COMPUTATIONAL FLUID DYNAMIC STUDY OF FLUID FLOW OF SINGLE PHASE IN MICROCHANNEL

Nik Hairie Asyari Hussin[1], Mohammad Ikmal Salim[2] and Amirah Mohamad Sahar[3*]

Pusat Pengajian Sains Matematik, Kolej Pengajian Pengkomputeran, Informatik Dan Media, Universiti Teknologi MARA (UiTM), 40450 Shah Alam, Selangor, Malaysia

[1]nikhairie00@gmail.com, [2]mikmalsalim23@gmail.com, [3*]amirahsahar@uitm.edu.my

(indicates the corresponding author)*

ABSTRACT

Computational Fluid Dynamics (CFD) is an analysis of a system using computer-assisted simulations to solve the governing partial differential equations of fluid flow, heat transfer, and chemical reactions. Microchannels are channels with hydraulic diameter less than 1 mm, and the flow condition depends on the surface condition of the channels. The single phase is the single medium used inside the microchannel for the simulations. Ansys Fluent Software is used to do numerical simulations of single phase flow in microchannels to understand the effect of geometrical parameters on velocity profile, friction factor and Reynolds number. ANSYS Workbench is used to create the geometry of the problem and its meshing, which is then imported into ANSYS and resolved by Ansys Fluent. Data reduction is followed by a solution method and convergence criteria. Simulation will stop after the residue for continuity reaches 0.001. The results obtained from the numerical simulations will be compared to the existing previous researcher data for validation. When comparing the results to existing correlation, the existing correlation's velocity profile has a more parabolic shape due to a smoother meshing surface. The velocity flow also worsens as the meshing surface gets rougher which is shown in the velocity contour along the channel. The predicted friction factor has a downward trend but has a difference where the existing correlation has a more linear downward trend. This is due to the number of cases being too few where the cases refer to the different velocity input. It can be concluded that a rougher mesh quality will reduce velocity in the channel which can be seen in the velocity profile and velocity contour along the channel. Next, the friction factor decreases as mesh quality becomes rougher. Additionally, when the friction factor decreases, Reynolds number will increase.

Keywords*: Computational Fluid Dynamics, Microchannels, Single-Phase Flow*

Received for review: 10-03-2023; Accepted: 09-04-2023; Published: 14-04-2023

1. Introduction

Computational fluid dynamics (CFD) is the analysis of a system using computer-assisted simulations, including related phenomena such as fluid flow, heat transfer, and chemical reactions. It was first used in the 1950s and since then, it has been developed increasingly (Norton, Tiwari, and Sun, 2013). Computational fluid dynamics (CFD) can be interpreted as a branch of fluid mechanics that solves and analyses fluid flow problems, utilising numerical methods and algorithms. Computational fluid dynamics (CFD) is a powerful and advanced numerical method to solve governing partial differential equations (PDEs) of mass, momentum and energy conservation in fluid flow and heat and mass transfer problems (Norton and Sun, 2010). In order to perform the calculations required to simulate the fluid surface interaction, defined by boundary conditions, computers need to be employed. High-speed supercomputers can be used for finding better and fast solutions. The fundamental basis of almost each and every CFD problem is connected with the Navier–Stokes equations. These define nearly any single-phase fluid flow. The Navier–Stokes equations can be simplified by removing certain parameters, which depict viscosity and lead to Euler equations. Further simplification can be done by removing the parameters that portray vorticity and consequently yield the full potential equations. After the simplifications are done, these equations can be then linearized so as to obtain the linearized potential equations. Visualising the resulting pressure, velocity vector, temperature distribution, and species concentration of liquid or solid form during the thermal process with attractive colour illustrations and animations can help interpret the physical phenomena that occur, improving the entire process and the quality of the product will be improved (Lemus-Mondaca, Vega-Gálvez, and Moraga, 2011). CFD helps in saving time during the designing process and therefore provides a cheaper and faster option compared to conventional testing for data acquisition. In real life tests, only a limited amount of quantities is measured at a time, while CFD can analyse all desired quantities at once with a high resolution in space and time. CFD analyses still only approximate a real physical solution, and a physical test is needed for verification purposes.

