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

May 28-June 1, 2017, Napoli, Italy

CHT-17-224

MODELLING OF HIGH-SPEED JET COOLING ON MICROSCALE

Tomer Rozenfeld[a*], Ashwin Kumar Vutha[b], Yingying Wang[c], Gennady Ziskind[a], Yoav Peles[c]
[a]Department of Mechanical Engineering, Ben-Gurion University of the Negev, Beer-Sheva, Israel.
[b]Department of Mechanical, Aerospace, and Nuclear Engineering, Rensselaer Polytechnic Institute, Troy, NY 12180, United States
[c]Department of Mechanical and Aerospace Engineering, University of Central Florida, Orlando, FL 32816, United States
[*]Corresponding author. Email: tomerroz@post.bgu.ac.il

ABSTRACT

In the present work, experimental and numerical studies were performed in order to investigate the cooling performance of a single-phase flow in micro-channel/slot-jet system. A three-dimensional numerical model of jet cooling was developed and implemented using commercial software ANSYS Fluent. The 3D conjugate conduction/convection heat transfer in the micro-channel simulations were used to complement experiments and to obtain detailed flow patterns of the jet, temperature, and heat flux distribution on the heater area, and fluid temperature distribution. The model has been verified in a preliminary study where its time-step and grid independency was established and validated vs. experiments.

INTRODUCTION

Thermal management of electronic devices has been a topic of growing interest as heat dissipation rates in processors and power electronics continue to increase while electronic devices become smaller and the design of effective cooling methods is becoming increasingly difficult. Impinging jets is a very effective cooling method that has been studied extensively at conventional scales (Zuckerman and Lior 2006) and more recently at the micro scale, showing very high heat transfer coefficients (Browne et al. 2010). In micro domains, the channel typically serve as a confinement to the jet, which may have beneficial effects on the flow field and heat transfer.

One of the first studies on the subject was by Zhuang et al. (1997), who investigated experimentally local heat transfer with liquid impingement flow in two-dimensional micro-channels. Experiment results indicated significant enhancement of convective heat transfer in Reynolds numbers between 70 and 4807. Sung and Mudawar (2008) investigated both experimentally and numerically a heat sink that combined a micro-channel and jets. The numerical model was developed for a micro-channel with an array of fourteen circular jets. The results indicated that by reducing the channel height, the impinging effect was stronger on the surface, resulting in a lower maximum surface temperature, but accompanying by more appreciable temperature gradients. A more recent work by Kim et al. (2015) studied numerically the effects of slot-jet length on the cooling performance of hybrid micro-channel/slot-jet module.

In the current work, a 3D numerical model was developed to solve a conjugate heat transfer in a micro-channel with a single phase flow through a long slot-jet. Simulations were used to complement experiments by obtaining detailed flow patterns of the jet, temperature and heat flux distributions on the heated impingement surface and heat transfer coefficient estimations.

MODELING

The experimental device was made of a 1-mm thick Pyrex and 400-µm thick silicon wafers. On the Pyrex wafer, four 100-nm thick resistance temperature detector (*RTD*) and a 1.5-mm × 0.4-mm heater were fabricated from titanium directly under the slot-jet. Jet orifice was etched by deep reactive ion etching (*DRIE*) on a silicon wafer, which was attached to the Pyrex wafer through a vinyl sticker (210-µm thick). A 1.9-mm × 14.8-mm × 210-µm micro-channel was formed by laser drilling into the sticker. The fluid enters through the nozzle inlet and impinges on the heated micro-channel lower wall, which was coated by a 2-µm thick silicon-oxide insulation layer, and exits through the two outlets located at both ends of the micro-channel. Experiments were performed for varying flow rates of Propylene glycol solution (40%) and different heat fluxes, where temperatures of the four *RTD*s were collected in order to study the impinging jet cooling characteristics.

