

A CFD Simulation Study on Pressure Drop and Velocity across Single Flow Microchannel Heat Sink

A. A. Razali^{*,a} and A. Sadikin^b

Faculty of Mechanical Engineering and Manufacturing, Universiti Tun Hussein Onn
Malaysia, 86400 Parit Raja, Malaysia

^{a,*}gd140100@siswa.uthm.edu.my, ^bazmah@uthm.edu.my

Abstract – This paper presents a numerical simulation of flow in a microchannel heat sink. The channel was defined as a dimension with less than 1.0 millimeter and greater than 100.0 micrometer. The ANSYS CFX 2015 was used to predict the flow in the microchannel. Besides, simulations were undertaken to determine the flow of the fluid within the microchannel in three different models. Therefore, three different models were employed for this study. The first model was a square-shaped channel with 0.5mm width and 0.5mm height constructed along 28.0mm channel length. The second and the third models were in rectangular shape. The differences between these models were their width and height of channel. The dimension for the second model was 0.75mm height and 0.5mm width (rectangular A), while the dimension for the third model was 0.5mm height and 0.75mm width (rectangular B). All the microchannel heat sink models had been simulated and showed results for pressure, temperature, and velocity inside the microchannel. The results were compared for each model and the data had been validated from published data. In addition, the initial velocity was set in a range between 0.1m/s and 0.5m/s. The highest pressure drop was recorded for the square microchannel. It was 58.12% higher than the pressure drop found in the rectangular microchannel with 0.75mm width, while 0.02% closer with the 0.75mm-height rectangular microchannel. Furthermore, there was a 33.34% of temperature difference, which had been higher for the square microchannel. Nevertheless, the highest velocity of 0.57m/s was recorded at the outlet of the microchannel. These had been consistent with other published data. **Copyright © 2015 Penerbit Akademia Baru - All rights reserved.**

Keywords: Microchannel, Pressure drop, Temperature difference

1.0 INTRODUCTION

Microchannels can be defined as channels with dimensions less than 1.0 mm and greater than 100.0 micron [1]. Besides, microchannels possess the very potential of wide applications in cooling high power density microchips in CPU system, micro power systems, and every many other large scale thermal systems requiring effective cooling capacity. This is a result of the micro size of the cooling system, which does not only significantly reduce the weight load, but also enhances the capability to remove greater amount of heat than any large scale cooling systems. As the size of the channel is reduced to micron (μm), the heat transfer coefficient can increase drastically from the original value [2]. Based on previous studies, the advantages offered by the microchannel have been determined due to their high surface-to-volume ratio,

as well as their small volumes. Other than that, in a microchannel heat sink, the channel (which is in micron size) is stacked together to increase the total surface area for heat transfer enhancement and to reduce pressure drop. Commonly, water or gas is chosen as coolant. The small channel will change the temperature of the fluid that flows across the microchannel.

Nonetheless, the major problem derived from past studies had been the difficulty in measuring the accuracy of pressure inside the channel. Therefore, almost all information pertaining to pressure has been based on the pressure measured at inlet and outside the channel, which is used to reduce the shear stress on the wall. A computational fluid dynamics (CFD) analysis constitutes a new “third approach” in a philosophical study and analysis fluid flow in microchannel heat sink [3]. In addition, the simulation can analyse fluid flow instead of pressure, temperature, and velocity across the microchannel heat sink.

Heat transfer in microchannel has been studied and compared with other researches. The purpose of developing this channel is for use in fluid control and heat transfer. Microchannels are very fine channels with width of normal human hair and are widely used for electronic cooling purpose [4]. In fact, many researchers have started this investigation with different methods, but with a common purpose. Moreover, many designs of microchannels have been studied, such as rectangular, trapezoidal, triangular, and circular shapes. Most of the previous studies used the rectangular-shaped microchannel because it is easier to fabricate. Other than that, rectangular cross-section shape is more beneficial for heat transfer [5]. Besides, there are varying values of pressure inside the microchannel. It is obviously clear that the high pressure region occurs at the entrance, while the low pressure region occurs at the outlet of the heat sink. This is due to the needs of high pressure to push the fluid flow along the direction of the microchannels out from the outlet plenum of the heat sink [6]. This happened due to change of dimension from the piping system to the microchannel. Since the present pressure measurements were made in the inlet and the outlet of the microchannel, it involved inlet contraction pressure loss and outlet expansion pressure recovery, in addition to pressure drop along the microchannel [7].

