Numerical verification of heat losses in experiments with microchannel under local heating

V V Belosludtsev.1,2
1 Kutateladze Institute of Thermophysics SB RAS, 630090, Novosibirsk, Russia
2 Novosibirsk State University, 630090, Novosibirsk, Russia

E-mail: v.belosludtsev@g.nsu.ru

Abstract. With the help of three-dimensional numerical simulation in FEM software package the experimental methodology of determining heat flux and heat losses during experiments in microchannel was verified. The obtained single-phase flow data on the heat transfer coefficient and wall temperature show good agreement between numerical simulation and experimental results.

1. Introduction
Recent advancements in manufacturing of microchips allowed quick progress in such areas as high-performance computing, artificial intelligence and Internet of things. To maintain rapid progress in areas related to high performance computing, further advancements in manufacturing of such microchips are needed. One of the most promising manufacturing trends in semiconductor industry is three-dimensional integration of microchips as stated in [1]. Scientists and engineers are paving the way for implementation of three-dimensional integration to increase efficiency and performance of electronics. Three-dimensional integrated circuits are already implemented in low energy applications such as systems on a chip in mobile devices and multilayer memory chips. Packaging advancements is one of the major trends of microelectronics development, but to be successfully implemented in high performance microchips new highly efficient cooling approach is needed [1]. One of the most convenient approach for thermal management of three-dimensionally stacked chips is microchannels inside those chips. Boiling in microchannels [2–4] should be able to remove high heat fluxes in 3D stacks of Thigh-performance chips. Except boiling, there are other ways of thermal management which can be implemented to remove high and uneven heat fluxes: 1) spray cooling [5], 2) micro-jet cooling [6], 3) cooling by falling liquid films [7, 8], and 4) cooling by evaporation of a thin liquid film shear-driven in a channel [9, 10]. Microchannel boiling has advantages such as capability to maintain uniform operating temperature due to phase change, easier to implement as compared to two-phase solutions such as shear-driven films and lesser pumping requirements as compared to spray cooling techniques.

Flow boiling on localized heat sources is not well studied unlike conventional flow boiling in channels with uniform heating. So that, it is important to carefully implement and improve experimental techniques to study flow boiling on localized heat sources to be able to determine correctly the main heat transfer characteristics such as heat transfer coefficient, wall temperature and critical heat flux. It follows that it is necessary to carefully examine and determine all experimental uncertainties, such as heat spreadings into test section and heat losses into the atmosphere.

The main purpose of this paper was to validate methodology for determination of heat flux and heat losses in our test section suggested in [11, 12] before proceeding to do experiments on boiling. To
validate methodology suggested in [11,12], conjugate laminar and turbulent single-phase heat transfer was numerically simulated in closed code FEM simulation software package.

2. Experimental setup
Schematic of experimental setup is presented in figure 1. Experimental setup includes the following main components: test section, digital data acquisition system, and working fluid supply circuit.

![Figure 1. Scheme of the experimental setup. 1 – pump Ismatec Reglo-ZS, 2 – plate heat exchanger HXP-193, 3 – thermostat Huber MPC K6, 4 – flow meter Titan Atrato Ultrasonic, 5 – working area, 6 – pressure sensor WIKA P-30, 7 – tank with working fluid.](image)

Milli-Q water was used as the working liquid and it was supplied to the test section by Ismatec Reglo Z-120 gear pump. The liquid flow rate was measured by ultrasonic flowmeter and it was adjusted by means of gear pump. The liquid passing through a plate heat exchanger, in which liquid was thermally stabilized to the required temperature with the aid of a thermostat. The working liquid temperature of 26°C was maintained and controlled by a thermocouple at the test section inlet. The deviation of the temperature of the working liquid at the inlet to the test section from 26°C during the experiments was no more than ± 0.5 °C. The experiments were carried out under normal pressure. Pressure at the inlet of the channel was measured by pressure sensors. The pressure drop along the channel in the experiments did not exceed 0.2 Bar.

