IOP Conference Series: Materials Science and Engineering

PAPER • **OPEN ACCESS**

Rectangular and Square Cross-Sections Microchannel Heat Sink CFD Simulation and Analytical Validation Using Liquid Water & Water-Aluminium Oxide Nanofluid as a Cooling Medium

To cite this article: Saurabh Narayan Pawar *et al* 2021 *IOP Conf. Ser.: Mater. Sci. Eng.* **1145** 012093

IOP Conf. Ser.: Mater. Sci. Eng. **1145** (2021) 012212 https://doi.org/10.1088/1757-899X/1145/1/012212

Retraction

Retraction: Rectangular and Square Cross-Sections Microchannel Heat Sink CFD Simulation and Analytical Validation Using Liquid Water & Water-Aluminium Oxide Nanofluid as a Cooling Medium (*IOP Conf. Ser.: Mater. Sci. Eng.* **1145** 012093)

Published 23 February 2022

This article (and all articles in the proceedings volume relating to the same conference) has been retracted by IOP Publishing following an extensive investigation in line with the COPE guidelines. This investigation has uncovered evidence of systematic manipulation of the publication process and considerable citation manipulation.

IOP Publishing respectfully requests that readers consider all work within this volume potentially unreliable, as the volume has not been through a credible peer review process.

IOP Publishing regrets that our usual quality checks did not identify these issues before publication, and have since put additional measures in place to try to prevent these issues from reoccurring. IOP Publishing wishes to credit anonymous whistleblowers and the Problematic Paper Screener [1] for bringing some of the above issues to our attention, prompting us to investigate further.

[1] Cabanac G, Labbé C and Magazinov A 2021 arXiv:2107.06751v1

Retraction published: 23 February 2022

Rectangular and Square Cross-Sections Microchannel Heat Sink CFD Simulation and Analytical Validation Using Liquid Water & Water-Aluminium Oxide Nanofluid as a Cooling Medium

Saurabh Narayan Pawar[1] Jugal Shrinivas Makam[1] Nilesh Balkishanji Totla[2]
[1]Student, MIT Academy of Engineering, Pune, India.
[2]Sr Assistant Professor, MIT Academy of Engineering, Pune, India.
[1]snpawar@mitaoe.ac.in [2]jsmakam@mitaoe.ac.in, [3]nbtotla@mech.maepune.ac.in

Abstract. In this study, cooling performance of copper material based microchannel heat sink was investigated using two different approaches which are CFD and Analytical. Microchannel heat sink with two different cross-sectional geometries of rectangular and square was considered for the present study. In the present work CFD simulation is carried out using two different cooling fluids which are liquid water and water-Al_2O_3 nanofluid. Nanofluid volume fraction of 0.3% was used for present study. Re number in between 200-1000 was used for the present study. For CFD simulation purpose heat sink of dimension of 25.4mm×25.4mm×2.384mm is considered in the study. Boundary condition of constant heat flux is assumed by providing heat flux at constant rate at bottom of the assembly. To compare between square and rectangular cross section microchannel heat sink, the hydraulic diameter is kept same in both the cases and CFD simulation was conducted. With using water- Al_2O_3 nanofluid as the working fluid the rectangular cross section is showing better performance in terms of cooling as compare to the square cross section. Drop in pressure results in rectangular section calculated using water Al_2O_3 nanofluid using both CFD and Analytical approach are in good agreement with difference of 13.4%

Keywords: Computational fluid dynamics, Heat transfer, Microchannel heat sink, ICEM-Fluent CFD.

