139

MULTI-PHYSICS MODELING OF SILICON-BASED MICRO-GROOVED HEAT PIPE

Serdar Taze[1], Barbaros Çetin[1]*, Zafer Dursunkaya[2]
[1]Mechanical Engineering Department, Microfluidics & Lab-on-a-chip Research Group
İhsan Doğramacı Bilkent University, 06800 Ankara, Turkey
[2]Dept. Mechanical Engineering, Middle East Technical University, 06800 Ankara Turkey
*Corresponding author: Fax: +90-312-266-4126, Email: barbaros.cetin@bilkent.edu.tr
Keywords: Micro-grooved, heat pipe, multi-physics modeling

ABSTRACT
Heat pipes have the advantage of transferring large amounts of heat between reservoirs with small temperature differences which makes them preferable for electronics cooling applications. Micro-grooved heat pipes promise the additional advantage of being adaptable to systems which need to be cooled with minimal contact resistance. In this study, a multi-physics computational model is developed to assess the thermal performance of a silicon-based micro-grooved heat pipe. The microfluidic platform consists of 50 rectangular micro-grooves on a silicon wafer with integrated chromium (Cr) microheaters. The phase change occurring within the micro-groove is included in the model as a convective heat transfer boundary condition at the channel wall. The convective heat transfer coefficients are obtained from another detailed study in which the 3-D heat transfer process in the solid and working fluid coupled with a 1-D analysis of momentum equation in a unit cell is solved. With the current computational model, an experimental set-up is designed, which will be used as the verification tool of the detailed computational model for a unit cell. The required heating and cooling conditions of the unit cell model can be realized with the proper design of the heater and the cooling channels. The present study demonstrates that the thermal performance of MHP can be predicted with global multi-physics modeling integrated with rigorous phase change model based on a unit cell.

NOMENCLATURE

d_s	thickness of the Cr layer
$\mathbf{J}$	current density
h	heat transfer coefficient (W/m^2·K)
k	thermal conductivity (W/m·K)
k_s	thermal conductivity of the Cr (W/m·K)
Q_{evap}	heat transfer at the evaporator region (W)
Q_{cond}	heat transfer at the condenser region (W)
Q_{heater}	heat transfer at the heating region (W)
Q_{cooler}	heat transfer at the cooling region (W)
Q_{cooler}	heat transfer to the surroundings (W)
q_{Joule}	heat generation due to Joule heating (W/m^3)
V	electrical potential
T	temperature (K)
T_∞	ambient temperature (K)
σ	electrical conductivity (S/m)
∇_t	gradient in tangential direction

INTRODUCTION
Considering the advances in integrated circuits (IC), new technological devices such as computers, smart phones and other applications are very promising for advancing the human standard of living. The main reason behind this improvement is the development in the microchip and IC technologies. The capabilities and performance of the microchips increased dramatically in recent decades and these improvements help produce multipurpose small scaled electrical devices for daily usage. Although these technological developments are scaling down the chips and increasing their

performance, there are limiting factors for further improvements, such as fabrication difficulties and high temperature rise in the microchips that can cause malfunction and a reduction in operational life. Insufficient cooling of microchips brings economical disadvantages compared to properly cooled microchips because overheating reduces the life of the system and increases the initial cost. Therefore, in literature alternative cooling methods are proposed by many researchers such as heat pipes, thermoelectric cooling, direct immersion pool boiling, spray cooling (liquid jet impingement), and forced convective boiling [1]. Although many of the listed cooling methods show good heat removal capability, the heat pipe has unique advantages such as simplicity, no additional component requirements, small amount of working fluid and high heat flux removal capacity. After many years of proposition of heat pipe, the idea of micro heat pipe (MHP) was first proposed by Cotter [2] in 1984, and widely studied to understand the performance parameters and improve the performance. Recent studies are focused on the mathematical modelling and experimental performance of micro grooved heat pipes. It is found out that the mathematical models and experimental results show good agreement [3]. In addition to that finite element analysis are conducted in MHP and Rahmat and Hubert [4] introduced FE simulations with ANSYS CFX-5.7.1 on a single triangular MHP to experience the two-phase flow (condensation and evaporation) and to determine the pressure gradient and velocity field. The observed results from the simulations were compared with those available in the literature and are in acceptable concurrence.

Although computational modelling has advanced over the decades, the modelling of phase change is still a challenging problem. In the assessment of the thermal performance of a heat pipe, modelling of the phase change is a critical step. In this study, a multi-physics computational model is developed to assess the thermal performance of a silicon-based micro-grooved heat pipe. The phase change occurring within the micro-groove is included in the model as a convective heat transfer boundary condition at the channel wall. The convective heat transfer coefficients for the phase change regions are obtained from another detailed study in which the 3-D heat transfer process in the solid and working fluid coupled with a 1-D analysis of momentum equation in a unit cell are solved.

