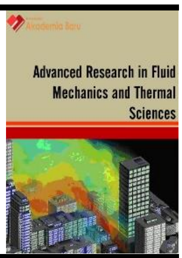
Penerbit Akademia Baru

Journal of Advanced Research in Fluid Mechanics and Thermal Sciences

Journal homepage: www.akademiabaru.com/arfmts.html
ISSN: 2289-7879



Computational Fluid Dynamic Simulation Analysis of Effect of Microchannel Geometry on Thermal and Hydraulic Performances of Micro Channel Heat Exchanger

Open Access

Chang Kai Fung[1], Mohd Fadhil Majnis[1,*]

[1] Department of Chemical and Petroleum Engineering, Faculty of Engineering, Technology and Built Environment, USCI University, 56000 Cheras, Kuala Lumpur, Malaysia

ARTICLE INFO

Article history:
Received 3 April 2019
Received in revised form 21 August 2019
Accepted 6 September 2019
Available online 25 October 2019

ABSTRACT

Micro Channel Heat Exchanger (MCHE) is one of the micro-scale equipment designed for higher efficiency in heat transfer. Among many factors, heat transfer rate and pressure drop are the most important considerations for performance of MCHE. Geometry of microchannel has significant impact to the heat transfer performance of refrigerant. In practice, rectangular and circular microchannel are most commonly used in the industry. To accurately and efficiently analyze the effect of microchannel geometry, heat transfer between refrigerant flow and air flow in segmental MCHE was modeled using Computational Fluid Dynamics (CFD) simulation. The data input for this study was obtained from the previous work which studied on the header part of automotive R134a microchannel condenser. This study was carried out to continue the previous simulation which focus on MCHE distributor. The overall performance that include both thermal and hydraulic were analyzed and compared between rectangular and circular MCHE. Parametric study to optimize the performance of MCHE was carried out to analyze effect of refrigerant velocity to the overall performance of MCHE.

Keywords:
Computational Fluid Dynamics; Distributor simulation; Geometry effect; Microchannel Heat Exchanger

1. Introduction

Nowadays, the application of MCHE has been increased in the Heating, Ventilation, Air-Conditioning & Refrigeration (HVAC&R) industry due to their high material usage and enhanced heat transfer characteristic. They are mostly used in the industries like microelectronics, aerospace, biomedical, robotics, telecommunications and automotive. The MCHE is a type of heat exchanger in which fluid flow in lateral confinements with typical dimension below 1mm which called as micro channel [1]. As surface area to volume ratio increasing, the effects of the force associated with the area (surface force, viscous forces, etc.) in micro channel will be strengthened, and this enhances the impact of axial heat conduction of micro channel wall [2]. However, there are still some challenges and limitation in improving MCHE's performance which are the uneven flow distribution, pressure

[*] *Corresponding author.*
E-mail address: fadhil@ucsiuniversity.edu.my (Mohd Fadhil Majnis)

drop and heat transfer rate. By reducing the channel dimensions to micro-scaled it will result in increment in pressure drop. Also, the heat transfer rate was limited by the heat transfer coefficient of the fluid used [3]. The performance of heat transfer and pressure drop between microchannel and minichannel has been compared by Dang *et al.,* [4] through simulation and experiment where result showed that microchannel heat exchanger is better with higher heat transfer rate. Herman *et al*., [5] found that an increase in fluid velocity can improve heat transfer performance but at the same time causing increase in the pressure drop. Design for enhanced tube with fin structure on its outer surface and helical pattern of grooves (rifling structure) on the inner bore of the tub can has higher heat transfer coefficient than conventional design [6]. Offset strip-fin is an optimal air-side arrangement for compact heat exchangers in terms of heat transfer, friction power, and compactness or size of heat exchanger [7]. Microchannel heat exchanger with fan-shaped cavities has higher heat transfer performance and the smaller pressure drop. The performance also depends on the deviation degree, coincidence degree and distribution of fan-shaped cavities [8]. Square-shaped microchannel heat sink has better pressure drop and heat transfer consistency than rectangular-shaped heat sink [9].

Computational fluid dynamics (CFD) is a science that can provide predictions that usually will be conducted under those conditions that defined in terms of flow geometry, physical properties of a fluid, the boundary and initial conditions of a flow field [10]. It also acts as a simulation tool in analyzing the fluid flows in accordance with its physical properties such as velocity, pressure, temperature, density and viscosity are conducted [11]. For instance, the Navier-Stokes equations are specified as the mathematical model of the physical case [12]. Besides, CFD also acts as a mathematical model of the physical case and a numerical method in analyzing the fluid flows. The mathematical model varies in accordance with the content of the problem such as heat transfer, mass transfer, phase change and chemical reaction [13].

