CFD Letters

Journal homepage:
https://semarakilmu.com.my/journals/index.php/CFD_Letters/index
ISSN: 2180-1363

Flow Distribution in Parallel Rectangular Multi Microchannels in Single Phase

Amirah Mohamad Sahar[1,*], Mohamad Shaiful Ashrul Ishak[2], Jan Wissink[3], Mohamed M. Mahmoud[3,4], Tassos G. Karayiannis[3]

[1] Faculty of Computer and Mathematical Sciences, Universiti Teknologi MARA, Shah Alam, Malaysia
[2] Faculty of Mechanical Engineering Technology, Universiti Malaysia Perlis, Malaysia
[3] College of Engineering, Design and Physical Sciences, Brunel University London, United Kingdom
[4] Faculty of Engineering, Zagazig University, Egypt

ARTICLE INFO

Article history:
Received 11 August 2022
Received in revised form 4 October 2022
Accepted 2 November 2022
Available online 11 January 2023

Keywords:
Flow distributions; maldistribution factor; parallel; microchannels; single phase flow

ABSTRACT

This study proposed newly designed inlet manifolds to manage non-uniformity in parallel multi microchannel heat sink by introducing edges with a curved shape were introduced in order to reduce flow-recirculation at the sharp edges. This resulted in a better flow distribution in the parallel channels. A comprehensible numerical study has been performed using ANSYS-Fluent and a three-dimensional computational domain, incorporating the effect of conjugated heat transfer, was employed in this study. R134a was used as the working fluid and copper was selected as the heat sink material. The dimensionless channel flow ratio and flow maldistribution factor were introduced to quantify the flow distribution inside individual channels and the uniformity of this flow distribution. A uniform flow distribution is achieved when the maldistribution factor value approaches 0.

1. Introduction

Microchannel heat sink is an advanced cooling technique which appears as a promising method that can provide high heat transfer rate due to small hydraulic diameter. Furthermore, microchannel heat sink is easy to be fabricated compared to other micro cooling device [1]. An innovation of microchannel heat sink design is required as fast development in electronic industry increases the power density rapidly. Therefore, basic problem in the conventional microchannel heat sink need to be identified first before the new microchannel design is proposed [2]. The design of a parallel microchannel heat sink requires an accurate depiction of microchannel flow. Two approaches are commonly used to describe fluid flow and heat transfer in microchannel heat sinks. The first employs simplified analytical methods, where the solid walls separating the microchannels behave like thin fins, the temperature of the fluid is uniform, heat transfer is one-dimensional and the heat transfer coefficient along the channel walls, evaluated from empirical heat transfer correlations, is constant.

* *Corresponding author.*
E-mail address: amirahsahar@uitm.edu.my

https://doi.org/10.37934/cfdl.15.1.6775

The other approach involves solving the three-dimensional Navier- Stokes and energy equations. Both approaches are based on prior understanding of fluid flow and heat transfer in macrochannels and are therefore valid only if the liquid flow in microchannels follows the same conservations laws as in macrochannels. The deviation reported in the parallel microchannel heat exchanger is due to experimental accuracy and the assumption of a uniform flow distribution in the data reduction model [3] and the maldistribution phenomenon is not examined well enough [4]. For example, a disagreement of the friction factor values reported in multiple microchannels may be due to the existence of a flow maldistribution in multiple microchannels [5-6]. The effect of flow maldistribution remains in experiments is incorporated into the value of the heat transfer coefficient. The assumption of a uniform flow distribution should no longer be accepted in multichannel flow [7].

Flow maldistribution is an uncalled phenomenon in single phase heat sink and accountable for the depreciation in the effectiveness of the system [8-9]. The flow maldistribution through the heat exchanger is generally associated with improper entrance configuration, due to poor header design and an imperfect passage-to-passage flow distribution in a highly compact heat exchanger caused by various manufacturing tolerances. Plenty of studies have been conducted to explore this important fundamental issue and focused on improving the performance of mini/microchannel heat sink by reducing flow maldistribution and a comprehensive review of the recent progress and future trends of microchannel heat sink were provided by Ghani *et al*., [10] and He *et al*., [11]. As concluded by Kumaraguruparan *et al*., [12] and the flow maldistribution between channels is due to flow separation or recirculation that occurred in the inlet header or manifold. due to the vortex flow, which lead to reduction of flow rate in the channels [13]. However, there were no explanation on the significant effect of flow separation or recirculation phenomenon that occurred in the inlet header. Hence, further research is needed to better understand and optimise the fluid flow distribution over parallel microchannels.

