Jurnal Teknologi

Full Paper

Numerical Study of The Effect of Area of Manifold and Inlet/Outlet Flow Arrangement On Flow Distribution in Parallel Rectangular Microchannel Cooling System

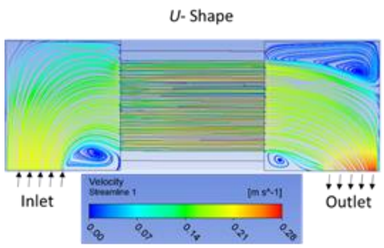
Article history
Received
21 January 2017
Received in revised form
31 May 2017
Accepted
17 August 2017

Amirah M. Sahar[a,b*], A. I. M. Shaiful[c]

[a]College of Engineering, Design and Physical Sciences, Brunel University London, UB8 3PH London, UK
[b]Communication Section, University Kuala Lumpur, British Malaysian Institute, Selangor, 53100 Gombak, Selangor, Malaysia
[c]School of Manufacturing Engineering, Universiti Malaysia Perlis, 01000 Kangar, Perlis, Malaysia

*Corresponding author
mepgabm@brunel.ac.uk

Graphical abstract



U-type Parallel microchannel

Abstract

Parallel microchannels have been widely used in cooling of compact electronic equipment due to large contact area with liquid and availability of large mass of fluid to carry away heat. However, understanding of flow distribution for microchannel parallel system is still unclear and there still lack of studies give a clear pictures to understand the complex flow features which cause the flow maldistribution. Generally, the geometrical structure of the manifold and micro channels play an important role in flow distribution between micro channels, which might affects the heat and mass transfer efficiency, even the performance of micro exchangers. A practical design of exchanger basically involves the selection of an optimized solution, keeping an optimal balance between gain in heat transfer and pressure drop penalty. A parallel microchannels configurations consisting inlet and outlet rectangular manifold were simulated to study flow distribution among the channels were investigated numerically by using Ansys Fluent 14.5. The numerical results was validated using existing experimental data and showed a similar trend with values 1% higher than experimental data. The influence of inlet/outlet manifold area and inlet/outlet arrangement on flow distribution in channels were carried out in this study. Based on the predicted flow non-uniformity value, ϕ, Z- type flow arrangement exhibits higher value of ϕ, which is 8%, followed by U-type, 2.6% and the I-type, 2.49%. Thus, a better uniformity of velocity and temperature distributions can be achieved in I-shape flow arrangement. The behavior of the flow distributions inside channels is due to the vortices that occurred at manifold. Besides comparing the pressure drop for case 1(D1) and case 2(D2), it is worth to mention that, as the area of inlet and outlet manifold decrease by 50%, the pressure drop is increasing about 5%. However, the inlet/outlet area of manifold on velocity and fluid temperature distributions was insignificant

Keywords: Computational fluid dynamics (CFD), single phase, parallel microchannels, flow maldistribution, manifolds

1.0 INTRODUCTION

Parallel microchannel have been widely use into micro heat exchangers applications due to large contact area with liquid and availability of large mass of fluid to carry away heat. However, parallel multichannel system, the biggest bottlenecks that hinder this configurations is to achieve a uniform flow distribution. A non-uniform flow also define as flow maldistribution is one of factor that have a significant effect on exchanger performance Kandlikar *et al*. [1]. Besides, it may cause to non-uniform temperature distribution, hence results in miscalculating the value of pressure drop and heat transfer value. [2] Moreover, the flow maldistribution can also lead to reduce the performance of micro cooling system due to uneven flow among parallel channel. [3]

There is many studies have been conducted to understand the behavior of flow distribution in multichannel configurations in order to obtain an optimal design for micro devices. A computational fluid dynamics (CFD) based optimization study was used by Tonomaru *et al*. [3] to obtain an optimal design for plate-fin micro device. Water was used as a working fluid in this study. The micro device was consisted five identical channels. They indicated that the flow uniformity in the microchannel is highly dependent on manifold shape, length and location of channels and inlet flow rate. The simulation results showed the uniform flow can be achieved at longest tested channel at lowest flow rates. However, flow mal-distribution was observed when the flow rate increasing. A similar conclusion was drawn by Pan *et al*. [5] in their numerical study. They pointed out that a relatively uniform velocity distribution among parallel microchannel could be reached at longer micochannel with smaller width. The effect of inlet/outlet manifold area and flow arrangement were also discussed in this study. It was found that a manifold with large area and when the direction of inlet/outlet ports was perpendicular to the microchannel plane give a better flow uniformity among channels.

