Proceedings of CHT-17
ICHMT International Symposium on Advances in Computational Heat Transfer

May 28-June 1, 2017, Napoli, Italy

CHT-17-142

NUMERICAL SIMULATION OF NUCLEATE BOILING IN MICROGRAVITY IN PRESENCE OF A SHEAR FLOW

Benjamin Franz[*,§] and Peter Stephan[*]
[*]Institute for Technical Thermodynamics, Technische Universität Darmstadt, Darmstadt, Germany
[§]Correspondence author. Phone: +49 6151 22271 Email: franz@ttd.tu-darmstadt.de

NOMENCLATURE

$\overrightarrow{f_{ST}}; \overrightarrow{f_g}$	Forces resulting from surface tension and gravity
H	Channel Height

INTRODUCTION

Nucleate Boiling is a very promising process for the cooling of electronic devices in space applications, which are continuously dissipating heat. It is not possible to employ correlations for the design of heat exchangers, which have been obtained in experiments under normal gravity conditions. The influence of gravity on local transport phenomena has to be understood in more detail. For that purpose a setup to investigate nucleate boiling in a shear flow experimentally called RUBI (Reference mUlti-scale Boiling Investigation) is planned and designed for operation aboard ISS. Prior to and accompanying the ISS experiments, numerical and experimental studies are performed on ground in order to gain a better understanding of phenomena occurring during the RUBI experiments.

Goals The main target of the simulations presented here is to examine the influence of system parameters such as flow velocity, heater power, subcooling and system pressure on the bubble detachment behaviour and the influence of the thermal boundary layer near the wall. It is to be investigated if bubble detachment can be established despite the absence of gravity. According to Siegel [1967] this could be achieved if the provided heat flux is high enough. Furthermore, it will be examined if a stable boiling process continuously dissipating heat is possible in weightlessness within a shear flow. If so, the overall heat transfer coefficient shall be determined and compared to that gained under normal gravity conditions. Full 3-D simulations of the two-phase flow including phase change, transient heat transfer between the liquid and the solid wall as well as evaporation at the three-phase contact line have to be conducted. This is challenging since the investigated geometry and high local mesh resolutions lead to a very large number of cells and the numerical methods used are rather computationally expensive. High attention has to be paid to the highest possible reduction of the computational domain within the experimental geometry and to a good parallelization of the solver.

NUMERICAL SETUP AND RESULTS

In the following section a short description of the simulation setup is given, followed by the numerical method used and the governing equations to be solved in the simulations.

Simulation Setup In Figure 1 the two computational domains of the simulations are presented. The upper part is the fluid domain, a section of the flow channel from the RUBI test cell. The lower part represents the heater. Both parts are coupled through boundary conditions. In order to reduce the number of mesh cells, the symmetry of the geometry is taken advantage of by imposing a symmetry boundary condition at the flow channel and heater center plane. The mesh is refined evenly once at the region where the flow channel and the heater meet and a second time at a region close to the nucleation site of the bubble, which is placed in the middle of the heater surface. In the most refined region each cell has an edge length of 62.5 µm. The meshes in the fluid and the solid domains are consistent at the solid-fluid interphase and refinement is performed simultaneously.

The material properties of the fluid are those of FC-72 for the liquid and those of barium fluoride for the heater, according to the experimental setup. For the calculations presented here saturated boiling is assumed with saturation conditions of FC-72 at a system pressure of 0.9 bar, this leads to a saturation temperature of 53.53 °C. All simulations are performed for a gravity level of 0.001 g.

The velocity profile of a Poiseuille flow given in Eq. (1) (with flow velocity u_x in x-direction, the maximum velocity in the middle of the channel $u_{x,max}$, the channel height H and the height coordinate z) is imposed as starting condition in the whole fluid domain and as boundary condition at the inlet (the inlet is on the left side, the outlet on the right side of the channel shown in Figure 1):

$$u_x(z) = 4u_{x,max}(1 - \frac{z}{H})\frac{z}{H} \tag{1}$$

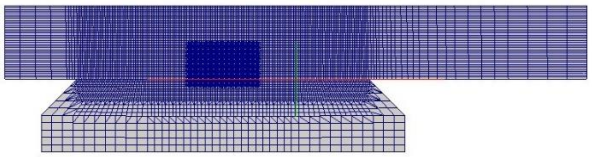

Figure 1. Computational Domain

Governing Equations The evaporating two phase flow is solved using a finite volume method combined with a Volume-of-Fluid method used for interface capturing. The equations that have to be solved in the numerical model are the conservation equations of mass, momentum, energy and volume fraction for incompressible, Newtonian fluids:

