Penerbit Akademia Baru

Journal of Advanced Research Design

Journal homepage: www.akademiabaru.com/ard.html
ISSN: 2289-7984

Hydrothermal Performance in a New Designed Hybrid Microchannel Heat Sink with Optimum Secondary Channel Geometry Parameter: Numerical and Experimental Studies

Open Access

Wan Mohd. Arif Aziz Japar[1], Nor Azwadi Che Sidik[1,*], M'hamed Beriache[2]

[1] Malaysian-Japan International Institute of Technology (MJIIT), Universiti Teknologi Malaysia (UTM) Kuala Lumpur, Jalan Sultan Yahya Petra, 54100 Kuala Lumpur, Malaysia
[2] Department of Mechanical Engineering, Faculty of Technology, University of Hassiba Benbouali, Algeria

ARTICLE INFO

Article history:
Received 17 January 2019
Received in revised form 20 March 2019
Accepted 22 March 2019
Available online 27 March 2019

Keywords:
Hybrid microchannel heat sink, secondary channel, optimal parameter, heat transfer, low Reynold number

ABSTRACT

Miniaturization and utilization of low-dimensional structures of recent electronic devices have witnessed some new micro cooling methods which can fulfil the cooling demand for the electronic devices. Microchannel heat sink (MCHS) is one of the micro cooling method 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 fabricate compare to other micro cooling methods. Due to fast development in electronic industry, hybrid microchannel heat sink with optimal design has received a great deal of attention in order to provide high heat transfer performance with acceptable pressure drop. However, most of the acceptable pressure drop has required high pumping power due high friction factor at high Reynold number in a proposed hybrid design. Therefore, the aim of this article to propose a new hybrid microchannel heat sink which can obtain the high heat transfer performance with minimal friction factor at low Reynold number by introducing optimum secondary channel geometry parameter in cavities-ribs microchannel heat sink. In this study, comparative analysis was conducted between proposed hybrid design with other related designs. Besides that, optimization of secondary channel geometry was conducted in order to achieve high heat transfer rate at low Reynold number. The comparative analysis revealed that secondary channel that connected between adjacent main channel increased the degree of flow mixing which contributed to the enhancement of heat transfer performance. Besides that, proposed hybrid design with optimum geometry parameter of secondary channel achieved the high overall performance at low Reynold number.

1. Introduction

Over the past decade, the investigation of fluid flow and heat transfer characteristic induced by natural convection on thermal performance becomes a most interesting topic in a cooling system.

* *Corresponding author.*
E-mail address: azwadi@utm.my (Nor Azwadi Che Sidik)

The effectiveness of the cooling system in such application is very important to keep the temperature of a structure or electronic device from exceeding limits imposed by needs of safety and efficiency. In recent years, rapid growth in the electronic industry has witnessed a new generation high performing dense chip packages in many modern electronic devices. However, the development of more compact electronic devices that will operate at high power density causes the thermal management of electronic devices becomes a very critical issue in the electronics industry due to lack of efficient technique to remove heat from the devices [1].

During the past 30 years, many methods have been proposed in open literature in order to improve overall performance of microchannel heatsink. Generally, the methods can be categorized into two groups, active method and passive method. Most of researcher has widely used the passive method due to its low cost and absence of moving part compared to active method [2]. All of the studies reviewed in open literature demonstrates the heat transfer performance could be enhanced by using single technique of passive method [3]. However, pressure drop issue become the main constrain in the innovation of microchannel heatsink. Nowadays, many researchers have used multiple technique of passive method in single phase flow for enhancement of microchannel performance. In 2013, L. Gong et al. [4] has analysed the performance of microchannel structured by dimple and wavy shape. They revealed that the presence of the dimple structure in wavy channel could enhanced the heat transfer performance and not apparently increase the flow resistance. Li et al. [5] has presented a numerical study to investigate the combined effect of ribs and cavities on fluid flow and heat transfer characteristic in hybrid design MCHS. The analysis shows the design has obtained the optimum PF of 1.62 at Re = 500 with the friction factor ratio of 2.8 – 3.0. This characteristic is very important because the higher friction factor will require the higher pumping power due pressure drop produced by MCHS. After a year, Li and his research team [6] again study the performance of hybrid design with different geometry shape of cavities and ribs. The analysis shows all designs in their study obtained PF less than 1.45 for all Reynold number. In 2017, Srivastava and his research team [7] try to investigate the combined effect of ribs and cavities in convergent-divergent hybrid microchannel. The result shows the hybrid design obtained the lower thermal resistance compare to hybrid design of rectangular channel with ribs and cavities. Pressure drop produced by the convergent-divergent hybrid microchannel is around around 7500 kPa – 10000 kPa for Re number of 400 to 500. Ghani et al. [8] studied the combined effect of ribs and secondary channels on hydrothermal performance in hybrid MCHS. The hybrid design obtained the PF of 1.9 – 2.0 with the friction factor of 6.0 – 7.0.

