



Parametric Study of Pin-Fins Microchannel Heat Exchanger

Mohammed Ziauddin, Fadi Alnaimat^(✉), and Bobby Mathew

Department of Mechanical Engineering, The United Arab Emirate University,
P.O. Box 15551, Al Ain, United Arab Emirates
f.alnaimat@uaeu.ac.ae

Abstract. MEMS heat exchangers use microchannels to increase the surface area of contact between the moving fluids. MEMS heat exchangers are typically utilized for processor chip cooling applications. This study investigates the influence of geometric and operating parameters on performance of a pin-fins microchannel heat exchangers. ANSYS Workbench is used for modelling. The mathematical model consists of the continuity equation, Navier-Stokes equation, and energy equation. Water is taken to the fluid in this study. The boundary conditions of the model consist of the inlet temperature of the fluids, inlet flow rate of the fluids, fluid velocity on the walls (set $v = 0$), outlet pressures (set to $P_{out} = 0$), and temperature gradient at the outlet (set equal to zero). The influence of hydraulic diameter, pin fins diameters, structural material, and Reynolds number on the effectiveness of heat exchanger is studied. Studies are carried out for Reynolds number varying between 100 and 2000. The materials considered for this study are stainless steel, silicon, and copper. The effectiveness is determined using the inlet and outlet temperatures of the fluids. The thermal performance of MEMS heat exchanger is influenced by the hydraulic diameter and pin-fin diameter. The thermal performance of the MEMS heat exchanger increases with reduction in Reynolds number.

Keywords: Effectiveness · Microchannel heat exchanger · Counterflow · Computational fluid dynamics (CFD)

1 Introduction

Heat exchangers are required in the refrigeration and air conditioning systems for heat transfer applications. With the demand of reduction in weight and increasing cost of manufacturing, technology improvement with Microchannel Heat Exchanger (MCHX) is very demanding [1]. The heat exchangers of micro dimensions have more advantages such as savings material, space, and meeting the demand of heat transfer [2]. The design can affect the performance if the heat exchanger is in parallel or counterflow arrangements. The performance is also expected to be varying. The use of microchannel heat exchangers is very recent from past decade. Samsung electronics have implemented the use of microchannel heat exchangers since 2006 in cooling model of air conditioning systems [3]. A study conducted exploring the performance of counter flow microchannel heat changers with different channel cross-sections found that increasing number of channels improved the effectiveness and pressure drop [4].

In literature, very limited work is covered in this direction studying microchannel heat exchanger whether it is through simulations or experiments. In this study, counter flow microchannel heat exchanger is studied with square cross-sections. The temperature variations of running hot fluid and running cold fluid are observed in order to calculate the effectiveness of the heat exchanger. Simulation approach is adopted using ANSYS Fluent and performing Computational Fluid Dynamics (CFD) simulation. Simulation is done in order to reduce cost and make use of computational tool to get satisfactory results controlling various parameters.

This could contribute in modifying and finalizing the model in order to prepare, manufacture, and test the microchannel heat exchanger experimentally in the future.

2 Mathematical Modeling

Main parameters that are involved in determining the performance of heat exchangers are the heat transfer rate of hot fluid and cold fluid. This could also be understood from observing the temperatures at the inlets and outlets of both fluids. The heat exchanger is expected to cool the hot fluid running at temperature less than 100 °C. Hence, the specific heat capacity is assumed to be constant in theoretical calculations for the cold and hot fluids. The capacity ratio is considered as shown in Eq. (3) value equal to 1 [5].

$$C_c = (\dot{m}C_p)_c \quad (1)$$

$$C_h = (\dot{m}C_p)_h \quad (2)$$

$$C_r = \frac{C_{min}}{C_{max}} = \frac{\min(C_c, C_h)}{\max(C_c, C_h)} \quad (3)$$

Where, C_c and C_h are the heat capacity rate of cold and hot fluids respectively, C_p is the specific heat at constant pressure, and \dot{m} is the mass flowrate. The rate of heat transfer can be determined by \dot{Q} from the below Eqs. (4) and (5).

$$\dot{Q} = C_c(T_{c,out} - T_{c,in}) \quad (4)$$

$$\dot{Q} = C_h(T_{h,in} - T_{h,out}) \quad (5)$$

$$\dot{Q}_{max} = C_{min}(T_{h,in} - T_{c,in}) \quad (6)$$

Where, $T_{c,in}$, $T_{c,out}$, $T_{h,in}$ and $T_{h,out}$ are the temperatures of cold and hot fluids at the inlets and outlets respectively. The log-mean temperature difference (LMTD) can be calculated using the following Eqs. (7), (8) and (9). The heat transfer effectiveness can be determined using NTU method and using the following Eqs. (10) and (6). The effectiveness is the ratio of the actual heat transfer rate to the maximum heat transfer rate possible [5].

