Master's thesis summary

Qian Bao

2016.3.20

This document is supposed to be a supplementary material to help you to better understand my research background and interests. Therefore, most contents are not constructed in a "from the ground up" way in order to avoid unnecessary tedious explanation. However any discussion will certainly be welcomed if you find anything unclear, confusing or incorrect in this document.

Thesis title: Study of Heat Transfer and Fluid Flow of Bubbles in a Microchannel
Advisor: Prof. S. Maruyama

1 Background

With the increase of operating frequency and integration of transistors, the power density of CPU is exceeding $10^6 W/m^2$ (as shown in Fig. 1) and reaches the limitation of traditional cooling technique. Innovative cooling technique is demanded to gain higher heat flux removal capability which will improve the performance of CPU and other electronic devices dramatically.

Most of today's cooling devices are utilizing forced convection of air or liquid, while boiling phenomena can evidently offer higher heat transfer coefficient, difficulty in control flow boiling phenomena makes it to be seldom used in the normal electronic product. Additionally, the normal scale boiling

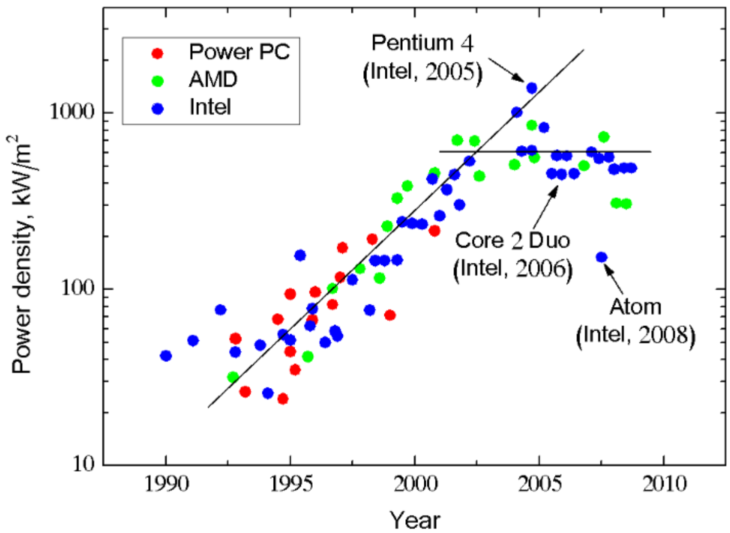


Figure 1 Power density of modern CPU [1].

is dominated by nucleate boiling and it is known that such cooling technique will fail if the heat flux exceeding critical heat flux is generated from the device surface because of the “dry out” of heat transfer surface.

In order to exceed this barrier and also improve the controllability, a cooling technique utilizing evaporation of liquid film in a microchannel has been proposed by our laboratory (at Tohoku Univ.). In a microchannel, as long as the nucleate boiling is suppressed by controlling channel diameter, expansion of a single bubble and evaporation of liquid film surrounding it will dominate the flow pattern. Although two-phase flow in a microchannel is being intensively studied in the last two decades, research focusing on the film behavior and heat transfer characteristic is far from sufficient and a deeper understanding is needed to develop such cooling devices.

The objectives of this study are to formulate a simulation program and by simulating bubble fluid flow, to understand fluid flow and heat transfer characteristic inside microchannel and help the development of innovative film evaporating cooling technique.

2 My approach to the project

My original plan was to build my own two-phase flow solver from scratch. The reason that we didn’t consider a commercial software package (like AN-

SYS Fluent) or opensource one (OpenFOAM) was:

- Commercial software may lack the flexibility in the future since the core solver is proprietary and no source code will be available for studying or modifying.
- Available opensource code like OpenFOAM has already a huge framework and complex class structure, which may be too time consuming to study it thoroughly.

As shown in Fig. 2,there are mainly 3 steps:

- Creating an incompressible, single-phase, laminar flow solver.
- Implementing interface tracking algorithm (VOF) [2].
- Implementing surface tension model (CSF) [3].

In the first year, a single-phase flow solver (implementing the SIMPLE algorithm [4]) written in Matlab language is successfully built. However, when I tried to implement the VOF method on top of my code, I noticed that the lack of consideration over program structure and software interfaces design was a disaster which made it almost impossible for me to further develop high-level algorithm.

On the other hand, thoroughly study over those programming aspects was something I could not afford as a master student. Thus in the second year, I changed the approach from building everything from scratch to utilizing available numerical code.

Results of each phase of my research will be described in the following sections.

