Journal of Thermal Engineering, Vol. 3, No. 5, pp. 1453-1465, October, 2017
Yildiz Technical University Press, Istanbul, Turkey

NUMERICAL STUDY OF PRESSURE DROP AND HEAT TRANSFER IN A STRAIGHT RECTANGULAR AND SEMI CYLINDRICAL PROJECTIONS MICROCHANNEL HEAT SINK

M. Zunaid[1,*], A. Jindal[1], D. Gakhar[1], A. Sinha[1]

ABSTRACT

The heat transfer and pressure drop characteristics of a straight rectangular and semi cylindrical projections microchannel heat sink were investigated. The heat sinks, made of copper, were aligned in row as a cluster of 21 microchannels, 231 µm wide and 713 µm deep. Water was used as cooling fluid and was made to flow through the channels. A three dimensional analysis was done for Reynolds number ranging between 200 to 1000 with constant heat flux of 10^6 W/m^2 defined relative to the platform area of the heat sink. The temperature rise and pressure drop of the fluid in straight microchannel heat sink was evaluated using ANSYS-CFX package and was validated using experimental data. Similar analysis was done for semi cylindrical projections microchannel heat sink by solving the conjugate heat transfer problem. A comparison of heat transfer and pressure drop between rectangular and semi cylindrical projections microchannel was done and the results were laid out. Results shows that heat transfer increases with the use of semi cylindrical projections microchannel heat sink.

Keywords: *Heat Sink, Microchannel, Heat Transfer, Reynolds Number, Conjugate Heat Transfer, Semi Cylindrical Projections*

INTRODUCTION

Microchannel heat sinks provide a cutting edge technology for removal of large amount of heat from a very small surface area. These are starting to challenge the use of traditional coil technology for heat exchange in many industries. Since the most recent decade, smaller scale machining technology is utilized at an expanded rate for the improvement of exceptionally productive cooling gadgets known as micro-channel heat sinks due to its greater advantages, for example, less coolant requirement and small size. Thus, the investigation of heat transfer and fluid flow in smaller scale channels which are two important parts of such gadgets, have pulled in additional considerations with expansive applications in both designing and performance issues. The construction of the heat sink is done from a material with high thermal conductivity, for example, copper or silicon. Smaller scale channels are created into its surface either by exact machining or miniaturized scale fabrication technology. These small scale channels have trademark measurements going from 10 to 1000 µm, and serve as flow entries for the cooling fluid. These micro-channels possess advantage of very small volume to surface area, small mass, large convective heat transfer coefficient. Miniaturization of electronic gadgets has become necessity of today. The reduced size of the microchips has led to an increase in the heat flux density, which causes overheating of gadgets and makes the general prosperity and working of these gadgets a major test for scientists. So there is a need to move onto profoundly proficient cooling innovation and heat dissipation strategies to meet the safety standards. The cross-sectional area of the heat sink serves as a course to transport liquid to and far from the wall of the channel. The study on single phase micro-channel heat sink has a very extensive coverage since the last two decades. Since the last two decades, micro-channel coolers having different dimensions and different substrate materials have been fabricated and tested with numerous coolants.

The micro-scale cooling and heat transfer was first studied by Tuckerman and Pease [1]. They fabricated a 1×1 cm^2 rectangular silicon wafer microchannel of depth 302 µm and width 50 µm with water as the cooling fluid. The heat sink was capable of dissipating 790 W/cm^2 with a pressure drop of 2.2 bar and maximum substrate temperature of 71˚C. This study demonstrated that electronic circuits can be effectively cooled by microchannels and supported further investigations. Peng and Peterson [2] experimentally investigated the single-phase forced convective heat transfer in micro-channel structures with small rectangular channels having hydraulic diameters of 0.133–0.367 mm for different types of geometric configurations. The results show that the laminar heat transfer is dependent upon the aspect ratio i.e. the ratio of hydraulic diameter to the center-to-center

This paper was recommended for publication in revised form by Kwok-wing Chau
[1]Department of Mechanical Engineering, Delhi Technological University, Bawana Road, Delhi, INDIA
**E-mail address: mzunaid3k@gmail.com*
Manuscript Received 30 May 2016, Accepted 23 August 2016

distance of micro-channels. Further investigations led by Fedorov and Viskanta [3] indicated that the average channel wall temperature along the flow direction was nearly uniform except in the region close to the channel inlet, where very large temperature gradients were observed. This allowed them to conclude that the thermo–properties are temperature dependent. Qu and Mudawar [4] performed experimental and numerical analysis of pressure drop and heat transfer in a single-phase microchannel (0.231 mm x 0.713 mm) at two heat flux levels. Deionized water was employed as the cooling liquid. Good agreement was found between the measurements and numerical predictions, validating the use of conventional Navier–Stokes equations for micro channels. The main objective of these studies is to develop heat transfer modeling tools that are essential to design and optimize heat sink geometry. To develop such optimized geometry, description of flow process in heat sink is essential. Heat transfer in micro-channel involves conduction of heat in the solid heat sink and convection to the coolant fluid. Numerical simulation methods are thus used to provide more accurate description of fluid flow and heat transfer through the microchannel. Using numerical simulation methods, Liu and Garimella [5] studied fluid flow and heat transfer in micro channels and confirmed that, the behavior of micro channels is quite similar to that of conventional channels. Their analysis showed that conventional correlations offer reliable predictions for the laminar flow characteristics in rectangular micro channels over a hydraulic diameter in the range of 244–974 μm. Sabbah et al. [6] observed that the prediction of heat transfer in micro-channels becomes difficult with increase in complicacy of the geometry of the micro-channels, requiring three dimensional analysis of heat transfer in both solid and liquid phases. Despite the small width of the channels, the conventional Navier Stokes and energy conservation equations still apply to the flow (as also observed by Qu and Mudawar [4]) due to the continuum of the working fluid where the channel width is many times larger than the mean free path of liquid molecules (water). Similar results were shown by Mokrani et al. [7] which showed that the conventional laws and correlations describing the flow and convective heat transfer in ducts of large dimension are directly applicable to the micro channels of heights between 500 and 50 microns. The effects of geometric parameters on the Nusselt number were illustrated by P. Mohajeri Khameneh [8] in a single-phase laminar flow and forced convective heat transfer of water in microchannels with the aim to obtain computational Nusselt number in laminar flow. This study showed that the average Nusselt number is enhanced by increasing the width or decreasing the height. Rebrov et al. [9] have reviewed the experimental and numerical results on fluid flow distribution and heat transfer. They found that the experiments with single channels are in good agreement with predictions using the published correlations. The review consists of two parts. In the first, the main methods to control flow distribution were reviewed. Several different designs of inlet/outlet chambers were presented together with corresponding models used for optimization of flow distribution. Nanofluids, which are fluids containing nanometer-sized particles, have emerged as one of the most effective coolants in microchannels. They exhibit enhanced thermal conductivity and the convective heat transfer coefficient compared to the base fluid. Roy et al. [10] studied heat transfer using nanofluids as a coolant inside a radial channel between two coaxial and parallel discs. The non-dimensional governing equations of mass, momentum and energy were solved using computational fluid dynamics. Results have shown that the inclusion of nanoparticles in a traditional coolant can provide considerable improvement in heat transfer rates, even at small particle volume fractions.

