Available online at www.ijapie.org

INTERNATIONAL JOURNAL OF ADVANCED PRODUCTION AND INDUSTRIAL ENGINEERING

IJAPIE-2017-01-114, Vol 2 (1), 23-28

IJAPIE
Connecting Science & Technology with Management.

A Journal for all Products & Processes.

Investigations of Fluid Flow and Convective Heat Transfer in Wavy Micro-Channel at Different Reynolds Numbers

Mohammad Zunaid[1*], Afzal Husain[2] , Anant Jindal[1], Avinash Gupchup[1]

[1](*Department of Mechanical Engineering, Delhi Technological University, Bawana Road, Delhi)*
[2](*Department of Mechanical and Industrial Engineering, Sultan Qaboos University, Muscat Oman)*
**Email: mzunaid3k@gmail.com*

Abstarct: *Microchannels' heat performance and overall characteristics can be improved by employing different fluid channel structures, different aspect ratios and various kinds of coolants. In this study a microchannel with a wavy fluid channel is analysed for its various characteristics like heat transfer and pressure drop. The conjugate heat transfer problem is solved for different values of Reynolds number. For three dimensional analysis the geometry is created in CAD software Solidworks and then imported in ANSYS for CFD simulation. The simulation model is first validated by comparing the computational results for straight microchannel with that of the experimental results. They are found to be in great agreement with the experimental results. A constant heat flux of 100 W/m^2 is applied at the heat sink bottom. With water taken as coolant and copper as material for heat sink, the heat transfer and pressure drop characteristics are analysed for different Reynolds numbers. It is found that both heat transfer and pressure drop increased with the employment of wavy channel as compared to straight microchannel for all values of Reynolds numbers.*

Keywords: *CFD, Heat sink, Microchannel, Wavy*

1. INTRODUCTION

The electronic devices have become more powerful and small due to the recent developments and growth in this industry. These small circuits also produce a large amount of heat which need to be dissipated efficiently and effectively. Microchannel heat sinks are used for removal of large amount of heat from a very small surface area. Due to its inherent advantages of small size, small coolant requirements and large heat dissipation, microchannels have replaced the conventional coil heat exchangers. Over the last decade, smaller scale machining technology is utilized for the improvement of exceptionally productive cooling gadgets known as microchannels. Thus, the investigation of heat transfer and fluid flow in smaller scale channels which are two important parts of such appliances,are important in both designing and performance issues.

