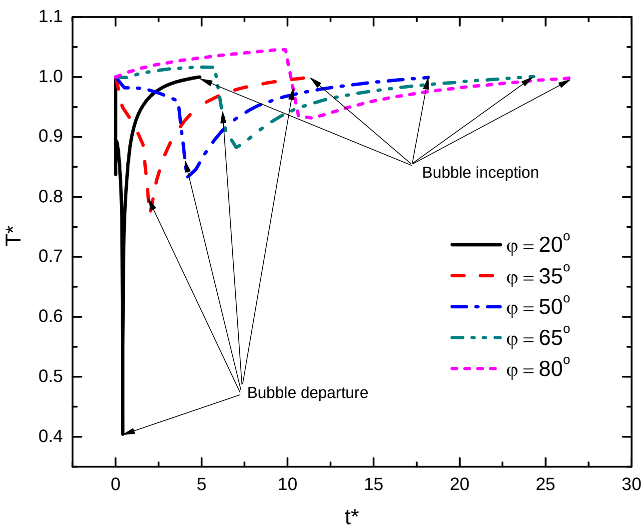


Figure 1: Temperature fluctuations at the cavity site for various contact angles during nucleate boiling of water on a stainless steel surface at one atmosphere pressure

The coupling of the solid substrate's thermal response to the simulation of nucleate boiling has offered insight into the dependence of growth and waiting periods on contact angle and gravity level.

References

Son G., Dhir V.K., Ramanujapu N., Dynamics and Heat Transfer Associated With a Single Bubble During Nucleate Boiling on a Horizontal Surface, Journal of Heat Transfer, Vol. 121 pp. 623-632 (1999)

Wlliams D.D.., Mesler R.B., The Effect of Surface Orientation on Delay Time of Bubbles from Artificial Sites during Nucleate boiling, A.I.Ch.E. Journal, pp. 1020-1024 (1967)

The effect of orifice on flow boiling instability in a single horizontal microtube

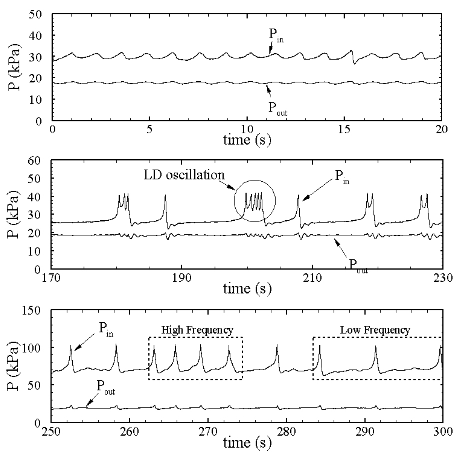
Yanfeng Fan, Ibrahim Hassan

Concordia University, Faculty of Engineering and Computer Science, Department of Mechanical and Industrial Engineering,
1515, St. Catherine St. West, EV4.139, Montreal, H3G 1M8 Canada
Ibrahimh@alcor.concordia.ca

Flow instability of boiling heat transfer in microchannels is a very important issue since it affects the local heat transfer characteristics and may cause boiling crisis. A promising attempt to reduce flow instabilities is by using inlet restriction; frequently defining as orifice.

Kandlikar et al. (2006) addressed the basic causes of instabilities and studied the effect of pressure drop elements. Their pressure drop elements were positioned at the inlet with area reduction of 50% and 4%. Results showed that the element with 50% reduction did not stop the reversed flow however it reduced its intensity; as for the element with 4% reduction oscillations vanished completely. Wang et al. (2008) proposed three types of inlet-outlet configurations in a parallel microchannel heat sink with trapezoidal cross-section. The orifice to the microchannel area was 20%. Results showed that reversed flow was eliminated; however, pressure drop was increased.

In this work, flow instability in a single microtube with inner diameter of 889 μm was investigated. Three area reductions of 20%, 35%, and 50% were selected. The effects of orifice on flow oscillation, pressure drop, and heat flux at the onset of flow stability under uniform heating were studied. Operating conditions ranged from 700 - 3000 $kg/m^2 \cdot s$ for the mass flux, 6 - 27 W/cm^2 for the heat flux, and dielectric coolant FC-72 was selected as the working fluid. In the absences of an orifice at the inlet, four oscillation types were observed at the onset of flow instability as shown in Figure 1; it was also noticed that the frequency of oscillation increased with the heat flux while keeping constant amplitude as seen in Figure 2. Adding the orifices helped in stabilizing the flow without generating significant pressure drop at the same operating conditions. The orifice of 20% area ratio showed better performance at low mass fluxes (< 1000 $kg/m^2 \cdot s$). Whereas, at high mass fluxes (> 2000 $kg/m^2 \cdot s$), the orifices with the area ratios of 50% and 35% were capable of efficiently stabilizing the flow or delaying the onset of flow instability as seen in Figure 3.



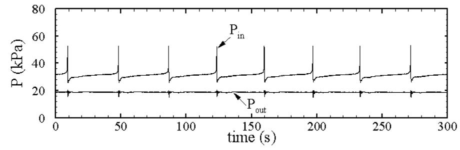


Figure 1: The pressure oscillation. (a) LD. (b) LDP. (c) TPD. (d) Pressure-drop oscillation.

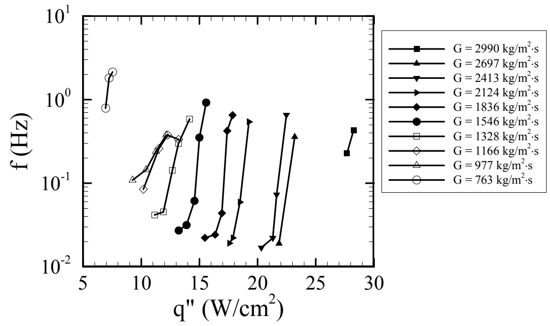


Figure 2: The effect of heat flux on oscillation frequency at different mass fluxes.

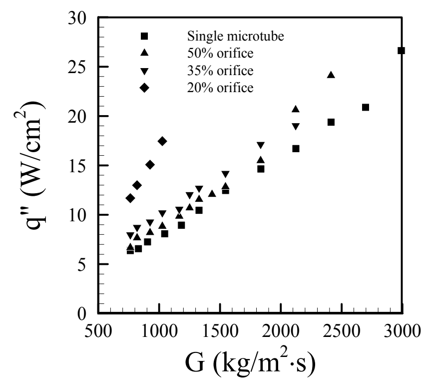


Figure 3: The heat flux at the onset of flow instability in the microtubes with different orifice sizes.

Considering the pressure drop, the orifice with large area ratio is recommended at high mass fluxes and the orifice with small area ratio is suitable for lower mass fluxes. Implementing orifice in the design of microchannel heat sink has the potential to stabilize the flow without active control.

References

Kandlikar, S. G., Kuan, W. K., Willistein, D. A., Borrelli, J., Stabilization of flow boiling in microchannels using pressure drop elements and fabricated nucleation sites, Journal of Heat Transfer Vol. 128 pp. 389-96 (2006)

Wang, G., Cheng, P., and Bergles, A. E., Effects of inlet/outlet configurations on flow boiling instability in parallel microchannels, Int. Journal of Heat and Mass Transfer Vol. 51 pp. 2267-2281 (2008)

Equilibrium estimation of the ground test loop system for interfacial behaviors and heat transfer characteristics in boiling two-phase flow (TPF)

Haruo Kawasaki, Kiyosumi Fujii, Kenichiro Sawada, Takashi Sakai, Soumei Baba, Yasuhisa Shinmoto, Hitoshi Asano, Osamu Kawanami, Koichi Suzuki, Haruhiko Ohta

JAXA Thermal System Group, 2-1-1 Sengen, Tukuba-shi, Ibaraki JAPAN
E-mail : kawasaki.haruo@jaxa.jp

Abstract

A two-phase mechanical pumped loop is appropriate for the thermal control system and the thermal management system which can treat a large amount of west heat. The two-phase mechanical pumped loops for ground use are general, while it is not general for space. As one of the reasons, gravity effects on the two-phase flow phenomena and on heat transfer characteristics have not been clarified in detail.

A project for the experiment on ISS which is named as "Interfacial behaviors and heat transfer characteristics in boiling two phase flow (TPF) has become an official project in JAXA. This experiment uses a two-phase mechanical pumped loop.

Ground test loop model (GM) for TPF

A ground test loop for TPF to verify the experiment on ISS was assembled as shown in Figure 1. The ground test loop was installed horizontally to simulate microgravity conditions.

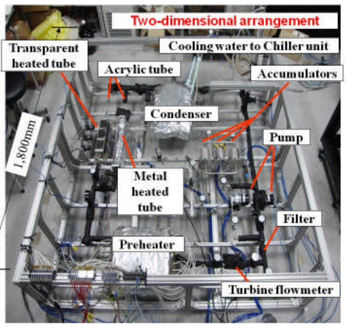


Figure 1: Schematic of ground test loop model (GM) for TPF

Equilibrium test

Assuming one day operation on the orbit, the thermal equilibrium test with 300W of heat load and with mass velocity of G=300 kg/s m^2 was conducted continuously for 8 hours. During the operation for 8 hours, there was little change in the temperature distribution in the test loop.

The fluid temperature at the outlet of metal heated tube was 67.95± 0.05 ^{o}C. The temperature was very stable and higher than the saturation temperature at atmosphere of FC-72. Inlet and outlet temperature of condenser was 61.80±0.05 ^{o}C and 22.4± 0.2^{o}C, respectively. All vaporized working fluid at the metal heated tube was condensed at the condenser, because the outlet temperature of condenser was lower than the saturation temperature. The pressure at the pump outlet was 155± 4 kPa, and the pressure in accumulator was 110.7 ± 0.1 kPa.

Surface temperature on the outlet side of metal heated tube was about 76 ^{o}C. A basic heat transfer data was obtained with high accuracy because of the adiabatic design surrounding the heated metal tube. There was the temperature difference between outlet of heated metal tube and inlet of condenser owing to the heat loss from the test loop to the atmosphere.

When the heat load on the metal heated tube was increased at G=300 kg/s m^2, the fluid temperatures in the test loop varied as shown as Figure 2.

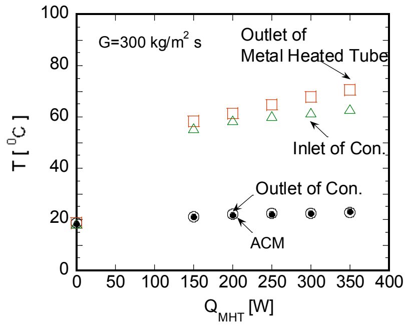


Figure 2: Effects of MHT heat load on the fluid temperatures.

The operating conditions in the each point of the test loop at Q_{MHT} =300W were plotted on the *P-T* diagram with saturated pressure curve of FC72 as shown in Figure 3. The pressure between outlet of the metal heated tube (MHT) and inlet of the condensation tube decreased according to the saturated pressure curve, owing to the flow resistance mainly at mixing chamber located between them. In this condition, the pump head was 48 kPa. If the pressure loss at mixing chamber can be reduced, the pump head decreases to 27 kPa at least, and the outlet temperature of MHT decreases from 68 ^{o}C to 63 ^{o}C. At higher heat load, the reduction of flow resistance by the modification or removal of the mixing chamber becomes more important to suppress the increase in both of the temperature at the MHP outlet and the pump head.

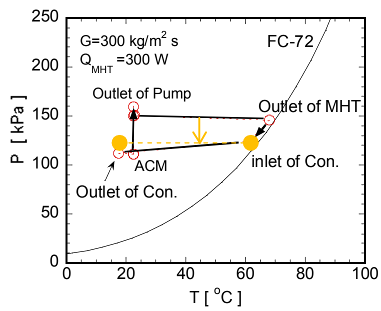


Figure 3: Relationship between temperature and pressure in the test loop

References

K.Fujii, H. Kawasaki, H. Ohta," Recent Progress of International Space Station KIBO Project - TPF-", Fifth International Topical Team Workshop on TWO-PHASE SYSTEMS FOR GROUND AND SPACE APPLICATIONS (2010)

Influence of surface tension models on the hydrodynamics of falling films in Volume of Fluid-Simulations

Christoph Albert, Dieter Bothe

Technische Universität Darmstadt
Center of Smart Interfaces
Mathematical Modeling and Analysis
Petersenstrasse 32, 64287, Darmstadt, Germany
albert@csi.tu-darmstadt.de
bothe@csi.tu-darmstadt.de

In this contribution, the hydrodynamics of laminar films flowing down a vertical plane, driven by the gravitational force, is investigated by means of Direct Numerical Simulation. Falling films are employed in a large variety of applications, e.g. in the food industry, in the pharmaceutical industry or in nuclear plasma reactors. Their main advantages are the good heat transfer characteristics and the small hold-up. Moreover, their large surface facilitates the mass transfer between liquid and gas.

The basic flow is modelled by the two-phase Navier-Stokes equations. Finite Volumes are used for the spacial discretization, and the interface is implicitly captured by the Volume of Fluid (VOF) method, together with the Piecewise Linear Interface Calculation (PLIC) technique, that is used to keep the interface sharp.

A falling film is hydrodynamically unstable, and perturbations will evolve into waves for every Reynolds number. In the numerical simulation, surface waves of a defined wave length are generated by an artificial excitation of the velocity at the inlet. At low excitation frequencies, these perturbations evolve into rollwaves of large amplitude, preceded by smaller capillary waves. At higher frequencies, they evolve into sinusoidal waves.

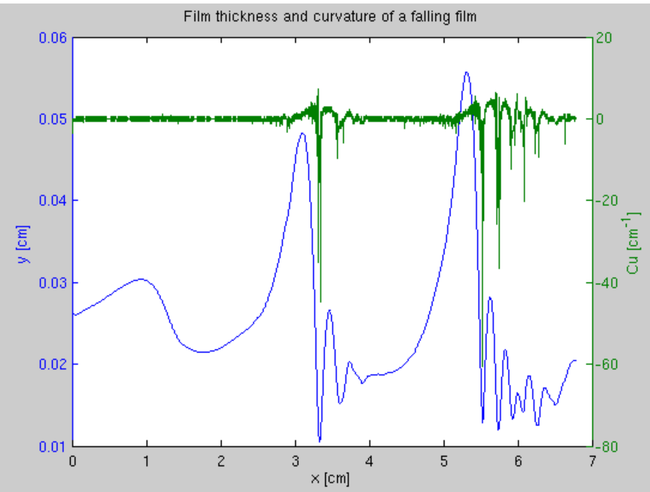


Figure 1:Film thickness and curvature of a falling film

The main numerical difficulty in the simulation of wavy falling films is that the curvature of the film surface in the capillary wave region is large. VOF simulations often have problems with large curvatures, since these tend to introduce spurious oscillations, so called parasitic currents, into the velocity field. These parasitic currents tend to increase in amplitude after grid refinement, and thereby prevent simulations with a grid sufficiently refined to resolve the relevant hydrodynamics.

Computations with two different approaches for the modelling of surface tension effects in VOF simulations are compared, namely the Continuum Surface Stress (CSS) model and the balanced Continuum Surface Force (bCSF) model with height functions. It is shown that the choice of the surface tension model heavily influences the vortex structure inside the film.

By careful comparison of the simulations to experimental and theoretical results, we show that the hydrodynamics in CSS simulations is not reliable, whereas bCSF delivers good results that converge after grid refinement.

The hydrodynamics of bCSF is thoroughly validated and can serve as a good basis for the simulation of heat and mass transfer processes inside a falling film.

This work is supported by the DFG International Research Training Group 1529 *Mathematical Fluid Dynamics* and the Center of Smart Interfaces at Technische Universität Darmstadt.

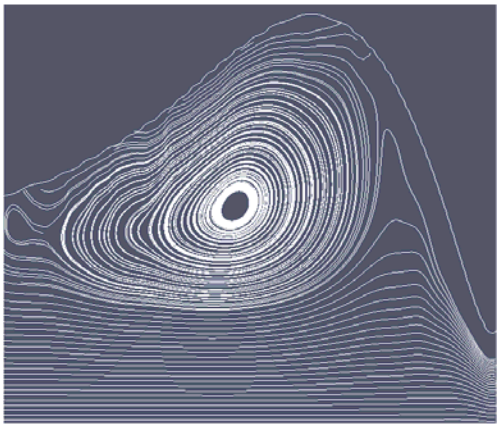


Figure 2: Vortex inside a roll wave, bCSF

Contribution to the rigorous 3D thermo-hydraulic modeling of grooved micro heat pipes

David Mikaelian and Pierre Colinet

Université Libre de Bruxelles, Faculty of Applied Sciences, Laboratory TIPs (Transfers, Interfaces and Processes)
Avenue F.D. Roosevelt 50, Brussels, 1050, Belgium
dmikaeli@ulb.ac.be and pcolinet@ulb.ac.be

Grooved heat pipes are heat transfer devices, based on a liquid-vapor phase change, with a very high effective thermal conductivity. Due to their high efficiency and their small size and low mass, heat pipes are especially used in the space field as cooling devices. Heat pipes are generally constituted by three sections: an evaporator section, an adiabatic section and a condenser section. Heat is applied to the evaporator section in order to vaporize the liquid, the vapor then flows towards the condenser section where it releases energy by condensing. Finally the liquid comes back from the condenser section towards the evaporator section by means of a capillary force due, in the case considered here, to the presence of axial grooves.

Many models have been developed in the past in order to investigate the functioning of heat pipes and to characterize their maximal heat transfer capacity. However, these models are either purely hydraulic, or purely thermal, or hydraulic combined with heat transfer coefficients, but to our knowledge there is a lack of models combining the rigorous resolution of the coupled hydraulic and thermal problems (including the solid). The objective of this work is to fill in this gap by modeling the phenomena of transport of matter, momentum and energy within a model heat pipe, starting from the full equations of continuity, Navier-Stokes and heat transport. This modeling enables to predict the profiles along the heat pipe for the height of liquid, the mean velocity of the flow and the temperature, for given heat flux distribution and total liquid volume. It also highlights the influence of the parameters of the heat pipe on its performances, including the maximal heat transfer capacity.

The system that is studied is a single triangular axial groove in microgravity conditions and is described in Fig. 1.

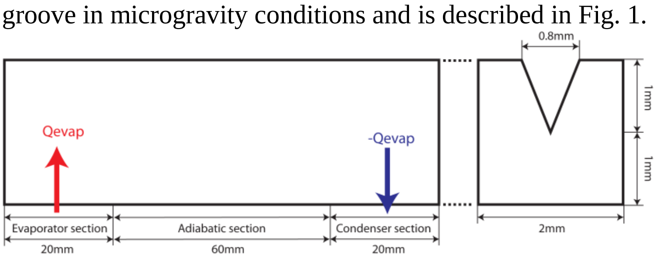


Figure 1 : Sketch of the model single-groove heat pipe

By combining the lubrication approximation and the one-sided approach with simulations of the flow and the heat transfer using the software COMSOL, our approach leads to the following 2nd-order differential equation for the scaled height $H(X)$ of liquid along the heat pipe:

$$Q(X) = C\frac{d}{dX}\left[H^2(X)\frac{dH(X)}{dX}\right]$$

where X is the (axial) coordinate along the heat pipe, $Q(X)$ the given heat flux profile along the heat pipe, and C is a negative constant that can be calculated by means of simulations on COMSOL, given the geometrical parameters of the heat pipe and the contact angle formed by the liquid in the groove (assumed constant).

By solving this equation for different uniform heat fluxes input in the evaporator section, a wetting angle of 30° and a volume of ethanol of $10^{-8}m^3$ inside the groove, we can establish the profiles along the heat pipe for the scaled height of liquid (see Fig. 2) and for the averaged axial velocity of the flow.

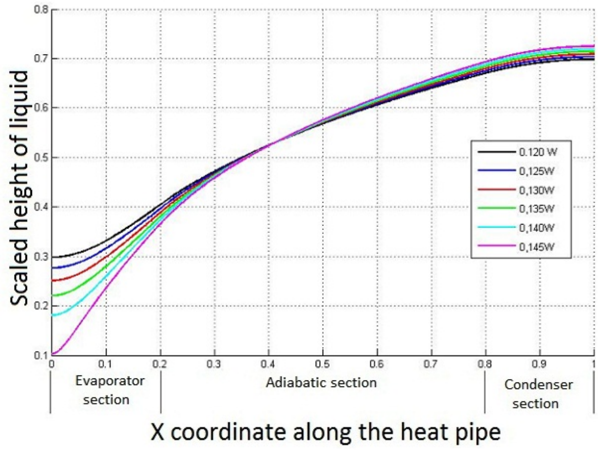


Figure 2: Profile of the scaled height of liquid along the heat pipe for typical operating conditions.

The equation of energy conservation can then also be solved by using the software COMSOL, for the liquid height distribution just determined. It leads to the temperature field in each section of the heat pipe, and hence to the 3D temperature field, both in the solid and in the liquid.

The proposed approach also enables to determine the maximum heat flux that can be applied in the evaporator section without reaching dry-out. This is done by examining the variation of the profile of the height of liquid along the heat pipe when we increase the heat flux applied to the evaporator section. Figure 2 illustrates this procedure. Expectedly, we can observe that the height of liquid in the evaporator decreases when the heat flux is increased

Finally, the parametric sensitivity of this maximum heat flux is also analyzed. Indeed, this heat flux depends on both geometrical and thermodynamical parameters, such as the semi-apex angle of the triangular groove and the wetting angle (here assumed to be constant and finite). Perspectives for generalization of the approach are also discussed.

Acknowledgements

The authors gratefully acknowledge financial support of the Fonds de la Recherche Scientifique – FNRS.

Latest developments of the Reference Multiscale Boiling Investigation experiment for the International Space Station

Olaf Schoele-Schulz [a)], Nils Schweizer [a)], Gerold Picker [a)],
Hans Ranebo [b)], Jan Dettmann [b)], Olivier Minster [b)], Balázs Tóth [b)], Josef Winter [b)], Christer Fuglesang [b)]

a) ASTRIUM Space Transportation GmbH, Friedrichshafen, Germany;
b) ESA-ESTEC, Noordwijk, The Netherlands

Institutes involved in the Reference Multiscale Boiling Investigation Spece Experiment:
Laboratoire IUSTI – UMR / CNRS 6595, Merseille, France;
Technical Thermodynamics, Technische Universität Darmstadt, Germany;
University of Pisa, Italy;
Institut de Mécanique des Fluides de Toulouse, France;
ENEA, Casaccia, Italy;
Ecole Polytechnique Fédérale de Lausanne, Switzerland;
Microgravity Research Center, Université Libre de Bruxelles, Belgium

Boiling is a two-phase heat transfer process where large heat fluxes can be transferred with small driving temperature differences. The high performance of boiling makes the process very interesting for heat transfer applications and it is widely used in industry for example in power plants, refrigeration systems, and electronics cooling. Nevertheless, due to the large number of involved phenomena and their often highly dynamic nature a fundamental understanding and closed theoretical description is not yet accomplished. The design of systems incorporating the process is generally based on empirical correlations, which are commonly accompanied by large uncertainties and, thus, has to be verified by expensive test campaigns. Hence, strong efforts are currently made to develop applicable numerical tools for a reliable prediction of the boiling heat transfer performance and limits.
In order to support and validate this development and, in particular as a precondition, to enhance the basic knowledge about boiling the comprehensive multi-scale experiment RUBI (Reference mUlti-scale Boiling Investigation) for the Fluid Science Laboratory on board the ISS is currently in preparation.
The RUBI experiment benefits from the absence of vapour buoyancy and natural convection in the high quality and long-term microgravity of the ISS. Effects and phenomena like thermocapillary convection that are hardly observable in normal gravity conditions can be investigated. Clearly predefined conditions particularly of the thermal boundary layer at the heating surface can be established without disturbances by natural convection.
The scientific objectives and requirements of RUBI have been defined by the members of the ESA topical team "Boiling and Multiphase Flow" and addresses fundamental aspects of boiling phenomena. The main objectives are the measurement of wall temperature and heat flux distribution underneath single vapour bubbles with high spatial and temporal resolution by means of IR thermography accompanied by the synchronized high-speed observation of the bubble shapes in various conditions.
To characterise the vapour properties and the thermal boundary layer around the bubble, a micro thermocouple array will measure the fluid temperature in the vicinity and inside of the bubbles. The current baseline array features four K-type thermocouples with a hot junction diameter of 45 microns.
A unique feature of the RUBI experiment is the possibility to study the effect of field forces on a single vapour bubble (e.g. bubble deformation, detachment, bubble-wall interaction, etc.). A gravitational-like field force will be created via an electric field above the heated surface and shear will be applied via a flow parallel to the heated surface.

Beside the introduction to the RUBI experiment concept the present contribution will focus on the actual technical status and performance of the experiment. Hardware concepts will be introduced and the results of first tests will be shown.
The industrial activities of the RUBI project are funded and the science team is supported by ESA.

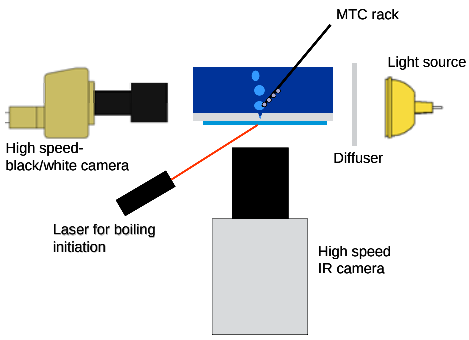


Figure 1: Experimental Setup of Reference Multiscale Boiling Experiment for ISS.

Corresponding Author:

Olaf Schoele-Schulz

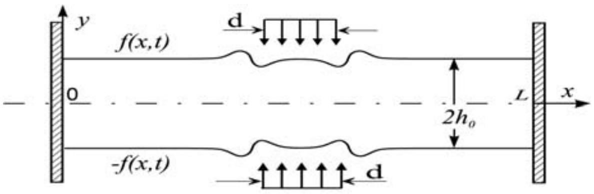
ASTRIUM Space Transportation GmbH
Orbital & Reusable Systems - Payloads
Dpt. TO52
88039 Friedrichshafen / Germany
Tel.: +49 / 7545 / 8-4039
Fax.: +49 / 7545 / 8-4429
olaf.schoele-schulz@astrium.eads.net
http://www.astrium.eads.net/

Features of the rupture of liquid film due the action of thermal load under microgravity conditions. The role of the Prandtl number.

A.S. Ovcharova

Lavrentyev Institute of Hydrodynamics SB RAS, Novosibirsk, Russia
ovcharova@hydro.nsc.ru

We investigate the influence the Prandtl number on the character of the film rupture under the action of thermal load. A weightless thin film is hanging between two flat rigid walls. The film has length L and thickness $2h_0$. At initial moment of time the film is in the rest. The thermal load acts onto the free surface of the film. In this case the film can change its form and have discontinuity. To investigate this process, the two-dimensional mathematical model based on the Navier-Stokes equations was applied.



The thermal load is modeled by prescribed temperature on the free surface in the form:

(1) $\theta(x,t) = \theta^*$, if $L/2 - mh_0 \le x \le L/2 + mh_0$

At the rest of the film surface $\theta(x,t) = 0$.

(2) $\theta(x,t) = \theta^*$, if $L/2 - mh_0 \le x \le L/2 + mh_0$

At the rest of the film surface $\partial\theta / \partial n = 0$.

Here, h_0 is the half of film thickness, m is a positive number which determines a width of thermal ray acting on the free surface of the film. To reveal the character of the film rupture we performed the computational investigations using parameters $Re = 1$; $Ca = 0.025$; $Mn = 30$; $Pr = 0.1 \div 10$. The ratio of film length to its half thickness $L/h_0 = 90$, $m=3$. Because the problem is symmetrical, we consider the half of domain only.

The investigations have shown: if the temperature on the film free surface is prescribed by type (1), i.e. it is predetermined at any time moment, then the lifetime, the character of the film rupture and the location of the free surface are not depend on the Prandtl number. At the same time, the isotherms have essential distinctions for different Prandtl numbers (see Fig. 1).

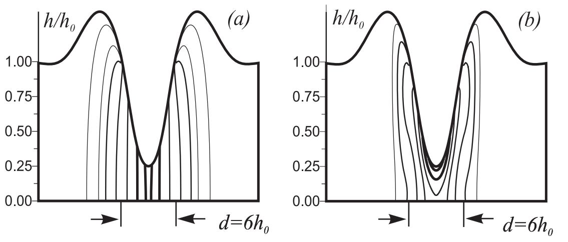


Figure 1. The location of free surface and isotherms for Prandtl numbers $Pr = 0.1$ (a) and $Pr = 10$ (b) at the same time moment. Here, θ^* is the half period of the sine in domain $d=6h_0$; max θ^*=0.5. At the rest of the free surface $\theta(x,t) = 0$.

If the temperature on the free surface is prescribed by type (2), Prandtl number plays a vital part. How it is shown in [1], the film rupture happens with the generation of the drop for such type of thermal load.

Figure 2 presents the results of calculations for case, when the temperature is determined in small domain $d=6h_0$ of film free surface only. At the rest of the film surface we could find it in the process of solution of the problem.

If the thermal load is intense enough (θ^* = 0.75), the film rupture occurs quickly and generated drops have insignificant distinctions for both Prandtl numbers. However, the more fast warming-up of the film no leads to the more quick rupture of the film. Opposite, the lifetime of the film $t^*(Pr = 0.1)$ is 1.5 time more than $t^*(Pr = 10)$ for Prandtl number $Pr = 10$.

The smaller the thermal load, the smaller the difference between the lifetimes of the film for both Prandtl numbers. At that, the location of free surface and form of drops have sharp distinction.