1. Introduction

In this modern era, use of smaller and compact devices has been increased tremendously. Modern computers that uses smaller electronic devices is the best example of it. Along with using this device, the problem of dissipation of heat has been increased also as if it is not removed then it might damage the component. Many researchers have presented their work regarding effective heat transfer and faster cooling of these devices so as to protect these devices using microchannels and minichannels. Microchannel and minichannel heat sink study has been carried out in various literature using different working fluids and different geometries for investigating the cooling performance and enhancing the heat transfer rate. [1] presented thermodynamic investigation results in circular tube for fully developed turbulent forced convection with water-Al_2O_3 nanofluid using entropy generation minimization method. Volume fraction of nanofluid in range of 0% to 6% was used and Reynolds number ranging from 5000 to 180000 were used. Using velocity and temperature fields obtained during CFD analysis the entropy generation rates were numerically determined. It has been proved that at each Reynolds number there is a cross sectional area which is optimum for which there is minimum entropy generation in tube and as with increase in Reynolds number the optimal area of cross section increases. [2] investigated nanofluid application on the system of parabolic trough collector. Using Al_2O_3/synthetic oil nanofluid and using Finite Element Method the multifield coupling simulation of PTC system was implemented for performance investigation. The Al_2O_3 particle concentration effects on PTC system were also studied. Good agreement was observed between experimental data and numerical results. Using Al_2O_3/synthetic nanofluid there is a great reduction in absorber maximum temperature and temperature gradient. [3] explored effects of various aspects which include heat source/sink location, number of tubes, average Nusselt number, Rayleigh number, volume concentration. By employing Finite Volume based control volume technique, governing equations are numerically solved. In the studied cavity, considerable enhancement in heat transfer was observed using nanofluids. [4] presented numerical study results of parabolic trough solar collector of high concentration ratio on the thermodynamic and thermal performance. In this study, 80° rim angle and 113 concentration ratio parabolic trough system was used. The thermal physical properties temperature dependency have been considered for both copper nanoparticles and base fluid. The combination of CFD and Monte-Carlo ray tracing procedures was used in numerical analysis. [5] numerically investigated different microstructures microchannels with forced convection heat transfer. In microstructural grooves vortices will appear. Using Nusselt number the microchannel geometries influence on performance of heat transfer was evaluated. The highest heat transfer performance was possessed by microstructure with V shaped groove.

[6] investigated numerically forced convection heat transfer of laminar flow in corrugated channel of trapezoidal section using copper water nanofluid. The Reynolds number ranging between 100 to 700 and volume fraction of nanoparticles in range of 0% to 5% was considered. The geometrical parameter effects like corrugated channel wavelength and amplitude have been presented. It was observed that with increasing volume fraction of nanoparticles the average Nusselt number increases with increase in pressure drop. [7] studied microchannel and minichannel using single phase liquid flow. Microscale liquid flow fundamental issue was addressed by author and relation for single phase liquid flow pressure drop calculations were given. Researchers in [8] conducted investigation experimentally and explored conventional sized channel based classical correlation validity for predicting single phase flow thermal behaviour in rectangular microchannel. Width of microchannels considered ranged between 194μm to 534μm. Deionized water was used for analysis with Reynolds number ranging from 300 to 3500. Classical and continuum approach based numerical predictions and experimental data were found to be in good agreement with showing 5% of average deviation. [9] used three nanofluid types namely carbon nanotubes-Ga, copper-Ga, diamond-Ga and carried out natural convection investigation of enhancement of heat transfer and generation of the entropy in cavity which is differentially heated. In this volume fraction of nanofluids is in between 0.01 to 0.15. Model of two-phase mixture is used for simulation of the nanofluid flow. Results showed that with Grashof number increment local entropy generation, heat transfer and convective intensity increases. [10] studied numerically heat transfer and analysis of fluid flow in channel having blocks at bottom wall. Nanofluid is used in this study. Channel inlet temperature of fluids are taken less than walls. Using control volume approach the governing equations are solved numerically. Results concluded that by using blocks on hot walls and using nanoparticles there is enhancement in channel heat transfer. By simulation, upto 60% enhancement in channel heat transfer is shown due to nanoparticles and blocks.

[11] investigated experimentally Foam/NEPCM composite heat transfer with phase change with foam porosity effect consideration. 47° Wall temperature reduction is provided which is maximum by Foam/NEPCM composite as compared to pure NEPCM. [12] employed CFD approach and studied serpentine tube heat exchanger for thermo hydraulic performance extensive exploration. The volume flow rates in the range of 1L/min to 5L/min was used. The Al_2O_3/water based nanofluid influence was broadly studied on thermo hydraulic performance using 1%, 3% and 5% nanoparticles concentrations. Higher performance of heat transfer was provided as compared to other cases using low to high serpentine tube. At the expense of pressure drop which is negligible the higher coefficient of heat transfer