Most of hybrid designs presented in open literature shows the optimum overall performance of MCHS was obtained at high Re number which has consumed high pumping power. Hence, research gap existed since there has no researcher that study about the optimum overall performance enhancement at low Re number which contributes to less pumping power consumption. The novelty of this research to propose a new potential hybrid design which can provide high heat transfer rate with low pumping power consumption. The objective of this research is to find the optimum geometry parameter of secondary channel in hybrid design microchannel heat sink which can provide high heat transfer rate with low pumping power consumption. In this study, the effectiveness of secondary channel geometry on the performance of hybrid microchannel heat sink and pumping power consumption are numerically studied by comparing the fluid flow and heat transfer characteristic in each enhanced microchannel heat sink designs. In present study, we extend our previous work [9] by optimizing the secondary channels geometry parameter in current design (TC-RR-SC MCHS) in order to enhance the overall performance of the MCHS at low Re number which contributes to the low pumping power consumption. Besides that, all analyses are extended to the wide range of Re number in order to find the optimum performance that can be achieved.

Comparative analysis between proposed design (TC-RR-SC MCHS) with related geometry such as CR MCHS, CR-RR MCHS, TC MCHS and TC-RR MCHS was conducted in order to analyse the performance enhancement that obtained by proposed design.

2. Methodology

Generally, current study is conducted by two different approaches, namely, numerical and experimental study. Numerical study is conducted to find the range of parameter that suitable for experimental study. Besides that, optimum geometry parameter also will be decided in this numerical study before the fabrication process of microchannel heat sink is conducted.

2.1. Numerical Approach

In the current research, a systematic numerical investigation (as shown in Figure 1) of hybrid design with secondary channel geometry is conducted at design analysis stage in order to analyse the effect of secondary channel geometry on hydrothermal performance. For the design development stage, CATIA software is used to design microchannel geometry that related with this research such as microchannel with conventional rectangular (CR), microchannel with rectangular rib (CR-RR), microchannel with triangular cavity (TC), microchannel with rectangular rib and triangle cavity (TC-RR), and microchannel with rectangular rib, triangular cavity and secondary channel (TC-RR-SC). In order to capture the fluid flow and heat transfer characteristic in three-dimensional model, FLUENT 17.0 is used to solve continuity equation, Navier–Stokes equation and energy equation. Pressure–velocity coupling is performed by finite volumes with the SIMPLEC algorithm. Results that obtained from simulation are compared with previous study result and theoretical formula. The acceptable error for this model is less than 10% as agreed by former researchers for simulation analysis [10-12]. After the comparative analysis is completed, another analysis is conducted at optimization process stage in order to find the optimum geometry parameter so as the maximum thermal performance can be obtained.

2.1.1. Design description

All designs in present study are made by copper material and consist of several microchannels. Generally, there have two methods which frequently adopted in numerical analysis, namely, computational domain of microchannel heat sink with single channel and computational domain of microchannel heat sink with single wall. These two methods have been adopted by many former research in order to save computational cost and time. In present study, single wall method is utilized in the numerical analysis. The computational domain for reference design, single passive technique design and hybrid passive technique design are shown in Figure 2, Figure 3 and Figure 4, respectively. The detail of parameter that illustrated in those figure is presented in Table 1.

2.1.2. Numerical analysis
2.1.2.1. Governing equation

In order to study the effect of secondary channel geometry on fluid flow and heat transfer characteristic, three dimensional numerical model is investigated according to the following assumptions:

a) Fluid flow can be assumed as continuum due to the Knudsen number (Kn) is less than (10^{-3}). So, Navier-stroke equation and non-slip boundary condition are applicable.
b) Fluid flow and heat transfer are in steady-state
c) The fluid is incompressible and Newtonian.
d) The flow is laminar
e) Thermophysical properties is constant
f) Viscous dissipation and gravitational are neglected
g) Radiation heat transfer is neglected.
h)

