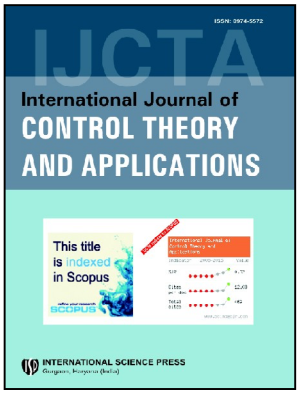


International Journal of Control Theory and Applications

ISSN : 0974-5572

Volume 10 • Number 11 • 2017

Experimental Study on a Concentric shaped Heat Sink with Alternate Flow Passages and its Comparison with CFD results of Straight Channel Heat Sink

Ravindra D. Jilte[#1] and M. Udayabhanukiran[2]

[#1]Department of Mechanical Engineering, Lovely Professional University, Phagwara, Punjab 144411, India
E-mail: rdjilte@gmail.com
[2]Department of Mechanical, KL University, Green Fields, Vaddeswaram, Guntur-522502, A.P, India
E-mail: mukkamalabhanukiran27@gmail.com

Abstract: The present study focuses on improved heat removal from a novel heat sink geometry as required for effective cooling of the devices to make them more efficient and capable in maintaining long life. The heat sink is manufactured by CNC machining, it comprises of four channels (width 4mm and depth 3.5mm) configured in concentric shape with alternate passage (slot of 3mm gap) for the fluid flow. Experiments on a concentric channel heat sink are conducted under constant heat dissipation rate with a mass flow rate ranging from 37.5 ml/min-181.8 ml/min. The experimental results of proposed heat sink is also compared with numerical results of straight channel heat sink of same hydraulic diameter and convective heat transfer area. Results include the comparison of thermal performance of two heat sink geometries under same heating conditions. in terms of thermal heat transfer rate, Nusselt number and pressure drop. The bifurcated motion introduced in the concentric shaped channel heat sink resulted in better momentum and heat transfer. The heat removal rate and heat transfer coefficient of proposed novel geometry is found to be about 12 % and 37 % higher compared to straight channel heat sink. Pressure drop for both the geometries are found to be negligible in the selected range of flow rate.

Keywords: Concentric channel heat sink, straight channel heat sink, water cooling, Heat transfer, Nusselt number, Pressure drop

I. INTRODUCTION

With increase in miniaturization of electronic devices conventional cooling methods such as air cooling is unable to meet the increasing demand in high heat removal rate. There is a requirement for properly designed effective cooling system for the devices to make them compact, more efficient, and capable in maintaining long life. For this purpose, use of heatsinks with micro/mini channels are found to be alternate solution.

The micro and mini channels are flow channels with air or liquid as working fluids. The possibility of mini and microchannels in heat sinks for high heat flux removal application was first proposed by [1]. The main reasons to focus on mini and microchannel based devices for various applications are due to decreased hydraulic diameter in micro channels the heat transfer coefficient increases thus provides tremendous cooling mechanism. Small channels when used with suitable fluids provides high heat transfer rates. Due to small size the components occupy less space, weight and easy to install. Ease of implementation and reduced material cost.

Many investigations have been reported on hydraulic and thermal behavior as well as heat transfer augmentations. Use of heat transfer augmented techniques in micro/mini channels for single phase cooling devices was reviewed by [2]. [3] also investigated on water cooled offset strip fin enhanced microchannel heat exchanger and found it has better performance compared to straight continuous channel walls. [4] carried numerical study on laminar flow and forced convective heat transfer in water cooled rectangular shaped microchannels sections having specific hydraulic diameters and distinct geometric configurations. Aspect ratio and hydraulic diameter affect the heat transfer rate of microchannels.

