Experimental setup for studying two-phase flows in micro- and minichannels at ultra-high heat fluxes: methodology and first experimental results

D V Zaitsev¹,² and V V Belosludtsev¹,²

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. The study of phase-change phenomena under high and ultra-high heat fluxes is urgent because of fast development of electronics and microelectronics. We have developed a test section with power of 3.5 kW with a heater of 1x1 cm² and adjustable geometry of the channel for achieving ultra-high heat fluxes in flow boiling and shear-driven liquid film experiments. The methodology of calculating heat losses in the test section is proposed and verified by flow boiling experiment versus another well studied test section. Observed trend of decrease of relative heat losses with increase in the heat flux makes it possible to assume that the heat flux as high as 2.5 kW/cm² can be reached by this test section.

1. Introduction

Study of phase-change phenomena under high and ultra-high heat fluxes is driven by electronics industry needs [1]. Volumetric heat fluxes in future three-dimensionally integrated chips is expected to reach values as high as 10 kW/m³ [2] with uneven distribution of the heat flux. In addition, there is a new generation of power electronics basing on GaN transistors and it has very good characteristics for high power conversion applications and, as such, it will require very effective cooling solutions. Development of power and microelectronics is slowed down by a inability of existing thermal management solutions to cope with the ultra-high heat fluxes.

Heat transfer in mini- and microchannels under high heat fluxes is being widely studied. Effects of microchannel aspect ratio on heat transfer is studied for different conditions [3-4] and in all cases authors note the effect of channel geometry on the flow boiling heat transfer characteristics. Boiling under heat fluxes above 1 kW/cm² was studied in [5-7]. Heat fluxes above 1 kW/cm² were achieved during flow boiling on diamond substrate covered by porous copper. In [6] flow patterns and heat transfer characteristics of boiling in microgaps under heat fluxes of up to 4 kW/cm² was studied for different channel heights but only in case of uniform heating from the wall. Importance of understanding of liquid-vapor interactions during boiling was demonstrated in [7]. Tapering of the channel allowed to achieve heat flux above 1 kW/cm², because such channel geometry promoted bubble expansion along flow and reduced pressure drop.

Observation of flow boiling hydrodynamics in mini- and microchannels is of great importance for understanding the boiling phenomena in such conditions. In [8-9] it was shown that bubble can reach typical dimensions of minichannels of in less than 1 ms, which could lead to heat transfer crisis. Therefore, it is necessary to be able to visualize boiling process in test facilities capable to achieve ultra-
high heat fluxes. In [9] the authors developed a technique to study the dynamics of vapor bubbles in a microchannel with the shooting speed of up to 775,000 frames per second.

In this paper we present a brief description of recently developed test section with power of 3.5 kW and first experimental results of flow boiling under moderate heat fluxes and compare obtained results with previous results obtained in a well-tested facility [10-12].

2. Experimental setup

A schematic of the setup is presented in Fig. 1. Working fluid is supplied by a pump with variable flow rate (1) through a heat exchanger (2) connected to a thermostat (3). Working fluid leaves the heat exchanger with a desired degree of subcooling, passes through an ultrasonic flow meter (4) and enters the test section (5). Excessive heat is dumped into the atmosphere by means of secondary fluid loop (7 - 8) which is also connected to the main fluid tank (6). The test section was designed to achieve ultra-high heat flux of up to 2.5 kW/cm². A schematic of the test section is presented in Fig. 2.

![Figure 1. Schematic of experimental assembly. 1 – Pump with variable flow rate, 2 – plate heat exchanger, 3 – thermostat with controller, 4 – ultrasonic flow meter, 5 – test section, 6 – fluid tank, 7 – auxiliary pump, 8 – plate heat exchanger connected to cooling tower.](image)

Heater block made of copper (1 in Fig. 2) is inserted to the stainless-steel plate (2) through asbestos gasket (6) which has low thermal conductivity, as low as 0.4 W/mK, combined with excellent working temperature up to 450°C. To ensure rigidity and to minimize the heat losses a TECAPEEK Natural plate with thermal conductivity below 0.27 W/mK is used as a base of the test section. The heat is generated in the copper heater by 7 cartridge heaters rated up to 500 W each, which gives 3500 W of total Joule heat. To additionally minimize heat losses, copper block was nickel plated on the outside. Thermal contact between cartridge heaters and heater block is provided through thin layer of liquid metal thermal interface material based on Gallium alloy. The copper heater block is wrapped with sheets of aerogel (9) with thermal conductivity of 0.019 W/mK at room temperature, greatly reducing heat losses into the atmosphere.

The literal sides of working channel are formed by PTFE inserts (5) with desired thickness and width, so that channel height and width can be adjusted by replacing the inserts. Top wall of channel is made
of optical glass with thickness of 15 mm allowing visualization of experiment. Fluid, represented in Fig. 2 as blue arrows, enters the channel from inlet nozzle (7), boils on the copper heater and leaves the test section at outlet (8).

To measure heat losses 9 thermocouples are installed in the stainless steel plate. Six thermocouples are placed along the flow as shown in Fig. 3, and three thermocouples are placed across the flow. Other six thermocouples (with diameter of 0.25 mm) are placed in 2 rows inside the copper heater head (Fig. 3) with distance between rows equal to 2 mm. These thermocouples are used to calculate the heat flux and wall temperature.