3 Phase 1: Creating a SIMPLE method simulator in Matlab from scratch

In this phase, a simple solver for single-phase, laminar flow implementing the SIMPLE algorithm was created from scratch (in Matlab). Navier-Stokes equations for incompressible flow are selected to model the problem:

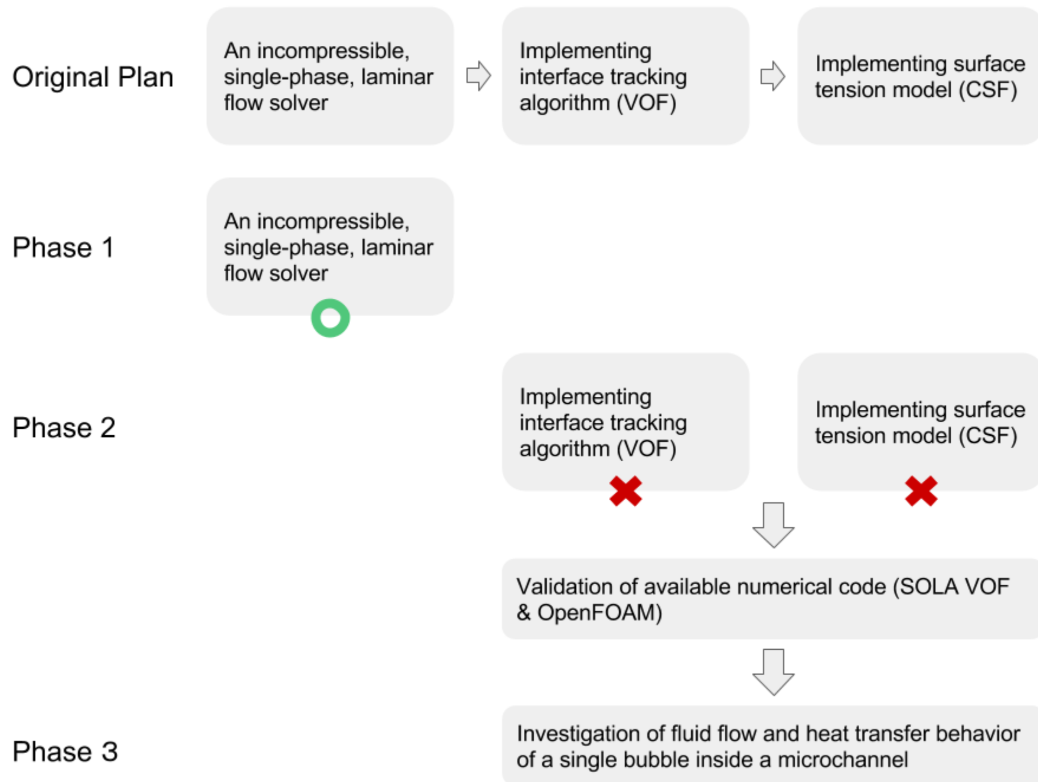


Figure 2 A diagram explaining my master's degree research.

$$\nabla \cdot \vec{v} = 0 \tag{1}$$

$$\frac{\partial(\rho\vec{v})}{\partial t} + \nabla \cdot (\rho\vec{v}\vec{v}) = -\nabla p + \mu\nabla^2\vec{v} + \rho g + F_b \tag{2}$$

$$\frac{\partial \rho c_p T}{\partial t} + \nabla \cdot (\rho c_p T\vec{v}) = \nabla \cdot \vec{q} \tag{3}$$

Several simulations are carried out to show the ability of this computation code. In order to validate the single-phase solver, a typical piple flow model is made as shown in Fig. 3. Also velocity profile on the exit was compared with solution of Hagen-Poiseuille flow.