With the introduction of CFD, it instantly became an important research tool in the engineering department. This is because of its advances in numerical solution methods and computer technology, geometrically complex cases, etc. Designers use CFD to verify their products that need to comply with the customer's specification design cycle, thus speeding up the development process. There are benefits that CFD offers such as being able to calculate pressure drops, heat transfer rates, design mass-flow rates and fluid dynamic forces such as lift and drag. People nowadays use CFD as one of the ways to reduce development cost. This is because they do not need to run physical experiments and tests to obtain the data for design that surely will be expensive. CFD simulations can act as an estimate before making any physical experiments that will have a chance to result in failure. Another reason is CFD provides the ability to theoretically simulate any physical condition that also can control the physical process while providing the ability to isolate specific phenomena for study.

Microchannels are channels with a hydraulic diameter of less than 1 mm, typically 1-99 μm. Microchannels are used for fluid control, heat transfer, and observation of cell migration. Its high surface-area-to-volume ratio makes it more efficient than its Marco equivalent, but its smaller size poses several challenges. The classification of microchannels is based on their shape and size. The most influential factor is determining the perfect shape with higher heat dissipation. In addition to channel dimensions, various flow regimes must be considered to determine the power factor. Microchannels are basically divided into circular and non-circular. In this basic classification, aspect ratio as a geometric parameter of microchannels plays the most important role along with surface properties. Flow conditions really depend on the surface conditions that determine what flow regime is expected. In the simulation, a rectangular microchannel will be used in order to compare the results with existing correlation.

Studies that were conducted in single-phase flow were carried out to check the validity of theories of fluid flow in microchannels and explain the reasons for the discrepancies reported in published literature. The validity of conventional theories have been tested in microscale channels, but there are some points to consider when assessing the coefficient of friction of single-phase flows. A crucial element in this study is to elicit all important factors in order to study their effects on fluid flow in microchannels.

The objectives of the study are to perform numerical simulations of single phase flow in microchannel using Ansys Fluent Software. Next, to analyse the effects of geometrical parameters on the velocity profile, friction factor, and Reynolds number. Additionally, to compare the predicted friction factor with the existing correlation.

2. Methodology

The research methodology that will be applied in order to achieve the objectives of this study starts with designing the physical model using Ansys software. The geometry of the microchannel and the cylindrical portion were both designed in Ansys Fluent Software. The microchannel with rectangular shape has a fixed length of 2mm, fixed width of 62mm and fixed height of 0.4mm. The cylindrical inlet plenum has a diameter of 2mm and a height of 8mm. The cylindrical outlet plenum also has the same height and diameter dimension. These two parts were then imported into ANSYS and the domain is discretized and resolved by Ansys Fluent.

Next, after designing the physical model, data reduction is the upcoming stage. CFD analysis was performed to study the properties of fluid flow in microchannels. There are assumptions that need to be made which include stable liquid flow, incompressible fluid, and constant fluid properties. In this stage, data will be reduced to a suitable volume while maintaining the quality of data.

After that, the solution method and convergence criteria needs to be set. By using Ansys Fluent, calculation was run to start the solver iterations. An iteration of 1,000 was set and the solution usually converged before reaching 1,000 iterations. Simulation is stopped when the residue for continuity reaches 1e-3.

The final stage, model evaluation phase, is divided into two parts of which is grid sensitivity check and simulation validation. A grid sensitivity check is needed for evaluation of the optimal grating resolution range derived from previous studies to eliminate the possibility of subjective judgmental bias in selecting grating conditions (Lee et al., 2020). Whereas, simulation validation is just a process of determining how well a simulation model and related data represent the real world in terms of the intended use of the model (John S. Carson, 2002).

For grid sensitivity check, a mesh sensitivity study consists of running the same simulation using grids of different resolutions and analysing how the convergent solution varies with each mesh. Due to limitations where 512,000 nodes is the maximum limit of testing, only mesh with body size above 0.15 is tested. Therefore, meshing of tetrahedra was tested with a body size of 0.15, 0.25 and 0.35. The mesh body size of 0.15 has 256,270 cells, 530,808 faces and 52,594 nodes. Next, the mesh body size of 0.25 has 112,119 cells, 235,262 faces and 24,621 nodes. The mesh body size of 0.35 has 54,790 cells, 116,267 faces and 12,822 nodes. However in simulation validation, the results will be compared with other existing data from similar experiments. The purpose of it is to increase the level of confidence from the inference of a simulation to an acceptable level.

