Penerbit
Akademia Baru

Journal of Advanced Research Design

Journal homepage:
https://akademiabaru.com/submit/index.php/ard
ISSN: 2289-7984

Numerical Analysis of Heat Transfer in Microchannel Heat Transfer in Microchannel Heat Sink using Flow Disruption

Muhammad Hazeer Khiralsaleh[1], Nor Azwadi Che Sidik[1,*], Kamyar Shameli[2]

[1] Department of Mechanical Precision Engineering, Malaysia-Japan International Institute of Technology, Universiti Teknologi Malaysia, 54100 Kuala Lumpur, Malaysia
[2] School of Medicine, Institute of Virology, Technical University of Munich, Munich, Germany

ARTICLE INFO

Article history:
Received 29 August 2023
Received in revised form 1 December 2023
Accepted 11 December 2023
Available online 21 December 2023

Keywords:
Microchannel; computational fluid dynamic; SOLIDWORKS; ANSYS

ABSTRACT

Microchannel heat sink is a passive heat exchanger that transfers the heat generated by an electronic or a mechanical device to a fluid medium, often a liquid coolant, where it is dissipated away from the device, thereby allowing regulation of the device's temperature at optimum levels. It is widely used in computers that are used to cool central processing units or graphic processors. In order to achieve great heat transfer performance of microchannel heat sink, passive method are used in this research particularly the flow disruption and secondary channel. A three-dimensional computational fluid dynamic (CFD) was conducted to study the characteristic of fluid flow and heat transfer in proposed design of microchannel heat sink with secondary channel and rectangular ribs (MCHS-SCRR) This research is to develop and optimize the heat transfer performance in a microchannel heat sinks to get high Nusselt Number low friction factor and high performance factor by optimizing the geometry design a simulation using certain software such as SOLIDWORKS and ANSYS. The result compared to previous research in this field of study.

1. Introduction

Every single electronic devices and hardware create excess heat and subsequently require thermal management to enhance reliability and avert premature failure. The total of heat output is equivalent to the power input, if there are no other energy interactions. There are a few techniques for cooling including different styles of heat sinks.

In the course of the most recent decade, micromachining innovation has been progressively utilized for the improvement of highly efficient cooling devices in view of its undeniable advantages, for example, less coolant requests and small dimensions [1]. Subsequently, the investigation of fluid flow and heat transfer in microchannels and chambers, which are two fundamental parts of such devices, have pulled in more considerations with expansive applications in both engineering and medical problems [2,3]. The greater part of the investigations is centred on the improvement of the

---

[*] *Corresponding author.*
E-mail address: azwadi@semarakilmu.com.my

https://doi.org/10.37934/ard.106.1.113

microchannel cooling performance to correlate the channel spacing (i.e. height and width) with its thermal performance characteristics. Although experimental investigations of microchannel cooling have been attempted previously [4], the technique has still been compelled by two key facts, firstly the present ability for machining microstructures and also the limitations of measurements on heat transfer parameter along the channel wall because of its three-dimensionality and micro-scale structures. As a result, numerical simulation have been progressively received for the examination of this sort of flow problems [5].

2. Literature Review

There are variety of microchannel heat sinks that has been produced since 1981 that had been proposed by Tuckerman and Pease [6]. Since then, a large and growing body of literature has investigated the performance of all kind of microchannel heat sinks in many ways including active and passive cooling techniques. In this research, passive cooling techniques are focused on.

2.1 Passive Cooling Method

Cooling is one of the main concerns in many different industries. In order to avoid creation of hot spots, heat has to be removed or it will shortens the life span of mechanical devices or even permanent damage of electronic components. According to Che Sidik [7] passive cooling methods are cost effective and more reliable than active cooling due to the absence of moving parts. The differences between passive and active cooling techniques is that passive cooling uses no energy while active cooling involves the use of energy as simplified in Figure 1.

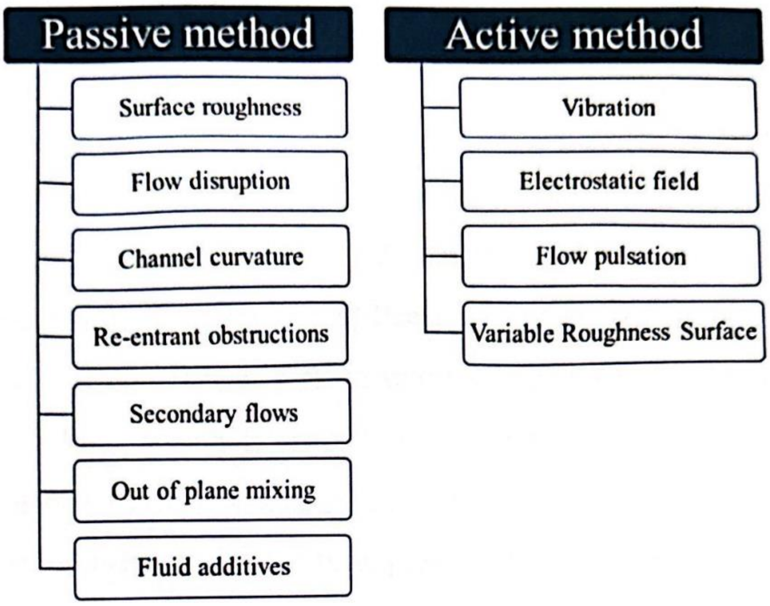


Fig. 1. Passive and cooling enhancement methods

2.2 Microchannel Heat Sink

Microchannel heat sink (MCHS) is one of the widely known high-tech devices that have been considered passive cooling methods especially for electronics cooling [7]. Microchannel heat sink (MCHS) is a device that dissipate heat fluxes which has been introduced by Tuckerman and Pease [6].