was provided using nanoparticles which are highly concentrated. [13] showed that by swirl flows, vortices and thermal boundary layer subsequent breaking the heat exchanger thermal efficiency is augmented by vortex generator. For this numerical study which is 3D validated vortex generator of triangular winglet type is selected. Working fluids which are air, water and nanofluids of two different types are analyzed with vortex generator which is non-central. [14] showed that less important role is played by O_2 and SO_2 in water and MEA losses than evaporation. For capturing CO_2 ionic liquids at room temperature are proposed potential candidates recently. IL to MEA aqueous solution addition reduced MEA and water losses. [15] stated that in many industrial applications the essential tool is surface cooling. In enhancement of heat transfer the important factor is area of the wetted surface utilization effectively. The vortex generator surface texture effects on vortex dynamics and heat transfer are studied, with pressure drop which is minimal the surface temperature found to reduce by vortex stretching. [16] have presented their work in microchannel heat sink using microchannel with rectangular and square cross section keeping the hydraulic diameter same in both of the cross sections. Liquid water was used as a cooling medium. As comparing with microchannel with square cross section, the better cooling performance was shown by rectangular cross section. In the present study, microchannel heat sink with rectangular and square cross section are considered and studied using two different types of fluid which are liquid water and water-Al_2O_3 nanofluid of 0.3% volume fraction. Using CFD approach both the cross sections are compared using both of the cooling fluids.

2. Physical Model & CFD Simulation

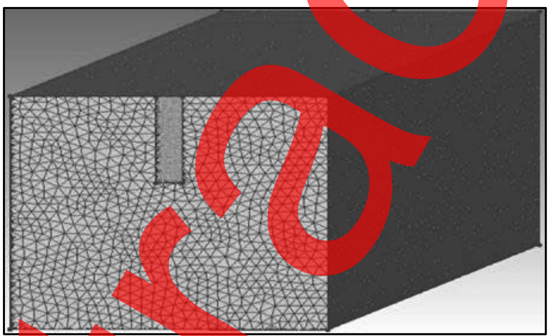

Figure 1. Rectangular microchannel heat sink model with meshing

CFD is a tool which is used to solve many engineering problems which involves fluid flow and heat transfer. CFD stands for Computational Fluid Dynamics. To get the heat sink machining was done on square block of copper with dimensions of 25.4 length, 25.4mm width and 70mm height. Now the dimension of rectangular section are considered to be 25.4mm length (L), 25.4mm width (W) and 2.384mm height (H) for CFD analysis in the present study. The rectangular section dimensions are same as that in [8]. There are N=10 straight parallel rectangular cross section microchannels in microchannel heat sink block same as that used in [8], but in the present study for CFD work, one rectangular cross section was considered and analysis was conducted as shown in Figure 1. The results obtained by CFD analysis of the rectangular and square heat sink was studied by using same hydraulic diameter in both the cases. The cooling fluids used in both the cases were liquid water and water-Al_2O_3 nanofluid of 0.3% volume fraction. The results of both the cross sections were compared using two cooling fluids mentioned. Re number was considered from 200 to 1000 for both of the cross sections.

Table 1. Dimensions of Rectangular and Square cross sections

Microchannel Dimensions	D_h	Height	Width
Rectangular	0.318 mm	H_r=0.884 mm	W_r=0.194 mm
Square	0.318 mm	H_s=0.318 mm	W_s=0.318 mm

The dimensions and notations used in table 1. Are same as those which were used in [16]. As shown in Figure 1. The fluid flows from inlet to outlet through the rectangular cross section microchannel. Constant heat flux was provided at bottom of the heat sink so that heated assembly temperature was brought down by cooling fluid by increment in temperature of cooling fluid from inlet to outlet. Using ICEM-CFD module the meshing of the assembly was done. Using GSF of 6 and tetra/mixed mesh type meshing was done. GSF stands for global scale factor. Fluent_V6 output solver and ANSYS solver was used. Keeping mesh density same the solid and fluid zones were meshed. The ICEM output was used and then using fluent software further analysis was done. It used navier stokes equation to solve the problem related to heat transfer and fluid flow. While analysis viscous laminar flow was selected and energy equation was kept on. Copper was selected as solid material.

3. Boundary conditions and assumptions:

1. Single phase laminar flow was used.
2. Working fluid and solid properties were treated as constant.
3. Three dimensional steady state fluid flow was considered.
4. Velocity values of 0.8149m/s, 1.6298m/s, 2.444m/s, 3.2596m/s, 4.0745m/s were obtained for Reynolds number of 200, 400, 600, 800, 1000 respectively using water-Al_2O_3 nanofluid of 0.3% volume fraction.
5. Outlet pressure was considered to be 0 Pa guage with pressure outlet condition.
6. 450000 W/m^2 heat flux was provided at bottom.
7. Insulation was considered at assembly top surface.