2.1 Equations

The single phase model equations consist of the equation of continuity, momentum equation. The equation of mass of conservation or continuity equation can be presented as such:

Conservation of mass (continuity):

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

Conservation of momentum:

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

Hydraulic diameter rectangular microchannel:

$$Dh = \frac{4(HchWch)}{2(Hch+Wch)} \tag{3}$$

Reynolds number:

$$Re = \frac{Pf\ VchDh}{\mu f} \tag{4}$$

The fanning friction factor is as follows:

$$f = \frac{\Delta PchDh}{2LchPfV^2ch} \tag{5}$$

where $\Delta Pch = Pch, in - Pch, out$

2.2 Tables and figures

Table 1: Reynolds number calculation

Density, P [kg/m^3]	Velocity inside the channel, V_{ch} [m/s]	Hydraulic diameter, D_h [m]	Fluid Viscosity, μ_f [kg/ms]	Reynolds Number, Re [-]
998.2	0.05	6.6667e-4	1.003e-3	33.1738
998.2	0.10	6.6667e-4	1.003e-3	66.3476
998.2	0.15	6.6667e-4	1.003e-3	99.5214
998.2	0.20	6.6667e-4	1.003e-3	132.6952

Table 2: Friction factor calculation

Mesh Body Size	Pressure Inlet, $P_{ch,in}$ [pa]	Pressure outlet, $P_{ch,out}$ [pa]	Length Channel, L_{ch} [m]	Density, P [kg/m^3]	Velocity Squared, V^2_{ch} [m^2/s^2]	Friction factor, f [-]
0.15	1021.0308	3.2359	0.002	998.2	0.0025	67.9754
0.25	1022.286	2.4427	0.002	998.2	0.0025	68.1122

0.35	926.00715	4.4133	0.002	998.2	0.0025	61.5504
0.15	2206.8554	10.7146	0.002	998.2	0.0050	36.6684
0.25	2242.1635	4.6531	0.002	998.2	0.0050	37.3591
0.35	1993.7216	6.5530	0.002	998.2	0.0050	33.1792
0.15	3519.3999	21.5067	0.002	998.2	0.0075	25.9570
0.25	3591.261	8.2925	0.002	998.2	0.0075	26.5884
0.35	3138.3518	6.0448	0.002	998.2	0.0075	23.2441
0.15	4935.3912	36.2671	0.002	998.2	0.0100	20.4498
0.25	5026.706	7.0140	0.002	998.2	0.0100	20.9531
0.35	4334.3454	1.8407	0.002	998.2	0.0100	18.0847

2.3 Model description

Three different meshing body sizes will be used in single channel configuration to examine the accuracy of different numerical schemes. Friction factor obtained will be used to compare with experimental data and conventional theory.

This model will be used to study the fluid flow. Fluid will enter from inlet plenum, *Vi* and flow out to the outlet plenum, *Po*. The inlet and outlet plenum has the same height labelled as *Hv* and a diameter labelled as *Dv*. In between, the liquid will be flowing through a channel with a dimension labelled as *Wc* x *Lc* x *Hc* which represents the width of the channel, the length of the channel and height of the channel respectively.

