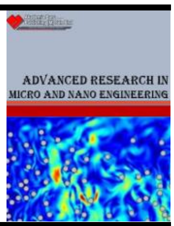
Penerbit Akademia Baru

Journal of Advanced Research in Micro and Nano Engineering

Journal homepage: https://akademiabaru.com/submit/index.php/armne
ISSN: 2756-8210



Heat and Flow Profile of Nanofluid Flow Inside Multilayer Microchannel Heat Sink

Mohamad Hafzan Mohamad Jowsey[1], Natrah Kamaruzaman[1,*], Mohsin Mohd Sies[2]

[1] School of Mechanical Engineering, Faculty of Engineering Universiti Teknologi Malaysia 81310 UTM Johor Bahru Johor, Malaysia
[2] School of Chemical Engineering, Faculty of Engineering Universiti Teknologi Malaysia 81310 UTM Johor Bahru Johor, Malaysia

ABSTRACT

The purpose of this study is to investigate the heat and flow profile of nanofluid flow inside multilayer microchannel heat sink. Two different cooling fluid; water and water-based Aluminum Oxide is compared. The heated surface of the heat sink is set as constant heat flux of 200W/cm^2 and mass flow rate is varied as 20, 40, 60 and 80kg/hr. From the results, it shows that the use of Al_2O_3 nanofluid is better in terms of heat transfer compared to water because the percentage of increment of Nusselt number is higher (83%) compared to the percentage of increment of friction factor (23.5%). Therefore, Al_2O_3 nanofluid is better in terms of heat transfer compared to water but the pressure drop is still a drawback as it is higher compared to water.

Keywords:
Aluminum Oxide, Multilayer microchannel heat sink, Thermal-hydraulic Performance

1. Introduction

Technologies in this century are integrated with computerized systems that are designed to work in a more complicated and complex manner. The past decade also shows a minimizing trend in terms of size of devices but still maintaining their purpose yet, increasing their functionality. Reduction in size and increased functionality, increases the workload. Hence, heat generation by devices raises a point of concern as it needs to be regulated for device to work optimally. Therefore, the use of micro channels has been adopted in order to regulate heat flux generated by electronic components [1,2]. Multilayered microchannel is more convenient to use compared to single layer microchannel due its better heat dissipating properties [3-5]. Vafai *et al.*, used a numerical study to show that two-layer microchannels had greater advantages than single layer microchannels [6].

The use of coolants also signifies the difference in heat transfer performance. Recent studies found improvement in terms of heat transfer when incorporating the use of nanofluids instead of water. Nanofluids are made by suspending nanoparticles in based fluid to be used as coolants. An investigation done by Wen and Ding [7] on the heat transfer enhancement of flowing nanofluid through copper tube with laminar flow showed that when 1.6% volume fraction of nanoparticles were dispersed in water, an enhancement up to 47% in terms of Nusselt number was observed.

* *Corresponding author.*
E-mail address: natrahk@utm.my

However, in terms of pressure drop, there seems to be an increase for nanofluids compared to water. Fotukian and Esfahany [8] in an investigation on pressure drop and convective heat transfer of nanofluid based in water with volume fraction less than 0.2% inside a circular tube at Reynolds < 35,000 resulted in a significant increase in heat transfer but at the same time the pressure drop of the nanofluid was much larger than the base fluid. Therefore, this study aims to see the heat transfer performance and pressure drop of nanofluids when incorporated with multilayered microchannels.

2. Methodology

This research will be conducted using a multilayered micro-channel which has a better thermal performance than a normal single layered micro-channel. However, in this research, the focus will be more on the working fluid which is Aluminium Oxide nanofluid coolants that flow through the micro-channel. This is because nanofluids are comprised of nano sized particles that have good thermal conductivity suspended in a base fluid [9].

The model of the multilayered microchannel was referred from Kamaruzaman *et al.*, [10]. Further research done on 3 arrangements of multilayer microchannel arrays (model 1, model 2 and model 3) found that the following model (model 2) was the most efficient. The arrangement of the model has 4 layers where the 1st and 3rd layer consists of 8 microchannel, while the 2nd and 4th layer had 7 microchannels placed in between the top layers. Therefore, for this study, the following model will be used in the analysis and simulation of nanofluids.