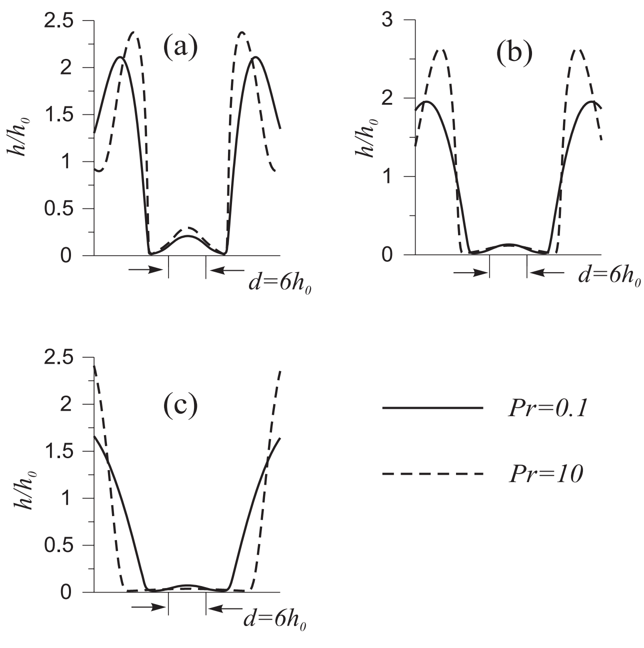


Figure 2. The location of the film free surface for the thermal load of different intensive:
(a): θ^*=0.75; (b): θ^*=0.5; (c): θ^*=0.25

References

A.S. Ovcharova Droplet Formation in the Rupture of a Liquid Film under Action of a Thermal Load, Fluid Dynamics, Vol. 46, No. 1 pp. 108-114 (2011)

The motion of two viscous heat conducting liquids in a cylindrical pipe

Victor K. Andreev

Institute of Computational Modeling RAS, Krasnoyarsk, Russia
E-mail: andr@icm.krasn.ru

The aim of the present work was to theoretical and numerical researches the motion of two immiscible viscous heat conducting liquids with an cylindrical interface [1]. Suppose that the densities ρ_j, $j=1,2$, the dynamic viscosities μ_j and the thermal diffusivities χ_j of the liquids are positive constants. We also assume that there are no external mass forces acting on the liquids. Under these assumptions, the equations of continuity, momentum, energy in the cylindrical coordinates admit one-parameter subgroup of transformations corresponding to the generator $\partial_z + A_j\partial_{\theta_j} - \rho_j f_j(t)\partial_{p_j}$, where A_j are constants and $f_j(t)$ are functions of time. The invariant solution should be sought in the form

$$u_j = v_j = 0, \qquad w_j = W_j(r,t),$$
$$p_j = -\rho_j f_j(t)z + P_j(t), \quad \theta_j = A_j z + T_j(r,t). \tag{1}$$

Solution (1) can be interpreted as follows. Suppose that on the interface $r=a$ between liquids the surface tension linearity depends on the temperature: $\sigma(\theta)=\sigma_0-\kappa\theta$, where $\kappa>0$ is constant. Initially, the liquids are at rest and occupy the cylindrical domains $0<r<a$ and $a<r<b$ respectively. At $t=0$ the temperature field $\theta_j = A_j z$ are created instantly in the whole domains. The thermocapillarity effect and pressure gradients $f_j(t)$ induce the unsteady motion of liquids. In this motion, the interface is represented by the cylindrical surface $r=a$ for all time and the trajectories are straight lines parallel to z axis. Substituting expressions (1) in the governing equations and taking into account the conditions on the interface $r=a$ we obtain the two initial boundary value problems for unknowns (W_1,W_2), (T_1,T_2). These problems can be solved successively.

The following results were obtained:

1) the analytical stationary solution $W_j^0(r)$, $T_j^0(r)$ is found. For some physical and geometric parameters the return flow can be arise due to thermocapillary force as shown in Fig. 1;
2) the a priori estimates of the velocity fields and temperature fields are obtained in uniconvergence space;
3) all functions W_j, T_j tend to the stationary state $W_j^0(r)$, $T_j^0(r)$ as $t\to\infty$ when the pressure gradient in the one of liquid tends to constant;
4) the volume flow rates through the domains are found and inverse problem with respect to pressure gradients is solved;
5) the influence of thermoconcapillarity effect and domains thickness on the flow type is investigated numerically.

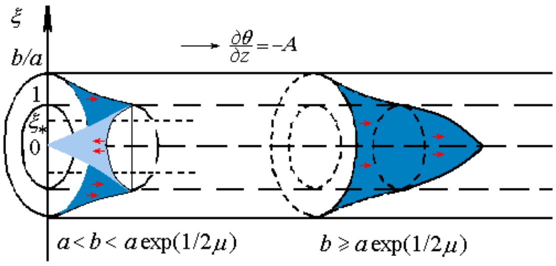


Figure 1: Scheme of steady flow in a pipe, $W_1^0(\xi_*)=0$, $\mu=\mu_1/\mu_2$

The work is supported by the Russian Foundation of Basic Research Grant # 11-01-00283 and the Siberian Branch of Russian Academy of Sciences on integrated project #65.

References

Andreev V.K., Bekezhanova V.B., Stability of nonisothermal fluids, Russia, Siberian Federal University. Krasnoyarsk. P. 356 (2010).

Slow sedimentation of viscoplastic double emulsion drops in a Newtonian fluid

Olga Lavrenteva, Irina Smagin and Avinoam Nir

Chemical Engineering Dept., Technion – Israel Institute of Technology
Haifa. 32000, Israel
ceolga@technunix.technion.ac.il

A compound drop (double emulsion droplet) consists of an inner drop enclosed in a shell of an immiscible liquid. Compound multiphase drops occur in a wide variety of natural processes and technologically significant applications. Double emulsions are employed for sustained and prolonged release of active ingredients in pharmaceutics cosmetics and food technology. Compound drop dynamics is relevant also to the deformation and migration of living cells

Many of applications listed above suggest an involved rheology of compound drop phases. E.g. filled hydrogel particles, which are advantageous to protect encapsulated lipids from degradation in food industry, are composed of known viscoplastic fluids. The latter constitute an important class of non-Newtonian materials. At relatively low levels of stress they behave as elastic or visco-elastic solids and exhibit little or no deformation. When the local stress exceeds a critical yield value they deform as non-Newtonian fluids. To our best knowledge, no theoretical studies of the compound drops with viscoplastic shells, that are the focus of the present paper, are available so far.

In this work we consider a double emulsion droplet consisting of viscoplastic shell and viscous core placed in a Newtonian fluid. Each of the 3 phases has its own density and the system moves and deforms under the action gravity and buoyancy. The material of the shell is a Bingham fluid with a constitutive equation

$$\begin{aligned} &\boldsymbol{\tau} = \left(\eta_{\mathrm{p}} + \frac{\tau_Y}{|\mathbf{D}|} \right) \mathbf{D}, && |\boldsymbol{\tau}| > \tau_Y, \\ &\mathbf{D} = \mathbf{0}, && |\boldsymbol{\tau}| \le \tau_Y, \end{aligned} \qquad (1)$$

where $\boldsymbol{\tau}$ denotes the viscous stress tensor and $\mathbf{D}$ is the rate of deformation tensor, with η_p and τ_Y being the plastic viscosity and the yield stress, respectively. | | is the second invariant of a tensor.

The governing parameter of the problem are the Reynolds number, the ratio of the equivalent radius of the core to that of the drop, *R*, the ratios of the viscosities, the Bingham number, *Bn*, that characterizes the ratio between viscous stresses in the system and the yield stress, two capillary numbers corresponding to the inner and outer interfaces the radii ratio, Ca_{in} and Ca_{out}, and 2 Bond numbers, Bo_{shell} and Bo_{core}.

Assuming that the Reynolds number is negligibly small we describe the fluid motion inside the drop and in the continuous fluid by the Stokes equations and solve the problem formulated above following the general scheme for the quasi-stationary deformations: Suppose that at a given time, the shape of the drop is known, then the velocity and stress fields are found from the steady Stokes equations which are solved making use of a variation of boundary integral method that was developed in Toose et al. (1999). The Green function for the Stokes equation is used and the non-Newtonian stress is treated as a source term. When the velocity at the interface is known, the boundary is advanced according to the kinematic condition and the new shape at the next time step is obtained.

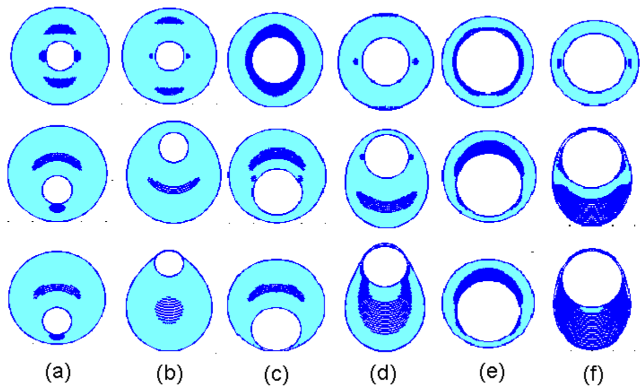


Figure 1: Various dynamical regimes of slightly deformable compound viscoplastic drop, $Ca_{\mathrm{in}} = Ca_{\mathrm{out}} = 1$, $Bo_{\mathrm{shell}} = 1$, $\lambda_{\mathrm{core}} = 1$, $Bn = 0.2$. (a) $Bo_{\mathrm{core}} = 3.6$, $R = 0.3$; (b) $Bo_{\mathrm{core}} = 7$, $R = 0.3$; (c) $Bo_{\mathrm{core}} = 1.5$, $R = 0.5$; (d) $Bo_{\mathrm{core}} = 3.6$, $R = 0.5$; (e) $Bo_{\mathrm{core}} = 1$, $R = 0.7$;(f) $Bo_{\mathrm{core}} = 2$, $R = 0.7$. $t = 0$, 50 and 100 for upper middle and lower rows, respectively. Unyielded zones are denoted by dark blue.

A series of computations of initially concentric spherical double emulsion drop, settling in the gravity field, was performed for various values of the governing parameters. We concentrate on the case of heavy shell, $Bo_{\mathrm{out}} > 0$, light core fluid, $Bo_{\mathrm{in}} > 0$, and the downwards motion of the drop. The behavior of the drop for negative *Bo* is discussed. Typical examples of the dynamics of slightly deformable drops are presented in the figure. One can see two different scenarios: With the passage of time, the core drop either reaches an equilibrium position inside the outer one and the compound drop continues to settle in a steady regime conserving the shape of the inner and outer interfaces, or the phases come to close proximity forming a thin shell that is anticipated to become unstable and break up.

We report on the dependence of the evolution scenario and on the configuration of steady states on the governing parameters with the special focus on the effect of the yield stress (Bingham number).

References

Toose E.M., Geurts B.J., and Kuerten J.G.M. A 2D boundary element method for simulating the deformation of axisymmetric compound non-Newtonian drops. Int. J. Numer. Meth. Fluids. Vol. 30, 653–674 (1999).

Interaction of Thermocapillary and Wave Instabilities in Heated Falling Liquid Film

E. A. Chinnov[1,3], E. N. Shatskii[1] and O. A. Kabov[1,2,3]
[1] S.S. Kutateladze Institute of Thermophysics, Russian Academy of Sciences, pr. Ac. Lavrentyeva 1, Novosibirsk, 630090 Russia, chinnov@itp.nsc.ru
[2] Universite Libre de Bruxelles, Chimie-Physique EP-CP165/62, Microgravity Research Center, Avenue F.D.Roosevelt 50, Bruxelles, B-1050, Belgium, okabov@ulb.ac.be
[3] Heat Transfer International Research Institute of ULB and IT SB RAS, CP 165/62, Avenue F.D. Roosevelt, 50, B-1050 Brussels, Belgium

It is known, two-dimensional (2D) waves in isothermal falling films are unstable with respect to 3D perturbations. It was established that the critical wavelength of instability with respect to transverse 3D perturbations decreases with increasing Reynolds number (Re) of the flow [1].

Presently, various mechanisms of rivulet formation in heated liquid films have been discovered and two special regimes of this process, thermocapillary (A) and thermocapillary-wave (B), have been described [2]. The wave structures in regimes A and B differ by the level of heat flux density necessary for their formation and by the character of the wavelength dependence on the heat flux density and the Reynolds number. Previously, the formation of regular wave structures in a thermocapillary regime A on the surface of a smooth liquids film flowing down a vertical wall containing small size heaters was discovered and studied at small Reynolds numbers. Under the action of thermocapillary forces directed against the flow (from heated to cold regions), the film exhibited thickening and, in the case of a threshold heat flux density, the flow disintegrated into vertical rivulets separated by thin film regions of a certain width Λ. The formation of rivulets in regime B takes place on the wavy surface of liquid films under the action of thermocapillary forces directed from relatively hot to cold regions. The rivulet flow structure gradually develops with growing heat flux and increasing distance from the upper edge of the heater.

This paper presents the results of an investigation of the evolution of hydrodynamic perturbations into thermocapillary-wave structures in a locally heated water film flowing down a vertical plate in non sufficiently studied interval of $5 < Re < 15$.

The experimental setup represented a closed circulation system including a reservoir with an electric pump, a working unit, and a filter. The working unit comprised a textolite base plate with a 150 × 150 mm^2 heater, temperature stabilizers, an extending board, and a film-forming device arranged in the upper part of the working unit. Distilled water was supplied by the pump to the film-forming device consisting of an accumulation chamber, a distributing unit, and a nozzle with a flat calibrated slit. The distance from the lower edge of the nozzle to the upper edge of the heater was X_n = 543 mm. The heated surface obeyed the boundary condition q = const. The initial temperature of water at the exit from the nozzle was 22 °C. The pattern of flow was monitored visually and the images were recorded using digital video and photo cameras. The instantaneous field of film thicknesses was measured using a fluorescent technique. The temperature field was determined using a high speed IR imager (Titanium 570M), which provided 640 ×512 pixel patterns (thermograms) of the temperature distribution over the liquid film surface at a total shot frequency of 115 Hz.

At a distance of 543 mm from the nozzle edge of the film-forming device to the upper edge of the heater, the film surface upstream the heater exhibited 3D horseshoe shaped waves with amplitudes exceeding those of smaller waves. In the absence of heating and at relatively low heat flux densities, the wave structure remained unchanged on the passage over a heater and the rivulets were not formed. An increase in the heat flux density to a sufficiently high level led to the onset of rivulet formation.

The mechanism of transformation of hydrodynamics waves into thermocapillary-wave structures is illustrated in the Figure. A pattern of periodic structure in regime B is formed on the film surface in residual layer, which retains between the waves. The horseshoe like wave (dashed line) is deformed into vertical dynamic inhomogeneities downstream, and split into rivulets spaced 13–16 mm from each other. Thus, the obtained patterns show how temperature inhomogeneities appear at the 3D front of a hydrodynamic wave and lead to deformation of the liquid film under the action of thermocapillary forces, which results in the formation of rivulets.

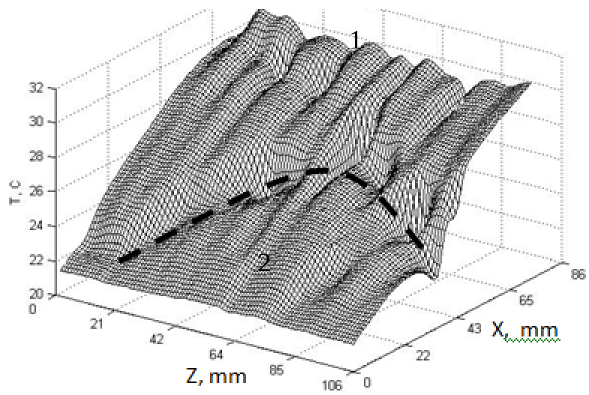


Figure: Temperature distribution on a heated film surface in residual layer (1) and horseshoe like wave (2), Re=10.5 q=0.73 W/cm^2.

References

1. S. W. Joo and S. H. Davis, Instabilities of Three-Dimensional Viscous Falling Films, J. Fluid Mech., Vol. 242 pp. 529-547 (1992).
2. E. A. Chinnov and O. A. Kabov, Jet formation in gravitational flow of a heated wavy liquid film, J. Appl. Mech. Tech. Phys., Vol. 44 pp. 708-71 (2003).

Future ESA experiments in Heat and Mass Transfer Research onboard the International Space Station

Balázs Tóth[1] on behalf of
ESA's Science, Payload Development and Operations teams, Science Team and Space Industry

[1] ESA-ESTEC, NL-2200AG Noordwijk, The Netherlands, E-mail: balazs.toth@esa.int
Science Team: Air Liquide (FR), AIST (JP), Alfa Laval (IT), Astrium GmbH (Bremen, DE), AUTH (GR), CCNY (US), CSI (DE), DIAS (IT), ENEA (IT), EPFL (CH), Euro Heat Pipes (BE), IMFT (FR), IUSTI (FR), LAPLACE (FR), MARS (IT), RAS (RU), SIT (JP), Thalès Aliena Space (FR), TUD (DE), UCLA (US), ULB (BE), UNIHEAT (IT), ZARM (DE), Universities of Alberta (CA), Bergamo (IT), Illinois (US), Kyushu (JP), Liège (BE), Maryland (US), Mons-Hainaut (BE), Padova (IT), Pisa (IT), Toronto (CA), Utsunomiya (JP)
Space Industry: Astrium GmbH (Friedrichshafen, DE), QuinetiQ and Lambda-X

Assessing two-phase heat transfer phenomena, where capillary forces play significant role is often challenging on ground. In the majority of the regimes gravity masks certain processes making the representation of the observations difficult. In some particular cases by creating small scale set-ups, the effect of the gravitational field can be minimised. Nevertheless, this solution imposes further challenges to the diagnostic systems, which are mostly optical.

Therefore, researchers often use microgravity condition, where the influence of gravity (i.e. natural convection, buoyancy) can be eliminated. To support such studies, the European Space Agency (ESA) makes its various microgravity platforms (drop tower, parabolic flights, sounding rockets, space missions) available for scientific purposes.

The aim of the present contribution is to provide a highlight of the two-phase flow heat transfer related experiment candidates, focusing on phase-change phenomena within the frameworks of the utilisation of the International Space Station (ISS). All of these experiments are planned to be conducted in the Fluid Science Laboratory of the Columbus module of the ISS.

CIMEX (Convection and Interfacial Mass EXchange) aims at studying evaporation induced convection and associated gas-liquid interface phenomena. Under ESA contract, EADS Astrium is responsible for the overall development, QuinetiQ (formally Verhaert Space) signes responsible to develop the Fluid Cell Assembly and Lambda-X takes care of the optical aspects of the experiment. The current concept features a liquid pool evaporator, where thermocapillary convection is triggered. The diagnostics include thremal sensors, Infrared thermography, Schlieren visualisation and tomographic Interferometry. The parameters of the experiment include the thermal boundary conditions, the evaporator pool depth, the liquid composition and the circulation properties (flow rate and non-condensable gas content) of the gas-phase.

The RUBI (Reference mUltiscale Boiling Investigation) experiment aims at studying the fundamentals of boiling by analysing the behaviour of single vapour bubbles. The hardware is being developed by EADS Astrium under ESA contract. Its main component is the boiling cell, where a thin resistive heater is implemented with an artificial nucleation site at its centre. The growing vapour bubble can be observed with a high-speed camera from the side and a retractable rack of microthermocouples to be able to study macro-scale properties. Micro-scale phenomena at the three-phase contact line can be analysed with an Infrared camera, observing the bottom side of the heater. A unique feature of RUBI is the possibility to study the effect of field forces on the vapour bubble. A scalable gravitational-like field force will be created via an electric field above the heated surface and shear will be applied by a flow parallel to the heated surface.

SAFIR (Single fin condensation: Film local measurements) focuses on the fundamentals of condensation on a single 2D-like fin profile. The aim is to characterise precisely the condensing liquid film and its distribution using CCD and Infrared cameras and Interferometry. Although, the focus is on condensation, the evaporator is planned to be instrumented as well and various parameters, such as surface roughness and structuring, heating uniformity and potentially shear-driven evaporation are considered to be assessed with particular attention to the film stability and rupture.

SELENE (SELf-rewetting fluids for thermal ENErgy management in space) intends to better understand the basic fluid dynamic and physico-chemical mechanisms in heat pipe systems, focusing on so-called self-rewetting fluids, whose surface tension increases with temperature and concentration. Transparent heat pipe models are foreseen, which allow the quantitative investigation of heat transfer performances and thus the validation of theoretical and numerical models.

To allow a relatively simple implementation of fundamentally similar two-phase flow loops that would study evaporation and condensation phenomena and in anticipation of interest in flow boiling, the concept of a two-phase flow thermal conditioner platform is under assessment. This system will provide for heating and cooling of the test cells of the interchangeable flow loops as well as commonly required (primarily optical) diagnostics. Such a concept could cover the objectives of EMERALD (Enhanced Methods for hEat tRAnsfer in a Loop heat pipe Demonstration) and DOLFIN (Dynamics Of Liquid FIlm/complex wall iNteraction) as well as other parts of the CIMEX programme (beyond the experiment mentioned above). Accordingly, in the loops micro, nanostructured and chemically treated surfaces can be tested in the presence of phase change. Either a given phenomenon could be focused on (i.e. evaporation, condensation or boiling) or loop behaviour can be investigated (e.g. a heat pipe system). Such a configuration would also enable the study in similar loops of various liquids.

Experimental study of the heat transfers in nucleate boiling: influence of the gravity level.

Mohamed Zamoum[1, 2*], Lounès Tadrist[2], Ouamar Rahli[2], Mohand Kessal[1], Hervé Combeau[3]

[1]Université M'hamed Bougara, Département Transport et Equipements des Hydrocarbures, Boumerdès, 35000, Algérie
[2]PUniversité d'Aix-Marseille (UI, II), Laboratoire IUSTI, UMR CNRS 6595, 13453 Marseille Cedex 13, France
[3]Institut Jean Lamour, Dép. SI2M, CNRS – Nancy-Université – UPV-Metz, F-54042 Nancy cedex, France
mzamoum@polytech.univ-mrs.fr

The purpose of this study is to contribute to a better understanding of the heat transfer in nucleate boiling regime. The aim is to determinate the characteristic parameters of the heat transfers with controlled operating conditions (thermodynamics of the fluid, incondensable, etc.) as well as the surface state. Experimental investigations are carried out in natural convection and nucleate boiling regimes.

The influence of the gravity is studied thought the effect of two parameters: inclined angle of the heating wall and fluid saturation pressure. The first one is realized by different orientations of the heating wall going from 0 to 180 °. The angle 0 ° corresponds to horizontal wall with a vertical heat flux of upward direction. The second is realized from the variation of the saturation conditions of the fluid from Tsat=15°C to Tsat=56°C.

An experimental device was realized to perform boiling experience of FC72 from flat heating wall in different conditions of gravity and pressure. The experimental device is conceived to control the thermodynamics conditions of the fluid and the heat transfer during natural convection and nucleate boiling regimes. To reach these objectives the device is constituted by several components, essentially the test cell and a boilingmeter (Fig. 1).

The test cell has a cubic design of 100mm size and contains a volume of 0,75 l. It is established by both structure in Teflon and six faces square in transparent Plexiglas of 100mm of size. The cell test is surrounded with several elements to ensure the control of the temperature and pressure saturation of the fluid.

The boilingmeter is an element of cylindrical shape conceived and made in the laboratory. It is essentially composed of three fluxmeters (Captec heating element), two copper pastilles and a heater resistive to provoke the heating flat walls of the boilingmeter by joule effect (Fig 1), two identical fluxmeters are placed on the upward and downward facing surfaces of the device. The fluxmeters have the a circular disk shape of 20mm of diameter and 0,4mm of thickness, instrumented by two sensors, flux and temperature to measure the heat flux exchange between the heat plate wall and the fluid and the temperature of the heating surface.

The first results presented in this paper aim at the determination of the boiling characteristic curve in nucleate boiling regime. They highlight the influence of the wall orientation on the heat transfer (Figure 2). The effect of the hysteresis phenomenon is also investigated in order to analyse the influence of the nucleation side density in the heat transfer characteristics. These results are compared with those proposed by various authors in the literature.

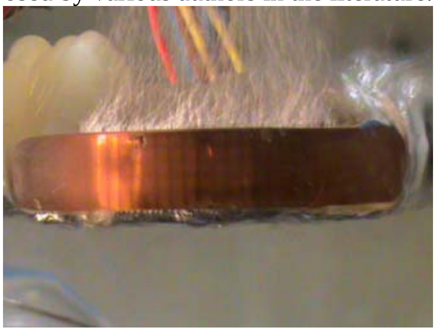


Figure 1: Boilingmeter

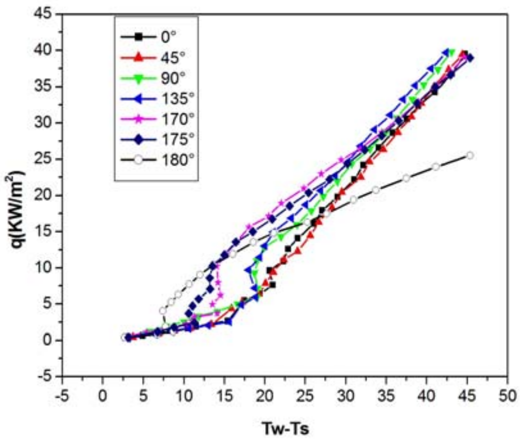


Figure 2: Heat transfer characteristics curve of nucleate boiling of FC-72 for several wall orientations at Psat=0,32bar

The experimental results will be detailed and discussed in the frame of the previous studies (1,2) and also in microgravity conditions where similar behaviours were reported (3).

References

1. K. Nishikawa, Fujita, Satoru, and Ohta. Effect of surface configuration on nucleate boiling heat transfer. International Journal of Heat and Mass Transfer, 27(9):1559–1571, 1984

2. J. C. Passos, L. F. B. Possamai, F.R. Hirata. Confined and unconfined FC72 and FC87 boiling on a downward-facing disc. Applied Thermal Engineering. 25 (2005) 2543-2554

3. O. Kannengieser, C. Colin, W. Bergez. Influence of gravity on pool boiling on a flat plate: Results of parabolic flights and ground experiments. Experimental Thermal and Fluid Science (2010), doi:10.1016/j.expthermflusci.2010.04.010

Experimental Study of Pool Boiling of FC-72 Over Smooth Surface Under Microgravity

Yan-Fang XUE[a], Jian-Fu ZHAO[b], Jin-Jia WEI[a *], Jing LI[b] and Dong GUO[a]

[a] State Key Laboratory of Multiphase Flow in Power Engineering, School of Energy and Power Engineering, Xi'an Jiaotong University.
28 Xianning Xilu, Xi'an 710049, China
[b] Key Laboratory of Microgravity (National Microgravity Laboratory) / CAS; Institute of Mechanics, Chinese Academy of Sciences.
15 Beisihuan Xilu, Beijing 100190, China
[**]E-mail:jjwei@mail.xjtu.edu.cn

With the rapid development of MEMS processing technology, the size of the electronic equipments is significantly reduced and the processing speed and memory capacity are substantially increased, resulting in a rapid increasing of power dissipation rate at the chip, module and system levels. Boiling heat transfer with phase change is an increasingly-popular alternative scheme for electronics cooling on earth due to its high heat transfer performance in comparison with single-phase heat transfer. For this reason, boiling heat transfer is expected to be also used as a cooling method for future space-based hardware. Boiling is a very complex process existing in many interrelated physical mechanisms. Although significant progress has been made in this field, the mechanisms of heat transfer responsible for enhancement and deterioration during nucleate boiling in microgravity are still unclear and even contrary until now. Opposite results may be obtained depending on the experiments by different researchers or under different experimental conditions (Lee 2002).

Experiments of highly subcooled pool boiling of FC-72 dissolved with air on a small scale smooth silicon chip with the dimensions of $10\times10\times0.5mm^3$ were studied in short-term microgravity condition utilizing the drop tower Beijing. The bubble behaviors and heat transfer of air-dissolved FC-72 on the silicon chip were obtained at the bulk liquid subcooling of 41 K and nominal pressure of 102 kPa. The identical ground experiments were also made for comparison. It should be pointed out that the identical ground ones did not achieve the critical heat flux in the present study to avoid damage to the test chip.

The boiling heat transfer performance at low heat flux region in microgravity condition is similar to that in normal gravity condition (Fig. 1), while bubbles couldn't depart from the heater surface throughout the whole microgravity period (Fig. 2(a)). The coalescence among these primary bubbles forms an isolated large bubble during the microgravity period, which also attaches to the heater surface throughout the microgravity period.

With further increasing heat flux to the fully developed nucleate boiling region, steady-state nucleate pool boiling was attained with slight deterioration of heat transfer in microgravity (Fig. 1). Coalescence occurs continuously among adjacent bubbles and then the coalesced large bubble can depart from the heating surface, which may mainly be caused by the oscillation of the coalesced bubble in lateral direction (Fig. 2(b)).

At the high heat flux regime near the critical heat flux in microgravity, the significant deterioration of heat transfer is observed (Fig. 1), and a large coalesced bubble forms quickly and almost covers the whole heater surface, followed by shrinking to an oblate in shape and smooth in contour due to the highly subcooled condensation (Fig. 2(c)). It is possible for the occurrence of local dry-out or transition to film boiling at the bottom of the large coalesced bubble.

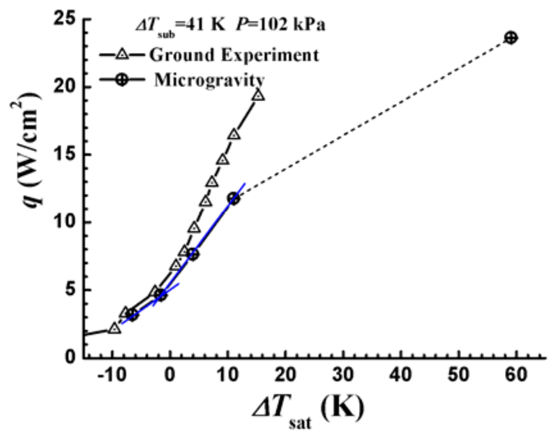


Figure 1: Boiling curves in ground and microgravity conditions.

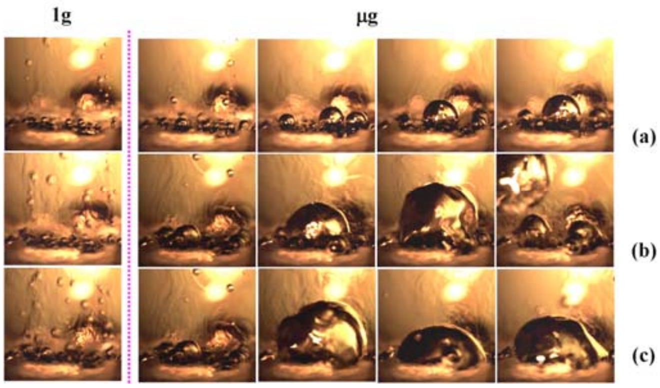


Figure 2: Variations of Bubble behaviors at different heat flux levels ((a) q=4.74 w/cm^2, (b) q=11.59 w/cm^2, (c) q=23.6 w/cm^2) under normal gravity and microgravity conditions. The images under microgravity condition are corresponding to 0.2, 1.2, 2.2 and 3.2 s, respectively.

Acknowledgement: We gratefully acknowledge the financial support from NSFC (No: 50806057 and 10972225) and the Key Laboratory of Microgravity/CAS for experiments utilizing the drop tower Beijing.