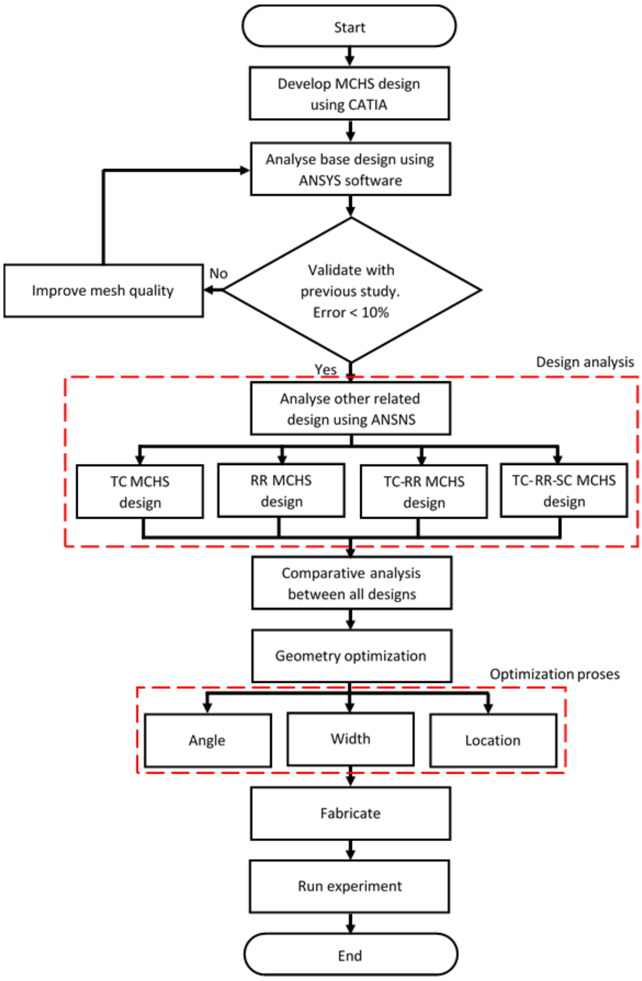


Fig. 1. Flow chart of methodology

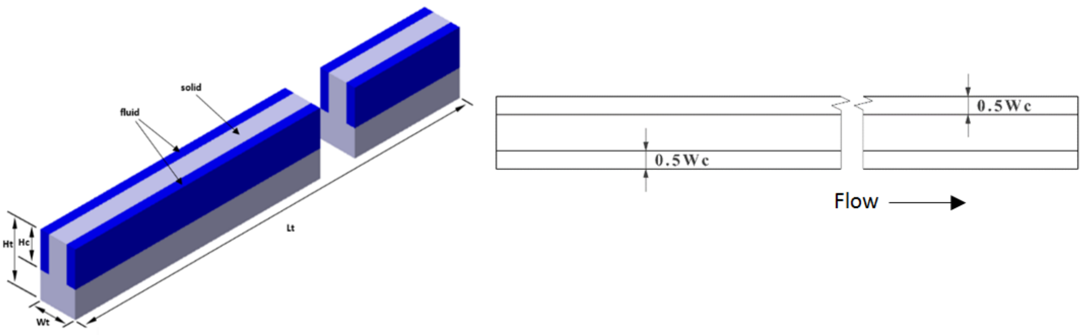
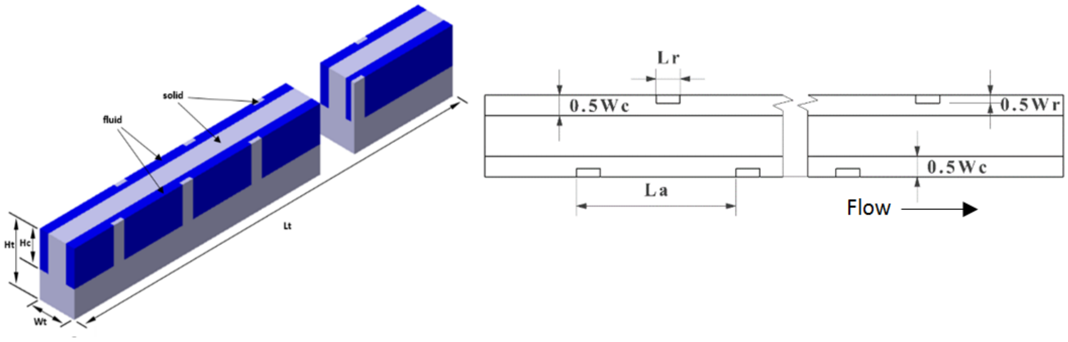
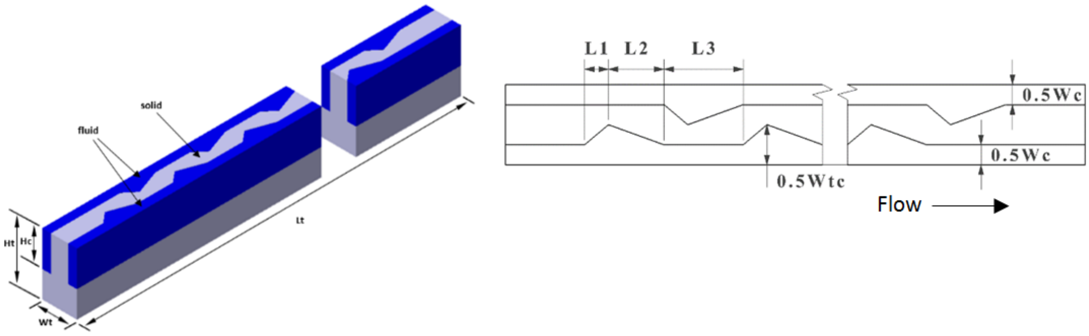


Fig. 2. Reference design (CR MCHS)

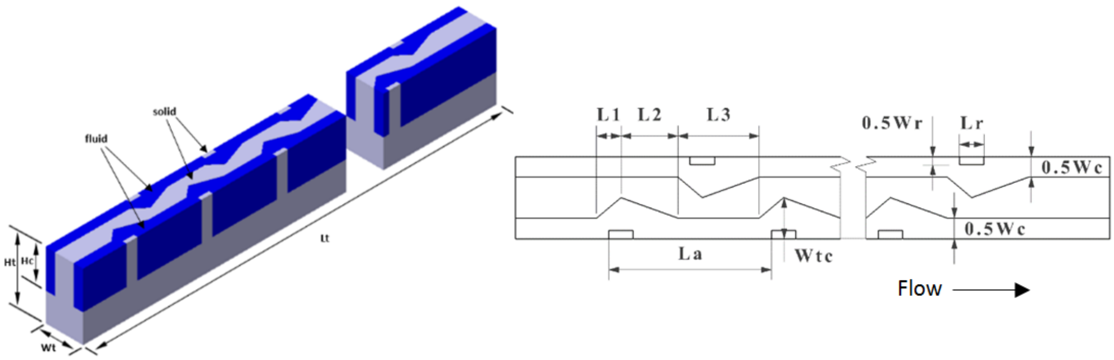


(a)

