EFFECT OF ANALYZING DIFFERENT SHAPES OF FIN ON THE HEAT TRANSFER CAPACITY OF MICRO-CHANNEL HEAT SINK THROUGH CFD

Md. Ishteyaque[1], Amitesh Paul[2]

Research Scholar[1], Professor[2]
Ram Krishna Dharmarth Foundation University, Bhopal

***Abstract*: Different types of micro-channel heat sink with different coolant are use now a days. Here in this work effect of different shapes of micro-channel heat sink fins was analysed. To analyse the effect of different shapes of fins here in this work four different shapes of fins are used that is rectangular, triangular, trapezoidal and cuboidal. To analyse the effect of change in Reynolds number on the performance of heat sink four different Re numbers was considered that are 100, 200, 300 and 400 and calculate the value of maximum temperature of heat sink and heat transfer coefficient in each case of analysis. Through analysis it is found that micro-channel heat sink with trapezoidal shapes fins shows the maximum heat transfer and have lowest temperature. In case of trapezoidal temperature and velocity distribution throughout the heat sink is more uniform as compared to another fins shape.**

1. Introduction

The need for removal of high heat flux with small area attracted many researchers to focus on Micro-Channel Heat Sink (MCHS). Currently, the MCHS is used for high heat flux applications like electronic cooling, because of its less weight, compactness and high heat transfer rate when compared to other microelectronic cooling systems. Figure 1.1. show that the microchannel heat sink geometry used in the microprocessor by Kandlikar (2005), where L is the length and W is the width of the microchannel. The microchannel comprises the family of the heat exchanger, which is used to transfer heat from high-temperature medium to low temperature medium. A microchannel can be defined as channels whose hydraulic diameter is less than 1 millimetre and greater than 1 micron. Above 1 millimetre the flow becomes macroscopic flows in which the coolant flows inside the channel to remove the heat by convection.

Micro-channel heat sink is a very efficient devise for heat transfer application, it is basically used where high specific heat transfer is required. it is basically used for small and sophisticated equipment's. So, it is very necessary to used highly efficient heat sink. To increase the heat transfer capacity of the micro-channel heat sink and to analysed the effect of different shapes of fins on heat transfer, here in this work four different shapes of fins and micro channels was analysed through CFD analysis using Ansys Fluent. It also analysed the effect of different Re numbers on heat transfer capacity of micro-channel heat sink. For each types of fins effect of change in Re number is analysed and calculates the maximum temperature and heat transfer coefficient. Rectangular shape micro-channel heat sink is analysed in chapter 3. Micro-channel heat sink with triangular, trapezoidal and cubical shapes fins was analysed in this chapter. This chapter also includes the temperature and velocity distribution for different micro-channel heat sink

2. Material used:

During the numerical analysis of micro-channel, in this work copper is considered for the manufacturing of micro-channel heat sink and water is used to flow inside the micro-channels. The properties of these materials considered during the numerical analysis is mention in the below table. Different materials can also be used for the manufacturing of heat sink, but as copper is considered in the base papers. So here also it considered copper.

Table.1 Properties of copper considered during the numerical analysis of micro-channels

Properties	Value
Density (kg/m)	8978
Specific heat (Kg/m^{3}K)	381
Thermal Conductivity (W/mK)	387.6

Table.2 Properties of Water considered during the numerical analysis of micro-channels

Properties	Value
Density (kg/m)	998.2
Specific heat (Kg/m^{3}K)	4182
Thermal Conductivity (W/mK)	0.6

Above materials are used during the numerical analysis of micro-channel heat sink. Water is commonly used as a coolant in most of the cases because it is cost effective easily available and environmentally friendly. Using normal water in micro channel can create problem of cavitation in long run. So generally demineralized water is used.

3. Validation of numerical model of micro-channel heat sink

Before performing the numerical analysis of micro-channels, it is very necessary to first validate the primary numerical analysis of micro-channel heat sink on the basis of different parameters that are considered in the previous research.