Tuckerman and Pease [1], were the first to study heat transfer in microchannel heat sink. They fabricated a 1×1 cm^2 rectangular microchannel employing water as the cooling fluid. The microchannel heat sink was able to dissipate 790 W/cm^2 at the expense of a pressure drop of 2.2 bar. This study demonstrated that electronic circuits can be effectively cooled by microchannels and supported further investigations. Peng and Peterson [2] also investigated the forced convective heat transfer in micro-channel structures with small rectangular channels having different hydraulic diameters of 0.133–0.367 mm for different types of geometric configurations experimentally. It showed that the laminar heat transfer is dependent upon the aspect ratio i.e. the ratio of hydraulic diameter to the center-to-center distance of micro-channels. Qu and Mudawar [3] carried out experimental and numerical analysis of pressure drop and heat transfer in a microchannel at two different heat flux levels. Water was employed as the coolant in the fluid channel. A good agreement was found between the experimental and numerical values, validating and assuring the use of conventional Navier–Stokes equations for microchannels. The main objective of this study was to develop heat transfer modelling tools that are essential to design and optimize heat sink geometry. Heat transfer in micro-channel involves conduction of heat in the solid heat sink and convection to the coolant fluid. Thus it is a conjugate heat transfer problem. Numerical simulation methods are thus used to provide more accurate description of fluid flow and heat transfer through the microchannel. Sabbah et al. [4] found out 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 phases (solid and liquid). Despite the small width of the channels, the Navier Stokes and conventional energy conservation equations still apply to the flow (as also observed by Qu and Mudawar [3]). Liu and Garimella [5] studied fluid flow and heat transfer, using numerical simulations, in micro channels and confirmed that the behaviour of microchannels is quite similar to that of conventional channels. It showed that the conventional correlations offer reliable predictions for the flow characteristics in rectangular micro channels for hydraulic diameters in the range of 244–974 µm. P.Mohajeri Khameneh [6] illustrated the effect of geometrical parameters on Nusselt number in a single-phase laminar flow and convective heat transfer in microchannels. This study showed that the average Nusselt number is increased or enhanced by increasing the width or decreasing the height of the fluid channel. Roy et al. [7] also studied heat transfer by employing nanofluids as a coolant for a radial channel between two coaxial and parallel discs. The governing equations of mass, momentum and energy were solved using computational fluid dynamics in ANSYS CFX. Results have shown that the inclusion of nanoparticles in a coolant can provide considerable improvement in heat transfer rates, even at small particle volume fractions. Chai, Xia and Wang [8] also studied heat transfer characteristics in a new microchannel geometry in which fluid channel was interrupted with ribs in the transverse microchambers. In the study they analysed the effect of such ribs on pressure distribution, velocity distribution and temperature distribution. They also studied the pressure drop and heat transfer characteristics in such microchannels. It was found that the ribs in the transverse microchambers can effectively increase local heat transfer coefficient along the flow direction. Xu et al. [9] numerically investigated the heat transfer and flow characteristics in microchannels with dimples. Studies on different geometric parameters of dimpled channel were conducted under constant Reynolds Number of 500 and a constant value of heat flux. Results showed that dimpled microchannels have better performance as compared to straight microchannels. Dang et al. [10], studied the effect of heat transfer and pressure drop of water in five rectangular-shaped microchannel heat sinks in order to optimize performance and design. They deduced that when inlet temperature and mass flow rate is kept constant, the performance of counter flow is always higher than parallel flow. Wang et al. [14] carried out numerical simulations to analyse the influence of geometric parameters on the heat transfer characteristics of rectangular, trapezoidal and triangular shaped microchannel heat sinks. They observed that among three kinds of microchannels, the rectangular cross-section microchannel has the lowest thermal resistance, followed by trapezoidal and then triangular cross-sections microchannels. Rebrov et al. [12] have studied the experimental and numerical results on heat transfer. It was found that the experimental results of single channels are in good agreement with predicted results using the given correlations. Firstly, the methods to control flow distribution were reviewed. Different designs of inlet/outlet chambers were presented along with corresponding models of flow distribution.In this present work, the heat transfer and pressure drop characteristics of a wavy microchannel are investigated numerically. As clear from the above mentioned literature review, the proposed geometry is new and requires in depth studies and results. To ensure the accuracy, fitness and reliability of the model, the simulation results of straight microchannel heat sink are compared with the experimental results of Qu and Mudawar [3]. Computational results are in good agreement with the experimental results, validating the conjugate heat transfer simulation model. Heat transfer and pressure drop characteristics for wavy microchannel are computed for various Reynolds number at a constant heat flux of $100 W/cm^2$ 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 variation in microchannel's characteristics like heat transfer and pressure drop with different Reynolds number are studied and analysed. The results are then used to evaluate the suitability of transport models in depicting the transport characteristics of single phase microchannel heat sinks.

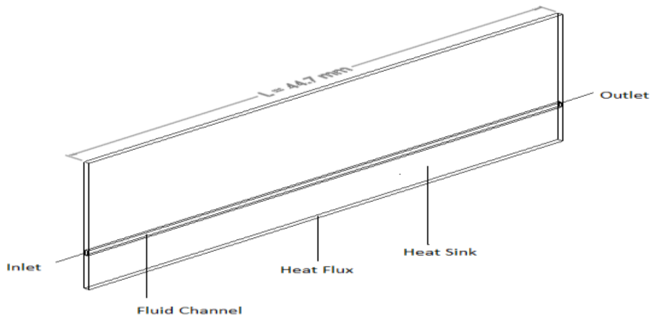
2. MODEL DESCRIPTIONS

2.1 GEOMETRIC CONFIGURATIONS

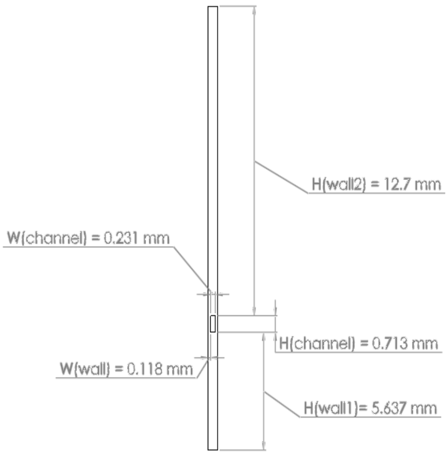
The geometrical configurations of both straight and wavy microchannel are given.