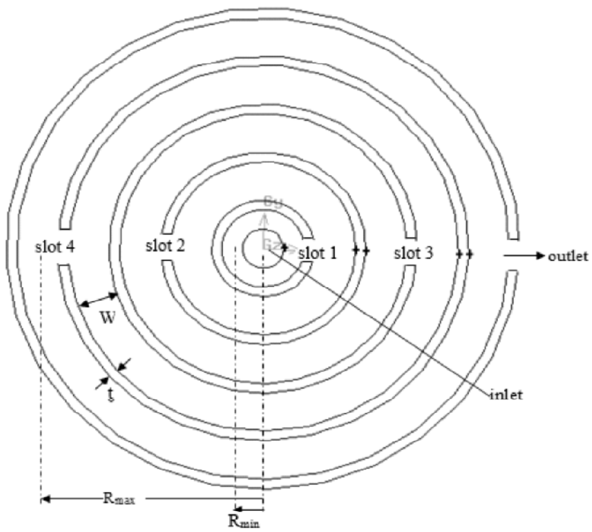
The use of nanofluids in mini/microchannels have been reported the literature. [5] numerically investigated on nanofluid (Al_2O_3–water) in horizontal microchannel. It was found that use of nano-particles in microchannel improves the heat transfer performance; Nusselt number and friction factor decreases due to viscous dissipation.

[6] analyzed the performance of microchannel heat exchanger and presented results considering the different shape and size of channels. [7] presented numerical study on mini channel heatsink subjected to constant heat flux wall condition. It was noticed that in narrow and deep channels the heat transfer performance was improved with relatively high-pressure drop. [8] reported that in mini-fin structures the convective heat transfer coefficient for water increased from 9-21 fold and for air from 12-38 fold compared to the empty plate channel. [9] conducted numerical study on grooved microchannel heatsinks to analyze the effect of geometrical specifications on laminar convective heat transfer. Trapezoidal grooved microchannel heatsinks (MCHS) has the optimum thermal design compared to rectangular and triangular grooved MCHS. [10] conducted numerical analysis on microchannel water block with pass variations. Heat transfer rate and accompanied with higher values of pressure drop was observed in 2-pass samples. [11] experimentally investigated the influence of fin spacing in different heat sinks for effective thermal management. [12] numerically analyzed the fluid flow and thermal characteristics of mini-channel heatsinks with non-uniform inlets. The total thermal resistance of mini-channel heat sink was found to be reduced from 9.9-13.1% using non-uniform baffles. [13] experimentally studied the cooling performance of sinusoidal wavy mini-channel heat sink and examined the effect of geometrical parameters and working fluids and observed effective thermal performance compared to straight mini channel heatsink. [14] suggested that that a continuum-based approach can be applied to estimate the heat transfer in microchannels. The use of serpentine microchannel (with square cross section) to characterize slug flow behavior has been reported by [15].

Based on the aforementioned studies, it is observed that most of the investigations carried out on thermal performance in a circular or rectangular straight mini-channel heat sinks and only few experimental or numerical studies were conducted on a spiral or concentric channels. In this work, thermal performance of concentric channel heat sink with alternate slots for the fluid flow is presented.

II. EXPERIMENTAL STUDY ON CONCENTRIC CHANNEL HEAT SINK (CCHS)

The straight channels are replaced with concentric channel with alternate slots. The inlet pipe is circular whereas outlet pipe is rectangular cross section. The water from the inlet pipe enters centrally and flow bifurcates after passing through the first passage (slot of 3mm width). It then flows through the first channel again bifurcates after passing through the second slot. (Figure 1). CCHS is made of copper material and was manufactured by CNC machining. Top of the heat sink was sealed by copper plate with a inlet pipe for the flow water through channel.



Parameters	Size
Total height of heat sink	5.5 mm
Thickness of heat sink base plate	2 mm
Total Diameter of heat sink	50 mm
Height of channel (H)	3.5 mm
Width of channel (W)	4 mm
Thickness of channel wall (t)	1 mm
Flow pasage slot for the water to flow	3 mm
Diameter of the water inlet pipe	5 mm
Dimensions of the water outet duct	3 × 3.5 mm
Width of the slots (passage)	3 mm

Figure 1: Concentric mini channel with alternate slots

Heating block is designed to supply the heat flux required to conduct the experiment. It comprised of tin sheet, calcium silicate board, ceramic fiber wool, nichrome wire and copper plate (Figure 2). Ceramic fiber wool and calcium silicate boards are used due to their better insulation properties. Nichrome wire is used as heating element.