References

Lee H.S., Mechanisms of Steady-State Nucleate Pool Boiling in Microgravity, Ann. N.Y. Acad. Sci., Vol. 974 pp. 447-462 (2002)

Recent Progress in JAXA Official Project of Boiling Two-Phase Flow Experiment onboard ISS

Kiyosumi Fujii[1], Haruo Kawasaki[2] and Haruhiko Ohta[3]

[1]JAXA ISS Science Project Office. 2-1-1 sengen Tukuba-shi, Ibaraki, JAPAN. E-mail : fujii.kiyosumi@jaxa.jp
[2]JAXA Thermal System Group. 2-1-1 sengen Tukuba-shi, Ibaraki, JAPAN. E-mail : kawasaki.haruo@jaxa.jp
[3]Kyushu University, Faculty of Engineering, Department of Aeronautics and Astronautics.
Motooka 744, Nishi-ku, Fukuoka 819-0395, JAPAN. E-mail : ohta@aero.kyushu-u.ac.jp

For the design of high-performance thermal management systems applied to space platforms, boiling and two-phase flow in microgravity has attracted much attention. Therefore, in order to mainly clarify gravity effects on heat transfer characteristics and establish database for microgravity flow boiling to provide fundamental information for the development of large-scale two-phase thermal management systems, we prepare for the experiments of boiling two-phase flow utilizing a long-term microgravity environment onboard Japanese experimental module "KIBO" in the International Space Station (ISS) as one of the JAXA official projects. We presented the situation of the project existing at the Fifth ITT Workshop in Kyoto, Sep. 2010[1]. In this presentation, up-to-date information of the preparation for the project is reported.

We designed or selected the specifications of individual components for the ISS experimental equipment, and **Figure1** shows the present configuration of the equipment designed by using those components. The equipment dimension is 800mm×650mm×500mm. The equipment is planned to accommodate in the working volume (WV) of a Multi –purpose Small Payload Rack (MSPR) developed as a multi-user experiment facility.

Also, In order to verify the feasibility of the ISS experiments, we assembled a ground model testing loop (GM) by improving the Bread Board Model (BBM) as shown in **Figure 2**. There are several objectives in the experiments using GM as follows: (1) verify the functions of all components selected for the Engineering Model (EM) of the experimental equipment, (2) determine the range of the experimental conditions in microgravity of the ISS experiment, (3) inspect the software for the electrical control of the experimental equipment etc.

Figure 3 shows examples of interfacial behaviors of two-phase flow observed in the transparent heated tube, where the inner of wall was uniformly coated with a gold film of a thickness of around 10nm.

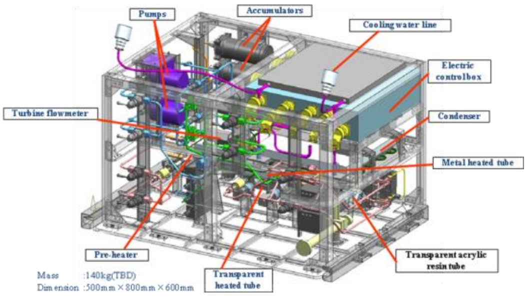


Figure 1: A schematic of **T**wo-**P**hase **F**low experimental equipment.

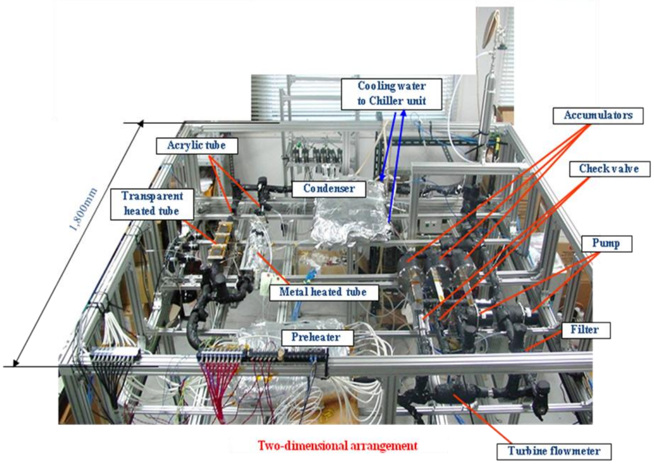


Figure2: Ground Model testing loop of **T**wo-**P**hase **F**low experiment.

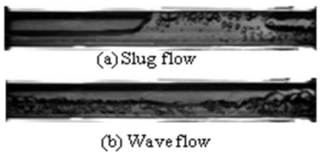


Figure 3: Interfacial behaviors of two-phase flow in a transparent heated tube.

Now we aim to launch the equipment by the H-II Transfer Vehicle (HTV) #6 in 2014 and are approaching to Preliminary Design Review (PDR) as scheduled in Table 1.

Table1: Schedule of the **T**wo-**P**hase **F**low experiment.

○ SDR (System definition review)	Apr. 2010 (approved)
○ PDR (Preliminary design review)	Jun. 2011
EM manufacturing and Test	
○ CDR (Critical design review)	May. 2012
FM manufacturing and Test	
○ PQR (post-qualification test review)	Jul. 2013
Preparation for flight	
○Launch on the HTV#6	Jul. 2014
○On orbit experiments	Aug.-Oct. 2014

Acknowledgement

The present authors express appreciation for the cooperation and powerful support by all project members participated from Kyushu Univ., Kobe Univ., Univ. of Hyogo, Tokyo Univ. Sci. Yamaguchi, IHI, IA, JAMSS and JSF.

References

[1] Haruhiko Ohta, Kiyosumi Fujii and Haruo Kawasaki, Preparation for Flow Boiling Experiment on Board ISS-Recent Progress, Fifth International Topical Team Workshop on TWO-PHASE SYSTEMS FOR GROUND AND SPACE APPLICATIONS, September 26-29, 2010, Kyoto, Japan

Heat Transfer Enhancement in Subcooled Boiling with Wetting Surface

Koichi Suzuki[1], Kazuhisa Yuki [1]and Haruhiko Ohta[2]
1. Dept. Mechanical Engineering, Tokyo Univ. Science-Yamaguchi
Daigaku-Dori 1-1-1, Sanyo-Onoda, Yamaguchi Pref.,756-0884 Japan. Email: suzuki@rs.noda.tus.ac.jp
2. Graduate School of Aerospace Engineering, Kyushu University
Moto-oka 44, Nishi-ku, Fukuoka-shi, Fukuoka Pref., 819-0395 Japan. Email;ohta@aero.kyusyu-u.ac.jp

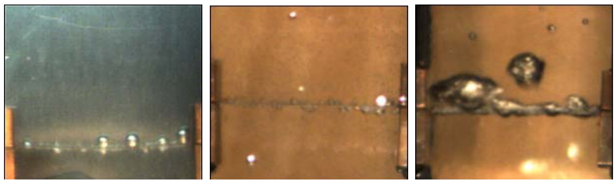
Wetting or hydrophilic property has been one of the important factors to control the efficiency of boiling heat transfer. The wetting accelerates the rate of solid–liquid contact and introduces the heat transfer enhancement. As considering today's energy and environmental conditions, the wetting in boiling can be expected as one of the eco-technologies. Especially, it may be effective for high heat flux transport technique using boiling heat transfer in space. However, the details of wetting have been still unknown in theoretical treatments and heat transfer practice.

Generally, three approaches have been considered for improving the wetting between solid and liquid. They are the coating of wetting materials on the heating surface, the microscopic treatment of the surface by irradiating ultraviolet beam (Takata al., 2003) or radioactive ray (Takamasa et al.,2003) and development of rewetting liquid (Abe, 2005). Actually, no wetting materials have been found for the high temperature use. In the second approach, the surface structure recovers in a short period after terminate of the irradiation. In the approach from liquid side, the heat transfer enhancement is expected by an effect of Marangoni convection induced in the bubble interface using binary mixtures of polyhydric alcohol and water.

Recently, a ceramics wetting material has been proposed by a certain glass company in Japan (Central Glass Co. Ltd. 1997-2002). The material is coated on the metal surface in thickness of 100μm and processed by firing the paste including micro-particles of metal oxides at 600ºC. The contact angle of water droplet on the aluminum or stainless surface coated with the wetting layer is less than 30 deg. In subcooled pool boiling of water, the wetting effect was remarkably observed for a stainless heater coated with the ceramics coatings in Fig.1 for an example. It was placed in horizontal and the size was 30mm in length, 1mm in width and 0.1mm in thickness. Heating was performed by applying DC under atmospheric condition. The burn-out heat flux (CHF) was observed remarkably higher than that of non-coating surface as shown in Fig.1. In the subcooled pool boiling of water with block wetting surface of 10mm in diameter, no remarkable differences were observed for the critical heat fluxes, however it was obtained at lower superheat of heating surface for the wetting surface.

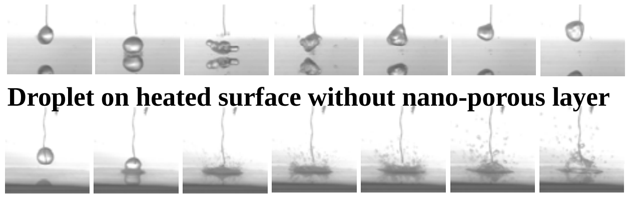
Recently, the heat transfer enhancement has been performed in boiling heat transfer using nano-particles or nano-materials. It may be a new technology for heat transfer enhancement in boiling. The authors have performed boiling test for the heating surface with nano-porous layer produced by a sintered metal for the purpose of heat transfer enhancement. A behavior of water droplet on the heated surface with nano-porous layer is shown in Fig.2 (Yuki et al.,2010). The droplet is broken explosively into many fine drops on the porous surface by violent boiling as shown in Fig.2. The experimental results show that the porous layer has wetting or hydrophilic property in boiling heat transfer.

The authors will introduce examples of heat transfer enhancement in subcooled boiling for the wetting surface with ceramics hydrophilic layer and nano-porous layer for practice.



Stainless surface $1.5MW/m^2$ CHF | **Wetting surface $1.5MW/m^2$** | **Wetting surface $3.4MW/m^2$ CHF**

Figure 1 Effect of ceramics wetting heater on subcooled pool boiling of water. Liquid subcooling = 10K



Droplet on heated surface without nano-porous layer

Droplet on heated surface with nano-porous layer

Figure 2 Effect of wetting heated surface with nano-porous layer on water droplet behavior

References

Takata, Y., Photo-induced Hydrophilicity and Its Applications to Phase Change Phenomena , Journals, Thermal Science and Engineering, (Heat Transfer Society of Japan), Vol. 10, No. 6, pp31-39 (2003).

Takamasa, T., Hazuku, T., Mishima, K., Okamoto, K., Mitsutake, T. and Morooka, S., Effect of Radiation Induced Surface Activation on Quenching Condition, Journals, Thermal Science and Engineering, (Heat Transfer Society of Japan), Vol. 11, No. 4, pp.33-41(2003).

Abe,Y., Self-rewetting Fluids-Beneficial Aqueous Solutions, Proceedings of the Interdisciplinary Transport Phenomena in Microgravity and Space Science Conference IV, , CR-ROM, ECI-ITP-05-26 (2005).

Central Glass Co. Ltd., Japanese patent: No.2599531(1997), No.3147251(2001), No. 3340149 (2002).

Yuki, K. and Suzuki, K., Cooling Technique of the Next Generation Power Devices by Boiling Heat Transfer with Nano-Porous Structure, Proceedings of the 21th ISTP, CD-ROM IS01-06 (2010).

Non-linear thermocapillary convection in half-zone liquid bridge - on-orbit experiments on Kibo aboard the ISS-

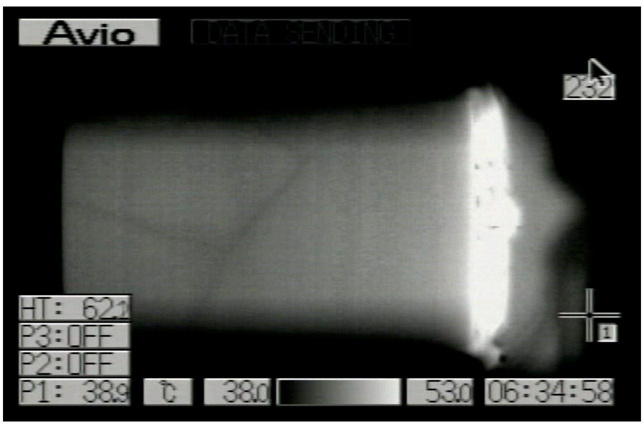
Ichiro Ueno[1], Koichi Nishino[2], Hiroshi Kawamura[3], Mitsuru Ohnishi[4] & Satoshi Matsumoto[4]

[1]Department of Mechanical Engineering, Faculty of Science & Technology, Tokyo University of Science
2641 Yamazaki, Noda, Chiba 278-8510, Japan
E-mail: ich@rs.tus.ac.jp
[2]Yokohama National University, [3]Tokyo University of Science, Suwa, [4]Japan Aerospace Exploration Agency (JAXA)

The long-duration fluid physics experiments have been carried out at 'Kibo,' the Japanese experimental module, aboard the International Space Station (ISS) since 2008. In these experiments, various aspects of thermocapillary convection in a half-zone (HZ) liquid bridge are examined. This series of experiments enable us to conduct lots of experiments that are impossible to run on Earth because of the gravity, and have been providing numbers of invaluable results. In 2010, thermocapillary convection in a HZ liquid bridge of 50 mm in diameter and 20-cSt silicone oil had been examined. During this term of experiment, the authors put a large temperature difference ΔT between the two rods, and succeeded to generate nonlinear convection fields in a HZ liquid bridge at three kinds of aspect ratios $\Gamma = H/R$, where H is the liquid bridge height and R the liquid bridge radius. The hydrothermal wave (HTW) proposed by Smith & Davis (JFM, 1983) is observed. Figure 1 indicates typical examples of non-linear behaviors of the HTW under a condition far beyond the transition point ($\Gamma = 2.50$, $\Delta T \sim 61.9$ K ($Ma \sim 17.4\times10^4$, $\varepsilon \equiv (Ma - Ma_c)/Ma_c \sim 7.47$)); (a) the side view of the liquid bridge observed with IR camera (heated disk is located at right-hand side, and cooled at left), and (b) temporal variation of the temperature deviation along the center line of the liquid bridge surface. One can see that the HTW basically propagates from the hot-end surface toward the cold-end surface. There also exists, however, components propagating from the cold to the hot. We pay our attention to the behavior of the HTWs, and analyze the chaotic behaviors from the surface temperature variation. We will introduce the chaotic thermocapillary convections and their transition scenario in the HZ liquid bridge.

References

Smith, M. K. & Davis, S. H., Instabilities of dynamic thermocapillary liquid layers. Part1.Convective instabilities, J. Fluid Mech. 132, pp.119-144 (1983).

(a)


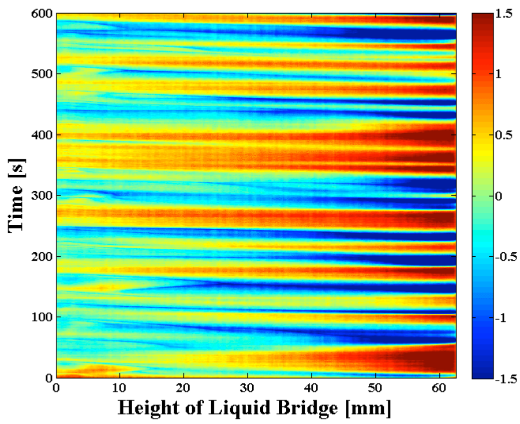
(b)


Figure 1: Typical examples of non-linear behaviors of the HTW under a condition far beyond the transition point ($\Gamma = 2.50$, $\Delta T \sim 61.9$ K ($Ma \sim 17.4\times10^4$, $\varepsilon \equiv (Ma - Ma_c)/Ma_c \sim 7.47$)); (a) side view of the liquid bridge observed with IR camera (heated disk is located at right-hand side, and cooled at left), and (b) temporal variation of the temperature deviation along the center line of the liquid bridge surface.

Interaction between bubbles and turbulence in monodisperse suspensions. Drop tower experiments

Pau Bitlloch, Xavier Ruiz[1], Laureano Ramírez-Piscina[2] and Jaume Casademunt

Departament d'Estructura i Constituents de la Matèria
Facultat de Física - Universitat de Barcelona
Av.Diagonal 647 - 08028 Barcelona (Spain)
bitlloch@ecm.ub.es

Efficient control of bubble formation and management in microgravity environments is an important aspect for multiple applications in rocket industry or Life Support Systems in space. On the other hand, from a fundamental point of view, the physics of bubble suspensions in turbulent flows, in particular, the two-way coupling between bubbles and turbulence is still poorly understood. In the absence of buoyancy, the situation may be in principle simpler but there are no much data available, in particular for monodisperse bubbly flows, due to the difficulty to produce large numbers of small bubbles with controlled size in microgravity.

Recently, a new bubble injection method has been successfully tested that allows the formation of monodisperse turbulent bubble jets in microgravity (Carrera et al. 2008, Arias et al. 2010). Here we present results of a new series of Drop Tower experiments that exploit the above bubble injection method to create a nearly uniform monodisperse bubble suspension carried by a turbulent duct flow.

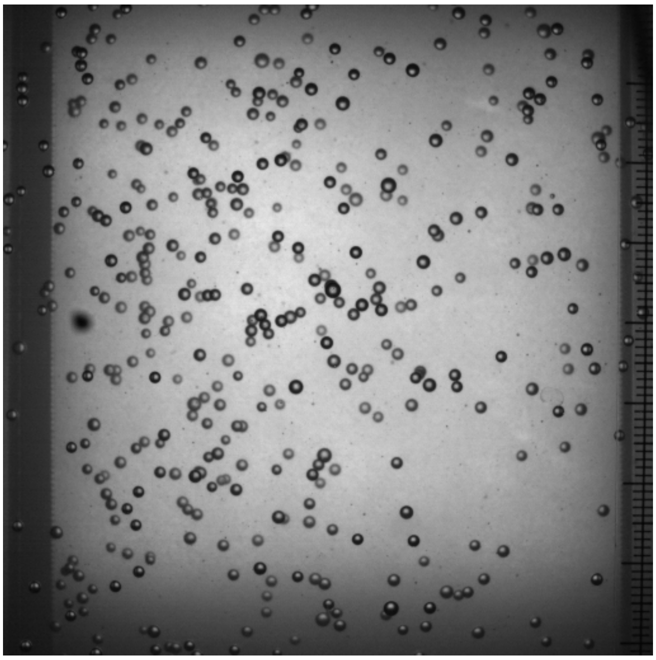


Figure 1: Snapshot of a monodisperse bubble suspension of prescribed bubble size in microgravity conditions

The experiment is designed to allow independent control of bubble size, bubble density and the degree of turbulence of the carrying flow. Typical duct Reynolds numbers used are up to 12000. Typical bubble diameter is 1.5 mm, that is, significantly larger than the Kolmogorov turbulent scale, but significantly smaller than the scale of most energetic eddies. Bubble densities keep mean bubble distances comparable to the size of most energetic eddies. This general scenario is designed to optimize spatial dispersion of bubbles while minimizing the degree of coalescence. The corresponding Webber number of bubbles is small enough to maintain the spherical shape, but they cannot be considered as point-like with respect to their interaction with the flow. Under these circumstances, a significant degree of two-way coupling between bubbles and turbulence is expected.

We have conducted a series of Drop Tower experiments with this setup at the ZARM facilities in Bremen. The experiment has proven successful in creating nearly monodisperse bubble suspensions in turbulent flows under microgravity, under controlled variations of the relevant parameters. Quantitative information is obtained through the appropriate digital image processing, based on particle tracking methods that yield trajectories of individual bubbles. As a first characterization, we have systematically measured the mean velocity profile of bubbles and the velocity root-mean-square dispersion for different experimental conditions, including variation of Reynolds number, bubble size and bubble density. In addition, the data provide, for the first time, a quantitative characterization of the decay of the pseudo-turbulence created by the bubbles during normal gravity, immediately before the drop experiment.

References

J. Carrera, X. Ruiz, L. Ramírez-Piscina, J. Casademunt, M. Dreyer, Generation of a Monodisperse Microbubble Jet in Microgravity, AIAA Journal Vol. 46 pp. 2010-2019 (2008)

S. Arias, R. González-Cinca, X. Ruiz, L. Ramírez-Piscina, J. Casademunt, Characterization of the performance of a minibubble generator in conditions relevant to microgravity, Colloids and Surfaces A: Physicochemical and Engineering Aspects 365, pp.52-55 (2010)

[1] Universitat Rovira i Virgili
[2] Universitat Politècnica de Catalunya

The dynamic bubble point

Michael Conrath and Michael Dreyer

Center of Applied Space Technology and Microgravity (ZARM),
University of Bremen, Am Fallturm, 28359 Bremen, Germany
Michael.Conrath@zarm.uni-bremen.de

The dynamic bubble point problem consists of a large bubble trapped at a porous screen immersed in a liquid that wets the screen. Due to gravity, the bubble is rather flat and blocks part of the screen. Upward liquid flow cross the screen leads to a pressure difference between its front and back side. As this pressure difference exceeds the bubble point threshold, the trapped bubble starts to break through the screen causing a number of small bubbles to emerge in chains from the upper side. These emerging bubbles form and detach in its typical manner. They affect the breakthrough and even bring it to a halt as the pressure difference falls below a detachment pressure which is innate to the screen pores. During breakthrough, the trapped bubble shrinks and correspondingly blocks the screen to a lesser extend. The liquid flow around the trapped bubble deforms it and tends to tear it flatter. Hydraulic losses in the circuit in which the trapped bubble is located cause a compression. The interplay of all these effects is a complex issue that is addressed in the present article. We develop a model comprising most of the effects and present it along with original experimental data.

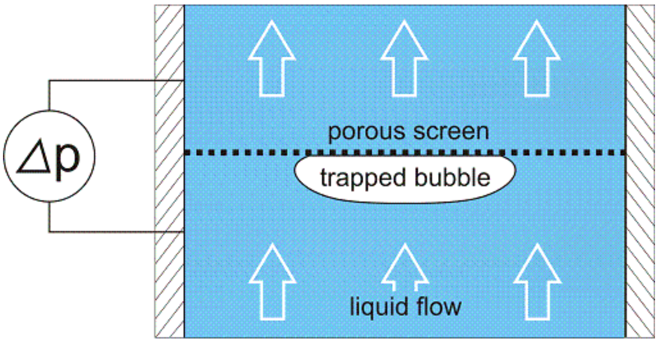


Figure 1: Dynamic bubble point problem consisting of a trapped bubble in liquid flow at a porous screen.

Figure 2: Experimental observation of the trapped bubble as it breaks through the porous screen at a sufficiently high liquid flow rate.

References

Michael Conrath, Michael E. Dreyer, The dynamic bubble point, submitted to Int. J. Multiph. Flow (2011)

The effect of particle sticking to solid matrix on a stability of a diffusion front in porous medium

T.P. Lyubimova[1], D.V. Lyubimov[2], B.S. Maryshev[1]

[1]Institute of Continuous Media Mechanics UB RAS, Computational Fluid Dynamics Laboratory
1, Koroleva Str, Perm, 614013, Russia
[2]Perm State University, Theoretical Physics Department, 15, Bukireva Str., Perm, 614990, Russia
lubimova@psu.ru

Due to the particle sticking to the solid matrix in porous medium, the drift of particles in this medium is not always properly described by the Gauss law. Different models were suggested for the description of the sticking effect. One of them is the fractal Mobile-Immobile Medium (MIM) model [1], it is the hydrodynamic limit of stochastic model. In the stochastic model the sticking time is random variable which is distributed according to the one-sided Levy stable law. The adequacy of MIM model is well confirmed by the experiments. However, due to the presence of the fractional derivatives in the equations, one needs to keep large-size-information on the admixture distribution history in course of the numerical simulations based on MIM model. An alternative approach is to use a two-phase phenomenological model accounting for the existence of two phases of admixture (stuck and unstuck particles) and implementing some kinetic law. In the present work, the simple second-order kinetic model [2] accounting for the saturation of immobile phase (stuck particles) is implemented.

In the present paper the effect of particle sticking to the solid matrix on a stability of planar diffusion front in porous medium is studied in the framework of Darcy-Boussinesq approximation. The case when heavy admixture diffuses from the upper boundary into the bulk of semi-infinite domain of porous medium in gravity field is considered. In this case one should expect the development of the Rayleigh-Taylor finger instability [3]. Characteristic size of the fingers along the front is defined by the wavelength of perturbations. Both MIM model and second-order kinetics model are used for the description of diffusion process. The time-scale for the development of perturbations at the front are small in comparison with the characteristic diffusion time scale which allows to implement the frozen coefficients method. According to this method, at first the concentration profile per unit of diffusion time is calculated, then the stability problem for the solutal convection is solved with fixed concentration profile.

The neutral curves in the parameter plane solutal Rayleigh number (**Rp_c**) – wave number of perturbations (**k***) obtained in the framework of two models described above are presented in Fig.1 for different parameters of the models. It is seen that in both models the stabilizing effect of the particle sticking is observed.

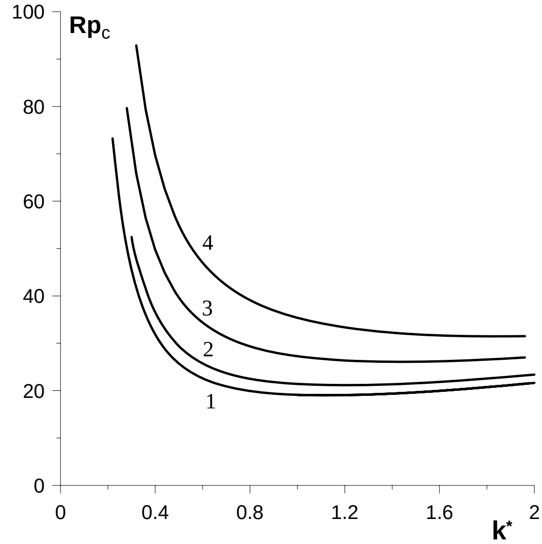


Fig.1. Neutral curves in the parameter plane solutal Rayleigh number – wave number of perturbations: 1 – classical diffusion model, 2 – MIM model with β=0.5, λ=3 and second-order kinetic model with a=1, b=1; 3 – MIM model with β=0.75, λ=3; 4 – second-order kinetic model with a=10, b=100; λ is the mobility parameter, β is the order of fractal derivative, a and b are the dimensionless parameters of adsorption and desorption

The work was made under financial support of Russian Foundation for Basic Research (grant 09-01-00618).

References

1. Schumer R Benson D A Meerschaert M M and Bauemer B 2003 Water Resour. Res. 39 (10) 1296.

2. Selim, H. M., and M. C. Amacher. 1997. Reactivity and Transport of Heavy Metals in Soils. CRC/Lewis, Boca Raton.FL (240 p).

3. Taylor, Sir Geoffrey Ingram, 1950, The instability of liquid surfaces when accelerated in a direction perpendicular to their planes, Proceedings of the Royal Society of London. Series A, Mathematical and Physical Sciences 201 (1065): pp. 192–196.

Enhanced Boiling Heat Transfer in Microgravity by Using Micro-Pin-Finned Surfaces

J. J. WEI[1*], J. F. ZHAO[2], Y. F. XUE[1] and J. LI[2]

[1]State Key Laboratory of Multiphase Flow in Power Engineering, School of Energy and Power Engineering, Xi'an Jiaotong University
28 Xianning Xilu, Xi'an 710049, China
[2]Key Laboratory of Microgravity (National Microgravity Laboratory) / CAS; Institute of Mechanics, Chinese Academy of Sciences
15 Beisihuan Xilu, Beijing 100190, China
E-mail: jjwei@mail.xjtu.edu.cn

Electronics cooling by using boiling heat transfer in space and in planetary neighbors has become an increasing significant subject due to its high efficiency in heat transfer. However, for the boiling heat transfer in microgravity, the buoyancy effect becomes weak, resulting in a longer stay time for the bubble departure. These may prevent the effective access of fresh bulk liquid to the heater surface in time, resulting in a lower boiling heat transfer performance at high heat flux. How to improve boiling heat transfer effectively in microgravity is an important issue.

Experiments were conducted to study the enhanced boiling heat transfer performance of FC-72 over square silicon chips in microgravity. A new surface structure, micro-pin-fin with the dimensions of 50×60 (PF50-60) μm^2(thickness, t $\times$ height, h), was fabricated on the chip surface. A smooth surface (chip S) was also tested for comparison. The microgravity was obtained by using Drop Tower Beijing in the National Microgravity Laboratory, Chinese Academy of Sciences. The facility satisfies the safety criteria to expose the apparatus in the drop capsule to different gravitational environments varying between microgravity (μg) ($10^{-2\sim-3}g_o$, g_o=9.81m/s^2) in the free falling period.

The vapor bubble behaviors of chips S and PF50-60 in microgravity at high heat fluxes are shown in Fig.1(a) and (b), respectively. The time for entering the microgravity condition is set to 0 s. It can be seen that before entering the microgravity condition, the bubbles generate and departure continuously from the heating surface due to the effects of buoyancy forces. However, at about 0.12s after entering the microgravity condition, the vapor bubbles begin to coalesce with each other to form several large bubbles, which attaches on the chip surface without departure due to the weakness of buoyancy force. The bubble number is much larger for chip PF50-60, indicating that the micro-pin-finned surface can provide larger number of nucleation sites for enhancing boiling heat transfer performance. With increasing time up to 1.82 s, the bubbles tend to maintain a hemispherical shape on chip S with a large contact area on the heater surface, and almost completely cover the heater surface, causing the mean heater surface temperature to rise continuously from checking the measured wall temperatures. However, the bubbles coalesce to form a large spherical bubble on chip PF50-60, which does not cause obvious increase of wall temperature. The capillary force generated by the interface between the large bubble and the liquid of the microlayer beneath the bubble drives plenty of fresh liquid to contact with the superheated wall for vaporization through the regular interconnected structures formed by the micro-pin-fins, as well as improves the micro-convection heat transfer by the motion of liquid around the micro-pin-fins. Therefore, there is no deterioration of boiling heat transfer, and the sufficient supply of bulk liquid to the heater surface guarantees the continuous growth of the large bubble. With further increasing time up to 3.50s, the vapor bubble still attaches on chip S, which prevent the bulk liquid from accessing the heater surface for evaporation, resulting in a significant increase of wall temperature. However, the bubble departs away from chip PF50-60 when the bubble size grows up to a certain value, resulting in no obvious change of wall temperature.