2.3 First Generation

Microchannel heat sink (MCHS) was introduced by Tuckerman and Pease [6] in 1981. Within these techniques, the microchannel heat sink has demonstrated high productivity when they have researched experimentally heat transfer in silicon microchannel heat sink. Tuckerman and Pease claims that the investigation stressed the ability of microchannel heat sink to remove heat up to 790 W/cm^2. The high surface area to volume which gives by microchannel has altogether contributed in expanding the thermal effectiveness. Moreover, the highly thermal conductivity and specific heat for fluids have likewise prompted to enhance heat transfer performance. Figure 2 depicts an example of an application of MCHS.

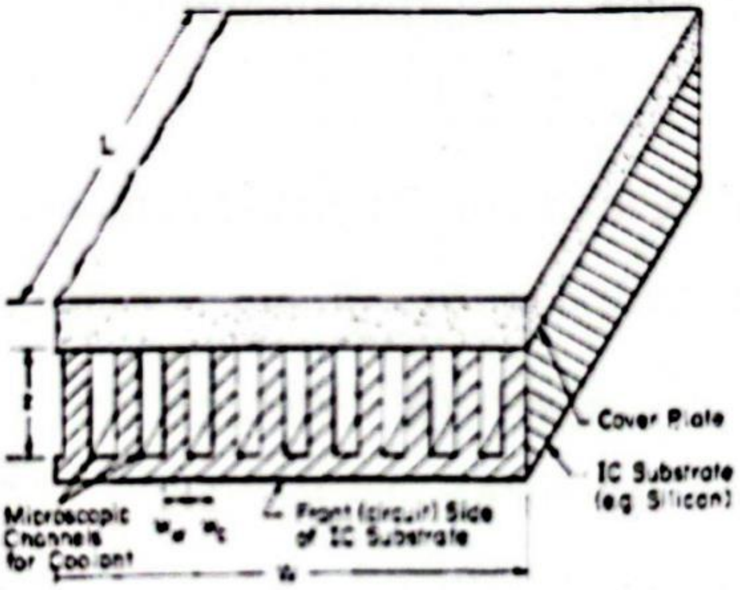


Fig. 2. Schematic view of component heat sink incorporated into an integrated circuit chip. For a 1 cm^2 silicon IC using a water coolant, the optimum dimensions are approximately w_w = wc 57 µm and z = 365 µm

2.4 Geometry Design

A considerable measure of studies have occupied the enthusiasm towards passive techniques which applied in conventional channels to be used in microchannel. From previous researches, it can be seen that most of early researchers have put their effort to improve the thermal performance of conventional MCHS by manipulating channel length, channel aspect ratio, wall thickness and even the cross-section shape of microchannel (e.g. circular, triangular and trapezoidal). Steinke and Kandlikar [8] proposed a few techniques which are enforceable in microchannels to enhance heat transfer in microchannel, for example, incorporation of mixing features to enhance the mixing flow, increase local heat transfer coefficient by breaking boundary layer using intermittent construction, incorporation of grooves and ribs. As needs be, numerous investigations have received these methodologies to enhance the thermal performance of microchannel heat sink.

In a research by Gunasegaran *et al.*, [9] while considering the Reynolds number ranges between 100-1000, they numerically analysed the effect of geometrical parameters on the heat sink performance particularly rectangular, trapezoidal and triangular microchannels. The researchers presented thus far provide evidence that the rectangular shaped microchannel with the smallest hydraulic diameter has the greatest value of heat transfer coefficient. Figure 3 shown the schematic diagrams of three s of channels.

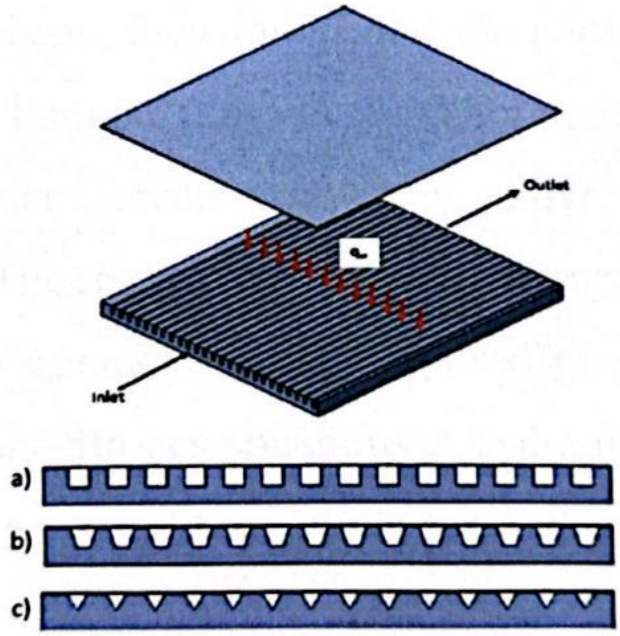


Fig. 3. Schematic diagrams of (a) Rectangular, (b) Trapezoidal and (c) Triangular microchannels

In another research, Weilin *et al*., [10] has conducted an investigation on the now characteristics of water through trapezoidal silicon microchannels with a hydraulic diameter ranging from 51 to 169 um. In their research, they first proposed a new correlation of roughness viscosity for trapezoidal microchannels. Weilin *et al*., [10] also point out that this viscosity should be considered in the equation of motion in a manner similar to the eddy viscosity in turbulent flow. Figure 4 shown the schematic diagram of trapezoidal microchannels with silicon substrate and pyrex glass cover.

