

PAPER • OPEN ACCESS

CFD Simulation and Analytical Validation of Rectangular and Square microchannel heat sink with liquid water as cooling medium

To cite this article: Saurabh Narayan Pawar and Nilesh Balkishanji Totla 2020 *IOP Conf. Ser.: Mater. Sci. Eng.* **998** 012030

View the [article online](#) for updates and enhancements.

CFD Simulation and Analytical Validation of Rectangular and Square microchannel heat sink with liquid water as cooling medium

Saurabh Narayan Pawar^a, Nilesh Balkishanji Totla^b

¹Student, MIT Academy of Engineering, Alandi, Pune-412105, India

²Senior assistant professor, MIT Academy of Engineering, Alandi, Pune-412105, India

^asnpawar@mitaoe.ac.in, ^bnbtotla@mech.maepune.ac.in

Abstract: In present work, both analytical and CFD investigation was performed to check the microchannel heat sink cooling performance, made up of copper with two different microchannel cross sections with shape of rectangle and square. CFD analysis is conducted with water as working fluid for both of the cross sections and Reynolds number ranging from 600 to 1000. A copper microchannel heat sink with bottom size of 25.4mm×25.4mm and height of 2.384mm is selected and CFD analysis was carried out and at bottom constant heat flux boundary condition is assumed. For comparison between two cross sections, the hydraulic diameter used is same for both. Rectangular cross section microchannel showed better cooling performance as compared to square cross section microchannel. Both analytical and CFD simulation results of pressure drop in rectangular microchannel heat sink are in good agreement with a difference of 18.9%.

Keywords: CFD, Heat sink, Microchannel, Rectangular, Square

NOMENCLATURE

L	Microchannel heat sink length, mm	\dot{m}	Mass flow rate, kg/s
W	Microchannel heat sink width, mm	C_p	Specific heat at constant pressure, J/kgK
H	Microchannel heat sink height, mm	V	Velocity, m/s
D_h	Hydraulic diameter, mm	Q	Heat rate, W
H_r	Height of rectangular microchannel, mm	Nu	Nusselt number
W_r	Width of rectangular microchannel, mm	Re	Reynolds number
H_s	Height of square microchannel, mm	A_s	Surface area, m^2
W_s	Width of square microchannel, mm	k	Thermal conductivity, W/mK
$T_{f,i}$	fluid temperature at inlet, K	h	Coefficient of Heat transfer, W/m^2K
$T_{f,o}$	fluid temperature at outlet, K	N	No of microchannels
$(\Delta T)_f$	Fluid temperature difference, K	GREEK SYMBOLS	
$T_{f,avg}$	Average fluid temperature, K	ρ	Density, Kg/m^3
$T_{w,avg}$	Average wall temperature, K	μ	Dynamic viscosity, Pa-s
(ΔT)	Average wall and average fluid temperature difference, K		
A_{cs}	Microchannel cross sectional area, m^2		

INTRODUCTION

The importance of minichannels and microchannels is ever increasing because of compact size and effective as well as efficient heat transfer performance in compact electronic circuits. Because of technological advancement, dependency on computers is increased hence use of smaller electronics circuits is essential in computers. Along with usage of circuits, problem of heat dissipation has also been raised and which has been successfully solved by using minichannels and microchannels. The amount of literature of the use of different geometries and different fluids to get better heat transfer effects in minichannels and microchannels has been increasing because it removes



Content from this work may be used under the terms of the [Creative Commons Attribution 3.0 licence](https://creativecommons.org/licenses/by/3.0/). Any further distribution of this work must maintain attribution to the author(s) and the title of the work, journal citation and DOI.

good heat amount from small area. For better cooling performance, the working fluid, geometry, material of the heat sink must be so chosen that better results are achieved. Many researchers have conducted their study in this field and have given good results about microchannels and minichannels. Rajesh Sharma, Amoljit Singh Gill and Vikas Dhawan [1] performed heat transfer study in single phase through microchannels. For microchannel heat sink copper material was used and water as cooling agent was used. They stated that classical approach and experimental results of microchannel heat transfer are in good agreement. They observed that with increase in velocity pressure drop increases. CFD numerical study was conducted by Oana Giurgiu, Angela Pleșa and Lavinia Socaciu [2] with two different minichannel models which were included in plate heat exchanger structure. The geometric characteristics influence comparative study of two plates on heat transfer intensification was conducted. Minichannels which were analysed had inclination angles of 30° and 60° . they stated that plate heat exchanger minichannel model with 60° inclination angle provide best heat transfer. Manoj Rao and Sameer Khandekar [3] presented experimental and computational study related to thermo-hydrodynamics of single phase liquid flow developing simultaneously. Seven parallel semi-circular channels array made by copper substrate was used. Experimental laminar and turbulent flow Po and Nu numbers matched with computational model value. Experimental Reynolds number was in range of 300-3200 and deionized, distilled and degassed water was used in this study. Dr. B. S. Gawali, V. B. Swami and S. D. Thakre [4] investigated rectangular cross section straight microchannel heat sink with theoretical and experimental approach. The distribution of temperature, pressure drop, coefficient of heat transfer were some of the selected criteria for study. Working fluid used was water and Reynolds number was in range of 200 to 700. They found that with increase in microchannel length, water temperature increases.