Channel geometry has significant impact to MCHE. Heat exchangers with two different layouts of micro column arrays (aligned and staggered) are fabricated and their performances are compared with respect to their heat transfer capabilities at a fixed mass flow. The staggered array of micro columns maximized heat transfer in a fixed active volume and at the mean time it was more preferred due to its minimized pressure drop across the heat exchanger [14]. Study from Hasan *et al.,* which conducted with several Reynolds number condition showed that at lower Reynolds number, *Re*, circular shaped channel has the highest effectiveness while at higher *Re* the triangular channels become the best followed by trapezoidal and then the circular. This is due to the effect of entrance region since the entrance length is longer in triangle and trapezoid than the circular channel. Relationship has made where the effect of entrance region increases with increase in Reynolds numbers. In general, the effectiveness decreases with increasing *Re* due to increase of flow velocity and at the same time decreasing the residence time inside the micro channel [15].

The objectives of this research focus on modeling the segmental microchannel heat exchanger and simulation of refrigerant and air flow to provide accurate prediction of heat transfer between these two streams. The modeling studies focus on the distributor of MCHE which consist of a set of fins and a microchannel tube bundle. The effects of heat transfer rate and pressure drop were investigated for both rectangular and circular microchannel with the same working fluids and boundary conditions. Finally, parametric study on effect of refrigerant velocity to overall performance was conducted in segmental MCHE.

2. Methodology of CFD of Evaluating Performance Micro Channel Design

Figure 1 shows overall flow chart of methodology. The CFD simulation of MCHE started with preliminary study where step of data collection for the data input of the MCHE modelling.

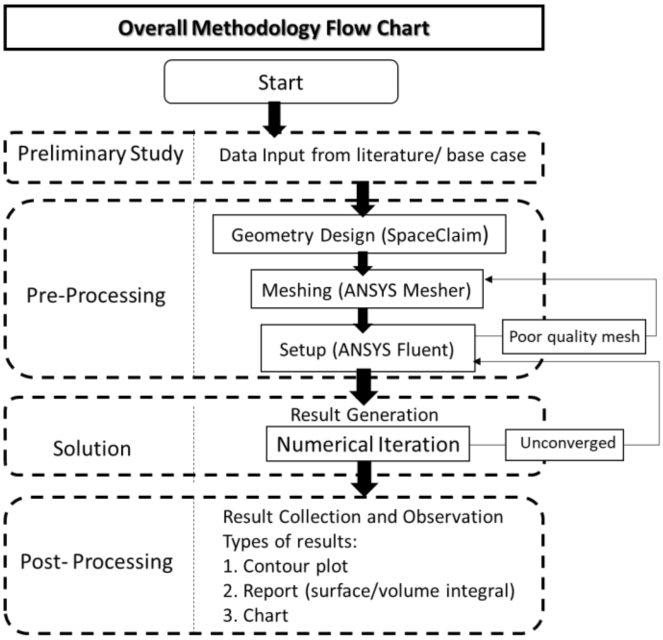


Fig. 1. Overall Methodology Flow Chart

2.1 Preliminary Study

Due to the difficulty of MCHE fabrication with limited availability of technology equipment, no experimental analysis has been done for this study. Therefore, the data input for this study was obtained from the research work by Huang *et al.,* [16] on the maldistribution of R134a stream in the header of MCHE. Table 1 shows the specification of the R134a automotive micro channel condenser and boundary conditions of working fluid.

Table 1
Specification of the R134a automotive micro channel condenser and boundary conditions of working fluid

Parameter	Value
Number of tubes	35
Tube length (mm)	660
Tube height (mm)	2
Tube width (mm)	17
Port height (mm)	0.77
Port width (mm)	1.24
Ports per tube (mm)	10
Vertical Spacing (mm)	10.89
Fin type	Louver
Fin Thickness (mm)	0.08
Fins per inch	17
Inlet header height (mm)	142.35
Inlet header width (mm)	20
Inlet header depth (mm)	12
Inlet connecting tube diameter (mm)	10.35
R134a inlet pressure (kPa)	980-1867
R134a inlet temperature (k)	338- 367
R134a inlet mass flow rate (g/s)	19.2- 29.7
Air volume flow rate (m3/s)	0.35- 0.71
Air inlet temperature (K)	297.5- 318.4
Air inlet RH (%)	20- 50

The CFD simulation of refrigerant (R134a) flows in the header of MCHE was performed by Huang *et al.,* [16] and the velocity contour plot obtained as illustrated in Figure 2. From the velocity contour plot, it can be concluded that the velocity of R134a was approximately constant at 6.5 m/s.