3.1 Development of numerical model

For performing the CFD analysis of heat sink, first it develops the CFD model. And apply the different boundary conditions as mention in Prajapati et.al [1]. For performing the numerical analysis of heat sink, first it has to develop the solid model of heat sink.

3.2 development of solid model of heat sink

The solid model of heat sink is developed on the basis of geometric parameters given in the base papers. For validating the numerical model, it is necessary to considered same geometric conditions as considered during the analysis performed by Prajapati et.al [1]. The geometric conditions on the basis of which the solid model of heat sink is developed is mention in the below table.

Table.3 Value of geometric parameters considered during analysis of heat sink

Parameters	Value (mm)
Heat Sink width (W)	3.7
Height of heat sink (H)	1.1
Bottom wall thickness (H_b)	0.1
Side wall width (W_W)	0.1
Channel Width (W_c)	0.5
Fin height (H_f)	0.8
Heat sink length (L)	15

Base on the above geometrical parameters solid model is made, the solid model of heat sink was made with the help of Ansys design modeler Software. For the validation case rectangular channel heat sink geometry is considered as mention by Prajapati et.al [1]. The solid model of micro-channel heat sink is shown in the below fig.

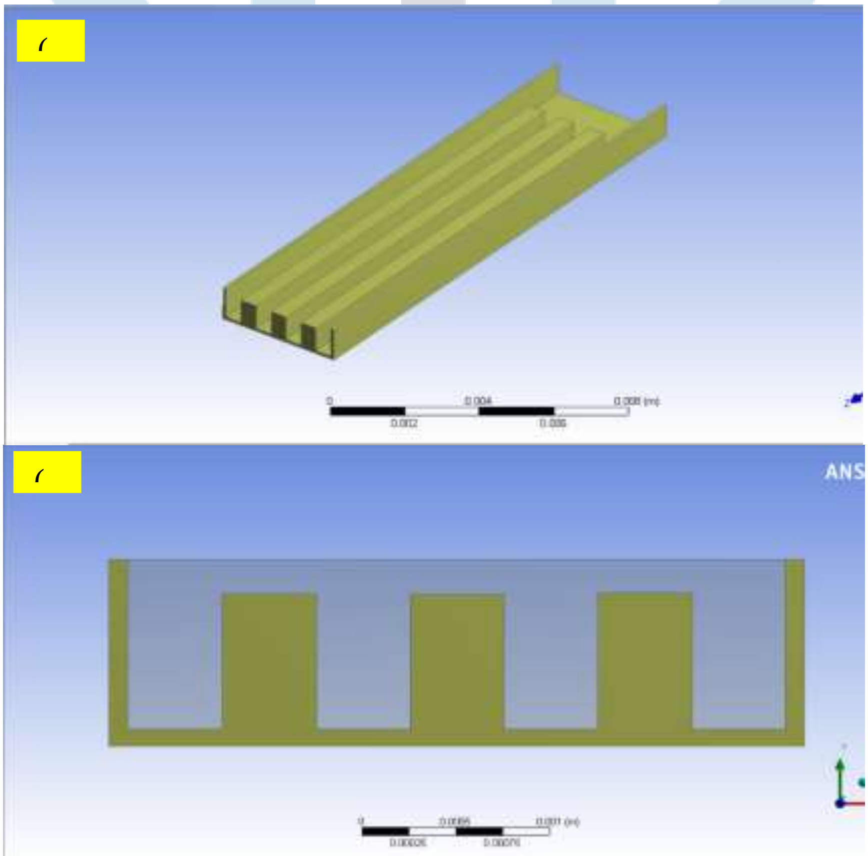

Fig.1 solid model of micro channel heat sink (A) complete view of geometry, (B) side view of the Geometry

From above fig. it is clearly shows that there are four micro-channels through which coolant fluid is flowing. Through solid model it also shows that micro-channels start 2 mm after the inlet position of coolant fluid so that flow can be stabilized after entering in to heat sink channel.