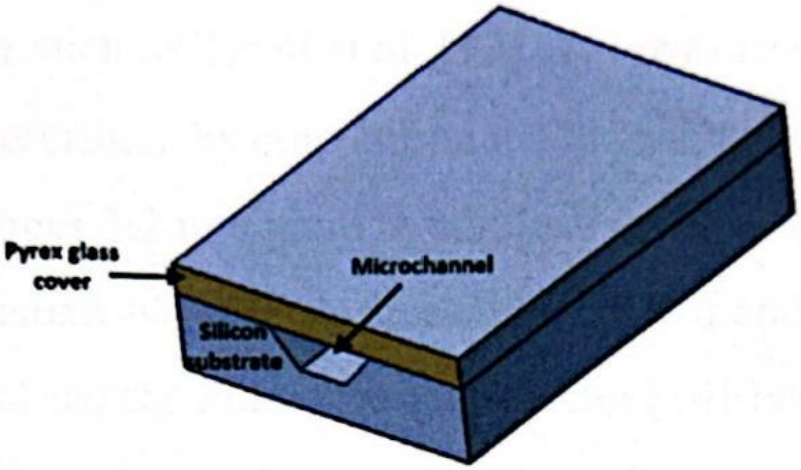


Fig. 4. Schematic diagram of trapezoidal microchannels with silicon substrate and pyrex glass

To determine the flow field and pressure drop for water flow in rectangular microchannels, Liu and Garimella [11] performed experimental and numerical investigations. In their analysis, they found that the conventional correlations offer reliable predictions for the laminar flow. In the other hand, Wang *et al*., [12] has investigate the laminar forced convection flow characteristics through a trapezoidal microchannel. The results shown that the comparison of wall temperatures and local Nusselt with the numerical study acquired good agreement indicated the validity of continuous Navier-Strokes equations at hydraulic diameter 155 um. Figure 5 shown the schematic diagram of trapezoidal microchannel proposed by Wang *et al*., [12].

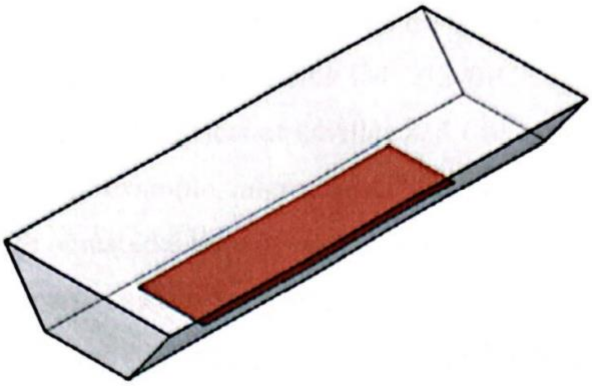


Fig. 5. Schematic diagram of trapezoidal microchannels proposed by Wang *et al*., [12]

Other researcher such as Tiselj *et al*., [13] also confirmed that the heat transfer characteristics can be described by conventional Navier-Strokes and energy equations for Reynold number ranges 3.2 to 64 and hydraulic diameter 160 um. Throughout this finding, it is also consistent with the earlier study by Wu and Cheng [14]. By using different cross-sectional aspect ratio, Wu and Cheng [14] had performed numerical and experimental study on laminar flow through smooth trapezoidal silicon microchannel and confirmed the validity Navier-Strokes equations even at hydraulic diameter as tiny as 25.9 um. Figure 6 shown the schematic diagram of trapezoidal silicon microchannel.

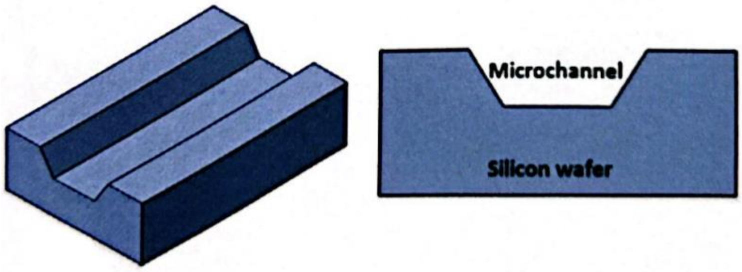


Fig. 6. Schematic diagram of trapezoidal silicon microchannel

The characteristics of fluid flow and heat transfer in microchannel heat sink with sinusoidal cavities and rectangular ribs (MCHS-SsCRR) have been examined numerically. The consolidated impacts of cavities and ribs are analysed through near with related geometries, for example, microchannel with rectangular ribs (MCHS-RR) and microchannel with sinusoidal cavities (MCHS-SsC). Moreover, an advancement examination has been performed to the geometrical parameters, for example, relative cavity amplitude (2), relative rib width (B) and relative rib length (T) to pick the ideal parameters that meet the cooling prerequisite of higher thermal performance in lower pressure drop [15].