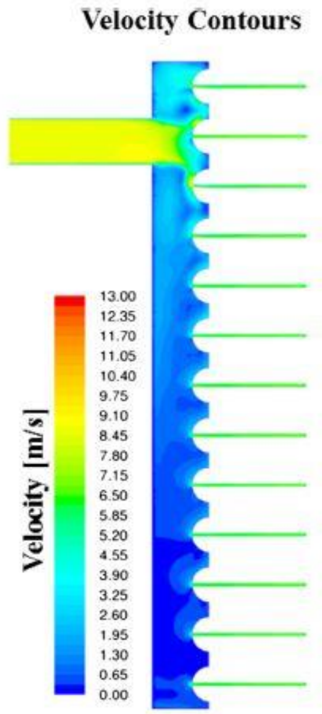


Fig. 2. Velocity contour plot of refrigerant flow in header of MCHE [16]

2.2 Geometry Design and Meshing

Segmental MCHE which includes three domains (microchannels, tube and fins) was developed according to the specifications listed in Table 1. The hydraulic diameter was calculated and the same hydraulic diameter was used for circular microchannel geometry design so that both of the geometries are studied under same Reynolds number, *Re* condition. The hydraulic diameter for both rectangular and circular microchannel are 0.95mm and same *Re* which is 1893. The *Re* for this study must be in laminar condition which *Re*<2300.

$$Dh = \frac{2(w \times h)}{(w+h)} \tag{1}$$

$$Re = \frac{\rho\, v\, Dh}{\mu} \tag{2}$$

Both geometries were developed by using SpaceClaim as shown in Figure 3. Then, meshing was done for both rectangular and circular microchannel domain with total of 479,043 cells in rectangular and 503,073 cells in circular microchannel. Quality of mesh shown in mesh metric of skewness which were 0.39 and 0.29 for rectangular and circular microchannel respectively. A simplified schematic of MCHE domain with mesh was shown in Figure 4 (a) rectangular (b) circular.

2.3 Model Setup and Solutions

The segmental MCHE was simulated using ANSYS Fluent. In Fluent, the governing conservation equations for the mass, momentum and energy are solved using the finite volume method and non-staggered grid discretization. The pressure-velocity coupled steady-state incompressible Navier-Stokes equations are solved using the SIMPLEC scheme and laminar model for both air and R134a flow. The boundary condition for R134a at microchannel inlet is at velocity of 6.5 m/s and temperature of 350K while the boundary condition of air at fin inlet is at velocity of 3 m/s and 300K. Assumption was made for outlet of microchannels and fins at atmospheric pressure condition. The solution was initialized with the patch of inlet velocity of 3m/s at fins inlet and 6.3 m/s at microchannels inlet. Patch function is to initially fill up the inlet surfaces evenly with specific velocity before the numerical solution. The energy and viscous model were used in iterating the solutions to convergence where rectangular and circular microchannel used 174 iterations and 168 iterations respectively to converged.

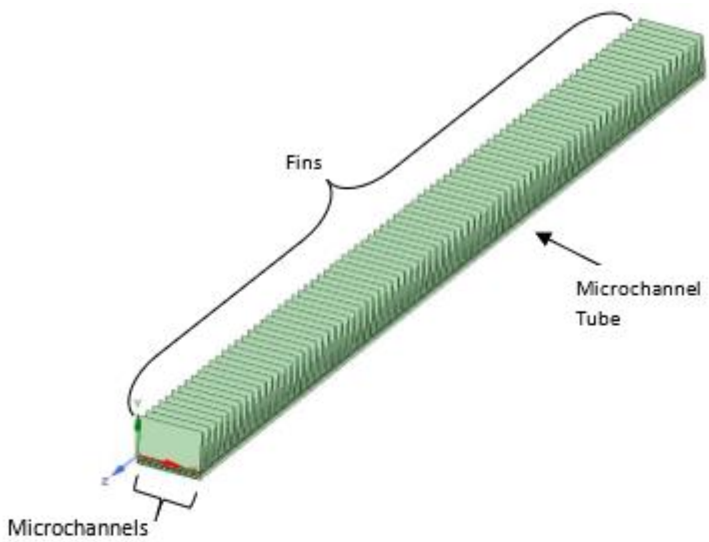
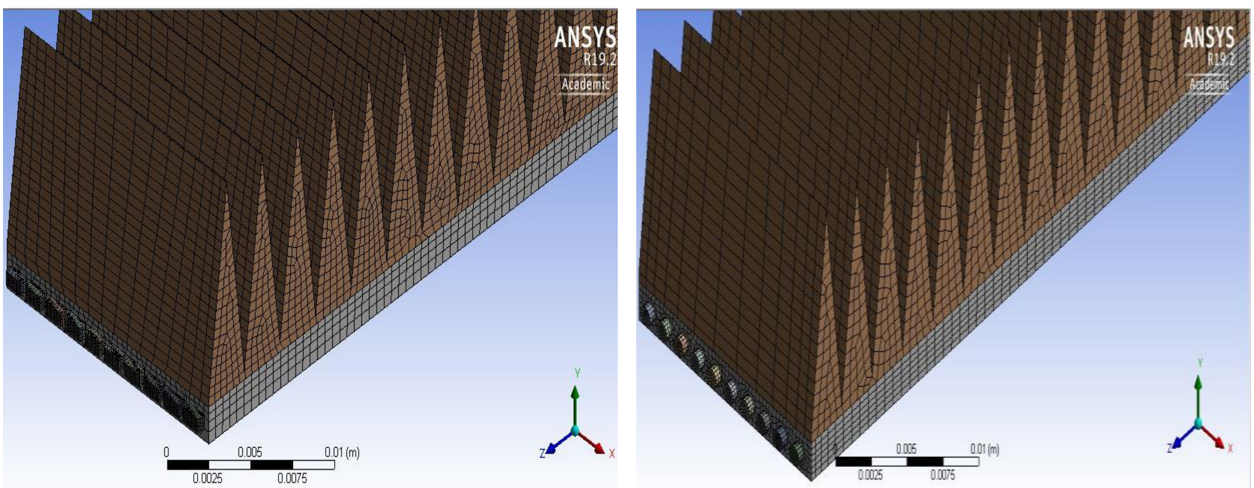