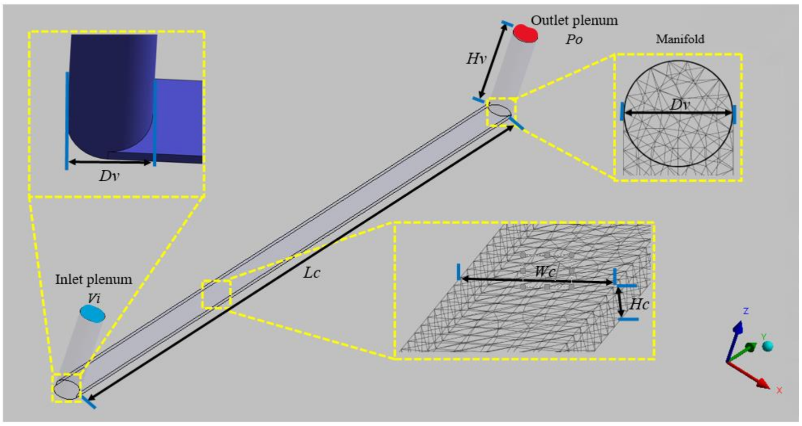


Figure 1. Computational domain of single microchannel system

3. Results and Discussion

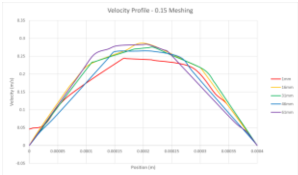
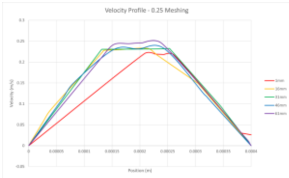
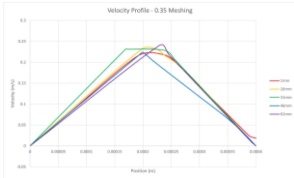
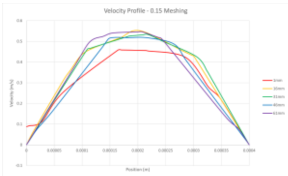
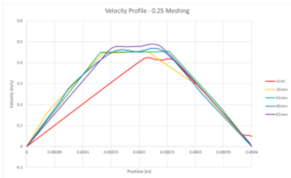
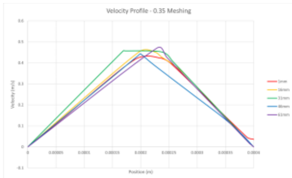
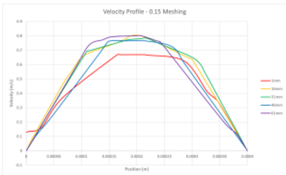
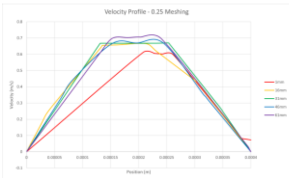
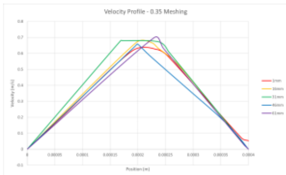
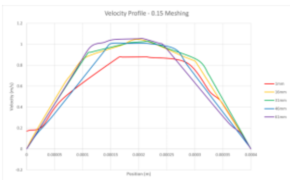
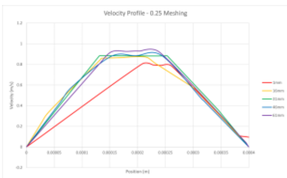
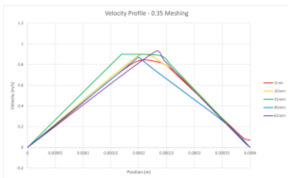
	Mesh body size = 0.15	Mesh body size = 0.25	Mesh body size = 0.35
Velocity input = 0.05 m/s	 (a)	 (b)	 (c)
Velocity input = 0.10 m/s	 (d)	 (e)	 (f)
Velocity input = 0.15 m/s	 (g)	 (h)	 (i)
Velocity input = 0.20 m/s	 (j)	 (k)	 (l)

Figure 2. Velocity profile for mesh body size = 0.15, mesh body size = 0.25 and mesh body size = 0.35

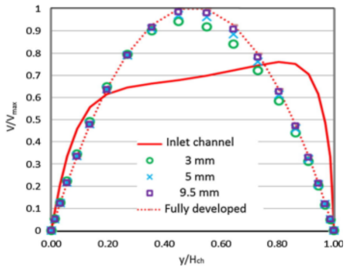
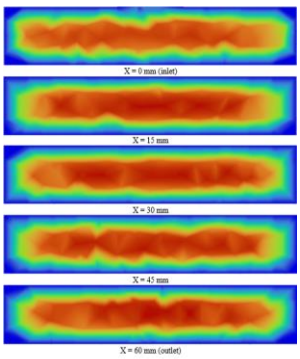
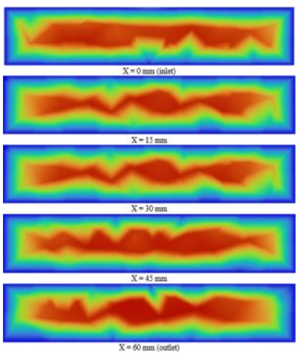
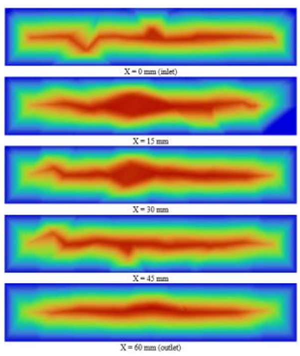


Figure 3. Development of the velocity profile for Re = 181, Dh = 0.56 mm and AR = 0.39. From "Effect of hydraulic diameter and aspect ratio on single phase flow and heat transfer in a rectangular microchannel" by Sahar, A. M., Wissink, J., Mahmoud, M. M., Karayiannis, T. G., and Ashrul Ishak, M. S. (2017).