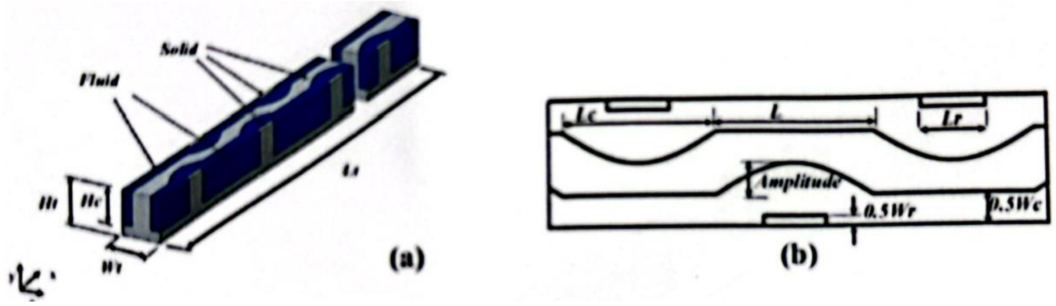


Fig. 7. Schematic diagrams of (a) MC-SsCRR and (b) Geometric parameters of MC-SsCRR

The hydrothermal performance of microchannel heat sink with secondary oblique channels in alternating directions and rectangular ribs MCHS-SOCRR has been studied numerically. The hybrid utilization features of ribs and secondary channels have been analyzed through comparative with related geometries such as microchannel with rectangular ribs MCHS-RR and microchannel with secondary channels MCHS-SOC. Moreover, an optimization analysis has been conducted to geometrical parameters to choose the optimal parameters that meet the cooling requirement of higher thermal performance in lower pressure drop. These parameters include; relative secondary channel width λ, relative rib width β and the angle of secondary channel θ [16].

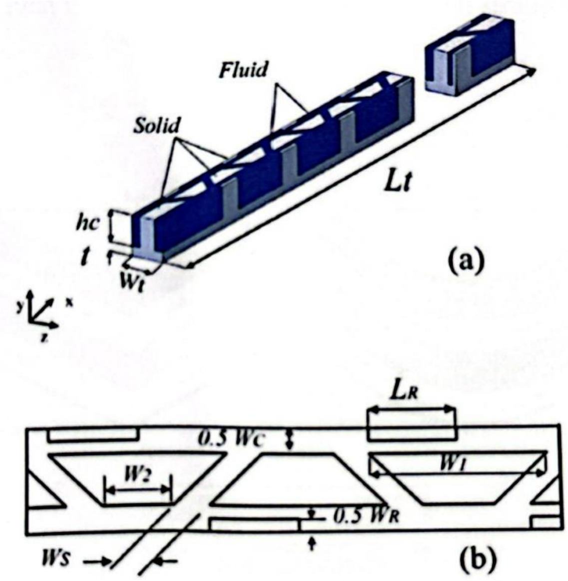


Fig. 8. Schematic diagrams of (a) MC-SOCRR and
(b) Geometric parameters of MC- SOCRR

Numerical simulations are also conducted to study the effect of channel shape on the thermal and flow fields of water flow in MCHS [17]. Three different channel shapes are considered zigzag, curvy and steps MCHS, and it is compared with straight and wavy MCHS. From the results, it is concluded that:

i. The temperature and the heat transfer coefficient of the zigzag MCHS is the least and greatest, respectively, among other channel shapes studied. The second best thermal performance is provided by curvy channels after the wavy channels.
ii. The pressure drop penalty for all channel shapes of MCHS is higher than the conventional straight MCHS.

iii. The zigzag MCHS has the highest value of pressure drop, friction factor and wall shear stress followed by the wavy, curvy and step MCHS, respectively.
iv. Thus, the zigzag MCHS is the best channel for the best thermal performance and the step MCHS is the best channel for the hydraulic performance with mode degradation of heat transfer compared to conventional straight MCHS.

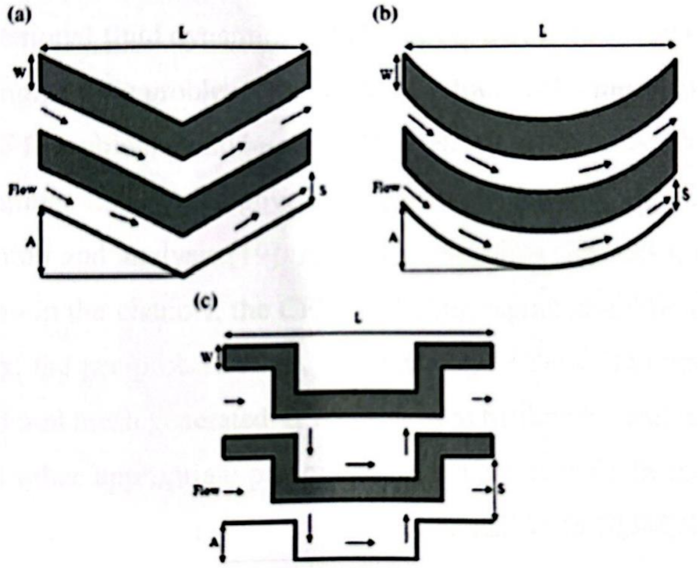


Fig. 9. A schematic of a heat exchange unite with (a) Zigzag channels, (b) Curvy channels and (c) Step channels

2.5 Flow Disruption in MCHS

The thermal boundary layer development in microchannel heat sink can be disrupted by introducing structures, dimpled ques, such as, reentrant cavities, porous medium, ribs and groove structures, dimpled surfaces, etc. These techniques disturb the thickening of boundary layers, enhanced mixing of flow at the leading edge resulting in an increased heat transfer. To fully understand this phenomenon, Hong and Cheng [18] analyzed the fluid flow characteristic and heat transfer in offset strip-fa microchannels heat sinks at different size of strip-fin for electronic cooling purpose. They successfully demonstrated the breakup of boundary layer and enhancement of heat transfer using the proposed design.