Balaji and Lakshminarayanan [6] study the effect of number and the inlet and outlet ports arrangement on flow distribution along the channels of micro heat exchangers by constructed a two dimensional model. From the obtained results, uniform flow distribution can be achieved using either an aligned single inlet and single outlet ports or single inlet port inline with the microchannels along with two outlet ports located at each corner of the micro heat exchanger.Jones, *et al*. [7] investigated flow distribution in 76 channels heat sink etched into silicon substrate by using infrared micro-particle image velocimetry. The average channel width and height of the each channel were 110μm and 371 μm, respectively. Flow maldistribution was observed at high flow rate, where, channels near the lateral edge have a substantially lower flow rate than middle channels. They stated that this might affect the heat transfer performance. Chein and Chen [8] conducted a numerical simulation to investigate the effect of maldistribution flow on microchannel heat sink performances. There was six different ways of inlet/outlet arrangements were studied; i.e I-, N-, S- D-, U-, V- types. In I-, N-, D-, and S- type fluid was supplied and leaved heat sinks horizontally, while, for U- and V-type, when fluid supplied to and leaves the heat sinks vertically. The highest temperature for all types heat sink studied located at the edge of the heat sink due to no heat dissipation by fluid convection in this region, while, the lower temperature was observed at the entrance zone of the microchannels due to high heat transfer coefficient. The uniform fluid flow among the channels can achieved when the fluid supplied and collected vertically in inlet and outlet plenum, respectively. By obtained the comparison of thermal resistance for all presented type heat sink, V-type showed the best performance among the others. Camilleri, *et al*., [9] mentioned that flow maldistribution was found affected by friction factor and the manifold structure, as the area ratio increases, the flow maldistribution become more pronounced to sudden enlargement from inlet pipe into manifold. Recently, Anbumeenakshi and Thansekhar [10], conducted an experiment study to identify an optimum manifold shape with inlet configuration in order to minimize the flow distribution in microchannel heat sink. Based on obtained results, at low flow rate, trapezoidal and triangle manifold give less flow maldistribution. However at high flow rate, rectangular a better uniformity can be achieved by using rectangular manifold.

From the aforementioned reviewed, the design of manifolds with inlet and outlet flow arrangement are an importance parameters in order to provide a high effectiveness micro heat exchangers. Previously, most studies only focused on the optimal design and neglected the effect of axial conduction that leads to non-uniform heat flux in microchannel flow [11]. Additionally, as mentioned by Baek *et al*. [12], the thermal performance of micro heat exchanger is degraded when, both parameters; axial conduction and flow maldistribution are occurred. In this paper, a numerical study is carried out to investigate the influences of inlet/outlet manifold area and inlet/outlet arrangement on flow distribution in channels.

2.0 METHODOLOGY

2.1 Description of Models

Figure 1 shows a computational domain of multichannel system by using the 3D full conjugated model. The dimensions of rectangular channels are 695 μm deep by 297 μm wide; while the fins are 695 μm deep by 209 μm wide. (see in Figure 2).

2.2 Numerical Method

A numerical simulations solving 3D Navier –Stokes equations was carried out to study the characteristics of fluid flow and conjugate heat transfer in microchannels. The following assumptions were adopted:

i) Steady state single fluid flow and heat transfer.
ii) Incompressible fluid.
iii) Negligible radiative heat transfer.
iv) Constant solid and fluid properties.
v) Negligible viscous dissipation and surface roughness effect.