Chai, Xia and Wang [11] studied the laminar flow and heat transfer characteristics in the interrupted microchannel heat sink with ribs in the transverse microchambers. In the study they examined the effect of such ribs on velocity contour, pressure distribution and temperature distribution. They also studied the local pressure drop and heat transfer characteristics in such microchannel heat sinks. Results showed that the ribs in the transverse microchambers could effectively prevent the decline of local heat transfer coefficient along the flow direction. A wavy channel structure was studied by Sakanova, Keian, Zhao [12] with the application of nanofluids. The effects of wavy amplitude, wavelength, volumetric flow rate and volume fraction of different types of nanofluids were studied. They found that with pure water as coolant the heat transfer performance of the wavy channel structure is improved, while the replacement of the pure water by nanofluids makes the effect of wavy wall unnoticeable. Xu et al. [13] numerically investigated the flow characteristics and heat transfer in microchannel with dimples. Different geometric parameters of dimpled channel were independently studied under constant Reynolds Number of 500 and a constant heat flux 1 W/mm2. It was found that in comparison to straight channels, dimpled surface reduced the local flow resistance and also improved thermal performance of microchannel heat sink.

Numerical simulations to investigate the influence of geometric parameters on the flow and heat transfer characteristics of rectangular, trapezoidal and triangular shaped microchannel heat sinks was carried out by

Wang et al. [14]. They found that among three kinds of microchannels, the microchannel with rectangular cross-section has the lowest thermal resistance, followed by trapezoidal and triangular cross-sections microchannels. Dang et al. [15], investigated the effect of heat transfer and pressure drop of water in five rectangular-shaped microchannel heat sinks with different configurations (S-type & I-type), in order to optimize performance and design. They showed that when inlet temperature and mass flow rate is kept constant, the performance index of counter flow is always higher than parallel flow. Further, the effect of hydraulic diameter on performance index was found to be more than that of substrate thickness (1.2 & 2mm).

Heat exchangers have often been subjected to thermodynamic optimization or entropy generation minimization like parallel flow, counter flow, cross-flow and phase change heat exchanger optimization. M.M. Awad & Y.S. Muzychka [16] have provided comprehensive review of the progress made in entropy generation minimization (EGM) in order to use it in internal and external structure heat exchanger. EGM combines basic concepts of heat transfer, fluid dynamics and thermodynamics with simple models like microchannel heat exchangers. These models are then used in practical devices and process optimization. Mohamed M Awad [17] has chronologically reviewed the research progress made on EGM (Entropy generation minimization). Entropy generation analysis as the evaluation parameter was used in many studies of the review. Different types of working fluids like water, engine oil, aniline, nanofluids and non-Newtonian were used in the studies. He has also made recommendations of future work for thermodynamic optimization of microchannels using entropy generation analysis. Dongmei Zhou and Timothy Rau [23] correlated numerical simulations and hand results with experimentally obtained results. They designed a methodology to improve the accuracy of the simulations.

In the present work, the pressure drop and heat transfer characteristics of a single-phase rectangular micro-channel with semi cylindrical projections are investigated numerically. It is evident from the above given literature review and to best knowledge of various studies that the numerical analysis of such geometry have not been done before. To ensure the accuracy and reliability of the model, the simulation results of straight microchannel heat sink have been benchmarked with Qu and Mudawar's experimental work [4]. It shows that the numerical data is in good agreement with the experimental data. Heat transfer and pressure drop characteristics for single phase rectangular microchannel with semi cylindrical projections are computed for various Reynolds number in the laminar flow. These are then compared with results of the single phase rectangular microchannel heat sink. An increase in heat transfer and pressure drop is seen for all Reynolds number in the laminar flow. The results are used to assess the suitability of macro transport models in depicting the transport characteristics of micro-channel heat sinks. A detailed description of the heat transfer characteristics is presented and discussed.

MODEL DESCRIPTIONS

Computational and Geometric Configurations

The geometrical configurations of both rectangular and semi cylindrical projections microchannel have been given.

Straight Rectangular Microchannel

A three dimensional CFD analysis for the heat transfer and pressure drop in a rectangular micro-channel is done.

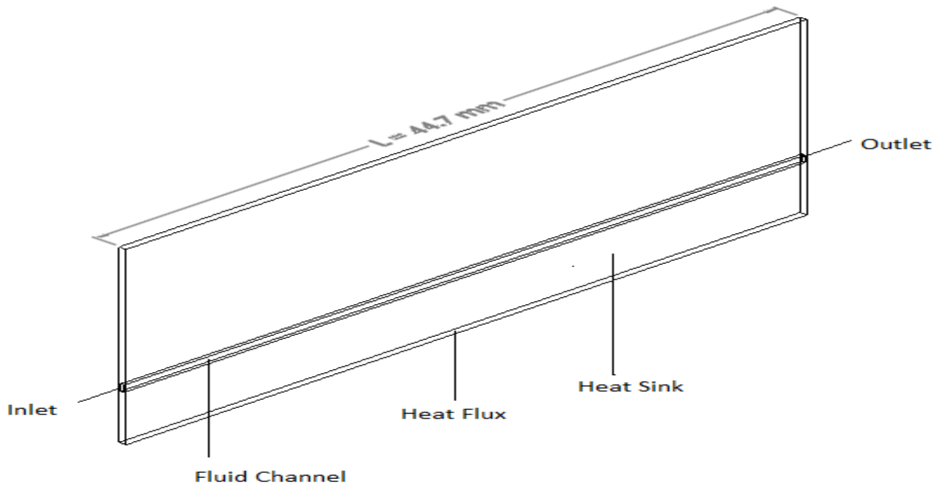
In order to validate this numerical study a reliable experimental study by Qu and Mudawar [4] is necessitated.