O. Abouali and N. Baghernezhad [5] investigated numerically rectangular and arc shaped two types of groove which are fabricated in microchannel surface. Microchannel heat sink having single phase laminar flow was studied numerically for heat transfer and pressure drop characteristic analysis. Result showed that higher heat removal flux is shown by arc groove as compared to rectangular groove. Results also showed that heat removal flux is higher for grooved microchannel with lower mass flow rate and higher wall thickness. Yang Liu, Jing Cui, WeiZhong Li and Ning Zhang [6] presented forced convection heat transfer numerical study in microchannel with different microstructure. They stated that with inlet Reynolds number heat transfer performance improves. They stated that highest heat transfer performance is of v-shaped grooved microstructure. Single phase liquid flow in microchannels and minichannels was studied by Satish G. Kandlikar [7]. Author addressed the fundamental issues in micro scale liquid flow and gave the relation to calculate pressure drop for single phase liquid flow. Experimental study was conducted for rectangular microchannels with single phase flow By Poh-Seng Lee, Suresh V. Garimella and Dong Liu [8]. Microchannels considered was having width in the range from $194\ \mu\text{m}$ to $534\ \mu\text{m}$. Test piece contained ten parallel microchannels made up of copper. Reynolds number in range from 300 to 3500 was considered and Deionized water was used. Average deviation of 5% was seen between experimental data and numerical predictions obtained on basis of classical, continuum approach. By applying condition of uniform wall temperature circumferentially and uniform wall Heat flux axially to rectangular cross section microchannels, convective heat transfer of laminar flow in entrance region was studied by Poh-Seng Lee and S V. Garimella [9]. Microchannels with different aspect ratio was chosen. Aspect ratio was defined as channel height divided by channel width. Paisarn Naphon and Osod Khonseur [10] conducted experiments to study heat transfer characteristics and pressure drop in microchannel heat sink with constant heat flux consideration. Reynolds number was selected in range from 200-1000 and heat flux in range from $1.80\text{--}5.40\ \text{KW}/\text{m}^2$. it is stated that heat transfer performance enhancement and variation in pressure drop are affected by shape and size of microchannel surface roughness irregularities.

P. Promvonge, S. Sripattanapipat, S. Tamna, S. Kwankaomeng and C. Thianpong [11] numerically studied laminar periodic flow in square channel with three dimensional isothermal wall and 45° inclined baffles fitted on one channel wall. Reynolds number in range from 100 to 1200 was used in this study. The enhancement factor was found to be higher for 45° baffle than 90° baffle for all baffle height and Reynolds number. G. Gamrat, M. Favre-Marinet and S. Le Person [12] used two different approaches to study roughness effect on heat transfer in microchannel with laminar flow. 3D numerical simulation and 1D RLM models were used. Both the models agreed to show that with relative roughness Poiseuille number and Nusselt number increases. Xiaojin Wei and Yogendra Joshi [13] stated that microchannel stack require less pumping power than single microchannel for removing

certain rate of heat at smaller flow velocity at inlet because of large area of heat transfer. Authors designed simple thermal resistance network to study stacked microchannel heat sink and overall thermal performance. Tomasz A. Kowalewski, Jacek Szumbariski and Slawomir Blonski [14] investigated channels with wavy walls for instability of viscous incompressible flow with theoretical, numerical and experimental approach. It has been shown that correctly chosen waviness of wall will lead to flow destabilization at surprisingly low Reynolds number by linear stability analysis. Jing ZHOU, Zhiheng YAN and Qiang GAO [15] applied microchannel heat exchanger to heat pump and air conditioning system used residentially and commercially. Results obtained from experiment shown that by at most 4% heat pump system capacity can be increased with using microchannel heat exchanger. Also by at most 4% energy efficiency can be increased.

So, large amount of work is being done in the field of microchannels and minichannels with different working fluid and cross sections. But In the present study, comparison between rectangular and square cross section microchannel is studied and presented using CFD solution and analytical approach.