Across all the figures, Figure 2 (a), (d), (g) and (j) shows a better developed velocity profile compared to the other figures. Figure 2 (b), (e), (h) and (k) shows a developing velocity profile but it does not have the parabolic shape. Figure 2 (c), (f), (i) and (l) have a similar situation but with even less of a parabolic shape. When velocity input is only 0.05 m/s in Figure 2(a), therefore the highest value of velocity in the channel almost reached 0.3 m/s. When velocity input is 0.20 m/s in Figure 2(j), the highest value of velocity in the channel surpasses 1 m/s. By comparing a particular mesh body size between Figure 2 (a), (d), (g) and (j), the graphs look similar with the only difference of velocity in the channel due to the difference in velocity input for each case. When comparing Figure 2 (a), (d), (g) and (j) to Figure 3, Figure 3 shows a better developed velocity profile overall. A cause that could affect Figure 2 (a), (d), (g) and (j) to not have a parabolic shape like Figure 3 is the mesh quality of the model. Due to a limitation where a maximum limit of 512,000 nodes is set, the mesh quality cannot be smoothen furthermore. The rougher the surface, the less the velocity profile has a parabolic shape. In Figure 3, there is also a phenomenon at the inlet where there is a huge loss of velocity. Since the inlet has a sharp edge corner, this phenomenon is probably caused by a flow separation in the inlet region of the microchannel. When compared with Figure 2(a), it does not have as much velocity loss at the inlet. This is probably due to the design of the model where the channel is 1 mm in the plenum as seen in Figure 1. Therefore, less dead spots appear in the plenum.

	Mesh body size = 0.15	Mesh body size = 0.25	Mesh body size = 0.35
Velocity input = 0.05 m/s	 (a)	 (b)	 (c)

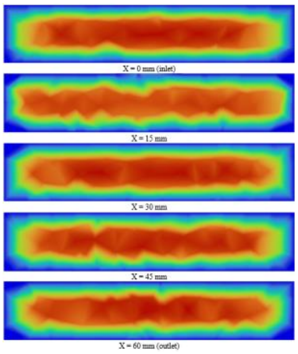
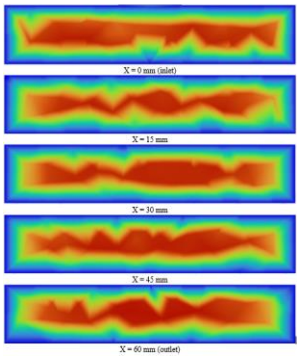
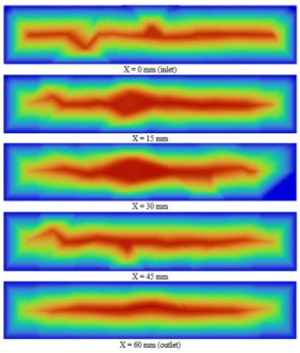
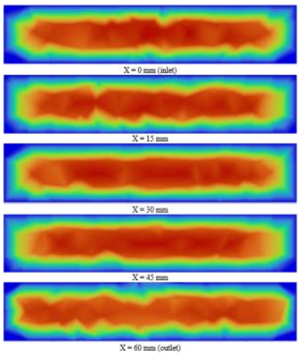
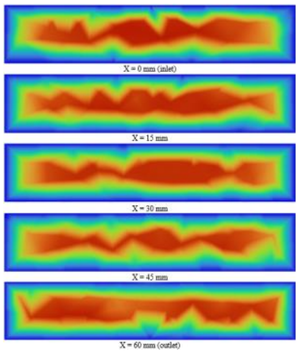
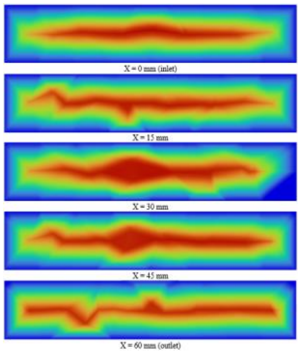
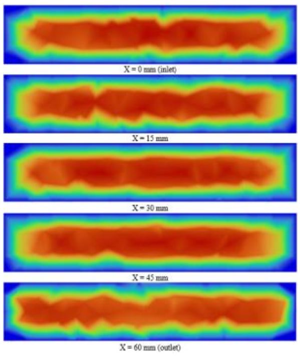
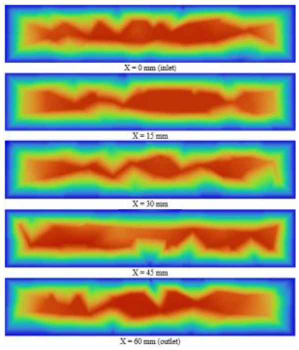
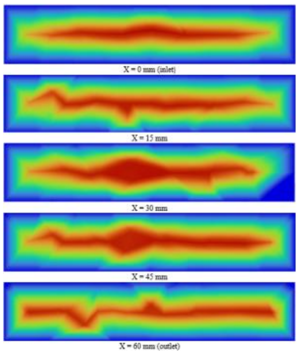
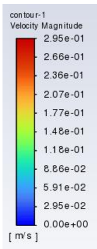
Velocity input = 0.10 m/s	 (d)	 (e)	 (f)
Velocity input = 0.15 m/s	 (g)	 (h)	 (i)
Velocity input = 0.20 m/s	 (j)	 (k)	 (l)
Reading			