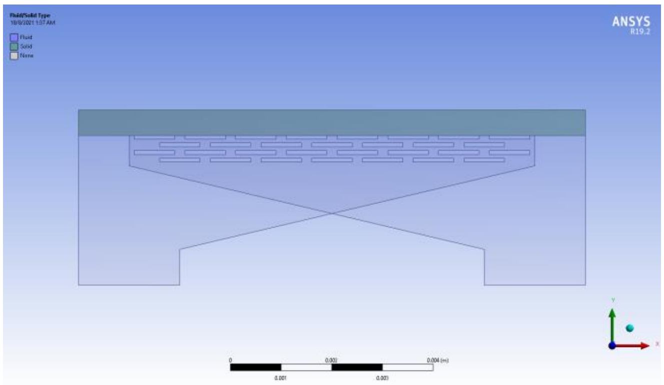


Fig. 1. Multilayer microchannel geometry

The dimension of the model is 2.95mm in depth, 1cm in length and the width is 0.6mm while the detailed dimension of the microchannel is 100um in height, 200 um in width with length of 800 um. The validation model and the tested model will be of the same shape geometry and arrangement as the reference model. The validation model will be tested with water to compare the accuracy with past research before proceeding with the tested model. The tested model will be using Aluminum Oxide Al_2O_3 based with water. The properties and material used for the multilayer microchannel are stated below:

Copper
Density : 8940 kg/m
Thermal Conductivity : 401 W/m.K (at 300K)
Specific Heat : 385 J/kg.K (at 298K)

In this research, a 4% - volumetric fraction of Aluminum Oxide Al_2O_3 based with pure water will be used whereby its properties are referred from Yu *et al.*, [11]. The properties of the water and Aluminum Oxide Al_2O_3 are as stated below:

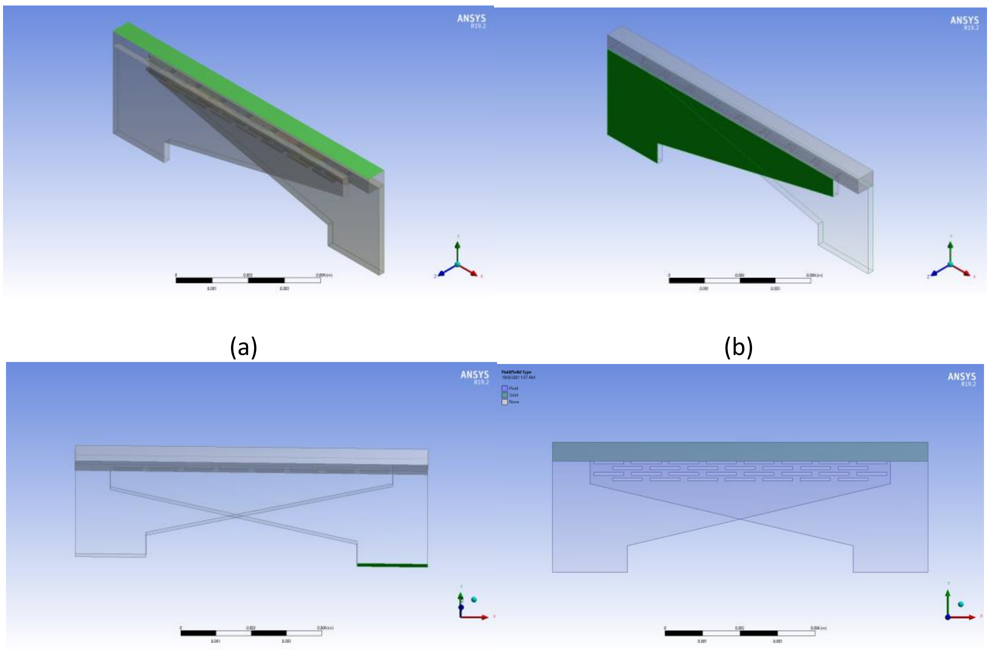
Table 1
Properties of water and Aluminum Oxide Al_2O_3 nanofluid at 27°C

Material	μ (kg/ms)	ρ (kg/m^3)	*Cp*(J/kg.K)	*k* (W/m.K)
Pure Water	1.003×10^{-3}	997.10	4179.00	0.6130
Al_2O_3Nanofluid with volume fraction ϕ = 4%	1.098×10^{-3}	1113.47	3706.84	0.6926