A three-dimensional numerical model of jet cooling was developed and implemented in this work using commercial software ANSYS Fluent. The model was developed according to the experimental micro-device dimensions, materials, and conditions. The micro-channel, nozzle, heater and the Pyrex substrate have the same dimensions as in the micro-device except for the micro-channel length and outlets. The micro-channel dimensions were 1.9 mm in width, 12 mm in length, and 210 µm height. The model did not include the micro-device circular outlets, therefore, the micro-channel length in the model was shorter and the outlets were located at its both ends. A 1-mm thick Pyrex substrate was attached to the lower side of the channel, and on top, a Titanium heater was modeled using thermal boundary condition of heat generation with dimensions exactly duplicating the experimental device. A 2-µm thick silicon-oxide insulating layer coated the entire upper surface of the Pyrex substrate including the heater and the temperature sensors, and served as the channel lower wall interfacing the impinging jet. Figure 1a presents the schematics of the entire model including the locations of inlet (coloured in green) and outlets (coloured in red). Figure 1b shows a magnification of the slot-jet, having dimensions of 1.485 mm in length and 57 µm in width in its top cross-section and 1.47 mm in length and 34 µm in width in its bottom cross-section.

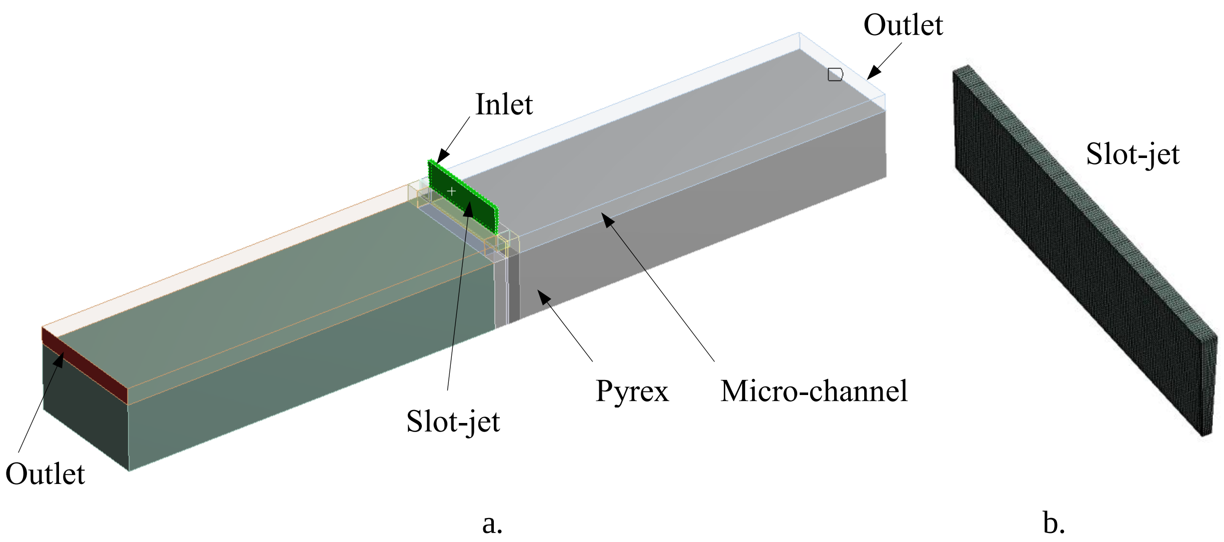

Figure 1. Schematics of the model (a) and the slot-jet (b).

RESULTS AND DISCUSSION

Figure 2 shows temperature contours on the heater surface for Re_j = 232 and power input of 0.6 W, which is equivalent to a nominal heat flux of 100 W/cm^2. The black squares represent the locations of the *RTD*s and the black and blue numbers are the experimentally measured and numerically predicted temperatures at those locations, respectively. A good agreement was found between the measured and numerically predicted temperatures with deviations of less than two degrees in most cases. The pressure drop recorded for the presented case was 196 kPa. Figure 3 shows contours of temperature and heat flux on the impingement surface, i.e., on top of the 2-μm silicon-oxide layer.