CAD MODEL

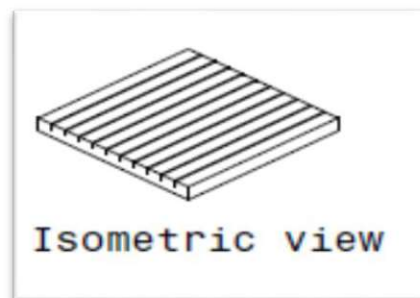


Figure 1: Isometric view of CAD model of rectangular microchannel heat sink

CFD which is Computational Fluid Dynamics, is a tool with which we can solve heat transfer and fluid flow related problems. The model of rectangular microchannel heat sink is drawn using CAD as shown in Fig 1. As given in [8], square copper block with dimensions of 25.4mm×25.4mm×70mm was used and machined to get heat sink. So in this present study, for CFD analysis purpose the dimensions of rectangular microchannel heat sink are taken to be L=25.4 mm, W=25.4mm, H=2.384mm. The dimensions of the rectangular microchannel heat sink are taken from [8]. The microchannel heat sink consists of N=10 parallel straight microchannels, same as that in [8], as shown in Fig 1. The fluid is allowed to flow through the microchannels for analysing the cooling performance as the base is given the constant heat flux. The working fluid used is water. For comparison purpose the CFD results of the rectangular microchannel was compared with square microchannel by keeping the hydraulic diameter of square microchannel same as that of the rectangular microchannel which is $D_h=0.318$ mm. The Reynolds number for current study was considered in range from 600 to 1000. The range of Reynolds number and working fluid was same for both rectangular and square microchannel. The number of microchannels was also same for both rectangular and square microchannel. The dimension of square and rectangular microchannel are shown in below table 1.

Table 1: Dimensions of rectangular and square microchannels

Dimensions of microchannels	height	width	D_h
Rectangular	$H_r=0.884$ mm	$W_r=0.194$ mm	0.318 mm
square	$H_s=0.318$ mm	$W_s=0.318$ mm	0.318 mm

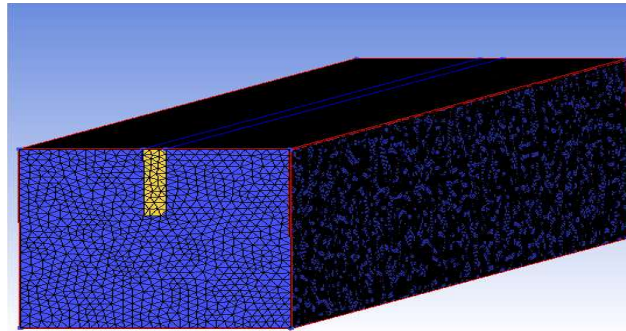


Figure 2: Meshed rectangular microchannel heat sink model

Fig 2. Represents rectangular microchannel heat sink after meshing, done using ICEM-CFD module. As shown in Fig 2. The central yellow zone is the microchannel with rectangular cross section. The liquid water flows through this zone from inlet to outlet and absorbs heat as constant heat flux was given to the base of the heat sink. Base is the bottom side in Fig 2.

For ease of CFD analysis, 1 microchannel was considered instead of 10 using symmetric boundary condition, as shown in Fig 2.

Boundary conditions and computational domain

For analysis purpose some of the assumptions are made for the present study:

1. The single phase laminar flow of working fluid is considered.
2. Steady state three dimensional fluid flow is considered.
3. All the properties of working fluid and material use as solid are considered to be constant.

The meshing of the model is done in ICEM-CFD module. For ease of analysis only one microchannel is considered out of ten and analysis is done. The mesh type is selected to be tetra/mixed and then using GSF meshing is done. GSF is global scale factor. It is defined as ratio of maximum length to minimum length of element in geometry. The grid independency was conducted by taking GSF value of 5 and 6 and it was found that for rectangular microchannel with $Re=600$, the pressure drop variation between GSF 5 and GSF 6 was found to be 6.01%, hence GSF 6 was chosen for study. The fluid and solid zones of the model were meshed with same mesh density. Output solver was considered to be Fluent_V6 and common structural solver ANSYS was selected.

The output of the ICEM-CFD module was then taken to the ANSYS Fluent software. This software solves the heat transfer and fluid flow related problems by using various standard equations of navier stokes equation. While solving, the energy equation was kept on and viscous laminar flow was selected. The material was selected to be copper as solid and liquid water as fluid. The various boundary condition are given as follows:

4. According to Reynolds number the inlet velocity was calculated for each flow. For Re number 600, 700, 800, 900 and 1000 the inlet velocities are 1.8924 m/s, 2.2078 m/s, 2.5232 m/s, 2.8386 m/s and 3.1540 m/s respectively.
5. The outlet was considered to be pressure outlet with gauge pressure value considered to be 0 Pa.
6. At bottom of the channel constant heat flux of 450000 W/m^2 was provided.
7. The top surface of the microchannel was assumed to be insulated.

The ANSYS fluent output was taken to CFD post module where post processing was done.