The experimental work done by Qu and Mudawar [4], on the test apparatus is modeled and simulated in the present study. In the test apparatus, the heat sink was fabricated from oxygen free copper. Water was moved through straight rectangular channels implanted in a test module. 21 rectangular smaller scale openings were machined into micro- channel surface by an accurate machining procedure (there are 21 parallel rectangular small scale directs in the module). The miniaturized scale openings were equidistantly divided inside the 1 cm heat sink width and had the cross-sectional measurements of 231 μm (width) by 712 μm (height).

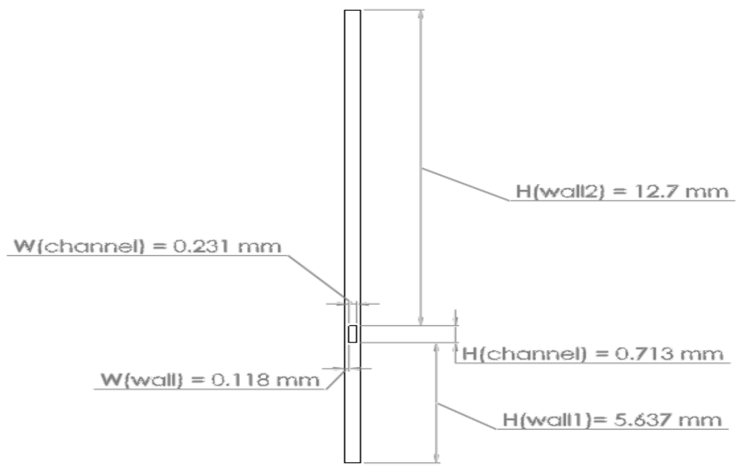
Since symmetry allows the results to be extended to the entire microchannel, a single microchannel with surrounding solid is used for numerical analysis. A schematic and cross section of the modeled microchannel structure used in this study is shown in Figure 1. Meanwhile, geometrical parameters of this model are presented in Table 1.

Table 1. Dimensions of the unit cell used for simulation

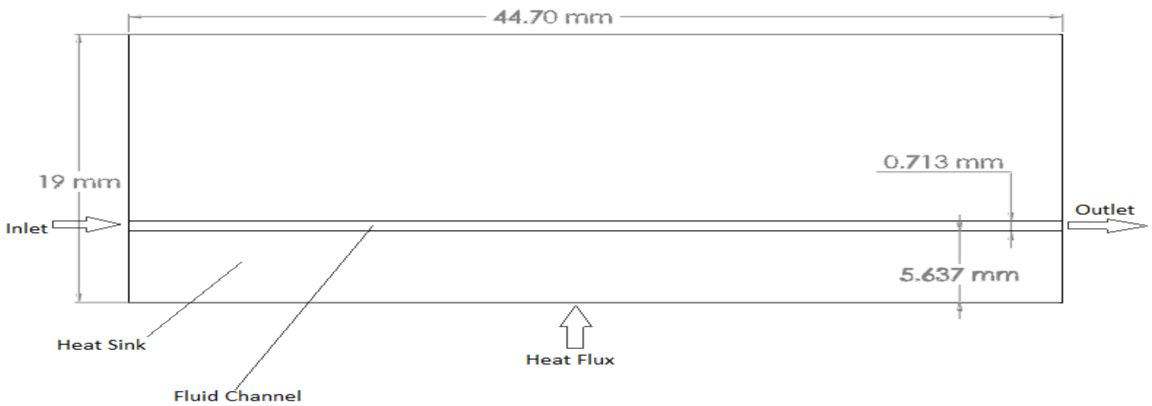
W_{wall}(μm)	$W_{channel}$(μm)	H_{wall2}(μm)	$H_{channel}$(μm)	H_{wall1}(μm)	L(mm)
118	231	12700	713	5637	44.7



(a)



(b)



(c)

Figure 1. Geometrical construction of the unit cell of straight microchannel heat sink used for simulation: - a) Isometric view, b) Side view, c) Front view

In order to increase the calculative capability, fine grids were applied to the microchannel volume and coarse grids to the rest of the heat sink.

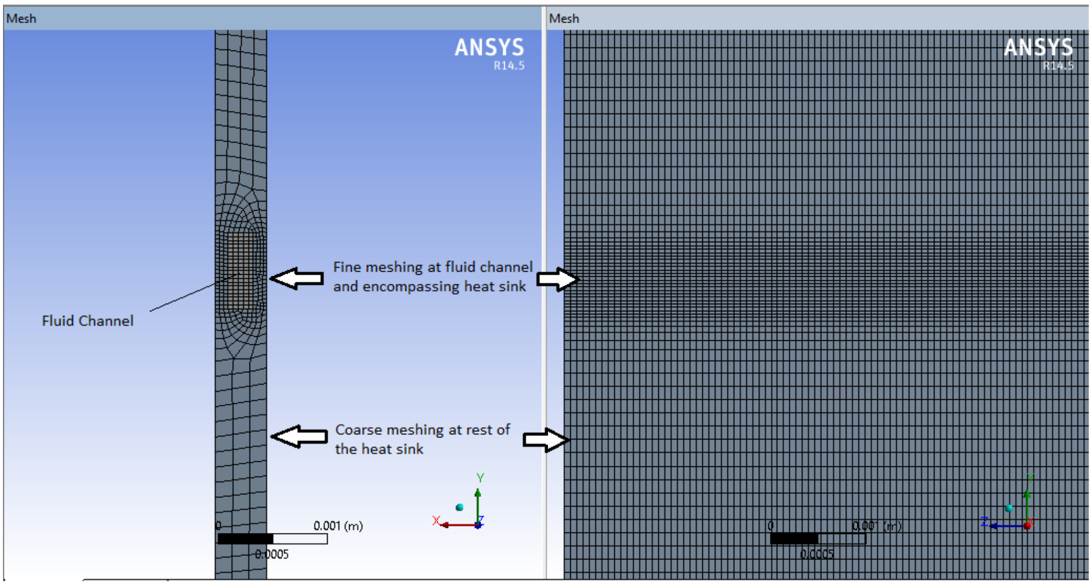


Figure 2. Sectional geometry and meshing grids of straight rectangular microchannel

Semi cylindrical Projections Microchannel

A schematic and cross section of the modeled microchannel structure used in this study is shown in Figure 3. The geometry has same external dimensions as that of the straight microchannel.

