IJCRT.ORG ISSN : 2320-2882

INTERNATIONAL JOURNAL OF CREATIVE RESEARCH THOUGHTS (IJCRT)

An International Open Access, Peer-reviewed, Refereed Journal

COMPUTATIONAL FLUID ANALYSIS OF WATER-COOLED HEAT EXCHANGER IN MICRO-CHANNELS WITH TRAPEZOIDAL RIBS

[1]Manish Kumar Upadhyay, [2]Dr. Ajay Kumar Singh, and [3]Dr. Parag Mishra
[1]Research Scholar, [2]Professor & Head, [3]Associate Professor
, Department of Mechanical Engineering,
Radharaman Institute of Technology and Science, Bhopal, Madhya Pradesh, India

Abstract - The main objective of this study is to enhance the thermal efficiency of rectangular Micro-channel heat sink (MCHS) by creating the interruption in the flow path using three identical trapezoidal ribs along the channel length. There was a several, thorough research on heat transfer performance and flow characteristics through Micro-channel heat sink for the different kind of ribs shapes and other parametrical configurations. In this study trapezoidal ribs are aligned from each other at an offset of 2.5mm while taking one rib at the Centre of the channel. The Reynolds number is used within the range of 200-1000, aiming to identify the best operating conditions for maximizing heat transfer efficiency. The study also scrutinized the relationship between fluid flow characteristics, specifically inlet velocities, and resulting heat transfer rates by determining the Nusselt number at various fluid inlet velocities. This indicates that the microchannels with trapezoidal ribs proved more efficient heat transfer enhancement and better thermal performance compared to the previous VR-MC model. The higher Nusselt numbers suggest improved cooling efficiency and enhanced heat transfer in practical applications utilizing microchannels with trapezoidal ribs.

Keywords: Microchannels, Reynolds number, Nusselt number, trapezoidal ribs.

I. INTRODUCTION

A small heat transfer component called a micro heat exchanger is created to transmit heat between two fluids. It is frequently employed in situations requiring high heat transfer rates and constrained area. The name "micro" alludes to the heat exchanger's diminutive size, with measurements generally in the micro- to millimeter-range. The two fluids usually travel over a number of tubes or tunnels in a micro heat exchanger's architecture. The channels can have different shapes, such as circular, rectangular, or serpentine, and can be stacked parallel or in a cross-flow pattern. Because the walls between the channels are often thin, there is a significant surface area for heat transmission while using less fluid overall. Micro heat exchangers have a wide range of practices, such as cooling electronics, recovering heat from chemical reactions, and regulating temperature in microfluidic devices. They differ from conventional heat exchangers in a number of ways, including rapid heat transfer rates, minimal pressure reductions, and small fluid volumes.

II. LITERATURE

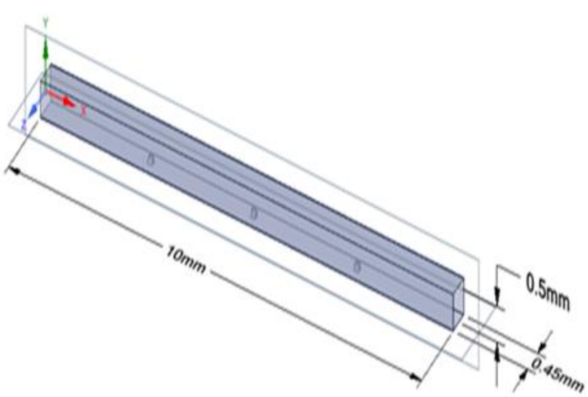
The Tuckerman et al. In this Research paper, author discussed about the issue of accomplishing compact, high-performance constrained fluid cooling of planar coordinates circuits has been examined. The convective heat-transfer coefficient(h) between the substrate and the coolant was found to be the essential hindrance to accomplishing low resistance of thermal. In the case of laminar flow in narrowed channels, the convective heat-transfer coefficient is inversely proportional to the channel width, and by this research paper we concluded that slimmer channels result in advanced heat-transfer coefficients, making microscopic channels required for operational cooling. This design approach aims to improve the convective heat transfer and improve the cooling performance.[1] Adams et al. In this research paper author states that an analysis was conducted on the forced convection of water in circular microchannels. The circular microchannels consists of diameters of 0.76 mm and 1.09 mm, and from this study we concluded that the Nusselt numbers for these microchannels were greater than what customary associations for larger channels would predict.[2]Ambatipudi et al. In this Research Paper, author discussed about the impact of many factors on the thermal performance of the device. Specifically, the effects of channel aspect ratio, Reynolds number (a dimensionless parameter indicating the flow regime), and the number of channels are studied, and by this research we concluded that the Nusselt number, which symbolizes the convective heat transfer, is greater for a system with a larger number of channels and a grander Reynolds number. This suggests that increasing the number of channels and the flow rate (indicated by a higher Reynolds number) enhances the heat transfer efficiency in the device.[3] Wu et al. In this Research Paper, author discussed about microchannel having cross-section as trapezoidal. The fluid is taken as deionized water. The hydraulic diameters of these microchannels range from 25.9 μm to 291.0 μm, and by this research paper we concluded that the friction constant of these microchannels is significantly influenced by the size of microchannels, defined as the ratio of the bottom width (Wb) to the top width (Wt) of the trapezoidal cross-section.[4] Rosaguti et al**.** In this research paper, author discussed about periodic serpentine microchannels with circular cross-sections, not pipes. They used computational fluid dynamics to evaluate the flow and heat transfer features at Reynolds numbers up to 200. we concluded that the forming of dean vortices induces significant enhancement in heat transfer as compared to the straight pipe.[5] Chein et al. In the research paper, author explored the fluid flow and heat transfer in microchannel heat sinks using numerical methods. They working the finite-volume scheme to solve the three-dimensional leading equations for fluid flow and heat transfer, and by this study we concluded that due to the variations in inlet/outlet of microchannels, the temperature distribution is different. Due to microchannel arrangement pressure drop is also different.[6] Sui et al. In this research paper, author's study is focused on numerically simulating the laminar liquid-water flow. The heat transfer occurs in three-dimensional wavy microchannels with a rectangular cross-section, by this research paper we concluded that the use of wavy microchannels with a rectangular cross-section could excellently enrich heat transfer performance through the generation of Dean vortices. These microchannels offered better convective fluid mixing compared to straight microchannels.[7] Mohammed et al. investigated that focuses on the numerical investigation of heat transfer and water flow characteristics in a wavy microchannel heat sink. The cross-section of heat sink is rectangular. The study studies many wavy amplitudes ranging from 125 to 500 μm and Reynolds numbers between 100 and 1000. Straight micro-channels are used to compare the results, and by this research paper we concluded that for considering constant cross-section of the channels, heat transfer performance is much better in case of wavy microchannels as compared to the straight microchannels and pressure drop is also much lesser in cash of the wavy micro-channels than straight micro-channels.[8] Chai et al. In this research paper, author discussed the properties of pressure drop and heat transfer features. It is established on several dimensions and sites of rectangular ribs in the transverse microchambers. They have used different size and position of rib, and by this research paper we concluded that for interrupted microchannels with ribs having optimum dimensions and the location for better heat transfer performance. For different Reynolds no. values it shows different performance. For a fixed dimension of rib there will be a optimum scope of operation. [9] Wang et al. In this research paper, author discussed that the inclination angle of the microchannel heat sink significantly affects the convective boiling heat transfer characteristics of the dielectric fluid, and by this research paper we concluded that A 45° upward inclination provides the highest heat transfer coefficient, while downward orientations lead to a deterioration in heat transfer performance. The findings suggest that slug velocity plays a crucial role in the observed heat transfer enhancement under inclined arrangements.[10] Anastasiou et al. In this research, author discussed that the effect of numerous fluid parameters, such as physical properties and flow rate, as well as the geometric appearances of the microchannels and the leaning angle on the geometrical appearances of the liquid phase. Since macroscopic associations are often not appropriate in the microscale, and by this research we concluded that the objective of the analyze is to improve predictive methods for estimating the geometrical appearances of the liquid phase, including the liquid film thickness, shape of the gas/liquid interface, and mean residence time.[11] Ahmed et al The numerical simulation inspects the effect of numerous geometrical constraints on laminar water flow and forced convection heat transfer appearances in a grooved microchannel heat sink. The study aims to optimize the design of an aluminum heat sink by allowing for four geometry variables: depth, tip length, pitch, and orientation of the cavities. These variables can alter the shape of the cavities from triangular to trapezoidal and then to rectangular, and by this research we concluded that the trapezoidal groove with specific dimensional ratios is the optimum design for better heat transfer with Nusselt no. rise of 51.59% as well as friction factor rise of 2.35%.[12] Yadav et al. In this research, author discussed about the improved heat transfer performance and reduced average surface temperature achieved through the use of extended surface microchannels. For the three different extended surface configuration considered for the analysis. i.e. upstream finned microchannel, downstream finned microchannel and complete finned microchannel and compared these all with the straight rectangular microchannel, and by this research we concluded that the at lower Reynolds no. upstream finned microchannel having much better heat transfer performance than rest others. Average temperature on the surface is also reduced significantly for microchannels with extended surface. [13] Aliabadi et al. In this research paper, author discussed about the cooling performance of a sinusoidal-wavy Mini channel heat sink with a cross-section of square shape. The study examines the effects of two specific geometrical parameters of the Mini channel: wave length and wave amplitude. The heat transfer and fluid flow features of the mini channels are achieved and associated with those of a straight Mini channel heat sink, and by this research we concluded that the heat transfer performance of the sinusoidal-wavy mini channel is much better as compared to the straight mini channel heat sink. Heat transfer coefficient as well as drop in pressure for sinusoidal-wavy mini channel heat sink increases proportionally. [14] Nandi et al. In this research paper, author discussed about numerical investigation of the unsteady laminar fluid flow and heat transfer, which are developing simultaneously inside a two-dimensional wavy microchannel, due to the velocity component which is varying sinusoidally at inlet. The flow in the microchannel was both thermally and hydrodynamically emerging, meaning that the temperature of the channel walls was kept unvarying while the flow was developing along the channel, by this research paper we concluded that velocity varying sinusoidally at inlet will have better heat transfer performance than that of the steady flow wavy microchannel. [15] Gholami et al. In this research paper, author discussed about the two-dimensional rectangular channel with a length-to-height proportion of 150. The main objective of the study was to scrutinize the impact of diverse rib shapes on the performance and heat transfer features of the nano-fluid flow in the quadrilateral microchannel. We also concluded that the rib shape is also one of major factors that affect the behavior of fluid and heat transfer performance. [17]