2.6 Numerical Simulation

Computational fluid dynamics (CFD) has the ability to deal with a wide range of simulating engineering problems related to heat transfer by means of the numerical solution The CFD problem could be tackled by general procedures: physical scenario, geometry, computational mesh, governing equation, solution algorithm, boundary condition, solution and analysis [19]. After understanding the heat transfer and fluid how phenomena in the channel, the CFD modelling region could be classified into a few major steps: the pre-process stage, the geometry of the CFD region constructed and a computational mesh generated. It was followed by the physical model, boundary conditions, and other appropriate parameters that were defined in the model's setup and solving stage. Finally the results could be obtained when FLUENT iterations led to converged results defined by a set of converged criteria. The temperature, heat transfer coefficient, and pressure drop across the pipe could be obtained throughout the computational domain in the post-process stage. FLUENT software is used to simulate governing equations of turbulent forced convection heat transfer with constant heat flux.

3. Methodology

In order to achieve the objectives of this paper which is to improve the heat transfer performance for microchannel heat sink, a new design will be simulate. In this section, details on the procedure and numerical conditions will be explained.

3.1 Software

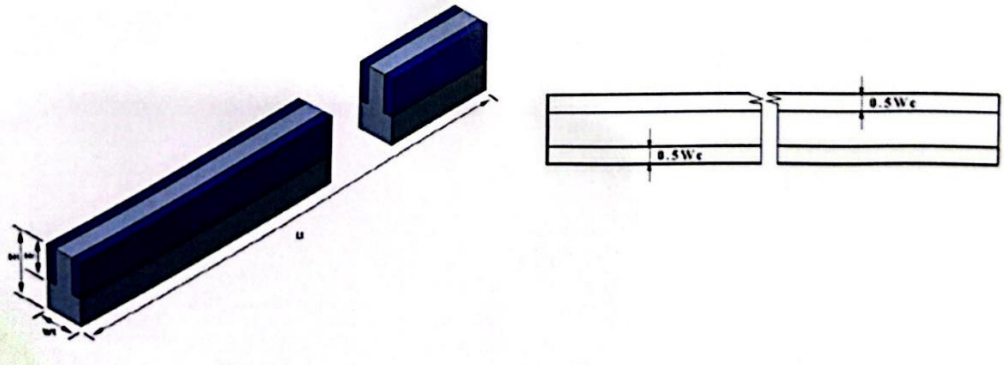
In this project, two software are used in acquiring and analyzing the data, namely:

(1) SOLIDWORKS
(2) ANSYS

SOLLDWORKS is a solid modeling computer-aided design (CAD) and computer-aided engineering (CAE) computer program that runs on Microsoft Windows. Particularly, this software used to design the shape or geometry of the model of microchannel heat sink. ANSYS is a software used to design products and semiconductors, as well as to create simulations that test a product's durability, temperature distribution, fluid movements and electromagnetic properties. This ware used to run simulation of heat transfer along the microchannel heat sink after designing according to proposed geometry.

3.2 Mathematical Modelling of Microchannel Heat Sink

The microchannel warm sink are made of copper. Keeping in mind the end god lo spar the computational cost, just a single symmetrical pieces of MCHS is embraced in present simulation. Figures 10 (a) and (b) outlines the computational domain with geometrical parameters for both of secondary channels and rectangular ribs (MCHS-SCRR) and comparing with straight rectangular microchannel MICH-SC individually. As appeared in Figures 10 (a) and (b), the main measurements MCHS such as total length Lt, total width *Wt* and total height *Ht* is 10 mm, 0.2 m and 0.35 mm respectively. The height of microchannel *Hc* is 0.2 mm and the width of the channel's wall which isolates between two channels is 0.1 mm. All the geometrical parameters are shown in Table 1.



(a) (b)

Fig. 10. Schematic diagrams of (a) MC-SCRR and (b) Geometric parameters of MC-SCRR

Table 1
Geometrical parameters of MCHS-SCRR

Geometrical parameters	Hc	Ht	Lt	Lr	La	L1	L2	L3	Wt	Wc	Wr	Wtc	Wsc	Asc (angle)
Value/µm	200	350	10,000	60	40	60	140	200	200	100	30	224	20	30°

3.3 Computational Domain and Boundary Condition

The computational has two segments, fluid and solid parts. The fluid part comprising of two chambers, each having two inlets/outlets and are connected with the flow channel. In view of these geometry, a multi-block structured mesh was produced and for each rectangular sub-domain, a “good quality" structured mesh was created to accomplish “good” numerical accuracy in terms of CPU requirements. Over the domains of the fluid and solid parts, mesh lines were associated easily to guarantee no numerical errors, because of interpolation, were introduced. A series of grids was used to obtain grid independent outcomes.