Figure 4. Contour of plane of positions for mesh body size = 0.15, mesh body size = 0.25 and mesh body size = 0.35

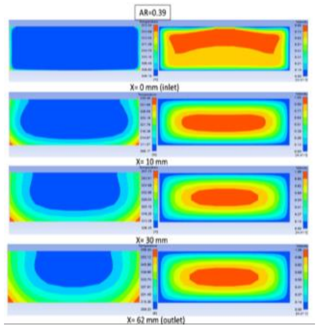
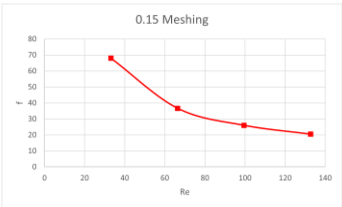
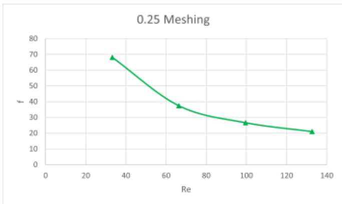
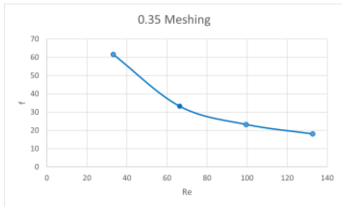


Figure 5. Velocity and temperature contour at AR = 10 and AR = 0.39 along the channel. From "Effect of hydraulic diameter and aspect ratio on single phase flow and heat transfer in a rectangular microchannel" by Sahar, A. M., Wissink, J., Mahmoud, M. M., Karayiannis, T. G., and Ashrul Ishak, M. S. (2017).

When comparing Figure 4 and 5, Figure 5 shows a better velocity flow. Figure 4(a) with a smoother surface compared to Figure 4(c), has better velocity flow. This shows that the better the mesh quality, the better the velocity flow. There is a phenomenon at the inlet of Figure 5 which is related with Figure 3. Due to flow separation, Figure 5 has some velocity loss. Figure 4 is potentially not as affected due to how the model was made where the channel is 1 mm in the plenum.

Mesh body size = 0.15	Mesh body size = 0.25	Mesh body size = 0.35
 (a)	 (b)	 (c)

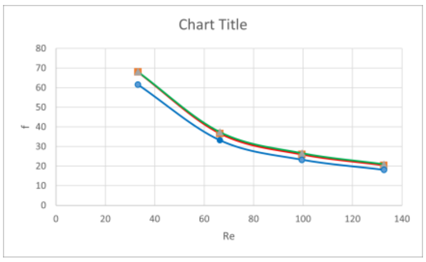
Mesh body size = 0.15, mesh body size = 0.25 and mesh body size = 0.35
 (d)

Figure 6. Friction factor against Reynolds number

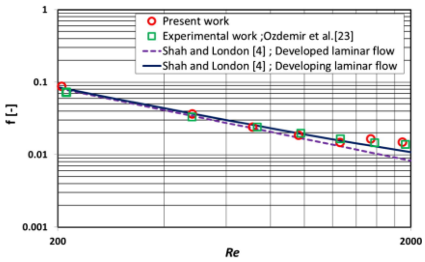


Figure 7. Friction factor with the experimental data and the predictions from correlations. From "Effect of hydraulic diameter and aspect ratio on single phase flow and heat transfer in a rectangular microchannel" by Sahar, A. M., Wissink, J., Mahmoud, M. M., Karayiannis, T. G., and Ashrul Ishak, M. S. (2017).