HEAT PIPE SPECIFICATIONS

The heat pipe is placed on a standard silicon substrate which is 525 μm thick and has a width and length of 40 mm and 90 mm, respectively. The system consists of silicon wafer, cooling channels embedded in PDMS, deposited Cr heaters and fused silica cover at the top. The Cr heaters serve as the evaporator, cooling channels serve as the condenser. The micro-grooves are on the silicon wafer. The transparency of fused silica will enable the visual inspection of the working fluid. The PDMS separator is also located between the silicon wafer and the fused silica cover that is chosen to attach the fused silica with silicon wafer, and form a space for the circulation of evaporated working fluid. For the simulations on the MHP, the system on which the thermal analysis is carried out consists of the heat pipe section, the Cr heater and the cooling section placed on the silicon substrate, as shown in Figure 1. The heat pipe channels are $200\,\mu\text{m} \times 200\,\mu\text{m} \times 75\,\text{mm}$ in size and 50 channels are present on the center of the silicon substrate, resulting in a total width of 20 mm. The lengths of the evaporator and condenser sections are $L_e = 45.75\,\text{mm}$ and $L_c = 29.25\,\text{mm}$, respectively. The cooling section is $L_{cooler} = 30\,\text{mm}$ in length and 20 mm in width. Cr heater is located at the bottom of the silicon substrate, and it has a serpentine geometry with 27 turn as shown in Figure 1. The thickness and width of the coated Cr is selected as 800 nm and 500 μm, respectively. The cooling channels embedded in a PDMS are located at the bottom of the silicon wafer as shown in Figure 1. Cooling water circulates inside the microchannels in the PDMS for heat removal. The cross-section of the cooling channels is $2\,\text{mm}\times 2\,\text{mm}$ and the total length is approximately 0.19 m. To obtain the required cooling capacity two sets of cooling channels are placed inside the PDMS. The fused silica is included in the simulations and it has standard dimensions of 500 μm thickness and width of 40 mm and length of 90 mm.

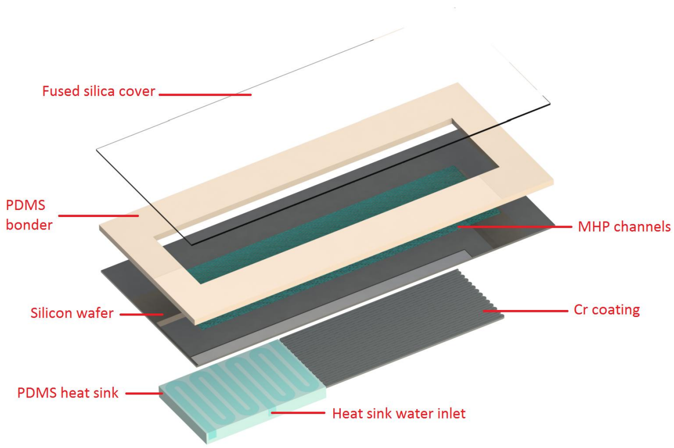

Figure 1
Computational domain

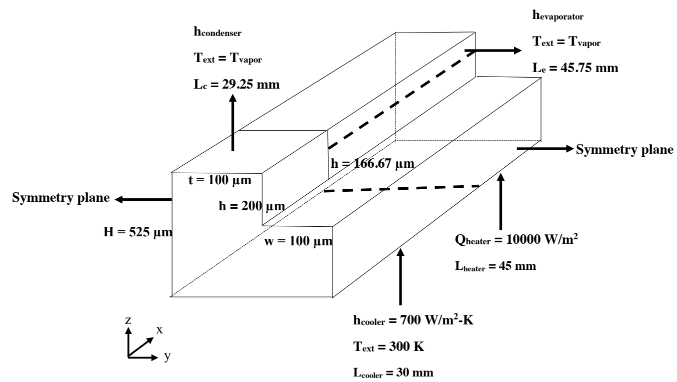

Figure 2
Computational domain for UCM

ANALYSIS AND MODELING

The 3-D multi-physics model contains different physical phenomena including heat transfer in solids and electrical resistive heating (Joule heating). The electrical field on the Cr heaters are modeled using the `Electric Current Shell Module` of the COMSOL. First step is the solution of the current conservation is solved on the shell with Eq. (1) where **J** represents current density.