Standard hydraulic boundary conditions of uniform inflow at the two chamber inlets were applied and the thermal boundary conditions for solving heat conduction equation were:

i. The heat flux was kept constant at 1,000,000 W/m^2 and the heat source were uniformly distributed over the fluid/solid interfaces;
ii. Water at ambient temperature (300K)
iii. Adiabatic wall condition were prescribed for the remaining surfaces

3.4 Numerical Method and Solution Procedure

In this research, the commercial computational fluid dynamics (CFD) package software ANSYS version 18.1 is utilized to complete the numerical simulation which includes conjugate heat transfer. The overseeing conditions for fluid flow were solved by the conventional finite volume methodology together with the pressure-velocity coupling method, ordinarily alluded to as the SIMPLE algorithm. The working liquid in the domain was thought to be incompressible, having laminar flow characteristics with consistent material properties. The buoyancy forces and radiation effects were neglected. Heat transfer, inside the solid blocks was solved for by the heat conducting equation constraints for the flow boundary conditions.

Simulations for a series of grids at different densities were completed in order to identify a baseline mesh, which would give grid-independent solutions. For every calculation, the convergence criteria for the streamwise, spanwise and divider typical speed segments were set to accomplish a leftover of 10^{-5}, and in most cases, the residual of spanwise velocity component achieved 10^{-6} after around 200 iterations. The final baseline mesh utilized 823,620 nodes and 771,768 elements for all simulations in streamwise, spanwise and wall-normal, respectively. Figure 11 illustrates the 3D geometry of MCHS-SCRR with mesh.

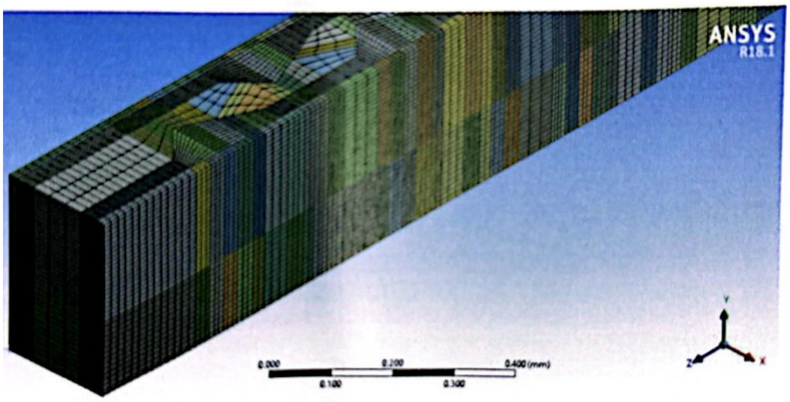


Fig. 11. Three dimensional geometry of MCHS-SCRR with mesh

3.5 Data Reduction

This segment introduces the relevant expressions that used to figure the characteristic of heat transfer and fluid flow in MCHS. The Reynolds number (Re) is expressed as follow,

$$Re = \frac{\rho u_m D_h}{\mu} \tag{1}$$

where D_h represent the hydraulic diameter and it is calculated as follow,

$$D_h = \frac{2H_c W_c}{H_c + W_c} \tag{2}$$

where ρ, μ_m, μ, H_c and W_c are fluid density, means velocity, dynamic viscosity, height of microchannel and width of microchannel respectively.

The apparent fiction factor in rectangular duct is given by

$$f_{app} = \frac{2D_h \Delta P}{L_t \rho u_m^2} \tag{3}$$

where L_t and ΔP are the total of microchannel and pressure drop across microchannel respectively.

The average Nusselt number is given by,

$$Nu_{ave} = \frac{h_{ave}(D_h)}{k_f} \tag{4}$$

where k_f is the thermal conductivity of the fluid.

4. Results and Discussion

4.1 Model Validation against the Theory

The numerical model described above was validated against a theoretical estimation for straight channel with symmetrical uniform heat Aux surfaces [19]. After obtaining the value for f_{app} of straight channel MCHS from the simulation results, it then is compared to Darcy friction factor. A comparison of friction factor by numerical prediction and the theory is shown in Figure 12.

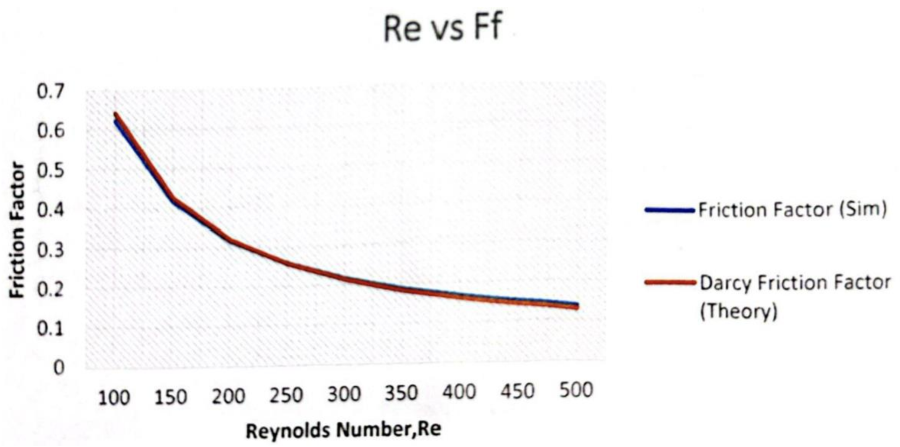


Fig. 12. Comparison of friction factor by numerical prediction and the theory

4.2 Nusselt Number and Pressure Drop

Nusselt number then compared between MCHS-SCRR and MCHS-SC. Here, numerical simulations were carried out at Reynolds Number with the range of 100 to 500, for which the local Nusselt number was defined as,