The further analysis was done using CFD post processing section.

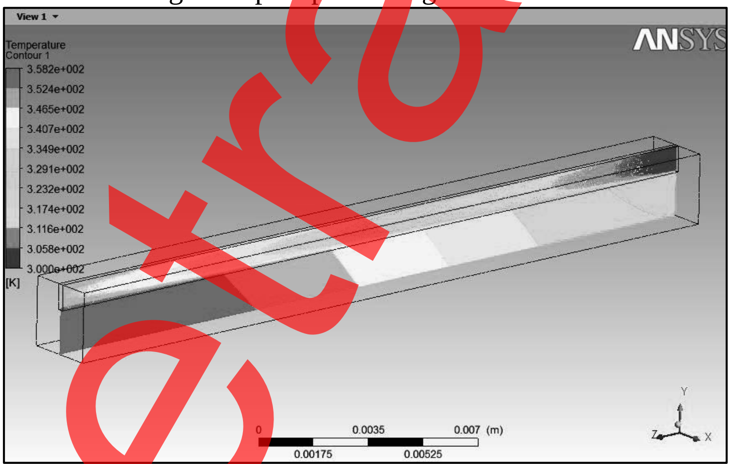

Figure 2. Temperature variation in rectangular cross section microchannel at 200 Reynolds number using water Al_2O_3 nanofluid of 0.3% volume fraction

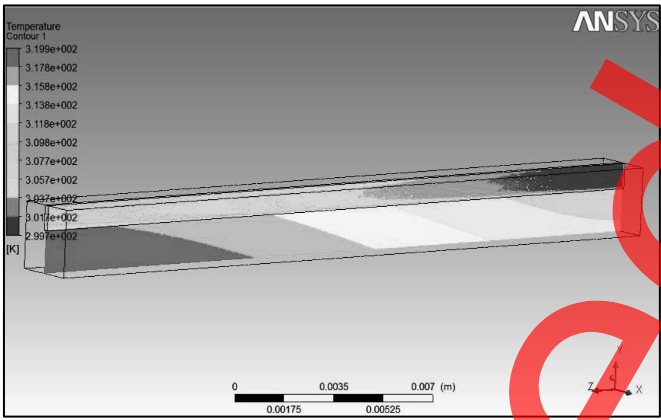

Figure 3. Temperature variation in rectangular cross section microchannel at 1000 Reynolds number using water-Al_2O_3 nanofluid of 0.3% volume fraction

Figure 2 and Figure 3. Represent temperature variation in rectangular cross section microchannel. As heat flux at constant rate was supplied at bottom of the assembly the assembly gets heated which is cooled by the working fluid flowing from inlet to outlet through the rectangular microchannel.

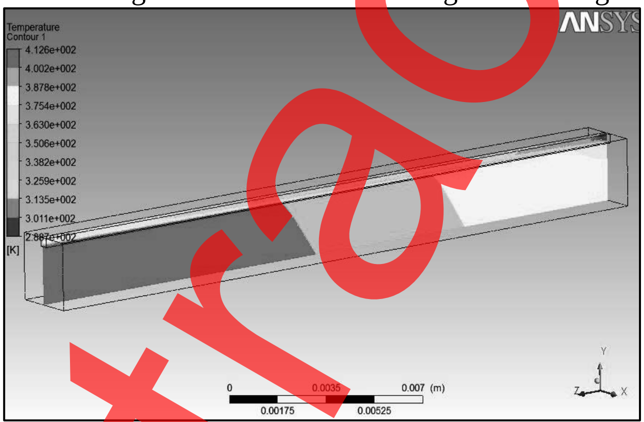

Figure 4. Temperature variation in square cross section microchannel at 200 Reynolds number using water Al_2O_3 nanofluid of 0.3% volume fraction