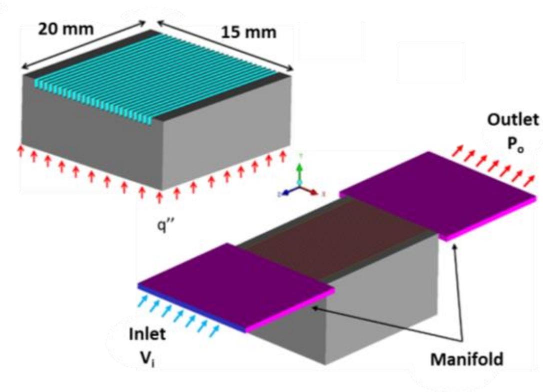


Figure 1 3D conjugated computational model of parallel multichannel

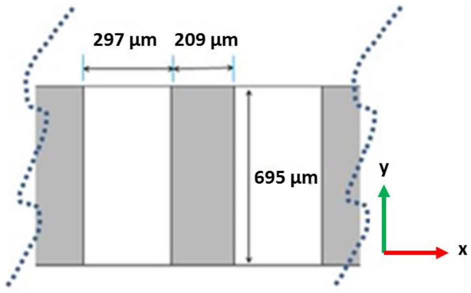


Figure 2 Channel cross-section domain of multichannel

Based on the above assumptions, the governing differential equations used to describe the fluid flow and heat transfer in the microchannel are given as:

Conservation of mass (continuity)

$$\nabla(\rho \vec{V}) = 0 \tag{1}$$

Conservation of momentum

$$\vec{V} \cdot \nabla(\rho \vec{V}) = -\nabla p + \nabla \cdot (\mu \nabla \vec{V}) \tag{2}$$

Conservation of energy for fluid

$$\vec{V} \cdot \nabla(\rho c_p T_f) = \nabla \cdot (k_f \nabla T_f) \tag{3}$$

Conservation of energy for solid

$$\nabla \cdot (k_w \nabla T_w) = 0 \tag{4}$$

At inlet manifold, a number of uniform inlet velocities were employed. A pressure outflow boundary condition was employed at the outlet. The no slip boundary condition was assigned for all wall boundaries. The heat loss through the top cover was considered to be negligible. The continuity of the temperature and heat flux is used as the conjugate boundary condition to couple the energy equations at the fluid and solid interface. The 3D model has been developed and the meshed is generated using Ansys ICEM and is further solved using Ansys Fluent 14.5. The SIMPLE scheme is used to resolve the pressure-velocity coupling. The flow momentum and energy equations are solved with a first-order upwind scheme. The hexa meshing grid scheme was used to mesh the system as shown in Figure 3. A highly compressed non-uniform grid near the channel walls was adopted in order to properly resolve viscous shear layers. Grid nodes were also concentrated along the axial direction in the entrance of the channel in order to properly resolve the flow and thermal development regions as adopted by Fedorov and Viskanta [13] and Qu and Mudawar [14]

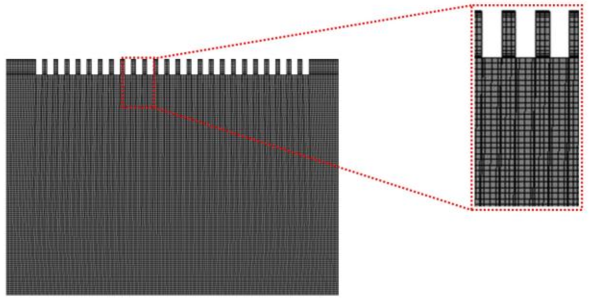
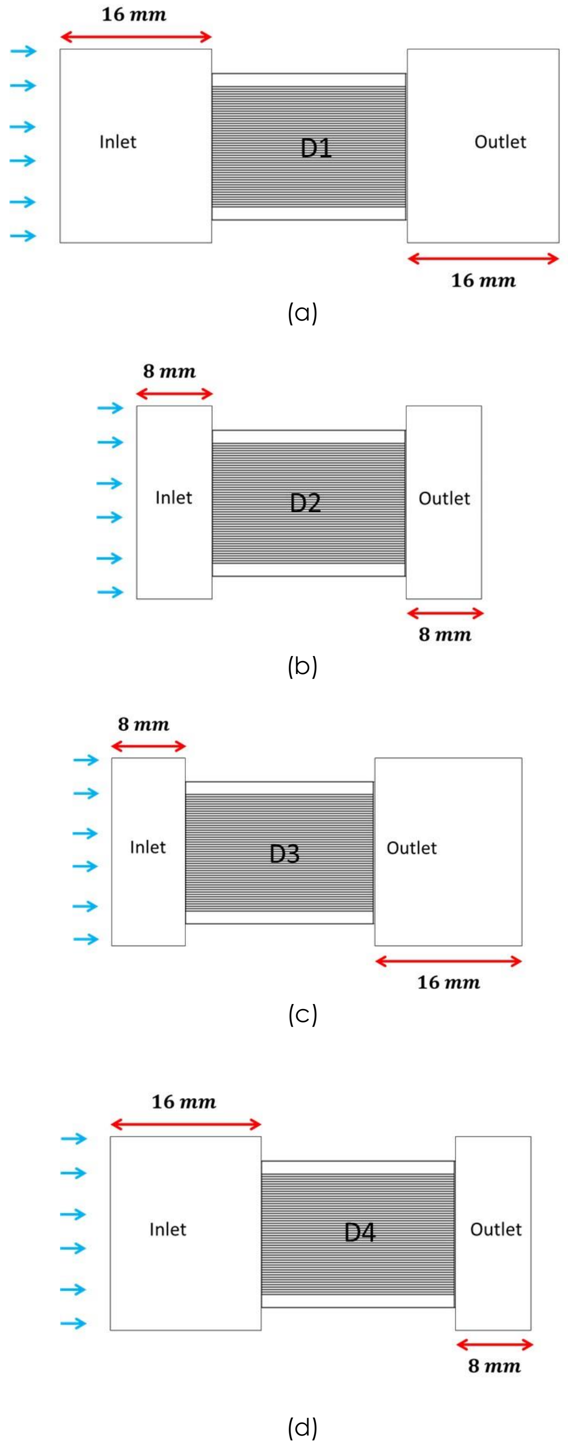