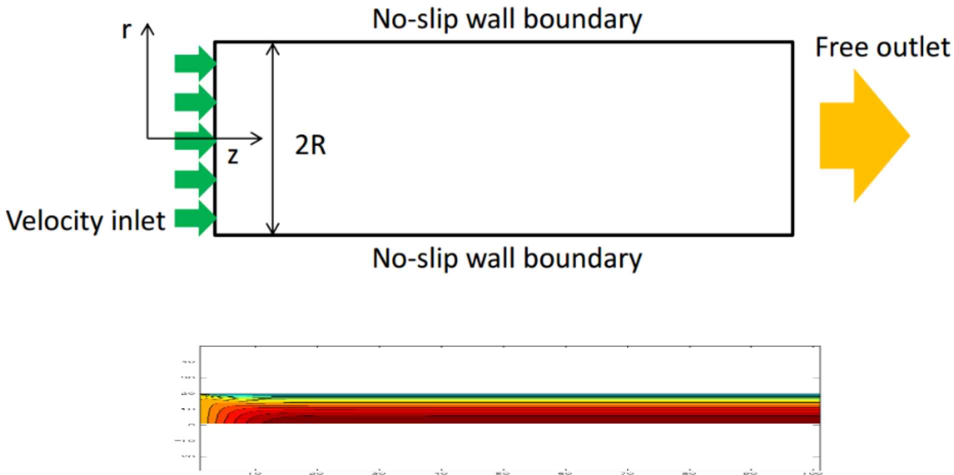


Figure 3 A test case for validation.

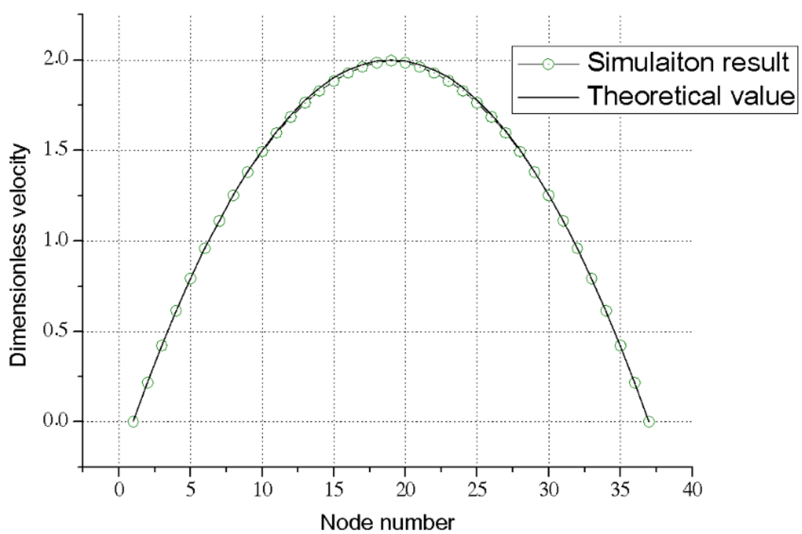


Figure 4 Velocity profile on the exit of pipe flow model.

4 Phase 2: Validation of SOLA-VOF and further results

As described before, I decided to change the approach in the second year of my master course. SOLA-VOF is a numerical solver package for two-phase flow which was originally developed at the Los Alamos National Lab in 1980s [5], and it's available for academical use recently.

To test the ability of the solver, I arranged several numerical experiments including the famous ”dam break” (in Fig.5) and not so famous ”steady bubble” (in Fig.6).

In the ”dam break” case, the calculated results showed good agreement with experimental data[6].

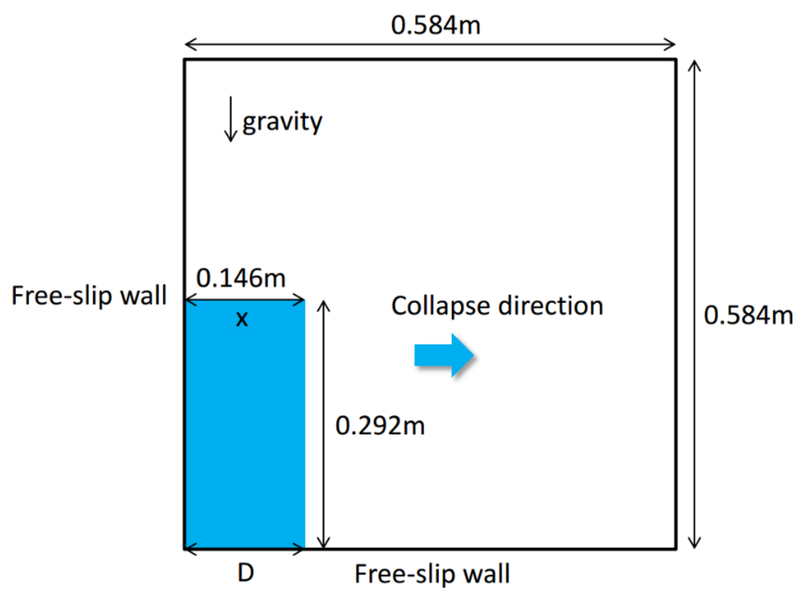


Figure 5 A typical dam break test case.