The micro-pin-fined surface structure can provide large thermal capillary force and small flow resistance, driving a plenty of bulk liquid to access the heater surface for evaporation in high heat flux region, which results in large boiling heat transfer enhancement. Since the thermal capillary force is no relevant to the gravity level, the micro-pin-fined surface appears to be one promising enhanced surface for efficient electronic components cooling schemes not only in normal gravity but also in microgravity conditions, which is very helpful to reduce the cooling system weight in space and in planetary neighbors

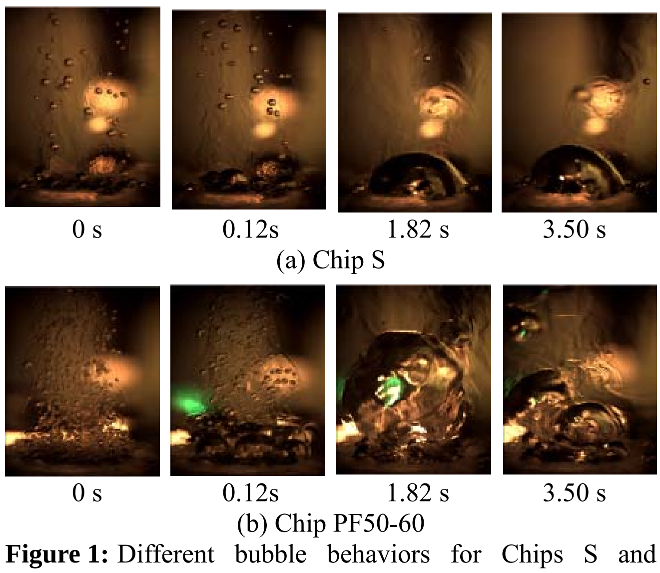


Figure 1: Different bubble behaviors for Chips S and PF50-60 in microgravity.

Acknowledgement: We gratefully acknowledge the financial support from NSFC (No:50806057 and 10972225) and the Key Laboratory of Microgravity/CAS for experiments utilizing the drop tower Beijing and the program for new century excellent talents in university (NCET-07-0680).

New approach to the studies of the Pulsating Heat Pipes

Vadim Nikolayev[1,2], Fabien Bonnet[1], Philippe Gully[1]

[1]SBT/INAC/CEA-Grenoble, 17 rue des Martyrs, 38054 Grenoble Cedex 9, France
[2]ESEME-CEA, PMMH-ESPCI,10 rue Vauquelin, 75231 Paris Cedex 05, France
vadim.nikolayev@espci.fr, http://www.pmmh.espci.fr/~vnikol/PHP.html

A heat pipe is a device for transferring the heat from hot to cold spots situated at a certain distance. It is a sealed tube containing liquid and gas phases of a pure fluid. The liquid is vaporized at the hot portion of the tube (evaporator) and the energy is stored as the latent heat of the liquid-vapor phase change. The vapor is then transported to the cold part (condenser) and released there during the inverse phase change. The pulsating heat pipe (PHP) is a simple capillary tube without any internal structure bent so that it meanders between condenser and evaporator. It is filled partially by the liquid and a sequence of bubbles and liquid plugs forms inside it (Figure 1). Spontaneous oscillations appear in such

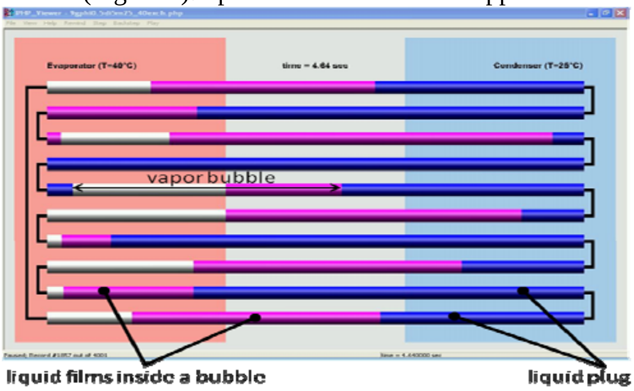


Figure 1: The PHP representation in the PHP_Viewer software developed for the postprocessing of the simulation results (Nikolayev, 2011). The liquid films partially cover the internal tube walls inside the vapor bubbles. Evaporator and condenser are shown with the red and blue rectangles.

a system when the difference of temperatures of evaporator and condenser becomes greater than a threshold. Due to the convective heat transport capability, the PHP appears to be the most efficient in comparison with the other types of the heat pipes. In particular, very small thermal resistances ~0.01K/W (Ayel et al., 2010) and very high transferred heat fluxes ~250 W/cm^2 (Maydanik et al. 2009). However, the PHP functioning is non-stationary and often chaotic. This complicates a lot the PHP understanding. There are several approaches to the PHP modeling. Because of the complexity of the PHP functioning, traditional approaches that seek establishing a correlation for the overall Nu as a function of other non-dimensional numbers (Re, etc.) are unfruitful. Only direct numerical simulations may be viable. A recently proposed approach of this type (Nikolayev, 2011) will be described. It reproduces some of the features observed experimentally like chaoticity and oscillation threshold. However it does not yet account for many of the physical phenomena that occur inside the PHP. What is the origin of the instability that causes the oscillations? What defines the oscillation threshold? What is the PHP thermodynamics? Is the vapor at saturation or can be compressed/expanded? How the menisci influence the pressure drop? What is the wetting film dynamics? What is the role of boiling in the PHP? Answering of all such questions are necessary will allow predicting the heat transfer in the PHP. They need to be studied in simplified systems like the single-branch PHP where there is only one meniscus that oscillates in a well defined area that can be observed (Das et al. 2010). Since the phase distribution in the single branch PHP is known in advance, the gas phase thermodynamics can be accessed experimentally. This is done recently in a cryogenic setup where the negligible radiative losses allow the gas temperature to be directly measured (Bonnet et al. 2011). Similarly, the theoretical analysis of the single branch PHP is much easier. In particular, the oscillation spectrum exhibits a characteristic frequency that has been recently established theoretically (Das et al. 2010). The developed theoretical model allows the role of different physical effects to be assessed.

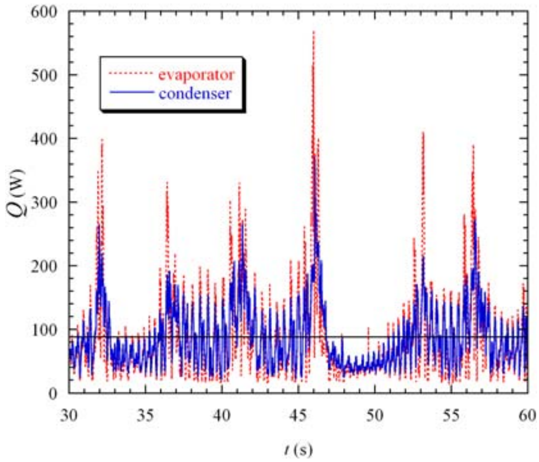


Figure 2: Typical simulated heat exchange rate evolution in the condenser and evaporator of the water PHP (Nikolayev, 2011).

References

V. Ayel, Y. Bertin, C. Romestant, G. Burban, Experimental study of pulsating heat pipes tested in horizonal and vertical positions, in : Proc. 15th International Heat Pipe Conf., Clemson, USA, 2010.

S.P. Das, V.S. Nikolayev, F. Lefevre, B. Pottier, S. Khandekar, J. Bonjour, Thermally induced two-phase oscillating flow inside a capillary tube, Int. J. Heat Mass Transfer 53 (2010) 3905-3913.

Y. F. Maydanik, V. I. Dmitrin, V. G. Pastukhov, Compact cooler for electronics on the basis of a pulsating heat pipe, Appl. Therm. Eng. 29 (2009) 3511 – 3517.

V. S. Nikolayev, A dynamic film model of the Pulsating Heat Pipe, ASME J. Heat Transfer (2011), accepted.

F. Bonnet, Ph. Gully, V. Nikolayev Experimental study of a single branch cryogenic Pulsating Heat Pipe: first results, Eurotherm seminar on gravitational effects on liquid-vapor phase change, Hyeres, France, 2011.

The CVB Experiment – Constrained Bubble Nucleation in Microgravity

Chatterjee[1], P.C. Wayner[1], Jr., J.L. Plawsky[1]
David F. Chao[2] , Ronald J. Sicker[2],
Tibor Lorik[3], Louis Chestney[3], John Eustace[3], Raymond Margie[3], John Zoldak[3]
[1]Rensselaer Polytechnic Institute, Troy, NY, USA,
[2]NASA Glenn Research Center, Cleveland, OH,
[3]Zin Technologies, Cleveland, OH, USA
E-mail: Plawsky@rpi.edu

The Constrained Vapor Bubble (CVB) is a prototype for a wickless, grooved heat pipe and is the first fluids experiment flown on the International Space Station as part of the US space program. The CVB experiment resides in the Fluids Integrated Rack (FIR) and along with the Light Microscopy Module (LMM) is designed to probe the details of the fluid mechanics underlying the operation of a heat pipe in space.

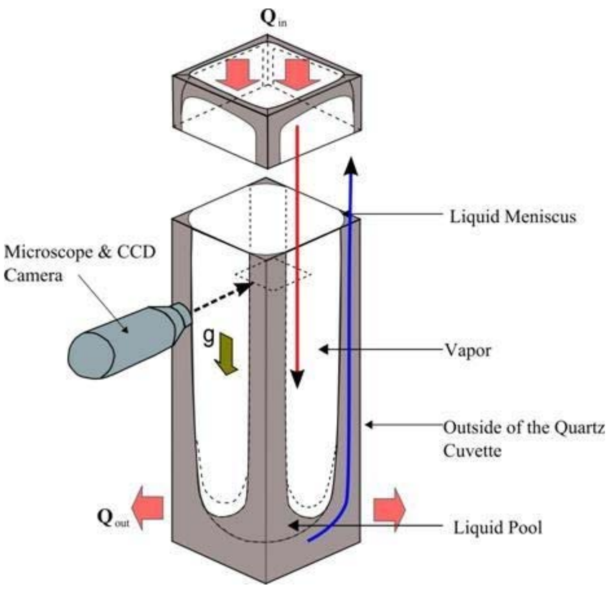


Figure 1. The Constrained Vapor Bubble and Liquid Pool

The CVB is essentially a wickless heat pipe. It consists of a fused silica cuvette that is partially filled with a working fluid - pentane in this case. The pentane forms a pool at one end and rises up the four corners of the cuvette due to capillary action (see Figure 1). A thin film of liquid adsorbs on the four flat faces of the cuvette; while the remaining volume of the cuvette is filled with pentane vapor. Thus, the central bubble volume is surrounded by liquid on all sides - the corners have a liquid meniscus while the flat faces have a thin adsorbed film. Since the bubble is effectively constrained by the walls of the cuvette, the experiment is named - Constrained Vapor Bubble.

Four CVB modules were run on the International Space Station. This paper deals with unexpected results from the run of the 20 mm module. Following launch, the vapor bubble of the CVB must be coaxed into its central position within the cuvette by subcooling the region behind the liquid pool and turning the heater on to drive the vapor toward the heater end. In general this process worked reliably however initial attempts with the 20 mm, the shortest module, failed and instead of aggregating the bubble, a series of individual nucleation events occurred as shown in Figure 2. These events occurred as superheats at the heater end of the device exceeded the 50 ˚C. Figure 3 shows temperature and pressure traces for one of these events and the purpose of this paper is to present the results and some analysis of the behavior.

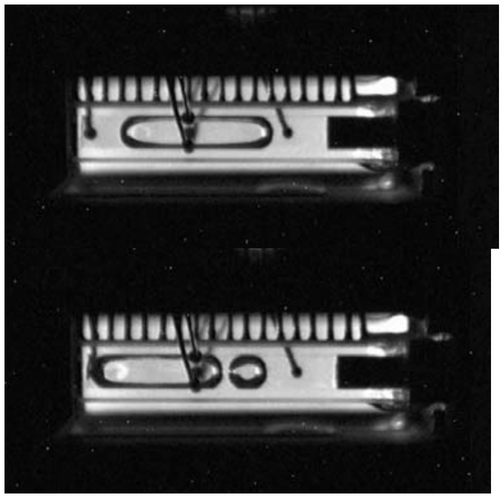


Figure 2 Nucleation and splitting of vapor bubble in the CVB

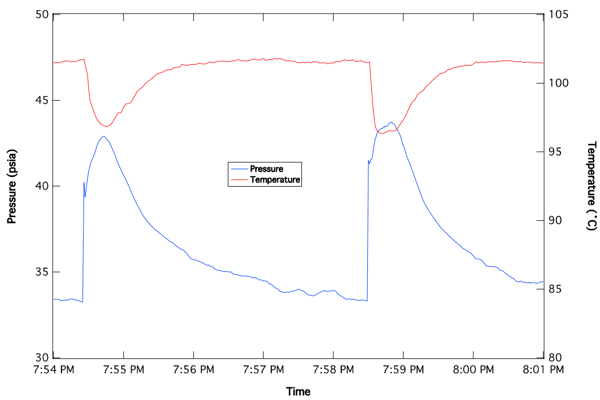


Figure 3 Pressure and temperature traces for two nucleation events.

References

[1] Chatterjee, A., et al., Constrained Vapor Bubble Experiment for International Space Station: Earth's Gravity Results. Journal of Thermophysics and Heat Transfer, 2010. 24(2), 400.

Development of High Efficient Heat Transport System for Electronics Devices by Using Flow Boiling

Kobayashi Hiroyuki[1], Yuka Asada[1], Shouhei Kanazawa[1], Yasuhisa Shinmoto[1], Haruhiko Ohta[1],
Masato Fukagaya[2], Yoshiyuki Abe[3], Mayumi Ouchi[3], Masahide Sato[4] and Ken-ichi Iimura[4]

[1]Department of Aeronautics and Astronautics, Kyushu University, 744 Motooka, Nishiku, Fukuoka 819-0395, Japan
*E-mail:koba@aero.kyushu-u.ac.jp
[2]SOHKi Co., Ltd., 2-42-4 Otemachi, Minato-ku, Nagoya, Aichi 455-0046, Japan
[3] National Institute of Advanced Industrial Science and Technology, 1-1-1 Umezono, Tsukuba, Ibaraki 305-8568, Japan
[4]Utsunomiya University, 7-1-2 Yoto, Utsunomiya, Tochigi 321-8585, Japan

Introduction

Heat generation density from data server is rapidly increased because of recent prevalence in communication systems supported by the development of electronic technology. Existing cooling system which consists of built-in cooling fan arrays and air conditioners, however, cannot meet cooling requirements in the near future. In addition, this system requires huge power consumption for the operation of air conditioners, and the power becomes around a half of total power consumption for the entire data center operation. The system newly proposed here utilizes the temperature difference between the possible semiconductor temperature and ambient temperature outside the data center, and the waste heat is directly transferred from the server rack to the outside air without using the air conditioners. Two-phase flow can realize the transfer of a large amount of waste heat minimizing the required temperature difference by the latent heat transportation. Additionally it can transport the waste heat along a long distance without the reduction of fluid temperature. This paper reports the performance of a developed cooling jacket directly attached to the semiconductors. It has a devised structure with auxiliary unheated channels for additional liquid supply to increase both of the heat transfer coefficient and critical heat flux.

Apparatus and experimental conditions

The structure of cooling jacket is shown in Fig. 1. The cooling liquid is supplied via sintered porous metal plates from the auxiliary unheated channels located on both sides of a main heated channel. The grooves machined on the heating surface support the liquid supply. The grooves have dimensions of 0.5mm in depth, 1mm in pitch and 90 deg in apex angle. The structure avoids dryout phenomena by the substantial reduction of heated length.

The heating surface with a length of 30mm and a width of 30mm was installed horizontally facing upwards in the cooling jacket and the gap size of narrow channel defined between the tip of grooves and a cover plate is kept at 2mm. FC72 was selected as test liquid to ensure electric insulation from semiconductors. The heat flux is evaluated ignoring the existence of the grooves, and surface temperature is defined at the tip of grooves. Because local dryout was frequently observed at upstream of the main heated channel, liquid supply from the auxiliary channels is regulated by adjusting the exposed area of porous metal plates keeping their porosity unchanged.

Experimental results and discussion

Experiments were conducted varying the exposed area $A_{suppl.}$ for liquid supply from auxiliary unheated channels as 28mm^2×2, 38mm^2×2, 48mm^2×2, 60mm^2×2 (Fully open). The values of critical heat flux are shown in Fig. 2. The highest value of CHF is obtained for $A_{suppl.}$ = 38mm^2×2. The critical heat flux for the case of fully open is the lowest due to the upstream dryout. The heating surface temperatures are compared in Fig. 3. The heating surface temperature evaluated at the center of the main heated channel is increased with the enlarged exposed area for liquid supply from auxiliary unheated channels. The result confirms the increased supply of liquid at downstream locations with increasing exposed area of sintered plates. An optimum distribution of liquid flow rate from the auxiliary channels seems to exist for the maximum heat removal avoiding the increase of surface temperature under the condition of a constant total flow rate of supplied liquid.

Acknowledgment

The experiments were conducted under the program of NEDO (New Energy and Industrial Technology Development Organization). The authors express appreciation for the financial support.

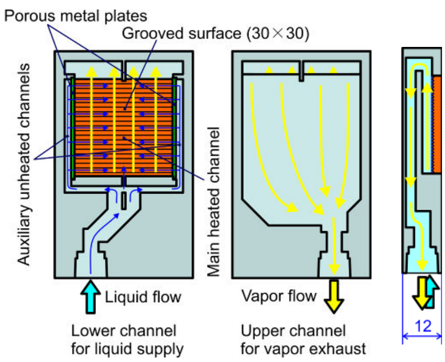


Figure 1: Structure of cooling jacket

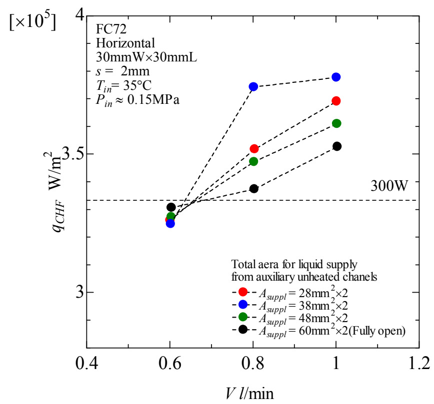


Figure 2: Critical heat flux

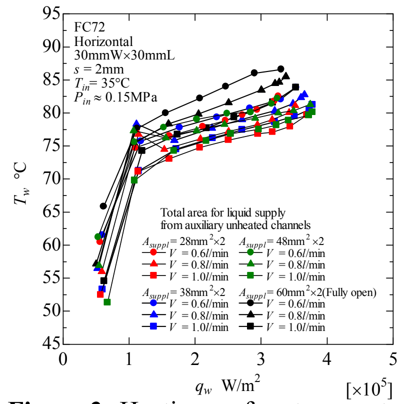


Figure 3: Heating surface temperature

Flow boiling in straight tubes in microgravity

De Malmazet Erik, Colin Catherine

Institut de Mécanique des Fluides de Toulouse, University of Toulouse
Allée du professor Camille Soula, Toulouse, 31400, France
colin@imft.fr

Flow boiling in microgravity is of a great interest for several practical applications. In the present study the relevance of using a circular tube evaporator for cooling electronic devices of a satellite is investigated. The main constraints are to minimize the pressure drop along the evaporator and to maintain the electronic components below a given critical temperature. The chosen refrigerant R245fa should be circulated at a saturation temperature close to 45°C in a tube of 12mm diameter with mass fluxes G between 50 and 200 $kg/m^2/s$. The quality x is around 0.3 at the evaporator inlet and 0.9 at the outlet. The wall heat flux ranges between 0.5 and 5 W/cm^2. In Figure 1, the range of liquid and vapour flow rates of the present study is plotted in a dimensionless flow pattern map. Re_{LS} and Re_{GS} are the liquid and vapour Reynolds numbers respectively, based on the tube diameter, the superficial velocities and kinematic viscosities of liquid and vapour. In figure 1, the flow pattern observed in microgravity for air-water flow (Bousman & Dukler 1993, Lowe & Rezkallah, 1999), Refrigerant R12 (Reinarts 1993) and Refrigerant R114 (Chen & al., 1991) are also plotted. For the present study, the flow pattern is expected to be annular.

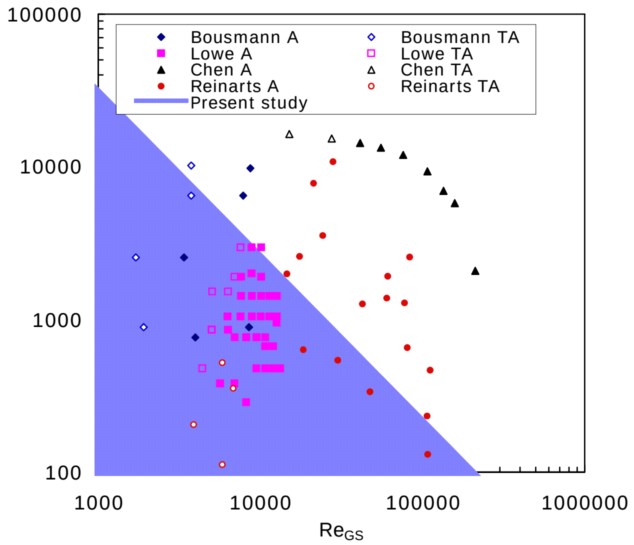


Figure 1: Flow pattern map for two-phase flows in microgravity at high qualities. A= annular flow, TA = transition between slug and annular flow. Blue line limit = present study.

In annular flow heat transfer is directly related to the liquid film thickness δ at the wall. In absence of liquid droplet entrainment, the film thickness can be expressed versus the void fraction in the pipe section α. In order to calculate the mean void fraction, the momentum balance equations for the liquid and the vapour phases can be solved after modelling the wall and interfacial shear stresses. The heat transfer through the turbulent liquid film can be calculated by assuming that inertia terms in the enthalpy balance equation are negligible. In microgravity the film thickness is smaller than in vertical upflow on ground and the heat transfer coefficient is therefore larger than in normal gravity. Several assumptions have been made in the modelling and more precise closure laws will be obtained from the experiments.

A two-phase flow loop for the study of flow boiling has been built at IMFT for performing experiments in vertical flow in normal gravity and in microgravity in parabolic flights. The test section is a sapphire tube of 6 mm dia. and 1mm thick with an ITO coating on its outer surface. The coating is heated by Joule effect and its temperature is measured in several locations by PT100 sensors. High-speed video pictures of the flow are taken with a PCO 1200HS imager. The pressure drop is measured along the test section with a differential pressure transducer Valydine P305D. The mean void fraction upstream and downstream the test section is measured by capacitance probes developed and carefully calibrated at IMFT. Joint measurements of pressure drop and mean void fraction allow to access to the wall and interfacial stresses. The wall heat transfer will be deduced from the wall heat flux and wall temperatures measurements. Results obtained in normal and microgravity conditions are compared to existing models. For the experiments, we use the refrigerant HFE7000 for safety reasons in the aircraft. The main characteristic dimensionless numbers are the same in the present experiments and the evaporator of the satellite.

References

Bousman W.S., Dukler A.E., Studies of gas-liquid flow in microgravity: void fraction, pressure drop and flow patterns, Proceedings of the 1993 ASME Winter Meeting, New Orleans, LA, December (1993.)

Chen, I., Downing, R., Keshock, E., and Al-Sharif, M.,, Measurements and Correlation of Two-Phase Pressure Drop under Microgravity Conditions, Journal of Thermophysics and Heat Transfer, 5 (4), pp. 514-523, (1991).

Lowe D.C., Rezkallah K.S., Flow regime identification in microgravity two-phase flows using void fraction signals, International Journal of Multiphase Llows, 25, pp 433-457 (1999).

Reinarts T.R., Adiabatic two-phase flows regime data and modelling for zero and reduced (horizontal) acceleration fields, PhD dissertation, Univ of Texas A&M (1993).

Non-Wetting drops as friction-less suspensions for space applcation

Fabio Peluso, Francesco Pezzuti, Salvatore Sorrentino

Telespazio SpA, A Finmeccanica/Thales Company
Sede di Napoli, Via E. Gianturco 31, 80146 Naples, Italy
fabio.peluso@telespazio.com

The phenomenon of permanent inhibition of coalescence due to termocapillary forces is known since several years. After their accidental detection during the space mission D1 while the science team was studying the behaviour of a silicon oil bridge in weightlessness, the phenomenon has been reproduced on ground with some limitations due to the presence of gravity that, mainly prevents large drops formation. However since then it has been extensively studied and scientific explanation provided and published in several papers. Lubrication due to air thus drives the phenomenon preventing the two drops actually being in contact.

In the present paper we want to show the potentiality of an industrial application of such physical phenomenon. Non coalescence inhibition may be fruitfully exploited also with a drop opposing to a solid surface; in this case one may assist to the "non wettability", simply establishing and keeping the drop and the solid body at a suitable temperature difference.

If more drops are formed in weightlessness, the non wettability allows a large and massive body (several kilograms, depending on the number of suspending drops) to be suspended by several drops. In such a way one may suspend a system without mechanical constraints, thus allowing its movement without friction. The application of such idea has brought to design a new class of facilities in which the scientific payload is located inside a sphere, this one being suspended by means of (at least) 4 drops. The sphere is thus isolated from vibrations transmitted by the main structure, and is also capable of rotating without friction following a force field external to the system. Data and commands are transmitted wireless to and from the sphere.

The system may follow, for instance, the residual gravity present on orbiting platforms, thus having a faint, continuous field affecting the experiment, instead of a rotating one.

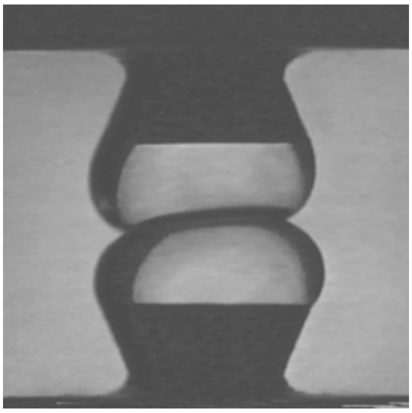


Figure 1: Picture showing non coalescing droplets of silicon oil. The effect is due to an air film trapped by themocapillary motions in between the two droplets..

A Pre-Qualification model of this new class of facilities, named GLOBE (that stays for Gas Lubricated Oil Bearings) is being currently under realization, thanks to a joint ESA-ASI funded project. An analytical model of the facility has shown that its performances are just as expected: the facility may actually provide a unique environment for those experiments designed for microgravity and that are particularly sensitive to vibrations and/or to rotating fields

References

G. Paul Neitzel and Pasquale Dell'Aversana, Non coalescence and non wetting behaviour of liquids, Annual Rev. Fluid. Mech.

Dell'Aversana, P, Banavar, JR, Koplik, J. 1996. Suppression of coalescence by shear and temperature gradients. Phys. Fluids 8:15-28

Dell'Aversana, P, Monti, R, Gaeta, FS. 1995. Marangoni flows and coalescence phenomena in microgravity. Adv. Space Res. 16:(7)95-(7)8

Dell'Aversana, P, Neitzel, GP. 1998. When liquids stay dry. Physics Today 51:38-41

Dell'Aversana, P, Tontodonato, V, Carotenuto, L. 1997. Suppression of coalescence and wetting: the shape of the interstitial film. Phys. Fluids 9:2475-85

Dell'Aversana, P, Tontodonato, V, Neitzel, GP. 2001. Coalescence pressure and load support measurements for non-coalescing and non-wetting systems.

Monti, R, Savino, R. 1997. Correlation between experimental results and numerical solutions of the Navier-Stokes problem for noncoalescing liquid drops with

Marangoni effects. Phys. Fluids 9:260-2
Monti, R, Savino, R, Tempesta, S. 1998. Wetting prevention by thermal Marangoni effect. Experimental and numerical results. Euro. J. Mech. B/Fluids 17:51-77

Napolitano, L, Monti, R, Russo, G. 1986. Marangoni convection in one- and two-liquids floating zones. Naturwiss. 73:352-5

Neitzel, GP, Dell'Aversana, P, Tontodonato, V, Vetrano, MR. 2000. Principles, limits and microgravity applications of self-lubricated liquids. First Int'l. Symp. Microgravity Res. Appl. Phys. Sci. Biotech., Sorrento, Italy.

Effect of Gravity on One-Component Two-Phase Flow Characteristics during Parabolic Trajectory Flight

Hitoshi Asano*, Yuuki Kanai*, Soumei Baba**, Yuto Wada**, Yasuhisa Shinmoto**, Haruhiko Ohta**, Tatsuya Hamada[+], Kenji Saruwatari[+], Osamu Kawanami[+], Kiyosumi Fujii[++]

*Kobe University, Department of Mechanical Engineering
1-1 Rokkodai, Nada, Kobe, 657-8501, Japan, E-mail asano@mech.kobe-u.ac.jp
** Kyushu University, Department of Aeronautics and Astronautics, 744 Motooka, Nishi, Fukuoka, 819-0395, Japan
[+]University of Hyogo, Department of Mechanical and System Engineering, 2167 Shosha, Himeji, Hyogo, 671-2280, Japan
[++]Japan Aerospace Exploration Agency, 2-1-1 Sengen, Tsukuba, Ibaraki, 305-8505, Japan

Two-phase flow loop system has been attracting attention as a cooling and temperature control system for space structure. The understanding of two-phase flow dynamics under micro gravity is necessary for design of the system. Experiments at Japanese Experimental Module "Kibo" in International Space Station are planned to clarify the effect of gravity on heat transfer and flow characteristics in a two-phase flow loop. Gear pump driven two-phase flow loop will be used in the ISS experiment. As a preliminary experiment to understand the effect of gravity change on the loop operation, two-phase flow loop experiments were conducted under the changing gravitational field during a parabolic trajectory flight of airplane. In this presentation, the effect of gravity on adiabatic one-component two-phase flow characteristics was evaluated.

Test section was a transparent acrylic tube with 4 mm in inner diameter. The test tube was placed horizontally in the longitudinal direction of the airplane. Fluorinert FC-72 (boiling point: 55.7 ºC, density ratio: ρ_L/ρ_G =120.5) was used as the working fluid. FC72 was supplied by a gear pump from a reservoir via a pre heater and two-types of the heat transfer test section. The experimental results on boiling heat transfer will be presented in the another presentations by Kawanami et al. and Baba et al. Flow behaviors were observed in horizontal direction by a high frame rate video camera (1000 fps, exposure time 100 µs, recording period 1.0 s) and a high sensibility CCD camera (30 fps, exposure time 10 µs, continuous video image).

Flow pattern observed under microgravity was classified into intermittent flow, semi-annular flow, and annular flow. Frames taken by the high frame rate video camera were shown for each flow pattern in Figs. 1 (a) to (c), respectively. For the intermittent flow, bubble shape difference between the head and end meant velocity difference between phases. In the semi-annular flow, a wave with low velocity and relatively large thickness was observed at the position indicated by the arrow in Fig.1 (b). The wave was not axial symmetry. For the annular flow, many ripple waves were observed between disturbance waves.

Flow pattern map of two-phase flows under microgravity is shown in Fig. 2. The solid line of $Fr^*=5$ is the boundary transition for vertically upward two-phase flows of FC72 in the same diameter tube under normal gravity. The tendency was almost the same as the results in our previous report at the last workshop in Kyoto, 2010.