This is particularly important in optimising the performance and efficiency of multichannel heat sinks. Therefore, in the present study, a new manifold design will be proposed as an effort to reduce the flow maldistribution in parallel microchannel heat sink and to examine the significant effect of flow separation and recirculation occurring in the inlet header or manifold on flow maldistribution.

2. Computational model

2.1 Model description

A three-dimensional model of a parallel microchannel heat sink was designed using Ansys ICEM 14.5. The computational setup is similar to the parallel microchannel heat sink that was employed in the experimental work conducted Fayyadh et al. [14] and combined with rectangular inlet and outlet manifolds (see Fig. 1). The working fluid enters horizontally in the inlet manifold and leaves horizontally through the outlet. The fluid is distributed to all the microchannels through the inlet manifold and collected by the outlet manifold. The length and width of both inlet and outlet manifolds are 16 mm and 15 mm, respectively. The depth of the manifolds is identical to the depth of microchannel, i.e., 0.695 mm in this case.

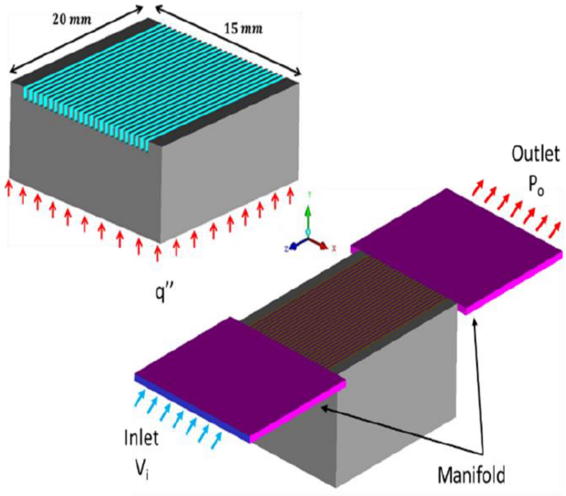


Fig. 1. Computational domain for multichannel simulations

2.2 Governing equations and boundary conditions.

To study the flow distribution over the parallel microchannels the following assumptions are taken:

i. The fluid flow is steady, single phase, laminar, incompressible and three dimensional.
ii. Properties of the solid and fluid are independent of temperature and pressure.
iii. Radiative heat transfer and surface roughness effects are negligible

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

Conservation of mass (continuity)

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

Conservation of momentum

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

Conservation of energy for the fluid

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

Conservation of energy for the solid

$$\nabla\cdot(k_w\nabla T_w) = 0 \quad (4)$$

All the governing equations (1)–(4) are solved with the help of ANSYS-Fluent 14.5 using a finite volume method (FVM). The SIMPLE scheme is used to resolve the pressure velocity coupling. Momentum and energy equations are discretized by a first-order upwind scheme. Based on the governing equations and operating conditions, a fixed inlet velocity is employed at the inlet and a pressure boundary condition is applied at the outlet. The properties of R134a, relevant to this research, are ρ = 1204 kg/m^3 , Cp = 1426 J/kg-K, k_f = 0.083 W/m-K and ν = 1.0 × 10−6 m^2 /s, while for solid copper the thermal conductivity is given by k_s = 387.6 W/m-K. The inlet temperature of the R134a fluid is fixed at 315 K, and the inlet velocity (Vi,measured at the inlet manifold) is varied from 0.03 to 0.1133 m/s. The average Reynolds number of channel in this study ranges from Re=200 to Re=850 and flow can be assumed laminar. A constant heat flux (22 kW/m^2) is prescribed at the base of the heat sink for all cases. All walls other than the interface and base are assigned with adiabatic and no-slip boundary conditions. When the residual values become less than 10^{-6} for the continuity, x-velocity, y-velocity, and z-velocity and less than 10^{-9} for the energy, the solutions were considered to be converged.