Furthermore, microchannel heat sinks dissipate large amounts of heat with relatively little surface temperature rise. To enhance the heat transfer in microchannel heat sink, it is necessary to study the simultaneous effects of various parameters, such as size of channel, shape of channel, fluid properties, pressure drop, and many more. In the microchannel, the local heat transfer coefficient varies with the channel size, cross section of channel, shape of channel, fluid properties, and fluid flow arrangement [8]. In the microchannel heat sink operation, the high temperature region should occur at the edge of the microchannel since there is no heat dissipation by fluid convection. The lower temperature region should occur at the middle regions of microchannel due to high heat transfer. Besides, the liquid flow in the microchannel can be regarded as continuum even in a very small channel. However, the liquid flow can become boiling when the wall temperature is higher than the vaporization temperature of the liquid [9].

From previous study, the smaller size of channels led to higher heat transfer, and thus, higher single-phase heat transfer coefficients were achieved at the entrance region of the microchannels. As the size of the channel reduces to micron or nano size, the heat transfer coefficient can increase thousand or million times the original value. Therefore, in this present study, the objective of this work was to identify the best shape for the channel to be applied in the microchannel heat sink based on pressure drop, temperature difference, and velocity. The simulation was performed with different velocity values, starting from 0.1m/s to 0.5m/s.

Besides, three different models were tested to measure pressure and temperature difference in the microchannel. The results of the simulation were compared with those obtained in past studies.

2.0 METHODOLOGY

The computational domain of microchannel is represented in three dimensional (3D). The microchannel heat sink had been modelled and simulated by using Computational Fluid Dynamics code ANSYS CFX. The channel was designed in three different models by using SolidWorks 2014. The first model was square in shape with a dimension of 0.5mm width and 0.5mm height. The second model was rectangular with a dimension of 0.5mm width and 0.75 mm height, which is also known as rectangular A. The last model was a rectangular with a dimension of 0.75mm width and 0.5mm height, which is known as rectangular B. The models consisted of inlet, microchannel region, fluid region, and outlet. The models are presented in figure 1.