Figure 3 Schematic and computational meshes of geometrical model

In this study, the influences of inlet/outlet manifold area and inlet/outlet arrangement on flow distribution in channels were carried out as shown in Figure 4 and Figure 5, respectively and channels area was remained constant. As shown in Figure 4, four different cases were conducted to investigate the influence of manifold area. In case 1, inlet and outlet manifold area were remain the same (D1) while in case 2 (D2), inlet and outlet manifold area were 0.5 time smaller than inlet/outlet manifold in case 1(D1).

In case 3 area of inlet manifold is smaller than outlet manifold (D3) and for last case, the area of inlet manifold is larger compared to outlet manifold (D4).



(a)

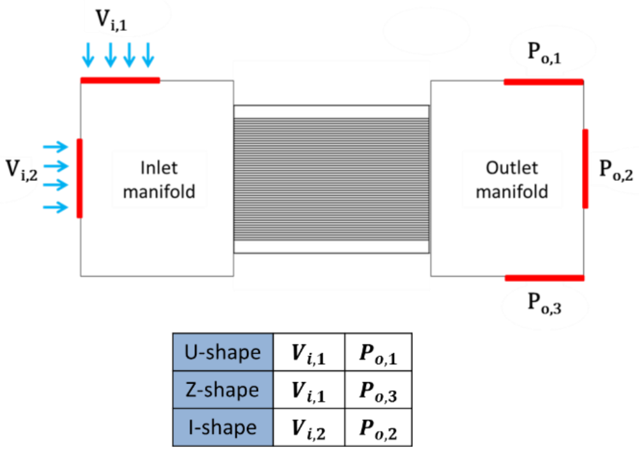
(b)

(c)

(d)

Figure 4 Four types of parallel microchannels with different configuration at inlet/outlet manifold area

As illustrated in Figure 5, two different velocity inlet and there pressure outlet were employed at inlet and outlet manifold, respectively. For U-shape of flow arrangement, $V_{i,1}$ and $P_{o,1}$ were employed at inlet and outlet manifold, respectively. However, for Z-shape, $V_{i,1}$ and $P_{o,3}$ were employed at inlet and outlet manifold and, $V_{i,2}$ and $P_{o,2}$ were employed at inlet and outlet manifold, respectively.



U-shape	$V_{i,1}$	$P_{o,1}$
Z-shape	$V_{i,1}$	$P_{o,3}$
I-shape	$V_{i,2}$	$P_{o,2}$

Figure 5 Schematic diagram of parallel microchannel with three different type of flow arrangement

3.0 RESULTS AND DISCUSSION

3.1 Validation of Numerical Results

The numerical results was validated using exiting experimental data [15]. A comparison of numerical and experimental temperature of bottom wall and temperature of fluid among channels is depicted in Figure 6. Based on figure, the numerical temperature of bottom wall and temperature of fluid showed a similar trend with values 1% higher than experimental data.