$$\nabla \cdot \mathbf{J} = 0 \tag{1}$$

Following this, the generated heat power per unit area on the Cr coating by the potential difference is determined by:

$$q_{Joule} = d_s \sigma \left| \nabla_t V \right|^2 \tag{2}$$

The Joule heating power is linked to the heat transfer in solids as a highly conductive layer boundary that makes the interconnection between the two physical phenomena. The heat transfer equation on the Cr coating can be written as:

$$\mathbf{n} \cdot (k \nabla T) = q_{Joule} + h(T_\infty - T) - \nabla_t \cdot (-d_s k_s \nabla_t T) \tag{3}$$

The first term is the heat flux through the body from the Cr thin film where **n** is the boundary normal vector and k is thermal conductivity. The third term represents convective heat flux that is transferred to the environment where h is heat transfer coefficient and T_∞ is the ambient temperature. The last term is heat conduction within the thin Cr layer.

The heat transfer in the solid region (silicon, PDMS, fused silica) is governed by the heat conduction equation:

$$\nabla \cdot (k_i \nabla T_i) = 0 \tag{4}$$

where k_i represent the thermal conductivity of different regions. At the outer surface of the MHP, to resemble the natural convection, heat transfer coefficient of $5\,\mathrm{W/m^2K}$ is assigned:

$$q = h(T_\infty - T) \tag{5}$$

The cooling of the MHP is performed by the heat sink in which cooling water flows through the square channels. The heat sink is implemented to the simulations by applying the convective heat transfer boundary condition at the walls of the cooling channels. The necessary heat transfer coefficients are determined using the appropriate fully developed Nusselt number value for constant wall temperature. Temperature of the flowing water which is the ambient temperature for the convective heat transfer occurring at the channel wall is taken as 300 K. The mass flow rate of the water is chosen such that the temperature at the inlet and exit of the cooling channel is 1 K which ensures constant ambient temperature.

For the convective heat transfer coefficients at the evaporator and condenser regions, a rigorous, detailed computational model on a unit cell (which will be called as Unit-Cell-Model (UCM) hereafter) was completed [5]. The unit cell used in the detailed model can be seen in Figure 2. In this model, 3-D heat transfer process in the solid and working fluid coupled with a 1-D analysis of momentum equation in a unit cell is addressed. Heat transfer coefficient data obtained from the model are implemented into the simulations of the entire MHP as convective heat transfer boundary condition. Figure 3 presents the phase change heat transfer coefficients and temperature distribution along the unit cell where the center of the MHP channels are located at the origin of the axial direction to handle the geometry easily. The same geometry is likewise used in the 3-D simulations.

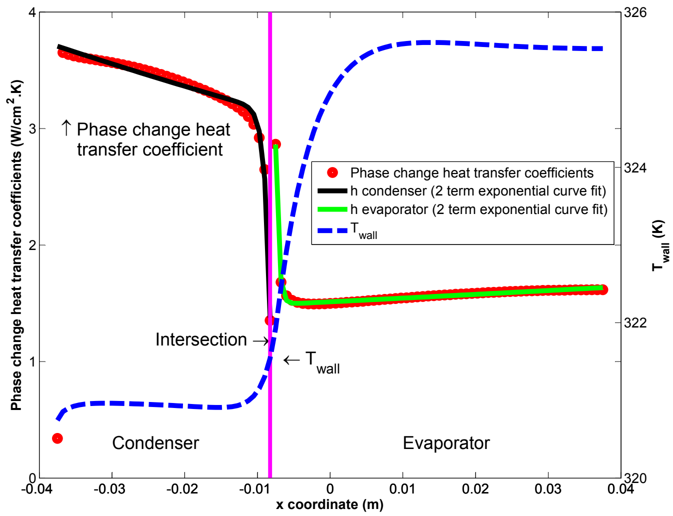

Figure 3
Phase change heat transfer coefficients and temperature distribution along a micro-groove

The global multi-physics computations are performed to decide on the design of the Cr heater and cooling channel geometry, and to assess the 3-D heat transfer effects when 50 channels are present. With the proper design of the MHP, an experimental set-up will be constructed to verify the results of the UCM. Therefore, it is important to realize the boundary conditions in the model on the evaporator and condenser regions. The boundary conditions for the evaporator and condenser region in UCM were constant wall heat flux and convective heat transfer with constant ambient temperature, respectively. With the proposed multi-physics model, a heater and cooling channel geometry are designed.