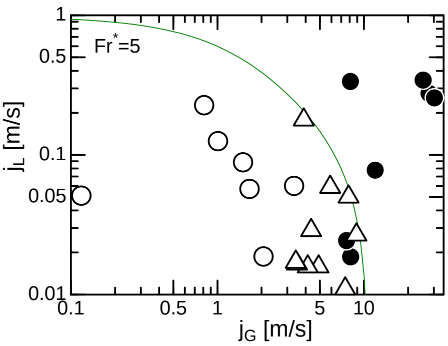
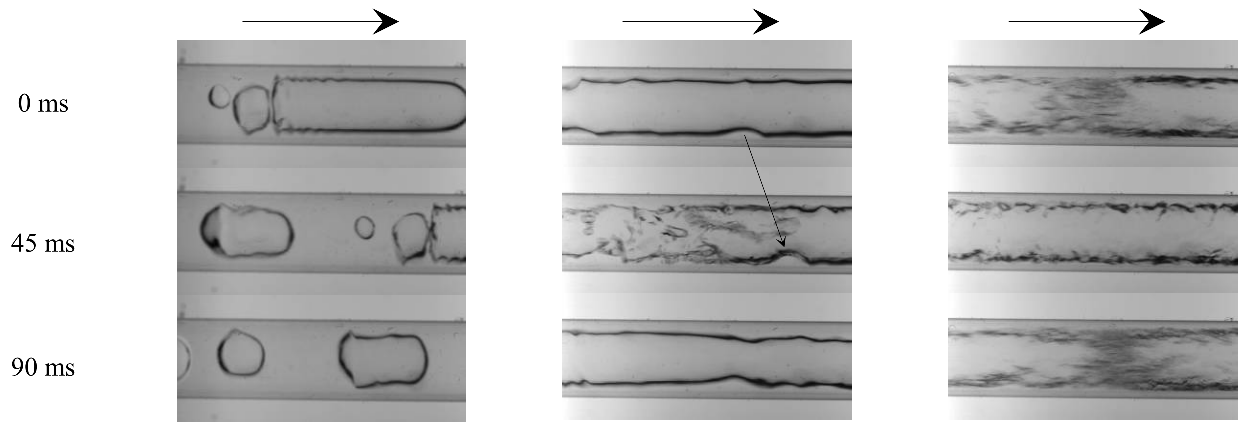


Figure 2: Flow pattern map (○ intermittent flow, △ semi-annular flow, ● annular flow)



(a) Intermittent flow
G=104 kg/(m^2・s), x=0.066
j_L=0.057 m/s, j_G=1.66 m/s

(b) Semi-annular flow
G=47.4 kg/(m^2・s), x=0.431
j_L=0.016 m/s, j_G=4.90 m/s

(c) Annular flow
G=188 kg/(m^2・s), x=0.288
j_L=0.078 m/s, j_G=11.9 m/s

Figure 1: Flow behavior observed by high frame rate video camera.

Rayleigh-Taylor instability of Newtonian and Viscoplastic fluids in microgravity

Aleksey Doludenko and Eduard Son

Joint Institute of High Temperatures, Russian Academy of Sciences
13, Izhorskaya, Moscow, 125412, Russia
adoludenko@gmail.com

Flows of mediums with the complicated rheology represent an important class of fundamental and applied problems. Fields of application include chemical, metallurgical, food industries and other fields. In practice such behavior of fluids is manifested everywhere and it is possible to give a lot of examples: liquid cement mortars, suspensions, various solutions of the polymeric compounds, solidifying lava, plasticine, heavy naphthas and avalanches, cosmetic creams and gels, liquid chocolate and various pastes. Even human blood is a viscoplastic fluid.

Blood can be considered as an example of structure in viscoplastic fluids. Early researches of blood viscosity were grounded on the fact that blood is a viscous fluid, but actually blood is a suspension of cells in a fluid. Non-newtonian properties of blood depend on many parameters: viscosity of plasma, deformation of erythrocytes, formation of aggregations and etc. At low shear rates viscosity is high: erythrocytes are packed in stacks and in the way of fluxion, at high shear rates red little bodies are arranged along a stream, and viscosity is minimum.

The Rayleigh-Taylor (RT) instability of a viscoplastic fluid is discussed. Bingham model (BM) as an effective rheological model including plastic effects is taken into account [1]. For numerical simulation one-mode disturbance of the contact surface between two fluids is considered. The main goal was to construct numerical 3D model and to obtain the relation between the development of instability and the yield stress.

In addition the Rayleigh-Taylor (RT) instability both of Newtonian and viscoplastic media is considered in 3D geometry with multi-mode disturbance of the contact interface in the field of microgravity. There are a lot of questions: what are the differences between Newtonian and viscoplastic fluids from the view of Raleigh-Taylor instability in microgravity (Fig. 1 and Fig. 2, for example, is for normal gravity)? What are the conditions forced the Bingham medium to the instability? Which properties of Bingham fluid are most important in the instability development?

In approach of "the deep water", neglecting viscous and surface effects, the width of the intermixed stratum grows under the law

$$h = \alpha A g t^2,$$

where g acceleration, t a time, A Atwood number. For Newtonian fluid the coefficient α is well known from the experiments. Its value varies from 0.05 to 0.09 approximately. For the Bingham fluid it is much less and depends on the yield stress.

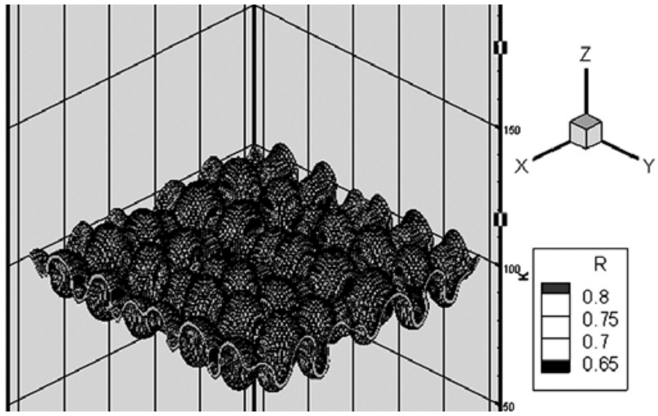


Figure 1: Bubbles of Newtonian fluid at small times of the Rayleigh-Taylor instability. Multi-mode approach.

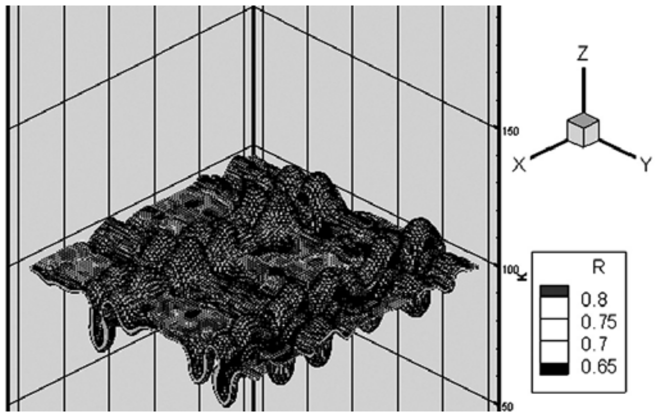


Figure 2: Bubbles of Bingham fluid at small times of the Rayleigh-Taylor instability. Multi-mode approach.

References

[1] Wang X., Gordaninejad F. Flow analysis of field–controllable, electro and magneto–rheological fluids using Herschel–Bulkley model // Journal of Intelligent Materials, Systems and Structures.— 1999.— Vol. 10, no.8.— Pp. 601 – 608.

Features of heat transfer in the pulse spray

Alexandr Nazarov, Anatoly Serov and Victor Terekhov

Institute of Thermophysics, Russian Academy of Sciences
prosp. Lavrentyev 1, Novosibirsk, 630090 Russia
nazarov@itp.nsc.ru, serov@itp.nsc.ru, terekhov@itp.nsc.ru

This paper represents experimental results on heat transfer between the pulse spray and flat heater. The effect of droplet grouping within the area of droplet-liquid mass of the pulse gas-droplet flow on heat transfer is shown.

The experimental setup (Figure1) consists of a heat exchanger with calorimeter and programmable controllable multinozzle source of the pulse spray with separate supply of liquid and gas phases.

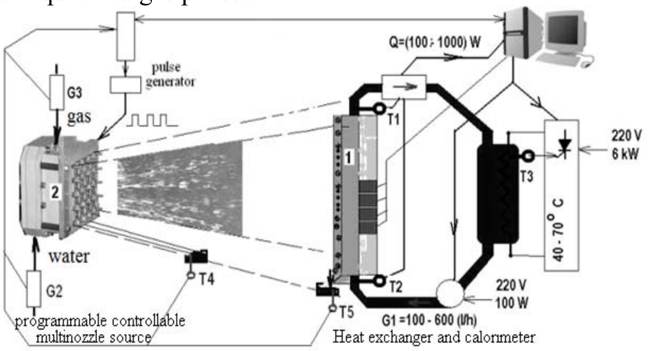


Figure1: Schematic of test facility.

Heat and mass transfer was measured under the atmospheric conditions and ambient temperature t_0 = 20 ^{0}C. The temperatures of air and liquid phases of aerosol were as follows: for liquid it was (7 ÷ 13)^{0}C; for air it was (20 ÷ 22)^{0}C; and the temperature of heat exchanger surface was t_w = const = 70 ^{0}C. The velocity of aerosol phases changed depending on the conditions at the initial distance (L ~ 60 mm): for air it changed within 0 ms^{-1} ÷ 25 ms^{-1}; and for water it varied within 1 ms^{-1} ÷ 20 ms^{-1}.

According to investigations of aerosol parameters, the liquid particles in the droplet area near the heat exchanger surface are separated predominantly into large drops (with the size of 120 ÷ 150 μm) and small ones (45 ÷ 50 μm). While moving from the source to the heat exchanger large drops group in the pulse "head", and the small ones group in its "tail", what leads to flow maldistribution in the region between the source and heat exchanger surface. Near the heat exchanger surface we can observe the extension of droplet area by the factor of 3-4 in comparison with duration of liquid electromagnetic valve opening. The local zones of increased concentration of liquid particles are formed within the droplet mass. Local maldistributions can be caused by redistribution of droplet mass concentration, splitting of droplets and their coalescence.

This maldistribution should effect the local heat transfer coefficient. The effect of local heat transfer maldistribution is especially obvious at evaporation cooling.

The typical curves of local heat flux for different velocities of the co-current air flow under the constant conditions of the liquid phase are shown in the figure: velocity is 5 ms^{-1}, opening of liquid valves of the source is characterized by frequency F_i = 1 Hz and duration T_i = 3 ms. In the diagram the heat flux curves are located successively with an increase in velocity of the co-current air flow. According to analysis of experimental data, maldistribution of the local heat flux is observed in the range of co-current gas flow velocities from 0 to 8 ms^{-1}. Under some conditions local maldistribution of heat flux can reach 50 % of its maximal value.

According to investigation of droplet grouping in the spray, with a rise of velocity of co-current air flow the pulse heat flux becomes predominantly uniform and its amplitude

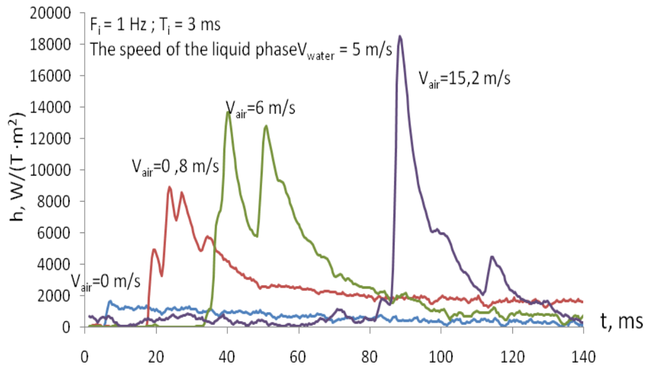


Figure2: The effect of velocity of co-current air flow on local heat flux.

can increase twice in comparison with the co-current gas flow of up to 8 ms^{-1} (see curve V_{air} = 15.2 ms^{-1}). This can be explained by the fact that with a rise of velocity of the co-current gas flow droplet mass redistributes. This leads to an increase in droplet concentration in the train "head", and when the droplet area meets the film surface deposited on heat exchanger, it breaks down intensively with formation of dry spots. Moreover, the clouds of small liquid particles are pressed to the surface by the air flow and interact both with the liquid film and dry spots on the heat exchanger surface, what adds efficiency to heat transfer.

References

V.I. Terekhov, A.F. Serov, A.D. Nazarov, and K.A. Sharov, Experimental investigation of the deposition of liquid droplets on the walls of a vertical cylindrical channel from wall gas-droplet jets, High Temperature, 2003, Vol.41, No.5, P.639-645.

Nazarov A.D., Serov A.F., and Bodrov M.V., Intensification of cooling by a pulsed gas-droplet flow. Equipment, parameters, and results. Technical Physics, Vol. 55, No.5, P.724–727 (2010).

Experiments of Flow Boiling Heat Transfer in Microgravity by using Parabolic Flight

Soumei Baba[1], Yuto Wada[1], Osamu Kawanami[2], Tatsuya Hamada[2], Kenji Saruwatari[2],
Hitoshi Asano[3], Yuuki Kanai[3], Yasuhisa Shinmoto[1] and Haruhiko Ohta[1]

[1] Kyushu University, Department of Aeronautics and Astronautics, 744 Motooka, Nishi-ku, Fukuoka, 819-0395, Japan,
E-mail: s_baba@aero.kyushu-u.ac.jp
[2] University of Hyogo, Department of Mechanical and System Engineering, 2167 Shosha, Himeji, Hyogo, 671-2280, Japan
[3] Kobe University, Department of Mechanical Engineering, 1-1 Rokkodai-cho, Nada, Kobe, Hyogo, 657-8501, Japan

Recent increase in the size of space platforms requires the management of larger amount of waste heat under high heat flux conditions and the transportation of it along a long distance to the radiator. Flow boiling applied to the thermal management system in space attracts much attention as promising means to realize high-performance heat transfer and transport. However, gravity effects on the two-phase flow phenomena and corresponding heat transfer characteristics have not yet been clarified in detail. In the present paper, flow boiling heat transfer characteristics in the microgravity environment are investigated by the parabolic flight experiments.

Outline of test loop is shown in Fig. 1. The experimental apparatus has two different heated test sections of a transparent heated tube and a metal heated tube, and unheated test sections of acrylic tubes placed backward of heated test sections. Inner diameter of these tubes is 4.0mm. Transparent heated tubes, which enable to heating, observation through the tube wall and the measurement of inner wall temperature simultaneously, is introduced to clarify the mechanisms of heat transfer by the relationship between liquid-vapor interfacial behaviors and heat transfer characteristics. The metal heated tube, on the other hand, is used for the measurement of critical heat flux and local heat transfer coefficients along a tube axis. The unheated sections enable to observe detailed liquid-vapor interfacial structures, especially of void fraction and distribution, and behaviors of annular liquid film, additionally to the measurement of pressure drop. Thermal conditions at the inlet of the test section are adjusted by a preheater. The total power input is removed in a condenser. Experiments on flow boiling with FC72 are conducted using cuppor heated tube. The experimental conditions are vertical upward flow, mass velocity G=40-673kg/m^{2}s, heat flux q=6-70kW/m^2, inlet pressure P_{in}=0.085-0.18MPa.

Figure 2 shows the heat transfer coefficient as a function of vapor quality under 1-g conditions in the experiment by using the metal heated tube. The transition from the region of nucleate boiling where the heat transfer coefficient is almost independent of vapor quality to the region of two-phase forced convection where the heat transfer coefficient is dependent on vapor quality is observed. At highest heat flux condition for G=60kg/m^{2}s, deterioration of heat transfer due to dryout is observed. The heat transfer coefficient is almost independent of mass velocity in the nucleate boiling region, and independent of heat flux in the two-phase forced convection region. In the nucleate boiling region, the relation of heat transfer coefficient α and heat flux q can be expressed as $\alpha \propto q^{0.78}$ as indicated by horizontal solid lines. The heat transfer coefficient is expressed as $\alpha = 260 G^{0.8}(1/X_{tt})^{0.5}$ in the two-phase forced convection region, where α and G are in W/m^{2}K and kg/m^{2}s respectively, and X_{tt} is Lockhart-Martinelli parameter (e.g. Dengler et al. 1956, Pujol et al. 1968).

The results in μg phase are shown in Fig. 3. In the nucleate boiling region the relation of heat transfer coefficient and heat flux can be expressed as $\alpha \propto q^{0.76}$. Heat transfer coefficients in microgravity are high compared to those under 1-g conditions. The heat transfer coefficient due to the two-phase forced convection is expressed as $\alpha = 260 G^{0.8}(1/X_{tt})^{0.5}$, i.e. the same relation for 1-g. Even at the vapor quality of unity for G=40 and 60kg/m^{2}s, deterioration of heat transfer due to dryout is not observed. The accuracy in the estimation of heat loss is to be improved by additional experiments.

References

Dengler, C.E., J.N. Addoms, Chemical Engineering Progress Symposium Series, Vol. 52 (18) pp. 95-103 (1956)
Pujol L., Stenning A.H., Symp. Ser. Can. Soc. Chem. Eng., Vol. 1 pp. 401-453 (1968)

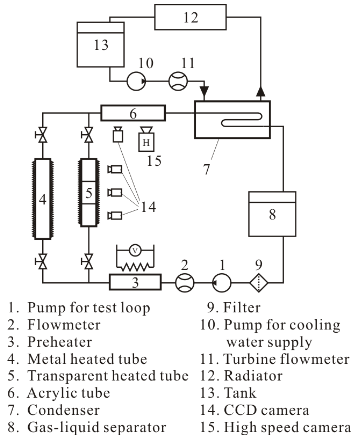


Figure 1: Experimental apparatus.

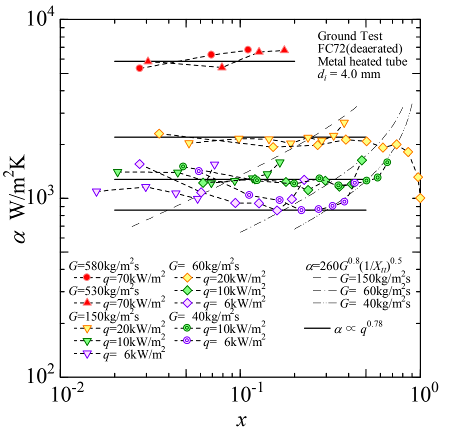


Figure 2: Heat transfer coefficient versus vapor quality under 1-g conditions.

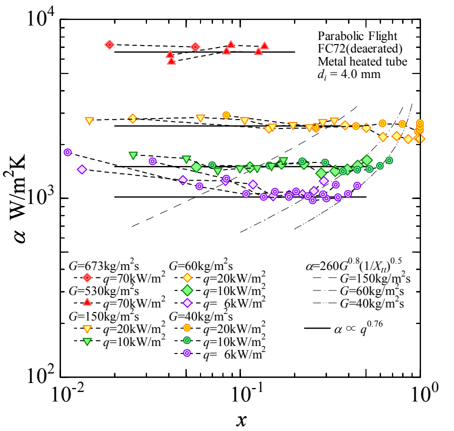


Figure 3: Heat transfer coefficient versus vapor quality under μg conditions.

Mathematical modeling of the convective flows with interfaces

Olga N. Goncharova[1,2], Carlo S. Iorio[3], Oleg A. Kabov[2,3]

[1] Altai State University, Faculty of Mathematics, Department of Differential Equations, Pr. Lenina 61, Barnaul, 656049, Russia
olga.n.goncharova@gmail.com
[2] Institute of Thermophysics, Russian Academy of Sciences, prosp. Lavrentyev 1, Novosibirsk, 630090 Russia
[3] Universite Libre de Bruxelles, Chimie-Physique EP-CP165/62, Microgravity Research Center, Av. Roosevelt 50, Bruxelles, Belgium
ciorio@ulb.ac.be, okabov@ulb.ac.be

The convective flows of the fluids in the open domains are often accompanied by the adjacent gas flows. The additional tangential stresses on a thermocapillary gas-liquid interface and the gas flow related evaporation effects should be taken into consideration in this situation. The physical experiments in the frame of CIMEX project of the European Space Agency should clarify some features of the convective flows of the fluids and their differences under conditions of the normal and low gravity. Mathematical modeling of convective fluid flows in the domains with free boundaries or with an interface subject to a condition of mass transfer is very important nowadays. The mathematical models of the various convective processes demand the study of the coupled problems of the gravitational and thermocapillary convection, interface processes including evaporation, heat and mass transfer processes in a co-current gas phase (Nepomnyashchy et al. 2002, Iorio et al. 2009, 2011).

In the present study, the common mathematical models and the general conditions to be applied at the interface between two interacting gas-liquid phases are presented. Scaling analysis and simplification of the mathematical models of convective fluid flows with evaporation is carried out. The three-dimensional exact solutions of the classical equations of convection (of the Oberbeck-Boussinesq equations) are presented (Pukhnachov 2000). Principal issues relating to well/ill posed initial boundary value problems of convection are discussed.

Our consideration is devoted to construction of the special exact solutions of the Oberbeck-Boussinesq equations under condition of action of a longitudinal temperature gradient. The liquid flows can be non-stationary and stationary. We confine ourself to investigation of a stationary solution in an infinite channel of a rectangular cross-section. The solution is characterized by dependence of the components of velocity on the transverse coordinates. The temperature and pressure have the terms similarly depending on the transverse coordinates. On the weakly deformed thermocapillary interface the kinematic and dynamic conditions and the conditions of continuity of velocity, temperature and thermal fluxes are assumed to be fulfilled. Analytical construction of the exact solutions of the stationary problems is added by the numerical investigations. By construction of a numerical algorithm the new unknown functions instead of the transversal components of the velocity are introduced. They are the stream function and vorticity. A possibility of a control of the convection mechanisms is investigated under conditions of normal gravity, microgravity and weightlessness.

The catalogues of the three dimensional flows in a channel are presented in the case of heat-insulated fixed boundaries and in the case of heating of one of the vertical walls. Creation of the transverse temperature gradient allows to change the character of flows and to study the heat transfer processes in the upper and lower fluids (or in the gas and fluid phases). The examples of the velocity fields are presented. The simulations presented in Fig. 1 are carried out for liquids like ethanol and nitrogen. Solutions allow demonstrating the three-dimensional effects in the non-axis-symmetrical problem. The differences in topology of flows in the lower and upper parts of the channel are confirmed by all the numerical experiments. The flows of both liquids can be qualitatively rather different: translational motion and progressively rotational flow are realized.

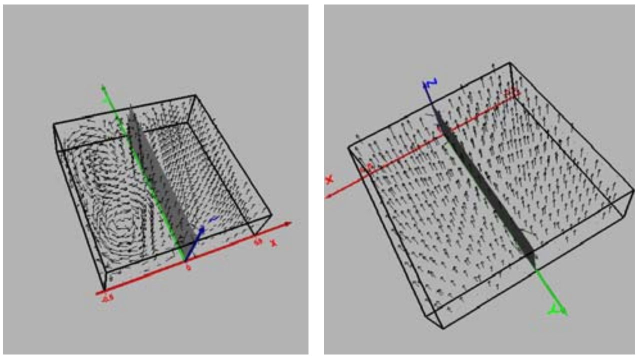


Figure 1: Velocity fields. Gr=10000, T=10, u=1 (left) Gr=1, T=10, u=1 (right). Here Gr is the Grashof number; T and u are the dimensionless longitudinal temperature gradient and ratio of the longitudinal and transverse characteristic velocities. The vector of gravity force is directed opposite to the X-axes.

Acknowledgments: The research has been supported by the Siberian Branch of Russian Academy of Sciences (Integrated projects No 64), the Russian Foundation for Basic Research (10-01-00007) and the European Space Agency (projects PRODEX CIMEX, Belgium).

References

Nepomnyashchy A.A., Velarde M.G., Colinet P., Interfacial phenomena and convection, Chapman and Hall / CRC. (2002)

Iorio C.S., Goncharova O.N., Kabov O.A., Study of evaporative convection in an open cavity under shear stress flow, *Microgravity sci. technol.*, Vol. 21(1), P. 313-319 (2009).

Iorio C.S., Goncharova O.N., Kabov O.A., Heat and mass transfer control by evaporative thermal pattering of thin liquid layers, *Submitted to Computational Thermal Sci*. (2011).

Pukhnachov V.V., Group-theoretical nature of the Birikh's solution and its generalizations. Book of Proc. "Symmetry and differential equations", Krasnoyarsk, P. 180-183 (2000) (in Russian).

NASA's Fluid Physics Program: Studies in the Early Stages of Development

John McQuillen and Brian Motil

Fluid Physics and Transport Branch
NASA Glenn Research Center
21000 Brookpark Road
Cleveland, OH 44135 USA

NASA's ISS Research Project is tasked with conducting both "exploration" and non-exploration research involving fluid behavior in the microgravity environment. While several fluid experiments are in the later phases of development and space-flight testing, such as the Capillary Flow Experiment, Investigating the Structure of Paramagnetic Aggregates From Colloidal Emulsions (InSPACE), Shear History Extensional Rheology Experiment (SHERE), Boiling Experiment Facility (BXF), and Coarsening in Solid-Liquid Mixtures (CSLM), there are several efforts in the early stages of development.

The Zero Boil-Off Tank Experiment (ZBOT) will examine the low gravity storage issues for volatile cryogenic liquids. To complement a computational fluid dynamic modeling study, ZBOT will use perfluoro-normal-Pentane as the liquid stimulant. Testing will include heating the bulk fluid within the tank and measure the self-pressurization from the thermal expansion of the liquid coupled with the increased evaporation rates. A second series of tests will examine the effect of mixing the fluid contents of the tank with an injection of cold liquid. Tank pressure, liquid temperatures, heater power and jet flow rate will be measured.

Observation and Analysis of Smectic Islands in Space (OASIS) will examine the behavior of freely suspended liquid crystals (FSLC) films FSLC films are formed from rod-shaped molecules that self-organize as bulk materials into fluid smectic LC phases: periodic stackings of layers in which each layer is a two dimensional (2D) fluid on the order of a molecular length in thickness. FSLC films exhibit a combination of physical characteristics that have made them uniquely exciting systems for the study of equilibrium and out-of-equilibrium phenomena in reduced dimensionality, for example liquid crystal ordering and fluctuations in two dimensions, and the effects of finite size on liquid crystal phase transitions. The microgravity experiments will explore thermocapillarity, the translational symmetry of the films and the absence of convection mitigating anomalous effects to enable detailed studies of the thermocapillarity of 2D fluids.

The Packed Bed Reactor Experiment (PBRE) will validate the hydrodynamics of two-phase flow in a packed bed reactor operating within a microgravity environment. Packed bed reactors are are critical components in life support systems; thermal control devices; fuel cells; and biological and chemical reactors because of the advantages that they offer including higher throughputs, compact design, operational flexibility and minimal power consumption. While these reactonrs are being currently being used, there are very few design and operational guidelines for the use of the porous media devices in a microgravity environment.

The Two-Phase Flow Separator Experiment (TPFSE) will examine the fluid mechanics on the efficiency of a cyclonic separator in reduced gravity. Through the tangential injection of a two-phase flow into a cylindrical-like geometry, it is possible to use the centrifugal acceleration to separate the gas and liquid phases. TPFSE will examine two different geometries to obtain baseline performance data that can be used to design these passive devices.

The Flow Boiling and Condensation Experiment (FBCE) will develop mechanistic models for flow conditions that would ensure insensitivity to gravity level for the important heat transfer mechanisms of flow boiling, dryout and condensation. The key objective is to identify the minimum flow requirements to ensure gravity independent flow boiling heat transfer, dryout and condensation.

An overview of these new efforts will be presented.

Evaporation of a thin liquid film in a microchannel

Yulia Kabova[1,2], Oleg Kabov[1,2,3,4], V.V. Kuznetsov[5] , T. Gambaryan-Roisman[1,6], P. Stephan[1,6]

[1] Center of Smart Interfaces, TU Darmstadt, Petersenstr. 32, 64287 Darmstadt,Germany
[2] Heat Transfer International Research Institute of ULB and IT SB RAS, CP 165/62, Avenue F.D. Roosevelt, 50, B-1050 Brussels, Belgium
[3] Institute of Thermophysics, Russian Academy of Sciences, prosp. Lavrentyev 1, Novosibirsk, 630090 Russia
[4] Universite Libre de Bruxelles, Microgravity Research Center, Avenue F.D.Roosevelt 50, Bruxelles, B-1050, Belgium
[5] Lavrentyev Institute of Hydrodynamics, RAS, prosp. Lavrentyev 15, Novosibirsk, 630090 Russia
[6]Institute for Technical Thermodynamics, Technische Universität Darmstadt, Darmstadt 64287, Germany

Gas-liquid systems with coupled heat and mass transfer are widely encountered in practice. Evaporation is the main heat transfer mechanism between liquid film and a gas phase at high heat fluxes. A number of experimental and theoretical works on getting in-depth understanding of the physics of nonisothermal thin liquid film flow driven by gravity and gas flow in a channel have been conducted, but complete understanding has not been achieved yet. In order to develop a deeper understanding of the 3D interface deformations mechanisms in heated thin films further investigations are needed.

Based on our previous investigations [1], 3D time-dependent two-sided mathematical model has been developed to understand the effects of nonuniform heating, Marangoni and evaporation on the hydrodynamics and heat transfer of shear-driven liquid films. The free interface is supposed to be deformable. Evaporated substance is assumed to be an impurity in a gas, which does not affect the thermodynamic properties. The gas is assumed to be incompressible. The transport processes in liquid and gas are described by the Navier-Stokes, continuity, energy and diffusion equations. The upper wall is adiabatic and impermeable. At the free gas-liquid interface the kinematic, dynamic and thermal boundary conditions as well as conditions of local thermodynamic equilibrium are posed. Gas is assumed to be not water-soluble. At the bottom wall no slip condition and thermal boundary conditions of two different types are posed. The case of prescribed wall temperature T=T(x, y, t) and the case of prescribed heat flux on the heater have been studied. Effects of surface tension, temperature dependent viscosity, gravity, evaporation and thermocapillarity are taken into account.

Thin layer approximation technique is used. The problem is reduced to five governing equations for film thickness, temperature distribution in the liquid and gas, concentration of vapor in the gas and gas pressure. The initial parameters such as the initial film thickness, tangential stresses and the pressure drop, were found out as the solution of the problem of isothermal laminar co-current flow in the channel - Couette flow with nonzero pressure drop. Temperature and concentration equations are solved in a nonsimplified form (taking into account convective terms), because convection and conduction mechanisms are equally essential for heat transfer at intensive local heating.