In the ”steady bubble” case, an air bubble is surrounded with water in a resting state. No gravity or other body force is included so it's predicted that no velocity in both gas and liquid phase should occur. However due to the surface tension model utilized in SOLA-VOF has a nature to generate unreal numerical velocity [3], it was found that the interface is under unstable condition and the original bubble shape may be destroyed.

With attention to this special phenomenon, several numerical experiments were conducted to show the behavior of single bubble inside a microchannel.

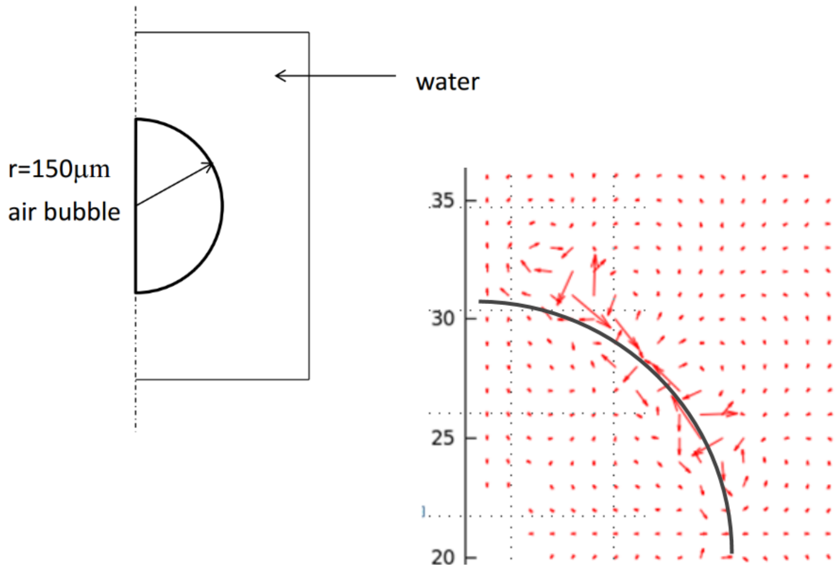


Figure 6 A ”steady bubble” test case and the parasitic current velocity observed.

Fig.7 shows the physical model and calculated results. The velocity of gas phase (air) and liquid phase (water) at the inlet is $1m/s$, and the diameter of the channel is $280\mu m$. The inlet velocity should be kept above certain value otherwise the parasitic current velocity may dominate the flow pattern and thus the results would be meaningless. It was also noted that due to the lack of grid resolution near the wall boundary, there may occur numerical ”dry out” in which the liquid film surrounding the bubble disappear as if the liquid film is evaporated by heat.

The bubble shape was compared with experimental result from other references[7] in Fig.8 and showed good agreement.

As a final step, energy equation was added to the solver to calculate the temperature distribution and heat exchange between the fluid and the wall. It should be mentioned that no phase change model was included and as a result, the bubble could not grow or shrink as in reality and the calculated heat flux was expected to be from experiment, too.

Fig.9 shows that the gas phase temperature was fixed on 100°Cand the temperature of overheat water was 105°C. The wall temperature was also fixed to 105°Cand thus heat would be absorbed from wall due to the exis-

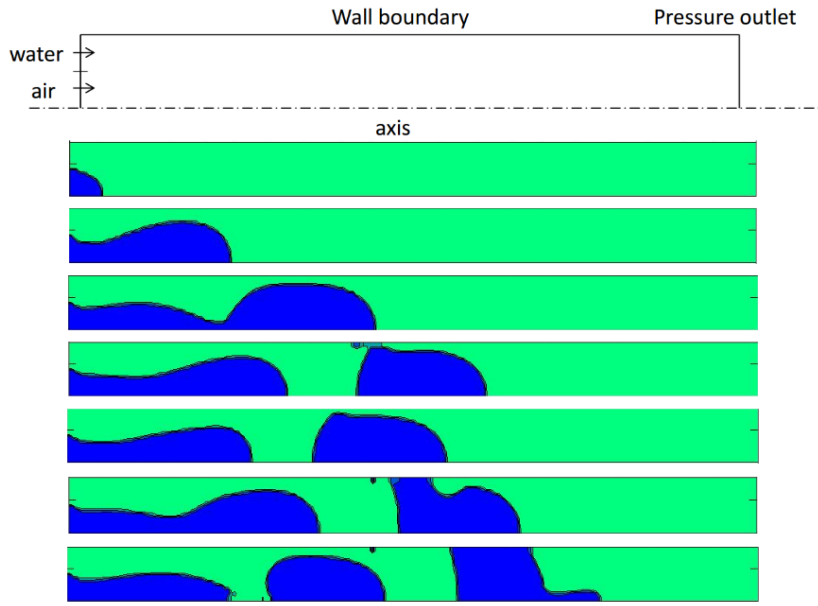
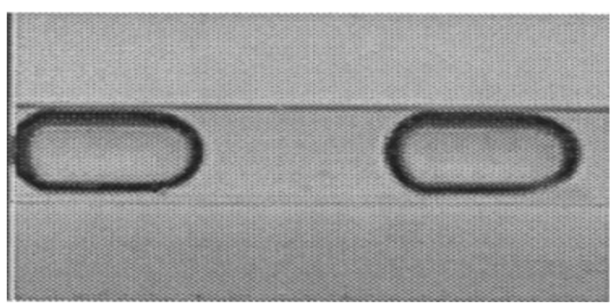
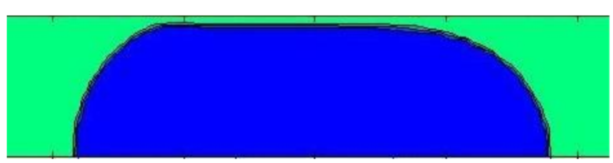