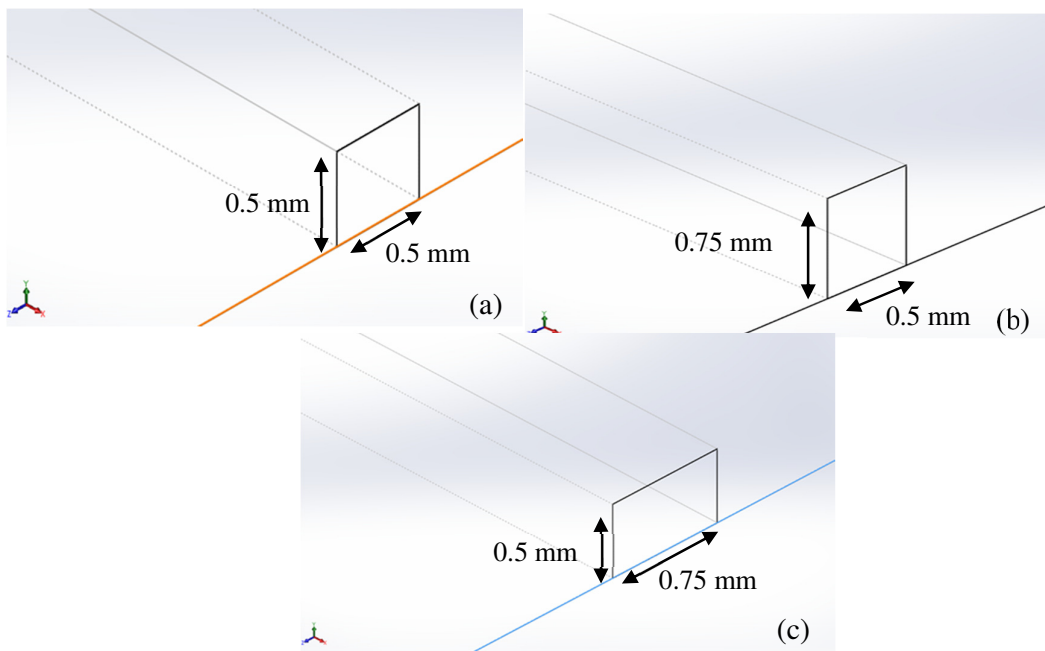


Figure 1: (a) Square shape; (b) Rectangular A (c) Rectangular B

On top of that, in order to obtain accurate results, mesh independence was carried out by varying the cell density of the model for similar boundary conditions. These indicators had been compared between course, medium, and fine meshed using grid independence test. The meshing was done between 100580, 118620, and 113120 elements. The pressure result for mesh level at the second level had been closer to the third level, which was varied by 0.2% only. Hence, a mesh level with 118620 elements was chosen for the rest of the analysis. Moreover, a high resolution scheme was selected as the solving option, while double precision was activated to improve accuracy. Meanwhile, the convergence criterion was set to 10^{-6} for residues. Figure 2 below shows the meshed model.

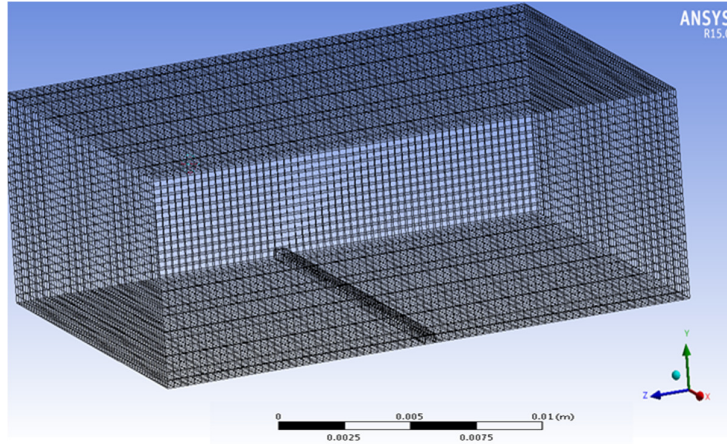


Figure 2: Meshed model

In addition, the models had been set to two domains; solid and fluid domains. At fluid domain, the material used was water, while the thermal energy was set for heat transfer option. Besides, subsonic flow regime was applied at the inlet with normal speed set for 0.1 m/s, 0.2 m/s, 0.3 m/s, 0.4 m/s, and 0.5 m/s. Moreover, a uniform heat flux (100 W/m^2) was applied at the bottom of the heat sink surface. The top of the microchannel heat sink was considered as adiabatic cover, while the top of the fluid domain was set as symmetry.

Besides, copper was used as solid material, whereas water in laminar flow regime was used as coolant fluid. The pressure drop was calculated based on data at inlet and outlet by using equation,

$$\Delta P = P_{in} - P_{outlet} \quad (1)$$

where ΔP is the pressure drop, while P_{in} is pressure inlet, and P_{outlet} is the pressure outlet. Temperature difference was calculated to determine the effect of shape, as well as the dimension of microchannel based on equation,

$$\Delta T = T_{outlet} - T_{inlet} \quad (2)$$

where ΔT is the temperature difference between temperature outlet (T_{outlet}) and temperature inlet (T_{inlet}). At the microchannel sections, the fluid temperature was taken as 300K, while the inlet velocity was calculated by using the following equation,

$$u_{in} = (Re \cdot \mu) / \rho \cdot D_h \quad (3)$$

The u_{in} is defined as inlet velocity; Re is Reynold number, μ is viscosity, ρ is density, and D_h is hydraulic diameter. In order to determine the cross-section of the microchannel, the hydraulic diameter, D_h , was calculated by using the equation depicted below,

$$D_h = 4 \cdot A = (2H_{ch} \cdot W_{ch} / H_{ch} + W_{ch}) \quad (4)$$

where A is the channel flow area, P is the channel wet perimeter, H_{ch} is the channel height, and W_{ch} is the channel width.

3.0 RESULTS AND DISCUSSION

The simulation was carried out to identify pressure drop, temperature difference, and velocity in the microchannel.

3.1 Validation Data

The experiment data published by [7] had been used for validation of the results obtained in this work. Figure 3 shows pressure drop across microchannel with different Reynold numbers.