Numerical results for the predicted friction factors were validated using the experimental data done by Sahar et al. (2017), M.R. Ozdemir et al. (2015) and the Shah and London (1978) correlations for laminar flow that is developing and that has developed. Figure 6(a) and 6(b) are very much alike. This shows that it has a similar pressure loss of the fluid in the pipe due to the interaction between the fluid and the pipe. Figure 6(c) shows it has a lower pressure loss of the fluid in the pipe compared to Figure 6(a) and 6(b). By comparing Figure 6 as a whole compared to Figure 7, it has a familiar trend where it has a downward trend. The main difference is that the downward trend in Figure 7 is more linear compared to Figure 6 overall. A reason it may not form a similar trend is because the number of cases are too few where the cases refer to the different velocity input.

4. Conclusion and Recommendation

A study of numerical simulation was carried out to investigate single phase fluid flow in a rectangular microchannel using water as a working liquid. The model consists of two cylinder plenums, one as inlet and one as outlet and a rectangular microchannel connecting both plenums. The simulation is done with the mesh body size of 0.15, 0.25, 0.35 and with four different velocity cases of 0.05 m/s, 0.10 m/s, 0.15 m/s and 0.20 m/s. What can be concluded are: A rougher mesh quality will reduce velocity in the channel which can be seen in the velocity profile. The friction factor decreases as mesh quality becomes rougher. As the friction factor decreases, Reynolds number will increase. The phenomenon where there is velocity loss at inlet is reduced probably due to the model where the channel is 1 mm in the channel. Based on the result obtained from the simulation. There's still plenty of thing that can be done for future studies in the same scope: Run the same experiment in turbulent flow and comparing with different type/properties of fluid.

Acknowledgement

The authors would like to thank Pusat Pengajian Sains Matematik, Kolej Pengajian Pengkomputeran, Informatik Dan Media, Universiti Teknologi MARA (UiTM) for supporting this research.

References

Ajeeb, W., Oliveira, M. S., Martins, N., & Murshed, S. S. (2021, November). Numerical approach for fluids flow and thermal convection in microchannels. In *Journal of Physics: Conference Series* (Vol. 2116, No. 1, p. 012049). IOP Publishing.

Alsaati, A. A., Warsinger, D. M., Weibel, J. A., & Marconnet, A. M. (2021). Vapor stem bubbles dominate heat transfer enhancement in extremely confined boiling. *International Journal of Heat and Mass Transfer*, *177*, 121520.

Carson, J. S. (2002, December). Model verification and validation. In *Proceedings of the winter simulation conference* (Vol. 1, pp. 52-58). IEEE.

Das, A. K., & Hiremath, S. S. (2022). Experimental and numerical analysis of thermohydraulic performance and entropy-generation in a rectangular microchannel for laminar and single-phase flow: Parametric study and multi-objective optimization. *Thermal Science and Engineering Progress*, *33*, 101375.

Dash, B., Nanda, J., & Rout, S. K. (2022). The role of microchannel geometry selection on heat transfer enhancement in heat sinks: A review. *Heat Transfer*, *51*(2), 1406-1424.

Femto Engineering. (2022, March 21). *An introduction to CFD: What, why and how*. Retrieved January 30, 2023, from https://www.femto.eu/stories/what-is-cfd/

Fung, C. K., & Majnis, M. F. (2019). Computational Fluid Dynamic Simulation Analysis of Effect of Microchannel Geometry on Thermal and Hydraulic Performances of Micro Channel Heat Exchanger. *Journal of Advanced Research in Fluid Mechanics and Thermal Sciences*, *62*(2), 198-208.