Wang et al. In this research paper, author discussed about how incorporation of bidirectional ribs in a microchannel heat sink can significantly improve heat transfer compared to the traditional vertical or spanwise ribs. It comprises of the both vertical and spanwise ribs simultaneously in order to become bidirectional ribs microchannel. The findings provide valuable insights for optimizing the design of microchannel heat sinks for improved cooling efficiency. For the research Reynolds no. has been taken between 100 and 1000. From this research we concluded that the Nusselt no. of bidirectional ribs microchannel is 1.4 to 2 times more than that of the vertical ribs microchannel and 1.2 to 1.42 times better than that of the spanwise ribs microchannel heat sink. [18] Yuan et al. In this research paper, author discussed about the design of non-uniform wavy microchannels, specifically the divergent wavy microchannel with the peak deviation position located away from the fluid entrance, can knowingly increase the heat transfer performance. The findings provide insights into the thermal-hydraulic behavior and enhanced heat transfer mechanisms in such microchannels, contributing to the development of more efficient microchannel heat sinks. From this research, we concluded that thermal performance of divergent wavy micro-channel having peak location far from fluid inlet shows better thermal performance as compared to the uniform and convergent wavy microchannels.[19]

Wang et al. In this research paper, author discussed about the flow and heat transfer features of a microchannel heat sink with truncated ribs on the sidewall. The numerical analysis covers a range of Reynolds numbers from 100 to 1000. From this research we concluded that in any of both the cases i.e. staggered or parallel cases, the Truncated ribs will increase the thermal performance near truncated intervals and decrease heavily gross pressure drop. Hence availability of truncated ribs can increase the gross thermal performance as well as reduction in pressure drop without loss in heat transfer.[20]

III. PRESENT WORK

A. *Objective*



Main objectives of this research are as follows:

1. The primary goal of this research is to improve thermal efficiency of the Micro-channel heat sink (MCHS).
2. To optimize the rate of heat transfer(Q) in microchannels with trapezoidal ribs by varying the Reynolds number within the range of 200-1000 to identify the optimal operating conditions that maximize heat transfer efficiency.
3. To study how different sizes and arrangements affect the heat transfer and the overall cooling performance in microchannels through computational methods.

B. GEOMETRY MODELING AND MESHING

Table 1: Thermo-physical properties of Water and Copper.

Property	Symbol	Water	Copper	Unit
Density	ρ	998.2	8978	kg/m^3
Specific heat at constant pressure	C_P	4182	381	J/kg-K
Dynamic viscosity	μ	0.00103		kg/m-s
Thermal conductivity	kf	0.6	387.6	W/m-K

Geometrical Inputs:

Height of the rectangular channel, (H) = 0.5mm

Width o the rectangular channel, (W) = 0.45mm

Cross sectional area(A)

A = H *W = 0.5* 0.45 = 0.000000225m^2

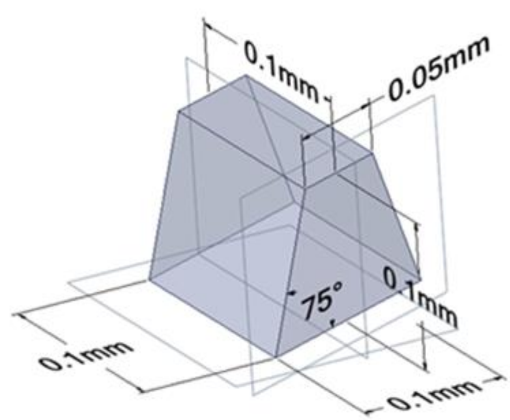
Hydraulic Diameter (D)