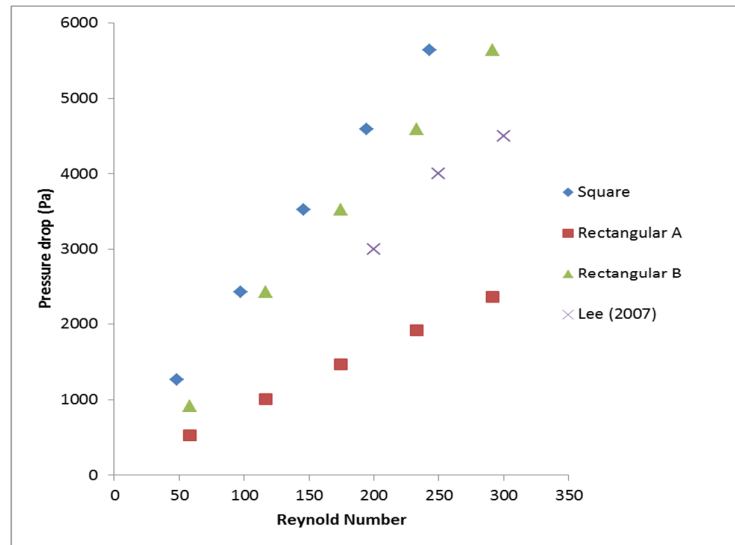


Figure 3: Pressure drop across the microchannel

The figure above shows pressure drop increased with Reynold number. In general, the simulation data are in agreement with the experiment data published. The different values from the published result had been the size of the microchannel used. The dimension of the experimental model was 0.215mm width by 0.821mm height, and in rectangular shape.

3.2 Contour Plot

Meanwhile, the contour plot shows different velocity, pressure, and temperature in microchannel. Figure 4 shows the contour plot for velocity across the microchannel. Figure 4 shows that velocity increased across from inlet to outlet of the microchannel. Since the inlet velocity was set at a low value, the fluid flow increased across along the small channel. However, the velocity changed due to the changes in water density with varying temperatures. Besides, the data recorded different pressure value along the microchannel.

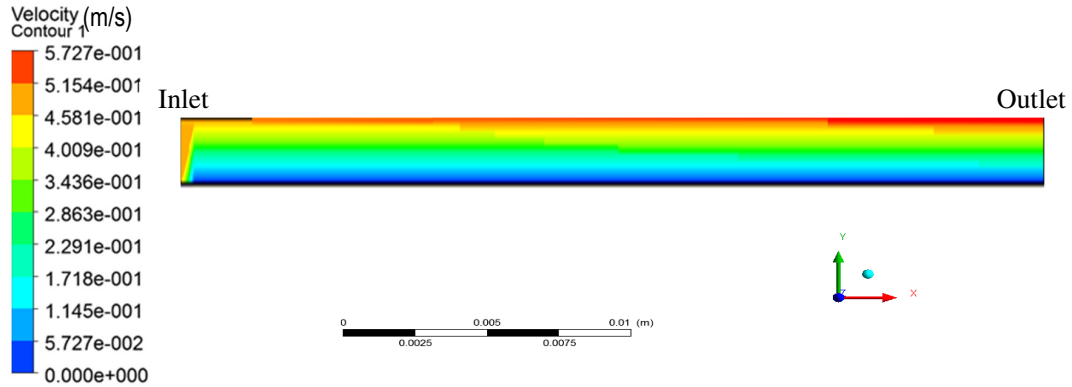


Figure 4: Contour plot for velocity across the microchannel

Figure 5 shows that the pressure decreased at the outlet of the microchannel. Nonetheless, high pressure was recorded at the inlet. The drop in pressure, as well as pressure losses, occurrences had been associated with the abrupt contraction at the channel inlet and the expansion at the channel outlet. This happened due to change of dimension from piping system to microchannel. However, it had been a different case for temperature at the microchannel. Figure 6 shows the contour plot for temperature at the outlet of microchannel.