4. Meshing:

For performing the numerical analysis of anybody first is get discretized in to number of bodies. While performing the numerical analysis, first it discretised the body in to number of nodes and elements. For optimizing the number of nodes and elements, discretization is performed with different tools to refine the mesh.as the number of nodes and elements increases the accuracy of the analysis also increases but after reaching certain value of Nodes and elements value of parameters will remain constant. Increasing number of nodes and elements also increases the computational time required for the simulation. The heat sink with simple mesh is shown in the below fig.

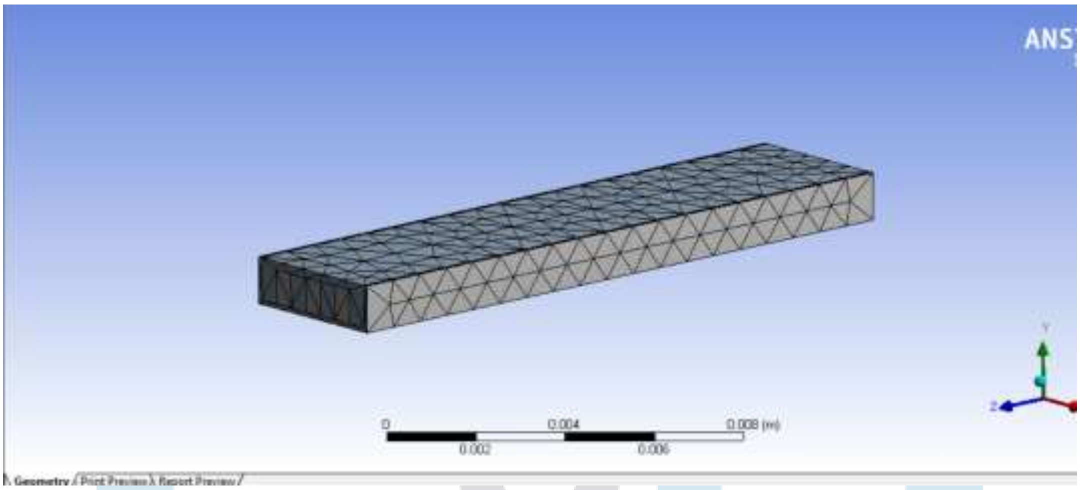

Fig.2 mesh of the micro-channel heat sink having 2909 numbers of elements

Above fig. shows the mesh of the geometry having very coarser mesh which will not give proper results during the numerical simulation. So, in order to gey the accurate result of the heat sink, use of different mesh refinement tools are used. The solid model of micro-channel heat sink having very fine mesh is shown in the below fig.

