

Optimal Thermal Design of Converged-Diverged Microchannel Heat Sinks for High Heat Flux applications

Dheepan Chakravarthi. M. K^{*}., Mutharasu. D, and Shanmugan. S.

Lab 335, School of Physics, University of Science Malaysia, 11800 Penang, Malaysia

*Corresponding author: Email: dheepan.mk@gmail.com

Abstract: With the advancements in aerospace technology, micro-electromechanical systems, hybrid data centres and microfluidics, the miniature size electronic chips in such applications are the need of the century. The major challenge in microelectronic chips is to eliminate the generated heat for stable and reliable operation of the devices. Microchannel heat sinks are efficient method to dissipate heat when the generated heat flux is more than 120 W/cm². The pressure drop and thermal resistance in the microchannel are the important parameters which determine the efficiency of the microchannel heat sink. The configuration of the microchannel in terms of thermal resistance and pressure drop is prior attention to design the microchannel heat sink. In this study, a converged-diverged (CD) microchannel heat sink was designed and optimized for the efficient pressure drop and thermal resistance condition.

Keywords: Insert two to five keywords that are descriptive of the work presented in the paper. Separate keywords by commas.

1. Introduction

With the advancements in aerospace technology, micro-electromechanical systems, hybrid data centres and microfluidics, the miniature size electronic chips in such applications are the need of the century. According to the Moore's law, the increase in operating temperature of the electronic boards results in deterioration of component's durability. Also, International technology Roadmap for semiconductors (ITRS,2009) alarmed that the thermal management in high performance chip packages will be crucial for intermittent and efficient operation of the parent device. Innovative as well as effective thermal solution is demanded by the growing technical advancements. Though the worldwide research has been conducted on different thermal heat dissipation, microchannel heat sinks are very promising in terms of high heat flux dissipation capacity and compatibility with the electronics

system. However, the performance of microchannel heat sinks is quite unsatisfactory and lot of research have been demanded in improving the microchannel system.

The research on Microchannels were began by Tuckerman et al [1]. Successive researches have been done to increase the heat transfer coefficients using single phase flow in microchannel with different aspect ratio. Singh et al [1,2] demonstrated the successful application of microchannel cooling for moderation of high heat flux transients such as p-n diodes, Metal-oxide semiconductor Field-Effect transistor (MOSFET) and resistance temperature detectors (RTDs). The problem of flow boiling, flow reversal, reduced critical heat flux, thermal stresses, pressure and temperature fluctuation in linear microchannels urged for different topology [4-7]. Many researches were done on microchannels with micro pin fins, surface roughness, and tortuous channels [8-13]. Nucleate boiling in the above mentioned modifications limits its applications. In general, only sinusoidal wall shape were considered in the design of microchannels [13]. Lee et al used diverging microchannels to mitigate the effects of flow instabilities that causes nucleate boiling [6]. The presence of critical angle which can reduce the pressure drop in single phase microchannel were reported by Agarwal [7].

Relatively few studies on the diverging/converging microchannels were done [14-16]. Louisos and Hitt and V. S. Duryodhan et al studies the convective heat transfer in diverging/converging microchannels [15,16]. The later proposed a methodology for calculating the characteristic length for calculating the characteristic length of varying cross section in microchannels. In their further studies [17,18], experimental demonstration of the isothermal wall condition for a constant heat flux condition were developed. Later, two cross sections, converging and diverging, of the microchannels were considered. The pressure drop and Nusselt number were compared in both the designs. In summary, the variation in cross section of the microchannels increases the heat transfer in

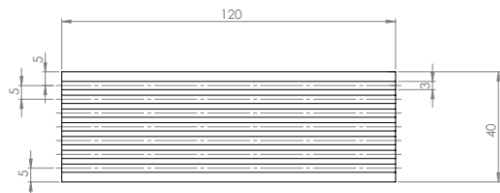
electronic devices of high heat flux. The flow instability in the microchannels were reduced.