2.3 Grid independence and validations

The hexa meshing grid scheme constructed in Ansys ICEM 14.5 was used to mesh the system as shown in Fig. 2. To properly resolve near-wall viscous shear layers, a highly compressed non-uniform grid was used inside the channels. Grid points 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 advised by Fedorov and Viskanta [15] and Qu and Mudawar [16]. Three different meshes (G1=215, 312 nodes, G2=516, 782 nodes, G3=818, 252 nodes) test the grid independence of the solution. Fig. 3 shows the variations in the velocity distribution over the 25 channels in the proposed arrangement heat sink obtained using the three meshes under the same volumetric flow rate and the same boundary conditions. It is observed that the velocity distribution for Grid 2 and Grid 3 is almost the same, with a maximum percentage difference of less than 3 %. Therefore, to save computational time and memory Grid 2 was used for all simulations.

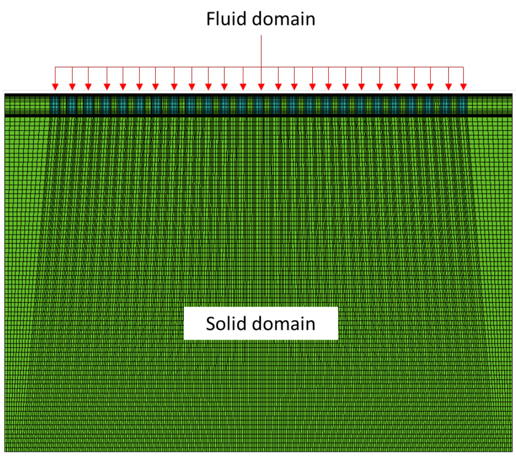


Fig. 2. Computational mesh of the geometrical model

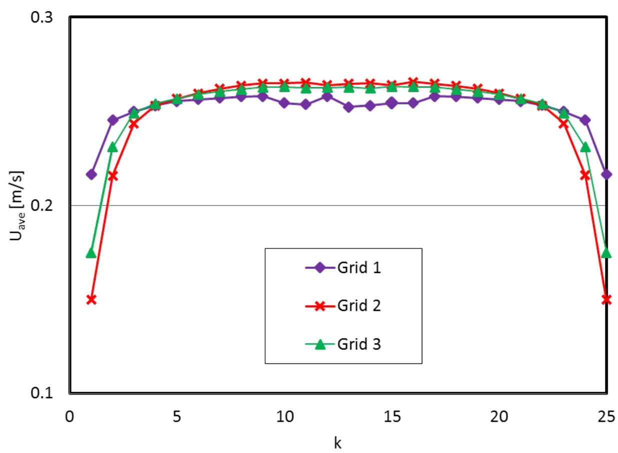


Fig. 3. Grid independency study (G1=215,312 nodes, G2=516, 782 nodes, G3=818, 252 nodes)

Numerical results were validated by comparison of the friction factor and average Nusselt number obtained in experiments [14]. To match the setup used in these experiments, especially for the validation runs a trapezoidal inlet and outlet manifold was used, while rectangular inlet and outlet manifolds were used to investigate the flow maldistribution for all other cases. Fig. 3 shows that the friction factor obtained in the simulation was in good agreement with experimental results. Compared to the correlation of Shah and London [17] however, the friction factor was slightly underestimated, though a similar trend was obtained. Also, the average Nusselt number obtained in the numerical simulations (see Fig. 4) was found to be lower than in the experimental data [14] and

Shah and London [17], which, as discussed in Sahar et al. [18], could be attributed to the effect of axial conduction on the heat transfer rate and the error arising from the assumption of uniform heat flux adopted in the experimental data reduction.

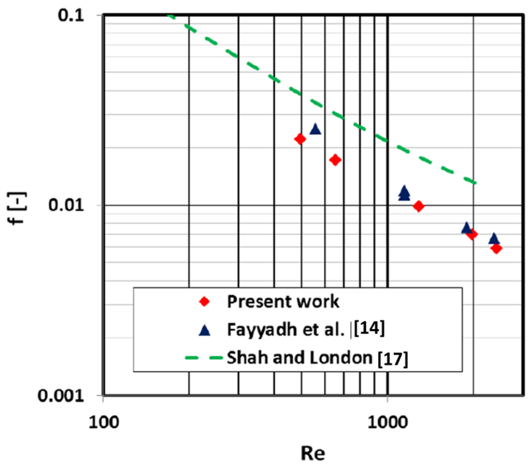


Fig. 4. Comparison of predicted friction factor with experimental results and correlations

