# DESIGN OF STACKED MICROCHANNEL HEAT SINKS USING COMPUTATIONAL FLUID DYNAMIC

Bella Jaya Chandana[1] and V.Vijaya Kumar[2]
[1]Department of Mechanical Engineering, ASRCE, Tanuku, JNTU Kakinada, India
[2] Assistant Professor, ASRCE, Tanuku, India

***ABSTRACT***
*The heat extraction problem is becoming an important factor in the development of micro-electronics. Over the last decade, micromachining technology has been increasingly used for the development of highly efficient cooling devices called heat sink because of its undeniable advantages such as less coolant demands and small dimensions. One of the most important micromachining technologies is micro channels. Hence, the study of fluid flow and heat transfer in micro channels which are two essential parts of such devices, have attracted more attentions with broad applications in both engineering and medical problems. Heat sinks are classified into single-phase or two-phase according to whether boiling of liquid occurs inside the micro channels*
*The microchannel heat sink is designed for electronic chips that could be effectively cooled by means of forced convection, by water flowing through the microchannels. This project presents a Computational Fluid Dynamics (CFD) flow simulation modeling, where water is made to flow across different cross-sectional microchannel heat sinks placed on heat source. The flow simulation of water and convective heat transfer are carried by using commercial software "Solid Works Flow Simulation". In the CFD process stacked microchannels of various cross-sections viz., rectangular, triangular, pentagonal and circular are designed, then by defining the computational domain, boundary conditions as inlet mass flow (1e-5 kg/s),outlet pressure ( static pressure) and heat flux(750w/cm2) and by considering the parallel and counter flow direction of water in different cross-sectional microchannels the temperature drop in heat sink, pressure drop across the channels and amount of volumetric heat transfer coefficient are analyzed and compared.*

***KEYWORDS:*** *Heat extraction, micromachining, stacked microchannel, computational fluid dynamics.*

## I. INTRODUCTION

Among the novel methods for thermal management of the high heat fluxes found in microelectronic devices, microchannels are the most effective at heat removal. The possibility of integrating microchannels directly in to the heat generating substrates makes them particularly attractive. The two important objectives in electronics cooling, minimization of the maximum substrate temperature and reduction of substrate temperature gradients can be achieved by the use of microchannels.

A large number of recent investigations have undertaken to study the fundamentals of microchannel flow as well as to compare the flow and heat transfer characteristics of microchannels with conventional channels. A comprehensive review of these investigations conducted over the past decade is presented in this chapter.

Studies on microchannel flows in the past decade are categorized in to various topics such as temperature, heat transfer in microchannels, Nusselt number, heat flux, comparison with flow in conventional channels, investigation of single phase and two-phase flows in microchannels, minichannels and small tubes, gas flow in microchannels, analytical studies on microchannel flows and design and testing of microchannel heat sinks for electronics cooling.

This chapter can be broadly classified under three categories. The first part of the survey deals with analytical studies. The second part of the survey deals with the numerical studies and the third part of the survey deals with the experimental studies.

# II. LITERATURE REVIEW

Non-uniform Temperature Distribution in Electronic Devices Cooled by Flow in Parallel Microchannels, was discussed by Hetsroni et al. [1]. Two-Phase Flow Patterns in Parallel Microchannels was studied by Hetsroni et al. [2]. They analyzed the effect of geometry on flow and heat transfer, finding that an increasingly uniform heat flux resulted in an increased irregularity of temperature distribution on the chip surface.

Baud [3] conducted an optimization study to minimize the temperature gradient and the maximum temperature for microchannel heat sink. It was demonstrated that further reduction in maximum temperature and temperature gradient could be achieved by varying the cross-sectional dimensions of the microchannel. The penalty of this method is the dramatically increased pressure drop due to the acceleration along the flow direction.

Zhao and Lu [4] presented an analytical and numerical study on the heat transfer characteristics of forced convection across a microchannel heat sink. Two analytical approaches are used by them: the porous medium model and the fin approach. In the porous medium approach, the modified Darcy equation for the fluid and the two-equation model for heat transfer between the solid and fluid phases are employed. Firstly, the effects of channel aspect ratio and effective thermal conductivity ratio on the overall Nusselt number of the heat sink are studied in detail. The predictions from the two approaches both show that the overall Nusselt number increases as aspect ratio increased and decreases with increasing thermal conductivity.

The effect of porosity on the thermal performance of the microchannel was also examined by them. They found that, whereas the porous medium model predicts the existence of an optimal porosity for the microchannel heat sink, the fin approach predicts that the heat transfer capability of the heat sink increases monotonically with the porosity.

They also studied the effect of turbulent heat transfer within the microchannel, and they found that turbulent heat transfer results in a decreased optimal porosity in comparison with that for the laminar flow. They proposed a new concept of microchannel cooling in combination with micro heat pipes, and the enhancement in heat transfer due to the heat pipes is estimated. Finally, they conducted two-dimensional numerical calculations for both constant heat flux and constant wall temperature conditions to check the accuracy of analytical solutions and to examine the effect of different boundary conditions on the overall heat transfer.