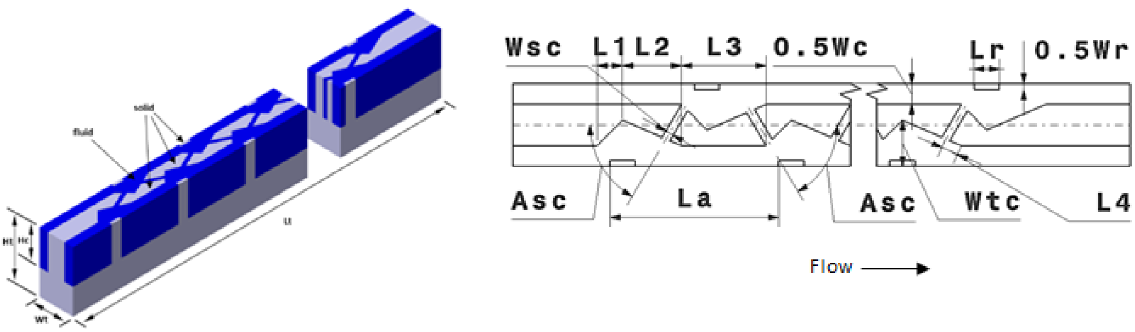


(b)

Fig. 3. Single passive technique design (a) CR-RR MCHS (b) TC MCHS



(a)



(b)

Fig. 4. Hybrid passive technique design (a) TC-RR MCHS (b) TC-RR-SC MCHS

Table 1
Geometry parameter description

Parameter	Value
Lt	10000 µm
Wt	200 µm
Ht	350 µm
Hc	200 µm
Wc	100 µm
Lr	60 µm
Wr	30 µm
La	400 µm
$L1$	60 µm
$L2$	140 µm
$L3$	200 µm
$L4$	10 µm, 25 µm, 40 µm
Wtc	112 µm
Wsc	20 µm, 30 µm, 40 µm, 50 µm
Asc	15°, 30°, 45°, 60°

These assumptions are applied on governing equations, namely, mass conservation, momentum and energy for convective heat transfer. Those equations are simplified and rewrite as follows:

Continuity equation:

$$\frac{\partial u}{\partial x}+\frac{\partial v}{\partial y}+\frac{\partial w}{\partial z}=0 \tag{1}$$

Where u, v and w are the velocity components in x, y and z-directions respectively.

Momentum equations:

$$u\frac{\partial u}{\partial x}+v\frac{\partial u}{\partial y}+w\frac{\partial u}{\partial z}=-\frac{1}{\rho_f}\frac{\partial p}{\partial x}+\frac{\mu_f}{\rho_f}(\frac{\partial^2 u}{\partial x^2}+\frac{\partial^2 u}{\partial y^2}+\frac{\partial^2 u}{\partial z^2}) \tag{2}$$

$$u\frac{\partial v}{\partial x}+v\frac{\partial v}{\partial y}+w\frac{\partial v}{\partial z}=-\frac{1}{\rho_f}\frac{\partial p}{\partial y}+\frac{\mu_f}{\rho_f}(\frac{\partial^2 v}{\partial x^2}+\frac{\partial^2 v}{\partial y^2}+\frac{\partial^2 v}{\partial z^2}) \tag{3}$$

$$u\frac{\partial w}{\partial x}+v\frac{\partial w}{\partial y}+w\frac{\partial w}{\partial z}=-\frac{1}{\rho_f}\frac{\partial p}{\partial z}+\frac{\mu_f}{\rho_f}(\frac{\partial^2 w}{\partial x^2}+\frac{\partial^2 w}{\partial y^2}+\frac{\partial^2 w}{\partial z^2}) \tag{4}$$

Where ρ_f and μ_f are the density and dynamic viscosity of the working fluid, respectively, and p is the pressure of the fluid.

Energy equation for the fluid

$$u\frac{\partial T_f}{\partial x}+v\frac{\partial T_f}{\partial y}+w\frac{\partial T_f}{\partial z}=\frac{k_f}{\rho_f c_{Pf}}\left(\frac{\partial^2 T_f}{\partial x^2}+\frac{\partial^2 T_f}{\partial y^2}+\frac{\partial^2 T_f}{\partial z^2}\right) \tag{5}$$

Where T_f is the fluid's temperature, c_{Pf} is fluid specific heat and k_f is fluid thermal conductivity.

Energy equation for the solid region:

$$0=k_s\left(\frac{\partial^2 T_s}{\partial x^2}+\frac{\partial^2 T_s}{\partial y^2}+\frac{\partial^2 T_s}{\partial z^2}\right) \tag{6}$$

where T_s is solid temperature and k_s is solid thermal conductivity.

2.1.2.2. Boundary condition

Boundary condition of hydrodynamic and thermal for all related designs in the present study are set as shown in Table 2. At the inlet of microchannel, the temperature of fluid is set to 300 K with uniform velocity for all designs. For the outlet condition, the pressure is set to atmospheric pressure (1 atm). A constant heat flux of 100 W/cm^2 is applied to the bottom of heat sink. At the fluid-solid interface, no-slip and no penetration are assumed.