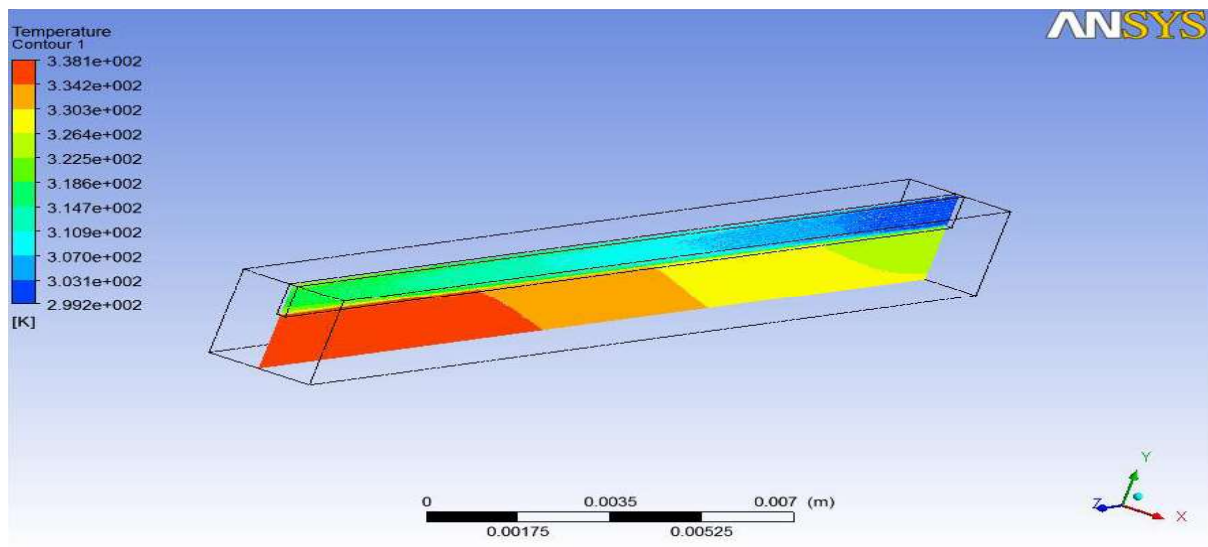


Figure 3: Temperature variation for rectangular microchannel heat sink for Reynolds number of 600

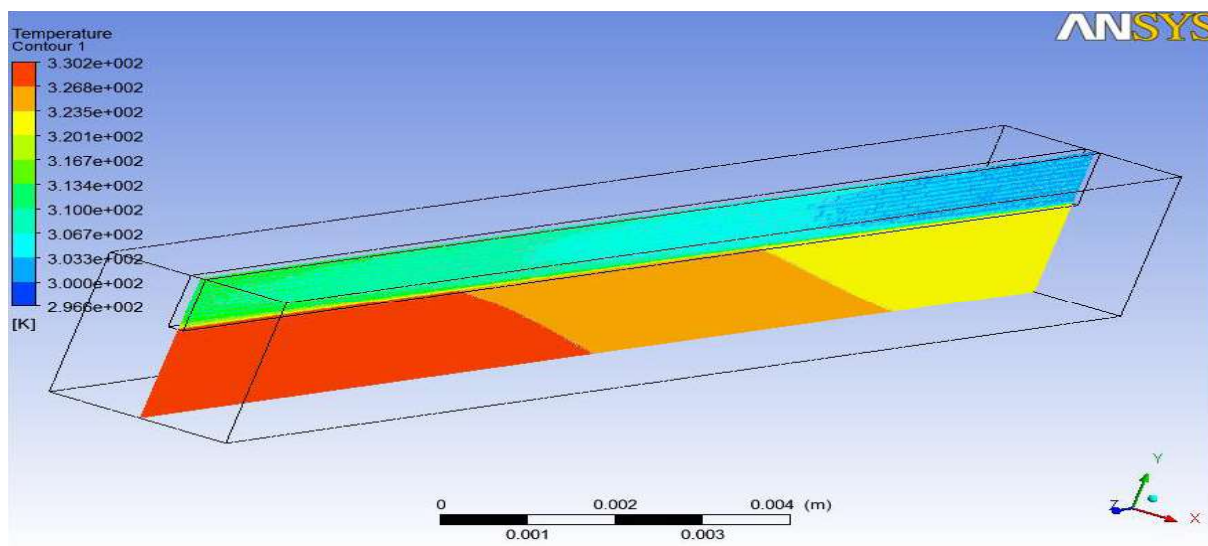


Figure 4: Temperature variation for rectangular microchannel heat sink for Reynolds number of 1000

The temperature variation for rectangular microchannel heat sink is shown in Fig 3. And Fig 4. For Reynolds number of 600 and 1000 respectively. For $Re=600$, the temperature range is 299.2 K to 338.1 K as shown in Fig 3. And that for $Re=1000$ is 296.6 K to 330.2 K as shown in Fig 4.

As per the colour contours for temperature shown in Fig 3. And Fig 4. It is cleared that temperature of the fluid increases from inlet to outlet as fluid absorbs heat as it passes through channel from inlet to outlet.