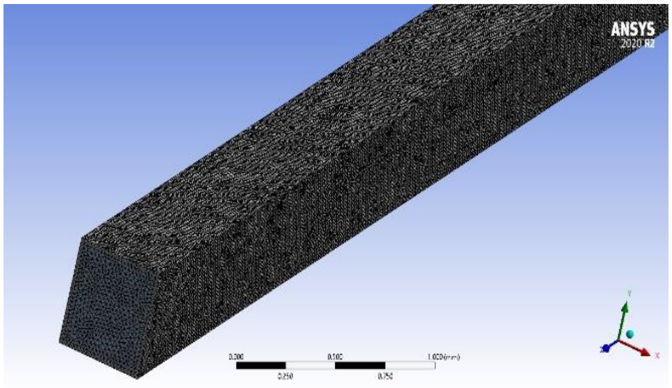
$$D = \frac{2\,W\,H}{W + H}$$

$$D = \frac{2(0.45 * 0.5)}{0.45 + 0.5}$$

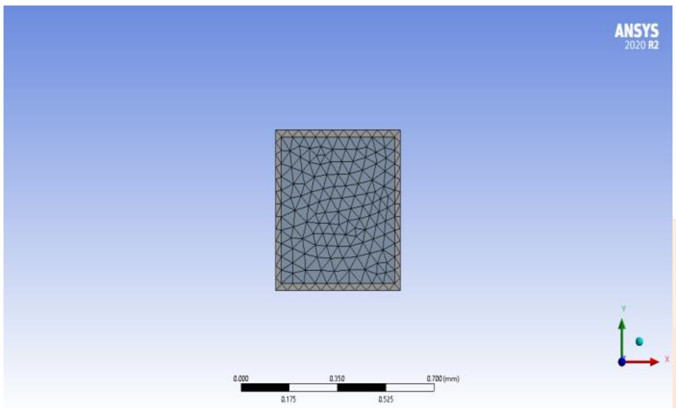
D = 0.4736842105mm

D = 0.00047368421m

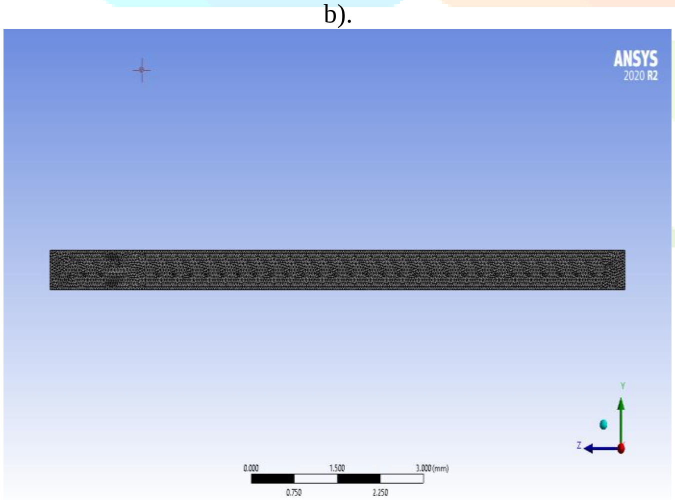




a).



b).



c).

Fig. 1 - Meshing created in ANSYS 2020 R2 (a) Isomeric view (b) front view (c) Side view.

Figure 1(a) shows the meshing of the model. It should be well-distributed throughout the domain to ensure that the solution is well-defined and doesn't depend on the location of the mesh. That's why we have used fine meshing.

Size of the mesh element/grid = 0.04mm

No. of mesh elements = 242605, no. mesh of nodes = 58661

Figure 1(b) and 1(c) represents the side view and front view of the meshed model respectively.

The commercial software ANSYS Fluent was used to conduct numerical simulations of fluid flow. Prior to the simulation, the computational domain was meshed in Fluent. Upon opening Fluent, 3D analysis was selected, and the exported mesh was read. The entire mesh was checked for aspect ratio and mesh quality. In the general menu of the setup, pressure-based and steady-state conditions were chosen, and gravity was taken 9.8 m/s in the downward y-direction. For the model selection, laminar flow was chosen, and both copper and water were designated as solid and fluid domains, respectively. The thermophysical properties of the solid and fluid were assumed to be constants and are listed in Table.

In this study, the fluid was supposed to enter the microchannel with a uniform velocity at varying Reynolds numbers. The inlet temperature of the fluid was preserved constant at 293 K and a heat flux of 1000000 W/m^2 was applied to the surface. The surface was subjected to a convective heat transfer condition with a convective heat transfer coefficient of 10 W/m^2-K at a free stream temperature of 293 K. A coupled heat transfer boundary condition was applied between the solid and fluid interface while the walls were measured adiabatic and exposed to a no-slip boundary condition.

C. Validation for microchannel with ribs

It's great to hear that the graphs obtained from your computational results are showing excellent results. Comparing them with numerical findings and experimental results is an important step in validating and verifying the accuracy of your work.

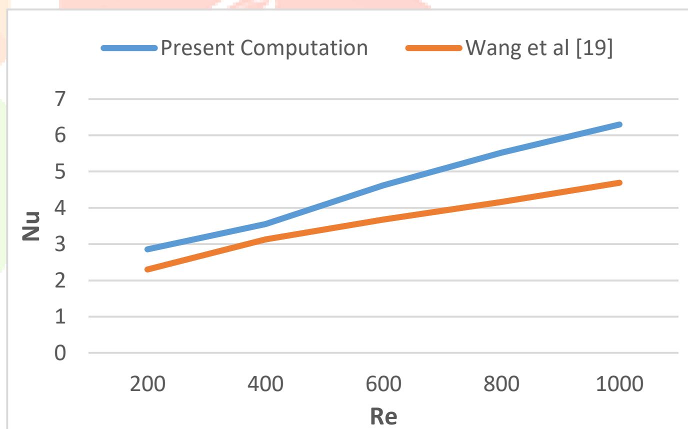


Fig.2-, graph showing computational and experimental data of Nusselt no. with varying Reynolds no

When you mention "Wang et al. [19]," it seems like you are referring to a specific research paper or study conducted by Wang and colleagues. By mentioning this reference, you are indicating that you have compared your computational results not only with numerical findings but also with experimental data published by Wang et al. Comparing your results with both numerical findings and experimental data adds credibility to your work. This is common in scientific research, as there can be various factors contributing to discrepancies between computational and experimental results, such as modelling assumptions, experimental conditions, or measurement errors. Overall, by comparing your computational results with numerical findings and experimental data from Wang et al., you are providing a comprehensive evaluation of your work and strengthening the validity of your findings.

IV. RESULT AND DISCUSSION

A. Surface Nusselt Number variation with Position along Z-axis

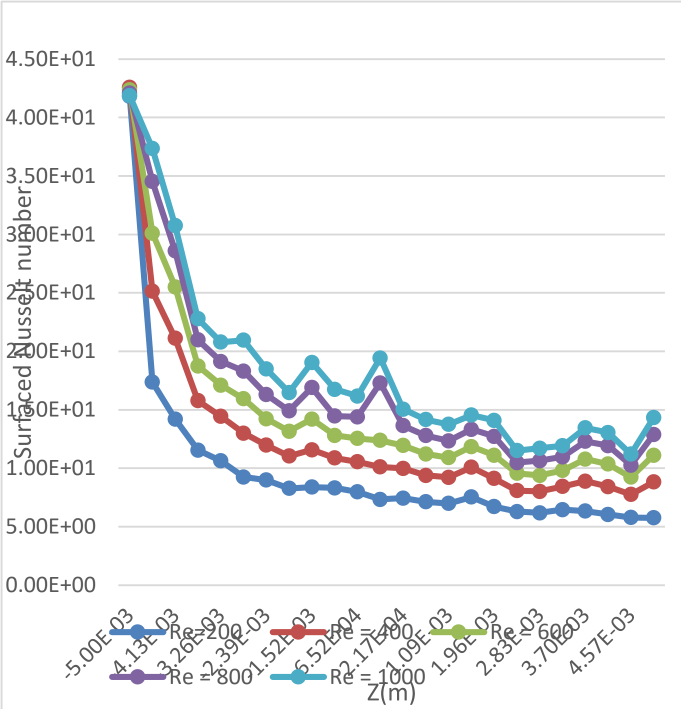


Fig. 3: Graph between Surface Nusselt number and position Z.