2.1.2.3. Grid independence test

This test is conducted in order to find the optimum mesh structure which can provides an accurate result in lower time cost. In this study, a meshing process for all related designs are generated by ICEM software where the structured mesh is applied in all zones with similar grid size. To ensure the quality of mesh structure, the minimum orthogonal quality of grid is controlled around 0.6 while the maximum skewness of grid around 0.5. Finite volume method is utilized to discretize the governing equation. The SIMPLEC algorithm was adopted to accomplish the pressure-velocity coupling. At the same time, the second order upwind scheme is used for convective term and second order central difference scheme is applied for diffusion term. Furthermore, convergence criterions are set to be less than 10^{-6} for continuity and less than 10^{-9} for energy.

In order to find the optimum mesh structure, finer grids are generated first and analysed in ANSYS FLUENT with the highest Re number in this study which is 800. The results that obtained from the analysis are validated with correlations that produced by former researchers. If the relative error for selected parameters are less than 10% [10-12], means that, the grids are finest enough to generate an accurate result. The number of element for the finest grids is selected as a reference value for a grid independence test. The relative error is calculated according to following equation:

$$e\%=\left|\frac{J_2-J_1}{J_1}\right|\times 100 \tag{7}$$

where (J) represents selected parameters, namely, local nusselt number, average friction factor, pressure drop and, temperature different between inlet and outlet. J_1 is represents the value of selected parameters which acquired from the finest grids, whereas J_2 represents the value of selected parameters obtained from other grids that more coarser than the finest grids.

Table 2
Boundary condition of thermal and hydrodynamic

Boundary	Location	Condition
Hydrodynamic	At the fluid-solid interface	No-slip and no penetration • $u = v = w = 0$ • $-k_s(\frac{\partial T_s}{\partial n}) = -k_f(\frac{\partial T_f}{\partial n})$ where n is the coordinate normal to the wall
	At inlet, $x = 0$	• $u_f = u_{in}$ • $v = w = 0$
	At outlet, $x = L_t = 10mm$	$p_f = p_{out} = 1atm$
Thermal	At inlet, $x = 0$	• $T_f = T_{in} = 300K$ (for water) • $-k_s(\frac{\partial T_s}{\partial x}) = 0$ (for solid)
	At outlet, $x = L_t = 10mm$	• $-k_f(\frac{\partial T_f}{\partial x}) = 0$ (for water) • $-k_s(\frac{\partial T_s}{\partial x}) = 0$ (for solid)
	At top wall, $z = Ht = 0.35mm$	• $u = v = w = 0$ • $-k_s(\frac{\partial T_s}{\partial z}) = 0$
	At bottom wall, $z = 0$	$-k_s(\frac{\partial T_s}{\partial z}) = q = 100W/cm^2$
	At side wall, $y = 0$	$\frac{\partial}{\partial y} = 0$ (symmetry)
	At side wall, $y = Wt = 0.2mm$	$\frac{\partial}{\partial y} = 0$ (symmetry)

2.1.2.4. Analytic techniques

In order to analyse the effect of each geometry design on hydro-thermal performance of MCHS, comparative analysis between CR-RR, TC, TC-RR and TC-RR-SC is conducted. Performance Evaluation Criteria (PEC) for all designs are measured based on the comparison with the corresponding straight rectangular channel (CR-design). Characteristic of thermal performance, hydrodynamic performance and overall performance are obtained by Nusselt number ratio and friction factor ratio. Overall performance for each design is obtained by performance factor, Pf that can be expressed as Nusselt number ratio over friction factor ratio to the power of one third. If the value of Pf is greater than 1, it means that the geometry design has the great thermal performance and dominates the drawback of pressure drop issue that generated in channels. The condition is vice versa for the value of Pf is lower than 1. The equation of can be written as:

$$Pf = \frac{Nu/Nu_o}{(f/f_o)^{(1/3)}} \tag{4}$$

Besides that, the augmentation entropy generation number will be used as performance indicator in order to analyse the enhanced performance of each geometry based on the reduction of flow and heat transfer irreversibility. The augmentation entropy generation number can be written as:

$$N_{s,a} = \frac{\dot{S}_{gen}}{\dot{S}_{gen,o}} \tag{5}$$

Where $\dot{S}_{gen,o}$ is refer to total entropy generation rate of reference design (CR-MCHS). All microchannel design that has entropy generation number less than unity, it can be assumed as enhanced design and thus can help to reduce the flow and heat transfer irreversibility. The enhanced design is described as has a better thermal performance from the view of second law of thermodynamics. These performance indicators will be analysed in excel 2016.

2.1.2.5. Plan for interpreting results

At the early stage of analysis, the simulation model is validated with theoretical data and previous study. This validation is to ensure the error of simulation model in acceptable range (less than 10%). Performance factor and augmentation entropy generation number are used as indicator to analyse the performance enhancement in enhanced microchannel heat sink. In order to understand the phenomena and mechanism behind the performance that obtained by the enhanced MCHS, following analysis is conducted:

a) Characteristic of velocity distribution profile
b) Characteristic of pressure distribution profile
c) Characteristic of temperature distribution profile
d) Bottom temperature distribution profile
e) Surface-to-volume ratio analysis

All the illustration will be presented by Tecplot 360 EX.