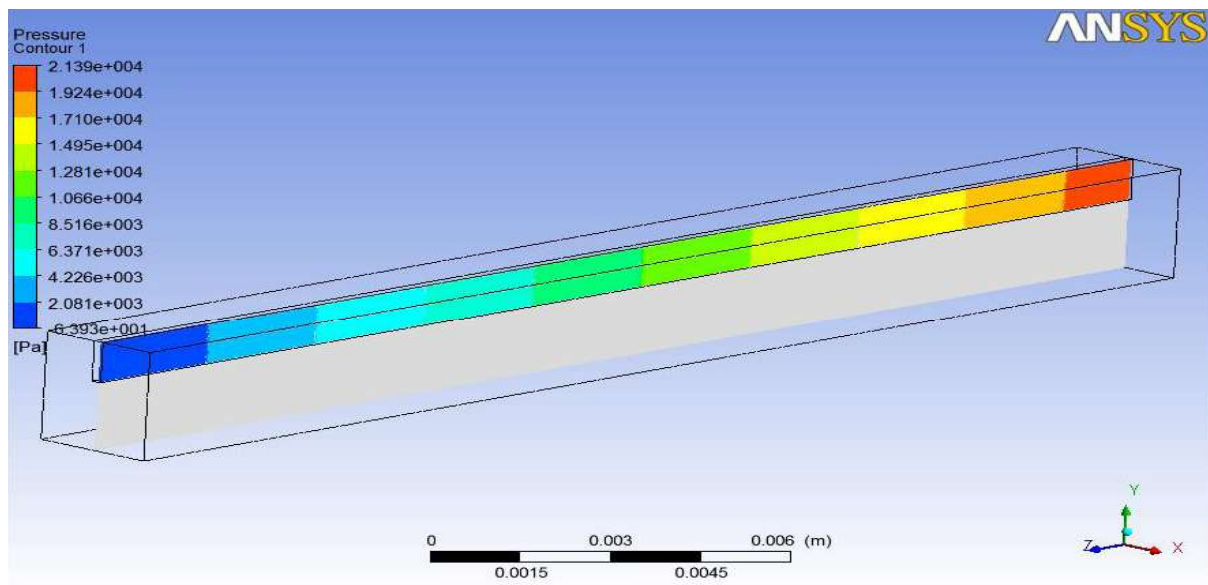


Figure 5: Pressure variation for rectangular microchannel heat sink for Reynolds number of 600

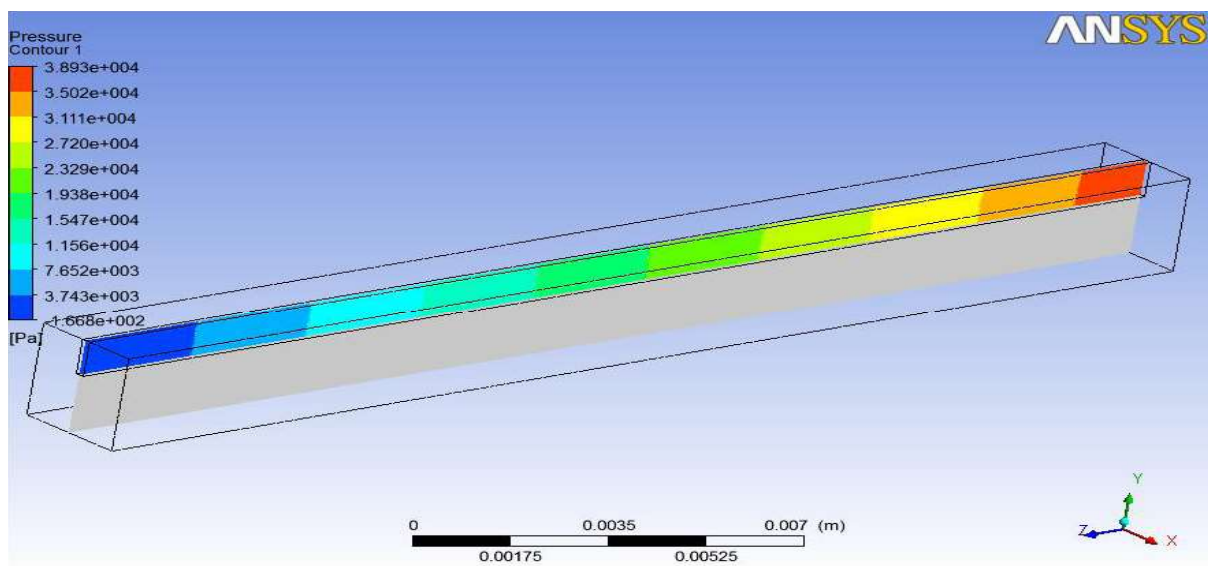


Figure 6: Pressure variation for rectangular microchannel heat sink for Reynolds number of 1000

Pressure variation for rectangular microchannel heat sink for $Re=600$ and $Re=1000$ is shown in Fig 5. And Fig 6. Respectively. From the pressure colour contour, it is observed that pressure drop of fluid takes place as fluid flows from inlet to outlet. This is because along flow direction loss of energy because of head losses takes place and that is why the pressure drop of fluid is sign of loss of energy.