Fig. 3. Geometry of MCHE computational domain



(a) (b)

Fig. 4. A simplified schematic of MCHE domain with mesh (a) mesh of Rectangular Microchannel domain, and (b) mesh of Circular Microchannel domain

2.4 Post-Processing

The results from Fluent including temperature difference, mass flow rate and pressure drop were then used for the performance analysis for both rectangular and circular microchannel. The performance analysis takes in consideration of thermal and hydraulic performance. Both geometries were lastly compared in term of the overall performance index, ɳ. Thermal performance was analyzed in term of the effectiveness of heat exchanger, ε in transferring the heat to other medium. The effectiveness, ε is the ratio of actual heat transfer rate, Q over maximum possible heat transfer rate, Q_{max}.

$$Q = m\ Cph\ (Thi - Tho) = m\ Cpc\ (Tco - Tci) \quad (3)$$

$$Qmax = Cmin\ (Thi - Tci) \quad (4)$$

$$\varepsilon = \frac{Q}{Qmax} \quad (5)$$

Hydraulic performance was analyzed in term of pressure drop where the pressure at inlet and outlet can be straight obtained from the results in simulation. The pressure drops in both fins and microchannels are the pressure difference between inlet and outlet.

$$\Delta \mathrm{P} = \mathrm{Pinlet} - \mathrm{P\ outlet} \quad (6)$$

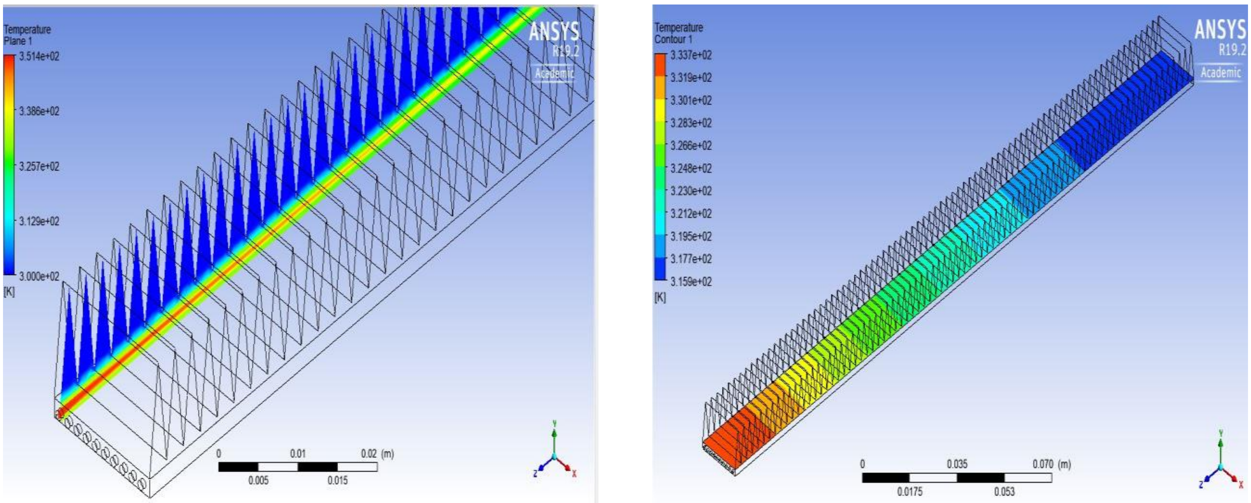
Overall performance of MCHE were analyzed in terms of both effectiveness and pressure drop. Where the term used for overall performance is the performance index, ɳ. Performance index is the ratio of effectiveness over pressure drop.