2.2. Experimental approach

After completing the numerical study, proposed design with optimum geometry parameter is fabricated. All independent variable parameters are set based on the studied parameter from numerical analysis. The heat flux at the bottom of the microchannel is set to 100 W/cm^2. Distilled water is used as a coolant in this experiment. The microchannel heat sink is fabricated by copper. This approach is conducted in order to verify numerical model of proposed design. The details of test rig and specification of apparatus that used in the present study are described in next section.

2.2.1. Test rig

There have six main parts in the test rig, namely, top cover, manifold, copper block, base, cartridge heater and cartridge heater holder. The top cover is used as insulation to prevent heat dissipate from the top of microchannel heat sink. In order to meet this requirement, material with low thermal conductivity such as transparent polycarbonate (Lexan) is used in top cover fabrication. For manifold, design and material selection is very important to ensure the manifold can maintain the flow distribution between each channels and has a minimum effect on the fluid flow characteristic. In present study, the manifold is made by Acetal (Polyoxymethylene (POM)) which highly recommended in fabrication of high precision part due to have a high stiffness, low friction and dimensional stability. The manifold is attached by inlet port, outlet port and 5 thermocouples holder. The inlet and outlet port are used to guide a fluid to enter and leave manifold. The thermocouples are used to measure the temperature of microchannel heat sink and fluid temperature at the inlet and outlet of microchannel heat sink.

Copper block is designed by 3 main section, namely, upper section, middle section and lower section. The proposed design of microchannel heat sink is machined on the top surface of upper section by CNC Micromachining. At the middle section, there has a plate which used as a holder between copper block body and manifold. While at the lower section, 4 holes are drilled across a middle section to upper section. Four cartridge heaters (OMEGA-HDC00024-250W-240V) are placed in those holes in order to supply a heat flux on the bottom of microchannel heat sink. The cartridge heaters are held up transversely to a cartridge heater holder which also made by Acetal. In order to prevent a heat flux dissipate from the copper block, base that used as connecter between manifold and cartridge heater holder is made by Acetal.

2.2.2. Apparatus

Table 3 shows the list of apparatus which used in the experiment set up as illustrated in Figure 5. Three main instruments, namely, thermocouple, pressure transducer and flow meter, that used for measurement is calibrated first before they can be used in data collection process.

Table 3
Apparatus specification

Apparatus	Model	Specification
Water Distillation unit	Favorit W4L	Power supply: • 220-240V • 50/60Hz • Single phase

		Water Supply Requirement: • 1 litre/min (Min pressure 3 p.s.i) (0.2 x 10^5 NM^{-2}) Distillate output: • 4 litres per hour, single distilled
Adjustable Pump	Peristaltic Pump: • FPU500 Pump Motor: • FPU5-MT-220	Peristaltic Pump: • Flow Rate: 0.5~2280 mL/Min • Response Time: 400 msec Pump Motor: • Maximum Back Pressure: 20 psi • Tube Wall: 1.5 mm (1/16") • Fluid Temperature Range: -46~ 149°C
Filter	Swagelok-SS-4TF-LE	Pore size: 15 micron
Flowmeter	OMEGA- FLR1010	Flow Range: • 100 to 1000 mL/min Max Pressure Drop: • 6 psi
Thermocouple	TJ40-CPSS-040G	Type: • T Sheath material: • 304 SS Sheath diameter: • 1mm Upper temperature: • 260°C
Data logger	LR8431-20	Power Sources: • 100 to 240 V AC, 50/60 Hz using AC Adapter Z1005
Pressure transducer	OMEGA-MMDWU030 USBHK3ME0T8A6CE	Transducer Type: • Differential Wet/Wet Unidirectional Range: • 200 kPa Accuracy: • +/-0.08%
Regulated DC power supply	APS3005DM	Input: • 220V±10% 50Hz Output: • Voltage: 0~30V regulable • Current: 0~5A regulable
Cartridge heaters	OMEGA-HDC00024W-240V	Sheath Length: • 38.1 mm Power: • 250 watts • 49 watt/cm^2 Voltage: • 240V

2.2.3. Procedure

Figure 6 shows the actual experiment set up in this study. This experiment consists of three main process such as preparation of distilled water, heating process and fluid flowing process. In preparation of distilled water, water distillation unit (Favorit 4L) is used to prepare the 40 littre distilled water. Heating process is the process to applied heat flux on the bottom of microchannel heat sink. The heat energy is generated by four cartridge heaters (HDC00024W-240V). The temperature changes of fluid and microchannel walls are measured by five thermocouples type-T that connected to a manifold. Two thermocouples are used to measure the fluid temperature at inlet and outlet of microchannel heat sink while three thermocouples are used to measure walls temperature of microchannel heat sink at three different locations. In fluid flowing process, Peristaltic pump and flow meter are used to control volume flow rate in microchannel heat sink while filter of 15 microns is used to prevent any particle that may clog the fluid flow in microchannel heat sink. Pressure different between inlet and outlet of microchannel that produced when the fluid flow through microchannel is measured by pressure transducer (MMDWU030USBHK3ME0T8A6CE). The pressure transducer can measure a water pressure up to 200 kPa.