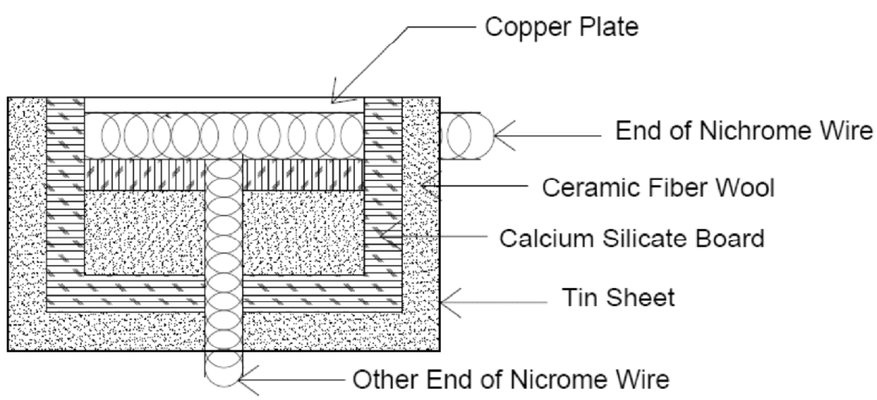


Figure 2. Cross-sectional view of heating block

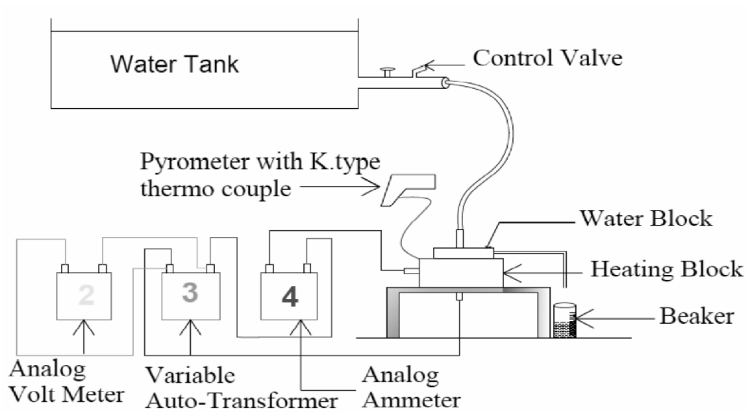


Figure 3. Line diagram of experimental set-up

The line diagram of the experimental set-up is shown in the Figure 3. It consists of a storage tank, a control valve, a variable auto-transformer, digital ammeter, digital voltmeter, pyrometer with k-type thermocouple, digital thermometer, timer, beaker and the test module.

The power supplied from variac-auto transformer to the heating block is adjusted and maintained at constant value. It is calculated as

$$P = VI \tag{1}$$

where V and I are voltage and current measured with digital voltmeter and ammeter. A small grooves were made on the top of copper plate of heating block in order to assemble the thermocouples with an accuracy of ± 1.5% of reading which is connected to pyrometer to measure the surface temperature of heating block. Flow rate was adjusted by control valve. Inlet and outlet temperatures of water are measured by digital thermometer with an accuracy of 1.5% ± 2^{0}C. During the experiment constant input power of 50 W is supplied to the heating block. Base temperature of heating block is recorded after attainment of steady state condition. Experiments are conducted at different flow rates. A set of three trials are conducted to check the repeatability of the experimental data.

III. NUMERICAL ANALYSIS OF STRAIGHT CHANNEL HEAT SINK (SCHS)

The performance study on concentric shaped heat sink is further extended and compared with conventional straight channel heat sink. CFD Software 'Fluent [16] was used to perform three-dimensional numerical simulation of SCHS. The 3-dimensional model (Figure 4) is generated using Gambit tool of CFD software Fluent.