$$Nu = \frac{h(D_h)}{k} \tag{5}$$

where *h* is the heat transfer coefficient. D with subscripts "*h*" stands for characteristic length. Parameter *k* is the local thermal conductivity. The predicted local Nusselt number from the simulation can be evaluated via the heat transfer coefficient which is,

$$h = \frac{q}{T_w - T_b} \tag{6}$$

where *h* is heat transfer coefficient and *q* for heat flux. *T* is the temperature with subscript '*w*' stands for wall and '*b*' for bulk. The comparison of Nusselt number by MCHS-SC and MCHS-SCRR is depicted in Figure 13.

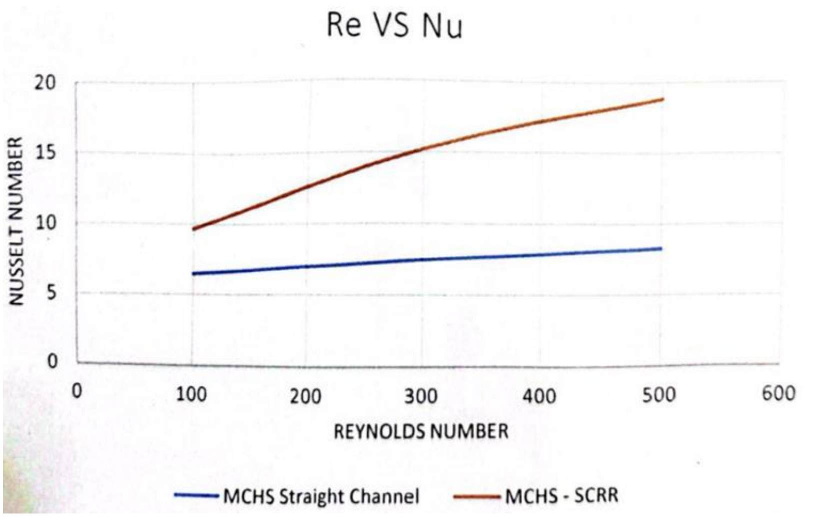


Fig. 13. Comparison of Nusselt number by MCHS-SC and MCHS-SCRR

The results of pressure drop that obtained from numerical simulation also compared between MCHS-SCRR and MCHS-SC as shown in Figure 14.

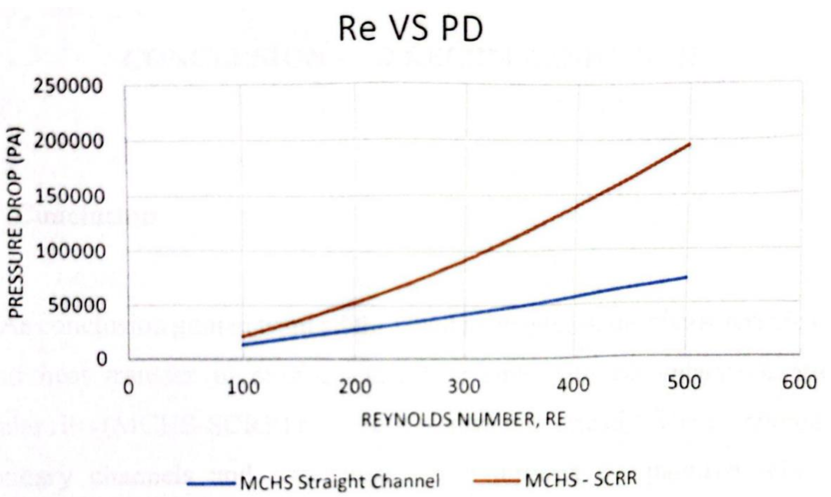


Fig. 14. Comparison of pressure drop by MCHS-SC and MCHS-SCRR

5. Conclusions

The characteristics of fluid flow and heat transfer in microchannel heat sink with secondary channels and rectangular ribs (MCHS-SCRR) have been studied numerically. The combined effects of secondary channels and ribs are analyzed through comparative with related geometries such as straight channel microchannel heat sink (MCHS-RR). The following conclusions can be extracted from the results:

i. The using of hybrid techniques between heat transfer augmentation methods such as secondary channels and rectangular ribs can be significantly enhanced overall performance in microchannel heat sink rather than straight channel.
ii. The combination of secondary channels and rectangular ribs contributed in increasing the high pressure drop which caused by ribs in MCHS-SCRR through providing smaller flow area.
iii. The proposed design promotes flow mixing and chaotic advection through the vortices formation in transverse direction of cavity.

iv. The current design provides larger heat transfer area comparative with relevant microchannel geometries which is MCHS-SC.

References

[1] Chandana, Bella Jaya, and V. Vijaya Kumar. "Design of stacked microchannel heat sinks using computational fluid dynamic." *International Journal of Advances in Engineering & Technology* 10, no. 3 (2017): 354.