$$\eta = \frac{\varepsilon}{\Delta \mathrm{P}} \quad (7)$$

3. Results

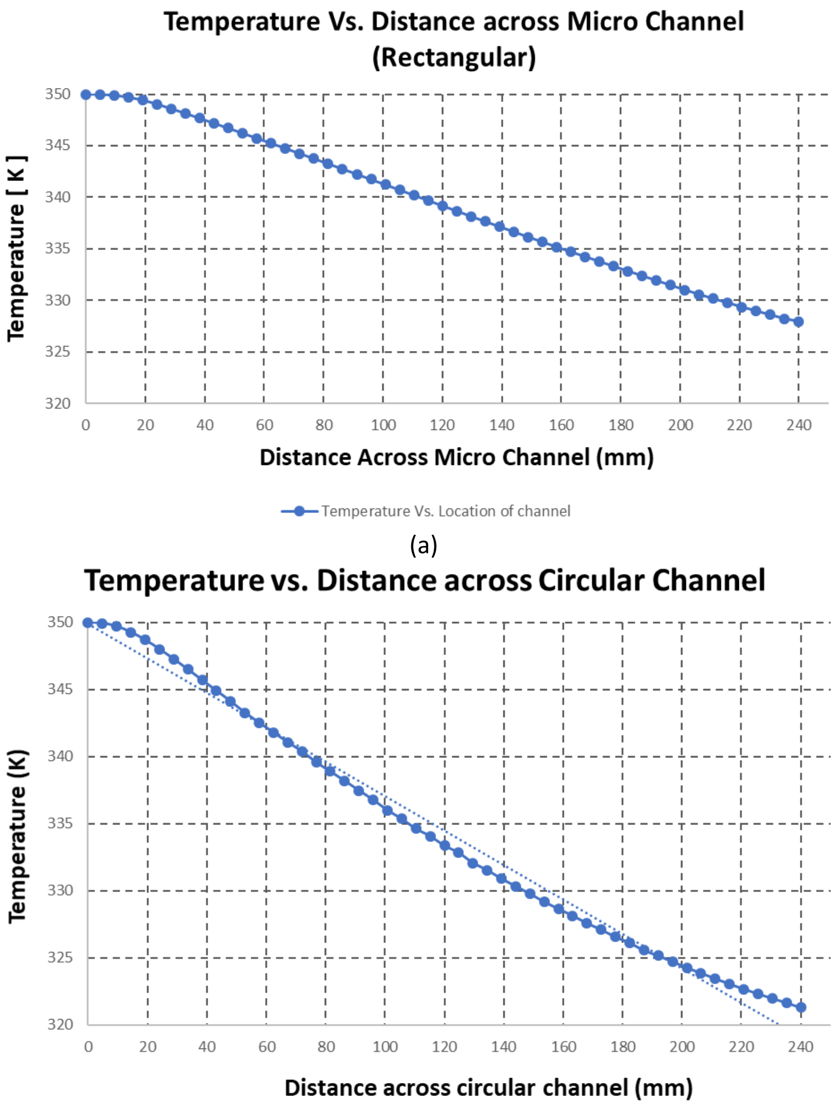
The analysis of the results of this simulation can be observed through the contour plot created in CFD-Post which includes velocity contour, pressure contour and temperature contour. In this study, the significant changes in temperature is observed between two streams which indicates the occurrence heat exchange. Figure 5(a) shows temperature contour at YZ plane of microchannel & fin domains and Figure 5(b) shows temperature contour at XZ plane of microchannel domain. From these two figures, the heat transfer trend can be observed clearly and precisely.

The changes in temperature of refrigerant, R134a along microchannel was shown in Figure 6(a) and (b) for rectangular microchannel and circular microchannel respectively. Figure 6(a) shows the temperature of R134a decreased constantly as it moved from the inlet to the end of the microchannel. Starting from the inlet 20mm, the temperature does not have any changes due to the heat just started to dissipate from R134a to the air. Overall the temperature dropped from 350K to the end(240mm) at temperature of 327.96K. Figure 6(b) shows from distance at inlet until 40mm the heat was transfer slowly from 350K to 345K and from distance of 40mm until 190mm the temperature decreased gradually from 345k to 325K and lastly the heat transferred slow down again from distance of 190 to end(240mm) with temperature change from 325K to 321.28K.