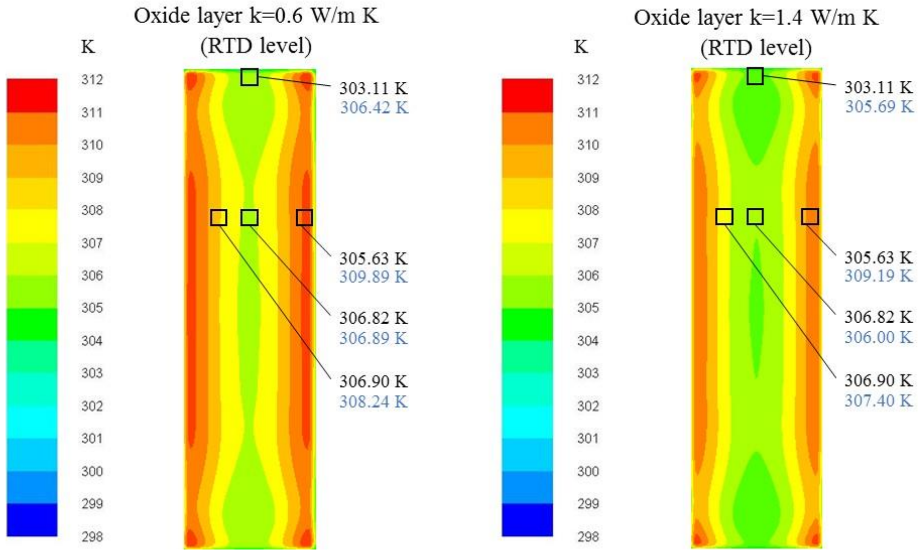

Figure 2. Typical contours of temperature on the heater surface for Re_j = 232 and heat flux of 100 W/cm^2.

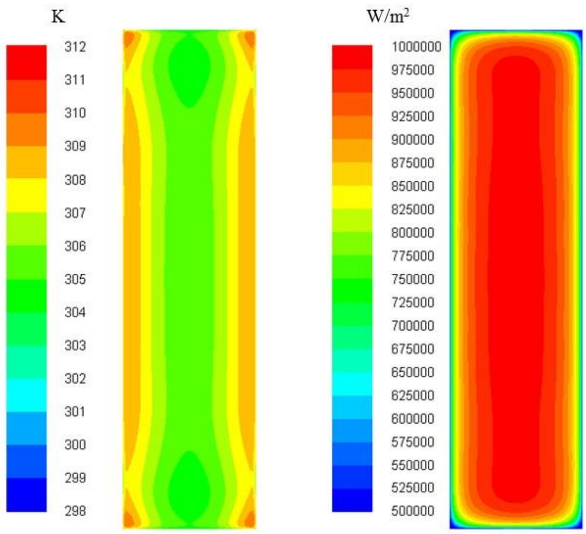

Figure 3. Typical contours of temperature on the heater surface with oxide layer k=0.6 W/m K for Re_j = 232 and heat flux of 100 W/cm^2.

Figure 4 shows temperatures along a line containing three *RTD*s as indicated in the schematic by a red dashed line. These results are for the same case as presented in Figures 2-3. Two different thermal conductivities of the silicon-oxide insulating layer were studied, namely 0.6 and 1.4 W/m K. The black triangles indicate the experimental results and the blue and orange numbers indicate the numerically obtained averaged heat transfer coefficient for the two different thermal conductivities. The black H-shaped symbols attached to the experimental results indicate the *RTD* dimension (in the horizontal direction) and the measurement uncertainty (in the vertical direction).