Figure 7 A microchannel model and numerical results showing the bubble shape.



Triplett experimental result



Numerical result in this study

Figure 8 Calculated bubble shape compared with experimental result.

tence of thin liquid film between gas bubble and wall boundary.

Fig.10 shows the calculated result of Nusselt number along the long-axis of the channel. The Nusselt number was defined as below to reflect the heat

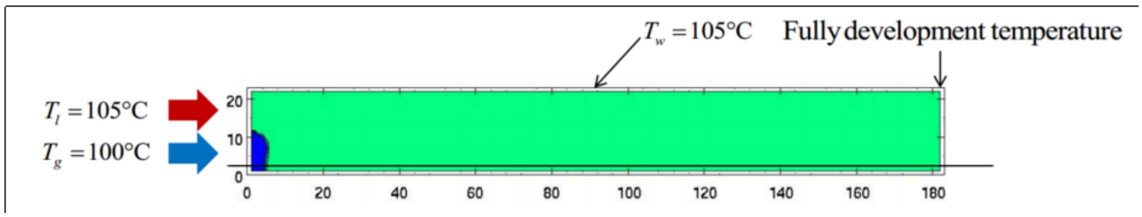


Figure 9 Temperature boundary condition used in this study.

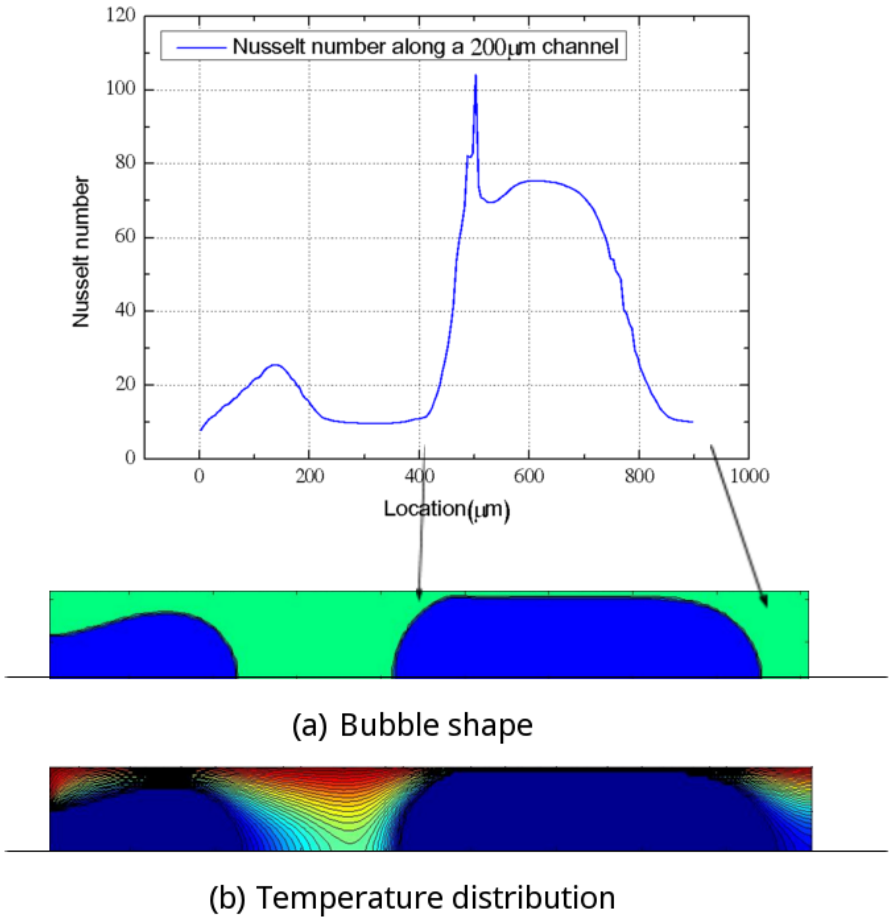


Figure 10 Calculated temperature distribution and derived Nusselt number.

exchange rate:

$$Nu = \frac{q_w}{T_w - T_b}\frac{d}{k_L} \tag{4}$$

And T_b is an artificial temperature used to represent the average temperature on a cross section of the channel.

It was found that the existence of liquid film greatly promoted the heat exchange between the fluid and wall, even there was no evaporation effect included in this study. And the spike of Nusselt number on the tail of the bubble suggested that the thickness of liquid film would be a key factor that affect the heat exchange rate.

5 Concluding remarks

The objectives of this study are to formulate a simulation program and by simulating bubble fluid flow, to understand fluid flow and heat transfer characteristic inside microchannel and help the development of innovative film evaporating cooling technique.

In the first phase of this study, I successfully built a single phase laminar flow solver in Matlab. In order to validate the single-phase solver, velocity profile on exit was compared with solution of Hagen-Poiseuille flow in pipe. Results showed good agreement but due to the lack of experience in software development and limitation of time, I decided to change the approach and utilize the available numerical code to continue the study.

In the second phase of this study, I studied the source code of SOLA-VOF method and validated it by conducting several experiments. Although some numerical issues including parasitic current and numerical dry out near wall boundary was found, by carefully setting up the calculation case, a single bubble flow inside a $280\mu m$ microchannel was successfully simulated. Expansion and acceleration of bubble caused by the evaporation of liquid film was neglected to simplify the phase change calculation procedure. Film thickness was found to be extremely important which affect the heat flux directly. Nusselt number and the heat flux result were used to evaluate the cooling ability in current simulation model.