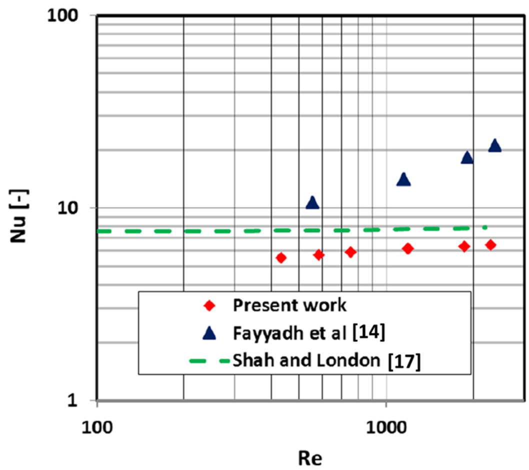


Fig. 5. Comparison of predicted Nusselt with experimental results and correlations

3. Results and discussion

In order to quantify the flow distribution over the parallel channels, the dimensionless channel flow ratio

$$\beta_k = {}^{Q_k}/_{Q_{total}} \tag{5}$$

is introduced, where Q_k (m^3/s) denotes the volume flow rate of channel *k* and Q_{total} (m^3/s) is the total volume flow rate. In order to quantify the overall flow maldistribution of the whole fluidic system, the maldistribution factor

$$MF = \sqrt{\frac{1}{N-1}\sum_{k=1}^{N}\left(\frac{\sigma_k-\bar{\sigma}}{\bar{\sigma}}\right)^2} \tag{6}$$

is introduced, where $\bar{\sigma}$ is the mean flow rate in the parallel micro channels, defined by

$$\bar{\sigma} = \frac{\sum_{k=1}^{N}\sigma_k}{N} \tag{7}$$

Note that a uniform flow distribution is achieved when the *MF* value approaches 0.

3.1 Proposed manifold design

The manifolds design appears to cause a flow restriction at the lateral edges resulting in reduced flow rates in the channels in this region due the large losses at sharp corners of the inlet manifold as shown in Fig.4 Therefore, to examine the significant influence of the separated flow occurring at the sharp corner in the inlet manifold on the flow distribution inside the microchannels, a heat sink with a curve edge (shown in Fig.5) was investigated numerically.

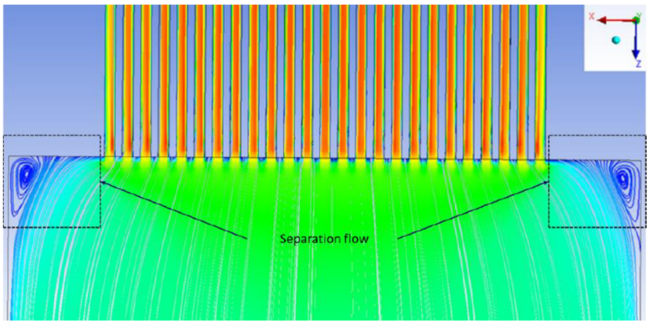


Fig. 6. Velocity streamlines of microchannel heat sink

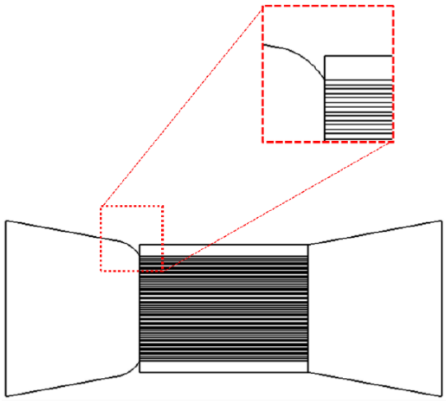
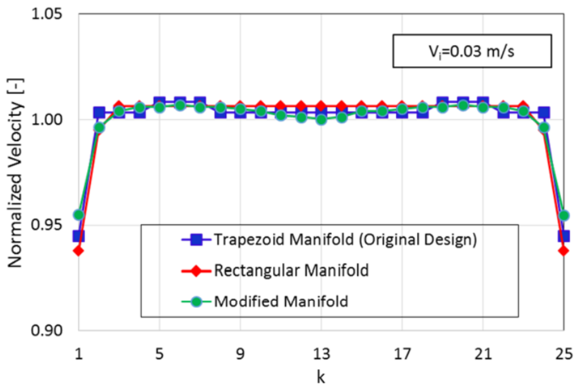
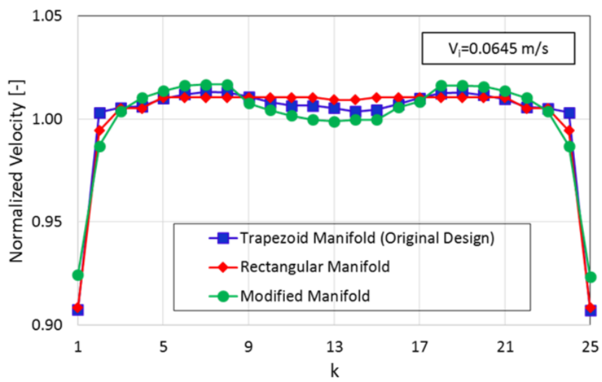