(a) (b)

Fig. 5. Temperature contour at (a) YZ plane of microchannel & fin domains, and (b) XZ plane of microchannel domain



(b)

Fig. 6. Graph of temperature vs distance across microchannel (a) Rectangular Microchannel, and (b) Circular Microchannel

Figure 7 shows the velocity profile of laminar flow in a smooth pipe whereby the velocity is highest at the middle of the flow and slowly decrease with the distance away from middle of the fluid flow until it touches the wall. Figure 8 is the velocity contour created in the microchannel in this study which exhibits the same velocity profile of laminar flow.

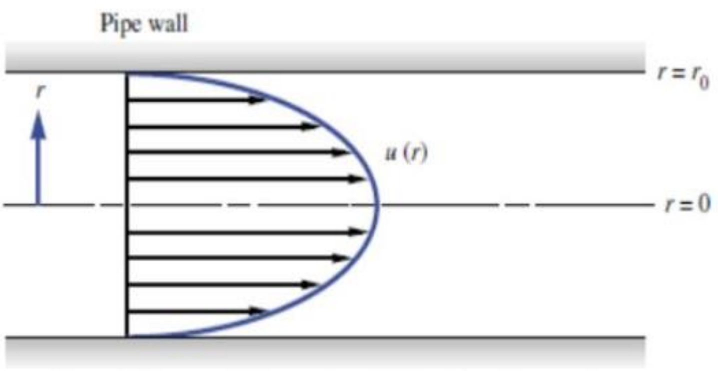


Fig. 7. Velocity profile of a laminar flow in a smooth pipe

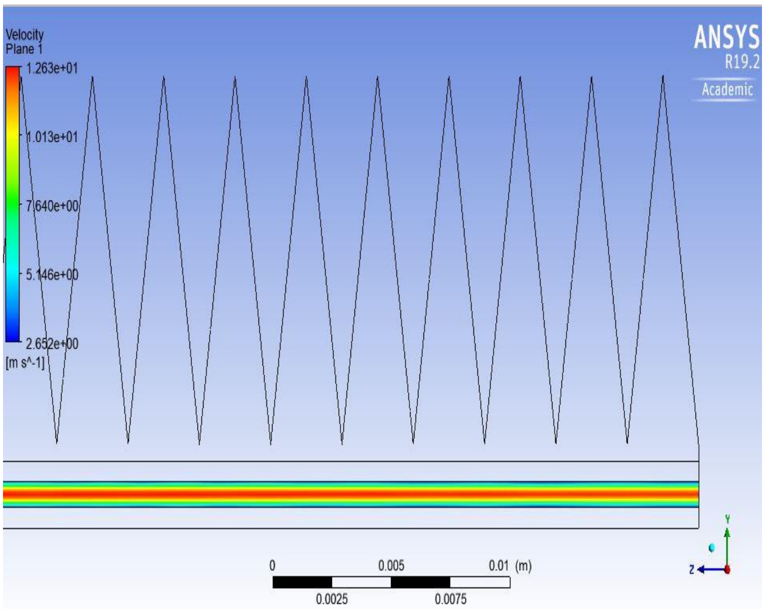


Fig. 8. Velocity contour of R134a flow in microchannel

To analyze and compare the performance of rectangular and circular microchannel heat exchanger, results were obtained, calculated and tabulated in Table 2. From Table 2, the performance of both rectangular and circular MCHE can be compared in terms of effectiveness, pressure drop and performance index. Circular microchannel has higher heat exchanger effectiveness (0.6284) but higher pressure drops (885.72 Pa). Compare to rectangular microchannel it has lower pressure drop (805.51 Pa) but lower heat exchanger effectiveness (0.5494). Therefore, the performance index was used as overall performance evaluation for both geometries where circular microchannel has higher performance index of 0.000709 m^2/N than rectangular microchannel with 0.000682 m^2/N. It was concluded that circular channel is a better geometry for MCHE.