$$\nabla \cdot (\rho\vec{u}) = \dot{\rho} \tag{2}$$

$$\frac{\partial \rho\vec{u}}{\partial t} + \nabla \cdot (\vec{u} \cdot \rho\vec{u}) = -\nabla p + \nabla \cdot (\mu \cdot \nabla\vec{u}) + \vec{f}_{ST} + \vec{f}_g \tag{3}$$

$$\frac{\partial \rho cT}{\partial t} + \nabla \cdot (\vec{u} \cdot \rho cT) = \nabla \cdot (\lambda \cdot \nabla T) + \dot{h} \tag{4}$$

$$\frac{\partial F}{\partial t} + \nabla \cdot (\vec{u} \cdot F) = \frac{\dot{\rho}}{\rho} F \tag{5}$$

The volumetric forces $\overrightarrow{f_{ST}}$ and $\overrightarrow{f_g}$ on the right hand side of Eq. (3) account for surface tension and gravity, respectively. The source terms on the right hand side of Eqs. (2), (4) and (5) account for phase change. The volume fraction field *F* from Eq. (5) determines the liquid volume fraction in each cell, it has a value between zero and unity in cells that contain part of the liquid vapor interface.

The simulations are conducted with the open source CFD toolbox OpenFOAM. Based on OpenFOAM's interFoam, Kunkelmann [2011] and Batzdorf [2015] developed a solver that additionally accounts for phase change, evaporation at the three-phase contact line and for the transient heat conduction between the solid heater and the fluid. Based on a work of Hardt and Wondra [2008], an evaporation model has been developed that allows for coarser mesh resolutions and unstructured meshes. Complementing the original VOF method, a means of explicit interface reconstruction through iso surfaces is integrated. In order to take evaporation at the three phase contact line into account, a parameterized subgrid model that calculates the one-dimensional heat flow in the contactline region based on the model by Stephan and Busse [1992] has been implemented. For details on the numerical model please refer to Kunkelmann [2011] and Batzdorf [2015].

As a first step a transient simulation of the liquid phase alone has been performed for each combination of flow velocity and heater power in order to produce a thermal boundary layer above the heater as a start condition for the boiling simulation. All of these pre-simulations have been conducted until a steady state has been reached in the sense that the temperature distribution in the liquid phase did not change over a simulated time of a few seconds.

Secondly, a simulation of nucleate boiling is performed. For that purpose, the volume fraction field is altered so that a vapor bubble with a very small radius pops up at the nucleation site with a given frequency. Adaptive mesh refinement is used to increase the grid resolution in the vicinity of the liquid-vapor interface in two stages.

Results In Figure 2 the temperature distribution from a two phase simulation for $u_{x,max}$ = 0.15 m/s and a heat distribution $\dot{q}$ = 20000 W/m² at an simulation time of t = 0.04 s is shown at the symmetry plane. The position of the nucleation site is shown by the white vertical line at the left side of the picture. One can see that at this state of the boiling process the thermal boundary layer is still a lot thinner at this position because of the bubble that has grown there and moved rightwards. On the right side of Figure 2 the z-wise distribution of the temperature above the nucleation site is shown.

Figure 3 shows the evolution of the thermal boundary layer thickness over time between nucleation and 0.04 s for the same set of parameters. It has been determined as the z-coordinate of the first cell z-wise from the nucleation site where the temperature of the bulk fluid is present. The values determined for the simulation times of t = 0.01 s and t = 0.02 s by this method are not significant, though, because the bubble has not moved far enough from the nucleation site and the vertical line is partwise situated within the bubble. Because of that those values are set to the initial thickness and significant values are given by the black square markers. However, one can recognize that a minimum of the thermal boundary layer thickness is reached at t = 0.03 s where the boundary layer starts to recover.

The results obtained so far show that the presented method is a suitable way to examine the influence of the boiling process on the thermal boundary layer above the heater. A next step is to determine and to compare the recovery times for different sets of flow velocities and heat distribution and to investigate the consequences of a bubble frequency that is faster than the recovery time for stable, permanent boiling. Furthermore, the influence of system pressure and subcooling on the overall boiling process has to be investigated in parameter studies.