Fig. 7. Schematic diagram of proposed manifold

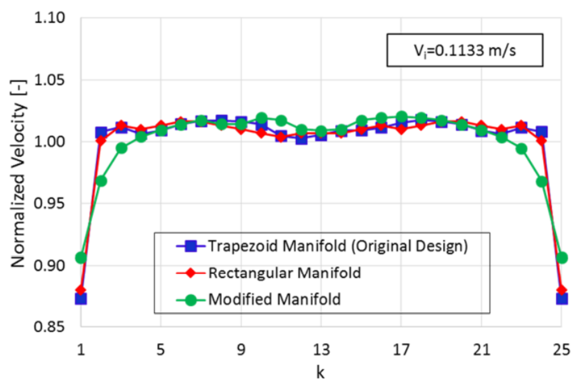
Fig. 8 shows the normalized velocity for three different manifolds design at the velocity inlet (a) V_{in}=0.030 m/s (b) V_{in}=0.0645 m/s (c) V_{in}=0.1133 m/s. A small flow maldistribution was observed at the lower flow rate. However, a significant flow maldistribution was observed at higher flow rate. Once again, for all three cases, the channels near the centerline having a greater flow ratio compared to the channels near the lateral edges of the heat sink. At a higher velocity inlet, the relative difference between the maximum and minimum flow ratio through the channels is approximately 12%, 13%, 6 % for trapezoid manifold, rectangular manifold, and modified manifold, respectively.



(a)



(b)



(c)

Fig. 8. Normalized velocity in each channel of (a) Trapezoid manifold (b) Rectangular manifold (c) Proposed manifold at different inlet conditions; (a) Vin=0.030 m/s (b) Vin=0.0645 m/s (c) Vin=0.1133 m/s

Therefore, to quantify the flow distribution for all three cases at various velocity inlet conditions, the value of MF has been evaluated and plotted in Fig. 9. Based on Fig.9 MF indeed increases when inlet velocity increases. It is also interesting to note that the modified manifold clearly provides a better uniform flow distribution through the heat sink, with maldistribution factors 0.014, 0.025, and 0.032 at inlet velocities 0.03 m/s, 0.0645 m/s and 0.1133 m/s, respectively, compared to the other two. Note that only small differences are observed between the trapezoid manifold and the rectangular manifold.

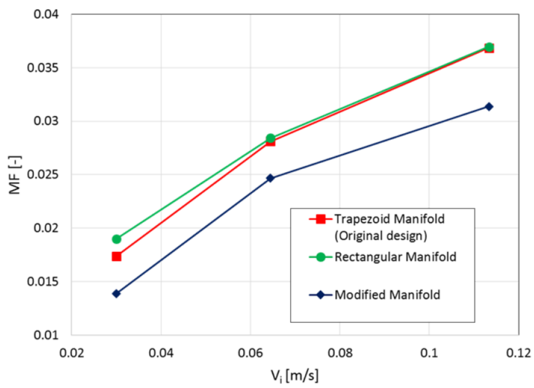


Fig. 9. Maldistribution factor for different inlet manifolds at various inlet velocity conditions