Table 2
Performance of Rectangular & Circular microchannel heat exchanger

MCHE Geometry Design	Rectangular		Circular	
Fluid	R134a Inlet	R134a Outlet	R134a Inlet	R134a Outlet
T (K)	350	322.5	350	318.6
P (Pa)	805.51	0	885.72	0
ΔT (K)	27.5		31.4	
Cp (J/kg.K)	1000.05		1000.05	
Q (J/s)	7.23		5.97	
ΔTmax(K)	50		50	
Qmax (J/s)	13.15		9.5	
Effectiveness, ε	0.5494		0.6284	
ΔP (Pa)	805.51		885.72	
Performance Index, n	0.000682		0.000709	

Next, parametric study on effect of refrigerant velocity to the performance index was conducted within laminar flow regime with refrigerant velocity of 3.5, 4.5, 5.5, 6.5 and 7.5 m/s in circular micro channel. The flow condition for each refrigerant velocity was tabulated in Table 3. The result of parametric study was plotted into graph in Figure 9.

Table 3
Flow condition of R134a in Circular microchannel

Velocity of R134a (m/s)	Density, ρ (kg/m3)	Hydraulic Diameter, Dh (m)	Viscosity, μ (Pa.s)	Reynold's Number
3.5	4.25	9.50E-04	1.39E-05	1019.57
4.5	4.25	9.50E-04	1.39E-05	1310.88
5.5	4.25	9.50E-04	1.39E-05	1602.18
6.5	4.25	9.50E-04	1.39E-05	1893.49
7.5	4.25	9.50E-04	1.39E-05	2184.79

From the graph, it shows the relationship of performance index is decreasing gradually with velocity of R134a. This can be explained with lower R134a velocity, longer residence time for R134a to dissipate the heat to the air steam and therefore the higher the heat transfer rate.

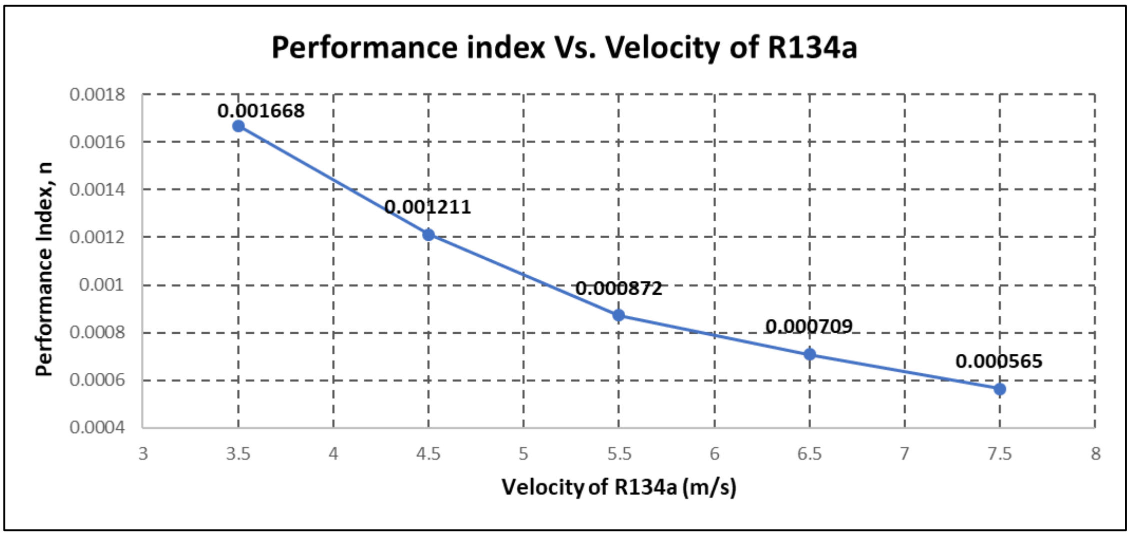


Fig. 9. Graph of performance index vs velocity of R134a

4. Conclusions

Overall, four objectives listed for this study which including

i. To develop the computational domain of Micro Channel Heat Exchanger (MCHE) with various micro channel geometry.
ii. To simulate the refrigerant flow for heat transfer in MCHE using ANSYS Fluent.
iii. To compare the results simulated in ANSYS Fluent for both geometries in terms of thermal and hydraulic performance.
iv. To conduct parametric study on the effect of refrigerant velocity to the heat transfer performance.

were successfully achieved with quality results which brought another useful research in micro channel heat exchanger (MCHE). This study has brought contribution specifically in geometry of MCHE which can help to increase the performance in real working industry like pharmaceutical, air conditioning system and automotive.

The modeling of MCHE with different geometry was completed using SpaceClaim and this brought the ease in study the geometry effect.

Next, the simulation of refrigerant and air flow for heat transfer was also successfully conducted using ANSYS Fluent. The contour plots showed in CFD-Post enhance the understanding of the heat transfer pattern and trend from refrigerant to air flow.

Lastly, all the CFD results help to identify the better geometry design of MCHE through the comparison of their overall performance. Circular micro channel has proven to be the better geometry due to its higher overall performance index. Parametric study on effect of R134a velocity also help in optimizing the performance of MCHE whereby lower R134a velocity has higher preference.