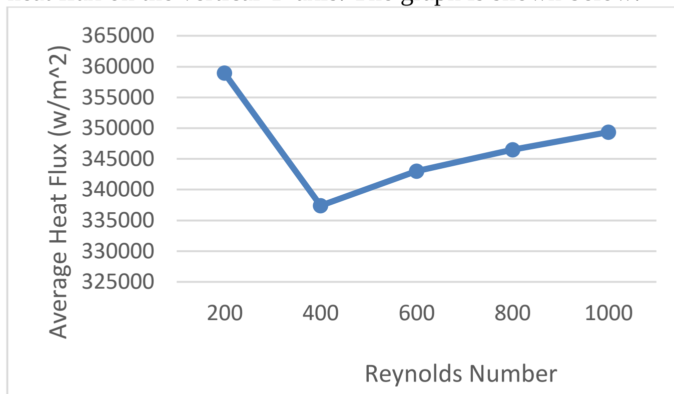
Graph 3: Variation of surface Nusselt no. with Z-axis for the different values of Reynold no. Position of lines on the microchannel surface has been taken from Z = -0.005 as inlet to the Z = 0.005. considering the more no. of lines on the surface of the microchannel along z-axis in order the analyse the accurate variation of surface Nusselt no. along the fluid flow. The graph illustrates the relationship between the position (Z) and the surface Nusselt number at a Reynolds number of 200 to 1000. The position is plotted on the horizontal X-axis, while the surface Nusselt number is plotted on the vertical Y-axis. From the graph, we can observe the following trends, As the position (Z) increases from negative values to positive values, the surface Nusselt number generally decreases.

Table 2: Reynolds Number, Average Heat Flux, Average Heat Transfer coefficient, Average Pressure, Average Nusselt Number.

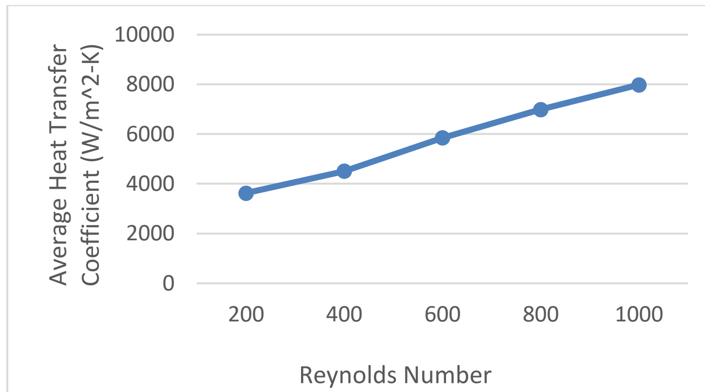
Reynolds Number [Re]	Average Heat Flux (W/m^2)	Average Heat transfer coefficient (W/m^2-K)	Average Pressure (Pa)	Average Nusselt number [Nu]
200	358915.8373	3620.428658	663.2061818	2.858
400	337382.8614	4501.734809	1874.922573	3.554
600	342996.8255	5847.149911	3547.536245	4.616
800	346488.1009	6988.246988	5634.293736	5.517
1000	349334.9964	7977.039585	8114.223391	6.298

Graph

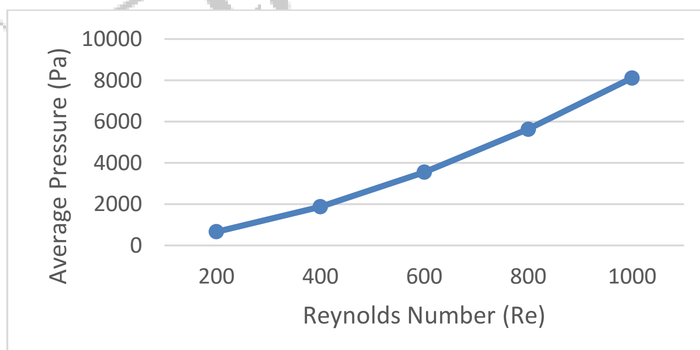
To create a graph based on the above data for different Reynolds numbers and the average heat flux, I will scheme Reynolds numbers on the horizontal X-axis and the average heat flux on the vertical Y-axis. The graph is shown below:



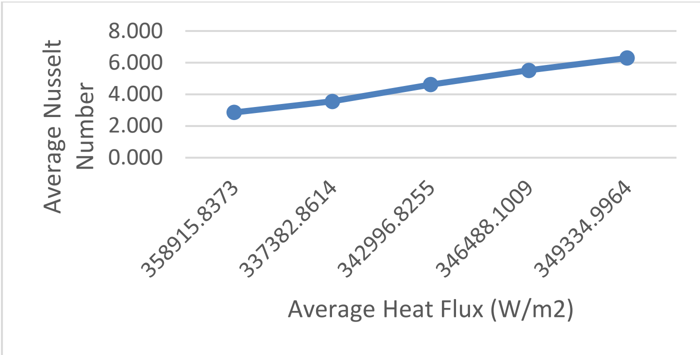
a).



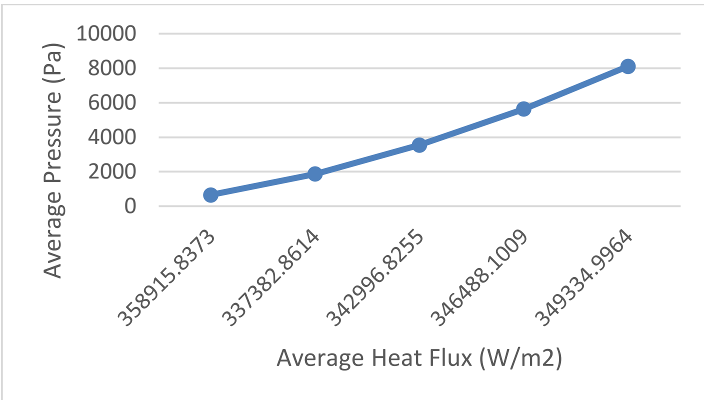
b).



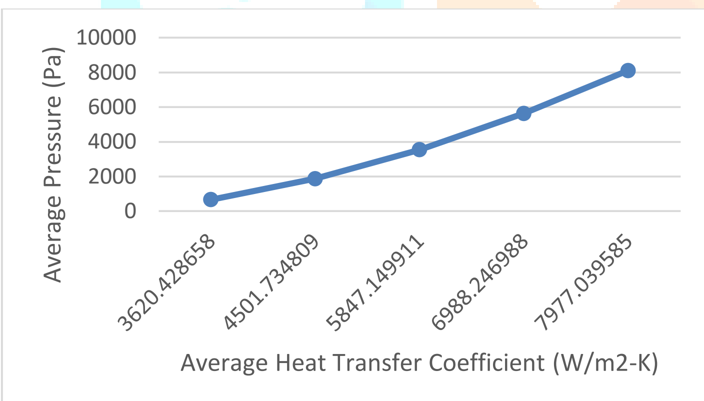
c).



d).



e).



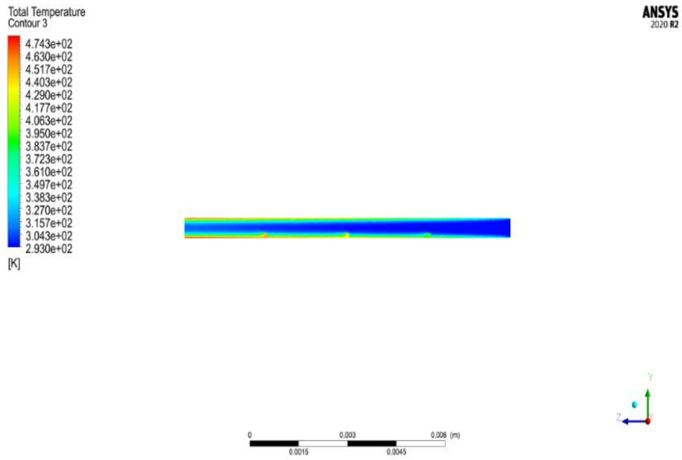
f).

fig 4(a). - graph showing variation of avg. heat flux with Reynolds no, 4(b). shows the variation of avg. heat transfer coefficient with Reynolds no, 4(c). shows the variation of avg. pressure with Reynolds no, 4(d). shows the variation of avg. Nusselt no. with avg. heat flux, 4(e). shows the variation of avg. pressure with avg. heat flux and, 4(f) shows the variation of avg. pressure with avg. heat transfer coefficient.