Modifications are made in the fluid channel. In the rectangular channel used for the fluid flow in straight microchannel, semi cylindrical projections were added, made of the same material as that of heat sink, placed alternatively on the upper and lower wall of the rectangular slot with its axis perpendicular to the direction of flow of fluid. The radius of the semi cylinder projections used, was taken as R=0.18 mm and length of the projections in same as the width of the rectangular channel. Meanwhile, geometrical parameters of this model are presented in Table 2.

Table 2 Dimensions of the semi cylindrical projections microchannel used for simulation

W_{wall}(μm)	$W_{channel}$(μm)	H_{wall2}(μm)	$H_{channel}$(μm)	H_{wall1}(μm)	L(mm)	R(mm)
118	231	12700	713	5637	44.7	0.18

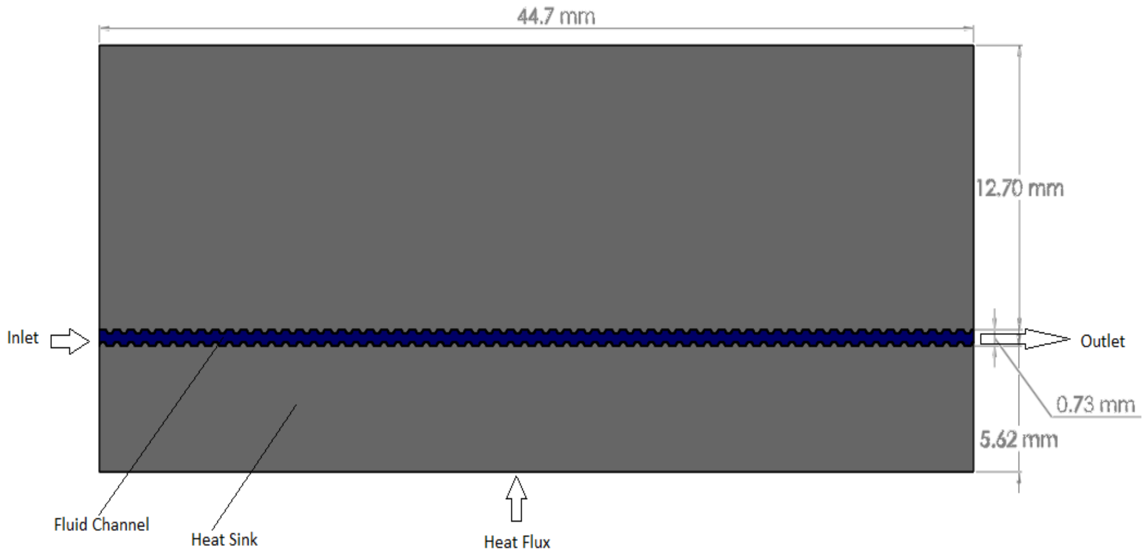
Governing Equations and Boundary Conditions

Several assumptions were considered before arriving on the governing equations and boundary conditions: -

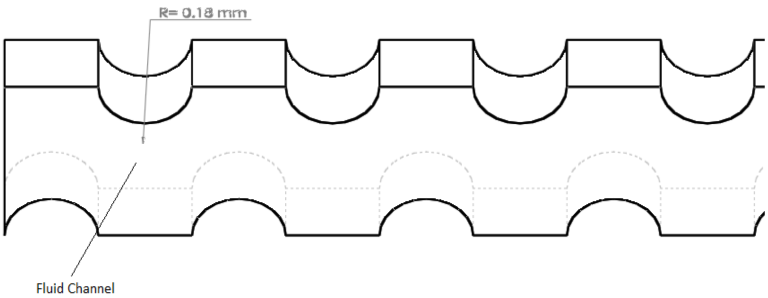
1. The process is steady and the fluid is incompressible.
2. The flow is laminar.
3. The body forces are neglected.
4. The side walls and the top wall of the microchannel structure are adiabatic.
5. Radiation heat transfer and natural convective heat transfer (due to air trapped in heat sink slots) are neglected.

Heat transfer in microchannels is a conjugate one, involving conduction in heat sink and convection to the coolant. One method to solve this conjugate heat transfer problem is to assume a combined computational domain (fluid and solid), [18], [19].

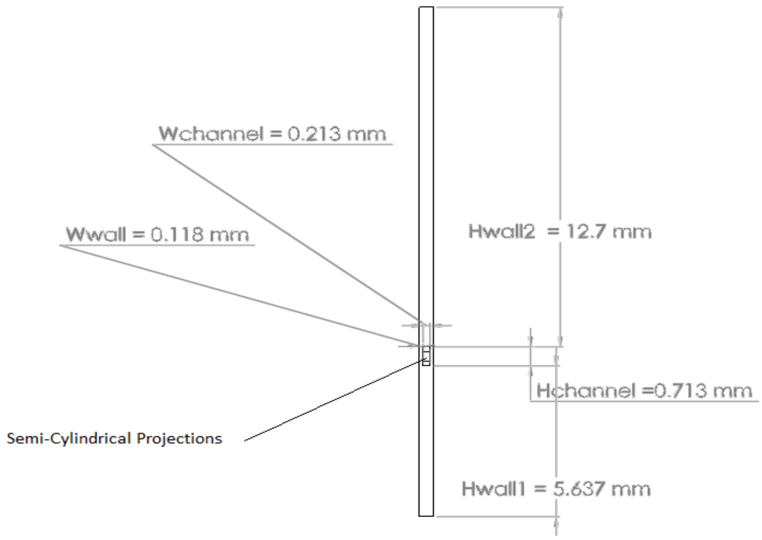
Based on the given assumptions following governing equations are used to describe the fluid flow characteristics and heat transfer.