The test section shown in Fig. 2 in longitudinal cross-section, consists of a stainless-steel flat plate with a flush-mounted copper heater rod having a square head of 1 × 1 cm which is serving as working surface. The rod is electrically heated from sides using a nichrome wire. The design of the heater provides close to constant temperature on the surface of the rod, \( T_{\text{wall}} = \text{const} \). The top wall of test section is formed by a transparent glass. Changeable inserts are placed between the working substrate and the glass cover, so that a channel with a desired height is formed. The height of the channel H was 0.6 mm
and it was measured by Micro-Epsilon IFC2451, unevenness of a channel was below 5%. The width of the channel was 34 mm, i.e. 3.4 times of the heater width. The working surface (stainless steel plate with copper rod) was roughly polished. The morphology of the working surface was analyzed using an atomic force microscope. The root mean square (RMS) surface roughness was found to be 0.5 µm. The heat flux was determined using thermocouples inserted into the test section (for more details see [11]),

**Figure 2.** Schematic of the test section. 1 – stainless steel plate, 2 – copper heater rod, 3 – PTFE seal, 4 – nichrome wire, 5 – heat-insulating coat of a heater, 6 – glass, 7 – liquid inlet nozzle.

3. **Numerical model**

The model in the numerical simulation package was made as close as possible to the real work area. Basic components, such as a steel plate and a copper heater, were introduced into the model, also a fluoroplastic O-ring (Fig. 2) and thermal contact resistance of the copper with steel were also introduced. The introduced thermal resistance reflected the actual contact resistance between the steel plate and the head of the copper heater as a result of sealing the joint of parts with a high-temperature sealant. The thermal resistance parameters - thickness (150 µm) and thermal conductivity of the layer (0.5 W / m * K) - were manually selected and remained unchanged for whole data set.

An experiment and numerical study were carried out at a fluid flow rate of 60–1500 ml/min (Re = 70–1700, determined by the hydraulic diameter of the channel), input heating power Q_J = 20–100 W, channel height of 0.6 mm and flow width of 34 mm. The liquid inlet temperature in the simulation was set equal to the experimental. For low Reynolds numbers, a laminar flow with conjugate heat transfer and no-slip condition at the channel boundaries was simulated, for high Reynolds numbers, the SST turbulence model (Shear Stress Transport [13]) was used, which is a low Reynolds number model and it does not require wall functions to be applied to the boundaries, therefore, it is one of the most suitable RANS model to correctly calculate heat and mass transfer. The use of the turbulent model for numerical simulation starting from a water flow rate of 750 ml/min (Re = 830, determined by the hydraulic diameter of the channel) gave the best agreement with the experimental data. This can be caused by an earlier transition to the turbulent heat transfer mode for rectangular channels with a small hydraulic diameter, as shown in [14], in which a completely turbulent heat transfer regime was observed starting from a Reynolds number of 400.

For the simulation, the Joule heat was set on the side surface of the copper heater, this heat was determined by the thermocouple measurements which was installed in the heater (Fig 2), as the Joule heat from the power source minus the heat loss from the nichrome coil into the atmosphere, caused by the imperfection of the heater insulation coat.

The mesh independence study was done prior main simulations and suitable grid parameters were found, for turbulent simulation average dimensionless wall distance was found to be 1.59.
Dimensionless wall distance should be of the order of 1; that is required by SST model to correctly predict quantities, such as heat flux from the wall without additional wall functions. Total number of mesh elements was slightly above 1 million in case of turbulent simulations.

4. Results and discussion
The results of numerical simulations in comparison with experimental data for the heat transfer coefficient (Fig. 3), wall temperature (Fig. 4–6) and the wall temperature profile versus the longitudinal coordinate (Fig. 7) were obtained for the given values of input power on the nichrome coil of the heater.

As it can be seen from Fig. 11, the numerical simulation results of the heat transfer coefficient are in good agreement with the experimental results, which indicates the correct approach of determining the true average heat flux from the surface of a copper heater in the experiment: the difference between the calculation and the experiment does not exceed 10%. Herewith, the total Joule heating power in this experiment is up to 3.5 times higher than the power from the surface of the heater, i.e. total heat loss in the experiment is up to 73%.