In this study, a combination of converging and diverging microchannel heat sink were considered. Converging-diverging (CD) microchannels were designed by Solidworks 15 and a three-dimensional numerical study is conceded for optimizing the CD microchannel dimensions using COMSOL Multiphysics 5.1. The CFD software is used to understand the heat distribution in the CD microchannel heat sink. Throughout this study, a constant heat flux of 120 W/cm^2 were applied on the surface of the heat sink and de-ionized water is considered as the working fluid. On comparing to the straight microchannel, the CD microchannel proved better results in terms of overall thermal resistance.

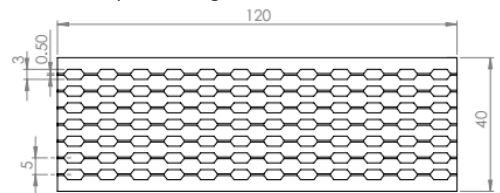
2. Numerical Modelling

2.1 Geometry description:

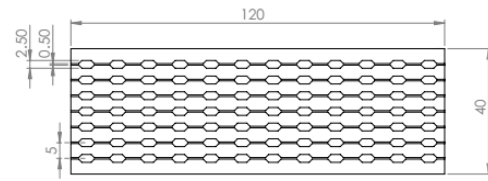
The configuration of converging-diverging microchannel used in the present study is as shown in Fig.1. The thermal performance of the proposed microchannel is compared with two of the basic straight microchannel which is also shown in the Fig.1(a) and (g). The material of the all the microchannel heat sink is Aluminium (Al 6061). In this study, the heat flux considered is same and constant which is applied on the surface of the microchannel heat sinks. The geometry considered in the computational domain is subjected to uniform wall conditions and boundary conditions. Deionized water is the working (cooling) fluid which enters the microchannels at 25°C and exits at environmental pressure.



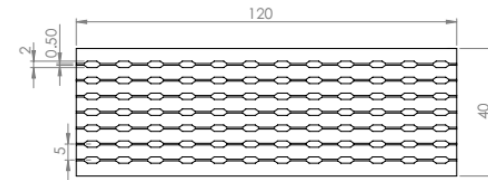
(a) 3000µm Straight microchannel heat sink



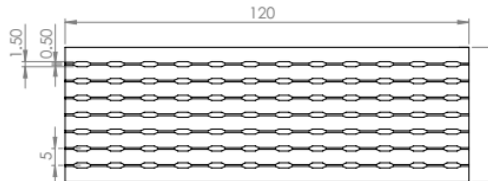
(b). 3000µm-500µm CD Microchannel heat sink



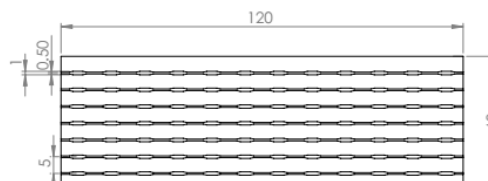
(c). 2500µm-500µm CD Microchannel heat sink



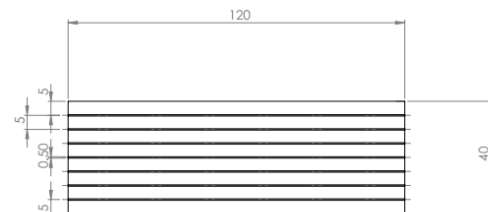
(d). 2000µm-500µm CD Microchannel heat sink



(e). 1500µm-500µm CD Microchannel heat sink



(f). 1000µm-500µm CD Microchannel heat sink



(g) 500µm Straight microchannel heat sink

Fig 1. Microchannel Heatsink considered in this study

The length, width and thickness of all the microchannel heat sink are 120mm, 40mm and 1.2mm respectively. The dimensions of the CD microchannel is to be optimized for higher heat dissipation. The optimization domain range for microchannel dimensions is $500\mu\text{m}$ (Minimum width) to $3000\mu\text{m}$ (Maximum width) with constant 51.34° converge-diverge angle. The height of the microchannel is adjusted to maintain constant hydraulic diameter ($D_h=0.44\text{mm}$). For the CD microchannel, the

maximum sectional variation is 3000 μ m-500 μ m as shown in Fig.1(b). In next design step, 2500 μ m-500 μ m is considered. Further, 2000 μ m-500 μ m, 1500 μ m-500 μ m and 1000 μ m-500 μ m are considered as shown in Fig 1(c-f). For the straight microchannel, 3000 μ m and 500 μ m are considered as shown in Fig. 1(a) and (g) respectively.