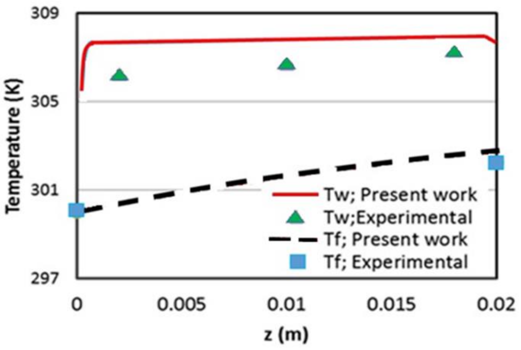
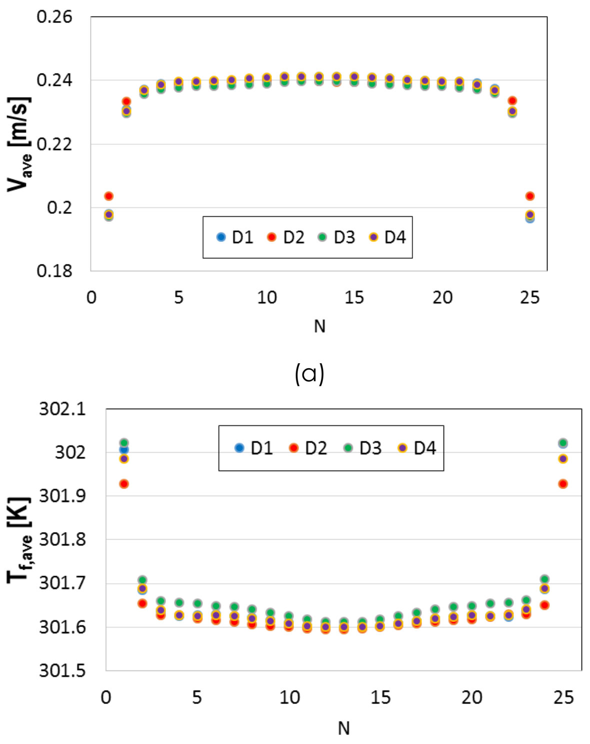


Figure 6 Comparison of numerical axial temperature of bottom wall and fluid with experimental data [15]

3.2 Influence of Inlet/Outlet Area of Manifold

In this section, the influence of inlet/outlet of manifold area on the flow distribution inside channels is discussed. Controlling area of manifold is equivalent to varying the intensity of different forces in manifold that governs flow distributions [16]. As mentioned before, four different cases were conducted and to compare the flow distributions for all cases. The comparison of average velocity and average fluid temperature distributions for all tested cases were depicted in Figure 7. Base on that figure, the first and the last channel, showed the lowest value for the velocity average, and the highest value for the temperature average, for all tested cases. Additionally, the difference between lowest and highest velocity is not more than 14% for all cases. From Figure 7, it worth mentioned that the effect of inlet/outlet area of manifold on velocity and fluid temperature distributions was insignificant.



(a)

(b)

Figure 7 (a) Velocity and (b) temperature average distributions against number of channels (N)

However, the effect of manifold area on flow distribution can be explained by investigating pressure drop among channel for all cases were depicted in Figure 8. According to numerical pressure drop showed Figure 8, for case 1, (D1) showed a lowest pressure compared to other cases. Moreover, by comparing the pressure drop for case 1(D1) and case 2(D2), it is worth to mention that, as the area of inlet and outlet manifold decrease by 50%, the pressure drop is increasing about 5%. In conclusions, large area of inlet/outlet manifold give a lower pressure drop compared to small manifold area.