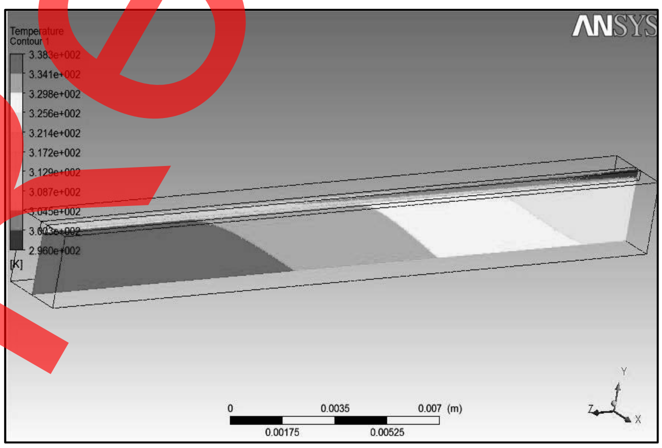

Figure 5. Temperature variation in square cross section microchannel at 200 Reynolds number using water-Al_2O_3 nanofluid of 0.3% volume fraction

Figure 4 and Figure 5. Represent the temperature variation in the square cross section microchannel. Fluid temperature increases as flows from inlet to outlet by absorbing heat from the heated assembly and it cools down the heated assembly.

4. Data Reduction from CFD Results

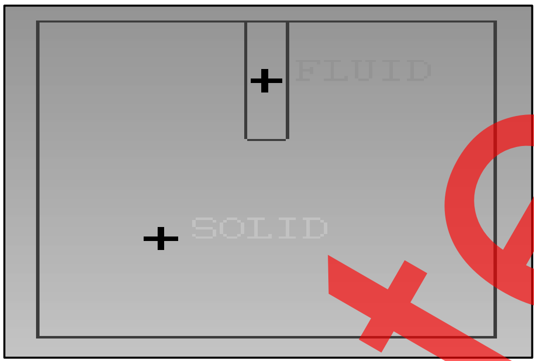

Figure 6. Rectangular microchannel heat sink front view

The Figure 6 was taken from [16]. Heat transfer takes place from the heated assembly to the cooling fluid by forced convection as fluid is flowing with certain velocity. For conduction CFD analysis, 2 zones were created namely solid and fluid zone. Lines were created in the assembly, 3 in fluid and 1 in solid part. From inlet to outlet the 3 lines were created at fluid section, one at top, one at middle and one at bottom.

Temperature difference between fluid inlet and outlet temperatures,

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

The average temperature of fluid was given as,

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

Average fluid temperature and average wall temperature difference was given as,

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

Mass flow of the fluid passing through the microchannel was calculated as,

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

The hydraulic diameter is given as,

$$D_h = (4A_{cs})/P \quad (5)$$

The formula for Reynolds number is,

$$Re = (\rho V D_h)/\mu \quad (6)$$

Heat absorbed by the cooling fluid as it flows through the microchannel is given as,

$$Q = \dot{m} C_p (\Delta T)_f \quad (7)$$

And the heat transfer coefficient associated with the above forced convection heat transfer process is given as,

$$Q = h A_s (\Delta T) \quad (8)$$

All Equations from (1)-(8) used are same as those which were used in [16].

5. Results & Discussions

Microchannel with different cross sections was used and between them comparison was conducted using liquid water and water-Al_2O_3 nanofluid of 0.3% volume fraction as a cooling medium. With both of the fluids the rectangular cross section was better with respect to the heat transfer as that of square cross section.

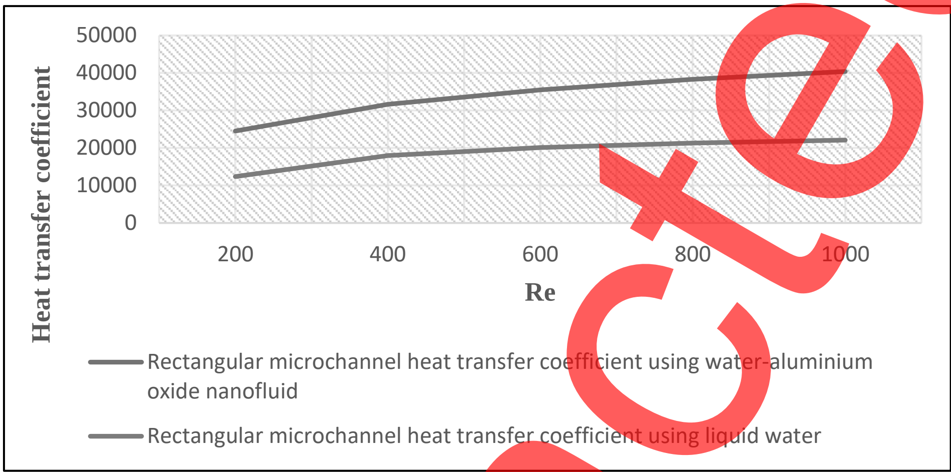

Figure 7. Comparison of heat transfer coefficient in rectangular microchannel using liquid water and water-aluminium oxide nanofluid