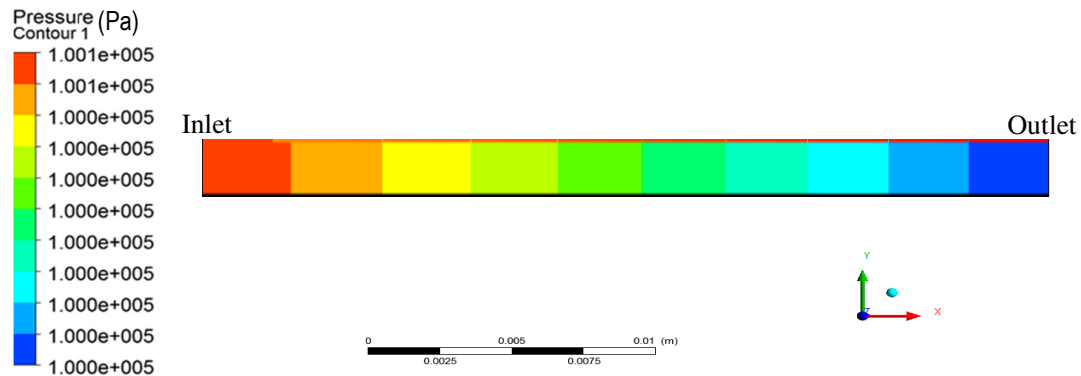


Figure 5: Contour plot for pressure in microchannel

Figure 6 shows that the temperature value was higher at the bottom of the microchannel. The water flow reacted with high temperature produced by heat flux at the bottom surface of the channel. At hot surface inside the channel, liquid was in contact with a thermally-conductive solid surface. However, the temperature decreased along with the height of the channel. This result showed that the micro size of the channel enhanced the capability to emit greater amount of heat in the microchannel heat sink.

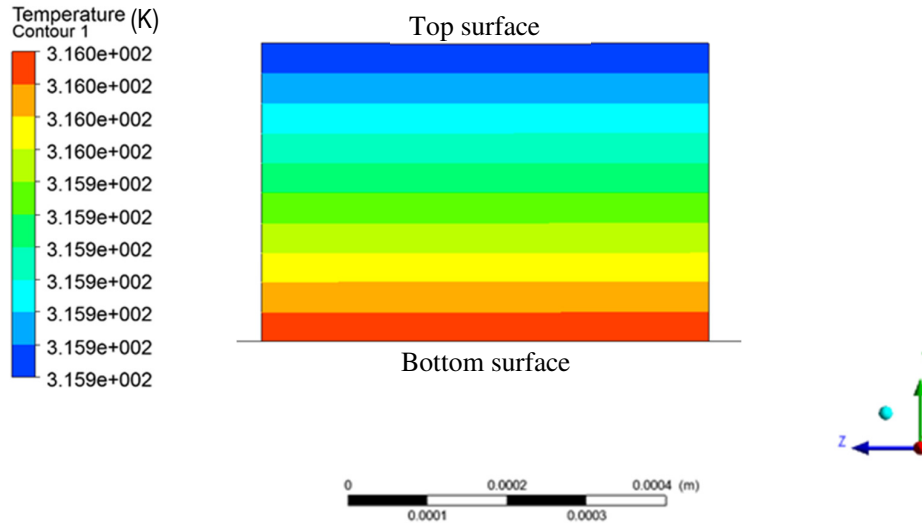


Figure 6: Contour plot for temperature in microchannel

3.3 Pressure Drop

The pressure usually increases at the inlet of microchannel and decreases at the outlet. Figure 7 shows the pressure drop in the microchannel.