The physics used in the simulations are `Heat Transfer in Solids` and `Electrical Currents, Shell`. The simulations are realized on a HP Z820 Workstation (Intel Xeon E5-2630 v2, 6 cores, 2.60GHz, 128GB RAM). The boundary conditions are convective at the evaporator, condenser and cooler sections, where heat transfer coefficients are specified. The external temperature for the convective heat transfer is taken as the vapor temperature which corresponds to the saturation temperature at a given pressure inside the MHP. The electrical potential on the Cr heater is introduced into the relevant physics. The phase change heat transfer coefficient data taken from UCM are implemented into heat transfer coefficient boundary condition of evaporator and condenser sections by an exponential curve fit. The results from the UCM together with the fitted curves along the axial direction are shown in Figure 3. The condenser heat transfer coefficient is introduced to the top surface of the MHP channels due to the fact that the condensation of working fluid generally takes place at the top of the channels. The evaporator heat transfer coefficient is implemented on side walls of the MHP, since most of the evaporation of the working fluid occurs on the side walls. Moreover, the heat transfer on the sections that are in direct contact with evaporated working fluid for the fused silica and PDMS bonder are simulated by defining a heat transfer coefficient which is determined using the Nusselt number correlations in accordance with the velocity and temperature of the vapor used in UCM. The material properties for silicon wafer, PDMS, fused silica, and chromium are already defined in the simulation environment except the thermal conductivity of silicon wafer, Poisson's ratio and relative permittivity of chrome which are specified as 130, 0.21 and 1.0, respectively. The thickness of the Cr layer is 800 nm in the shell module. At the outer surface of the MHP, convective heat transfer coefficient of $5\,\mathrm{W/m^2K}$ – which is typical for natural convection – is applied. The mesh dependency is also checked, and it is found that total of 18 million DOF generates mesh independent results. A typical run takes approximately 45 minutes of CPU with a 48 GB of physical memory usage.

RESULTS AND DISCUSSION

Selection of Cr heater geometry: UCM uses

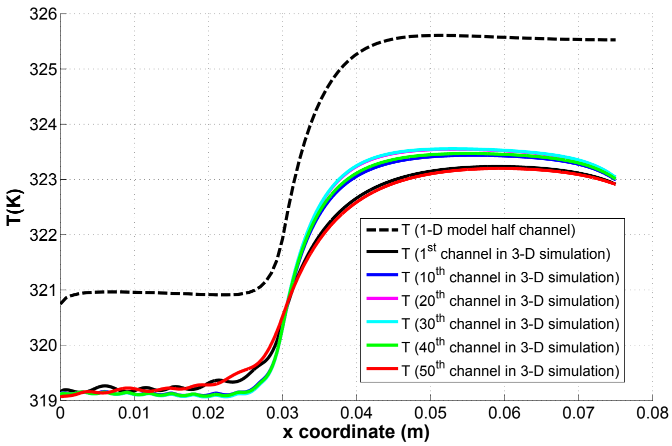

Figure 4
The temperature distribution along the micro-grooves

Table 1
Heat transfer data at different regions

	UCM (T_{vapor} = 321.8 K)	Global model (T_{vapor} = 319.9 K)
Q_{evap}	8.11 W	7.48 W
Q_{cond}	8.10 W	7.47 W
Q_{heater}	9.0 W	9.0 W
Q_{cooler}	8.98 W	8.29 W
Q_{loss}	——	0.69 W

uniform heat flux at the heater section; therefore, in order to render a uniform heat flux, different heater configurations are analyzed. The simulations start with a single-piece Cr heater geometry with various corner length and the result of the analysis showed that the uniform heat flux as a result uniform temperature profile is difficult to obtain with this geometry; therefore, a serpentine geometry is proposed and approximate uniform surface temperature distributions are obtained. In the light of these findings, the serpentine Cr heater geometry with 84 V electrical potential difference is selected for MHP simulations to obtain the required heater power.

Cooling unit design: For the cooling unit, water is selected as the working fluid. To simulate the boundary conditions in UCM, cooling channels having a square cross section with 2 mm width and 1 mm spacing between channels are used. Two parts of cooling section are used that have a total of 8 turns with a total length of 0.19 m. The cooling channels are separated to obtain a uniform heat removal rate in the cooling channel and equally distribute the mass flow rate to each channel. The volumetric flow rate of the cooling water is found to be 32.4 mL/min for each channel for the required cooling capacity.