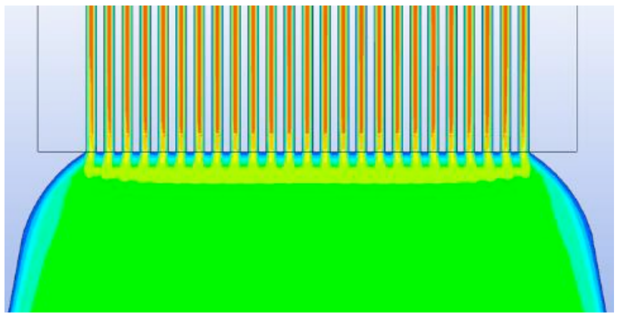


Fig. 10. Velocity contour in the inlet proposed manifold

The reasons for the better flow performance of the proposed manifold design are clearly illustrated in Fig.10. As mentioned previously, the formation of secondary flow at corners in the inlet manifold was identified as the root cause of the reduction of flow ratio in the channels at the lateral edges. As noted earlier by Jones et al. [19] a recirculation region identified in the inlet manifold constricts the available space into which the fluid is distributed, which resulting in higher localized velocities and larger pressure drops within the manifold. The fluid feeding the channels near the centreline has a more direct path through the manifold, so that the central channels receive a larger portion of the flow than channels near the lateral edge.

4. Conclusions

In this chapter, the numerical study was conducted to study the flow distribution inside a multichannel configuration with twenty-five parallel channels. The numerical study was carried out using R134a as a working fluid. As a validation of present numerical work, a detailed description of the experimental facility that was modelled was given in Fayyadh et al. [14] and friction factor and Nusselt number were compared. A newly design for the inlet manifold was proposed in this study, where edges with a curved shape were suggested in order to reduce the occurrence of flow recirculation at the sharp edges. The numerical results for the proposed manifold indicate that there is no secondary flow occurring in the inlet manifold so that the mass flow rate remains almost the same in all channels. In fact, the relative difference in mass flow rates reduces to 6%. Similar results were found by Siddique *et al*. [20], hence, it can be concluded that the proposed manifold distributes flow more uniformly in channels than the other collector shapes considered.

Acknowledgement
The authors would like to express their appreciation to the Ministry of Higher Education Malaysia and Research Management Centre Universiti Malaysia Perlis (project number: FRGS/1/2018/TK07/UNIMAP/02/2) for awarding a research grant to undertake this project. The authors also acknowledge the Faculty of Mechanical Engineering Technology for the research facilities and space to conduct this research.