$$\Delta T_1 = T_{h,in} - T_{c,out} \quad (7)$$

$$\Delta T_2 = T_{h,out} - T_{c,in} \quad (8)$$

$$\Delta T_{lm} = \frac{\Delta T_1 - \Delta T_2}{\ln\left(\frac{\Delta T_1}{\Delta T_2}\right)} \quad (9)$$

$$\varepsilon = \frac{\dot{Q}}{\dot{Q}_{max}} \quad (10)$$

ANSYS Fluent simulation software is used to design and analyze microchannel heat exchanger with square microchannels and rectangular fins inside the channel. The modeled section of the heat exchanger is shown in Fig. 1. Overall dimension of the analyzed unit heat exchanger is 0.5 mm in width, 0.5 mm in height and 2 cm in length. Two square microchannels were modelled with 100 μm by 100 μm cross-sectional area. One carrying hot fluid and other carrying cold fluid in a counter flow arrangement. Each channel was attached with 20 fins of 20 μm by 20 μm square cross section that were equidistant through located through the length of the channel. The working fluid used is water. The Reynolds number was varied from 0.0005 to 50 and the effectiveness was calculated for each case.

2.1 Counterflow Microchannels

In Fig. 1, the CAD model of the heat exchanger is shown that is consisting of the hot fluid domain on the top channel and cold fluid domain in the bottom channel.

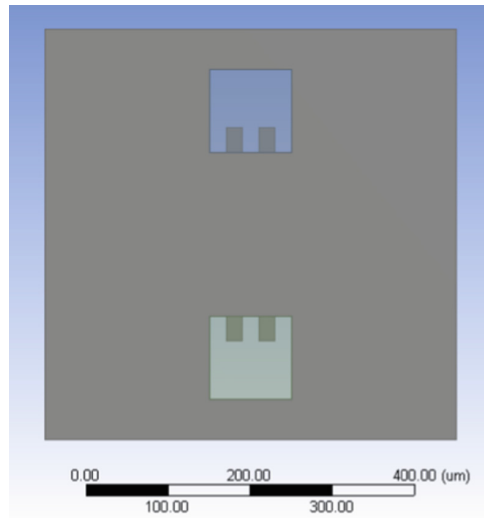


Fig. 1. CAD Model of microchannel heat exchanger.

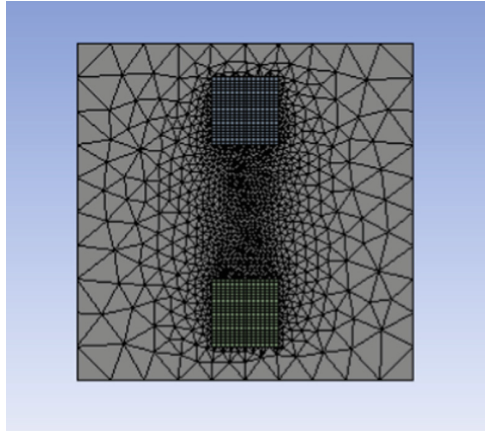


Fig. 2. Meshed model of microchannel heat exchanger.

In Fig. 2, the meshed model is shown, and it was done applying mesh sizing tool through sizing the edges. Mesh elements were made smaller and coarse near the microchannel edge compared to other regions ensuring more elements located at the hot and cold fluids domains.

3 Setup

The model was analyzed and solved as steady state conditions using energy equations and viscous model (realizable k-epsilon model considering standard wall functions). For the fluid domain, water-liquid was considered as the working fluid. For the shell domain, aluminum material was considered in the model. The temperature of hot inlet was kept at 70 °C, whereas, the temperature of the cold inlet was set at 15 °C. The thermal conditions were kept as convection at the shell wall with heat transfer coefficient as 20 W/m².K.

4 Data Collection

The results were mainly collected to obtain the centerline temperatures of the hot fluid and cold fluid domains. As the Reynolds number was varied from 5 to 300 for the hot fluids and respective velocity obtained from capacity ratio was inputted for cold fluid velocity in the counterflow. The temperature distributions were recorded, and effectiveness was calculated for each case based on the outlet temperature readings. For viewing the contours of temperature distribution in both fluid and shell domain, a plane was created in the mid-section of the heat exchange where it can show the temperature contours of both domains. However, only one legend could be viewed, so the hot fluid temperatures were viewed as those were more significant.