The parameters set for this research is with constant heat flux throughout the surface of the heated block, with increasing mass flow rate from the inlet and pressure gauge set to zero at the outlet. The parameters and that boundary condition is the model is illustrated as below:

Table 2
Parameters setting for the simulation

Mass flow rate (kg/hr)	20	40	60	80
Heat flux, (W/cm^2)	200	200	200	200



(a) (b) (c) (d)

Fig. 2. (a) Heating surface, (b) inlet (left) and outlet (right), (c) symmetry walls and (d) microchannel arrangement

The output parameters that are obtained from the simulation includes temperature at inlet, outlet and surface in contact with microchannel. Besides that, the values of pressure and velocity are also taken at the inlet and outlet. These raw data are then used to solve the governing equations related to the heat transfer and pressure drop. The governing equations that are involved in the process of numerical analysis are as below.

Heat flux equation

$$q = mC_p\Delta T \tag{1}$$

$$q = hA_s\Delta T \tag{2}$$

Where q is the heat flux, m is the mass flow rate, h is the coefficient of convective heat transfer, A_s is the surface area of microchannel in contact with the heated surface, C_p is the specific heat and ΔT is the temperature difference.

Nusselt number,

$$N_u = \frac{h\,D_h}{k} \tag{3}$$

Hydraulic diameter,

$$D_h = \frac{2H_cW_c}{H_c + W_c} \tag{4}$$

Reynolds number,

$$Re = \frac{u\,D_h}{v} \tag{5}$$

Pressure drop,

$$\Delta P = \frac{2f_{app}x\rho u^2}{D_h} \tag{6}$$

Where k represents the thermal conductivity, H_c represents that height of channel, W_c is the width of channel, u is the average velocity, v is the kinematic viscosity, f_{app} is the apparent friction factor, x is the channel length, and ρ is the density of fluid.

Grid independent testing is done by applying mesh at critical points that are going to be studied such as the internal channel walls, inlet and outlet, the heated wall with uniform heat flux and the insulated adiabatic wall throughout the microchannel. The meshing applied will be further fined because the finer the meshing, the more accurate the result will be. The element sizing of first meshed model was then further increades until the values of temperature outlet converged and no longer changed. The difference of the results between the first to the converged mesh is approximately 0.21% more accurate. This will help enhance the accuracy of the of the desired results.

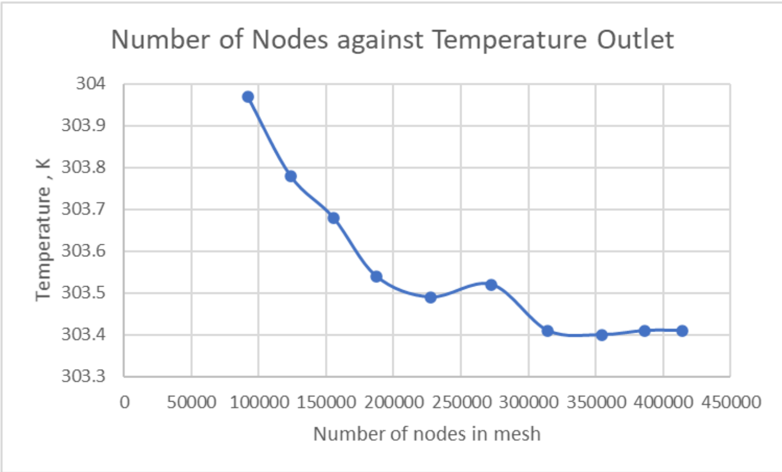


Fig.1. Grid independence test

3. Results and Discussion

The visualization of flow can be seen in terms of velocity and temperature. The following Figure 3 shows the velocity profile of the flow from the inlet to the outlet of the model. The fluid closer to the inlet experience a higher velocity as compared to the microchannel that is further away from the inlet. The fluid has to travel a further distance to reach the microchannel hence losing its velocity along the way. The fluid that flows out of the microchannel experience higher velocity and as fluid converges toward the outlet, the velocity of fluid is at the highest at this point.