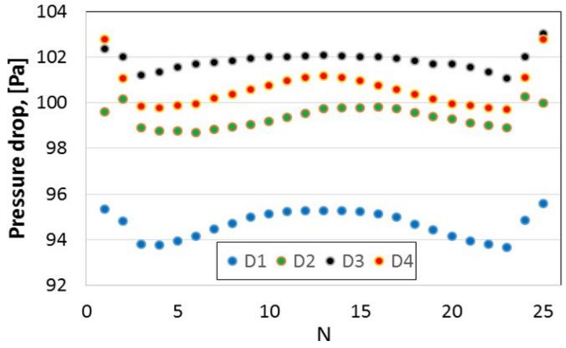


Figure 8 Pressure drop distributions against number of channels (N)

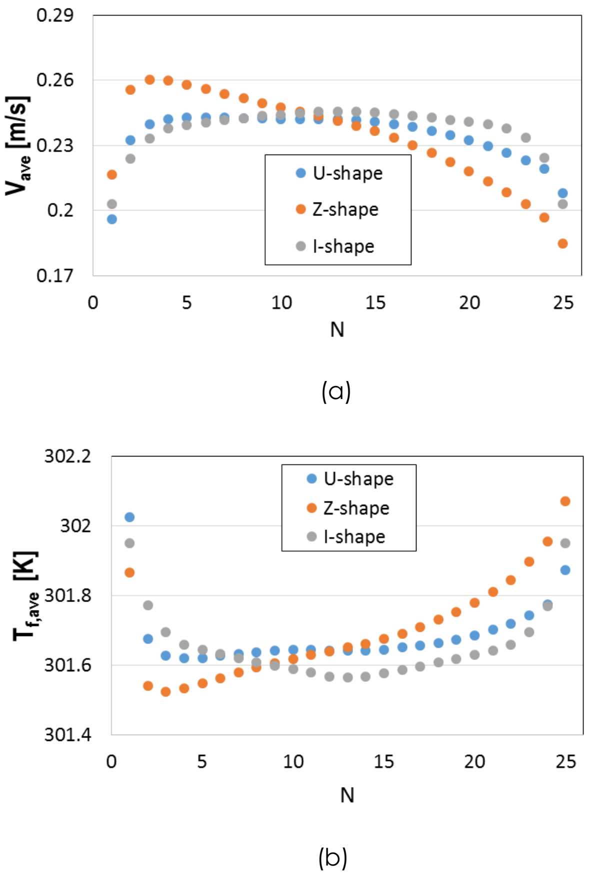
3.3 Influence of Inlet/Outlet Arrangement

Flow distributions of three different flow arrangements; U-, Z- and I- shapes are discussed in this section. Average velocity and average temperature distributions inside channels for all tested flow arrangement were illustrated in Figure 9. From this figure, it is shown that the distributions of average velocity and average temperature distributions are approximately symmetric among channels at I-shape flow arrangement. Similar results was also found by Kumaran *et al.* [17] A predicted the flow non-uniformity factor, ϕ, defined as:

$$\phi = \{(V_1^2 + V_2^2 + V_3^2 + \ldots + V_n^2)/n\}^{1/2} \times 100\ \% \qquad (5)$$

where,

$V_1, V_2, V_3, \ldots, V_n$ = { (Average flow rate – Channel flow rate)/ Average flow rate} , is calculated in this study. The smallest value of flow uniformity factor showing a better uniform flow. Based on the predicted values, ϕ, and the Z- shape flow arrangement exhibits higher value of ϕ, which is 8%, followed by U-shape, 2.6% and the I-shape, 2.49%. Therefore, it is worth to note that, a better uniformity of flow distribution also can be achieved in I-shape of flow arrangement.