As shown in Figure 7. The values of heat transfer coefficient expressed in W/m^2K obtained for rectangular microchannel using nanofluid are higher compared to the values obtained with liquid water. The heat transfer coefficient values for rectangular microchannel using liquid water at Reynolds number of 600, 800 and 1000 are taken from [16]. The percentage increase in heat transfer coefficient using nanofluid in rectangular microchannel was observed to be 49.5%, 43.09%, 43.3%, 44% and 45% for Reynolds number of 200, 400, 600, 800 and 1000 respectively and average percentage increment of 45%.

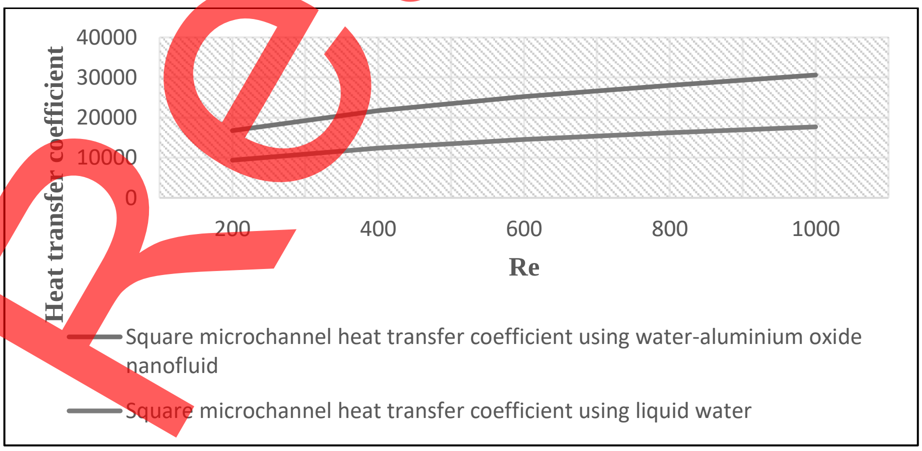

Figure 8. Comparison of heat transfer coefficient in square microchannel using liquid water and water-aluminium oxide nanofluid

As shown in Figure 8. In each case the nanofluid is giving better heat transfer coefficient values expressed in W/m^2K as compared to the liquid water in square cross section microchannel. The heat transfer coefficient values at Reynolds number of 600, 800 and 1000 for square cross section using liquid water are taken from [16]. The percentage increase in heat transfer coefficient values using nanofluids as compare to using water for Reynolds number of 200,400, 600, 800 and 1000 in square microchannel was observed to be 44%, 42.9%, 42%, 42% and 42% respectively with average percentage increment of 42.5%.

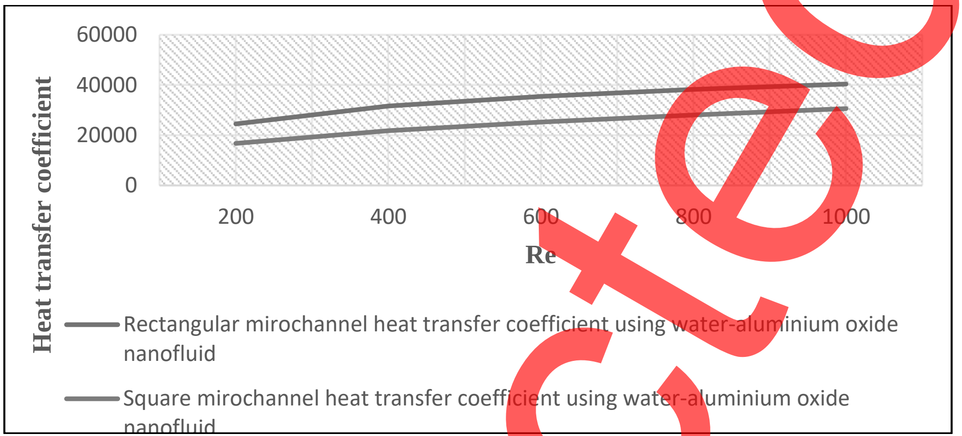

Figure 9. Comparison of heat transfer coefficient in rectangular and square microchannel using water-aluminium oxide nanofluid