Deformations of gas-liquid interface, evolution of speed and temperature fields are computed (see Fig.1). Calculations are carried out for water driven by the Nitrogen gas. Channel height and initial film thickness are equal to 250 μm and about 50 μm, respectively. Liquid and gas Reynolds numbers as well as heat flux or temperature on the heater are varied. Substrate is heat-insulated, i.e. heating intensity is equal to zero on the substrate, excluding heater region.

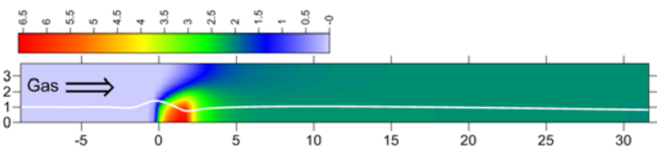


Figure 1: Temperature and gas-liquid interface profile on the axis of symmetry at y=0. Gas is moving in X-direction, heater upper edge is located in the origin of coordinates. Heater: 2 mm x 1.2 mm, $q=1W/cm^2$. Re=1, Re_{gas}=7,5.

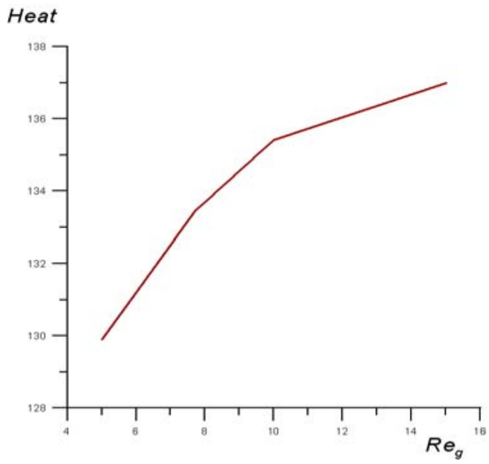


Figure 2: Total heat dissipation versus gas Reynolds number. Re=1, Ma=3.291 (at given temperature on the heater T=20ºC).

Our investigations have shown that governing mechanism of the two-phase heat transfer is evaporation. In the microchannel with evaporation Newtons law is inapplicable because its value depends on many parameters of the problem. But assumption that shear stress at the free surface is constant could be used in modeling. It is found out that total heat dissipation depends nonlinearly on gas Reynolds number with coefficient in the power less than 1 (Fig.2).

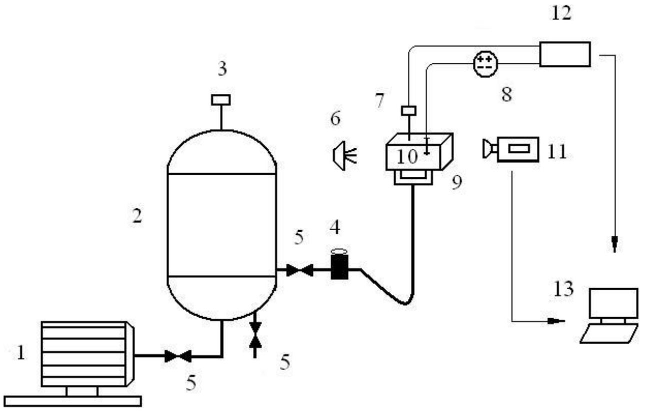
References: [1] Kabova Yu. O., Kuznetsov V.V., Kabov O.A. The effect of gravity and shear stress on a liquid film driven in a horizontal minichannel at local heating, Microgravity sci. technol. 2009, Vol. 21, Suppl. 1, P. S145-S152.

Evaporating-freezing Phenomena of Water Droplets during Quick Depressurization

Wang-Fang Du, Jian-Fu Zhao*

Key Laboratory of Microgravity (National Microgravity Laboratory)/CAS; Institute of Mechanics, Chinese Academy of Sciences
15 Beisihuan Xilu, Beijing 100190, China
*E-mail: jfzhao@imech.ac.cn

Flashing evaporation may occur in two cases. One is that the liquid is heated quickly so its temperature is much higher than the saturation temperature corresponding to the system pressure, the other is that the system pressure is suddenly dropped to a value far below its saturation pressure corresponding to the system temperature. Liquid discharging into vacuum environment during space orbital flight is one of important applications of the second case. Generally, the liquid jet quickly breaks into small droplets, and the rapid evaporation leads to a quick decrease of the droplet temperature, which may eventually lead to freezing of the droplets and possible blockage of the discharge pipe, and then cause the failure of the discharging process. The objective of this study is to experimentally investigate flashing evaporation process of water droplets released into vacuum.



1 - vacuum pump, 2 - vacuum chamber, 3,7 - pressure transducer, 4 - electro-magnetic valve, 5 - ball valve, 6 lighting, 8 - temperature transmitter, 9 - test vessel, 10 - thermocouple, 11 - high speed camera, 12 - NI data acquisition system, 13 - computer

Fig. 1: Schematic diagram of experimental system

A schematic drawing of experimental system is shown in Fig. 1. A droplet was hung in the test vessel with a thin thermocouple. Then the air inside the test vessel was evacuated quickly by opening the electro-magnetic valve connecting to the vacuum chamber. As soon as the quick depressurization began, small bubbles usually emerged within the droplet, which may come from not only the internal air separation, but also the thermocouple junction due to the existence of embryo core of nucleation center. The liquid droplet temperature dropped quickly, which was caused not only by the expanding of air around it but also by the quick evaporation on its surface. After the temperature exceeds some low limit, the droplet was frozen from the surface. Generally, the surface of the frozen droplet was not smooth. The coagulation contact surface moved from the droplet surface to its center. The liquid transformed the solid state, releasing the coagulation latent heat. Then the temperature rise to a value near the freezing point. After the whole droplet was frozen, its temperature dropped again due to the surface sublimation. Sometimes, the droplet may split during the freezing process, which was caused by the increase of the pressure inside the frozen surface.

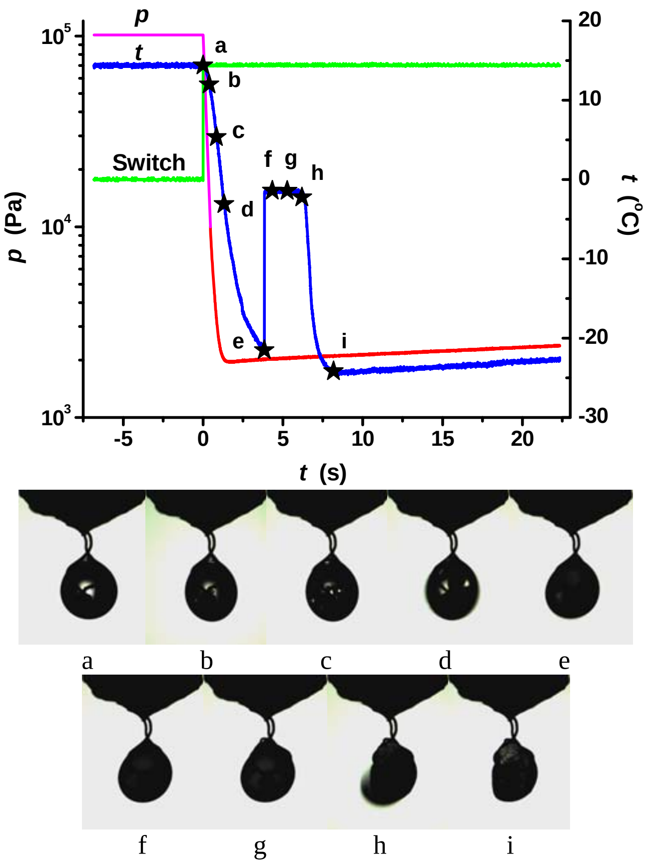


Fig. 2: Typical evaporating-freezing process of water droplets during quick depressurization

The thermal-dynamical behaviors of the evaporating-freezing process of water droplets during quick depressurization were also analyzed and discussed in the present work.

Acknowledgement: This work was supported by the Knowledge Innovation Program of the Chinese Academy of Sciences.

Sixth International Conference on TWO-PHASE SYSTEMS FOR GROUND AND SPACE APPLICATIONS, Cava de' Tirreni, Italy, September 25-28, 2011

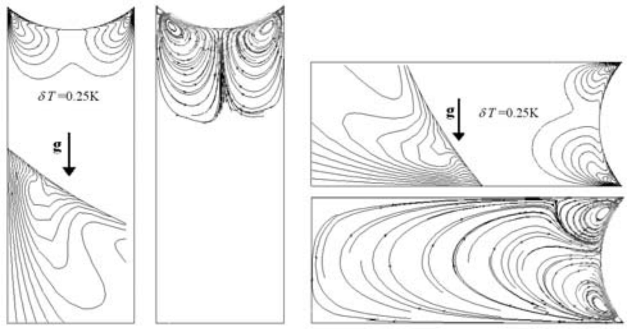
Numerical Simulation of Thermocapillary Flow Induced by Non-uniform Evaporating on the Meniscus in Capillary Tubes

Bo Lan[a], You-Rong Li[a,b], Dong-Ming Mo[a] and Deng-Fang Ruan[a]

[a] College of Power Engineering, Chongqing University, Chongqing 400044, China
[b] Key Laboratory of Low-grade Energy Utilization Technologies and Systems, Ministry of Education, Chongqing 400044, China
E-mail: liyourong@cqu.edu.cn

Understanding the evaporation characteristics on the meniscus in capillary tube is the key factor in design of micro-cooling devices, such as heat pipes, capillary pumped loops, compact and printed heat exchangers. Buffone *et al* (2005) performed a series of investigations on the evaporation process from a liquid meniscus formed in capillary tube with various sizes by using infrared thermography and micro-particle image velocimety technique. A very strong convection in the liquid is observed. The non-uniform evaporation from the meniscus leads to a temperature gradient along the interface causing a surface tension gradient, which is the driving mechanism for the observed convection. Dhavaleswarapu *et al* (2007) investigated steady buoyant-thermocapillary convection near an evaporating meniscus by using micro-particle image velocimetry technique for tube diameter ranging from 75 to 1575μm. A transition from a pure two-dimensional thermocapillary flow to a three-dimensional buoyant -thermocapillary flow is observed with the increase of tube diameter for horizontal tube. Wang *et al* (2008) investigated the mass transport inside an open micro-tube by using a generalized model which coupled the evaporation at a liquid-air interface with the vapor diffusion processes in air. The three-dimensional flow structure in the micro-tube is simulated with the effect of Marangoni convection, buoyancy and the influx of fluid to the interface. For horizontal tubes of diameter 100μm or larger immersed in a water bath, flow asymmetry due to buoyancy is observed.

In the present work, three-dimensional numerical simulation was developed to investigate the effects of tube size, evaporating heat flux and buoyancy on thermocapillary flow on the meniscus in capillary tubes. Capillary tube radius ranged from 0.1 to 1mm and the working liquid was methanol. The effects of tube size, evaporating heat flux and buoyancy on thermocapillary flow were investigated. The results show that the non-uniform evaporation on the meniscus leads to two opposite temperature gradients along the radial direction, which generate two thermocapillary flow vortexes under the meniscus. The lowest temperature on the meniscus decreases and the largest velocity increases with an increase in tube size. For horizontal capillary tube, the isotherms and path-lines in the vertical mid-plane lose symmetry, which is attributed to the combined buoyancy and thermocapillary effects, as shown in Fig. 1. For the vertical capillary tube, with increasing average evaporating heat flux, the steady axisymmetrical flow will undergo a transition to a steady asymmetrical flow, then turn to be a three-dimensional oscillatory flow, as shown in Fig. 2. In this case the critical evaporating heat flux is found to be 48125 W/m^2.



(a) vertical tube (b) horizontal tube

Figure 1: Meridional views of isotherms and path-lines for tubes of radius 1mm under different gravity conditions

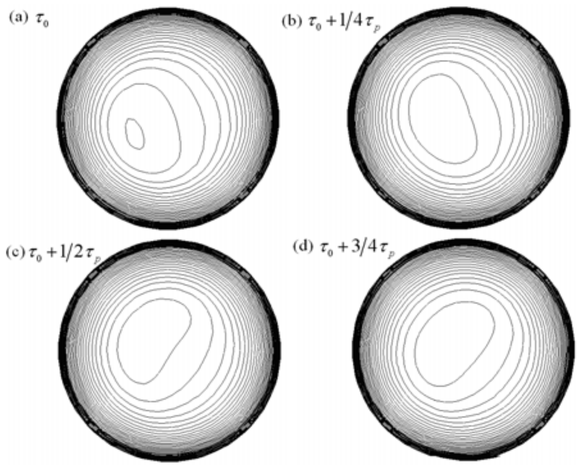


Figure 2: Temperature oscillation on the meniscus interface during a period at q_0=51985 W/m^2, n=6 and r_0=1mm.

References

Buffone, C., Sefiane, K.: Experimental investigation of self-induced thermocapillary convection for an evaporating meniscus in capillary tubes using micro-particle image velocimetry. Phys. Fluids **17**, 052104 (2005)

Dhavaleswarapu, H. K., Chamarthy, P., Garimella, S. V.: Experimental investigation of steady buoyant -thermocapillary convection near an evaporating meniscus. Phys. Fluids **19**, 082103 (2007)

Wang, H., Murthy, J. Y., Garimella, S. V.: Transport from a volatile meniscus inside an open microtube. Int. J. Heat Mass Transfer **51**, 3007-3017 (2008)

Evaporation of a water drop on an extended or narrowed heating surface

Mebrouk Ait Saada[1], Salah Chikh[1], Lounès Tadrist[2]

[1] Faculté de Génie Mécanique et de Génie des Procédés, USTHB
B.P.32, El Alia, Bab Ezzouar 16111 Alger, Algeria
[2] Aix-Marseille Université (U1, U2), Polytech'Marseille, Laboratoire IUSTI, CNRS UMR 6995,
Technopôle de Château Gombert, 5 rue Enrico Fermi, 13453 Marseille, France
Corresponding author: m_aitsaada@yahoo.fr

Evaporation of sessile drops on a hard support is a process which occurs in several situations of everyday life and is encountered in many fields of engineering such as metallurgical applications, electronic industries and fire suppression systems. A full understanding of physics and controlling mechanisms of this evaporation requires a trough analysis of heat and mass transfer across the interfaces as well as well as the interaction between the three phases coexisting.

Many authors investigated the evaporation of a sessile drop which usually occurs under a constant wetting radius and a variable contact angle or inversely, Erbil et al. (1999). Thermal properties of the substrate are shown to have an effect on the evaporation process. The experimental results of David et al. (2007) showed that the evaporation rate of sessile drops is limited by substrate thermal properties especially for high evaporation rates. Theoretical and numerical models were developed and used to study this problem Dunn et al. (2009).

In this work, a numerical approach is adopted to investigate heat and mass transfer phenomena for an evaporating water drop on a heated subtract. A diffusion model is developed, it accounts for heat conduction in both liquid and solid phases and for diffusion in the gas phase. The variation of the saturation concentration with temperature at the free surface of the drop is included for coupling the problems for the vapor concentration in the atmosphere and the temperature in the liquid and the substrate. The numerical results analyze in particular the influence of the heating zone of the substrate and give a comparison between the evaporation of a pinned drop and that of a de-pinned drop, to see the figure of the evaporation rate.

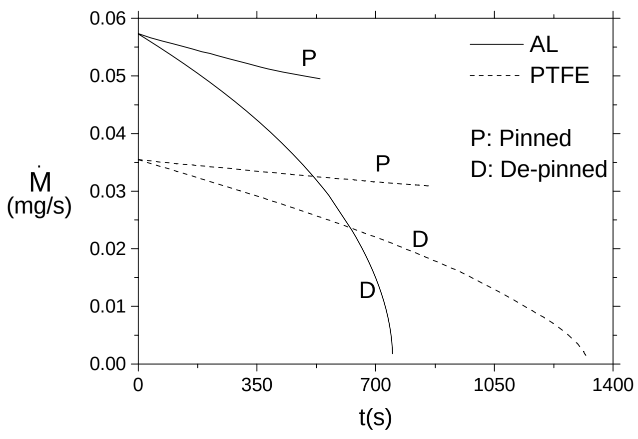


Figure 1: Evaporation rate versus time.

References

Erbil H.Y., McHale G., Rowan S.M. and Newton M.I., Analysis of evaporating droplets using ellipsoidal cap geometry, *J. Adh. Sci. Technol.*, 13, pp. 1375-1391 (1999).

David S., Sefiane K. and Tadrist L., Experimental investigation of the effect of thermal properties of the substrate in the wetting and evaporating of sessile drops, *Colloids and surfaces A: Physicochemical and Engineering Aspects*, 298, pp. 108-114 (2007),.

Dunn G.J., Wilson S.K., Duffy B.R., David S. and Sefiane K., The strong influence of substrate conductivity on droplet evaporation, *J. Fluid Mech.*, 623, pp. 329-351 (2009).

Modeling of vortex precession in water-air flow

A.A. Dekterev[1], A.A. Dekterev[1,2], A.A. Gavrilov[1,2]

[1]Institute Of Thermophysics SB RAS
Lavrentyev Ave. 1, 630090, Novosibirsk, Russia

[2]Siberian Federal University,
79 Svobodny Prospect, 660041, Krasnoyarsk, Russia
dek_art@mail.ru

Swirl flows are widespread in various energy and technical devices, including the applications with two-phase liquid-gas media. Feature of swirling two-phase flows is the separation of gas and liquid phases. Under the influence of centrifugal force gas is collected near the vortex axis, forming air core. Such vortices can be both stationary and precessing. The phenomenon of vortex core precession in single-phase flows is widely investigated theoretically and experimentally, but the features of two-phase swirling flows are yet poorly studied. In this work, based on numerical simulations we studied the effect of gas loading on the characteristics of flow in the vortex chamber. We used experimental data obtained on stand of Institute of Thermophysics SB RAS (Shtork et al. 1999) as a basis for our calculations. Figure 1 shows the computational domain of the vortex chamber.

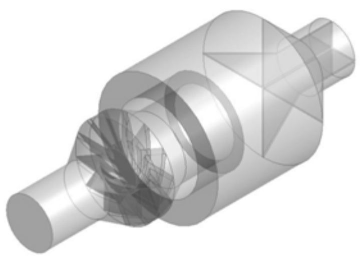


Figure 1: Geometry of computational domain

For solving the problem the in-house CFD code SigmaFlow and commercial packages were used. The flow was modeled using Reynolds-averaged Navier-Stokes equations. For turbulence modeling the Spalart-Allmaras detached eddy simulation (DES) method was used. Results of our colleagues have shown that this model gives good results for modeling swirled flows (Sentyabov et al. 2011). Simulation of the gas phase is carried out using the two-fluid Eulerian approach. The pressure coupling scheme was SIMPLEC and the algebraic multigrid solver was used as the equation solver. The implicit scheme of second order accuracy for time approximation was used.

In order to test models of incompressible flow the unsteady calculations of single-phase flow were performed. Water flow rate varied from 1 l/s to 3 l/s. Figure 2 shows a plot of precession frequency of the vortex core versus the water flow rate. It is evident that the dependence is close to linear. The calculation results are in good agreement with the experimental data.

In the second stage the two-phase flow calculations were carried out. Water flow rate was 2 l/s, air flow rate at the inlet varied from 0 to 0.5 l/s.

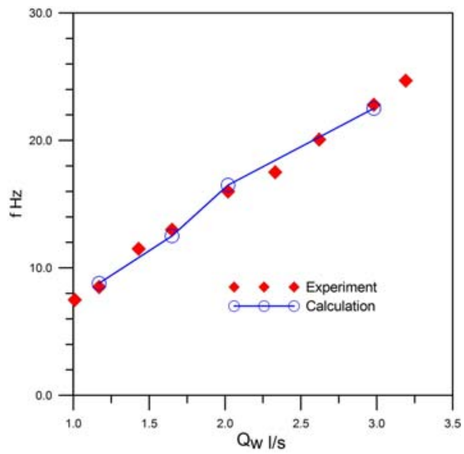


Figure 2: Frequency of precession vs water flow rate (single phase flow)

The results presented on Figure 3 show that used mathematical models provide a good agreement between calculation and experimental values for frequency of precessing liquid-air flow.

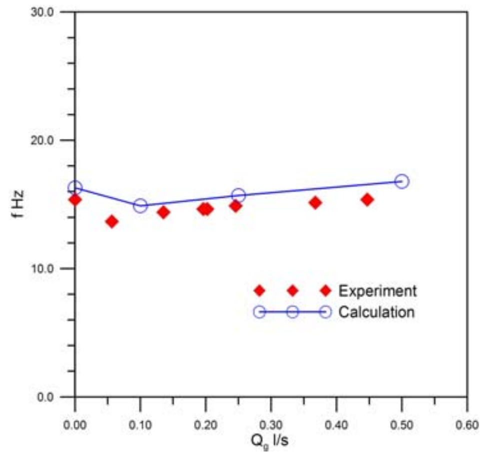


Figure 2: Frequency of precession vs gas flow rate (two-phase flow, Q_w=2 l/s)

References

Shtork S.I., Cherny I.S. Experimental study of the integral characteristics of swirl gas-liquid flow. Trans. Intern. Symp. "Actual Problems of Physical Hydroaerodynamics". Novosibirsk, Russia, 19 - 23 April 1999, p. II – 108 (1999)

Sentyabov A.V., Gavrilov A.A., Dekterev A.A. Investigation of turbulence models for swirled flows simulation. Thermophysics and aeromechanics. Vol 1, pp. 81-93 (2011)

Detached Eddy Simulation of cloud cavitation on a NACA0015 hydrofoil

Andrey Gavrilov, Alexander Dekterev

Institute of Thermophysics of SB RAS, Krasnoyarsk branch
50/44 Akademgorodok, ICM SB RAS, TORINS, Krasnoyarsk, 660036, Russia
gavand@yandex.ru

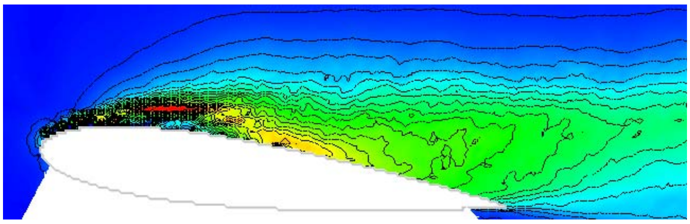
An unsteady turbulent cavitation flow over a hydrofoil at angle of attack is numerically simulated using three-dimensional incompressible, homogeneous multiphase Reynolds Averaged Navier-Stokes (RANS) method. To simulate cavitating flows, we adopt multi-phase single-fluid model including mass fraction transport equation. Two mass transfer models, also called cavitation models, are incorporated as source terms to model the transfer of mass from liquid to vapour and back (Singhal et al., 1997). In two-phase flows, the turbulence models might affect the cavity structure and cavitation dynamics. In the present work Unsteady Reynolds Averaged Navier-Stokes and Detached Eddy Simulation (DES) modeling are examined with the intent of elucidating the differences that these modeling approaches have in the prediction of flow structures and integrated quantities. The DES formulation in this study is based on a modification of the RANS SST model of Menter (Travin et al., 2002), such that it reduces to RANS close to solid walls and to Large Eddy Simulation away from the wall.

One of the main computational difficulties for cavitation is the large density ratio between the liquid and the vapor phases. The SIMPLE-like pressure-based method developed by (Senocak and Shyy, 2003) is used for time-dependant computations. This modification of SIMPLE procedures is achieved through the inclusion of a pressure-density coupling scheme into the pressure correction equation. The pressure–correction equation exhibit a convective–diffusive (compressible) nature in phase change regions and purely diffusive nature (incompressible) in the pure liquid or vapor phase.

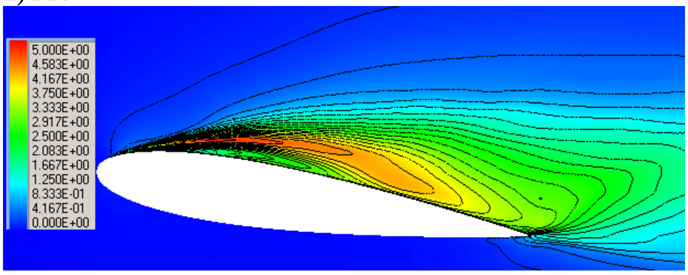
Two test cases are used for the validation of the mass transfer models: a two dimensional NACA6602 hydrofoil and a three dimensional NACA0015 hydrofoil. The 2D NACA6602 case is primarily used for analysis of the influence of model parameters, such as the grid resolution, the mass transfer models and the numerical interpolation schemes, on the simulation results. The unsteady cavitation flow field including transient cavity shedding is reproduced. The cavitation patterns, such as leading edge cavitation inception and reentrant jets, are reproduced well and show good comparison with the well-known experimental data.

A fully three-dimensional DES simulation was performed for NACA0015 hydrofoil at the attack angle 8.5° and cavitation numbers of 1.2. Highly vortex fluid motion such as a cavitation cloud was observed. There is a strong interaction between the large-scale vortex and cavitation. The separated shear layer rolls up, thus turning into a large-scale vortex. The large-scale vortex yields a low-pressure region at its center, while in the low-pressure region bubbles grow and remain.

Results from a DES and Particle Image Velocimetry (PIV) in the middle section of a cavitating NACA0015 hydrofoil were compared (Fig.1). The experiments and CFD simulations also captured a region of back flow (reentrant jet). Downstream, the agreement was not as good, but the same trends were still in evidence.



1) PIV



2) DES

Figure 1: Root mean square of the streamwise velocity fluctuations in the middle section.

References

Singhal, A.K., Vaidya, N. and Leonard, A.D., Multidimensional simulation of cavitating flows using a PDF model for phase change, ASME Fluids Engineering Division Summer Meeting, ASME Paper FEDSM97-3272 (1997).

Travin, A., et al., Physical and numerical upgrades in the Detached-Eddy Simulation of complex turbulent flows. In: Friedrich, R., Rodi, W. (eds.) Fluid Mechanics and its applications. Proc. Of EUROMECH Colloquium 412. Advances in LES of Complex Flows, vol. 65, pp. 239–254, (2002).

Senocak, I. and Shyy, W., A Pressure-Based Method for Turbulent Cavitating Flow Computations, J. of Comp. Physics, vol.176, pp. 363-383, (2002).

Turbulent flow in the T - shaped microchannel

Andrey V. Minakov[1,2], Valery Ya. Rudyak[3], Andrey A. Gavrilov[2], Alexander A. Dekterev[1,2],

1 Siberian Federal University, Kirensky 79, 660074, Krasnoyarsk, Russia
2 Institute of Thermophysics SB RAS, Lavrentieva, 1, 630090, Novosibirsk, Russia
3 Novosibirsk State University of Architecture and Civil Engineering, Leningraskaya, 113, 630008, Novosibirsk, Russia
Email: tov-andrey@yandex.ru

Fluid mixing in microchannels is an extremely important problem in various applications of microflows. In macroscopic flows, mixing usually occurs in a turbulent flow regime. Microflows, however, are usually laminar, and corresponding Reynolds numbers are not so large. However, in fairly large microchannels, the Reynolds number can be a few hundreds. These flows have many interesting features. The purpose of this work was to study the flow regimes and mixing efficiency of in a T-type micromixer at the high Reynolds numbers. The Reynolds numbers were varied from one to one thousand. The cross section of the mixing channel was 100 µm×200 µm, and its length was 1400 µm. The transverse inlet channels were symmetric to the mixing channel, and their cross-section was 100 µm×100 µm, and the total length was 800 µm. The simulation results lead to the following main conclusions.

At Reynolds numbers Re < 5, stationary vortex-free flow occurs in the mixer. At Reynolds numbers 5 < Re < 150, stationary symmetric vortex flow occurs in the mixer. In this flow regime, two symmetric horseshoe vortices form at the end of the mixer, which propagate into the mixing channel. The flow is totally symmetric and stationary. Despite the presence of horseshoe vortices at the entrance to the mixing channel, the interfaces remains virtually plane (Fig. 1).

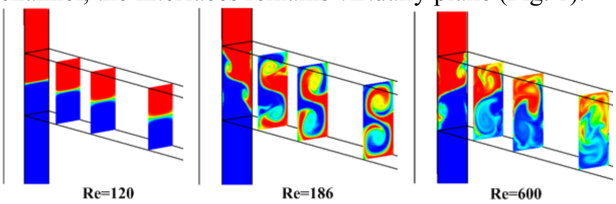


Figure 1: Mixing in the T-type micromixer.

At Reynolds numbers 150 < Re < 240, stationary asymmetric vortex flow occurs in the mixer. The horseshoe vortices formed at the entrance lose symmetry and rotate by 45 degrees to the central longitudinal plane of the mixer (see Fig. 1). Because of this, two equally swirling vortices form in the mixer. The flow nevertheless remains stationary. The presence of vortices in the mixer dramatically increases the mixing efficiency. It increases by a factor of 25 compared to the symmetric regime (Re <150) see Fig. 2 (left). The simulation results are in good agreement with experimental data [2].

At Reynolds numbers 240 < Re < 400, nonstationary periodic flow occurs in the mixer. The pulsation frequency of the velocity (and other characteristics) agrees with experimental data [3] with an accuracy of 1–2%. At Reynolds numbers 400 < Re < 1000, the flow in the mixer becomes stochastic. The S-shaped vortex structure observed at lower Reynolds numbers is destroyed (see Fig. 1). The destruction of the coherent structure leads to a sharp decrease in the mixing efficiency. Nevertheless, at such Reynolds numbers, one still cannot argue that a turbulent flow regime occurs. In the Fig. 2 the dependence of the mixing efficiency and pressure drop on the Reynolds number is presented.

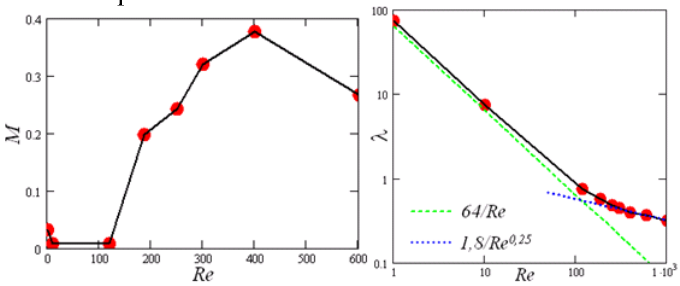


Figure 2: Mixing efficiency (left) and pressure coefficient.

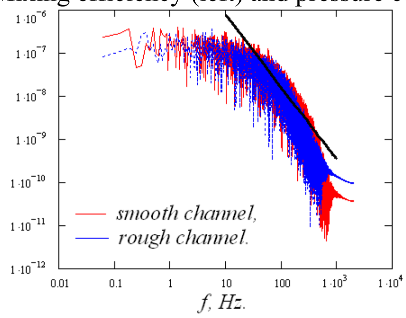


Figure 3: Spectrum of turbulent flow pulsations for Re=600.