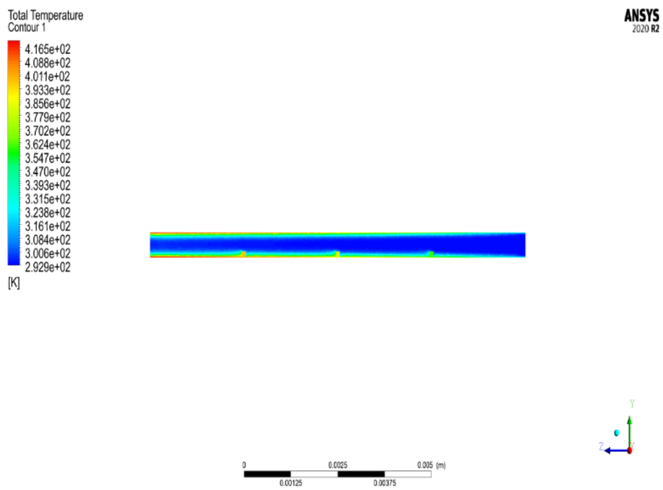
- From graph 4(a) we can conclude the as the Reynolds number surges from 200 to 1000, the average heat flux generally declines. The drop in average heat flux with growing Reynolds number suggests that as fluid flow becomes more turbulent (higher Reynolds number), the convective heat transfer becomes less efficient, resulting in a lower heat flux.
- From the graph 4(b) we can conclude that as the Reynolds number surges from 200 to 1000, the average heat transfer coefficient generally rises. The surge in the average heat transfer coefficient with increasing Reynolds number shows that as fluid stream turns out to be more turbulent (higher Reynolds number), the convective heat transfer becomes more efficient, causing in a greater heat transfer coefficient.
- From the graph 4(c) we can conclude that as the Reynolds number surges from 200 to 1000, the average pressure generally rises. The surge in average pressure with growing Reynolds number suggests that as fluid flow becomes more turbulent (higher Reynolds number), it results in increased pressure drop or resistance within the system.
- From the graph 4(d) we conclude that as the average Nusselt number surges, the average heat flux generally rises. The average Nusselt number characterizes the convective heat transfer efficiency, while the average heat flux denotes the amount of heat transferred per unit area. Therefore, a higher average Nusselt number indicates better convective heat transfer, leading to a higher average heat flux.
- From the graph 4(e) we can conclude that as the average pressure surges, the average heat flux generally rises. The increase in average heat flux with increasing average pressure suggests that higher pressure conditions contribute to more efficient heat transfer.
- From the graph 4(f) we can conclude that when the average pressure rises, due to the average heat transfer coefficient generally rises. The surge in average heat transfer coefficient with growing average pressure advises that higher pressure conditions contribute to more efficient heat transfer.

B. Impact of the rib on the temperature contours

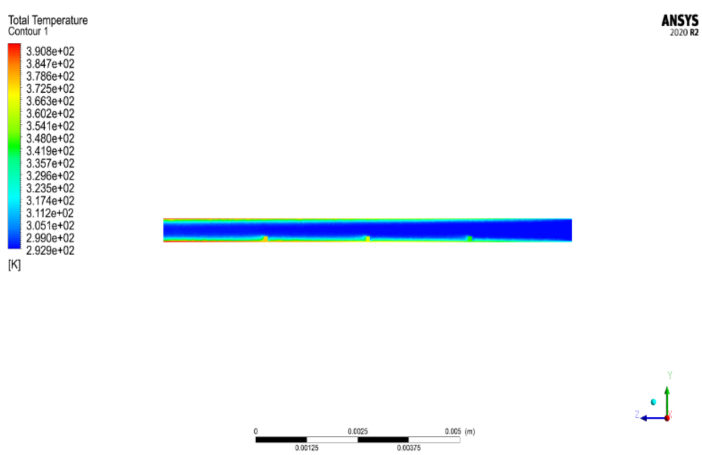
Temperature contours display the variation of temperature at different locations within the simulated domain. These contours are graphical representations that use colour gradients or isolines to illustrate the temperature values across the surfaces or sections of interest. They help engineers and researchers understand how temperature changes within the simulated system, allowing them to identify areas of high or low temperatures and analyse heat transfer patterns. Temperature contours in ANSYS Fluent are generated by solving the leading calculations for fluid stream and heat transfer, accounting for properties such as convection, conduction, and radiation. By visualizing temperature gradients, users can gain insights into thermal behaviour, identify hotspots or regions with inadequate cooling, and make informed design decisions to optimize the system's thermal performance. There are different temperature contours are taken at different number of Reynolds number.



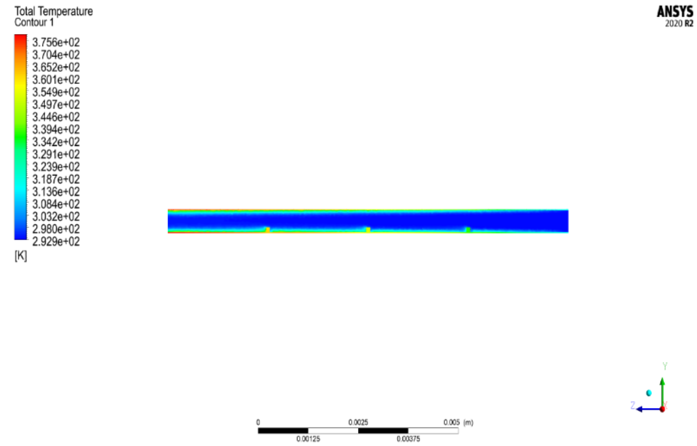
a).



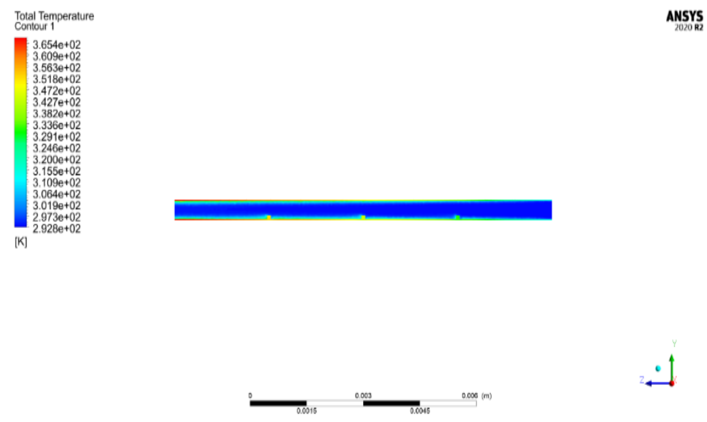
b).



c).



d).