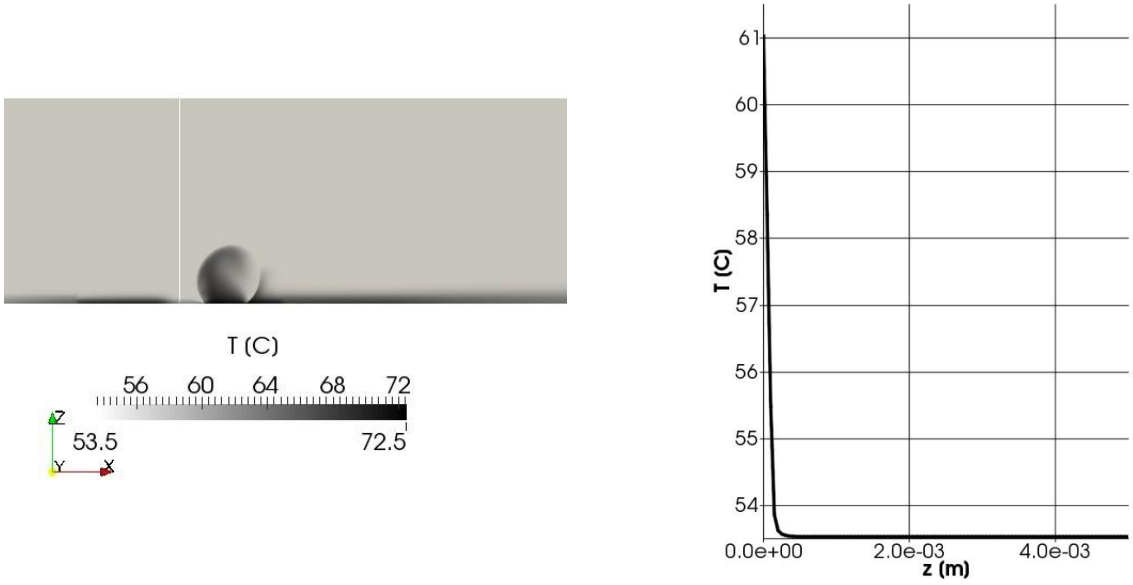

Figure 2. Temperature Distribution at a Simulation Time of 0.04 s (left) and the z-wise distribution at the nucleation site (right)

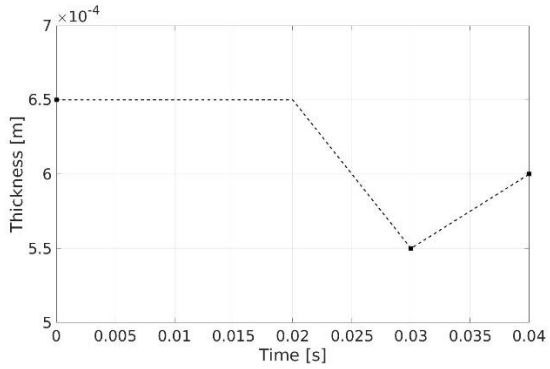

Figure 3. Evolution of the Thermal Boundary Layer Thickness at the Nucleation Site over Time

REFERENCES

Batzdorf, B. [2015], Heat Transfer and Evaporation during Single Drop Impingement onto a Superheated Wall, *PhD Thesis*, Technische Universität Darmstadt.

Hardt, S. and Wondra, F. [2008], Evaporation Model for Interfacial Flows Based on a Continuum-Field Representation of the Source Terms, *Journal of Computational Physics*, Vol. 227, pp. 5871-5895.

Kunkelmann, C. [2011], Numerical Model and Investigation of Boiling Phenomena, *PhD Thesis*, Technische Universität Darmstadt.

Siegel, R. [1967], Effects of Reduced Gravity in Heat Transfer, *Advances in Heat Transfer (J.P. Hartnett/ T.F. Irvine Jr., Ed.)*, Vol. 4, pp. 143-228, Academic Press, New York.

Stephan, P. and Busse, C. [1992], Analysis of the Heat Transfer Coefficient of Grooved Heat Pipe Evaporator Walls, *International Journal of Heat and Mass Transfer*, Vol. 35, pp. 383-391.