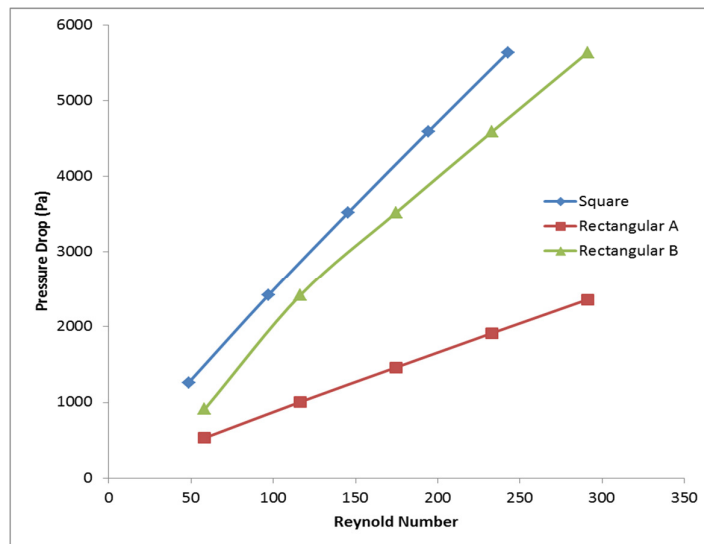


Figure 7: Pressure drop in microchannel

The figure above shows that the pressure drop was increased due to the increased Reynold number. The square-shaped microchannel had the highest value compared to the one with a rectangular shape. The pressure drop value for the rectangular-shaped microchannel with 0.75mm width was 0.02% closer to the one with a square shape. Besides, it had been obvious that higher Reynold number affected the pressure drop. This was because high pressure with high velocity pushed the fluid to flow along the microchannel out from the outlet region.

Besides, when fluid with constant properties flows through the channel, a linear relationship is expected between pressure drop and Reynolds number. A gradual slope change in the pressure drop variation with Reynolds number was attributed to the increasing contraction and expansion pressure losses at the inlet and the outlet of the microchannel.

3.4 Temperature Difference

It had been similar a similar case with temperature difference in microchannel. Figure 8 shows the temperature difference based on velocity inlet.

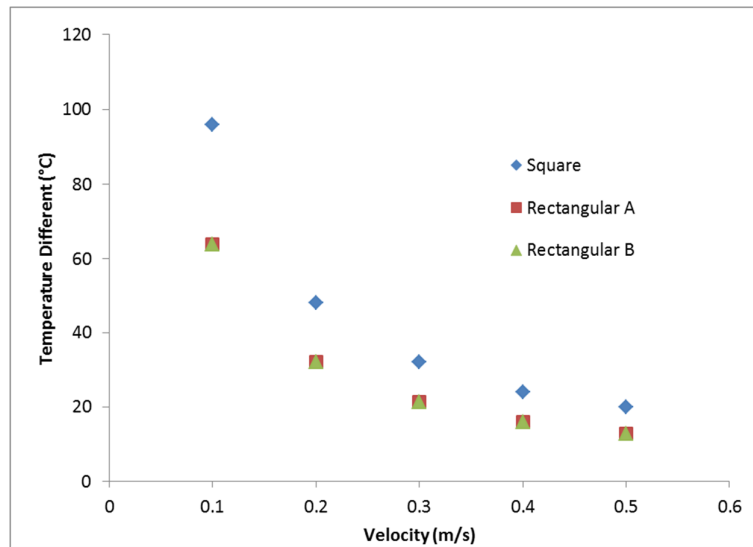


Figure 8: Temperature differences between three different models of microchannel

The figure above shows that the temperature difference decreased when the velocity increased. The square-shaped microchannel had larger value than both the rectangular microchannels. Besides, there was a 33.34% of difference between the square microchannel and both the rectangular-shaped ones. As expected, the heat sink temperature increased along the channel. Moreover, fluid that flowed with low velocity had been linked with high heat flux produced at the bottom surface of the microchannel. This was attributed to the thin thermal boundary layer at the microchannel region. Besides, the varied heat flux surrounded the channel periphery, approaching zero in corners where the flow was weak for rectangular channel.

3.5 Velocity

The velocity across the microchannel had different values based on the size and the dimension of the microchannel. Figure 9 shows the differences of velocity with different models. Figure 9 also shows the velocity of water flow inside the microchannel at different positions. The highest velocity achieved was 0.57 m/s. There was only a rectangular-shaped A with 0.75mm height that had reached the highest velocity at 0.54m/s. This situation shows that fluid flow velocity increased consistently at channel with the same dimension. Since the flow regime was in laminar flow, it had been characterized by the high momentum diffusion and low momentum convection. The height of the channel was limited to 0.5mm for this simulation. On top of that, the inlet position started with low velocity and it increased instead of getting small dimension for fluid flow across the microchannel.