This study aims at finding the optimal design of the CD microchannel for which the overall heat distribution in the heat sink is high. Estimation of total thermal resistance of the microchannel heat sink provides the evident information regarding the performance of the proposed microchannel heat sink.

2.2 Mathematical modelling:

The heat flux supplied to the microchannel heat sink is carried away by the inlet coolant. Only a small fraction of heat supplied gets transmitted to the surrounding or environment through convection and radiation. A heat loss analysis was carried by Duryodhan et al to calculate the minimum percentage of heat lost other than the forced convection in the microchannels [16]. In this analysis, two steps were carried out. First, the proposed microchannel heat sink is simulated without the inlet water flow for a particular inlet heat flux on surface of the microchannel heat sink. Second, the microchannel is simulated with the inlet water flow to attain the same heat flux. The difference in the temperature on the surface of the heat sink indicates the heat loss by natural convection and radiation. In this study, the heat loss in assumed to be 12%.

The following are the equations to determine the heat carried away by the fluid in the microchannels:

Total heat in the tank is given by:

$$Q_{\text{tank}} = Q_{\text{in}} - (Q_{\text{channel}} + Q_{\text{loss}}) \quad (1)$$

Heat in the channel is calculated by:

$$Q_{\text{channel}} = m c_p (T_{\text{out}} - T_{\text{in}}) \quad (2)$$

The heat carried away by the channel is given by:

$$h_{\text{out}} = q_{\text{heat}} / (T_f - T_s) \quad (3)$$

The heat flux in the channel is given by:

$$q_{\text{heat}} = Q_{\text{channel}} / A_s \quad (4)$$

For the fluid flow in the microchannels, the following assumptions is made in this study:

- i. 3D, steady state, incompressible flow
- ii. Effect of gravity is in normal direction.
- iii. Thermophysical properties of the fluid is set as piecewise linear function and has not effect on temperature difference.
- iv. Viscous effect of the fluid is not considered.

To evaluate the thermal performance of the heat sink, calculation of total thermal resistance provides us clear understanding [19]. The total thermal resistance is given by:

$$R_{\text{th}} = (T_s - T_{\text{in}}) / Q_{\text{in}} \quad (5)$$

2.3 CFD simulation:

A three-dimensional steady state simulation were conducted using COMSOL Multiphysics 5.1. The computational domain was set as 0.45m x 0.45m x 0.45m. The following assumptions and boundary conditions are made for the simulation:

1. Fluid flow is laminar in the computational domain.
2. All surfaces were non-radiant surface.
3. Process of heat transfer was under steady state condition.
4. Inlet wall is “velocity inlet” at 25°C while the outlet wall is “environmental pressure”
5. Heat flux is constant which is 120 W/cm² and applied on the surface of the microchannel heat sink.
6. Material assumed were homogeneous and isotropic.

2.4 Conjugate Heat transfer Module:

The classical Navier stokes equations: continuity equation, momentum equation and energy equation are solved by solver in the software.

Mass conservation or continuity equation:

$$\nabla \cdot (\rho V) = 0 \quad (6)$$

Momentum equation:

$$\rho \frac{DV}{Dt} = -\nabla p + \mu \nabla^2 V + \rho g \quad (7)$$

Thermal energy conservation equation:

$$\rho C_p \frac{DT}{Dt} = \nabla \cdot (k \Delta T) + \frac{DP}{Dt}$$

For fluid:

$$\frac{\partial}{\partial x} (\rho u T) + \frac{\partial}{\partial y} (\rho v T) + \frac{\partial}{\partial z} (\rho w T) = \frac{k}{C_p} \left(\frac{\partial^2 T}{\partial x^2} + \frac{\partial^2 T}{\partial y^2} + \frac{\partial^2 T}{\partial z^2} \right) + S_e \quad (8)$$

where S_e indicates the heat source term for the radiative heat transfer which is not considered in this study.