As future work, numerical method to suppress the ”parasitic current” will be necessary since the existence of this unreal numerical velocity limited the computable velocity range. Including the evaporation model will make it possible to simulate actual bubble growth and shrink instead of air bubble, which will certainly increase the value of the numerical results and may suggest interesting investigation for further experimental research.

References

[1] Junosuke Okajima. “Phase change phenomena and heat transfer in a microchannel and their application to biological cooling system”. In: Tohoku university Phd Dissertation (2011).

[2] C.W Hirt and B.D Nichols. “Volume of fluid (VOF) method for the dynamics of free boundaries”. In: Journal of Computational Physics 39.1 (1981), pp. 201–225.

[3] J.U Brackbill, D.B Kothe, and C Zemach. “A continuum method for modeling surface tension”. In: Journal of Computational Physics 100.2 (1992), pp. 335–354.

[4] S. Patankar. Numerical Heat Transfer and Fluid Flow. Series in computational methods in mechanics and thermal sciences. Taylor & Francis, 1980.

[5] B.D. Nichols et al. “SOLA-VOF: A Solution Algorithm for Transient Fluid Flow with Multiple Free Boundaries”. In: Los Alamos National Scientific Laboratory (1980).

[6] W. J. Moyce J. C. Martin. “Part IV. An Experimental Study of the Collapse of Liquid Columns on a Rigid Horizontal Plane”. In: Philosophical Transactions of the Royal Society of London. Series A, Mathematical and Physical Sciences 244.882 (1952), pp. 312–324.

[7] K.A. Triplett et al. “Gas–liquid two-phase flow in microchannels Part I: two-phase flow patterns”. In: International Journal of Multiphase Flow 25.3 (1999), pp. 377–394.

Nomenclature

μ	Dynamic viscosity.
ρ	Fluid density.
$\vec{q}$	External heat flux.
$\vec{v}$	Flow velocity vector.
c_p	Specific heat of fluid.
d	Diameter of the microchannel.
F_b	Other body forces.
g	Gravity (vector).
k_L	Heat conduction of the fluid.
Nu	Nusselt number.
p	Pressure.
q_w	Heat flux on the microchannel wall.
T_b	Average temperature on cross section of a microchannel.
T_w	Wall temperature of a microchannel.