5 Results and Discussion

The results are plotted against the hot fluid Reynolds number. Observing through the contours, it was clear that the amount of temperature loss in the hot fluid was similar to the temperature gained by the cold fluid. For Reynolds number equal to 5, the results are illustrated in the below Fig. 3, the legend shows the temperatures of hot fluid with maximum of 343 K in the inlet and 321.06 K in the outlet.

The centerline temperatures of hot and cold fluid domains are plotted for each of the Reynolds number in Figs. 4 and 5 respectively.

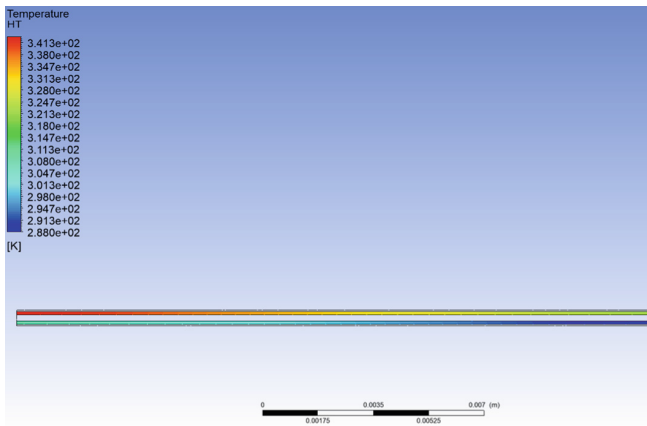


Fig. 3. Temperature contours (section view)

These results also showed that the temperature variations are closely opposite to each other indicating the temperature drop in the hot fluid side is close value of the temperature increased in the cold fluid side. Figures 4 and 5 provide an evidence that the heat capacity of both hot and cold fluids remains similar which is leading to assumption of capacity ratio equal to 1. In order to calculate the effectiveness, the value of \dot{Q} obtained from either Eqs. (4) or (5) will give precise results.

According to Figs. 4 and 5, the Reynolds number of 100 showed less temperature difference with respect to inlet and outlet. Heat transfer is effective in the small thermal entrance length as compared to heat transfer observed for Reynolds numbers 5 to 25. The change in temperature is greater from Reynolds number 5 to 25. For all the cases of Reynolds numbers, the temperature profile is observed as linear from inlet to outlet. It can also be depicted that by further increase in Reynolds number beyond 50 would not result in effective temperature as temperature difference from inlet to outlet gets lower. As the effectiveness is the ratio of actual to maximum heat transfer rate, the decrease in temperature difference from inlet to outlet can decrease the heat transfer effectiveness.

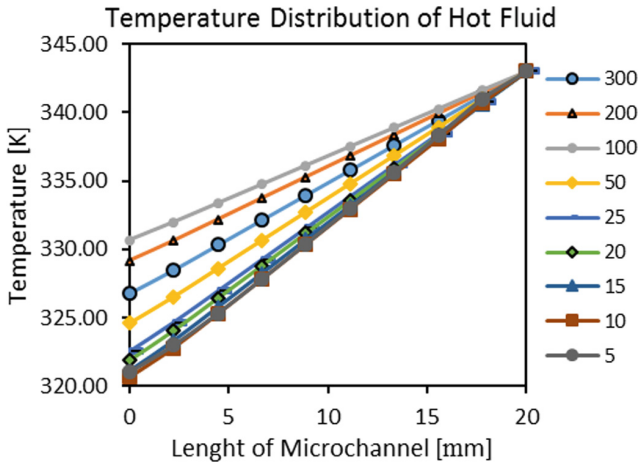


Fig. 4. Temperature distribution of hot fluid.

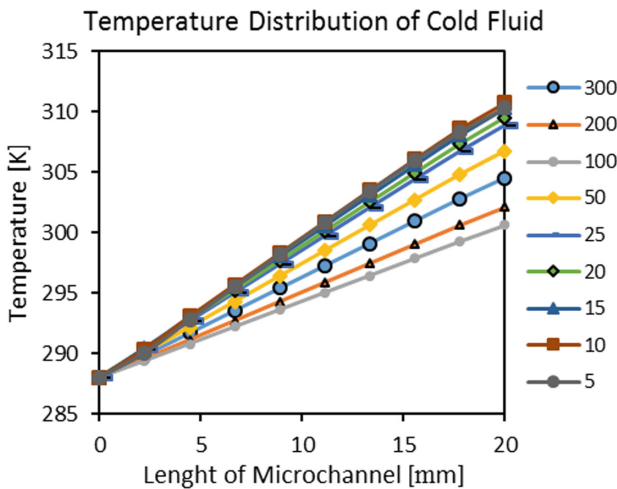


Fig. 5. Temperature distribution of cold fluid.