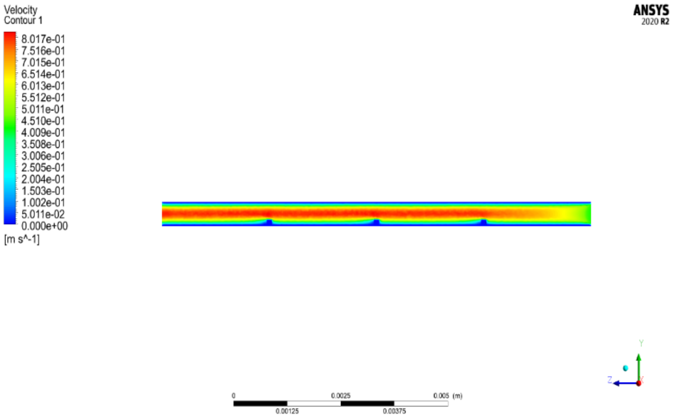


(e)

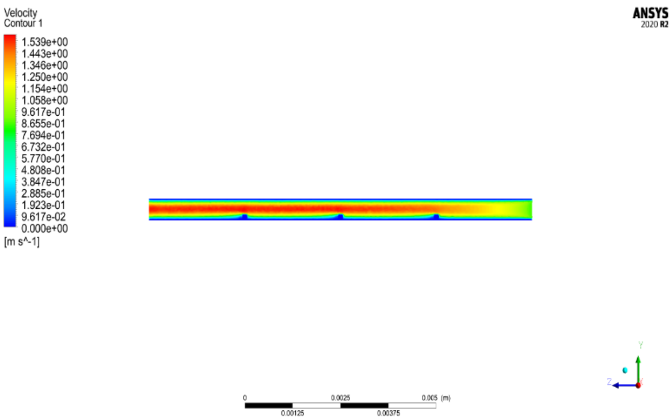
fig 4.11 shows the total temperature contour at (a) Re = 200, (b) Re = 400, (c) Re = 600, (d) Re =800 and (e) Re = 1000.

C. Impact of the rib on the velocity contours

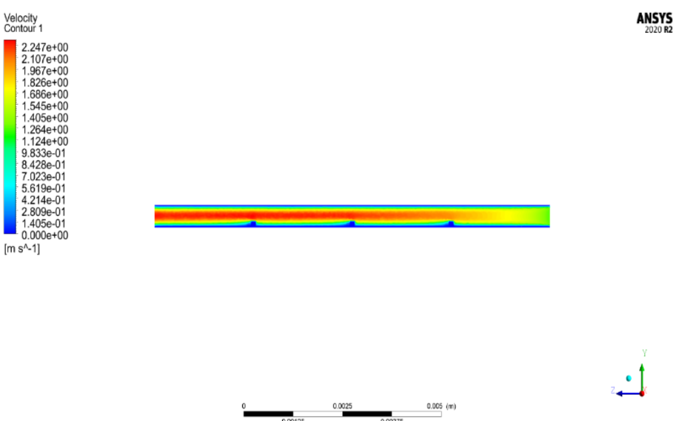
Velocity contours display the magnitude and direction of fluid velocities at different locations within the simulated domain. These contours use colour gradients or isolines to represent the velocity values across surfaces or sections of interest. Velocity contours in ANSYS Fluent are generated by solving the governing equations for fluid flow, taking into account factors such as fluid viscosity, turbulence, and pressure gradients. By examining velocity gradients, users can analyse flow patterns, identify areas of recirculation, turbulence, or flow separation, and make informed decisions to optimize the system's performance. These are different velocity contour are taken at different number of Reynolds number.



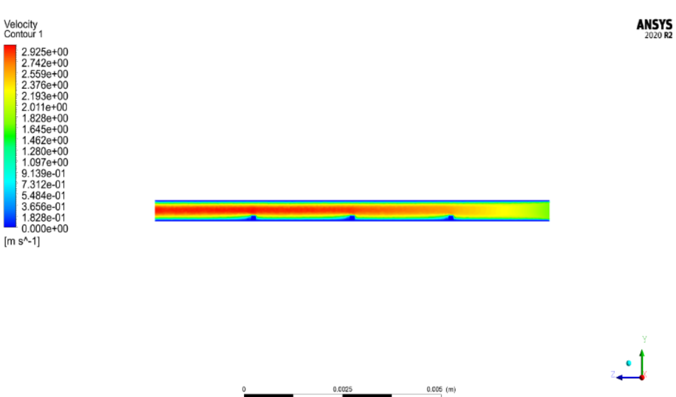
a).



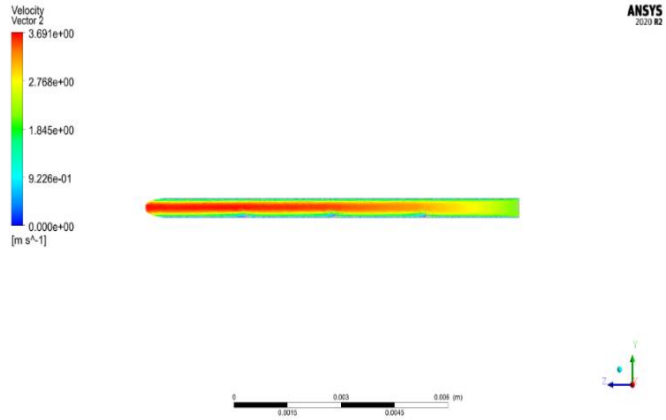
b).



c).



d).

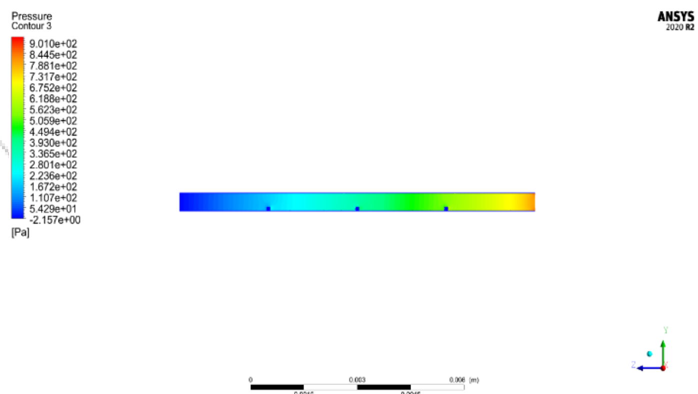


(e)

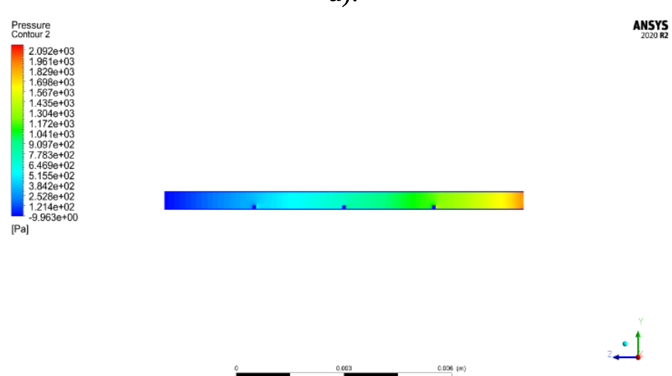
fig 5 shows the velocity contour at (a) re = 200, (b) re = 400, (c) re = 600, (d) re =800, and (e) re =1000