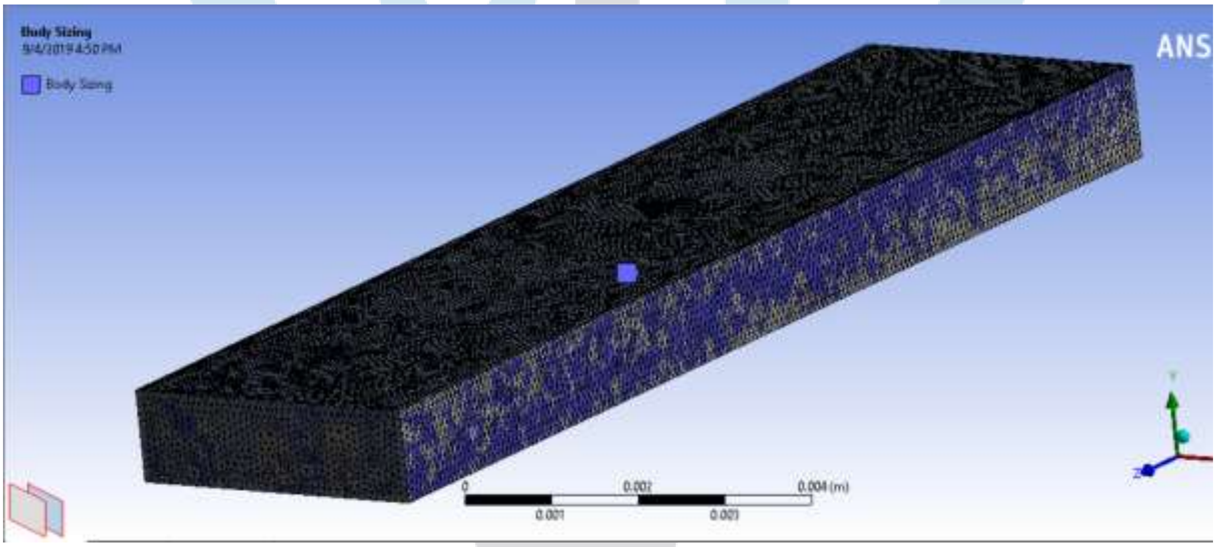

Fig.3 mesh refinement of complete body

Above fig. shows the refinement of heat sink, in order to get refine mesh different tools were used. The detail of mesh is shown in the below fig., and it also contains the number of nodes and elements used during the numerical analysis. In this work, 746337 number of Elements and 153453 nodes is use for the numerical analysis.

5. Validation

For validating the CFD model of micro channel heat sink, here it considered rectangular micro-channel heat sink as considered by Prajapati et.al [1]. Here CFD analysis of heat sink is performed with the help of Ansys Fluent. After applying different boundary conditions simulation was run at different Re number, the value of heat transfer coefficient, temperature and velocity distribution throughout the heat sink and maximum temperature of heat sink is calculated. The temperature and velocity distribution for different velocity of water is shown in the below case.

5.1 For Re = 100

In this case water is entering inside the micro-channel with 0.057 m/s velocity and different boundary conditions was applied on different component of heat sink. The temperature and velocity distribution for this case is shown in the below fig. the value of heat transfer coefficient value which 5460 W/m^2K for rectangular micro-channel at 100 Re number, it also shows the velocity contours of vertical plane made at 7.5 mm from the inlet position of water.