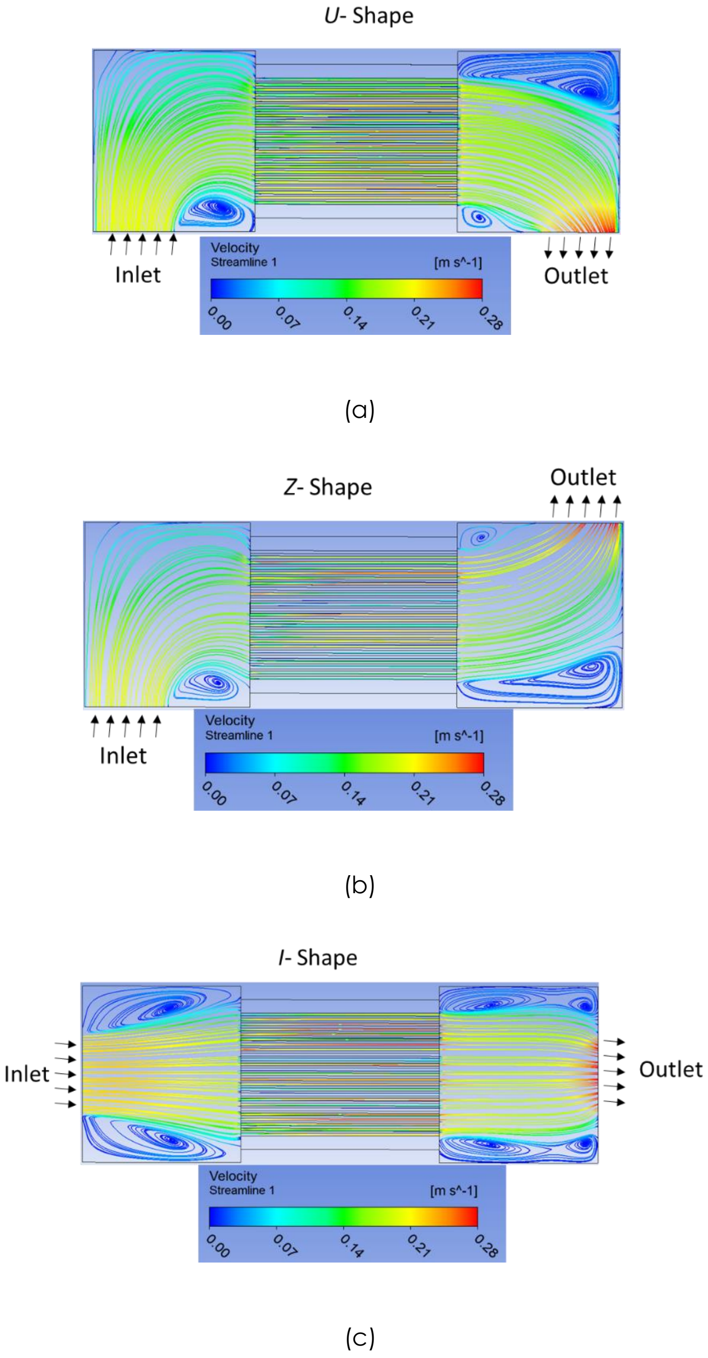


(a)

(b)

Figure 9 Comparison of (a) velocity and (b) temperature average distributions for U, Z, I flow arrangement

Velocity streamline for all flow arrangements tested in this study, were illustrated in Figure 10. As shown in Figure 10, a vortices were clearly appeared at sharped edge of manifolds and it appeared at different spot of manifolds depend on type of flow arrangement. The vortices at manifold might be influence the of flow distributions inside microchannels. It also worth to mention that, the symmetrical distribution of velocity and temperature distribution at I-shape flow arrangement might due to symmetric shape of vortices that appeared at inlet manifold.



(a)

(b)

(c)

Figure 10 Velocity contour streamline for U, Z, I flow arrangement

4.0 CONCLUSION

The flow distribution of twenty-five parallel microchannels with rectangular inlet and outlet manifold was simulated in three dimensional models by Ansys Fluent 14.5. The influences of inlet/outlet manifold area and inlet/outlet arrangement on flow distribution in channels were carried out. The simulations results show the inlet/outlet area of manifold on velocity and fluid temperature distributions was insignificant. However, the large areas of inlet/outlet manifold tested in this study give a lower pressure drop. Moreover, a better uniformity of velocity and temperature distributions were achieved in I-shape flow arrangement. It is worth to mention that, the inlet/outlet flow arrangements showing a significant effect on velocity and temperature distributions among microchannels. Besides, the behavior of the flow distributions inside channels is due to the vortices that occurred at manifold.

Acknowledgement

The authors would like to thank Ministry of Higher Education Malaysia for providing research grant under Fundamental Research grant scheme (FRGS – 9003-00537). Special thanks also to School of Manufacturing Engineering, Universiti Malaysia Perlis for providing space and facilities to undertake this work.

References

[1] S. Kandlikar, Z. Lu, W. Domigan, A. White, M. Benedict. 2009. Measurement of Flow Maldistribution in Parallel Channels and Its Application to Ex-Situ and In-Situ Experiment's in PEMFC Water Management Studies. *Int. J. Heat and Mass Transfer.* 52: 1741-1752.