Also in this work we studied the influence of wall roughness structure of flow at high Reynolds numbers, see fig.3. The significant effect of roughness on flow and mixing, and the laminar-turbulent transition has been shown.

This work was supported in part by the Russian Foundation for Basic Research (grant No. 10-01-00074) and the Federal Special Program "Scientific and scientific-pedagogical personnel of innovative Russia in 2009-2013" (projects No. P230, 14.740.11.0579).

References

Minakov, A.V., Rudyak, V.Ya., Gavrilov, A.A., & Dekterev, A.A. (2010) On optimization of mixing process of liquids in microchannels. J. of Siberian Federal Univ. Math. & Phys., 3, No. 2, 146-156.

Hoffmann, M., Schluter, M., & Rubiger N. (2006) Experimental investigation of liquid–liquid mixing in T-shaped micro-mixers using µ-LIF and µ-PIV. Chemical Engineering Science, 61, 2968-2976.

Dreher, S., Kockmann, N., & Woias, P. (2009) Characterization of Laminar Transient Flow Regimes and Mixing in T-shaped Micromixers. Heat Transfer Engineering, 30, 1-2, 91-100.

Experimental investigation on rapid evaporation of R-113 liquid during depressurization

Qiaoling ZHANG, Qincheng BI*, Jiangbo WU

Xi'an Jiaotong University, State Key Laboratory of Multiphase Flow
W 28 Xianning Rd, Xi'an, 710049, China
qcbi@mail.xjtu.edu.cn

Introduction

The objective of this study is to experimental investigate on rapid evaporation process of supercritical fluid R-113 during depressurization. Although the phenomena of fluid releasing into vacuum or low pressure have been discussed by many researchers(Owen et al. 1991, Hindmarsh et al. 2002), the pressure and temperature are not too high, especially the supercritical fluid to the low pressure few. Non-thermodynamic equilibrium phenomena such as rapid evaporation will occur in the process of depressurization, which is more complicated in supercritical fluid.

In this paper, the typical temperature transition of R-113 surface during rapid evaporation is experimental investigated, and the effects of initial environmental pressure and initial droplet temperature on temperature transition are summed up based on experimental data.

Experimental apparatus and method

A schematic diagram of experimental system is shown in Fig.1. During an experiment, R-113 liquid was deposited in a small round tube with a diameter 10mm in the test vessel. The predetermined pressure was provided by the high-pressure nitrogen gas, and the process of quick depressurization was started by opening the magnetic valve. Typical temperature history on the surface of liquid was measured by thermocouple, the environmental pressure in the test vessel was recorded by pressure transducer, and the shape variation was observed by high speed camera.

Results and discussion

Compare with the emission to the vacuum or low pressure, the non-thermodynamic equilibrium process of supercritical fluid emission is more complicated. Figure 2 compares the same initial temperature and the different initial ambient pressure effecting on the surface temperature of R-113, including both supercritical and subcritical pressure range. It was found that with the higher ambient pressure, the surface temperature of the liquid reduced faster, the minimum temperature being lower correspondingly. The temperature on the supercritical condition decreases much larger than that of sub-critical. In addition, the pressure change in the test vessel can be divided into two stages: firstly, a stage with fast pressure dropping for the first 1s after opening the valve. For this stage, a great deal of liquid quickly evaporates, absorbing the latent heat which causes the temperature of liquid surface decreases rapidly; secondly, the pressure reaches a constant value, and the liquid temperature decreases mainly due to the low temperature of the test vessel. In the second stage, because of the heat transfer between low ambient temperature and the liquid surface temperature, the temperature change was lagged.

Based on the experimental data, the influence of initial ambient pressure and initial liquid temperature are summed up. With the high initial ambient pressure and initial liquid temperature, the evaporation rate becomes fast, and the amount of the evaporation liquid increases.

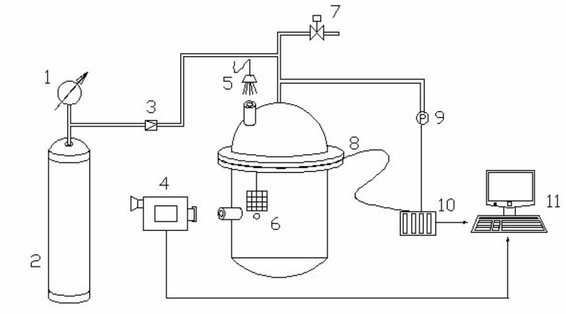


Figure 1: Schematic diagram of experimental system.
(1, pressure gauge, 2, nitrogen bottle, 3, relief valve , 4, high speed camera, 5, spotlight, 6, test vessel, 7, magnetic valve, 8, thermocouple, 9, pressure transducer, 10, NI data acquisition system, 11, computer.)

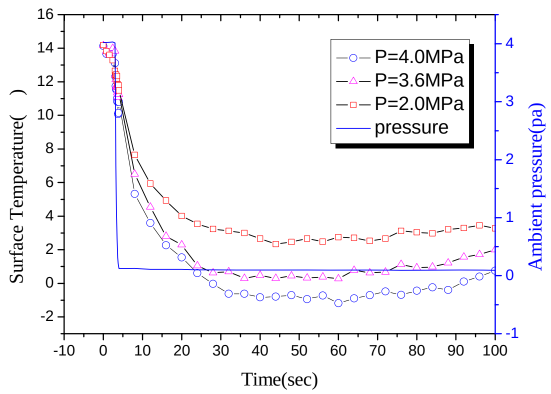


Figure 2: The influence of environmental pressure on surface temperature transitions of R-113 liquid.

Acknowledgements

The research was supported by NSFC- RFBR International Cooperation Project (No. 50711120546).

References

Owen, I., Jalil, J.M., 1991, "Heterogeneous flashing in water drops", International Journal of Multiphase Flow,17(5), pp.653-660

J.P. Hindmarsh, A.B. Russell, X.D. Chen, Experimental and numerical analysis of the temperature transition of a suspended freezing water droplet, Int. J. Heat Mass Transfer. Vol. 46,pp.1199-1213 (2002).

Vibration-Induced Attraction of a Particle towards a Wall in Microgravity – The Mechanism of Attraction Force

Mehrrad Saadatmand[1], Masahiro Kawaji[1,2], and Howard H. Hu[3]

1. Dept. of Chemical Engineering and Applied Chemistry, University of Toronto, Toronto, ON M5S 3E5, Canada
E-mail: mehrrad.saadatmand@utoronto.ca, kawaji@ecf.utoronto.ca
2. Dept. of Mechanical Engineering, City College of New York, New York, 10031 USA
3. Dept. of Mechanical Engineering and Applied Mechanics, University of Pennsylvania, Philadelphia, PA, USA
E-mail: hhu@seas.upenn.edu

The effects of small vibrations on a particle oscillating near a solid wall in a fluid cell, relevant to material processing such as crystal growth in space, have been investigated by direct numerical simulations. Simulations have been conducted for a solid particle suspended in a fluid cell filled with a fluid of 1 cSt viscosity, vibrating sinusoidally in a horizontal direction. The mechanism for this attraction force as well as some examples showing the effects of this force on the particle are presented in this paper. The simulations revealed the existence of a vibration-induced force attracting the particle towards the nearest cell wall which varied with the cell vibration frequency. The predicted flow patterns around the particle unveiled an accelerated flow in the gap between the particle and the nearest wall as well as a pressure decrease in accordance with Bernoulli's principle, which would result in the attraction force.

Visual Experiment Researches for Bubbling Process in water

Lixi Zhang1, Fangchao Han2 and Shiyuan Gao3

School of Engine and Energy, Northwestern Polytechnical University
127 Youyi Street, Xi'an, 710072, China
1Email:zhanglixi@nwpu.edu.cn, 2Email: 170438183@qq.com, 3Email: gaoshiyuaner@163.com

The micro-process of air bubbling in humidifier are observed and measured by high-speed photography experiments. The experiment system is shown in Fig.1. At low air speed, the bubbles are independent each other in vertical moving upward state, which are shown in Fig.2. At higher speed, the collision, broken and swing of bubbles are happened during the process of their upward movement, it can be seen in Fig.3.

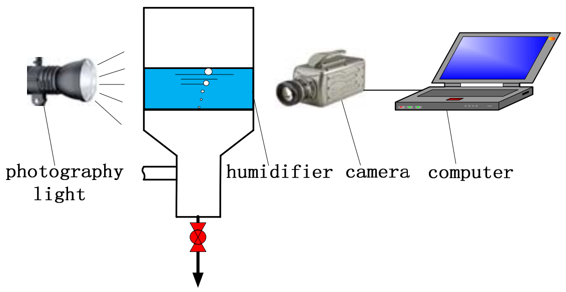


Figure 1: Scheme of photography system for bubbling

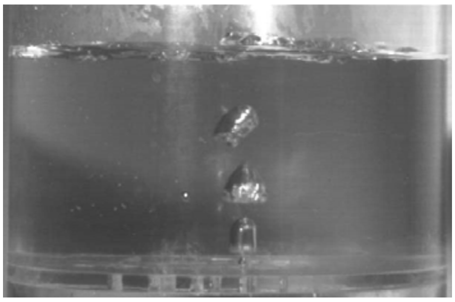


Figure 2: Bubbles moving state at low speed

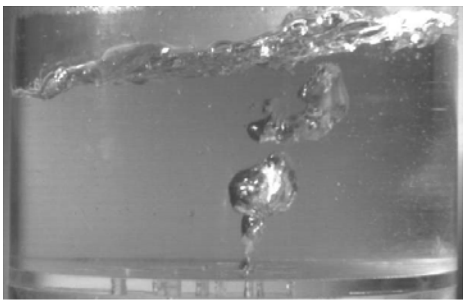


Figure 3: Broken and swing of bubbles at higher speed

The stage of bubble's formation, growth, separation and movement on a sieve hole in humidifier are shown in Fig.4, it is indicated that the separating stage of bubble from the hole is obviously, so that the formation process of air bubble in water can be classified to the second-stage model of bubble formation. With the water level increased in humidifier, the air speed through the sieve hole would be decreased. Basing on the experimental data, the fitted relationship between the average bubble volume with the air speed and the sieve diameter is obtained, and it's indicated that the sieve diameter is the main factor affecting the bubble volume. Under the different conditions, the relative ratio of bubble's maximum cross-section to the humidifier's cross-section *R* would be increased with the bubble rising in water. One of the results of *R* is shown in Fig.5, which is under the diameter of sieve hole is 2mm and the height of water lever is 50mm.

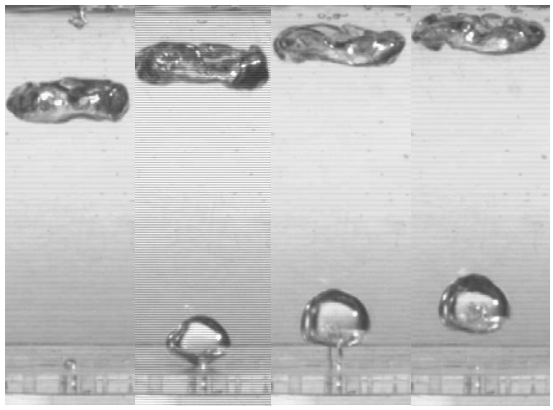


Figure 4: Bubble's formation, growth and separation

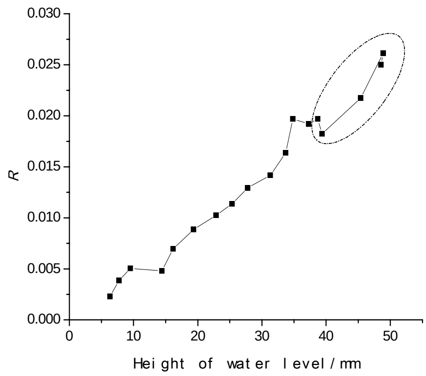


Figure 5: *R* changed with height water lever

References

L.Zhang, M.Shoji, Aperiodic bubble formation from a submerged orifice, Chem. Eng. Sci., Vol. 56 pp. 5371-5381(2001)

Marrucci G., A theory of coalescence, Chem Eng ScI, Vol. 24 pp. 975-985(1969)

Analysis the Change of Parameters in Bubbling Humidifier

Lixi Zhang1, Wenbin Chen2 and Hefei Zhang3

School of Engine and Energy, Northwestern Polytechnical University
127 Youyi Street, Xi'an, 710072, China
1Email:zhanglixi@nwpu.edu.cn, 2Email: wenbinchen88@163.com, 3Email: zhfyu705@163.com

It is taken the single-stage bubbling humidifier in the solar desalination process as a research object which is shown in Fig.1. In the different conditions of supplied heat and initial humidifying temperature in humidifier, the remaining water quantity, the instantaneous humidifying capacity and the water temperature in it are calculated and analyzed by differential equations of energy balance and mass balance in unsteady process. It's known that the water temperature and the instantaneous humidifying capacity in humidifier are changed significantly with the time increased in the initial humidifying stage.

When the initial temperature of humidifying is 323.15K and he heat supplied to water Q_s is 1000*W*,, the relationships of T_2 and M_s change with time are shown in Fig.2 and Fig.3

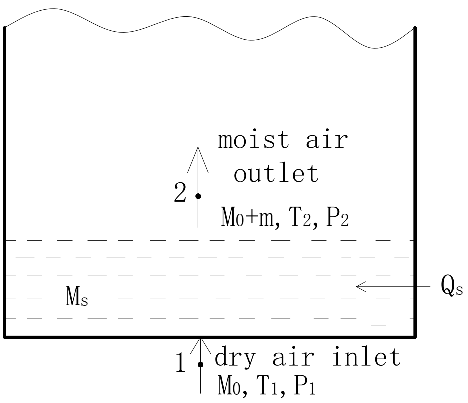


Fig.1 Schematic diagram of energy balance in humidifier

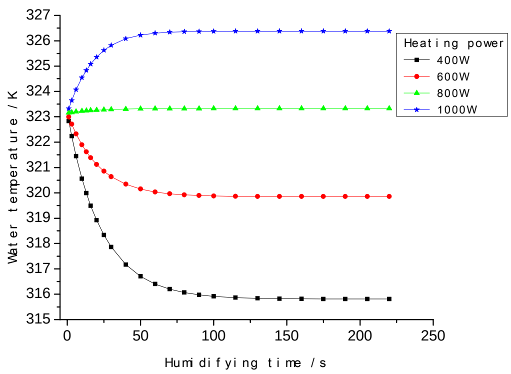


Fig.2 Temperature of water in humidifier changed with time

When the height of water level is 1cm and the heat supplied to water is 1000W, the times of all water vaporized in humidifier under different initial humidifying temperature are shown in table 1. It's indicated that the time needed for all water vaporized is decreased with the initial humidifying temperature increased.

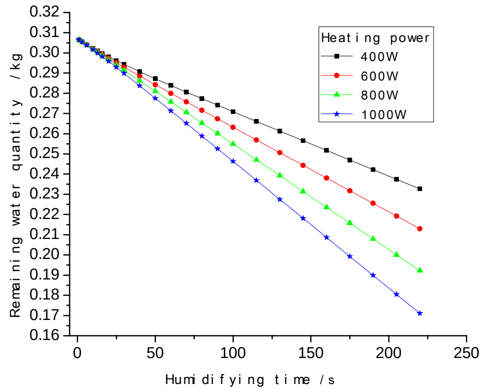


Fig.3 Water quantity changed with time

Table 1 Time needed for all water vaporized /s

temp. /K	303.15	313.15	323.15	333.15	343.15
time /s	563	526	490	458	429

Above all, the increasing or decreasing of these parameters depends on that the heat supplied to water is higher or lower than the heat absorbed by water vaporizing. If the supplied heat is higher, the water temperature and the instantaneous humidifying capacity would be increased; otherwise they would be decreased or unchanged. After a certain time of humidifying, the water temperature and the instantaneous humidifying capacity tend to be stable; under different supplied heat, the remaining water quantity is decreased approximately by liner with humidifying time increased, and the larger the supplied heat is, the faster the remaining water quantity decreased, that means the humidifying capacity is more.

This research is important to correctly predict the change of water temperature, remaining water quantity and instantaneous humidifying capacity in humidifier, and it is benefit to guide the operation of humidifying process.

References

Lixi Zhang, Guangping Cheng, A new process of solar desalination based on humidification-dehumidification cycle, International Multi-Conference of Engineers and Computer Scientists 2009, Vol.II pp. 1908-1910(2009)

S.A.El-Agouz, M.Abugderah, Experimental analysis of humidification process by air passing through seawater, Energy Conversion and Management, Vol. 49 pp:3698-3703(2008)

Ke Li, Chemical Engineering Thermodynamics, Beijing，Petroleum Industry Press,（1985）

Flow boiling instability in micro-channel heat sink.

Vladimir V. Kuznetsov, Alisher S. Shamirzaev, Igor A. Kozulin

Institute of Thermophysics SB RAS
Ave. Academician Lavrentyev 1, Novosibirsk, 630090, Russia
E-mail vladkuz@itp.nsc.ru

Two-phase mini- and micro-channel heat sinks are successfully applied for high heat-flux dissipation by various technologies. Successful application of the two-phase micro-channel heat sink requires fundamental understanding of hydrodynamic and thermal aspects of the phase change in small channels. Nowadays the mini/micro channel studies of flow boiling heat transfer especially at low mass fluxes, demonstrate the contradictory data. As noted in (Guodong et al 2008), the flow instabilities has essential influence to the flow pattern in two-phase heat sink and can reduce the flow boiling heat transfer.

In the current study the local heat transfer coefficients at upward flow boiling of the water were measured along stainless steel heat sink. The heat sink includes 10 parallel channels of the 640x2050 μm cross-section and 120-mm length. The local heat flux and wall temperatures along heat sink were measured. Heat transfer coefficients were determined from these data using the fin efficiency model. Temporal instabilities of the wall temperature were recorded during experiments. The steam generator in front of the measurement section allows us to measure the local heat transfer coefficients in a wide range of mass qualities. Two phase flow with prescribed vapor quality was fed into the heat sink through capillary with 1 mm ID. An experimental investigation was performed for the mass flow rate in range of 8 to 90 kg/m^{2}s and heat fluxes of 20 to 170 kW/m^2. The adiabatic flow pattern map for gas-water upward flow at the similar micro channel was obtained also.

For different initial vapor quality x_0 the maximum of the heat transfer coefficient was observed at the inlet of heat sink and it was decreased along its length, see Figure 1. Here the data for mass flow rate 34 kg/m^{2}s and 84.7 kg/m^{2}s are presented as circles and triangles accordingly. High values of the heat transfer coefficients at the inlet of heat sink caused by good initial liquid distribution in this area. The calculations of the heat transfer coefficient according to (Kandlikar, 2010) and (Kuznetsov, 2010) models are presented in Figure 1 also. For small vapor quality the calculations are close to experimental data for mass flow rate 84.7 kg/m^{2}s and differ from the data for mass flow rate 34 kg/m^{2}s. Kandlikar's model predicts the heat transfer coefficient deterioration along the heat sink but both models don't describe the influence of initial quality to the heat transfer coefficients. We suppose that heat transfer deterioration along the heat sink occurs due to liquid redistributions in the channels by capillary forces.

The measurements of wall temperature variation in time show that for mass flow rate 34 kg/m^{2}s the standard deviation of wall temperatures has a minimum at the inlet, it increases along the heat sink and decreases at the exit. Maximum value of standard deviation of the wall temperature reaches 1 K. With grow of mass flux the wall

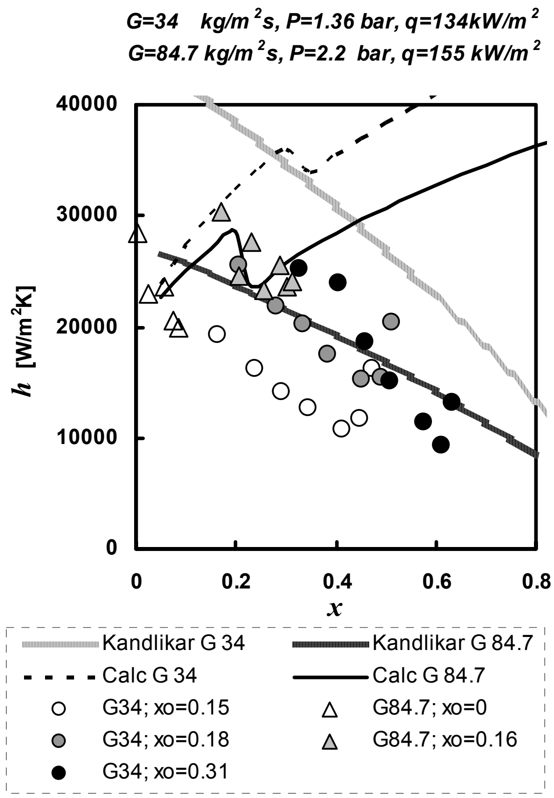


Figure 1: Local heat transfer coefficient variation along the micro-channel heat sink for different inlet vapor quality.

temperature becomes stable. For mass flow rate 84.7 kg/m^{2}s the wall temperature is stable everywhere and its standard deviation is close to the measurement accuracy. However, the deterioration of heat transfer along heat sink occurs both for stable and unstable wall temperature modes.

The authors are grateful to the Russian Foundation for Basic Research for the financial support (grants No. 07-08-12212-ofi and 11-08-01140-a) and "Rosnauka" (state contract N 02.740.11.0054).

References

Guodong Wang, Ping Cheng, A.E. Bergles, Effects of inlet/outlet configurations on flow boiling instability in parallel microchannels, International Journal of Heat and Mass Transfer 51 (2008) 2267–2281

S.G. Kandlikar, Similarities and Differences Between Flow Boiling in Microchannels and Pool Boiling // Heat Transfer Engineering, 31(3):159–167, 2010.

Kuznetsov V.V. Heat and Mass Transfer with Phase Change and Chemical Reactions in Microscale // Proceedings of the International Heat Transfer Conference IHTC14, August 8-13, 2010, Washington, DC, USA, Keynote IHTC14-22570.

The investigation of hydraulic jump disruption in liquid film flow on a surface of a rotating disk

Ivan Kabardin, Vladimir Meledin, Sergei Dvoinishnikov

Institute of Thermophysics SB RAS, Lavrentyev Ave, 1, Novosibirsk, 630090, Russia
kabardin@itp.nsc.ru

Many investigations have been performed in the past on flow and heat transfer characteristics in thin liquid films due to the fact that high heat transfer rates can be achieved in thin films. Even though the heat transfer to thin films falling down on a vertical plate is quite high, much higher heat transfer rates can be realized when the film is generated by the impingement of a liquid jet onto a surface. This is due to the fact that the film velocities are typically greater than those of a falling film. In order to understand the major factors affecting the heat transfer performance, hydrodynamic characteristics of a thin film with a free surface flowing over stationary and rotating disk surfaces need to be first understood. One of interesting phenomena of liquid film flow on a surface of a rotating disk is a hydraulic jump. It occurs when a liquid jet falls vertically on a horizontal surface. In this phenomena film thickness remain thin till the some value of radius. After that film thickness rise abruptly. Investigators[1][2][3] have described experimental and theoretical investigations of a hydraulic jump as hydraulic jump forms, an estimation of hydraulic jump radius, the dependences of hydraulic jump radius from coefficients of viscosity, surface tension etc. But there is not an investigation the disruption of hydraulic jump under the influence of centrifugal force.

In this work it was investigated the structure of hydraulic jump on a surface of a rotating disk and it was investigated the disruption of a hydraulic jump under the influence of a centrifugal force at different values of a flow rate of liquid

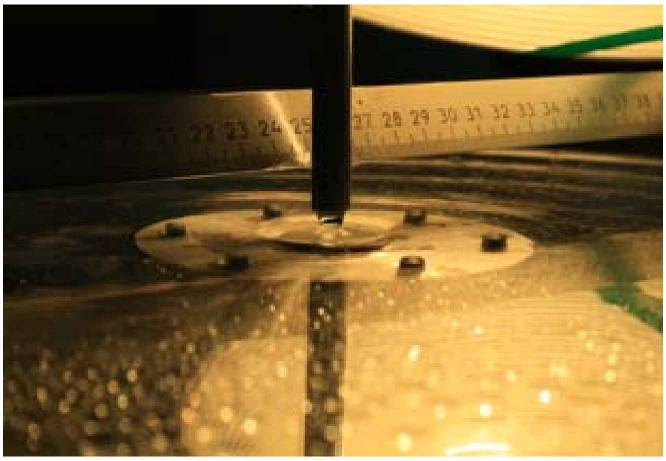


Fig.1 The photo of hydraulic jump. Volumetric flow rate is 7 ml/s

In the work liquid emerges from a constant flow rate tank on a surface of a rotating disk. Liquid emerges from a nozzle in a form of a free liquid jet. Liquid spread over the surface of a rotating disk and formed a liquid film. At low values of rotational speed and low values of flow rate a liquid film thickness remained very small and same value of radius liquid film thickness rose abruptly. The photo of hydraulic jump is presented on the fig.1. The rotating disk surface was illuminated with light of a projector. Light of a projector reflected from a surface of a film and formed an image on a screen. The image of liquid film was registered with a digital camera and then it was processed by a personal computer. Obtained pictures were in a perspective projection. On a personal computer images were transformed from a perspective projection to a horizontal projection. From processed images the mean diameter of a hydraulic jump were measured. The diameter of the disk was 400 mm. Rotational speed was varied from 0,05 revolution per second till 1 revolution per second. As working liquid it was used distilled water. The liquid flow rate was varied from 0 till 50 milliliters per second.

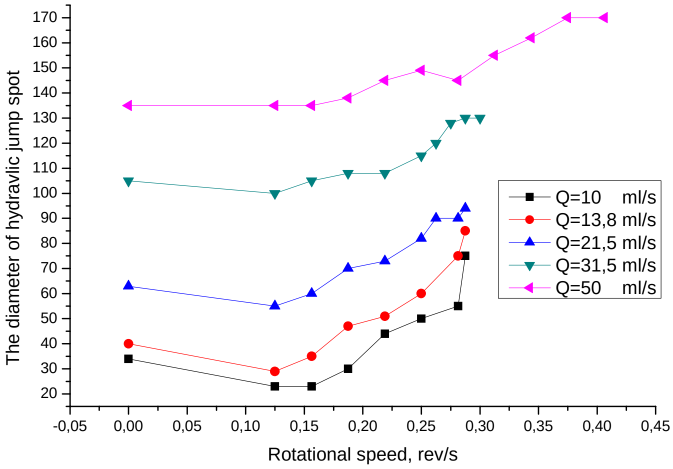


Fig. 2 The dependence of the diameter of the radius of hydraulic jump from rotational speed at different flow rates.

There were investigated structure of hydraulic jump on a surface of a rotating disk and the disruption of a hydraulic jump under the influence of a centrifugal force at different values of a flow rate. The dependence of the diameter of the radius of hydraulic jump from rotational speed at different flow rates is presented in fig.2.

References

1. John W. M. Bush, Jeffrey M. Aristo, The influence of surface tension on the circular hydraulic jump// J. Fluid Mech. - 2003. - Vol. 489. - P. 229–238.
2. John W. M. Bush, Jeffrey M. Aristoff, A. E. Hosoi An experimental investigation of the stability of the circular hydraulic jump// J. Fluid Mech. - 2006. - Vol. 558. - P. 33–52.
3. Thomas R. N. Jansson, Martin P. Haspang, Kåre H. Jensen, Pascal Hersen, Tomas Bohr Polygons on a Rotating Fluid Surface// Physical review letters. - 2006. - PRL 96, 174502 - P.1-4

Thermocapillary Deformation of Gravity-Driven Liquid Film under Thermal Wave Propagation

Oleg Sharypov[1,2] and Pavel Kuibin[1,2]

[1]Kutateladze Institute of Thermophysics, Siberian Branch of Russian Academy of Sciences
Lavrentyev av., 1, Novosibirsk, 630090, Russia
[2]Novosibirsk State University
Pirogova str., 2, Novosibirsk, 630090, Russia
sharypov@itp.nsc.ru

Experimental study of gravity-driven thin liquid film deformation under local heating shows the existence of steady-state regimes with two- or three-dimensional flow structure (Kabov et al. 1996). The previous theoretical study (Sharypov & Kuibin 2008) confirms the possibility of steady-state 2-D flow regime in thin horizontal layer with moving local heat source. It was shown that even under the zero-gravity condition the motion of local heater provides the existence of steady-state regime without film breakdown and dry spot formation (Sharypov & Kuibin 2009*a*). The impact of different physical mechanisms in the layer deformation was analyzed in the work (Sharypov & Kuibin 2009*b*). It was shown that inertial force can play the main stabilizing role for long-wave deformations of the free surface. The calculated stream function revealed the presence of a vortex in the layer when the temperature gradient wasn't too small. Those results were obtained solving the problem with surface temperature known from experiment. This allowed one to eliminate thermal part of the general problem. Solution of conjugated thermal and hydrodynamic problem meets the difficulties with boundary condition statement: the distributions of the temperature and the heat flux at the substrate are unknown within the heat release zone. These distributions are not measured in experiments. An approach to avoid these difficulties was proposed in the work (Sharypov & Kuibin 2010), where both stabilizing mechanisms were taken into account: gravity-driven film flow and motion of heat release zone (in opposite direction). In this case the resulting velocity profile in accompanying frame is "combined" (superposition of the parabolic profile (due to gravitation), the uniform one (due to motion of heat release zone) and the linear one (due to thermocapillary effect)). At the present work we use that approach to obtain further detailed information on the flow structure of non-isothermal film with the "combined" velocity profile.

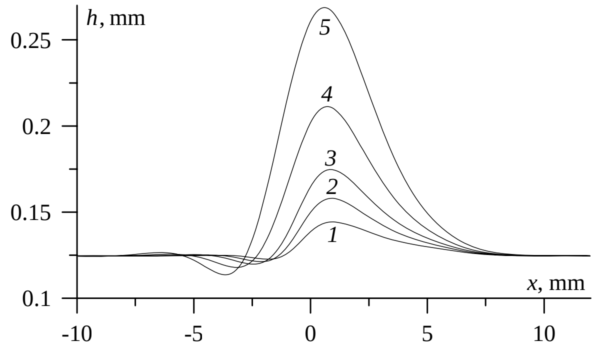


Figure 1: Film deformation at different inclination angles under constant flow rate condition.

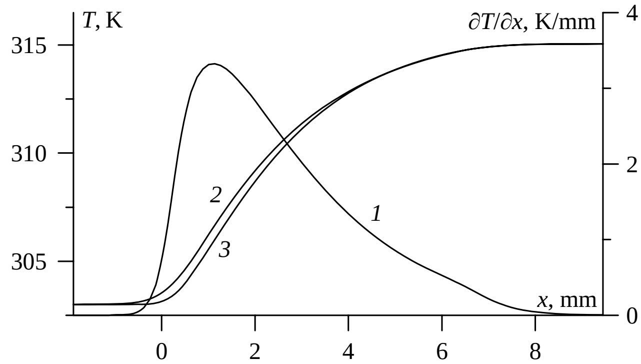


Figure 2: Temperature gradient (1) and temperature at the free surface at vertical (2) and horizontal (3) substrate under constant flow rate and heat release.