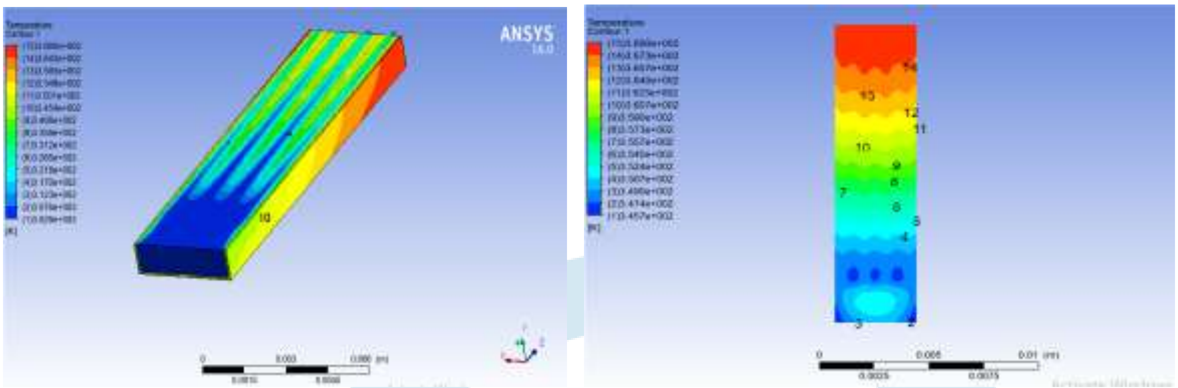

Fig.4 shows the temperature distribution throughout the body of heat sink

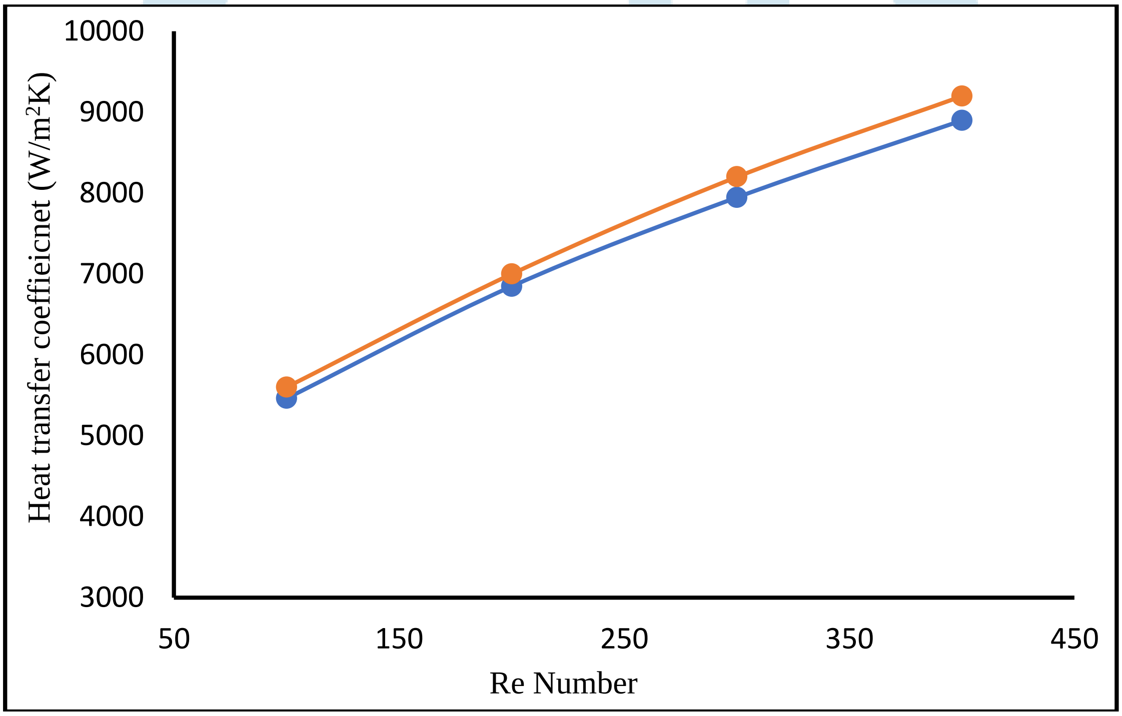

Fig.5 comparison of value of heat transfer coefficient at different Re numbers

From the above comparison graph it is found that the value of heat transfer coefficient for different Re Number calculate through numerical analysis is close to the value given in the base paper at a particular Re number. The error percentage is in under 10 percent for each case. So, it is concluded that the numerical analysis of micro-channel heat sink is correct. For more clarity maximum temperature of heat sink at different Re number is also compared.

6. Comparison of different types of fins shapes used in micro-channel heat sink:

After performing the numerical analysis of micro-channel heat sink having different shapes of fins at different Reynolds numbers. It compares the value of heat transfer coefficient for different shapes of fins at different Reynolds numbers. Here in this work, for analysing the temperature distribution it also compared the value of maximum temperature of heat sink at different velocity of water. Through comparison, optimum heat sink fins can be find out and also it gives clear idea about the temperature and velocity variation of different heat sink. The comparison of temperature and heat transfer coefficient are mention in the below table.