The case study presented in this paper focus on modeling and simulation of microchannel condenser where this method can be applied to microchannel evaporator. Also, this study focuses on the geometry of microchannel in affecting the performance of MCHE and future study can be extended to study effect of fins angle and fins configuration. The proposed geometry can be used as reference in designing better microchannels and improve overall performance of MCHE.

Acknowledgement
Appreciation to UCSI PSIF grant Proj-In-FETBE-056, supervisor, Mr Fadhil and Co-supervisor, Dr. M. Taufik M. Sallehuddin (Petronas Research Sdn. Bhd.) for the supports and guidance in completing this study.

References

[1] Gregory Swift, Albert Migliori, and John Wheatley. "Construction of and measurements with an extremely compact cross-flow heat exchanger." *Heat transfer engineering* 6, no. 2 (1985): 39-47.

[2] Huazhao, L., Feng, W., Huang, L. J., & Yuan, G., 2015. *U.S. Patent No. 9,115,939*. Washington, DC: U.S. Patent and Trademark Office.

[3] Satish G. Kandlikar, and William J. Grande. "Evaluation of single-phase flow in microchannels for high heat flux chip cooling—thermohydraulic performance enhancement and fabrication technology." *Heat transfer engineering* 25, no. 8 (2004): 5-16.

[4] Thanhtrung Dang, and Jyh-Tong Teng. "Comparisons of the heat transfer and pressure drop of the microchannel and minichannel heat exchangers." *Heat and mass transfer* 47, no. 10 (2011): 1311-1322.

[5] Cila Herman, and Eric Kang. "Heat transfer enhancement in a grooved channel with curved vanes." *International Journal of Heat and Mass Transfer* 45, no. 18 (2002): 3741-3757.

[6] Ratnesh Tiwari, Rohit S. Andhare, Amir Shooshtari, and Michael Ohadi. "Development of an additive manufacturing-enabled compact manifold microchannel heat exchanger." *Applied Thermal Engineering* 147 (2019): 781-788.
[7] Naef A.A. Qasem, and Syed M. Zubair. "Compact and microchannel heat exchangers: A comprehensive review of air-side friction factor and heat transfer correlations." *Energy conversion and management* 173 (2018): 555-601.
[8] Minqiang Pan, Hongqing Wang, Yujian Zhong, Minglong Hu, Xiaoyu Zhou, Guanping Dong, and Pingnan Huang. "Experimental investigation of the heat transfer performance of microchannel heat exchangers with fan-shaped cavities." *International Journal of Heat and Mass Transfer* 134 (2019): 1199-1208.
[9] Razali, A. A., and A. Sadikin. "CFD simulation study on pressure drop and velocity across single flow microchannel heat sink." *Journal of Advanced Research Design* 8, no. 1 (2015): 12-21.
[10] Hu, Howard H. "Computational fluid dynamics." In *Fluid mechanics*, pp. 421-472. Academic Press, 2012.
[11] Schaldach, G., L. Berger, I. Razilov, and H. Berndt. "Computer simulation for fundamental studies and optimisation of ICP spray chambers." *ISAS (Institute of Spectrochemistry and Applied Spectroscopy) Current Research Reports, Berlin, Germany* (2000).
[12] Jameson, Antony, Wolfgang Schmidt, and Eli Turkel. "Numerical solution of the Euler equations by finite volume methods using Runge Kutta time stepping schemes." In *14th fluid and plasma dynamics conference*, p. 1259. 1981.
[13] Bengt Sundén. "Computational heat transfer in heat exchangers." *Heat Transfer Engineering* 28, no. 11 (2007): 895-897.
[14] J. J. Brandner, E. Anurjew, L. Bohn, E. Hansjosten, T. Henning, U. Schygulla, A. Wenka, and K. Schubert. "Concepts and realization of microstructure heat exchangers for enhanced heat transfer." *Experimental Thermal and Fluid Science* 30, no. 8 (2006): 801-809.
[15] Mushtaq I. Hasan, Abdul A. Rageb, M. Yaghoubi, and Homayon Homayoni. "Influence of channel geometry on the performance of a counter flow microchannel heat exchanger." *International Journal of Thermal Sciences* 48, no. 8 (2009): 1607-1618.
[16] Long Huang, Moon Soo Lee, Khaled Saleh, Vikrant Aute, and Reinhard Radermacher. "A computational fluid dynamic and effectiveness-NTU based co-simulation approach for flow mal-distribution analysis in microchannel heat exchanger headers." *Applied Thermal Engineering* 65, no. 1-2 (2014): 447-457.