# III. DESCRIPTION OF THE COMPUTATIONAL DOMAIN

## 3.1 Parallel flow Computational domain

A single stacked rectangular microchannel having the dimensions of 18μm × 6μm, that represents the channel height *h* and width *w*. The bottom surface of heat sink at $y = 0$ is uniformly heated with a constant heat flux of 750 W/cm$^2$ and at the top surface at $y = H$ is well insulated and also the adiabatic conditions are applied at the other boundaries. Fluid flowing through the channel is at temperature 20ºC on account of static pressure. The direction of the fluid flow is parallel to the *z*-axis as shown in Fig.1. The flow is assumed to be laminar and both hydro-dynamically and thermally fully developed. Also the thermo physical properties are assumed to be constant. The flowing fluid in the microchannels is considered as water.

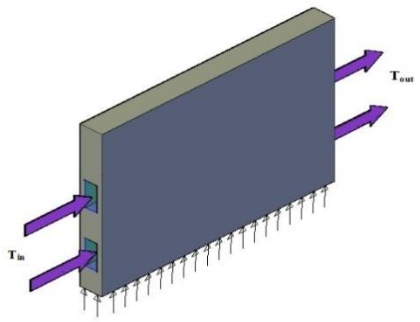


**Fig.1:** Computational domain of parallel flow single stacked rectangular microchannel heat sink

### 3.2 Counter flow Computational domain

A single stacked rectangular microchannel having the dimensions of 18µm × 6µm, that represents the channel height $h$ and width $w$. The bottom surface of heat sink at $y = 0$ is uniformly heated with a constant heat flux of 750 W/cm$^2$ and at the top surface at $y = H$ is well insulated and also the adiabatic conditions are applied at the other boundaries. Fluid flowing through the channel is at temperature 20ºC on account of static pressure. The direction of the fluid flow is in opposite direction along the $z$-axis as shown in Fig.2. The flow is assumed to be laminar and both hydro-dynamically and thermally fully developed. Also the thermo physical properties are assumed to be constant. The flowing fluid in the microchannels is considered as water.

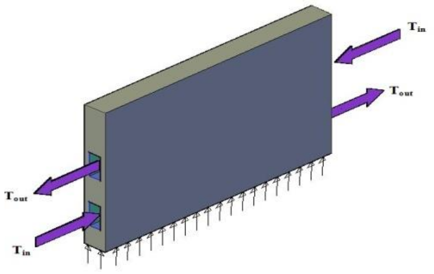


**Fig.2** Computational domain of counter flow single stacked rectangular microchannel heat sink.

## IV. CFD MODELING AND GOVERNING EQUATIONS

The three-dimensional fluid flow and heat transfer in different cross-sectional microchannels (viz,. Rectangular, Triangular, Pentagonal and Circular) heat sinks are analyzed using water as the cooling fluid. A schematic structure of a rectangular microchannel heat sink is shown in Fig.3.The micro-heat sink model consists of a 10 mm long ($L$=10 mm) silicon wafer having a width of $w$ = 810 µm, a depth of $h$ = 700 µm and are separated by 20 µm wall thickness. A uniform heat flux is applied at the bottom surface of the heat sink.

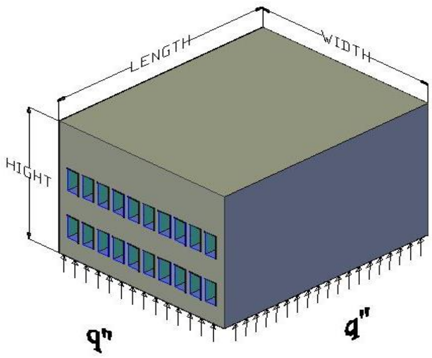


**Fig.3** Schematic of the stacked rectangular microchannel heat sinks.

### 4.1 Heat Sink Design

The computational domain will initially starts with the design of different cross-sectional stacked microchannel heat sinks. The design parameters of different cross-sections are mentioned with the help of a sample sectioned rectangular stacked microchannel heat sink as shown in following fig.4. It represents the height, width and spacing between channels by the design considerations.