![Figure 3](image-url). Comparison of the heat transfer coefficient for numerical simulation (CFD) and experiment (Exp). The legend shows the Joule heating power $Q_J$ during the experiment.
Figure 4. The comparison of numerical (CFD) and experimental (Exp) wall temperature measured by therocouple T$_3$. The legend shows the Joule heating power $Q_J$ during the experiment.

Figure 5. The comparison of numerical (CFD) and experimental (Exp) wall temperature measured by thermocouple T$_4$. The legend shows the Joule heating power $Q_J$ during the experiment.
Figure 6. The comparison of numerical (CFD) and experimental (Exp) wall temperature measured by thermocouple $T_5$. The legend shows the Joule heating power $Q_J$ during the experiment.

Figure 7. Wall temperature versus the longitudinal coordinate, comparison of experiment (Exp) and numerical simulation (CFD). Fluid flow rate - 1500 ml/min. The legend shows the Joule heating power $Q_J$ during the experiment. Zero of longitudinal coordinate “X” is taken at the liquid inlet.
The readings of thermocouples $T_3$ and $T_4$ closest to the heater (see Fig. 4-5), as well as the last thermocouple after the heater $T_5$ (Fig. 5), are in good agreement with experimental data. The discrepancy between the numerical simulation and experimental values of the thermocouple $T_2$ (Fig. 7) is probably caused by heating of the insulating textolite case of the experimental setup; to take this effect into account, more complex numerical model is necessary. Temperature of thermocouple $T_1$ (Fig. 7) is fully determined by liquid inlet temperature, therefore there is no difference between simulation and experiment.

In Fig. 7 comparison of the wall temperature profile obtained by numerical simulation with experimental data is presented. The experimental wall temperature in the center of the heater is in good agreement with the numerically simulated temperature curve for the entire data set. As it can be seen from the graph, the calculated temperature of the surface of the heater rises along the flow, which is associated with the heating of liquid as it moves along the heater.

**Conclusions**

Experiments on convective heat transfer of subcooled water flow in a mini-channel under local heating, were conducted. Verification of the experimental methodology of evaluating the heat flux and heat loss by numerically simulating heat transfer in the substrate is performed. The obtained results of complex three-dimensional numerical simulation of heat transfer coefficient and wall temperature are in good agreement with the experimental results. The approach shown in this article can be used for verification of heat losses in more complex experiments.

**Acknowledgments**

The study was performed under the support of Russian Scientific Foundation (Grant No. 18-79-10258).

**References**

[1] International Roadmap for Devices and Systems 2020 *Institute of Electrical and Electronics Engineers*

[2] Thome J 2004 *International Journal of Heat and Fluid Flow* **25**

[3] Zaitsev D, Tkachenko E, Belosludtsev V, Kreta A, Kabov O 2018 *Journal of Physics: Conference Series* **1105** 012142

[4] Belosludtsev V, Zaitsev D 2020 *AIP Conference Proceedings* **2212** 020011

[5] Kim J 2007 *International Journal of Heat and Fluid Flow* **28**

[6] Robinson J, Kempers R, Colenbrander J, Bushnell N, Chen R 2018 *Applied Thermal Engineering* **136**

[7] Chinnov E A, Kabov O A, Marchuk I V, Zaitsev D V 2002 *International Journal of Heat and Technology* **20** 69–78

[8] Zaitsev D V, Semenov A A, Kabov O A 2016 *Thermophysics and Aeromechanics* **23** 625–8

[9] Tkachenko E, Zaitsev D 2016 *MATEC Web of Conferences* **72** 01114

[10] Zaitsev D, Tkachenko E and Kabov O 2017 *EPJ Web of Conferences* **159** 0054

[11] Zaitsev D V, Tkachenko E M 2019 *Journal of Physics: Conference Series* **1369** 012058

[12] Tkachenko E M, Zaitsev D V 2019 *AIP Conference Proceedings* **2135** 020057

[13] Menter F R, Kuntz F, Langtry R 2003 *Turbulence Heat and Mass Transfer* **4** 625–32

[14] Peng X F, Peterson G P 1994 Experimental Heat Transfer An International Journal **7**(4), 265–83