2.1.1 STRAIGHT MICROCHANNEL

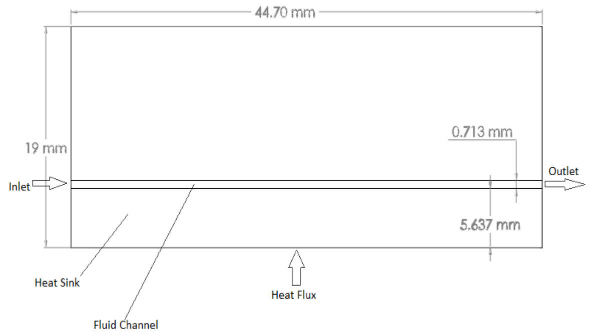
In the experimental work done by Qu and Mudawar [3], the microchannel's characteristics were investigated by a test apparatus. In the test apparatus, the heat sink was fabricated from copper. Water was used as a coolant and was moved through straight rectangular channels implanted in a module. A total of 21 rectangular openings were machined into micro- channel surface by some micro machining procedure (there are 21 parallel rectangular and straight small scale directs in the module). The miniaturized scale openings were equidistantly divided inside the heat sink and had the cross-sectional dimensions of 231 μm (width) by 712 μm (height). As symmetry allows the results to be extended to the entire microchannel, a single microchannel of same cross section with surrounding solid is used for numerical analysis. Fig. 1 depicts the geometrical configurations of the straight microchannel. The geometrical parameters of this model are presented in Table 1.



(a)



(b)



(c)

Figure 1: Geometrical construction of straight microchannel: a) Isometric view, b) Side view, c) Front view

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

2.1.2 WAVY MICROCHANNEL

The schematic diagram of the wavy microchannel is depicted in Fig. 2. The geometry has same external dimensions as that of the straight microchannel. The wave dimensions were decided according to the length of the channel. A sinusoidal wave (Eqn. 1) was constructed in the fluid channel and the amplitude of the wave was taken as 0.15 mm while the wavelength was taken as 2 mm.

$$y = A\sin(2\pi x/\lambda) \quad (1)$$

Here, 'A' is wave amplitude and 'λ' is wavelength.

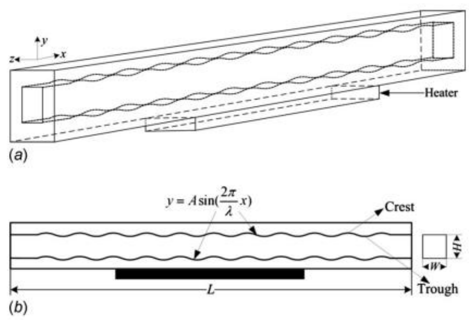


Figure 2: - Geometrical construction of wavy microchannel: a) Isometric view b) Front view

2.2 GOVERNING EQUATIONS AND BOUNDARY CONDITIONS

The assumptions that were taken before arriving on the governing equations and boundary conditions are as follows:
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.

Microchannels heat transfer problem involves two heat transfers. First is the conduction in heat sink and second is convection to the coolant. One method to solve this conjugate heat transfer problem is to assume a combined computational domain. The classical fin analysis method is applied to solve this problem in which heat sink walls are modelled as fins.

Based on the given assumptions, the following governing equations are used to describe the flow characteristics.

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

$$\frac{\partial\rho}{\partial t} + \nabla.(\rho\vec{v}) = S_m \quad (2)$$

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

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

$$\frac{\partial}{\partial t}(\rho\vec{v}) + \nabla.(\rho\vec{v}\vec{v}) = -\nabla p + \nabla.(\bar{\bar{\tau}}) + \rho g + \vec{F} \quad (3)$$

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: -

$$\partial/\partial t\,(\rho E) + \nabla.(v^{\rightarrow}\ (\rho E + p)\ ) = \nabla.(k\_eff\ \nabla T + \sum\_J ⟦h\_j\ (J\_j\ )^{\rightarrow}⟧ + (\tau\_eff\ )^{=}.v^{\rightarrow}\ ) + S\_h \quad (4)$$

where, k_{eff} is the effective conductivity ($k+k_t$), where k_t is the turbulent thermal conductivity, J_j is the diffusion flux of species J, S_h denotes the chemical reaction heat, and some other sources of heat.

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

Hydraulic Boundary Conditions: -
1: - At channel outlet, mass flow rate boundary condition is applied.
2: - Uniform velocity at the inlet of the channel.
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.