Temperature distribution: The temperature distribution on the MHP system is generated and analyzed to observe the matching between the finite element simulation results with the UCM simulations. The temperature profile along the top corner of the 1^{st}, 10^{th}, 20^{th}, 30^{th}, 40^{th} and 50^{th} MHP channels together with 1-D model are presented in Figure 4. The wall temperature in UCM is between 320.7 K and 325.8 K, whereas in MHP simulations, the wall temperature changes between 323.6 K and 318.9 K. The reason behind this difference is that the UCM model is well insulated; however, in 3-D global model, the effect of natural convection and heat transfer in MHP chamber which is fused silica surface and PDMS bonder surface that are contact with working vapor, are taken into account. Additionally, in the global model, the 3-D effects mainly due to the conduction heat transfer through the Si wafer have a significant influence on the heat transfer mechanism. The oscillations in temperatures of the condenser section is clearly seen in Figure 4 that is because the cooling system has a serpentine geometry and it has 1 mm space between channel walls. However, the temperature change is not critical and serpentine channels are a more suitable way to transfer heat uniformly from the cooler section. In temperature profiles on the MHP channels, except 1^{st} and 50^{th} channel located at the corners, the wall temperatures show the same trend as seen in the temperature profile in the 10^{th}, 20^{th}, 30^{th} and 40^{th} channels. 1^{st} and 50^{th} channel temperatures show slightly different profiles especially in the evaporator region due to heat transfer from Cr heater to Si wafer. The serpentine geometry of the Cr heater has turning corners at the beginning and end of the evaporator section that is why temperature decrease is observed at that locations.

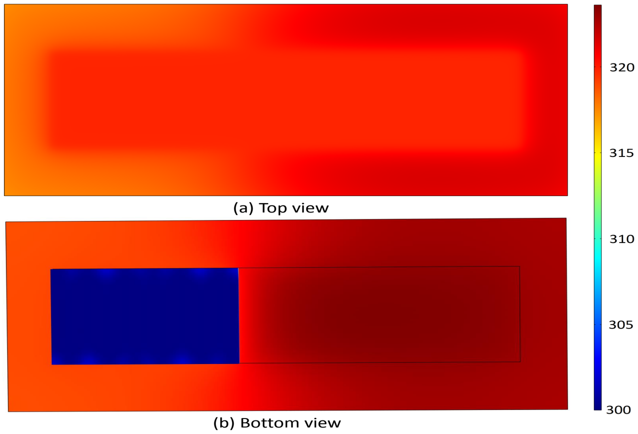

Figure 5
The temperature distribution in MHP

The heat transfer rates obtained in the global and UCM are tabulated in Table 1. It can be realized that heat transfer rates are closer to each other at condenser and evaporator sections in UCM where the vapor temperature is 321.8 K. When the 3-D simulation applied for this vapor temperature, it is observed that the heat transfer rates in evaporator and condenser sections are equal and deviate from the results of UCM due to 3-D heat conduction effects in MHP. Therefore, the equivalence of condenser and evaporator heat rates are obtained with changing vapor temperature inside the system which is determined as 319.9 K. For that case the condenser and evaporator heat rates are 7.48 W and 7.47 W and the error between two values is found to be less than 1%. The temperature profile on the top and the bottom surface of the MHP can be seen in Figure 5. Figure 5–(a) shows the temperature distribution on top of the MHP. The condenser and evaporator sections are distinguishable. The temperature on the heat sink and Cr heater can be seen in Figure 5–(b) where blue section is the PDMS heat sink and red section is the Cr heater.

CONCLUSIONS

In this study, a global multi-physics model is present to predict thermal performance of a MHP. The results of a detailed UCM is introduced into the global model as a boundary condition. An electrical heater and cooling channel section is designed to realize the boundary conditions used in UCM. The global computational model guide to decide the design parameters for the MHP system together with the temperature distribution on the complete system for the experimental setup. The fabrication and the experimentation of the MHP together with the electrical heaters and the cooling channels will be our future research direction.

ACKNOWLEDGMENTS

This study was supported by Turkish Scientific and Technical Research Council, Grant No. 213M351.

REFERENCES

1. L. Zhang, K. E. G., and Kenny, T. W., 2004. *Silicon Microchannel Heat Sinks Theories and Phenomena*. Springer-Verlag Berlin Heidelberg, pp. 1–11.

2. Cotter, T. P., 1984. “Principles and prospects of micro heat pipes”. In Proceedings of the 5th Int. Heat Pipe Conference, Tsukuba, JAPAN, pp. 328–332.

3. Lefèvre, F., Rullière, R., Pandraud, G., and Lallemand, M., 2008. “Prediction of the temperature field in flat plate heat pipes with micro-grooves - experimental validation”. *International Journal of Heat and Mass Transfer,* **51**(15–16), pp. 4083–4094.

4. Rahmat, M., and Hubert, P., 2010. “Two-phase simulations of micro heat pipes”. *Computers & Fluids,* **39**(3), pp. 451–460.

5. Odabasi, G., 2014. “Modeling of multidimensional heat transfer in a rectangular grooved heat pipe”. PhD thesis, Middle East Technical University, Turkey, June.