All the process is started with the preparation on 40 littre distilled water. Next, inlet tube is connected to Peristaltic pump, filter of 15 microns and flow meter in series arrangement as shown in Figure 6 before it connected to inlet manifold. After that, the outlet of manifold is directly connected to water collected tank by clear tube. Five thermocouples and pressure transducer are connected to manifold body and the inlet-outlet of manifold, respectively. Pump motor is switched on and desired volume flow rate is set before run the experiment. After the desired volume flow rate is obtained, the cartridge heater is switched on. The temperature changes of fluid and channel walls that measured by thermocouples are monitored from data logger. All the temperature and pressure that measured by thermocouple and pressure transducer are collected after both parameters reach steady state condition.

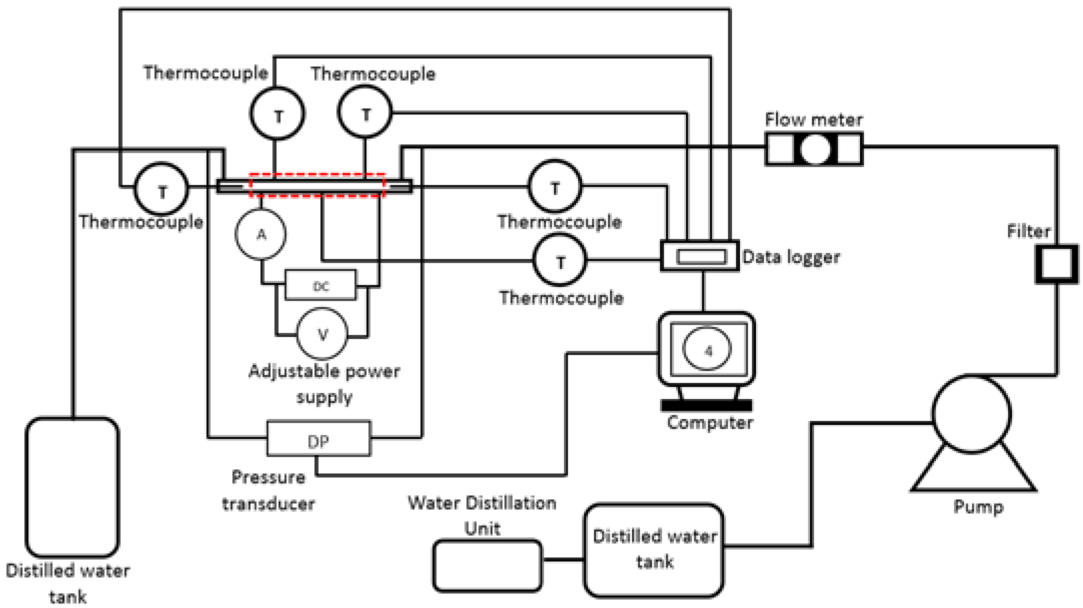


Fig. 5. Schematic diagram

Fig. 6. Experiment set up

3. Result and discussion

Overall performance for each design is obtained by performance factor, Pf that can be expressed as Nusselt number ratio over friction factor ratio to the power of one third. If the value of Pf is greater than 1, it means that the geometry design has a great thermal performance and can dominates the penalty of pressure drop issue that produced in channel. The condition is vice versa for the value of Pf is lower than 1. Figure 7 Shows the variation of performance factor with Reynold number for different geometry structure. It can be seen that CR-RR has the better performance than TC design for the Reynold number lower than 160 due to flow interruption by ribs. However, the performance that achieved by CR-RR has been predominated by TC at the Reynold number higher than 160. Although the Nusselt number ratio of CR-RR is higher than TC, pressure drop that increases with velocity in CR-RR has degraded the overall performance of CR-RR design.

In general, at the low Reynold number (Re < 150), overall performance of TC-RR is the highest compared to other design. However, the presence of secondary channel in TC-RR-SC that has similar design with TC-RR (for ribs and cavities) has shown a superior performance for 150 ≤ Re ≤ 800. The maximum value of Pf for TC-RR-SC is 1.67 at 700 ≤ Re ≤ 800, while the maximum value for TC is 1.58 at Re = 800. Pf value for TC-RR has slightly increased and persist to its maximum value of 1.36 at Re = 150 then reduce to 1.03 at Re = 800. Unfortunately, the performance of CR-RR has reduced as Reynold number increases.