2.3 DATA REDUCTION

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

$$Re = \rho v D_h / \mu \quad (5)$$

where, ρ is density, v is average velocity ,µ is dynamic viscocity,D_h is hydraulic diameter of channel.
Mass flow rate is calculated as: -

$$\dot{m} = \frac{Re.A.\mu}{D_h} \quad (6)$$

Where hydraulic diameter is given as,

$$D_{h=}\frac{4A}{P} \quad (7)$$

where, A=area of cross section of microchannel [$W_{channel}$ x $H_{channel}$] and
P= wetted perimeter[2 x ($W_{channel}$ +$H_{channel}$)]

3. RESULTS AND DISCUSSIONS

3.1 MODEL VALIDATION

To ensure that the simulation model produces desired and accurate results, the simulation results of straight microchannel are compared with the experimental results of Qu and Mudawar's [3].

3.1.1 PRESSURE DROP

The numerical values of pressure drop along the channel are compared with the experimental values.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 as shown in Table 2.

Table 2: Comparison of experimental and computational values of pressure drop

Reynolds number	Experimental pressure drop (bar)	Computational pressure drop (bar)
400	0.10	0.11
600	0.17	0.15
800	0.23	0.22
1000	0.32	0.29
1200	0.41	0.366

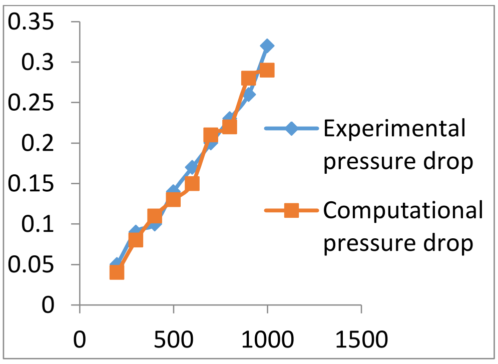


Figure 3: - Pressure drop vs Reynolds number for straight rectangular geometry.

3.1.2 TEMPERATURE RISE

For further validation the computational temperature rise is compared with the experimental temperature rise. Figure 5 shows experimental and numerical temperature rise results for varying Reynolds numbers. As shown in Table 3, the computational values are found to be in good agreement with the experimental values.

Table 3: - Comparison of experimental and computational values of temperature rise

Reynolds number	Experimental temperature rise (°C)	Computational temperature rise (°C)
400	22	20
600	16	14
800	10	10
1000	8	6
1200	6	4

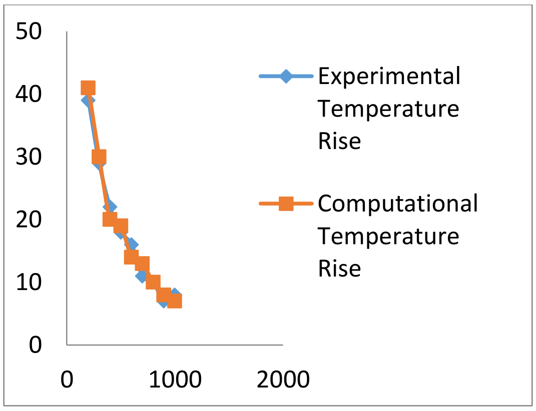


Figure 4: Temperature Rise Vs Reynolds Number for straight rectangular microchannel

3.2 WAVY MICROCHANNEL CHARACTERISTICS

In this section various flow characteristics like heat transfer and pressure drop will be studied for wavy microchannel heat sink at different values of Reynolds number. A constant heat flux of 100W/cm^2 is applied at the heat sink bottom and the values of pressure drop and temperature rise are evaluated for different sets of Reynolds number.

3.2.2 PRESSURE DROP

Pressure drop across the fluid channel in wavy microchannel heat sink is evaluated using ANSYS CFX package. A comparison is made between the pressure drop in straight micro-channel and wavy microchannel. Table 4 depicts the pressure drop for both wavy and straight microchannel heat sink at different Reynolds number.

Table 4: Pressure drop for wavy and straight microchannel heat sink

Reynolds number	Wavy pressure drop(bar)	Straight pressure drop(bar)
400	0.125	0.11
600	0.208	0.15
800	0.291	0.22
1000	0.413	0.29
1200	0.533	0.36

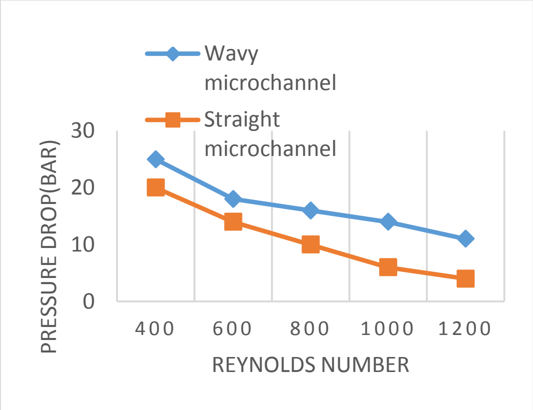


Figure 5: Comparison of Pressure drop for straight and wavy microchannel heat sink