From Figure 9. We can conclude that rectangular microchannel is giving better performance as that of square section while using nanofluid as cooling medium. The percentage increase in heat transfer coefficient values expressed in W/m^2K using rectangular microchannel as compared to square using nanofluid as a cooling medium was observed to be 31.6%, 31%, 29%, 26.7% and 24% for the Reynolds number of 200, 400, 600, 800 and 1000 with average percentage increment of 28.4%.

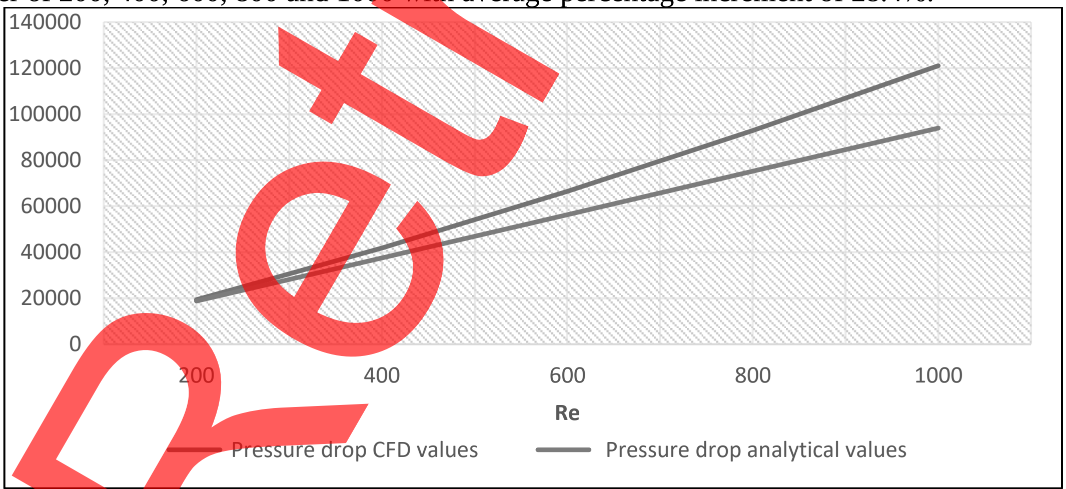

Figure 10. Pressure drop values in rectangular cross section using water-aluminium oxide nanofluid

The values of pressure drop in Pa are expressed in Figure 10. Using CFD and Analytical approach. With the flow pressure drop increases because of the frictional head loss. The analytical values of pressure drop are calculated using formula given in [7]. The percentage variation between the pressure drop values obtained using CFD Analytical approach for Reynolds number of 200, 400, 600, 800 and 1000 was observed to be 3.05%, 10%, 15.2%, 19.12% and 19.9% with average variation of 13.4%.

6. Conclusion

Present study was conducted using liquid water and water-Al_2O_3 nanofluid with 0.3% volume fraction. As compared to liquid water the nanofluid is showing better heat transfer performance. The rectangular is better to use as compared to square cross section microchannel as heat will be effectively removed by rectangular cross section because heat transfer coefficient values obtained are higher for rectangular than square cross section.

1. Average percentage increment in heat transfer coefficient values in rectangular microchannel with water-Al_2O_3 nanofluid as cooling medium as compared to the liquid water was obtained to be 45%.
2. Average percentage increment in heat transfer coefficient values in square microchannel with water-Al_2O_3 nanofluid as cooling medium as compared to the liquid water was obtained to be 42.5%.
3. Average percentage increment in heat transfer coefficient values in rectangular microchannel as compared to the square microchannel with water-Al_2O_3 nanofluid as cooling medium was obtained to be 28.4%.
4. Average pressure drop variation in rectangular microchannel with water-Al_2O_3 nanofluid as cooling medium using analytical and CFD approach was observed to be 13.4%.
   So it is better to use rectangular as that of square microchannel and nanofluid as compared to liquid water.