[2] Ho, Chih-Ming, and Yu-Chong Tai. "Micro-electro-mechanical-systems (MEMS) and fluid flows." *Annual review of fluid mechanics* 30, no. 1 (1998): 579-612. https://doi.org/10.1146/annurev.fluid.30.1.579

[3] Chován, Tibor, and András Guttman. "Microfabricated devices in biotechnology and biochemical processing." *TRENDS in Biotechnology* 20, no. 3 (2002): 116-122.https://doi.org/10.1016/S0167-7799(02)01905-4

[4] Gromoll, Bernd. "Micro cooling systems for high density packaging." *Revue générale de thermique* 37, no. 9 (1998): 781-787. https://doi.org/10.1016/S0035-3159(98)80004-4

[5] Gao, Puzhen, Stéphane Le Person, and Michel Favre-Marinet. "Scale effects on hydrodynamics and heat transfer in two-dimensional mini and microchannels." *International journal of thermal sciences* 41, no. 11 (2002): 1017-1027. https://doi.org/10.1016/S1290-0729(02)01389-3

[6] D.B. Tuckerman, R. Pease (1981). High-performance heat sinking for VISI, IEEE Electron Dev. Lett. 2 126-129. https://doi.org/10.1109/EDL.1981.25367

[7] Che Sidik, Nor Azwadi, Muhammad Noor Afiq Witri Muhamad, Wan Mohd Arif Aziz Japar, and Zainudin A. Rasid. "An overview of passive techniques for heat transfer augmentation in microchannel heat sink." *International Communications in Heat and Mass Transfer* 88 (2017): 74-83. https://doi.org/10.1016/j.icheatmasstransfer.2017.08.009

[8] Steinke, Mark E., and Satish G. Kandlikar. "Review of single-phase heat transfer enhancement techniques for application in microchannels, minichannels and microdevices." *International Journal of Heat and Technology* 22, no. 2 (2004): 3-11. https://doi.org/10.1115/ICMM2004-2328

[9] Gunnasegaran, Prem, H. A. Mohammed, N. H. Shuaib, and Rahman Saidur. "The effect of geometrical parameters on heat transfer characteristics of microchannels heat sink with different shapes." *International communications in heat and mass transfer* 37, no. 8 (2010): 1078-1086. https://doi.org/10.1016/j.icheatmasstransfer.2010.06.014

[10] Weilin, Qu L., Gh Mohiuddin Mala, and D. Q. Li. "Pressure-driven water flows in trapezoidal silicon microchannels." *International Journal of Heat and Mass Transfer* 43, no. 3 (2000): 353-364. https://doi.org/10.1016/S0017-9310(99)00148-9

[11] Liu, Dong, and Suresh V. Garimella. "Investigation of liquid flow in microchannels." *Journal of Thermophysics and heat transfer* 18, no. 1 (2004): 65-72. https://doi.org/10.2514/1.9124

[12] Wang, Guodong, Liang Hao, and Ping Cheng. "An experimental and numerical study of forced convection in a microchannel with negligible axial heat conduction." *International Journal of Heat and Mass Transfer* 52, no. 3-4 (2009): 1070-1074. https://doi.org/10.1016/j.ijheatmasstransfer.2008.06.038

[13] Tiselj, Iztok, G. Hetsroni, B. Mavko, A. Mosyak, E. Pogrebnyak, and Z. Segal. "Effect of axial conduction on the heat transfer in micro-channels." *International Journal of Heat and Mass Transfer* 47, no. 12-13 (2004): 2551-2565. https://doi.org/10.1016/j.ijheatmasstransfer.2004.01.008

[14] Wu, H. Y., and Ping Cheng. "Friction factors in smooth trapezoidal silicon microchannels with different aspect ratios." *International journal of heat and mass transfer* 46, no. 14 (2003): 2519-2525. https://doi.org/10.1016/S0017-9310(03)00106-6

[15] Ghani, Ihsan Ali, Natrah Kamaruzaman, and Nor Azwadi Che Sidik. "Heat transfer augmentation in a microchannel heat sink with sinusoidal cavities and rectangular ribs." *International Journal of Heat and Mass Transfer* 108 (2017): 1969-1981. https://doi.org/10.1016/j.ijheatmasstransfer.2017.01.046

[16] Ghani, Ihsan Ali, Nor Azwadi Che Sidik, Rizal Mamat, G. Najafi, Tan Lit Ken, Yutaka Asako, and Wan Mohd Arif Aziz Japar. "Heat transfer enhancement in microchannel heat sink using hybrid technique of ribs and secondary channels." *International Journal of Heat and Mass Transfer* 114 (2017): 640-655. https://doi.org/10.1016/j.ijheatmasstransfer.2017.06.103

[17] Mohammed, H. A., P. Gunnasegaran, and N. H. Shuaib. "Influence of channel shape on the thermal and hydraulic performance of microchannel heat sink." *International Communications in Heat and Mass Transfer* 38, no. 4 (2011): 474-480. https://doi.org/10.1016/j.icheatmasstransfer.2010.12.031

[18] Hong, Fangjun, and Ping Cheng. "Three dimensional numerical analyses and optimization of offset strip-fin microchannel heat sinks." *International Communications in Heat and Mass Transfer* 36, no. 7 (2009): 651-656. https://doi.org/10.1016/j.icheatmasstransfer.2009.02.015