InternacionalWeb.com. (n.d.). *Advantages of computational fluid dynamics*. PRE Technologies. Retrieved January 30, 2023, from http://www.pretechnologies.com/services/computational-fluid-dynamics/advantage

Jouhara, H., & Meskimmon, R. (2018). An investigation into the use of water as a working fluid in wraparound loop heat pipe heat exchanger for applications in energy efficient HVAC systems. *Energy*, *156*, 597-605.

Korasikha, N. R., Sharma, T. K., Rao, G. A. P., & Murthy, K. M. (2021). Recent Advancements in Thermal Performance Enhancement in Microchannel Heatsinks for Electronic Cooling Application. In *Heat Transfer-Design, Experimentation and Applications*. IntechOpen.

Lee, M., Park, G., Park, C., & Kim, C. (2020). Improvement of grid independence test for computational fluid dynamics model of building based on grid resolution. *Advances in Civil Engineering*, *2020*, 1-11.

Malekjani, N., & Jafari, S. M. (2018). Simulation of food drying processes by Computational Fluid Dynamics (CFD); recent advances and approaches. *Trends in food science & technology*, *78*, 206-223.

Peng, X. F., Peterson, G. P., & Wang, B. X. (1994). Frictional flow characteristics of water flowing through rectangular microchannels. *Experimental Heat Transfer An International Journal*, *7*(4), 249-264.

Qu, W., & Mudawar, I. (2002). Experimental and numerical study of pressure drop and heat transfer in a single-phase micro-channel heat sink. *International journal of heat and mass transfer*, *45*(12), 2549-2565.

Raman, R. K., Dewang, Y., & Raghuwanshi, J. (2018). A review on applications of computational fluid dynamics. *Int. j. LNCT*, *2*(6), 137-143.

Rosa, P., Karayiannis, T. G., & Collins, M. W. (2009). Single-phase heat transfer in microchannels: The importance of scaling effects. *Applied Thermal Engineering*, *29*(17-18), 3447-3468.

Sahar, A. M., Özdemir, M. R., Fayyadh, E. M., Wissink, J., Mahmoud, M. M., & Karayiannis, T. G. (2016). Single phase flow pressure drop and heat transfer in rectangular metallic microchannels. *Applied Thermal Engineering, 93*, 1324-1336.

Sahar, A. M., Wissink, J., Mahmoud, M. M., Karayiannis, T. G., & Ishak, M. S. A. (2017). Effect of hydraulic diameter and aspect ratio on single phase flow and heat transfer in a rectangular microchannel. *Applied Thermal Engineering*, *115*, 793-814.

Shah, R. K., & London, A. L. (2014). *Laminar flow forced convection in ducts: a source book for compact heat exchanger analytical data*. Academic press.

Sun, H., Fu, H., Yan, L., Ma, H., Luan, Y., & Magagnato, F. (2022). Numerical Investigation of Flow and Heat Transfer in Rectangular Microchannels with and without Semi-Elliptical Protrusions. *Energies*, *15*(13), 4927.

Therma. (n.d.). *Understanding microchannel heat exchangers & their use cases*. Retrieved July 15, 2022, from https://www.therma.com/microchannel-heat-exchangers/

Tuckerman, D. B., & Pease, R. F. W. (1981). High-performance heat sinking for VLSI. *IEEE Electron device letters*, *2*(5), 126-129.

Tusar, M., Ahmed, K., Bhuiya, M., Bhowmik, P., Rasul, M., & Ashwath, N. (2019). CFD study of heat transfer enhancement and fluid flow characteristics of laminar flow through tube with helical screw tape insert. *Energy Procedia*, *160,* 699-706.

Wei, H., Wu, T. F., Sekar, D., Cronquist, B., Pease, R. F., & Mitra, S. (2012, December). Cooling three-dimensional integrated circuits using power delivery networks. In *2012 International Electron Devices Meeting* (pp. 14-2). IEEE.

Xu, B., Ooti, K. T., Wong, N. T., & Choi, W. K. (2000). Experimental investigation of flow friction for liquid flow in microchannels. *International Communications in Heat and Mass Transfer*, *27*(8), 1165-1176.

Zhang, J., Diao, Y. H., Zhao, Y. H., & Zhang, Y. N. (2014). An experimental study of the characteristics of fluid flow and heat transfer in the multiport microchannel flat tube. *Applied Thermal Engineering*, *65*(1-2), 209-218.