The following inferences can be made out from the given computational values:

- A linear graph is expected between pressure drop and Reynolds number because of the constant fluid properties. The reason for slope change can be given to the temperature dependency of viscosity. For a fixed inlet water temperature, the outlet temperature decreases with increase in the Reynolds number (Fig. 5), and hence increases water viscosity and thus results in a greater pressure drop.
- As seen from Fig. 5, the pressure drop of water across the wavy fluid channel is found to be greater than that of the straight micro-channel for every Reynolds number. The increase in pressure drop can be attributed to the crest and troughs present in the fluid channel. They work as obstacles in the fluid flow and hence increase pressure losses.
- A trend change at Reynolds number 600 can be attributed to the conversion from laminar to transition region, also suggested by Peng and Peterson [2].

3.2.3 TEMPERATURE RISE

Temperature rise across the fluid channel in wavy microchannel heat sink is evaluated using ANSYS CFX package. A comparison is made between the temperature rise in straight micro-channel and wavy microchannel. Table 5 depicts the temperature rise for both wavy and straight microchannel heat sink at different Reynolds number.

Table 5: - Temperature rise for wavy and straight microchannel heat sink

Reynolds number	Wavy temperature rise (°C)	Straight temperature rise (°C)
400	25	20
600	18	14
800	16	10
1000	14	6
1200	11	4

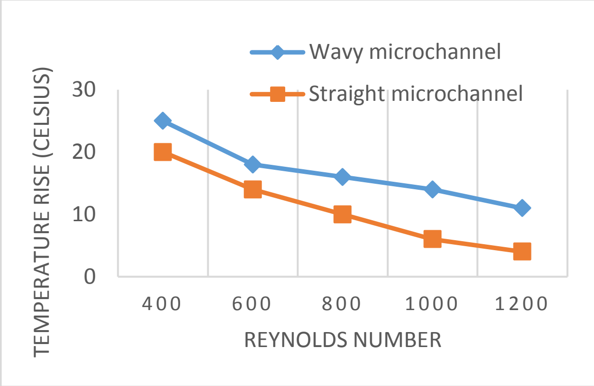


Figure 6: Comparison of Temperature rise for straight and wavy microchannel heat sink.

The following observations and inferences are made from the evaluated computational values:

- The heat transfer decreases with increase in Reynolds number as the velocity increases.
- As seen from the Fig. 6, the temperature rise of water from inlet to outlet in wavy microchannel is found out to be greater in comparison of straight microchannel for every Reynolds number for a constant heat flux of 10^6 W/m^2.
- The structure analysed increases the convection heat transfer area and hence the heat transfer.
- This wavy structure periodically interrupts thermal boundary layer along the flow direction and hence increase the heat transfer.

4. CONCLUSIONS

In the current work, the conjugate heat transfer problem in a wavy microchannel geometry is solved by numerical simulations with the assumption of single phase flow. Numerical analysis of flow characteristics and heat transfer is conducted and results are analysed. These results are then compared with a straight rectangular microchannel heat sink to assess the suitability of the new geometry. Based on the results following conclusions can be made-

- The heat transfer in wavy microchannel heat sink is greater than that of straight rectangular microchannel heat sink. This wavy structure increases the convection heat transfer area and hence the rate of heat transfer. It also interrupts thermal boundary layer periodically along the flow direction, which also results in greater heat transfer.
- Increment in heat transfer can also be credited to the increased mixing of fluid in the wavy channel.
- The effect of wavy fluid channel on pressure drop is obvious and pressure drop increases with increase in Reynolds number. The pressure drop of water across the wavy fluid channel is found to be greater than that of the straight micro-channel for every Reynolds number. The increase in pressure drop can be attributed to the crest and troughs present in the fluid channel. They work as obstacles in the fluid flow and hence increase pressure losses.
- 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 conventional energy equations and Navier- Stokes equation can easily

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] 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.
[4] 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.
[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] P. Mohajeri Khameneh . A numerical study of single-phase forced convective heat transfer with flow friction in microchannels, International Journal of Engineering 25 (2012) 79-88.
[7] 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.
[8] 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.
[9] 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 .
[10] 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.
[11] 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
[12] 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.
[13] Mohamed M Awad, A review of entropy generation in microchannels, Advances in Mechanical Engineering 2015, Vol. 7(12) 1–32
[14] 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.