(a)



(b)



(c)

Figure 3. Geometrical construction of the unit cell of semi cylindrical projections microchannel heat sink used for simulation: - a) Front View, b) Fluid Channel, c) Side View

The equation for conservation of mass, or continuity equation, can be written as follows:

$$\frac{\partial \rho}{\partial t} + \nabla.(\rho\vec{v}) = S_m \tag{1}$$

The equation written above is a general equation for conservation of mass. Here Sm ¬ is the mass added from any user defined sources.

Momentum conservation in a reference frame which is non accelerating can be written as: -

$$\frac{\partial}{\partial t}(\rho\vec{v}) + \nabla.(\rho\vec{v}\vec{v}) = -\nabla p + \nabla.(\bar{\bar{\tau}}) + \rho g + \vec{F} \tag{2}$$

Where, p is the static pressure, τ is the stress tensor, F and ρg are the external forces on body and gravitational force on body.

The energy conservation equation is given as: -

$$\frac{\partial}{\partial t}(\rho E) + \nabla.(\vec{v}\,(\rho E + p)\,) = \nabla.\left(k_{eff}\nabla T + \sum_{J} h_j\vec{J_J} + \overline{\overline{\tau_{eff}}}.\vec{v}\right) + S_h \tag{3}$$

where, k_{eff} is the effective conductivity ($k+k_t$), where $_{kt}$ is the turbulent thermal conductivity, defined on the basis of turbulence model being used, and J j is the diffusion flux of species J. On the right-hand side of the equation the first three terms represent energy transfer due to species diffusion, conduction, and viscous dissipation, respectively, S_h denotes the chemical reaction heat, and some other sources of heat.

As, we have assumed a unitary computational domain, we only need to specify boundary conditions for the unit cell, which are given as: -

Hydraulic Boundary Conditions

1. Uniform velocity at the inlet of the channel.
2. At channel outlet, mass flow rate boundary condition is applied.
3. Zero velocity at all other solid boundaries.
4. No slip at the surface.

Thermal Boundary Conditions

1. A uniform heat flux of 10^6 W/m^2 at the bottom wall of the heat sink.

Table 3 Zone Specification for straight rectangular microchannel

Heat sink front wall	Wall
Heat sink top wall	Wall
Heat sink back wall	Wall
Heat sink bottom wall	Heat flux
Heat sink right wall	Wall
Heat sink left wall	Wall
Channel entry	Static pressure
Channel outlet	Mass flow outlet
Default Interface	Wall

Data Reduction

The relation between the hydraulic diameter and Reynolds number in a channel is given by

$$Re=\rho v D_h/\mu \tag{4}$$

where, ρ is density, v is average velocity ,µ is dynamic viscocity, D_h is hydraulic diameter of channel, [20], [21].

Mass flow rate is calculated as

$$\dot{m} = \frac{Re.A.\mu}{D_h} \tag{5}$$

Where hydraulic diameter is given as,

$$D_{h=}\frac{4A}{P} \tag{6}$$

where, A=area of cross section of microchannel [$W_{channel}$ x $H_{channel}$] and

P= wetted perimeter[2 x ($W_{channel}$ +$H_{channel}$)]

RESULTS AND DISCUSSIONS

Model Validation

To ensure the accuracy and reliability of the model, the simulation results of straight microchannel heat sink have been benchmarked with Qu and Mudawar's experimental work [4].

Pressure Drop

For the validation of the pressure drop, the experimental pressure drop along the channel is compared with the computational pressure drop.

Figure 3 shows the comparison of experimental and numerical pressure drop results for varying Reynolds numbers. The computational values are found to be in good agreement with the experimental values.

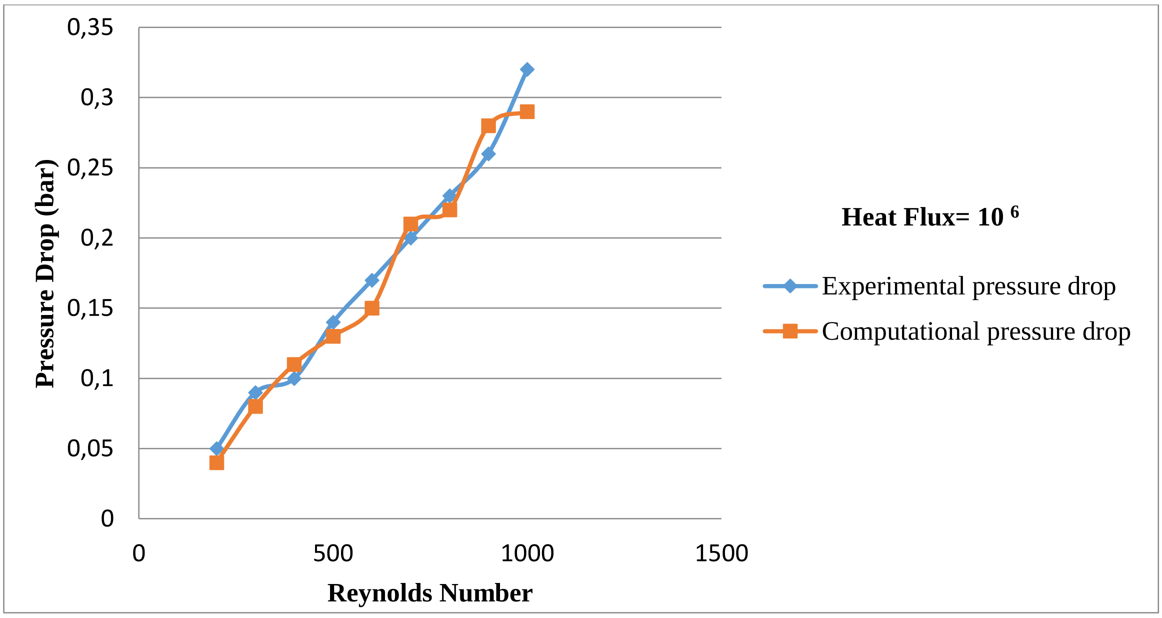


Figure 4. Pressure drop vs Reynolds number for straight rectangular geometry.