For solid:

$$\left(\frac{\partial^2 T}{\partial x^2} + \frac{\partial^2 T}{\partial y^2} + \frac{\partial^2 T}{\partial z^2}\right) = 0 \quad (9)$$

2.5 Optimization Module:

Global least square objective function was used as the optimization criteria for which the list of temperature values were imported using tabulated function for parameter column. This study aims at finding the optimized width of the CD section and hence, parameter declared in the objective function was maximum and minimum width. Optimization solver was used to solve the objective function. BOBYQA method was selected as the solver.

Once the materials and boundary conditions are defined, the designed geometries are meshed using the free tetrahedral cells along with individual boundary layer mesh for solid-liquid interface. The total cells in the domains are 4,177,360 for which the deviation in results were less than 0.8%. Thus, mesh sensitivity is considered to be maximum for this study. Hence, the CFD model is considered to be highly meshed [20]. The residual error is set as 10^{-6} for maximum convergence of the solution. The change in velocity, heat flux, temperature (solid and fluid) and pressure drop can be manipulated using the post processor in the software.

3. Results and Discussion

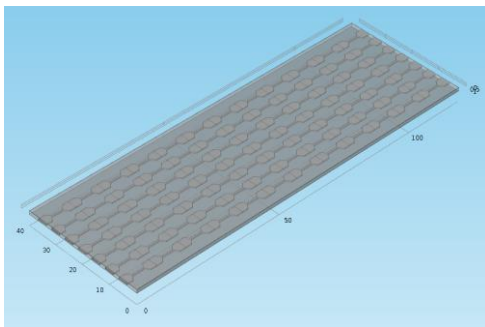


Fig. 2. CAD model of CD microchannel

Fig. 2 shows the imported CAD model of CD microchannel. This section explains the pressure drop and thermal resistance obtained from the numerical simulation.

From the optimization solver, it was found that the increase in width of the CD section in microchannel increases its thermal performance. 3000-500 μ m CD microchannel shows superior performance against the other microchannel considered in this study. Hence, results of 3000-500 μ m were discussed in further explanation.

3.1 Pressure Drop:

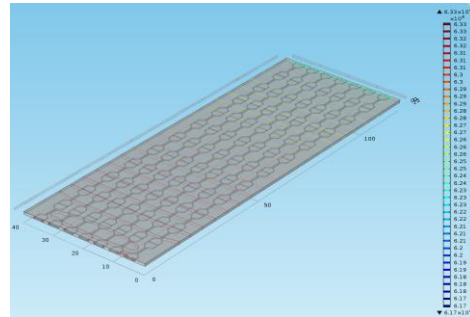


Fig 3. Pressure drop in CD microchannels

The pressure drop in the straight microchannels are constant in the microchannels. This is due to the streamline fluid flow for which the velocity is high. Fig.3 shows the variation in pressure drop across the CD microchannel heat sink. Unlike straight channels, the pressure drop is non-uniform due to varying cross-section in the microchannels. Fig.4 shows the detailed CD section of a single unit in the microchannel. L_2 and L_4 are the converging and diverging section respectively. The cross-sectional area varies in these two section whereas the area of L_1 , L_3 and L_5 are same due to uniform cross-section.

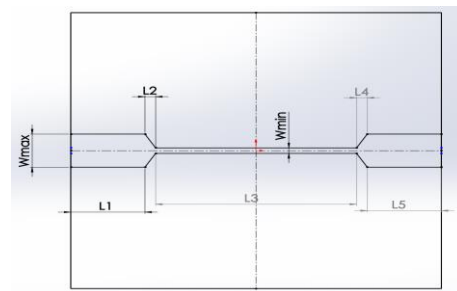


Fig.4 CD section in the microchannel

To be noted, the pressure varies across each unit of the CD section. This is because, the pressure drops when there is a change in velocity in the cross section. Fig.5 explains the phenomenon of pressure drop in the CD microchannels. In section L_1 , the working fluid enters at section with pressure P_1 . When the fluid enters section 2,