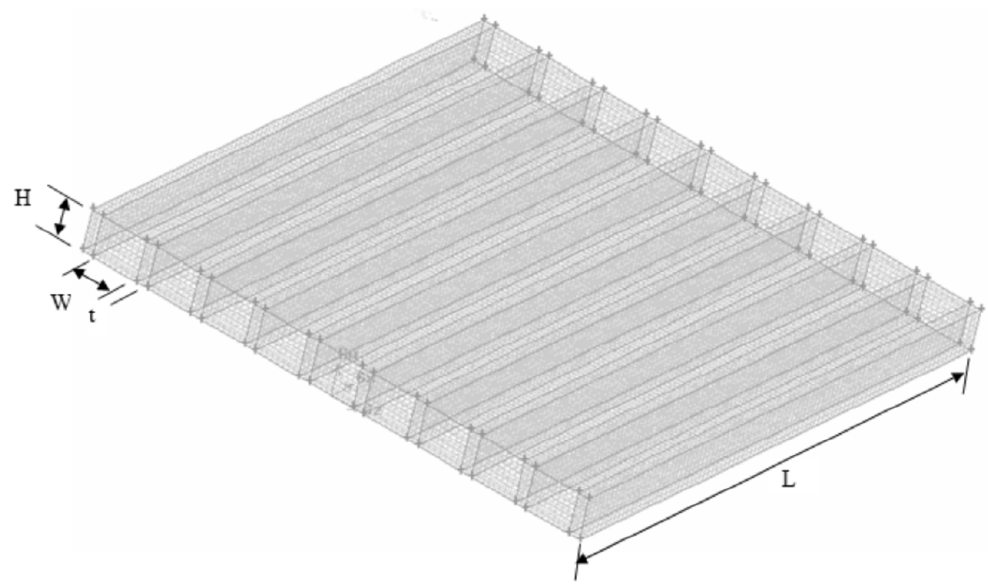


Figure 4: Three-Dimensional geometry of SCHS modeled in GAMBIT

The dimensions of straight channel heat sink are chosen to maintain the same hydraulic diameter (D_h) and convective heat transfer area (A_{conv}) as that of proposed geometry.

The hydraulic diameter for non-circular flow passage is given by

$$D_h = \frac{4Ac}{P} = \frac{4HW}{2(H+W)} = \frac{2HW}{H+W} = 3.733 \text{ mm}$$

where, H is the depth of the cavity and W is width of the cavity (Figure 4).

Numerical scheme adapted in the present study are summarized as below: steady state laminar model is considered wherein the continuity, momentum, and energy equations are used to get the solution. The semi-implicit pressure linked equation (SIMPLE) scheme of the Fluent software is used. The SIMPLE algorithm consists of a pressure- velocity coupling and is used for steady state simulations. The momentum and energy solutions controls are second order upwind type. The convergence criteria for residuals of continuity and velocity equations were set with a value of 10^{-6} and 10^{-9} for energy equation .The results are obtained once the solutions are converged.

IV. RESULTS AND DISCUSSION

The thermal performance of both the geometries are presented in the following section. The mass flow rates ranging from 37.5 – 181.8 ml/min was considered during the experimentation as well as numerical simulation.

(A) Comparison of observed temperatures and heat transfer rates

In concentric channel heat sink, constant mass flow rate is maintained throughout its passage from inlet to outlet while in case of straight channel heat sink mass flow rate is supplied through the header and assumed to be equally distributed among 9 channels. The area averaged surface temperature and water outlet temperature for straight channel heat sink are obtained numerically for a constant heat flux. These are compared with measured temperature of concentric channel heat sink for different mass flow rates as shown in Figure 5. It is observed that both the channel wall temperature and water outlet temperature reduced for higher mass flow rates. The flow rate in each straight rectangular channel is low, consequently, the surface temperature and water outlet temperature are found be higher.

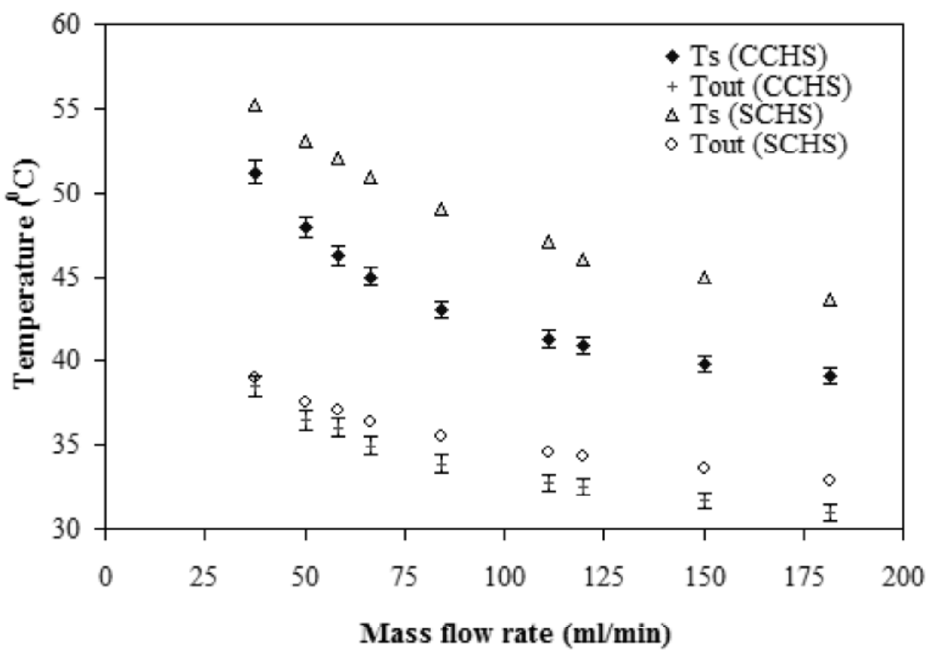


Figure 5: Surface and fluid temperatures for concentric and straight channel heat sinks