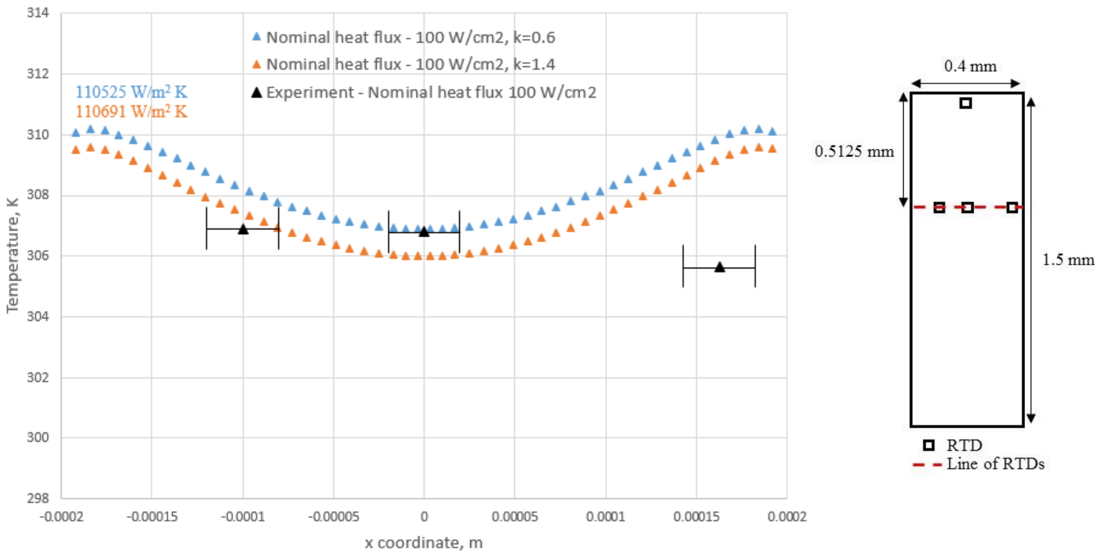

Figure 4. Temperature along line of *RTD*s for Re_j = 232 and heat flux of 100 W/cm^2.

CONCLUSION

The current work presented a 3D numerical model of single-phase flow in a micro-channel with a slot-jet. The model was developed and implemented using a commercial software ANSYS Fluent and aimed at solving a 3D conjugate heat transfer problem in the micro-channel. The numerical results obtained in this study were compared with experimental results, demonstrating a good agreement, and also allowed for further investigation of the flow and thermal characteristics of the suggested system.

ACKNOWLEDGEMENT

This work was partially sponsored by the Office of Naval Research, Award No. N62909-15-1-2025, under DARPA BAA 14-42.

REFERENCES

Shin, J.H., Rozenfeld, T., Vutha, A., Wang, Y., Ziskind, G. and Peles, Y. (2016), Local heat transfer coefficients measurement under micro jet impinging using Nitrogen gas (N_2), *Proceedings of the ASME 2016 Summer Heat Transfer Conference*, July 10-14, Washington, DC, USA.

Zuckerman N., and Lior N., “Jet impingement heat transfer: physics, correlations, and numerical modeling,” Advances in Heat Transfer, 39, 2006.

Browne E.A., Michna G.J., Jensen M.K., and Peles Y., "Experimental investigation of single-phase microjet array heat transfer," Journal of Heat Transfer, 132 (4), 041013, 2010.

Zhuang Y., Ma C.F., and Qin M., "Experimental study on local heat transfer with liquid impingement flow in two-dimensional micro-channels," International Journal of Heat and Mass Transfer, Vol. 40.17, pp. 4055-4059, 1997.

Sung M.K., and Mudawar I., "Single-phase hybrid micro-channel/micro-jet impingement cooling," International Journal of Heat and Mass Transfer, 51, pp. 4342–4352, 2008.

Kim C.B., Leng C., Wang X.D., Wang T.H., and Yan W.M., “Effects of slot-jet length on the cooling performance of hybrid microchannel/slot-jet module,” International Journal of Heat and Mass Transfer, 89, pp. 838–845, 2015.