CFD CALCULATION

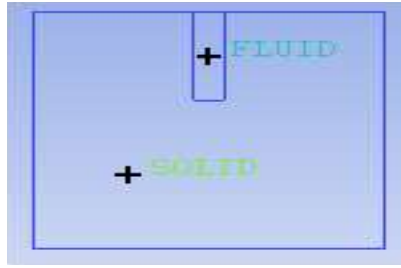


Figure 7: Front view of rectangular microchannel heat sink

In this study, convective heat transfer takes place and heat was passed to liquid water flowing through the microchannel which was being used as cooling medium. As shown in Fig 7. 2 zones were created in model, one was fluid zone and another solid zone. For analysis in CFD post, 4 lines were drawn out of which 3 are in fluid zone and one is in the solid zone. 3 lines were drawn in fluid zone from inlet to outlet exactly at a distance of half of the microchannel width from one microchannel wall, first at top, second at centre and third at base of fluid zone. One line in solid zone was drawn at bottom of solid zone exactly below the 3 lines in fluid zone. These lines were used for analysis in CFD post.

$$(\Delta T)_f = T_{f,o} - T_{f,i} \quad (1)$$

Then the average fluid temperature was calculated as,

$$T_{f,avg} = (T_{f,i} + T_{f,o})/2 \quad (2)$$

The temperature difference between fluid average and wall average temperature was calculated as,

$$(\Delta T) = T_{w,avg} - T_{f,avg} \quad (3)$$

Mass flow rate was calculated as,

$$\dot{m} = \rho A_{cs} V \quad (4)$$

Then heat transfer to fluid from inlet to outlet was calculated as,

$$Q = \dot{m} C_p (\Delta T)_f \quad (5)$$

Heat gained by fluid through convection is same as that of heat transfer through solid material through conduction which is,

$$Q = h A_s (\Delta T) \quad (6)$$

Then Nu number was calculated as,

$$Nu = (h D_h) / k \quad (7)$$

The Reynolds number was calculated as,

$$Re = (\rho V D_h) / \mu \quad (8)$$

Whereas the hydraulic diameter is calculated as,

$$D_h = (4 A_{cs}) / P \quad (9)$$

RESULT AND DISCUSSION

In present study, many factors like coefficient of heat transfer, pressure drop, Nusselt number are studied and comparison between rectangular and square microchannel heat sink is presented in the form of graph. Rectangular microchannel heat sink is showing good results as compared to square microchannel heat sink.

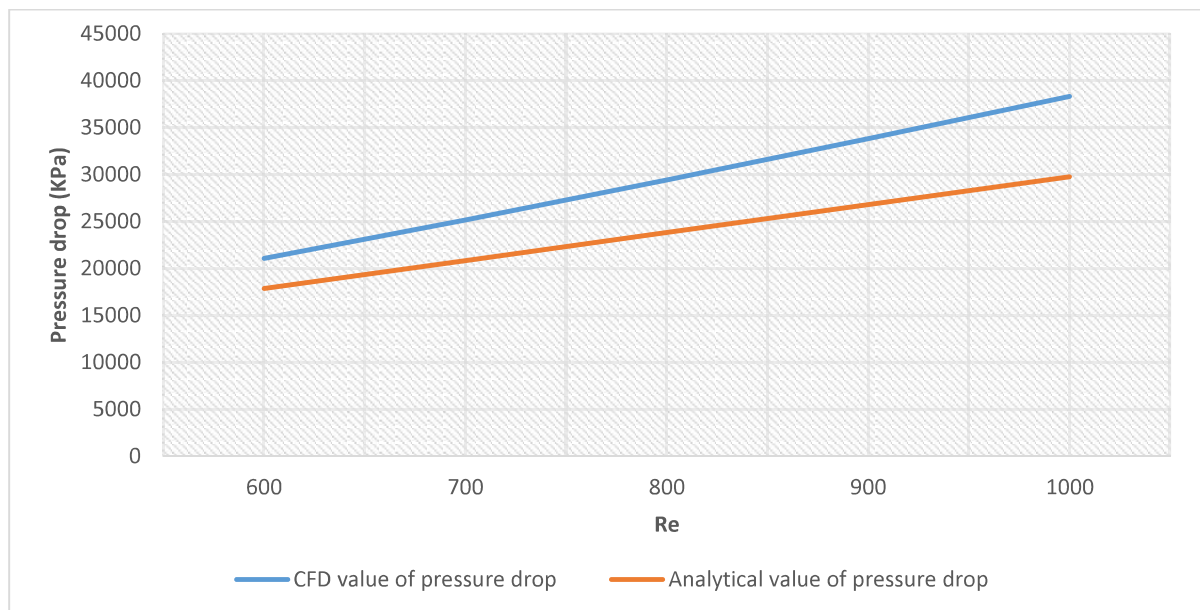


Figure 8: CFD and Analytical pressure drop value comparison of rectangular microchannel

As shown in Fig 8. The CFD value are on little higher side than that of analytical values of pressure drop for rectangular microchannel for each and every Reynolds number. The analytical value are calculated using formula in [7]. Pressure drop takes place along the flow direction as energy decreases and it is converted to head loss mainly due to friction. After comparison between values obtained, the percentage variation between CFD and analytical value for Reynolds number of 600, 700, 800, 900 and 1000 were observed to be 15.21%, 17.17%, 19.06%, 20.78% and 22.34% respectively. The average percentage variation after considering five percentage variation value was observed to be 18.9%.