The heat transfer rate of flowing fluid (Q_w) in case of concentric channel heat sink is calculated by eq.(2) whereas eq. (3) is used for straight channel heat sink.

$$Q_{w,CCHS} = m_{tot} c_{pw} \left(T_{out} - T_{in}\right)_{\mathrm{CCHS}} \tag{2}$$

where m_{tot} is the mass flow rate of water in kg/s, c_{pw} is the specific heat of water in J/kgK, T_{wi} *and* T_{oi} are inlet and outlet temperatures of water.

$$Q_{w,SCHS} = \sum_{i=1}^{i=n} m_i c_p \left(T_{out} - T_{in}\right)_{channel,i} \tag{3}$$

where $m_i = m_{tot}/n$, n = number of rectangular channels.

Estimated results of heat transfer rate for different values of mass flow rate is plotted in Figure 6. The bifurcated motion introduced in the concentric shaped channel heat sink resulted in better momentum and heat transfer. As expected from Eq. 2, heat removal rate increased with mass flow rate. The heat removal rate for concentric channels is about 2.2-11.9 % higher.

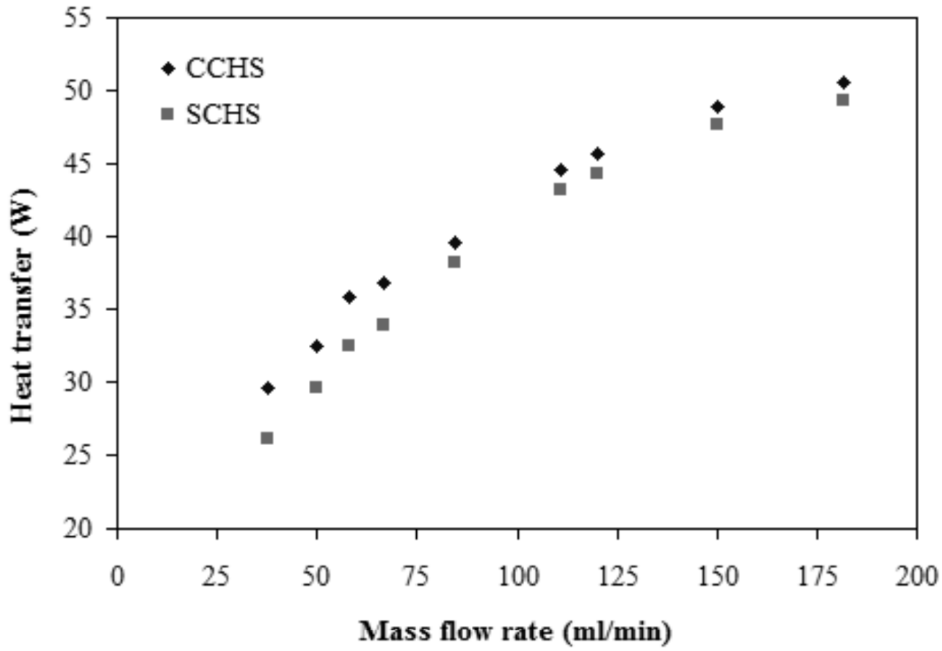


Figure 6: **Comparison of heat transfer rate of concentric and straight channel heat sinks**

(B) Comparison of Nusselt number

The heat transfer coefficient is calculated based on Eq. 4.

$$h = \frac{Q_w}{A_{s,3*}} \left(\frac{1}{T_s - \frac{T_{in} + T_{out}}{2}} \right) \tag{4}$$

where Q_w is heat transfer rate calculated based on Eqn 2 and Eqn 3 respectively for concentric and straight channel heat sink. $A_{s,3*}$ = area of the channel wall which participates in convective heat transfer. In both the geometries, three walls of the channels (one bottom and two vertical walls) are heated surfaces.