A linear relationship is expected between pressure drop and Reynolds number owing to the constant fluid properties. The reason for slope change can be attributed to the temperature dependency of viscosity. For a constant inlet water temperature, the outlet temperature decreases with increase in the Reynold number (Figure 5), which increases water viscosity and hence results in a greater pressure drop.

The trend of graph changes after 600 Reynolds number. This can be attributed to the conversion from the laminar to transition region at 600 Reynolds number, as suggested by Peng and Peterson, [2].

Temperature rise

For further validation the experimental temperature rise is compared with the computational temperature rise. Figure 5 shows the comparison of experimental and numerical temperature rise results for varying Reynolds numbers. The computational values are found to be in good agreement with the experimental values.

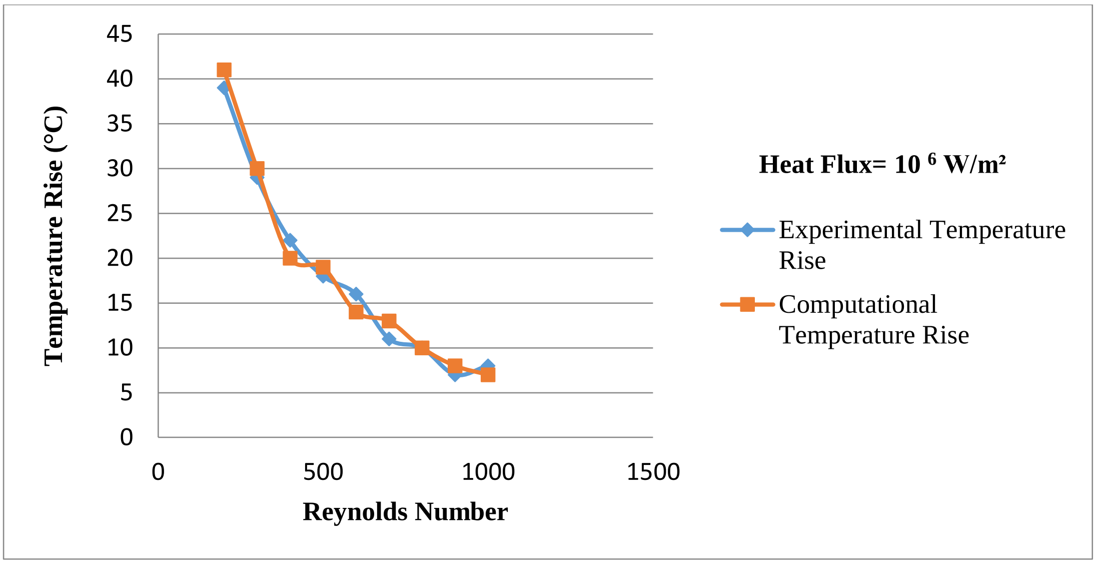


Figure 5. Temperature Rise Vs Reynolds Number for straight rectangular geometry

As expected the heat transfer rate decreases with increase in Reynolds number. But it can be clearly seen that the amount of decrease, falls drastically by increasing the velocity or the Reynolds number. The temperature rise can be given by the following function, [22]: -

$$\text{Temperature rise}: \quad 265.67\ v^{-0.4997} \tag{7}$$

Comparison of Pressure Drop in Two Geometries

Pressure drop across the fluid channel in semi cylindrical projections microchannel heat sink is evaluated using ANSYS CFX package. A comparison is made between the pressure drop in straight micro-channel and semi cylindrical projections micro-channel.

Clearly, as seen from the graph, the pressure drop of water across the fluid channel is found to be greater than that of straight micro-channel for every Reynolds number (200-1000) for constant heat flux of 10^6 W/m^2. The increase in pressure drop is obvious due to the presence of semi cylindrical projections. These projections hinder the fluid flow in the microchannel and hence increase the friction factor. These hindrances lead to a higher pressure loss. The reason for slope change can be attributed to the temperature dependency of viscosity. For a constant inlet water temperature, the outlet temperature decreases with increase in the Reynolds number (Figure 5), which increases water viscosity and hence results in a greater pressure drop. A trend change at Reynolds number 600 can be attributed to the conversion from laminar to transition region, [2].

Comparison of Temperature Rise in Two Geometries

The major goal in this section is to evaluate the temperature rise of water from inlet to outlet in the semi cylindrical projections micro-channel using CFD simulations. Then, an exact comparison is made between the temperature rise in straight micro-channel and semi cylindrical projections micro-channel.

Clearly, as seen from the graph, the temperature rise of water from inlet to outlet in semi cylindrical projections micro-channel is found out to be greater than that of straight micro-channel for every Reynolds number (200-1000) for constant heat flux of 106 W/m2.

The reasons for this behavior are as follows

- The structure analyzed increases the convection heat transfer area.
- This structure interrupts thermal boundary at a regular interval along the flow direction and hence increase the heat transfer.
- Increment in heat transfer can also be attributed to increase in fluid mixing with the help of proposed microchannel structure (semi cylindrical projections).