References
[1] Japar, Wan Mohd Arif Aziz, Nor Azwadi Che Sidik, R. Saidur, Natrah Kamaruzaman, Yutaka Asako, Siti Nurul Akmal Yusof, and Mohd Nizam Lani. "Hybrid Microchannel Heat Sink with Sustainable Cooling Solutions: Numerical Model Validation." *CFD Letters* 14, no. 4 (2022): 91-117. https://doi.org/10.37934/cfdl.14.4.91117
[2] Japar, Wan Mohd Arif Aziz, Nor Azwadi Che Sidik, Natrah Kamaruzaman, Yutaka Asako, and Nura Mu'az Muhammad. "Hydrothermal performance in the Hydrodynamic Entrance Region of Rectangular Microchannel Heat Sink." *Journal of Advanced Research in Numerical Heat Transfer* 1, no. 1 (2020): 22-31.
[3] Kumaran, R. Manikanda, G. Kumaraguruparan, and T. Sornakumar. "Experimental and numerical studies of header design and inlet/outlet configurations on flow mal-distribution in parallel micro-channels." *Applied Thermal Engineering* 58, no. 1-2 (2013): 205-216. https://doi.org/10.1016/j.applthermaleng.2013.04.026
[4] Dąbrowski, Paweł. "Thermohydraulic maldistribution reduction in mini heat exchangers." *Applied Thermal Engineering* 173 (2020): 115271. https://doi.org/10.1016/j.applthermaleng.2020.115271
[5] Peng, X. F., G. P. Peterson, and B. X. Wang. "Frictional flow characteristics of water flowing through rectangular microchannels." *Experimental Heat Transfer An International Journal* 7, no. 4 (1994): 249-264. https://doi.org/10.1080/08916159408946484
[6] Peng, X. F., and G. P. Peterson. "Convective heat transfer and flow friction for water flow in microchannel structures." *International journal of heat and mass transfer* 39, no. 12 (1996): 2599-2608. https://doi.org/10.1016/0017-9310(95)00327-4
[7] Solovitz, Stephen A., and Jeffrey Mainka. "Manifold design for micro-channel cooling with uniform flow distribution." *Journal of fluids engineering* 133, no. 5 (2011). https://doi.org/10.1115/1.4004089
[6] Kim, Jungchul, Jeong Heon Shin, Sangho Sohn, and Seok Ho Yoon. "Analysis of non-uniform flow distribution in parallel micro-channels." *Journal of Mechanical Science and Technology* 33, no. 8 (2019): 3859-3864. https://doi.org/10.1007/s12206-019-0729-8
[8] Kumar, Sanjeev, and Pawan Kumar Singh. "Effects of flow inlet angle on flow maldistribution and thermal performance of water cooled mini-channel heat sink." *International Journal of Thermal Sciences* 138 (2019): 504-511. https://doi.org/10.1016/j.ijthermalsci.2019.01.014
[9] Siddiqui, Osman K., Mohannad Al-Zahrani, A. Al-Sarkhi, and Syed M. Zubair. "Flow Distribution in U-and Z-Type Manifolds: Experimental and Numerical Investigation." *Arabian Journal for Science and Engineering* 45, no. 7 (2020): 6005-6020. https://doi.org/10.1007/s13369-020-04691-4
[10] Ghani, Ihsan Ali, Nor Azwadi Che Sidik, Natrah Kamaruzzaman, Wira Jazair Yahya, and Omid Mahian. "The effect of manifold zone parameters on hydrothermal performance of micro-channel HeatSink: A review." *International Journal of Heat and Mass Transfer* 109 (2017): 1143-1161. https://doi.org/10.1016/j.ijheatmasstransfer.2017.03.007
[11] He, Ziqiang, Yunfei Yan, and Zhien Zhang. "Thermal management and temperature uniformity enhancement of electronic devices by micro heat sinks: A review." *Energy* 216 (2021): 119223. https://doi.org/10.1016/j.energy.2020.119223
[12] Kumaraguruparan, G., R. Manikanda Kumaran, T. Sornakumar, and T. Sundararajan. "A numerical and experimental investigation of flow maldistribution in a micro-channel heat sink." *International Communications in Heat and Mass Transfer* 38, no. 10 (2011): 1349-1353. https://doi.org/10.1016/j.icheatmasstransfer.2011.08.020
[13] Wang, Chi-Chuan, Kai-Shing Yang, Jhong-Syuan Tsai, and Youn Chen. "Characteristics of flow distribution in compact parallel flow heat exchangers, part II: Modified inlet header." *Applied Thermal Engineering* 31, no. 16 (2011): 3235-3242. https://doi.org/10.1016/j.applthermaleng.2011.06.003
[14] Fayyadh, Ekhlas M., Mohamed M. Mahmoud, K. Sefiane, and T. G. Karayiannis. "Flow boiling heat transfer of R134a in multi microchannels." *International Journal of Heat and Mass Transfer* 110 (2017): 422-436. https://doi.org/10.1016/j.ijheatmasstransfer.2017.03.057

[15] Fedorov, Andrei G., and Raymond Viskanta. "Three-dimensional conjugate heat transfer in the microchannel heat sink for electronic packaging." *International Journal of Heat and Mass Transfer* 43, no. 3 (2000): 399-415. https://doi.org/10.1016/S0017-9310(99)00151-9

[16] Qu, Weilin, and Issam Mudawar. "Experimental and numerical study of pressure drop and heat transfer in a single-phase micro-channel heat sink." *International journal of heat and mass transfer* 45, no. 12 (2002): 2549-2565. https://doi.org/10.1016/S0017-9310(01)00337-4

[17] Shah, Ramesh K., and Alexander Louis London. Laminar flow forced convection in ducts: a source book for compact heat exchanger analytical data. Academic press, 2014.

[18] Sahar, Amirah M., Mehmed R. Özdemir, Ekhlas M. Fayyadh, Jan Wissink, Mohamed M. Mahmoud, and Tassos G. Karayiannis. "Single phase flow pressure drop and heat transfer in rectangular metallic microchannels." *Applied Thermal Engineering* 93 (2016): 1324-1336. https://doi.org/10.1016/j.applthermaleng.2015.08.087

[19] Jones, Benjamin J., Poh-Seng Lee, and Suresh V. Garimella. "Infrared micro-particle image velocimetry measurements and predictions of flow distribution in a microchannel heat sink." *International Journal of Heat and Mass Transfer* 51, no. 7-8 (2008): 1877-1887. https://doi.org/10.1016/j.ijheatmasstransfer.2007.06.034

[20] Siddique, Ayyaz, Bhaskar J. Medhi, Amit Agrawal, Anugrah Singh, and Sandip K. Saha. "Design of a collector shape for uniform flow distribution in microchannels." *Journal of Micromechanics and Microengineering* 27, no. 7 (2017): 075026.