For straight channel heat sink, $A_{s,3*} = n\times(2H+W) = 4158$ mm^2, for CCHS, As,3* = 4066 mm^2

Nusselt number is defined as

$$N_u = \frac{hD_h}{k} \tag{5}$$

Heat transfer coefficient calculated for both the geometries showed it is function of flow and surface geometry of the channel. The heat transfer coefficient of proposed geometry is 32-37.56% higher compared to straight channel heat sink.

Reynolds number (R_e) is defined as

$$R_e = \frac{\rho U D_h}{\mu} = \frac{\dot{m} D_h}{\mu A_c} \tag{6}$$

where ρ is density, U is velocity and μ is dynamic viscosity and thermal conductivity of water. The properties of water are evaluated at bulk temperature $(T_{in}+T_{out})/2$.

Nusselt number for both geometries are compared for different Reynolds number as shown in Figure 7. Nusselt number for concentric shaped channel heat sink is ~25-32% higher compared to straight channel.

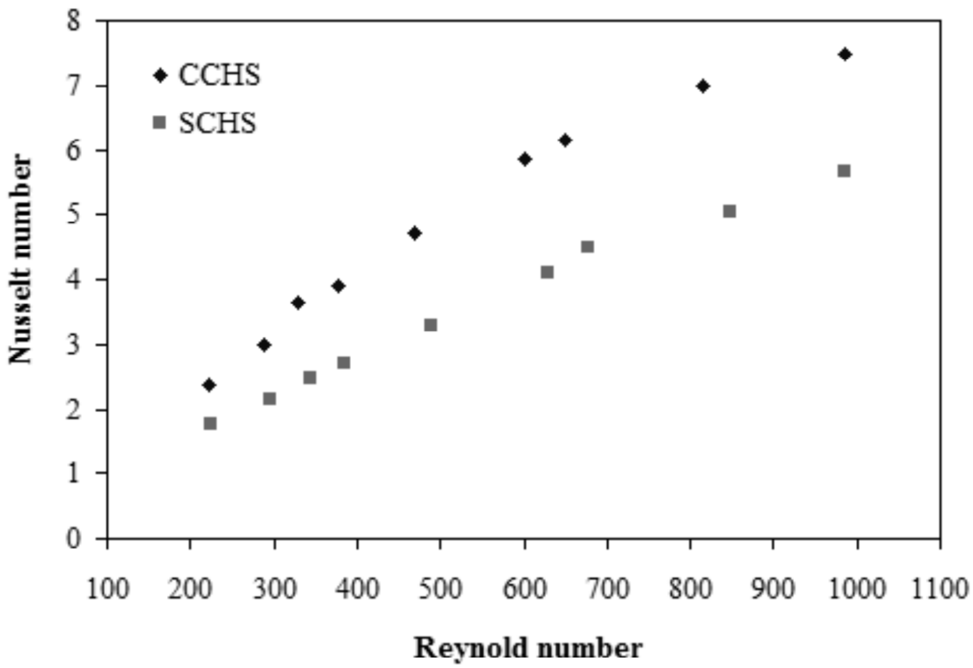


Figure 7: Nusselt number of concentric and straight channel heat sinks with Reynolds number

(C) Comparison of Pressure drops

The hydrodynamic (L_h) and thermal entry length(L_t) are calculated using Eq. 7

$$L_h = 0.05 \,\mathrm{Re}\, D_h \tag{7}$$

$$L_t = 0.1 \,\mathrm{Re}\,\mathrm{Pr}\, D_h \tag{8}$$

Pressure drop occurs when a fluid flows through a small channel. So the system requires pumping power to overcome the pressure drop. Pressure drop in rectangular channels [17] is given by Eq. 9 as

$$\Delta p_{SCHS} = \frac{2(f\ \mathrm{Re})\mu UL}{D_h^2} + K(\infty)\frac{\rho U^2}{2} \tag{9}$$

where K(∞) is the Hagenbach factor.

$$K(\infty) = 0.6796 + 1.2197\alpha c + 3.308\alpha_c^2 - 9.592\alpha_c^3 + 8.9089\alpha_c^4 - 2.9959\alpha_c^5 \tag{10}$$

[18] provided the following equation for a rectangular channel.