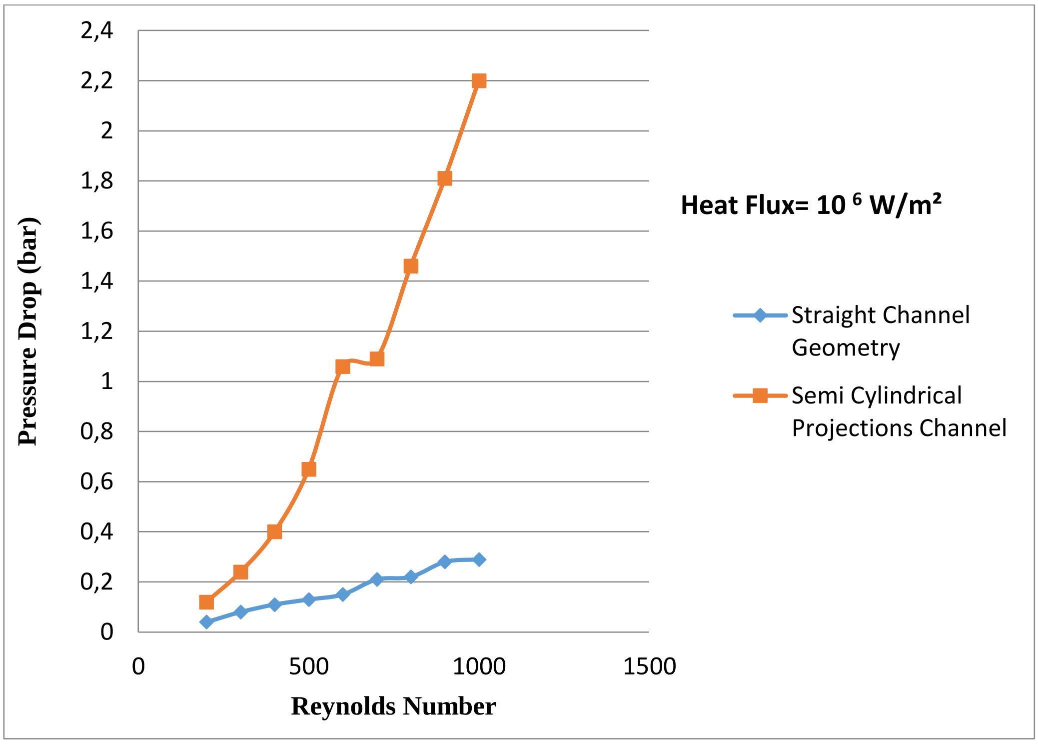


Figure 6. Pressure drop vs Reynolds number comparison of straight and semi cylindrical projections microchannel heat sink

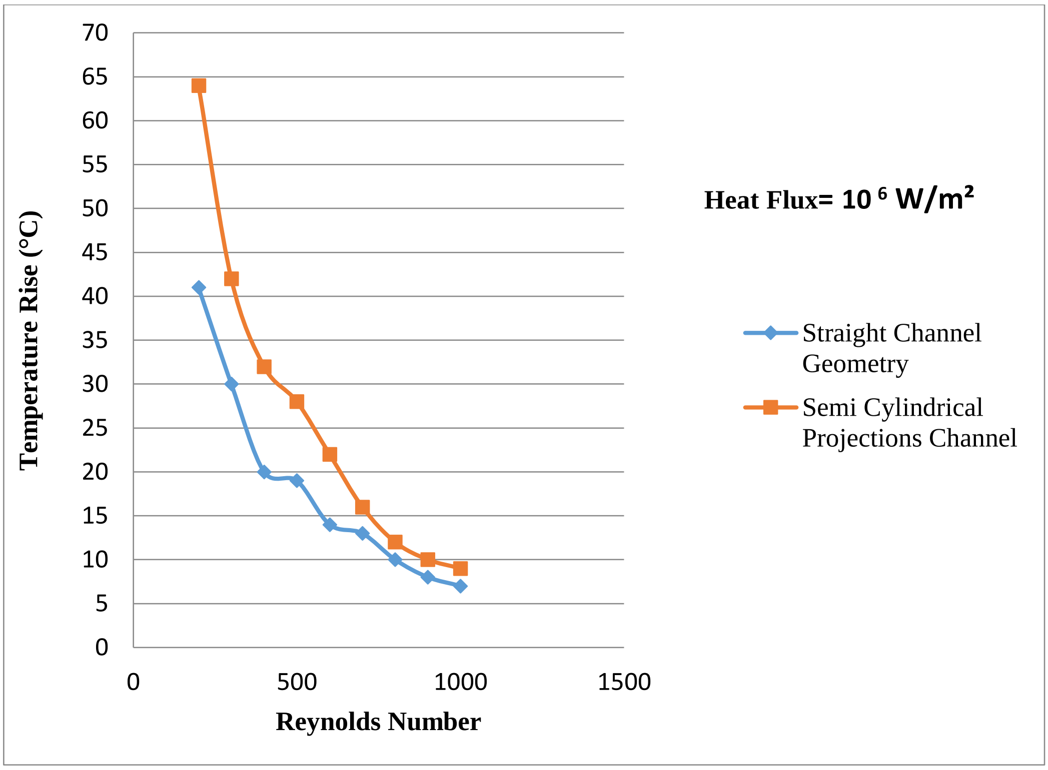


Figure 7. Temperature rise vs Reynolds number comparison of straight and semi cylindrical projections microchannel heat sink

CONCLUSIONS

In this paper, we have investigated, by numerical simulation the conjugate heat transfer problem in a totally new microchannel geometry (semi cylindrical projections) with the assumption of single phase flow. Numerical analysis of flow properties and heat transfer is conducted and results are laid out. These results are compared with that of a rectangular microchannel heat sink to assess the design of the new geometry. Based on the results following conclusions can be made-

- The heat transfer in semi cylindrical projections microchannel heat sink is greater than that of rectangular microchannel heat sink. This structure increases the convection heat transfer area. It also interrupts thermal boundary layer periodically in the flow direction and hence increases the heat transfer.
- Increment in heat transfer can also be attributed to increase in fluid mixing with the help of proposed microchannel structure (semi cylindrical projections).
- The effect of semi cylindrical projections on pressure drop is obvious and pressure drop increases with increase in Reynolds number. These projections hinder the fluid flow in the microchannel and hence increase the friction factor. These hindrances lead to a higher pressure loss compared to rectangular microchannel heat sink.
- The computational values for heat transfer and pressure drop are found to be in good agreement with that of the experimental values and hence proves that the energy equations can predict the flow and heat transfer characteristics of a microchannel.
- The heat transfer in straight rectangular microchannel heat sink decreases with increase in Reynolds number as the flow velocity increases. The amount of decrease, falls drastically by increasing the velocity.
- The pressure drop in straight rectangular microchannel heat sink increase with increase in Reynolds number. The change in slope can be attributed to the temperature dependency of viscosity of water.