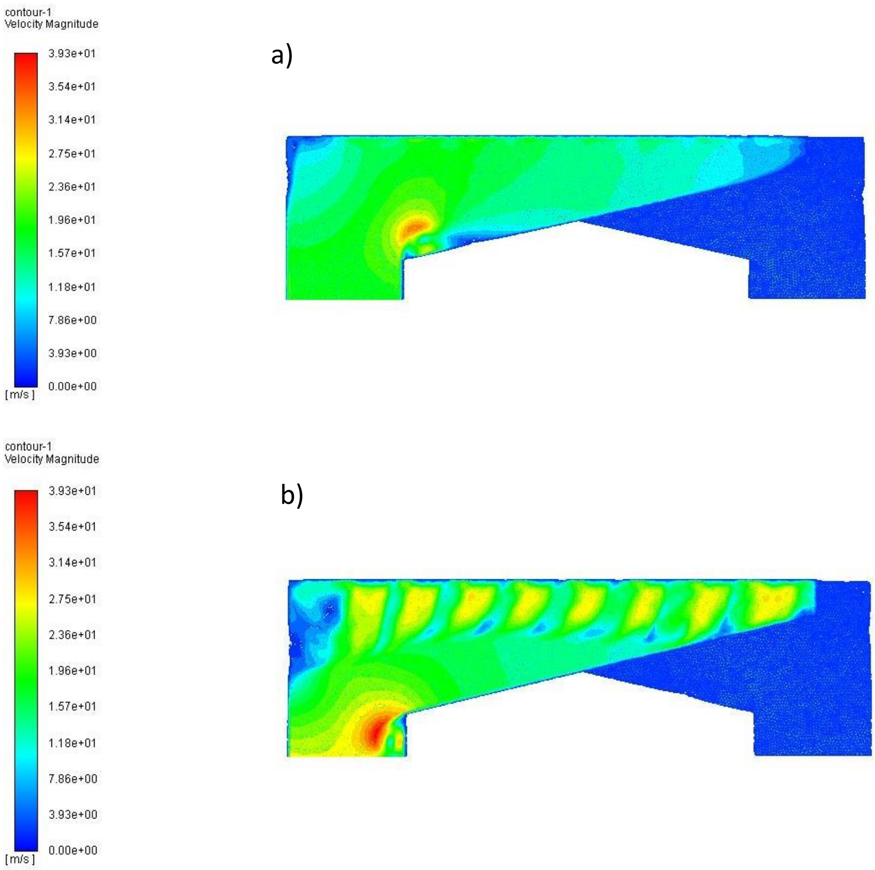


Fig. 3: (a) velocity flow at inlet and (b) velocity flow at outlet

The heated surface is cooled by the coolant that flows through the microchannel. At this point, the residual heat from the surface that is in contact with the fluid will be absorbed by the coolant and make its way through the outlet distribution channel. Figure 4 below shows the heat flow from heating block to the microchannel. The heat flow in the microchannel is seemed to be uneven where more heat is being transferred at the right section of the model than the left part. A hotspot is located at the right most corner of the model. This is due to the flow of coolant out of the microchannel and causes stagnation point. At this point the velocity flow is very small and causing the heated coolant to accumulate at this area before gradually flowing out of the outlet distribution channel.