$$f\ \mathrm{Re} = 24\left(1 - 1.3553\alpha_c + 1.9467\alpha_c^2 - 1.7012\alpha_c^3 + 0.9564\alpha_c^4 - 0.2537\alpha_c^5\right) \tag{11}$$

Aspect ratio (±) is given by, $\alpha = \frac{H}{W} = 0.875$

Pressure drop relation available for spiral coil [19, 20] is modified for concentric shaped mini channels and given by

$$\frac{\Delta p_{CCHS}}{2\rho V^2}\left[\frac{WD_h^{1/2}}{R_{\max}^{3/4}(R_{\max} - R_{\min})^{3/4}}\right] = 49\left(\frac{\rho V D_h}{\mu}\right)^{-0.67} \tag{12}$$

where R_{max} and R_{min} is the maximum and minimum radius of the CCHS respectively.

The pressure drop is calculated for both the geometries and plotted in Figure 8 The pressure drop increased for higher flow rates. The pressure drop is higher (almost 100 times more) in concentric shaped channel heat sink but in both the geometries its value is negligibly small.

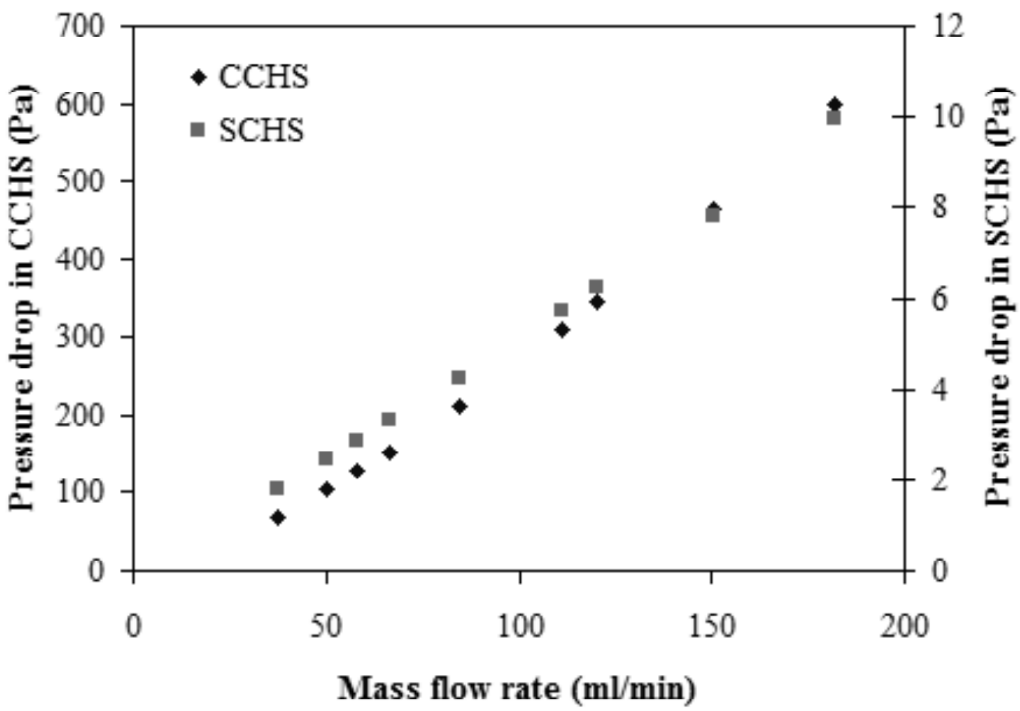


Figure 8: Pressure drop of concentric and straight channel heat sinks with mass flow rate

V. CONCLUSION

In the present work the thermal performance of water-cooled concentric shaped channel heat sink have been studied. The study was extended to compare its performance with conventional straight channel heat sink. The proposed geometry (concentric shaped heat sink) showed better thermal performance as compared to straight channel heat sink. The heat removal rate is about 12% higher as compared to conventional straight channel heat sink. The increase of heat transfer rate is accompanied with a penalty on high value of pressure drop, friction factor, and pumping power as compared to straight channel heat sink. Since in the operating flow rates the magnitude of pressure drop and pumping power both are found to be negligible, the proposed geometry is more preferable in applications where less circulation of cooling medium is required.

REFERENCES

[1] Tuckerman D B, Pease R F. "High-performance heat sinking for VLSI". IEEE Electron Device Letters pp. 126–129, 1981.