D. Impact of the rib on the Pressure

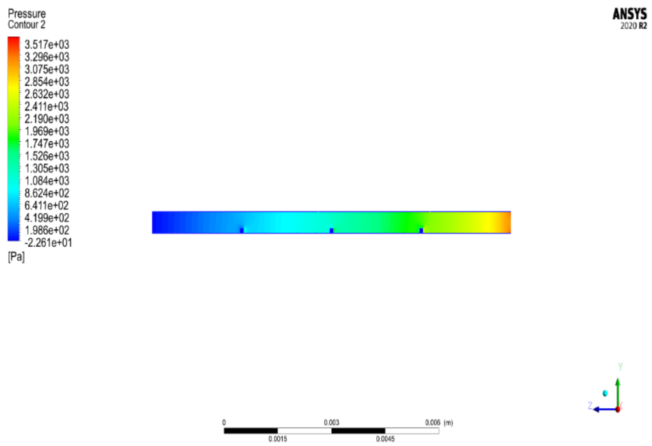
Pressure contours display the variation of pressure values at different locations within the simulated domain. These contours use colour gradients or isolines to depict the pressure values across surfaces or sections of interest. By visualizing pressure contours, engineers and researchers can gain insights into the pressure distribution, understand the flow behaviour. Pressure contours in ANSYS Fluent are obtained by solving the governing equations for fluid flow, considering factors such as fluid viscosity, fluid density, and velocity distribution. By analysing pressure contours, users can identify pressure gradients, pressure drops, flow restrictions, and regions of high or low pressure that may impact the performance or safety of the system under consideration. These insights enable engineers to optimize designs, identify potential issues, and make informed decisions to improve the efficiency and reliability of the simulated system.



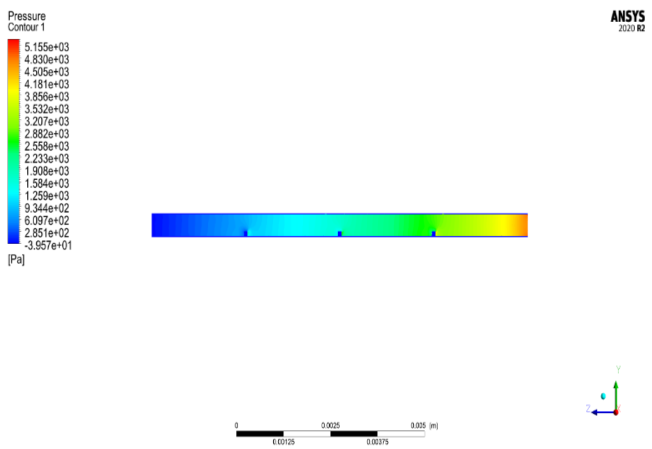
a).



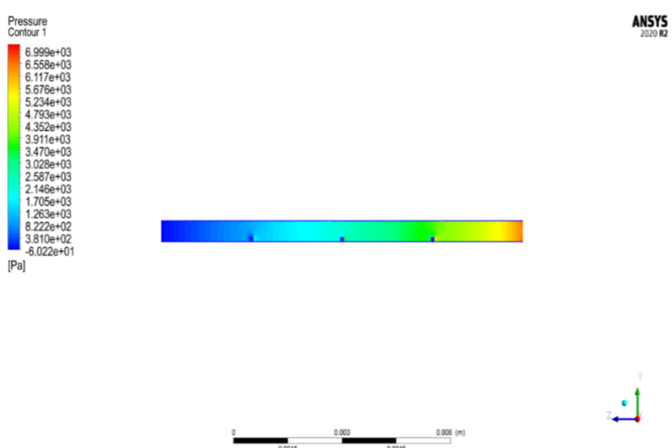
b).



c).



d).

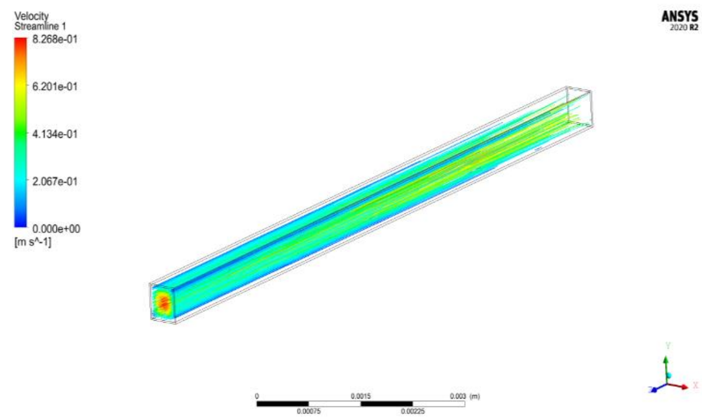


(e)

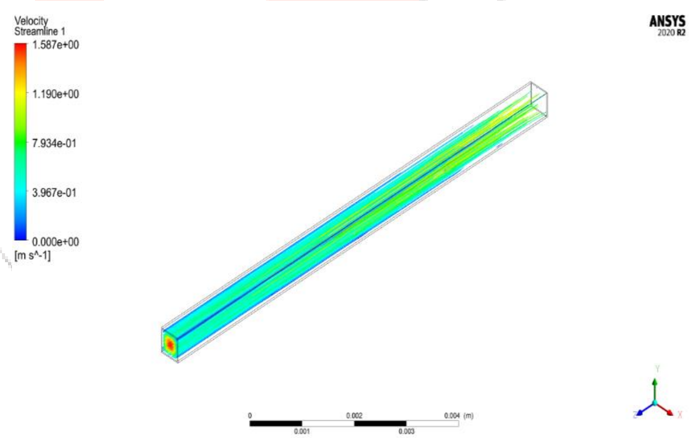
Fig 6: shows the Pressure contour at (a) Re = 200, (b) Re = 400, (c) Re = 600, (d) Re =800
and (e) Re = 1000

E. Impact of the rib on the streamline on fluid

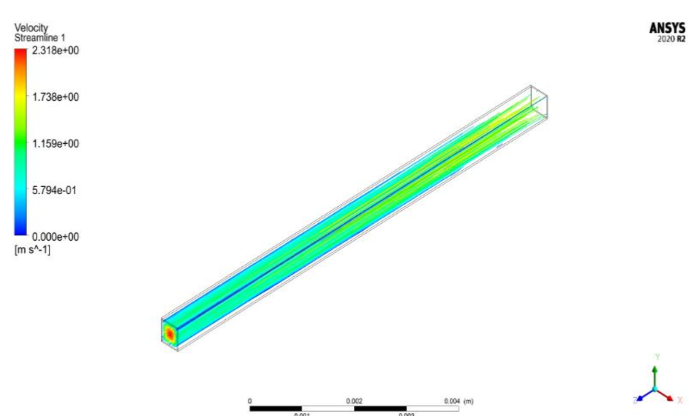
Streamline contours in ANSYS Fluent denote to the graphic image of fluid flow paths within a computational domain. These contours use lines or ribbons to represent the paths traced by fluid particles at different locations within the domain. By visualizing streamline contours, engineers and researchers can gain insights into the flow behaviour, understand the flow paths, and identify regions of interest such as stagnation points, separation zones, or recirculation areas. Streamline contours in ANSYS Fluent are generated by solving the governing equations for fluid flow, considering factors such as fluid velocity, viscosity, and boundary conditions. The software computes the trajectories of fluid particles based on these parameters and displays them in a manner that helps users visualize the flow paths within the simulated system. By examining streamline contours, users can analyse the flow patterns, identify areas of interest, such as regions of strong mixing or flow separation, and optimize designs for improved fluid flow characteristics. Streamline contours provide valuable information about the behaviour of the fluid, including its direction, speed, and patterns of flow, enabling engineers to make informed decisions regarding the design and optimization of their systems.