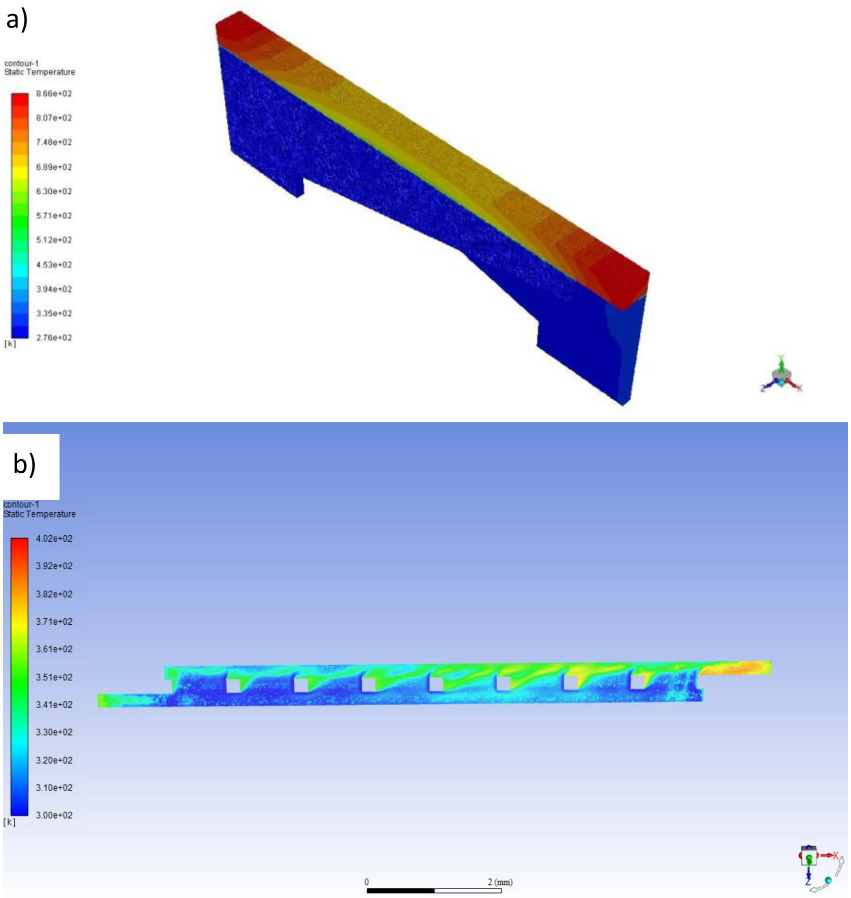


Fig. 4.(a) heat flow on the heating block and (b) cross section of microchannel in contact with heating block

Figure 5 shows the temperature difference between surface temperature and the fluid bulk mean temperature. The temperature difference for the lowest mass flow rate (20 kg/s) is similar for both water and Al_2O_3. However, as the mass flow rate increasing, the Al_2O_3 nanofluid experienced a much lower temperature difference compared to water. This due to the density difference between both fluids. Higher fluid density contributes to a higher Reynolds number which indirectly increase the heat transfer rate to the fluid. This is also shown by the following Figure X of Nusselt number versus Reynolds number

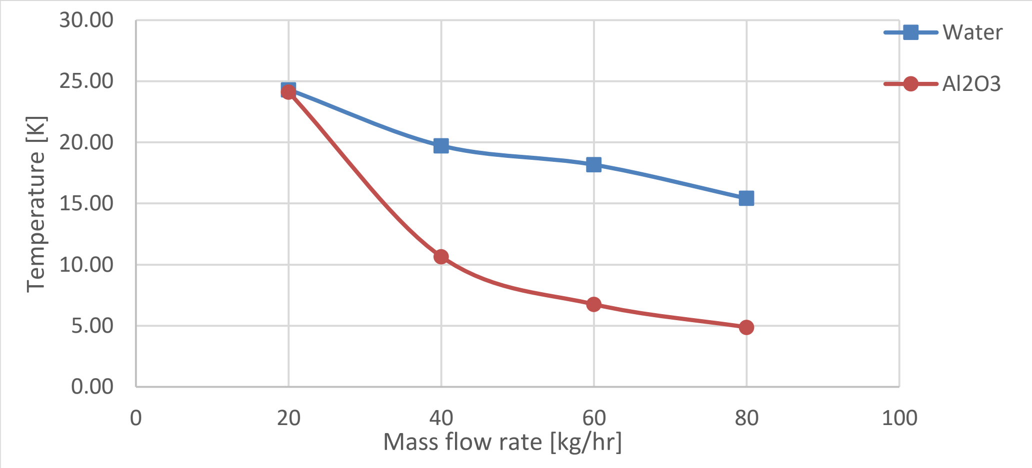


Fig. 5: Temperature difference against mass flow rate

Both nanofluid and water shows an increment of Nusselt number as the Reynolds number increase. However, as the mass flow rate increases, the average velocity increase together with the Reynolds number show a significant increment of Nusselt number in nanofluid compared to water. The increment of heat transfer capability is also related to the thermal conductivity of the nanofluid. High thermal conductivity reduces the thermal resistance of the flow and increase the capability of the fluid to transfer heat. Nusselt number range of both fluids can be seen in Figure 6 below.