[2] Steinke M E, Kandlikar S G. "Review of single-phase heat transfer enhancement techniques for application in micro-channels, mini-channels and microdevices". Heat and Technology vol. 22(2), pp. 3-11. 2004.

[3] Steinke M E, Kandlikar S G. "Single-phase liquid heat transfer in plain and enhanced Micro-channels". Proceedings of ICNMM2006, Fourth International Conference on Nanochannels, Microchannels and Minichannels, Limerick, Ireland. June 19-21, 2006.

[4] Khameneh P M, Mirzaie I, Pourmahmoud N, Majidyfar S. "A numerical study of single-phase forced convective heat transfer with flow friction in microchannels". International Journal of Engineering Transactions A: Basics vol. 25(1), pp. 79-87, 2012.

[5] Ramiar A, Ranjbar A A. "The effects of viscous dissipation and variable properties on nano fluids flow in two dimensional micro channels". International Journal Engineering Transactions A: Basics vol. 24(2), pp. 131-142, 2011.

[6] Hasan M I, Rageb A A, Yaghoubi M, Homayoni H. "Influence of channel geometry on the performance of counter flow microchannel heat exchanger". International Journal of Thermal Sciences vol. 48, pp. 1607–1618, 2009.

[7] Xie X L, Liu Z J, He Y L, Tao W Q. "Numerical study of laminar heat transfer and pressure drop characteristics in a water-cooled mini channel heat sink". Applied Thermal Engineering vol. 29, pp. 64–74, 2009.

[8] Jiang P, Ruina Xu. "Heat transfer and pressure drop characteristics of mini-fin structures". International Journal of Heat and Fluid Flow vol. 28, pp. 1167–1177, 2007.

[9] Ahmed H E, Ahmed M I. "Optimum thermal design of triangular, trapezoidal and rectangular grooved microchannel heat sinks". International Communications in Heat and Mass Transfer vol. 66, pp. 47–57, 2015.

[10] Choi, Jin Tae, Kwon, Oh Kyung. "A numerical study of the heat transfer and fluid flow micro-channeled water block for computer CPU cooling". Journal of Mechanical Science and Technology vol. 25 (10). pp. 2657-2663, 2011.

[11] Saad Ayub Jajja, Wajahat Ali,Hafiz Muhammad Ali, Aysha Maryam Ali. "Water cooled mini-channel heat sinks for microprocessor cooling: Effect of fin spacing". Applied Thermal Engineering vol. 64, pp. 76-82, 2014.

[12] Liu, Jianlin. "Numerical study on performances of mini-channel heat sinks within-uniform inlets". Applied Thermal Engineering. 2016; vol. 9, pp. 856–864.

[13] Aliabadi Khoshvaght-Aliabadi, Sahamiyan M, Hesampour M, Sartipzadeh, O. "Experimental study on cooling performance of sinusoidal–wavy minichannel heat sink". Applied Thermal Engineering vol. 9, pp. 50–61, 2016.

[14] Poh-Seng Lee, Suresh Garimella , Dong Liu. "Investigation of heat transfer in rectangular microchannels". International Journal of Heat and Mass Transfer vol. 48, pp. 1688–1704, 2008.

[15] Fabiana Cairone, Salvina Gagliano, Maide Bucolo. "Experimental study on the slug flow in a serpentine microchannel" . Experimental Thermal and Fluid Science vol. 76, pp. 34–44, 2016.

[16] Fluent 6.3.26 User Guide. 2006.

[17] Steinke M E, Kandlika S G. "Single-phase liquid friction factors in microchannels". ASME, Third International Conference on Microchannels and Minichannels, Toronto, Canada, June 13–15, 2005.

[18] Ali S, Seshadri C. Pressure Drop in Archimedean Spiral Tubes, Industrial and Engineering Chemistry Process Design and Development vol. 10 (3), pp. 328-332, 1971.

[19] Djordjevic M L, Velimir P, and Macic M V. Pressure drop and stability of flow in Archimedean spiral tube with transverse corrugations. Thermal Science vol. 20(2), pp. 579-591, 2016.

[20] Fan Y, Lee P S, Li-Wen Jin, Zhang, Ding-Cai. A parametric investigation of heat transfer and friction characteristics in cylindrical oblique fin mini channel heat sink. International Journal of Heat and Mass Transfer vol. 68, pp. 567–584, 2014.