The Fig. 6 shows the plot of heat transfer effectiveness with respect to Reynolds number for the analyzed unit of microchannel of heat exchanger. From observation, the effectiveness is increasing from 5 to 10 and then decreasing with higher Reynolds number. For Reynolds number 5, 10, and 15 the effectiveness remains above 0.65. The highest value of 0.706 heat transfer effectiveness was found for the Reynolds number equal to 10. That is corresponding to 4.13 m/s for hot and 4.048 for cold fluid. This range of the Reynolds number can yield the most effective performance of the microchannel heat exchanger.

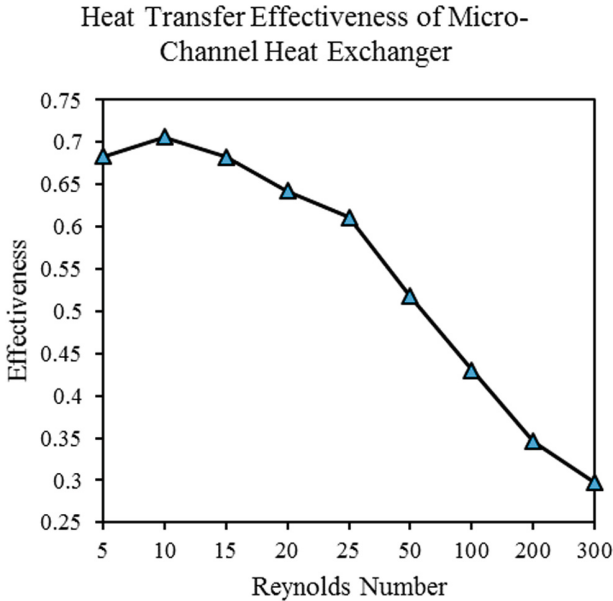


Fig. 6. Heat transfer effectiveness.

6 Conclusions

In this study, the simulation using ANSYS Fluent was achieved. Because there is still less work upcoming in this field of research, it is always better to start with most feasible tools. The results obtained in this work show that heat capacity of both hot and cold fluids remains similar which is leading to assumption of capacity ratio equal to 1 for the counterflow arrangement. The effectiveness is high in certain range of the Reynolds numbers and these can be considered as best operating conditions for good performance of microchannel heat exchanger. This simulation study provided a basis for analyzing the microchannel heat exchanger and indicating best operating conditions for analyzed unit.

In upcoming follow up work in this direction, simulation of different shapes of microchannels, and studying variations in the other parameters including the dimensions of the microchannel itself, and the impact of using different material is expected to be explored as well. Also, experimental analysis is expected to be covered later after subsequent CFD analysis for validating. Moreover, the effect of velocity and effectiveness of the heat exchanger can be observed. All these will surely add to credibility in this field of research adding to literature.

Acknowledgments. The authors acknowledge financial support received from the UAE University for Grant no. 31R153, and the ADEK Award for Research Excellence (AARE) for Grant no. 21N220-AARE18-089.

References

1. Hana, Y., Liu, Y., Li, M., Huang, J.: A review of development of micro-channel heat exchanger applied in air-conditioning system. In: 2nd International Conference on Advances in Energy Engineering (ICAEE2011) (2012)
2. Khan, M.G., Fartaj, A.: A review on microchannel heat exchangers and potential applications. *Int. J. Energy Res.* **35**(7), 553–582 (2010)
3. Hayase, G.: Development of Micro Channel Heat Exchanger for Residential Air-Conditioners. In: International Refrigeration and Air Conditioning Conference, Purdue (2016)
4. Hasana, M.I., Rageb, A.A., Yaghoubi, M., Homayoni, H.: Influence of channel geometry on the performance of a counter flow microchannel heat exchanger. *Int. J. Therm. Sci.* **48**(8), 1607–1618 (2009)
5. Fakheri, A.: Heat exchanger efficiency. *J. Heat Transf.* **129**, 1268–1276 (2007)