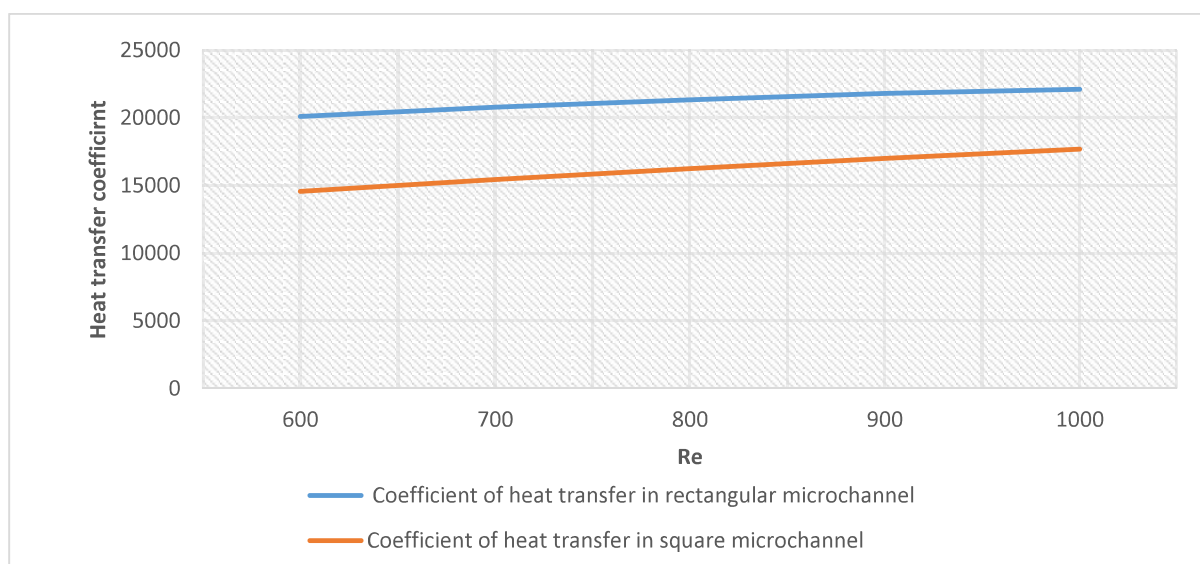


Figure 9: Rectangular and Square microchannel heat transfer coefficient value comparison

As shown in Fig 9. The rectangular microchannel is showing high coefficient of heat transfer value as compared to square microchannel for each and every Reynolds number. Coefficient of heat transfer is the main factor as it shows how much effectively heat is transferred between solid and adjacent flowing fluid. Hence we can say that

rectangular microchannel is better as compared to square microchannel. The results obtained in [5] shows different trend and states that higher heat removal flux is shown by arc groove as compared to rectangular groove. Coefficient of heat transfer shows effectiveness of the system as it represents amount of heat removed effectively by cooling fluid from desired section.

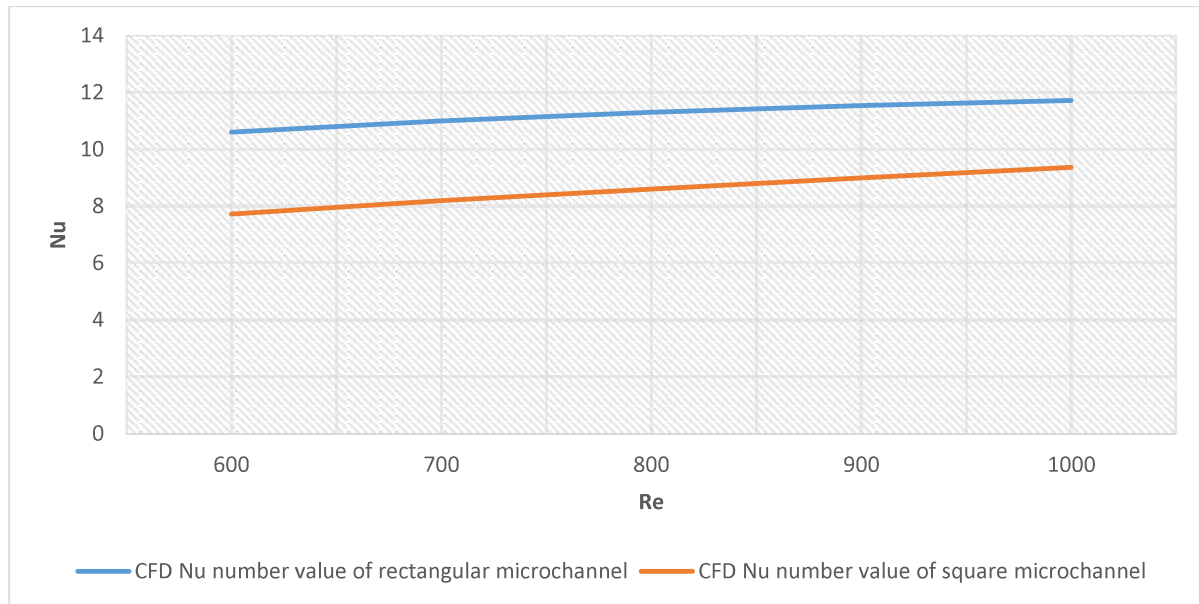


Figure 10: Rectangular and Square microchannel Nu number value comparison

As shown in Fig 10. Rectangular microchannel is showing higher Nusselt number values as compared to square microchannel for each and every Reynolds number. Nusselt number is directly proportional to coefficient of heat transfer hence the values are showing same trend as shown in Fig 9. Nusselt number is a dimensionless number used in heat transfer analysis.