It is shown numerically that changing of the velocity profile (heat source velocity increase with slope decrease under other equal conditions: fixed flow rate, film thickness and heat release) leads to dramatic amplification of thermocapillary deformation of the film, see Fig. 1 (the lines correspond to different slope angles: 1 – 90°, 2 – 45°, 3 – 15°, 4 – 4°, 5 – 0°). The temperature distributions differ slightly for vertical and horizontal substrates if the flow rate and heat release are the same, see Fig. 2.

References

Kabov O.A., Marchuk I.V. and Chupin V.M., Thermal imaging study of the liquid film flowing on vertical surface with local heat source, Russ. Journ. Eng. Thermophys., Vol. 6 pp. 105-138 (1996)

Sharypov O.V. and Kuibin P.A., Microgravity: effect of a moving local heater on liquid film structure, Microgravity sci. technol., Vol. 20 pp. 237-241 (2008)

Sharypov O.V. and Kuibin P.A., Heat-wave induced vortex in a thin liquid layer, Int. Rev. Chem. Eng. Vol. 1 pp. 158-163 (2009*a*)

Sharypov O.V. and Kuibin P.A., 2D flow structure in a thin liquid layer under thermal wave propagation, Microgravity sci. technol., Vol. 21 pp. S321-S324 (2009*b*)

Sharypov O.V. and Kuibin P.A., Effect of motion of a local heat source on thermocapillary deformation of a thin liquid film flowing down under the action of gravity, Tech. Phys. Lett., Vol. 36 pp. 683-686 (2010)

Experimental Study of the International Space Station Contamination by its Orientation Thrusters Jets

Vyacheslav Yarygin[1], Yury Gerasimov[2], Andrey Krylov[2], Victor Prikhodko[1], Igor Yarygin[1]

[1] Kutateladze Institute of Thermophysics SB RAS,
Lavrentieva Ave 1, Novosibirsk, 630090, Russia
yarygin@itp.nsc.ru
[2] Korolev Rocket and Space Corporation ENERGIA,
Lenin Str. 4a, Korolev, Moscow area, 141070, Russia

The joint ejection of gas and near-wall liquid film from a supersonic nozzle into vacuum (Fig. 1) is studied experimentally. This study was motivated by solving the problem of external contamination of space vehicles surfaces, including the International Space Station (ISS), by jets of its orientation thrusters (OT), in which the fuel film is used for nozzle walls cooling (Gerasimov and Yarygin, 2007). Operation of these thrusters is accompanied by periodic ejection of burnt and unburnt fuel fractions, including the droplet ones (contaminants). Ejection of fractions occurs into almost full sphere – from 0 to 180° relative to the jet axis. This is firstly caused by peculiarities of gas and liquid flows into vacuum. Construction elements of the orbital station which get into the flow field of exhaust plume are undergone to mechanical and physical-chemical effect and this is undoubtedly a negative factor. Besides there is a risk of contaminant penetration into the station on astronaut suits after their spacewalks. Now great attention is paid to the problem of the ISS contamination.

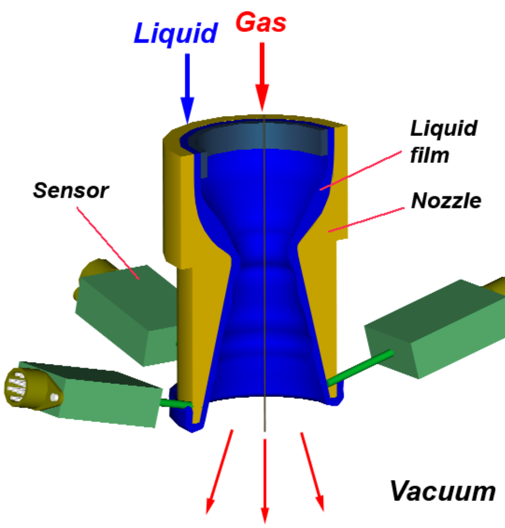


Figure 1: Supersonic nozzle with near-wall liquid film.

The main attention in the work was given to statement of researches on modeling in the vacuum chamber of ISS contamination processes caused by OT, and also to the analysis of contamination minimization methods.

It was shown that the typical angle of jet divergence, defined via relative jet impulse, could be used as the main modeling parameter of OT exhaust plume. This angle should be equal in experimental and real conditions. The choice of liquids, modeling real components of rocket fuel, is approved.

The description of experimental setup and near-wall liquid film thickness and velocity measurement technique in supersonic nozzle is given. The structure of gas-droplet flow arising behind exit cross-section of supersonic nozzle in vacuum is obtained. Appearance of two typical areas of a droplet phase flow behind nozzle exit – the central *1* and the peripheral *2* (Fig. 2) is established experimentally. Reasons of their appearance are revealed and parameters of droplets in these areas are estimated.

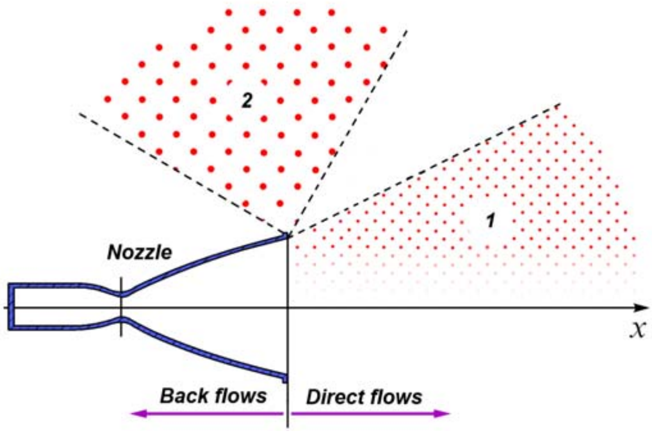


Figure 2: Flow structure behind supersonic nozzle.

Arising of backflows which cause contamination of space station external surface is shown.

The analysis of backflows control methods for contamination reducing is given. It is shown that use of special gas-dynamical protective devices – screens, mounted on exit part of a nozzle, allows reducing essentially value of back (contaminating) flows.

The satisfactory correspondence of obtained experimental results with on-orbit observations results is shown.

References

Yu.I. Gerasimov, V.N. Yarygin. Problems of Gas-Dynamical and Contaminating Effect of Exhaust Plumes of Orientation Thrusters on Space Vehicles and Space Stations // Proceedings 25th International Symposium on Rarefied Gas Dynamics (RGD25), pp. 805-811 (2007).

Formation of Ultra-Disperse Particles from Micro Droplet Flows and their Study

Igor Yarygin, Victor Prikhodko, Vyacheslav Yarygin, Mark Andreev

Kutateladze Institute of Thermophysics SB RAS,
Lavrentieva Ave 1, Novosibirsk, 630090, Russia
yarygin@itp.nsc.ru

At the present time gas-droplet flows are widely used in various heat- and mass-transfer apparatuses and devices. A detailed overview of experimental and theoretical investigations of gas-droplet flows shows that they are carried out in rather wide ranges of regime parameters on temperature, pressure and composition. At the same time it is possible to state that the field of scientific and practical applications of gas-droplet flows continuously extends. One of the modern and actively developing directions is obtaining and study of micro droplet flows (Moseler and Landman, 2000), and formation of ultra disperse particles, including nanoparticles, of various purposes from these flows (Tyree and Allen, 2008).

The main idea of the method consists in obtaining droplets with the required size (ideally with monodroplet distribution function of particles on sizes) of substance solution and subsequent evaporation of solvent in a bearing gas flow. Thus it is enough to control and operate the dimension of initially formed droplets (with known initial solution concentration) to obtain required dispersion of substance. It is much easier to carry out such control as the droplet dimensions exceed dry particles dimensions by 2-3 orders. At the same time independent measurements of particles parameters, formed after droplets evaporation, are important too.

Different methods of generation and parameters control of micro droplet flows, including ejection into vacuum, are discussed in the report. The description of the experimental setup for micro droplets generation and their diagnostics is given.

With the use of the method of light scattering by particles (Mie scattering) the average dimensions of multicomponent liquid droplets are measured and their evolution in the gas flow is studied.

Distribution functions of particle on sizes are obtained with use of aerosol particles spectrometer. Fig.1 illustrates for example the distribution function in conditions when NaCl was used as nano-forming substance with a weight concentration in solution of 1/2000. Possibility of the formed ultra disperse particles parameters control by changing the initial size of droplets and solution concentration is proved experimentally (Fig. 2).

A comparison of the measured average size of nanoparticles with those calculated from measurements of average droplet size is carried out.

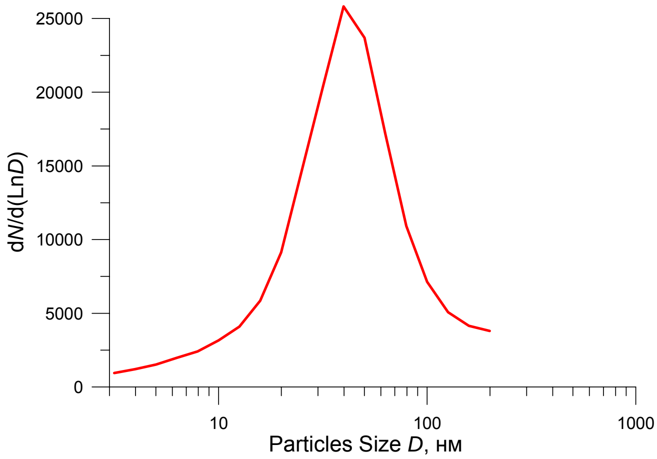


Figure 1: Distribution function of particles on sizes

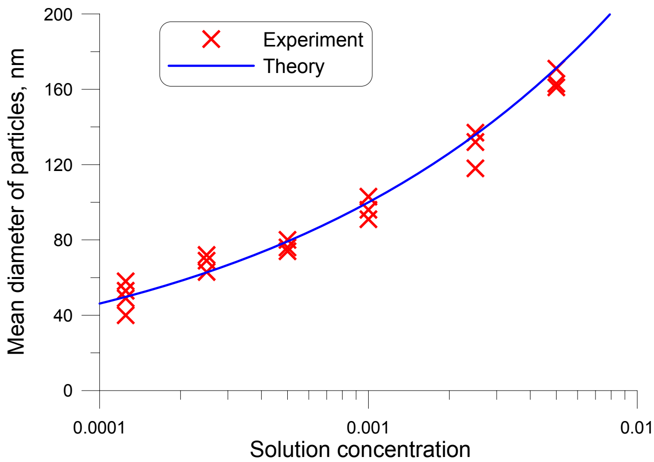


Figure 2: Dependence of particle's sizes on solution concentration

This work was supported by the Council of the President of the Russian Federation for Support of Young Russian Scientists and Leading Scientific Schools (grant MK-182.2010.8).

References

Michael Moseler and Uzi Landman. Formation, Stability, and breakup of nanojets. Science, Vol.289, No.5482, pp.1165-1169, (2000)
Corey A. Tyree and Janathan O. Allen. Foam droplet separation for nanoparticle synthesis. Journal of Nanoparticles Research, Vol.10, No.3, pp 465-473 (2008)

Capillary Channel Flow

The CCF experiment on the International Space Station

Michael Dreyer, Aleksander Grah, Joerg Klatte and Peter Canfield

University of Bremen, Production Engineering, Center for Applied Space Technology
Am Fallturm, Bremen, 28359, Germany
Michael.dreyer@zarm.uni-bremen.de

Mark Weislogel and Yongkang Chen

Portland State University, Maseeh College of Engineering and Computer Science
PO Box 751, Portland OR 97207, USA

The CCF experiment performed first operations between January and March 2011 onboard the International Space Station (ISS). The purpose of the Capillary Channel Flow experiment (CCF) is to investigate the behavior of capillary flows through open capillary channels in an environment of compensated gravity. The international science team controlled the experiment from a ground station located at ZARM in Bremen, Germany. One of the most pressing challenges is to determine the maximum forced flow rates that are possible before the free surface of the liquid becomes unstable and collapses - a phenomenon that in fluid mechanics is referred to as 'choking'.

The CCF experiment is a joint German (DLR – ZARM) and American (NASA – PSU) endeavor. It is supervised and supported by NASA's Glenn Research Center. The experimental unit was constructed in Germany by Astrium Friedrichshafen and in April of 2010 was transported to the ISS onboard flight STS-131 of the space shuttle Discovery. Supervised by Astrium engineers at Marshall Space Flight Center (MSFC) in Huntsville, Alabama, NASA astronauts installed the unit into the Microgravity Science Glovebox onboard the ISS. Having passed initial tests, CCF operations have commenced successfully.

For years, scientists have been developing and improving methods to use capillary channels liquid transport whilst preventing contamination of the liquid with gas bubbles. This particular experimental setup has already been tested in the drop tower at ZARM and in suborbital rocket flights, but these experiments were extremely limited by the duration of the state of reduced gravity, typically not longer than a few seconds or minutes.

Onboard the ISS, the team is able to utilize the compensated gravity environment to perform a great deal more and longer experiments. They are able to vary parameters such as channel length, flow rates, and even accelerations and oscillations of flow. The experiment is equipped with numerous sensors and a high-speed camera which produces an abundance of data that was down-linked to the ground station at ZARM. The newly acquired data will be used to validate current mathematical models of capillary flows helping optimize the development of liquid management systems in space.

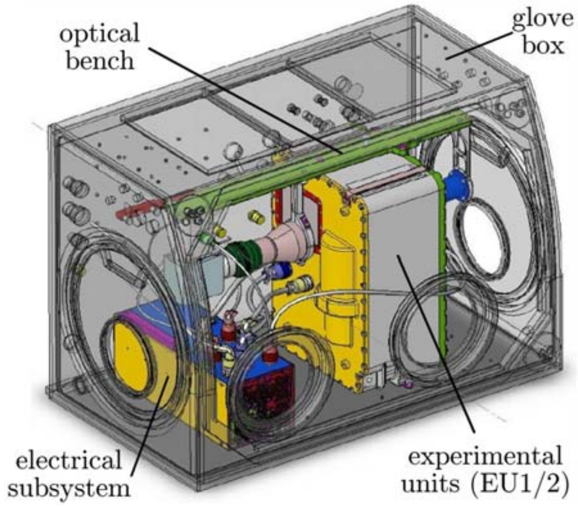


Figure 1: The CCF experiment in the Microgravity Glovebox onboard the International Space Station.

The talk aims at a general introduction to the subject of a flow channel in space. The fundamental equations and boundary conditions, and the tools to solve them, will be explained. The scientific hardware and its functionality, the scientific and technical stimuli as well as the measurements techniques will be introduced. A description of the operational scenario will be concluded with first results of steady, unsteady and oscillatory flows.

Grah, A., Dreyer, M. E.: Dynamic stability analysis for capillary channel flow: One-dimensional and three-dimensional computations and the equivalent steady state technique, Phys. Fluids, 22, 014101 (2010).

Grah, A., Haake, D., Rosendahl, U., Klatte, J., Dreyer, M. E.: Stability limits of unsteady open capillary channel flow, J. Fluid Mech., 600, 271-289 (2008).

Klatte, J., Haake, D., Weislogel, M. M., Dreyer, M. E.: A Fast Numerical Procedure for Steady Capillary Flow in Open Channels, Acta Mech., 201, 269-276 (2008).

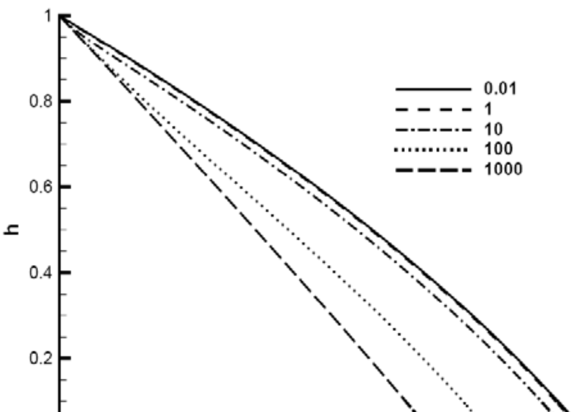
Analysis of 'Surface Thermal Capacity' Effects on Maranogni Instability of Evaporating Thin Liquid Layers

Rong Liu, Qiu-Sheng Liu*

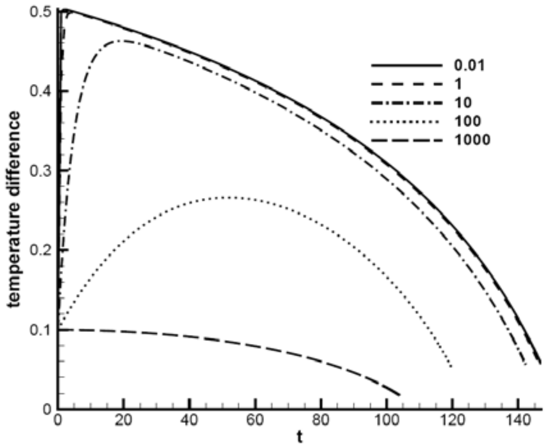
Key Laboratory of Microgravity, Institute of Mechanics, Chinese Academy of Sciences,
Bei-Si-Huan-Xi Road, #15, Beijing 100190, China
*E-mail:liu@imech.ac.cn

In most of previous works, the energy boundary condition only takes into account the contributions of the evaporation latent heat and the heat conduction in the liquid. However, a newly measured surface property-'surface thermal capacity' $c\sigma$ has been found in recent studies (Duan et al. 2005, Das, et al. 2007). Duan and Ward's experiments in 2005 predicted a measured surface property - the 'surface thermal capacity'. If this property is introduced into the energy boundary condition at liquid-vapor interface, the energy conservation principle can be satisfied during steady state. Classical theories considered only thermo-capillary effects on the shear stress balance equation. However, the contribution of surface tension to the surface energy balance has been completely ignored. In this paper, we performed a linear analysis of the Marangoni- Benard instability in a horizontal static liquid thin layer during evaporation. In the boundary condition of vapor-liquid interface, both the 'surface thermal capacity' property and the surface tension are considered.

The influence of the 'surface thermal capacity' property on the Marangoni-Benard instabilities in evaporating liquid thin layers have been investigated through linear instability analysis. A one-sided model with a deformable evaporating liquid-vapor interface is proposed. The moving boundary problem of the basic state is studied using numerical method. The results show that the 'surface-thermal capacity' will significantly influence the evaporation time and the temperature difference in the liquid layers.



(a)



(b)

Figure 1: The depth h (a) and the temperature difference (b) with time for various dimensionless surface-thermal capacity G .

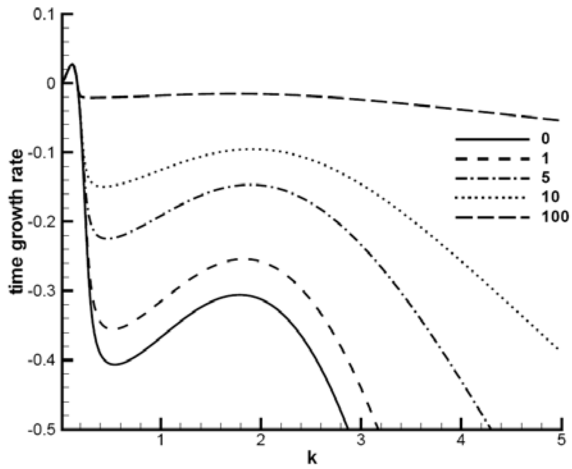


Figure 2: time growth rate versus wave number for various G.

References

F. Duan, C. A. Ward, Surface excess properties from energy transport measurements during water evaporation, Phys. Rev. E, Vol.72, pp. 056302-056312 (2005)

K. S. Das, C. A. Ward, Surface thermal capacity and its effects on the boundary conditions at fluid-fluid interfaces, Phys. Rev. E, Vol.75, pp.065303-065306 (2007)

Nanofluids with self-rewetting behaviour for space and terrestrial heat pipes

Daniele Mirabile Gattia[1], Raffaele Savino[2], Roberto Di Paola[2], Renzo Marazzi[1], Marco Vittori Antisari[1]

1 ENEA, UTTMAT, Via Anguillarese 301, Rome 00123, Italy
daniele.mirabile@enea.it
2 University of Naples Federico II, DIAS Dept., P.le V. Tecchio 80, Naples, 80125, Italy

Modern electronic systems require improvements in thermal management. Studies on binary or multi-component heat transfer fluids with peculiar properties have risen in importance due to their application in the improvement of heat transfer performances.

In particular an interesting class of fluids, called "self-rewetting fluids", demonstrated interesting surface tension properties. The classification of these fluids is due to a surface tension which increases with temperature and concentration. Self-rewetting fluids include dilute aqueous solutions of high carbon number alcohols (e-g- Butanol, Pentanol, Exanol, Heptanol. etc.) that show surface tension minima versus temperature, and the temperature dependence turns out to be positive in the higher temperature region beyond the minima. Since in the course of liquid/vapour phase change, self-rewetting fluids behaviour induces a rather strong liquid inflow (caused by both temperature and concentration gradients) from the cold region (where liquid condensates) to the hot evaporator region, these fluids have been proposed and investigated as new heat transfer fluids for advanced heat transfer devices, e.g. heat pipes or heat spreaders for terrestrial and space applications.

Thermophysical properties like surface tension, wettability and thermal conductivity, at different temperatures, have been measured not only for binary mixtures, but also for a number of ternary aqueous solutions with relatively low freezing point and for nanoparticles suspensions (so called nanofluids). The use of an appropriate suspended material could enhance the properties of the base self-rewetting fluid.

The present work is dedicated to the study of the thermophysical properties (figure 1) of nanofluids based on water/alcohol solutions with suspended carbon nanostructures. In particular single-wall carbon nanohorns (SWNH), synthesized by a homemade apparatus with an AC arc discharge in open air (Mirabile Gattia et al. 2007), have been used as suspended material. The use of arc discharge has demonstrated that it is possible to produce this carbon nanostructure in a simple and cheap way. SWNH have been produced by an alternating current (AC) fed arc discharge with a symmetric configuration of the electrodes. By this way it has been maximized the quantity of evaporated material which can be collected as fine soot.

An interesting result is that nanofluids prepared from initial self-rewetting fluids exhibit the same anomalous positive surface tension gradient with temperature as binary self-rewetting solutions.

Thermal tests on bare and groove heat pipe show that heat pipes filled with self-rewetting nanofluids show better thermal performances in comparison to the heat pipe filled with ordinary self rewetting fluids also in terms of maximum temperature reached at the evaporator. These results suggest the utilisation of self-rewetting nanofluids as innovative working fluids for heat transfer applications.

The paper discusses the results of these investigations and laboratory characterization tests.

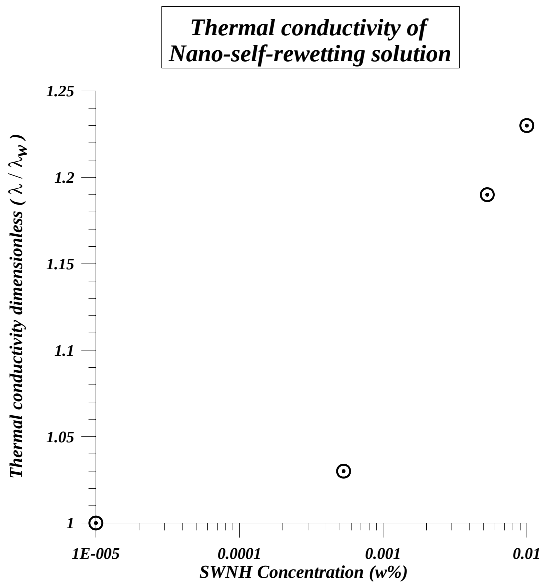


Figure 1: Thermal conductivity measurements of "Nano-self-rewetting" solution based on dilute aqueous solution of butanol, dimensionless with thermal conductivity of pure water

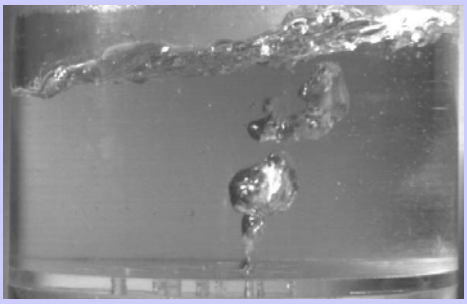
The potential interest of the proposed studies stems from the large number of possible industrial applications, including space technologies and terrestrial applications, such as cooling of electronic components. Some interesting applications may be taken into consideration which take advantage of the peculiar properties of the nanofluids here presented, for example the development of advanced wickless heat pipes for utilization in reduced gravity environments (Savino et al. 2008, Savino et al. 2009).

References

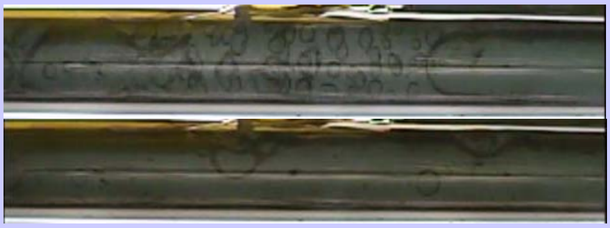
Mirabile Gattia D., Vittori Antisari M., Marazzi R., Nanotechnology, Vol. 18, pp. 255604-255610 (2007)

Savino R., Abe Y., Fortezza R., Acta Astronautica, Vol. 63 pp. 24-34 (2008)

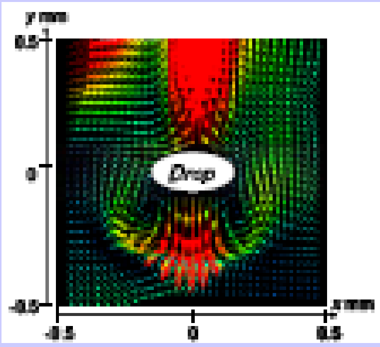
Savino R., Cecere A., Di Paola R., Int. Journal of Heat and Fluid Flow, Vol. 30 pp. 380-388 (2009).



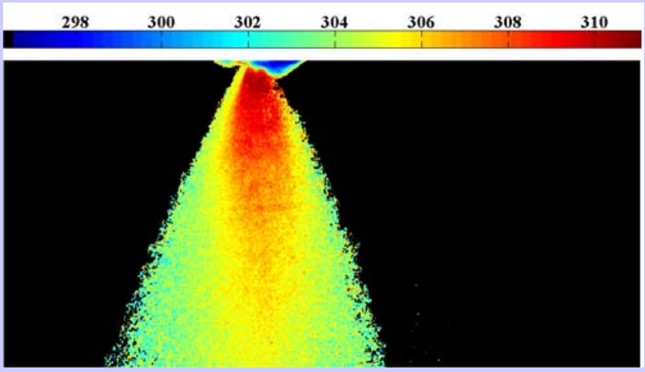
Zhang et al.



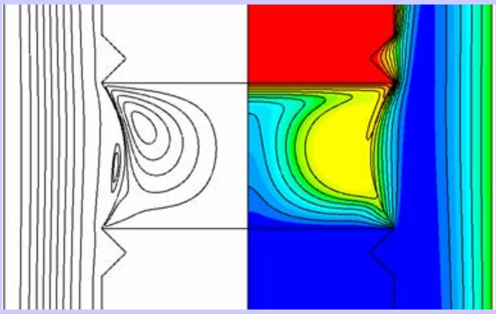
Kawanami et al.



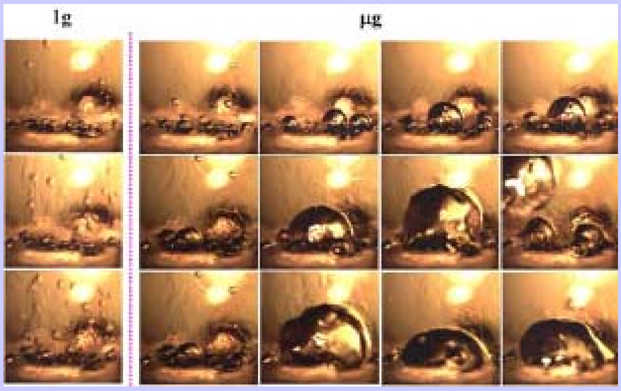
Ishii et al.



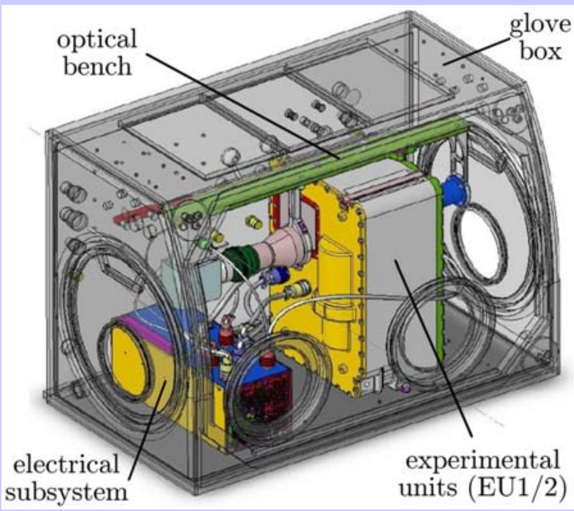
Vetrano et al.



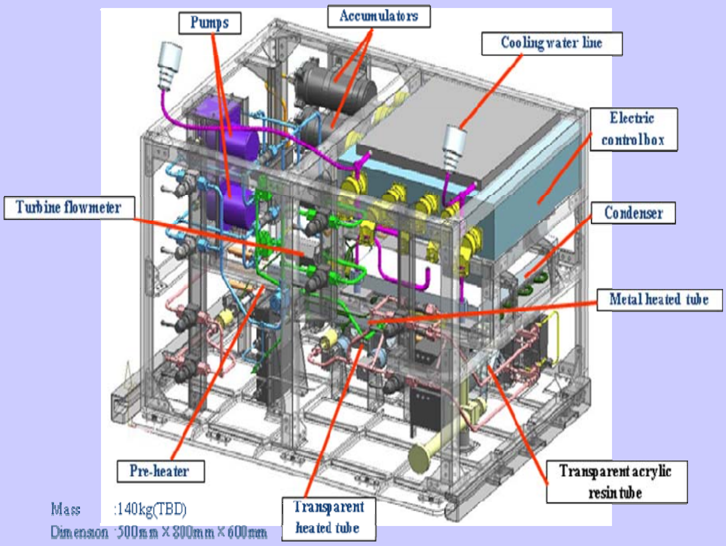
Gaponenko and Shevtsova



Xue et al.



CCF, Microgravity Glovebox ISS, Dreyer et al.



Fujii et al.