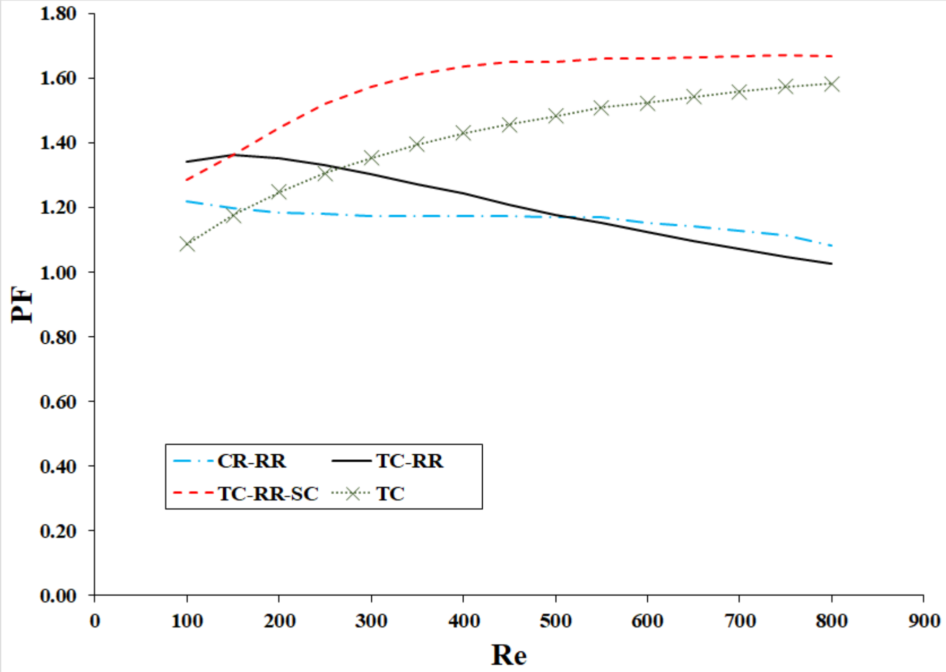


Fig. 7. Variation of Performance factor with Reynold number for different geometry structure

4. Conclusion

In present study, the effectiveness of secondary channel geometry in hybrid microchannel heat sink is investigated systematically in order to find the optimum geometry parameter of secondary channel that can provide the highest heat transfer rate at low Re number which can contributes to less pumping power consumption. This study reveals that, by introducing secondary channel, overall performance of microchannel heat sink can achieve the higher heat transfer rate in the wide range of Re number. The secondary channel geometry that attached between adjacent channel has increased the degree of flow mixing and thus improve a heat transfer coefficient which contributes to the enhancement of heat transfer performance. After all analyses are completed, experiment analysis will be started.

Acknowledgments
Authors wish to thanks Universiti Teknologi Malaysia for supporting this research activity under Takasago grant (R.K130000.7343.4B314).

References
[1] Farsad, E., S. P. Abbasi, M. S. Zabihi, and J. Sabbaghzadeh. "Numerical simulation of heat transfer in a micro channel heat sinks using nanofluids." *Heat and Mass Transfer* 47, no. 4 (2011): 479-490.
[2] Ny, G., N. Barom, S. Noraziman, and S. Yeow. "Numerical study on turbulent-forced convective heat transfer of Ag/Heg water nanofluid in pipe." *J. Adv. Res. Mater. Sci* 22, no. 1 (2016): 11-27.
[3] Lee, Y. J., P. S. Lee, and S. K. Chou. "Enhanced thermal transport in microchannel using oblique fins." *Journal of Heat Transfer* 134, no. 10 (2012): 101901.
[4] Gong, Liang, and Bo Wei. "The Characteristics of Fluid Flow and Heat Transfer in Wavy, Dimple and Wavy-Dimple Microchannels." In *Applied Mechanics and Materials*, vol. 394, pp. 173-178. Trans Tech Publications, 2013.
[5] Li, Y. F., G. D. Xia, D. D. Ma, Y. T. Jia, and J. Wang. "Characteristics of laminar flow and heat transfer in microchannel heat sink with triangular cavities and rectangular ribs." *International Journal of Heat and Mass Transfer* 98 (2016): 17-28.

[6] Li, Yifan, Guodong Xia, Yuting Jia, Dandan Ma, Bo Cai, and Jun Wang. "Effect of geometric configuration on the laminar flow and heat transfer in microchannel heat sinks with cavities and fins." *Numerical Heat Transfer, Part A: Applications* 71, no. 5 (2017): 528-546.

[7] Srivastava, Pankaj, Anupam Dewan, and Jugal K. Bajpai. "Flow and heat transfer characteristics in convergent-divergent shaped microchannel with ribs and cavities." *International Journal of Heat and Technology* 35, no. 4 (2017): 863-873.

[8] 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.

[9] Japar, Wan Mohd Arif Aziz, Nor Azwadi Che Sidik, and Shabudin Mat. "A comprehensive study on heat transfer enhancement in microchannel heat sink with secondary channel." *International Communications in Heat and Mass Transfer* 99 (2018): 62-81.

[10] Wang, Ya-Zhou, Yi-Xin Hua, and Hua Meng. "Numerical studies of supercritical turbulent convective heat transfer of cryogenic-propellant methane." *Journal of Thermophysics and Heat Transfer* 24, no. 3 (2010): 490-500.

[11] Hua, Yi-Xin, Ya-Zhou Wang, and Hua Meng. "A numerical study of supercritical forced convective heat transfer of n-heptane inside a horizontal miniature tube." *The Journal of Supercritical Fluids* 52, no. 1 (2010): 36-46.

[12] Han, Chang-Liang, Jing-Jie Ren, Wen-Ping Dong, and Ming-Shu Bi. "Numerical investigation of supercritical LNG convective heat transfer in a horizontal serpentine tube." *Cryogenics* 78 (2016): 1-13.