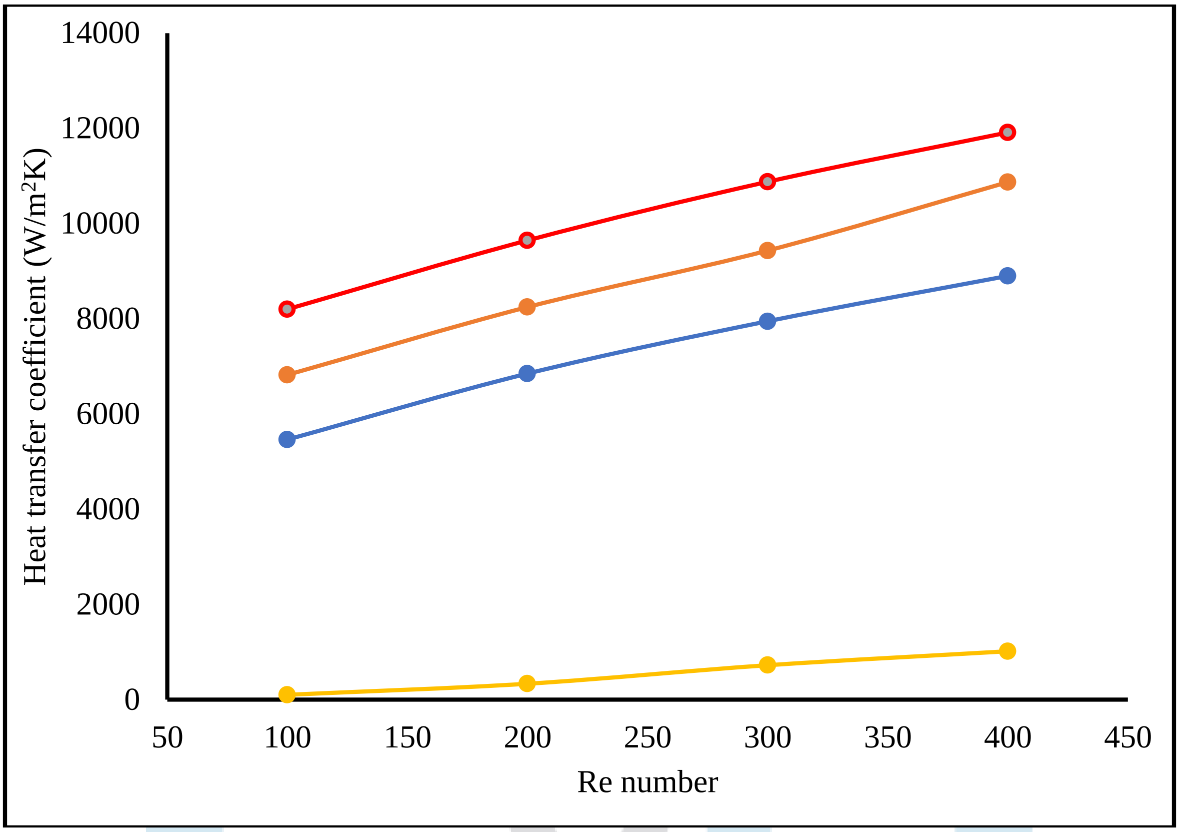

Fig.6 comparison of value of heat transfer coefficient for different Re number

Above graph shows the comparison of value of heat transfer coefficient for different Reynolds number. Through above graph it is found that the variation of heat transfer coefficient with respect to Re number shows the linear variation. The value of heat transfer coefficient for micro-channel heat sink having trapezoidal fin shows the maximum heat transfer coefficient. Whereas heat sink having triangular and rectangular shows less value of heat transfer coefficient as compared to trapezoidal fins, but it is higher then the cuboidal fins. Micro-channel heat sink having cuboidal shapes of fin shows much variation as compared to other three shapes of fins. Which means that, very less amount of heat get transfer from the heat sink. It is conclude that heat sink having trapezoidal fins shows maximum heat transfer.