the change in cross section increases the velocity. When the fluid exits section 2, the velocity decreases on entering section 3 which increases the pressure. Therefore, the pressure variation occurs in each section of the CD microchannels. The viscous forces along the walls of solid-fluid boundary reduces due to formation of flow recirculation vortices as shown in Fig.5. At the exit furrows of the channels, there is symmetric vortices formed despite high mainstream velocity flow. These vortices induces the hot fluid from the main stream flow to mix and recirculate with the cold fluid [13]. Hence, the viscous forces on the walls reduces maintaining low pressure drop in the CD microchannel.

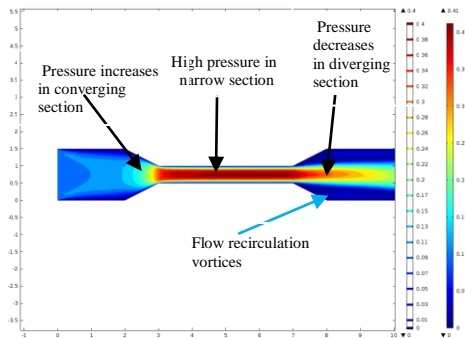


Fig.5 Velocity contour in CD section

Fig.6 shows the comparison of pressure drop in various microchannels under study. It is inferred that pressure drop in proposed CD microchannels is less than both the straight microchannel considered. This is affirmed by the velocity contour as explained earlier. It is important to note that the increase in width of CD microchannel decreases the pressure drop proportionately. To be noted, the vortices formation at the exit furrows of CD section is predominant for larger width of microchannel. At lower width, deterioration of vortices occurs due to increased flow velocity.

For 120 W/cm² heat input and flow velocity of 0.1 m/s, the pressure drop in 3000-500 μ m is 53% lower than in 3000 μ m straight microchannels. The pressure drop reduction is obvious for corresponding increase in maximum width of CD section except for 1000-500 μ m microchannel. The pressure drop in 1000-500 μ m microchannel is 7% be higher than 500 μ m straight microchannel. Hence, the maximum width of CD microchannel should be above

1000 μ m for supremacy in vortices formation and improvement in thermal performance.

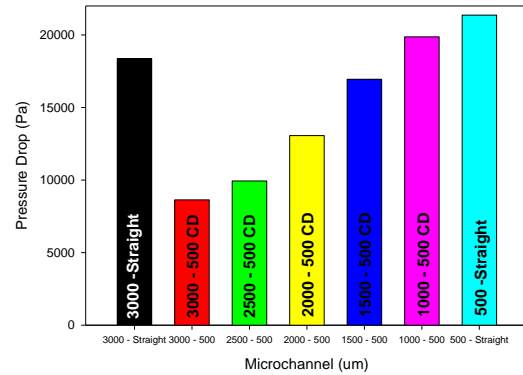


Fig. 6 Pressure drop in microchannels

3.2 Thermal resistance:

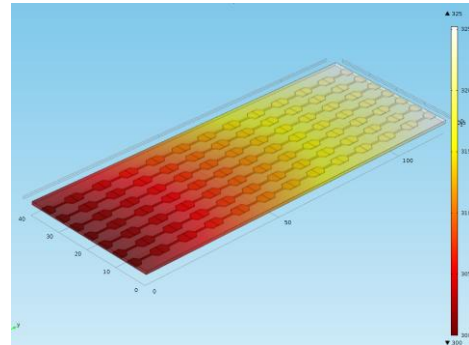


Fig 7. Temperature distribution in microchannels

Fig.7 shows the temperature distribution in the microchannel heat sinks under study. The reduction in maximum temperature of CD microchannel is due to enhancement of heat transfer coefficient of fluid flow. The increase in heat transfer can be correlated to the vortice formation as shown in Fig.5. The solid-fluid boundary in the channel wall is perturbed which enhances the heat transfer. It is reinstated that the flow recirculation from the vortices enhances the periodical mixing of hot fluid and cold fluid along the mainstream flow. The increase in width of CD microchannel increases the recirculation and reduces the overall surface temperature of the heat sink. However, nominal width should be selected in order to neglect the effects of nucleate boiling. The major advance in CD microchannels is that the converging-diverging section in the channels helps in mitigating the nucleation effects which was observed in wavy microchannels [5]. Limiting

the nucleation helps in increasing the heat transfer rate [13].