[2] Zhai, Y., Xia, G., Li, Z. and Wang, H. 2017. Experimental Investigation and Empirical Correlations of Single and Laminar Convective Heat Transfer in Microchannel Heat Sinks. *Experimental Thermal and Fluid Science*. 83: 207-214.

[3] Pistoresi, C., Fan, Y. and Luo, L. 2015. Numerical Study on the Improvement of Flow Distribution Uniformity among Parallel Mini-channels. *Chemical Engineering and Processing: Process Intensification*. 95: 63-71.

[4] O. Tonomura, S. Tanaka, M. Noda, M. Kano, S. Hasebe, I. Hashimoto. 2004. CFD-based Optimal Design of Manifold in Plate-fin Microdevices. *Chem. Eng. J*. 101: 397-402.

[5] S. Balaji, S. Lakshminarayanan. 2006. Improved Design of Microchannel Plate Geometry for Uniform Flow Distribution. *The Canadian J. of Chem. Eng*. 84: 715-721.

[6] M. Pan, X. Shao, L. Liang. 2013. Analysis of Velocity Uniformity in a Single Microchannel Plate with Rectangular Manifolds at Different Entrance Velocities. *Chem. Eng. Technol.* 36(6): 1067-1074.

[7] Jones, B. J., Lee, P.-S. and Garimella, S. V. 2008. Infrared Micro-particle Velocity Measurements and Predictions of Flow Distribution in Microchannel Heat Sink. *International Journal of Heat and Mass*. 51: 1877-1887.

[8] Chein, R. and Chen, J. 2009. Numerical Study of Inlet/Outlet Arrangement Effect on Microchannel Heat Sink Performance. *International Journal of Thermal Sciences.* 48: 1627-1638.

[9] Camilleri, R., Howey, D. and McCulloch, M. 2015. Predicting the Flow Distribution in Compact Parallel Flow Heat Exchangers. *Applied Thermal Engineering*. 90: 551-558.

[10] Anbumeenakshi, C. and Thansekhar, M. R. 2016. Experimental Investigation of Header Shape and Inlet Configuration on Flow Maldistribution in Microchannel. *Experimental Thermal and Fluid Science*. 75: 156-161

[11] Sahar, A. M., Özdemir, M. R., Fayyadh, E. M., Wissink, J., Mahmoud, M. M. and Karayiannis, T. G. 2016. Single Phase Flow Pressure Drop and Heat Transfer in Rectangular Metallic Microchannels. *Applied Thermal Engineering*. 93: 1324-1336.

[12] Baek, S., Lee, C. and Jeong, S. 2014. Effect of Flow Maldistribution and Axial Conduction on Compact Microchannel Heat Exchanger. *Cryogenics*. 60: 49-61.

[13] A. G. Fedorov, R. Viskanta. 2000. Three-dimensional Conjugate Heat Transfer in the Microchannel Heat Sink for Electronic Packaging. *Int. J. Heat and Mass Transfer*. 43: 399-415.

[14] W. Qu, I. Mudawar. 2002. Experimental and Numerical Study of Pressure Drop and Heat Transfer in a Single-phase Microchannel Heat Sink. *Int. J. Heat and Mass Transfer*. 45: 2549-2565.

[15] Fayyadh, E. M., Mahmoud, M. M., Sefiane, K. and Karayiannis, T. G. 2017. Flow Boiling Heat Transfer of R134a in Multi Microchannels. *International Journal of Heat and Mass Transfer*. 110: 422-436.

[16] Siva V, M., Pattamatta, A. and Das, S. K. 2013. Investigation on Flow Maldistribution in Parallel Microchannel System for Intergrated Microelectronic Device Cooling. *IEEE Transactions on Components, Packaging and Manufacturing Technology*. 4(3): 2156-3950.

[17] Kumaran, R. M., Kumaraguruparan, G. and Sornakumar, T. 2013. Experimental and Numerical Studies of Header Design and Inlet/Outlet Configurations on Flow Mal-Distribution in Parallel Micro-Channels. *Applied Thermal Engineering*. 58(1): 205-216.