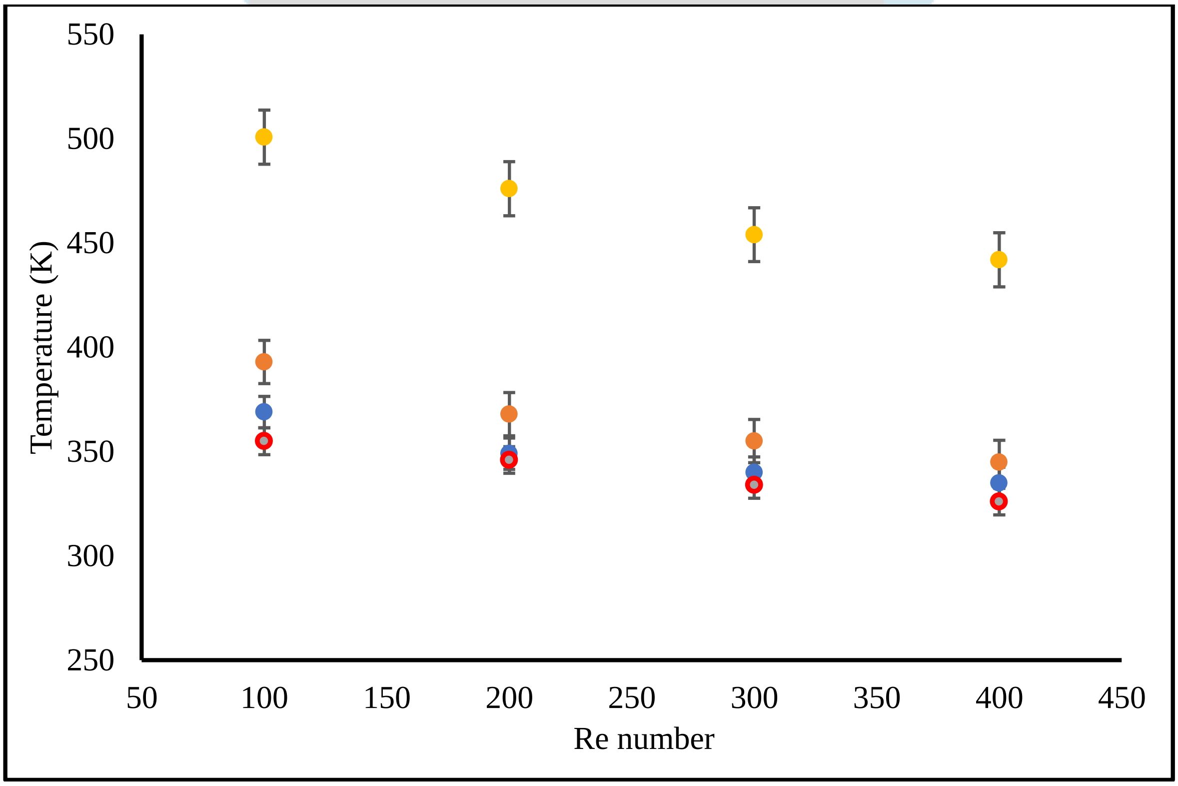

Fig.7 comparison of value of maximum temperautre of heat sink at different Re number

The variation of temperature with respect to Re number for different shape of fins was shown in the above fig. Through graph it is found that the value of maximum temperature for heat sink having trapezoidal fins shows the minimum value ascompared to other. The temperature variation for cuboidal fins shows much higher temperature value as compared to other fins. There is very less variation in temperature in between triangular, rectangular and trapezoidal shape of fin. From above comparision graphs it is concluded that micro-channel heat sink shows optimum heat transfer.

7. Conclusion:

Numerical analysis of heat sink is performed using CFD model through Ansys Fluent. Temperature of heat sink starts decreasing with increase in temperature whereas velocity distribution inside the micro-channel also starts varying. Through analysis it is found that the heat transfer capacity of heat sink having trapezoidal fins shows maximum value as compared to other. Micro-channels heat sink with rectangular, triangular and trapezoidal shape fins shows marginal variation in temperature and heat transfer coefficient. Whereas heat sink with cuboidal shows much variation. It is necessary for manufacturer to analyse the flow behaviour of fluid inside the micro-channel; therefore velocity variation is very important. Temperature distribution is also necessary to accurately design the solid model of heat sink.

References

[1] Yogesh K. Prajapati (2019), "Influence of fin height on heat transfer and fluid flow characteristics of rectangular microchannel heat sink", International Journal of Heat and Mass Transfer 137 (2019) 1041–1052.