![Figure 2](image2.jpg)

**Figure 2.** Longitudinal cross-section of the test section. 1 – Copper heater block, 2 – stainless steel plate, 3 – TECAPEEK base, 4 – glass top wall, 5 – PTFE insert, 6 – asbestos gasket, 7 – fluid inlet nozzle, 8 – fluid outlet, 9 – aerogel insulation, 10 – cartridge heaters.

![Figure 3](image3.jpg)

**Figure 3.** Thermocouple placement in the working plate and heater block.

### 3. Data reduction and results

To accurately calculate heat flux from the heater it is needed from the total heat to subtract two main parts of heat losses - to the atmosphere and heat spreadings into the stainless-steel plate. Since two rows of thermocouples (T_{upper} and T_{lower} in Fig. 3) are placed right below the stainless-steel plate there is no need in calculation of heat losses into the atmosphere. Heat spreading into the working plate can be decomposed into three terms - along the flow direction, counter to the flow direction and athwart to the...
flow direction. Total heat loss into the stainless-steel plate can be obtained using the thermal conduction law:

\[ Q = \frac{S}{4} \cdot k \cdot \frac{(T_3 - T_2)}{\Delta x} + \frac{S}{4} \cdot k \cdot \frac{(T_4 - T_5)}{\Delta x} + \frac{S}{2} \cdot k \cdot \frac{\Delta T_{side}}{\Delta x} \]

Where \( S \) is the side area of a cylinder passing between the closest thermocouples pair in the stainless-steel plate, with the height of the cylinder being equal to the thickness of the stainless-steel plate. Since the heat spreading in athwart direction should be symmetrical, factor of 1/2 is used in the third term instead of 1/4. \( \Delta T_{side} \) is temperature difference in athwart direction to the flow; \( k \) – thermal conductivity of the stainless-steel, \( \Delta x \) – distance between pair of thermocouples. Then we can subtract the total heat into the plate from the heat obtained by using thermal conduction law with average temperatures of upper and lower row of thermocouples in heater block head, and thus we can obtain the average heat flux from the heater wall into the working fluid.

Figure 4. a) Critical heat flux for new and old test section. Channel width is 32 mm, channel height is 1 mm for both sections. b) Boiling curves for the new test section for different flow rates.

To verify the above proposed methodology of calculation of the heat flux, a flow boiling experiment was done in the channel with the height of 1 mm and width of 32 mm and compared to the experiment in exactly the same conditions conducted using an old test section. The methodology in old experimental setup was thoroughly verified experimentally and numerically [10-12]. As can be seen from Fig. 4a, maximum deviation between data (up to 40%) occurs at low flow rates below 60 kg/m²s and at high flow rates deviation of experimental data is within experimental error. Flow boiling curves behavior with increase of flow rate and heat flux (Fig. 4b) is also comparable to known studies, for example [7]. With the increase of the heat flux on the heater the total heat losses (losses to the atmosphere from the heating cartridges and heat spreadings into the stainless-steel plate) are reduced to 30% at heat flux of 250 W/cm² and above (Fig. 5). This trend should continue up to the moment where radiation heat losses start to be not negligible [10]. Basing on this trend we suggest that our test section is capable to provide heat fluxes as high as 2.5 kW/cm².
**Figure 5.** Total heat losses (losses to the atmosphere from the heating cartridges and heat spreadings into the stainless-steel plate) versus heat flux.

### 4. Conclusions

A test section with adjustable geometry of the channel was developed for achieving ultra-high heat fluxes in flow boiling and shear-driven liquid film experiments. The methodology of calculating heat flux and heat losses in the test section is proposed and verified by flow boiling experiment versus another well studied test section. Observed trend of decrease of relative heat losses with the heat flux, combined with the Joule heating power of 3500 W, makes it possible to assume that heat flux of 2.5 kW/cm² and beyond could be reached by this test section.

### Acknowledgments

This work is supported by the Russian Science Foundation under grant № 19-19-00695.

### References

[1] Kabov O A, Zaitsev D V, Cheverda V V, Bar-Cohen A 2011 Experimental Thermal and Fluid Science 35(5)
[2] Bar-Cohen A, Wang P 2012 The Journal of Heat Transfer 134 (5)
[3] Fu B R, Lee C Y, Pan C 2013 International journal of heat and mass transfer 58 (1-2), 53-61
[4] Markal B, Aydin O, Avci M 2016 International Journal of Heat and Mass Transfer 93 130-143
[5] Palko J W et. al. 2017 Advanced Functional Materials 27(45)
[6] Nasr M H et al. 2017 International Journal of Heat and Mass Transfer 108
[7] Kalani A, Kandlikar S G 2015 Applied Physics Letters 107(18)
[8] Zaitsev D, Tkachenko E, Belosludtsev V, Kreta A, Kabov O 2018 Journal of Physics: Conference Series 1105 (1)
[9] Belosludtsev V V, Zaitsev D V 2020 AIP Conference Proceedings 2212 020011
[10] Zaitsev D V, Tkachenko E M, 2019 Journal of Physics: Conference Series 1369 012059
[11] Tkachenko E M, Zaitsev D V 2019 AIP Conference Proceedings 2135 020057
[12] Belosludtsev V V, Tkachenko E M, Zaitsev D V 2021 AIP Conference Proceedings 2337 020001