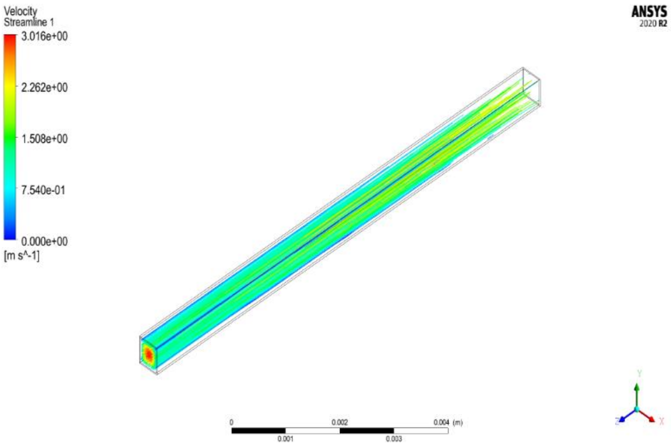
(a)



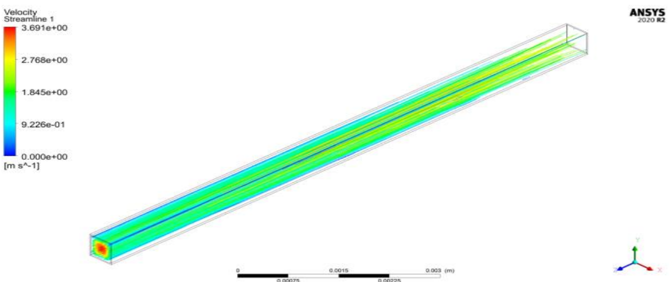
(b)



(c)



(d)



(e)

fig 4.14 shows the velocity streamline at (a) Re = 200, (b) Re = 400, Re = 600, (d) Re =800 and (e) Re = 1000

V. CONCLUSION

In this study, the attention was on studying models of microchannels with Trapezoidal ribs. The aim was to explore the heat transfer performance of these models under several conditions using numerical simulations. Microchannels with Trapezoidal ribs are a specific type of microchannel design that incorporates rib structures with trapezoidal cross-sections. These rib structures are intended to enhance heat transfer by increasing the surface area available for heat exchange and promoting fluid mixing.

- Upon comparing the computational result data of Reynolds number versus Nusselt number obtained in the present study it is evident that the Nusselt numbers in the present study are observed to be approximately 11% to 25% higher than those reported by Wang et al. [19].
- This assessment recommends that the heat transfer performance of the microchannels with Trapezoidal ribs, as examined in the present study, is more efficient in terms of heat transfer enhancement than the VR-MC model examined by Wang et al. [19].
- We can observe the following trends: When Reynolds number increase, average heat flux decreases due to this phenomenon fluid flow becomes turbulent, the capable of convective heat transfer becomes less, due to this heat flux becomes lower.
- We can observe that when Reynolds number increases in the graph, average heat transfer coefficient also increases respectively due to this fluid velocity becomes higher, the effect of convective heat transfer increased, causing in a greater heat transfer coefficient.

ACKNOWLEDGEME T

The authors are grateful to Director, Dr. R K Pandey, Professor and Head of Department, Mechanical Engineering, Dr. Ajay Singh, Associate Professor, Dr, Parag Mishra, Radharaman Institute of Technology and Science Bhopal, for their support and guidance.

REFERENC S

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

[2] Adams, T. M., Abdel-Khalik, S. I., Jeter, S. M., & Qureshi, Z. H. (1998). An experimental investigation of single-phase forced convection in microchannels. International Journal of Heat and Mass Transfer, 41(6-7), 851-857.

[3] Ambatipudi, K. K., Rahman, M. M. (2000). Analysis of conjugate heat transfer in microchannel heat sinks. Numerical Heat Transfer: Part A: Applications, 37(7), 711-731.

[4] Wu, H. Y., & Cheng, P. (2003). Friction factors in smooth trapezoidal silicon microchannels with different aspect ratios. International journal of heat and mass transfer, 46(14), 2519-2525.are

[5] Rosaguti, N. R., Fletcher, D. F., Haynes, B. S. (2005). Laminar flow and heat transfer in a periodic serpentine channel. Chemical Engineering & Technology: Industrial Chemistry-Plant Equipment-Process Engineering-Biotechnology, 28(3), 353-361.

[6] Chein, R., & Chen, J. (2009). Numerical study of the inlet/outlet arrangement effect on microchannel heat sink performance. International Journal of Thermal Sciences, 48(8), 1627-1638.

[7] Sui, Y., Teo, C. J., Lee, P. S., Chew, Y. T., Shu, C. (2010). Fluid flow and heat transfer in wavy microchannels. International Journal of Heat and Mass Transfer, 53(13-14), 2760-2772.

[8] Mohammed, H. A., Gunnasegaran, P., Shuaib, N. H. (2011). Numerical simulation of heat transfer enhancement in wavy microchannel heat sink. International Communications in Heat and Mass Transfer, 38(1), 63-68.

[9] Chai, L., Xia, G., Zhou, M., Li, J., Qi, J. (2013). Optimum thermal design of interrupted microchannel heat sink with rectangular ribs in the transverse microchambers. Applied Thermal Engineering, 51(1-2), 880-889.

[10] Wang, C. C., Chang, W. J., Dai, C. H., Lin, Y. T., & Yang, K. S. (2012). Effect of inclination on the convective boiling performance of a microchannel heat sink using HFE-7100. Experimental thermal and fluid science, 36, 143-148

[11] Anastasiou, A. D., Gavriilidis, A., Mouza, A. A. (2013). Study of the hydrodynamic characteristics of a free flowing liquid film in open inclined microchannels. Chemical Engineering Science, 101, 744-754.

[12] Ahmed, H. E., & Ahmed, M. I. (2015). Optimum thermal design of triangular, trapezoidal and rectangular grooved microchannel heat sinks. International Communications in Heat and Mass Transfer, 66, 47-57.

[13] Yadav, V., Baghel, K., Kumar, R., Kadam, S. T. (2016). Numerical investigation of heat transfer in extended surface microchannels. International Journal of Heat and Mass Transfer, 93, 612-622.

[14] Khoshvaght-Aliabadi, M., Sahamiyan, M., Hesampour, M., & Sartipzadeh, O. (2016). Experimental study on cooling performance of sinusoidal–wavy minichannel heat sink. Applied Thermal Engineering, 92, 50-61.

[15] Nandi, T. K., Chattopadhyay, H. (2013). Numerical

investigations of simultaneously developing flow in wavy microchannels under pulsating inlet flow condition. International communications in heat and mass transfer, 47, 27-31.

[16] Odesola, I. F., Ashaju, A. A., Ige, E. O. (2017). Simulation of Fluid flow and Thermal Transport in Gravity-dominated Microchannel. International Journal of Scientific and Engineering Research, 8(1), 994-1001.

[17] Gholami, M. R., Akbari, O. A., Marzban, A., Toghraie, D., Shabani, G. A. S., & Zarringhalam, M. (2018). The effect of rib shape on the behavior of laminar flow of oil/MWCNT nanofluid in a rectangular microchannel. Journal of Thermal Analysis and Calorimetry, 134(3), 1611-1628.

[18] Wang, G., Qian, N., & Ding, G. (2019). Heat transfer enhancement in microchannel heat sink with bidirectional rib. International Journal of Heat and Mass Transfer, 136, 597-609.

[19] Yuan, D., Zhou, W., Fu, T., & Liu, C. (2020). Experimental and numerical investigation of heat and mass transfer in non-uniform wavy microchannels. International Journal of Thermal Sciences, 152, 106320.

[20] Wang, G., Chen, T., Tian, M., & Ding, G. (2020). Fluid and heat transfer characteristics of microchannel heat sink with truncated rib on sidewall. International Journal of Heat and Mass