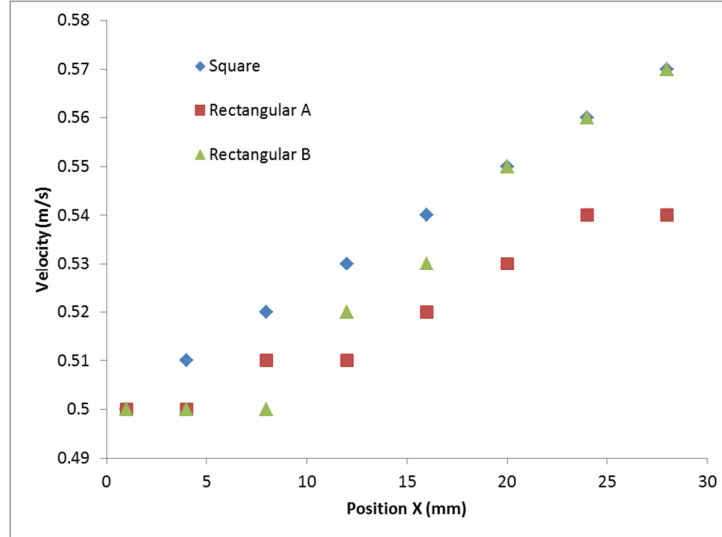


Figure 9: Velocity across microchannels with different models

4.0 CONCLUSION

The simulation results had shown pressure drop, temperature difference, and velocity across the three different models of microchannel heat sinks. The data had been validated with experimental data published. Based on the results, the square-shaped model demonstrated better results than those with rectangular shapes. Besides, the pressure drop results showed that the square-shaped microchannel had good agreement with values rather than rectangular shape. In addition, the velocity inside the square-shaped microchannel also increased more consistently compared to those with rectangular shape. Meanwhile, the single flow of microchannel offered advantages in using square-shaped microchannel based on temperature, pressure, and velocity across the microchannel heat sink. The low velocity had been used from 0.1m/s to 0.5m/s, which had been in laminar form. Most importantly, the findings were found to be in close agreement with the past study [7].

ACKNOWLEDGEMENT

This work had been supported by Research Acculturation Grant Scheme (RAGS) (No. R028).

REFERENCES

- [1] P.R. Parida, Experimental investigation of heat transfer rate in micro-channels, MSc thesis, Indian Institute of Technology Guwahati, India, 2007.

- [2] C.W. Liu, H.S. Ko, C. Gau, Microchannel heat transfer, in: A., Belmiloudi (Ed.), Heat Transfer - Theoretical Analysis, Experimental Investigations and Industrial Systems, InTech, 2007, pp. 77-156.
- [3] A.K. Pandey, A computational fluid dynamics study of fluid flow and heat transfer in a micro channel, MTech thesis, National Institute of Technology Rourkela, India, 2011.
- [4] K.V. Sharp, R.J. Adrian, J.G. Santiago, J. I. Molho, Liquid flow in microchannels, in: M. Gad-el-Hak (Ed.), The MEMS Handbook, vol. I, second edition, CRC Press, Boca Raton, 2006, pp. 10.1-10.45.
- [5] D. Gupta, V. Venkataraman, R. Nimje, CFD & thermal analysis of heat sink and its application in CPU, International Journal of Emerging Technology and Advanced Engineering 4 (2014) 198-202.
- [6] P, Gunnasegaran, N.H. Shuaib, H.A. Mohammed, M.F. Abdul Jalal, E. Sandhita, Heat transfer enhancement in microchannel heat sink using nanofluids, in: L. Hector Juarez (Ed.), Fluid Dynamics, Computational Modeling and Applications, InTech, 2012, pp. 287-326.
- [7] J. Lee, I. Mudawar, Assessment of the effectiveness of nanofluids for single-phase and two-phase heat transfer in micro-channels, International Journal of Heat and Mass Transfer 50 (2007) 452-463.
- [8] D.B.S. Gawali, V.B. Swami, S.D. Thakre, Theoretical and experimental investigation of heat transfer characteristics through a rectangular microchannel heat sink, International Journal of Innovative Research in Science, Engineering and Technology 3 (2014) 15631-15640.
- [9] J. Lee, I. Mudawar, Low-temperature two-phase microchannel cooling for high-heat-flux thermal management of defense electronics, IEEE Transactions on Components and Packaging Technologies 32 (2009) 453-465.