Conclusion

As per the results obtained in each case, rectangular microchannel values are higher than that of square microchannel.

1. Heat transfer coefficient values were higher in each case for rectangular microchannel as compared to square microchannel so efficient heat transfer will take place in rectangular microchannel to the cooling fluid.
2. the rectangular microchannel heat sink is better to use for better cooling performance as Nusselt number values were higher in each case for rectangular microchannel as compared to square microchannel.
3. CFD pressure drop value of rectangular microchannel were higher in each case as compared to analytical values calculated using formula in [7].

So as per the results for better cooling performance it is better to use rectangular microchannel as compared to square microchannel. In the present study, the effects of piping and fitting and minor head losses present in experimental analysis are not considered. So these factors can have effect on the values obtained in the present study.

References:

- 1) Rajesh Sharma, Amoljit Singh Gill and Vikas Dhawan, "EXPERIMENTAL ANALYSIS OF HEAT TRANSFER AND FLUID FLOW IN MICRO-CHANNEL HEAT SINK", International Journal of Recent advances in Mechanical Engineering (IJMECH) Vol.5, No.3, August 2016.
- 2) Oana Giurgiu, Angela Pleșa, Lavinia Socaciu, "Plate heat exchangers - flow analysis through mini channels", Energy Procedia 85 (2016) 244 – 251
- 3) Manoj Rao, Sameer Khandekar, "Thermo-hydrodynamics of developing flows in a mini-channels array", 19th National and 8th ISHMT-ASME heat and mass transfer conference, Jan3-5, 2008, JNTU, Hyderabad, India.
- 4) Dr. B. S. Gawali , V. B. Swami , S. D. Thakre, "Theoretical and Experimental Investigation of Heat Transfer Characteristics through a Rectangular Microchannel Heat Sink", International Journal of Innovative Research in Science, Engineering and Technology, Vol. 3, Issue 8, August 2014
- 5) O. Abouali, N. Baghernezhad, "Numerical Investigation of Heat Transfer Enhancement in a Microchannel With Grooved Surfaces", Journal of Heat Transfer ,APRIL 2010, Vol. 132 / 041005-1
- 6) Yang Liu, Jing Cui, WeiZhong Li, Ning Zhang, "Effect of Surface Microstructure on Microchannel Heat Transfer Performance", Journal of Heat Transfer ,DECEMBER 2011, Vol. 133 / 124501-1
- 7) Satish G. Kandlikar, "Single-Phase Liquid Flow in Minichannels and Microchannels", Heat Transfer and Fluid Flow in Minichannels and Microchannels, 2014 Elsevier Ltd.
- 8) Poh-Seng Lee, Suresh V. Garimella , Dong Liu, "Investigation of heat transfer in rectangular microchannels", International Journal of Heat and Mass Transfer 48 (2005) 1688–1704
- 9) Poh-Seng Lee, S V. Garimella, "Thermally Developing Flow and Heat Transfer in Rectangular Microchannels of Different Aspect Ratios", International Journal of Heat and Mass Transfer, August 2005
- 10) Paisarn Naphon , Osod Khonseur, "Study on the convective heat transfer and pressure drop in the micro-channel heat sink", International Communications in Heat and Mass Transfer 36 (2009) 39–44
- 11) P. Promvongse , S. Sripattanapipat , S. Tamna , S. Kwankaomeng , C. Thianpong, "Numerical investigation of laminar heat transfer in a square channel with 45° inclined baffles", International Communications in Heat and Mass Transfer 37 (2010) 170–177
- 12) G. Gamrat , M. Favre-Marinet, S. Le Person, "Modelling of roughness effects on heat transfer in thermally fully-developed laminar flows through microchannels", International Journal of Thermal Sciences 48 (2009) 2203–2214
- 13) Xiaojin Wei and Yogendra Joshi, "Optimization Study of Stacked Micro-Channel Heat Sinks for Micro-Electronic Cooling", IEEE TRANSACTIONS ON COMPONENTS AND PACKAGING TECHNOLOGIES, VOL. 26, NO. 1, MARCH 2003
- 14) Tomasz A. Kowalewski, Jacek Szumbariski, Slawomir Blonski , "LOW-REYNOLDS-NUMBER INSTABILITY OF THE LAMINAR FLOW BETWEEN WAVY WALLS", Proceedings of the Sixth International ASME Conference on Nanochannels, Microchannels and Minichannels ICNMM2008 June 23-25, 2008, Darmstadt, Germany
- 15) Jing ZHOU, Zhiheng YAN, Qiang GAO, " Development and Application of Micro Channel Heat Exchanger for Heat Pump", 12th IEA Heat Pump Conference (2017) P.3.1.5