The phenomenon of increased heat transfer is due to increased heat transfer coefficient of the fluid flow in the CD microchannels. Primarily, the rate of heat transfer is high due to velocity rise in the converging channel. Secondly, the rate of heat transfer in the fluid-solid boundary of the microchannels channels is increased due to alternative pressure drop and pressure rise. Apart from these, the surface area of heat transfer is high and therefore, the rate of heat transfer is also high. Altogether, high heat transfer rate contributes for low thermal resistance which is the significant quality of an efficient microchannel heat sink.

The thermal resistance (R_{th}) of microchannel heat sink is calculated by eqn.(5). Fig. 8 shows the comparison of R_{th} . It can be observed that the total thermal resistance of the CD microchannel heat sink is lower than that of the straight microchannel heat sinks. In detail, for a constant inlet velocity of 0.1 m/s, the thermal resistance of CD microchannel heat sink is 74% lower than the 0.5mm straight microchannel heat sink and 65% lower than the 3mm straight microchannel heat sink. It should be noted that the total thermal resistance decreases with increase in width of CD section. This means that when the change in CD cross section, the pressure drop and thermal resistance decreases significantly. On comparison with the microchannel heat sink considered by Baodong Shao et al [19] of the similar size (120mm x 40mm x 1.2mm), the CD microchannel heat sink illustrates that the thermal resistance is 38% lesser. This adds value to the proposal of CD microchannel as an efficient one.

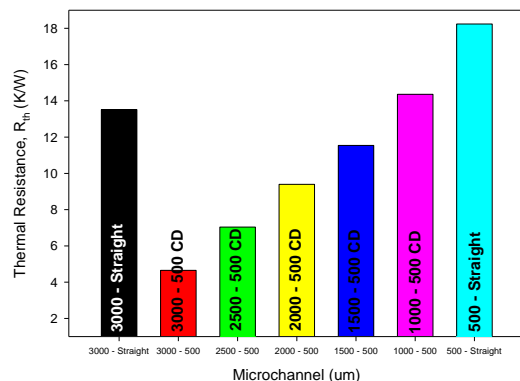


Fig. 8 Thermal resistance in microchannels

4. Conclusion

An innovative geometry of the microchannel is designed and simulated in this study. The influence of converging diverging microchannel on heat transfer is proven for superior thermal performance. The thermal resistance was found to be low for the microchannel dimension of 3000 μ m-500 μ m and this is considered as the optimal dimension for the CD microchannel heat sink. For the proposed microchannel, the pressure drop is calculated to be 53% lower than higher than the 3mm straight microchannel. Converging Diverging section in the CD microchannel prompts for non-nucleated flow and periodic velocity rise which contributes for increased heat transfer. This can be proven by reduction in thermal resistance of CD microchannel heat sink. R_{th} of the CD microchannel heat sink is 4.652 K/W. Hence the proposed CD microchannel heat sink illustrates superior thermal performance.

Acknowledgments

The author gratefully acknowledges Institute of Post-graduate Studies, Universiti Sains Malaysia for funding the research through USM Fellowship.

References

1. D.B. Tuckerman, F.W. Pease, "High-performance heat sinking for VLSI", *IEEE Electron Device Lett.* EDL-2 (1981) 126–129.
2. S.G. Singh, A. Agrawal, S.P. Duttgupta, "Reliable MOSFET operation using twophase microfluidics in the presence of high heat flux transients", *J. Micromech. Microeng.*, 21 (2011) 105002.
3. S.G. Singh, S.P. Duttgupta, A. Agrawal, "In-situ impact analysis of very high heat flux transients on non-linear p-n diode characteristics and mitigation using on-chip single-phase and two-phase microfluidics", *J. Microelectromech. Syst.* 18 (6) (2009) 1208–1219.
4. N.R. Rosaguti, D.F. Fletcher, B.S. Haynes, "Laminar flow and heat transfer in a periodic serpentine channel with semi circular cross section", *Int. J. Heat Mass Transfer*, 49 (2006) 2912–2923.