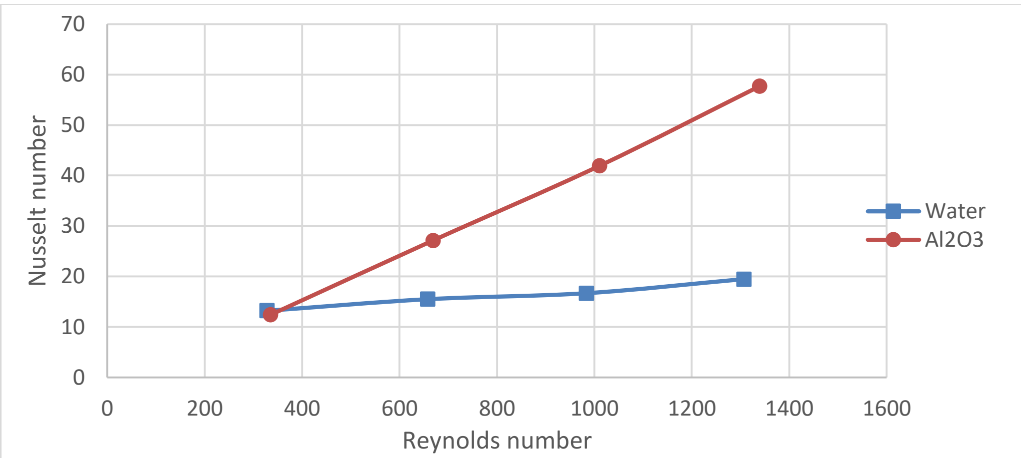


Fig. 6. Nusselt number against Reynolds number

In terms of pressure drop as shown in Figure 6, at the lowest mass flow rate (20kg/hr) there is no significant difference of pressure drop for both water and Al_2O_3 nanofluid. However, as the mass flow rate increases, the pressure drop of Al_2O_3 nanofluid increase exponentially up to an average of 14% higher than that of water. This means that at the end of the flow, which is the outlet, the pressure loss in Al_2O_3 nanofluid is more than water. This is expected as Al_2O_3 has higher viscosity compared to water.

In the case of apparent friction factor, nanofluid has approximately 23.5% higher compared to water. This is because apparent friction factor has relations regarding friction that occur over specified length across the wall surface of the channel and related to the wall shear stress. The presence of nanoparticles in causes it to contact the surface of the wall and producing friction and causes depreciation of momentum of fluid due to the increase in wall shear stress. The apparent friction factor of both fluids can be seen in Figure 8.