NOMENCLATURE

D_h	Hydraulic Diameter of Channel, [mm]
F	Force, [N]
g	Gravitational Acceleration, [m^2/s]
H	Coefficient of Convective Heat Transfer, [W/(m^2 K)]
$H_{channel}$	Height of Channel, [µm]
H_{wall1}	Height of Wall 1, [µm]
H_{wall2}	Height of Wall 2, [µm]
J_j	Diffusion Flux of Species J, [$m^{-2}s^{-1}$]
k_{eff}	Effective Thermal Conductivity, [W/(m K)]
k_t	Turbulent Thermal Conductivity, [W/(m K)]
M	Mass Flow Rate across Channel, [kg/s]
P	Static Pressure, [bar]
Re	Reynolds Number
S_h	Chemical Reaction Heat, [J]
S_m	Mass Added from any user Defined Sources, [kg]
T	Time,[s]
T	Temperature,[K]
V	Average Velocity,[m/s]
$\vec{v}$	Velocity Vector, [m/s]
$W_{channel}$	Width of Channel, [µm]
W_{wall}	Width of Wall, [µm]

Greek Symbols

ρ Density, [kg/m^3]

μ Dynamic Viscosity, [kg/(m s)]

τ Stress Tensor, [N/m^2]

REFERENCES

[1] D.B. Tuckerman, R.F.W. Pease, "High-performance heat sinking for VLSI", IEEE Electron. Dev. Lett. EDL-2 (1981) 126–129.

[2] X.F. Peng, G.P. Peterson, "Convective heat transfer and flow friction for water flow in microchannel structures", Int. J. Heat Mass Transfer 39 (1996) 2599–2608.

[3] A.G. Fedorov, R. Viskanta, "Three-dimensional conjugate heat transfer in the microchannel heat sink for electronic packaging", Int. J. Heat Mass Transfer 43 (2000) 399–415.

[4] I. Mudawar, W. Qu, "Experimental and numerical study of pressure drop and heat transfer in a single-phase micro-channel heat sink", Int. J. Heat and Mass Transfer 45(2002) 2549-2565.

[5] Dong Liu and Suresh V. Garimella, Investigation of Liquid Flow in Microchannels, Journal of Thermophysics and Heat Transfer, 18(2004) pp.65-72.

[6] Sabbah, R., Yagoobi, J., & Hallaj, S. A., Experimental and numerical investigation of heat transfer characteristics of liquid flow with micro-encapsulated phase change material, *ASME Summer Heat Transfer Conference,* 2 (2008) 495-501.

[7] Omar Mokrani, Brahim Bourouga, Cathy Castelain, Hassan Peerhossaini, Fluid flow and convective heat transfer in flat microchannels, Int. J. Heat Mass Transfer 52 (2009) 1337–1352

[8] P. Mohajeri Khameneh et al. A numerical study of single-phase forced convective heat transfer with flow friction in microchannels, International Journal of Engineering 25 (2012) 79-88.

[9] Evgeny V. Rebrov, , Jaap C. Schouten, Mart H.J.M. de Croon Single-phase fluid flow distribution and heat transfer in microstructured reactors, Chemical Engineering Science66(2011)1374-1393.

[10] Gilles Roy, Cong Tam Nguyen, Paul-René Lajoie, Numerical investigation of laminar flow and heat transfer in a radial flow cooling system with the use of nanofluids, Superlattices and Microstructures 35 (2004) 497–511.

[11] Lei Chai, Guo Dong Xia, Hua Sheng Wang, Laminar flow and heat transfer characteristics of interrupted microchannel heat sink with ribs in the transverse microchambers, International Journal of Thermal Sciences 110 (2016) 1-11.

[12] Assel Sakanova, Chan Chun Keian, Jiyun Zhao, Performance improvements of microchannel heat sink using wavy channel and nanofluids, International Journal of Heat and Mass Transfer 89 (2015) 59–74.

[13] Minghai Xu, Hui Lu, Liang Gong, John C. Chai, Xinyue Duan, Parametric numerical study of the flow and heat transfer in microchannel with dimples, International Communications in Heat and Mass Transfer 76 (2016) 348–357 .

[14] Hongtao Wang, Zhihua Chen, Jianguo Gao, Influence of geometric parameters on flow and heat transfer performance of micro-channel heat sinks, Applied Thermal Engineering 107 (2016) 870–879

[15] Thanhtrung Dang, Jyh-tong Teng, Jiann-cherng Chu, Tingting Xu, Suyi Huang, Shiping Jin and Jieqing Zheng (2012). Single-Phase Heat Transfer and Fluid Flow Phenomena of Microchannel Heat Exchangers, Heat Exchangers - Basics Design Applications, Dr. Jovan Mitrovic (Ed.), InTech, DOI: 10.5772/32989.

[16] M.M. Awad, Y.S. Muzychka, Thermodynamic optimization, heat exchangers – basics design applications, in: J. Mitrovic (Ed.), Heat Exchangers – Basics Design Applications, vol. 1, ch. 1, InTech, Rijeka, pp. 3-52, InTech, ISBN 978-953-51-0278-6, 2012.

[17] Mohamed M Awad, A review of entropy generation in microchannels, Advances in Mechanical Engineering 2015, Vol. 7(12) 1–32

[18] S.V. Patankar, A numerical method for conduction in composite materials, flow in irregular geometries and conjugate heat transfer, in: Proceedings of 6th International Heat Transfer Conference, vol. 3, 1978, pp. 297–302.

[19] S.V. Patankar, Numerical Heat Transfer and Fluid Flow, Hemisphere, Washington, DC, 1980.

[20] Yunus A.Cengel, John M.Cimbala, Fluid Mechanics, Third edition, McGraw Hill Education, New York(2013)

[21] Frank M. White, Fluid Mechanics, Seventh edition, McGraw Hill Education (2011)

[22] N. Amanifard, M. Fakhrai, A. Arzpeyma, Y. Mojib and M. Abazare, Numerical Analysis Of Fluid Flow And Heat Transfer In Microchannels, Journal of Applied Mathematics Islamic Azad University Lahijan, Autumn 2004, Page 58-70.

[23] Dongmei Zhou, Timothy Rau, Thermal Design Methodology and Prediction of Heat Sink Performance,

Journal of Thermal Engineering, Yildiz Technical University Press, Istanbul, Turkey Vol. 2, Special Issue 4, No. 4, pp. 826-836, July, 2016.

[24] Erdal Çetkin, Inverted Fins for Cooling of a Non-Uniformly Heated Domain , Journal of Thermal Engineering, Yildiz Technical University Press, Istanbul, Turkey Vol. 1, Issue No. 1, pp. 1-9, January, 2015.