5. Y. Sui, C.J. Teo, P.S. Lee, Y.T. Chew, C. Shu, "Fluid flow and heat transfer in wavy microchannels", *Int. J. Heat Mass Transfer*, 53 (2010) 2760–2772.
6. P.C. Lee, C. Pan, "Boiling heat transfer and two-phase flow of water in a single shallow microchannel with a uniform or diverging cross section", *J. Micromech. Microeng.*, 18 (2008) 025005.
7. Agrawal, V.S. Duryodhan, S.G. Singh, "Pressure drop measurements with boiling in diverging microchannel", *Front. Heat Mass Transfer* 3 (2012) 013005.
8. Kosar, Y. Peles, "Thermal-hydraulic performance of MEMS-based pin fin heat sink", *J. Heat Transfer*, 128 (2006) 121–131.
9. Y. Peles, A. Kosar, C. Mishra, C-J. Kuo, B. Schneider, Forced convective heat transfer across a pin fin micro heat sink, *Int. J. Heat Mass Transfer*, 48 (2005) 3615–3627.
10. M.M. Rahman, Measurements of heat transfer in microchannel heat sinks, *Int. Commun. Heat Mass Transfer*, 27 (2000) 495–506.
11. H.Y. Wu, P. Cheng, An experimental study of convective heat transfer in silicon microchannels with different surface conditions, *Int. J. Heat Mass Transfer*, 46 (2003) 2547–2556.
12. N.R. Rosaguti, D.F. Fletcher, B.S. Haynes, Laminar flow and heat transfer in a periodic serpentine channel with semi-circular cross section, *Int. J. Heat Mass Transfer*, 49 (2006) 2912–2923.
13. Gong, Liang, Krishna Kota, Wenquan Tao, and Yogendra Joshi. "Parametric numerical study of flow and heat transfer in microchannels with wavy walls." *Journal of Heat Transfer* 133, no. 5 (2011): 051702.
14. Xuan, Xiangchun, and Dongqing Li. "Particle motions in low-Reynolds number pressure-driven flows through converging–diverging microchannels." *Journal of Micromechanics and Microengineering*, 16, no. 1 (2005): 62.
15. W.F. Louissos, D.L. Hitt, Heat transfer and viscous effects in 2D and 3D micronozzles, in: *37th AIAA Fluid Dynamics Conference*, June 25–28, 2007
16. V.S. Duryodhan, Abhimanyu Singh, S.G. Singh, Amit Agrawal, "Convective heat transfer in diverging and converging microchannels", *International Journal of Heat and Mass Transfer*, 80 (2015) 424–438.
17. Duryodhan, V. S., S. G. Singh, and Amit Agrawal. "Liquid flow through a diverging microchannel." *Microfluidics and nanofluidics* 14, no. 1-2 (2013): 53-67.
18. Duryodhan, V. S., Abhimanyu Singh, Shiv Govind Singh, and Amit Agrawal. "A simple and novel way of maintaining constant wall temperature in microdevices." *Scientific reports* 6 (2016).
19. Baodong Shao, Lifeng Wang and Jianyun Li, "Application of thermal resistance network model in optimization design of micro-channel cooling heat sink", *International Journal of Numerical Methods for Heat & Fluid Flow* Vol. 19 Nos. 3/4, 2009 pp. 535-545.
20. Raypah, Muna E., M. K. Dheepan, Mutharasu Devarajan, and Fauziah Sulaiman. "Investigation on thermal characterization of low power SMD LED mounted on different substrate packages." *Applied Thermal Engineering*, 101 (2016): 19-29.