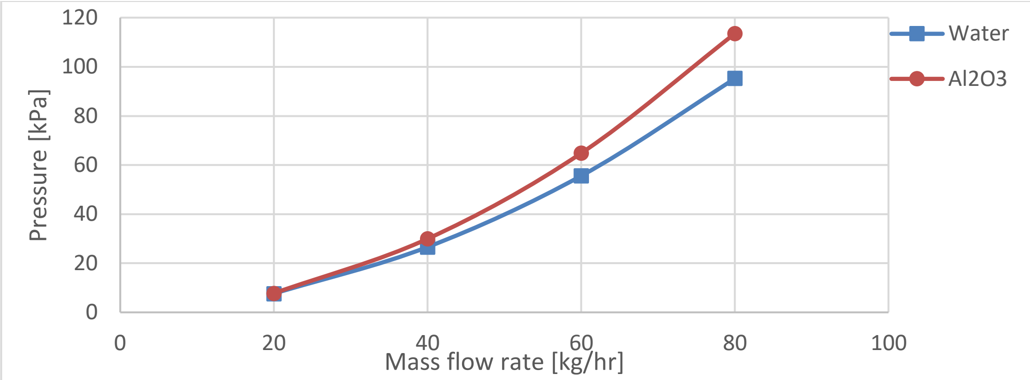


Fig. 7. Pressure drop against mass flow rate

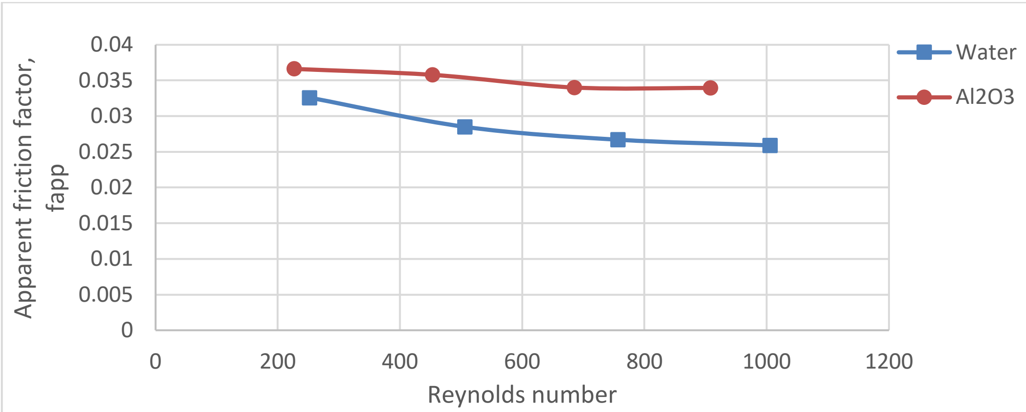


Fig. 8. Apparent friction factor against mass flow rate

4. Conclusion

This study is comparing the differences portrayed by using pure water and Aluminium Oxide nanofluid as coolants on a multilayer microchannel. The simulation is performed using ANSYS software and parameters of constant heat flux of 200W/ with an increasing mass flow rate from 20, 40, 60 up to 80kg/hr. From the simulation done, it is noticed that the used on nanofluid poses better heat transfer performance compared to water. This can be seen from the amount of heat dissipated from the surface of microchannel in contact with the heating block up to the outlet distribution channel. The Nusselt number of nanofluid shows a higher value compared to water. However, in terms of pressure drop, water has better advantage as the pressure drop of water is lower than that of nanofluid. This is because nanofluid has higher friction factor compared to water. Overall, it can be said that nanofluid has good heat dissipating properties however pressure drop is still that should be taken into consideration.

References

[1] Brighenti, Flavio, Natrah Kamaruzaman, and Juergen J. Brandner. "Investigation of self-similar heat sinks for liquid cooled electronics." *Applied thermal engineering* 59, no. 1-2 (2013): 725-732.

[2] Cheng, Y. J. "Numerical simulation of stacked microchannel heat sink with mixing-enhanced passive structure." *International communications in heat and mass transfer* 34, no. 3 (2007): 295-303.

[3] Liu, Guojie, Bin Zhang, Yunliang Zhang, and Chunsheng Guo. "Modeling of double-layer triangular microchannel heat sink based on thermal resistance network and multivariate structural optimization using firefly algorithm." *Numerical Heat Transfer, Part B: Fundamentals* 77, no. 5 (2020): 417-428.

[4] Elqady, Hesham I., Ali Radwan, Abdallah YM Ali, Mohammed Rabie, Essam M. Abo-Zahhad, Shinichi Ookawara, M. F. Elkady, and A. H. El-Shazly. "Concentrator photovoltaic thermal management using a new design of double-layer microchannel heat sink." *Solar Energy* 220 (2021): 552-570.

[5] Lu, Sainan, and Kambiz Vafai. "A comparative analysis of innovative microchannel heat sinks for electronic cooling." *International Communications in Heat and Mass Transfer* 76 (2016): 271-284.

[6] Vafai, Kambiz, and Lu Zhu. "Analysis of two-layered micro-channel heat sink concept in electronic cooling." *International Journal of Heat and Mass Transfer* 42, no. 12 (1999): 2287-2297.

[7] Wen, Dongsheng, and Yulong Ding. "Experimental investigation into convective heat transfer of nanofluids at the entrance region under laminar flow conditions." *International journal of heat and mass transfer* 47, no. 24 (2004): 5181-5188.

[8] Fotukian, S. M., and M. Nasr Esfahany. "Experimental investigation of turbulent convective heat transfer of dilute γ-Al2O3/water nanofluid inside a circular tube." *International Journal of Heat and Fluid Flow* 31, no. 4 (2010): 606-612.

[9] Choi, S. US, and Jeffrey A. Eastman. *Enhancing thermal conductivity of fluids with nanoparticles*. No. ANL/MSD/CP-84938; CONF-951135-29. Argonne National Lab., IL (United States), 1995.

[10] Kamaruzaman, Natrah, Mohd Farhan Wisley, Ummikalsom Abidin, Mohsin Mohd Sies, and Mohd Ridhwan Mohd Ruslan. "Effect of Channel Arrangement on the Thermal Hydraulic Performance of Multilayer Microchannel Arrays." *CFD Letters* 12, no. 8 (2020): 17-25.

[11] Yu, Kitae, Cheol Park, Sedon Kim, Heegun Song, and Hyomin Jeong. "CFD analysis of nanofluid forced convection heat transport in laminar flow through a compact pipe." In *Journal of Physics: Conference Series*, vol. 885, no. 1, p. 012021. IOP Publishing, 2017.