[2] Ammar A. Hussain, Basim Freegaha, Basima Salman Khalafa, Hossein Towsyfyan (2019), "Numerical investigation of heat transfer enhancement in plate-fin heat sinks: Effect of flow direction and fillet profile", Case Studies in Thermal Engineering 13 (2019) 100388.

[3] Dawei yang, Yan wang, Guifu ding, Zhiyu jin, Jinhong zhao & Guilian wang 2017, ' Numerical and experimental analysis of cooling performance of single-phase array microchannel heat sinks with different pin-fin configurations', Applied thermal engineering, vol. 112, pp. 1547-1556.

[4] Si-NingLi, Hong-NaZhang, Xiao-BinLi, QianLi, Feng-ChenLi, ShizhiQian & Sang WooJoo 2017, 'Numerical study on the heat transfer performance of non-Newtonian fluid flow in a manifold microchannel heat sink', Applied Thermal Engineering, vol. 115, pp. 1213-1225.

[5] Lei Chai , Guo Dong Xia & Hua Sheng Wang 2016, 'Numerical study of laminar flow and heat transfer in microchannel heat sink with offset ribs on sidewalls', Applied Thermal Engineering, vol. 92, pp. 32–41.

[6] C. Anbumeenakshi, M.R. Thansekhar, (2016),"On the Effectiveness of a Nanofluid Cooled microchannel heat sink under non-uniform heating condition",http://dx.doi.org/10.1016/j.applthermaleng.2016.11.144.

[7] Anurag Dahiya, Satbir Singh Sehgal & Harpreet Singh 2016, 'CFD and Heat Transfer Analysis of Microchannel Heat Sink with Different Inlet and Outlet Flow Arrangements', Indian Journal of Science and Technology, vol. 9, no. 36, pp. 436-447.

[8] Hamdi E Ahmed, Ahmeda, MI, Islam MF Seder & Salman, BH 2016, 'Experimental investigation for sequential triangular doublelayeredmicrochannel heat sink with nanofluids', International Communications in Heat and Mass Transfer, vol. 77, pp. 104-115.

[9] Benjamín Herrmann - Priesnitz, Williams R Calderón - Muñoz, Alvaro Valencia & RodrigoSoto 2016, 'Thermal design exploration of a swirl flow microchannel heat sink for high heat flux applications based on numerical simulations', Applied Thermal Engineering, vol. 109, pp. 22-34.

[10] Weerapun Duangthongsuk, Somchai Wongwises (2015),"A Comparison of the Heat Transfer Performance and Pressure Drop of Nanofluid- Cooled Heat Sinks with Different Miniature Pin Fin Configurations", Experimental Thermal and Fluid Science (2015), http://dx.doi.org/10.1016/j.expthermflusci.2015.07.019.

[11] Assel Sakanova, Chan Chun Keian & Jiyun Zhao 2015, 'Performance improvements of the micro-channel heat sink using wavy channel and naofluids', International Journal of Heat and Mass Transfer, vol. 89, pp. 59-74.

[12] Ngoctan tran, Yaw – jan chang, Jyh tong & Thanhtrancy dang 2015, 'Numerical and Experimental Investigations on Heat Transfer of Aluminum Microchannel Heat Sinks with Different Channel Depths', International journal of mechanical engineering and robotics, vol.4, no. 3, pp. 204-208.

[13] Daeseok Jang, Se-Jin Yook, Kwan-Soo Lee (2014)," Optimum design of a radial heat sink with a fin-height profile for high-power LED lighting applications", Applied Energy 116 (2014) 260–268.

[14] Daeseok Jang, Se-Jin Yook, Kwan-Soo Lee (2014)," Optimum design of a radial heat sink with a fin-height profile for high-power LED lighting applications", Applied Energy 116 (2014) 260–268.

[15] Yue-Tzu Yang, Kuo-Teng Tsai, Yi-HsienWang & Shih-Han Lin 2014, 'Numerical study of microchannel heat sink performance using nanofluids', International Communications in Heat and Mass Transfer, vol. 57, pp. 27-35