References

[1] Aggrey Mwesigye, Zhongjie Huan, *Thermodynamic analysis and optimization of fully developed turbulent forced convection in a circular tube with water–Al_2O_3 nanofluid*, International Journal of Heat and Mass Transfer **89** (2015), pp 694-706.

[2] Yanjuan Wang, Jinliang Xu, Qibin Liu, Yuanyuan Chen, Huan Liu, *Performance analysis of a parabolic trough solar collector using Al_2O_3/synthetic oil nanofluid*, Applied Thermal Engineering **107** (2016), pp 469-478.

[3] Yulin Ma, Mojtaba Jamiati, Alireza Aghaei, Mohammad Sepehrirad, Amin Dezfulizadeh, Masoud Afrand, *Effect of differentially heated tubes on natural convection heat transfer in a space between two adiabatic horizontal concentric cylinders using nano-fluid*, International Journal of Mechanical Sciences **163** (2019), p 105148.

[4] Aggrey Mwesigye, Zhongjie Huan, Josua P. Meyer, *Thermal performance and entropy generation analysis of a high concentration ratio parabolic trough solar collector with $Cu - Therminol^{®}$ VP-1 nanofluid*, Energy Conversion and Management **120** (2016)), pp 449-465.

[5] Yang Liu, Jing Cui, WeiZhong Li, Ning Zhang, *Effect of Surface Microstructure on Microchannel Heat Transfer Performance*, Journal of Heat Transfer, DECEMBER 2011, **133** / , pp 124501-1.

[6] M.A. Ahmed, M.Z. Yusoff, N.H. Shuaib, *Effects of geometrical parameters on the flow and heat transfer characteristics in trapezoidal-corrugated channel using nanofluid*, International Communications in Heat and Mass Transfer **42** (2013) , pp 69-74.

[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) , pp 1688–1704.

[9] Xiaoming Zhou, Yanni Jiang, Xunfeng Li, Keyong Cheng, Xiulan Huai, Xidong Zhang, Hulin Huang, *Numerical investigation of heat transfer enhancement and entropy generation of natural convection in a cavity containing nano liquid-metal fluid*, International Communications in Heat and Mass Transfer **106** (2019) , pp 46-54.

[10] H. Heidary, M.J. Kermani, *Heat transfer enhancement in a channel with block(s) effect and utilizing Nano-fluid*, International Journal of Thermal Sciences **57** (2012) , pp 163-171.
[11] W.Q. Li, S.J. Guo, L. Tan, L.L. Liu, W. Ao,*Heat transfer enhancement of nano-encapsulated phase change material (NEPCM) using metal foam for thermal energy storage*, International Journal of Heat and Mass Transfer **166** (2021) , p 120737.
[12] M. Awais, M. Saad, Hamza Ayaz, M.M. Ehsan, Arafat A. Bhuiyan, *Computational assessment of Nano-particulate (Al_2O_3/Water) utilization for enhancement of heat transfer with varying straight section lengths in a serpentine tube heat exchanger*, Thermal Science and Engineering Progress **20** (2020) , p 100521.
[13] Man-Wen Tian, Saleh Khorasani, Hazim Moria, Samira Pourhedayat, Hamed Sadighi Dizaji,*Profit and efficiency boost of triangular vortex-generators by novel techniques*, International Journal of Heat and Mass Transfer **156** (2020) , p 119842.
[14] H. Anandakumar and K. Umamaheswari, A bio-inspired swarm intelligence technique for social aware cognitive radio handovers, Computers & Electrical Engineering, vol. **71**, pp. 925–937, Oct. 2018. doi:10.1016/j.compeleceng.2017.09.016
[15] R. Arulmurugan and H. Anandakumar, Early Detection of Lung Cancer Using Wavelet Feature Descriptor and Feed Forward Back Propagation Neural Networks Classifier, Lecture Notes in Computational Vision and Biomechanics, pp. 103–110, 2018. doi:10.1007/978-3-319-71767-8_9.
[16] Saurabh Narayan Pawar, Nilesh Balkishanji Totla,*CFD Simulation and Analytical Validation of Rectangular and Square microchannel heat sink with liquid water as cooling medium*, 3rd International Congress on Advances in Mechanical Sciences, IOP Conf. Series: Materials Science and Engineering 998 (2020) 012030, IOP Publishing,doi:10.1088/1757-899X/998/1/012030.