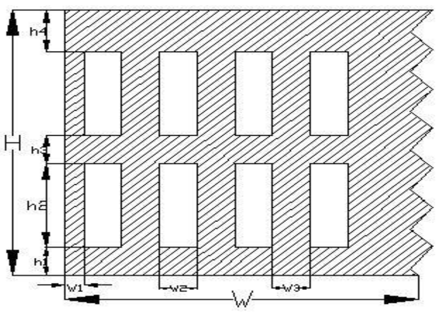


**Fig.4** sectional view of rectangular stacked microchannel

The dimensions of other cross-sections are mentioned as follows,

**Table.1** Geometric dimensions of the stacked microchannels

| Section Cross-section | $h_1$ (μm) | $h_2$ (μm) | $h_3$ (μm) | $h_4$ (μm) | $w_1$ (μm) | $w_2$ (μm) | $w_3$ (μm) |
|---|---|---|---|---|---|---|---|
| Rectangular | 18 | 18 | 14 | 2 | 02 | 06 | 02 |
| Triangular | 18 | 07 | 11 | 28 | 02 | 06 | 02 |
| Pentagonal | 18 | 12 | 10 | 30 | 02 | 05 | 02 |
| Circular | 21 | 05 | 16 | 33 | 02 | 05 | 02 |

## 4.2 GOVERNING EQUATIONS

The governing equations are continuity, momentum and energy equations, which are derived from fundamental principles of heat and fluid flow. The equations are posed to implement SIMPLE (Semi-Implicit Method for Pressure Linked equation) algorithm. Here no-slip assumptions are made.
"According to LANGHAAR EQUATION"
The length required to fully developed laminar flow entrance length = 0.057.$Re.D_h$
= (0.057.105.86.58) μm= 518.18 μm<10 mm
So fully developed laminar flow is valid.
Some simplifying assumptions are required before applying the conventional Navier-Stokes and energy equations to the model. The major assumptions are:

(1) Steady state flow and heat transfer,
(2) Incompressible fluid,
(3) Laminar flow,
(4) Uniform wall heat flux,
(5) Constant solid and fluid properties (thermo-physical properties)
(6) Negligible radiation of heat transfer,
(7) Negligible superimposed natural convective heat transfer.

## 4.3 Continuity Equation

$$\frac{\partial u}{\partial x}+\frac{\partial v}{\partial y}+\frac{\partial w}{\partial z}=0$$

Momentum Equation (Navier-Stroke Equation)

X – Momentum Equation:

$$\rho\left(u\frac{\partial u}{\partial x}+v\frac{\partial u}{\partial x}+w\frac{\partial u}{\partial x}\right)=-\frac{\partial P}{\partial x}+\mu\left(\frac{\partial^2 u}{\partial x^2}+\frac{\partial^2 u}{\partial y^2}+\frac{\partial^2 u}{\partial z^2}\right)$$

Y – Momentum Equation:

$$\rho\left(u\frac{\partial v}{\partial x}+v\frac{\partial v}{\partial x}+w\frac{\partial v}{\partial x}\right)=-\frac{\partial P}{\partial y}+\mu\left(\frac{\partial^2 v}{\partial x^2}+\frac{\partial^2 v}{\partial y^2}+\frac{\partial^2 v}{\partial z^2}\right)$$

Z – Momentum Equation:

$$\rho\left(u\frac{\partial w}{\partial x}+v\frac{\partial w}{\partial x}+w\frac{\partial w}{\partial x}\right)=-\frac{\partial P}{\partial z}+\mu\left(\frac{\partial^2 w}{\partial x^2}+\frac{\partial^2 w}{\partial y^2}+\frac{\partial^2 w}{\partial z^2}\right)$$

## 4.4 Energy Equation

$$\left(u\frac{\partial T}{\partial x}+v\frac{\partial T}{\partial x}+w\frac{\partial T}{\partial x}\right)=\frac{1}{\alpha}\left(\frac{\partial^2 T}{\partial x^2}+\frac{\partial^2 T}{\partial y^2}+\frac{\partial^2 T}{\partial z^2}\right)$$

# V. CFD ANALYSIS ON SOLID WORKS FLOW SIMULATION

This analysis has been carried out using commercial software "SOLIDWORKS". The reason behind using this software purpose is:

a) It is not possible to do a planned design experimentally due to experiment limitations.
b) At the same time it is imperative to do the above for the logical conclusion of the work.

c) Basic CFD based study is becoming popular with its increased reliability owing to extensive research work done on turbulence modeling and massive effort put by software developer for these models.
d) Development of specific software for CFD application for this study is also not possible in such a limited time. At the best, a particular geometry with simple turbulence model for one parameter could have attempted.

In view of this, use of commercial software like FLUENT, SOLIDWORKS etc., are the optimum choice.

## 5.1 Basic Steps in CFD Analysis

Basic step involved in flow simulation using CFD software can be summarized as

a) Generation of geometric model.
b) Creating lids.
c) Developing flow simulation wizard.
d) Defining material to heat sink.
e) Defining boundary conditions.
f) Defining the goals

## 5.2 Generation of Geometric Model

The geometric model has been created and meshing is done in SOLID WORKS software. The design of the microchannel heat sink is made by taking the reference dimensions of the channels that we have extracted from literature survey for this project. The geometry has been made by first creating the rectangular surface. After creating it is made to extrude to required length of heat sink. Then channels are created and cut along one surface to form a microchannel heat sink.

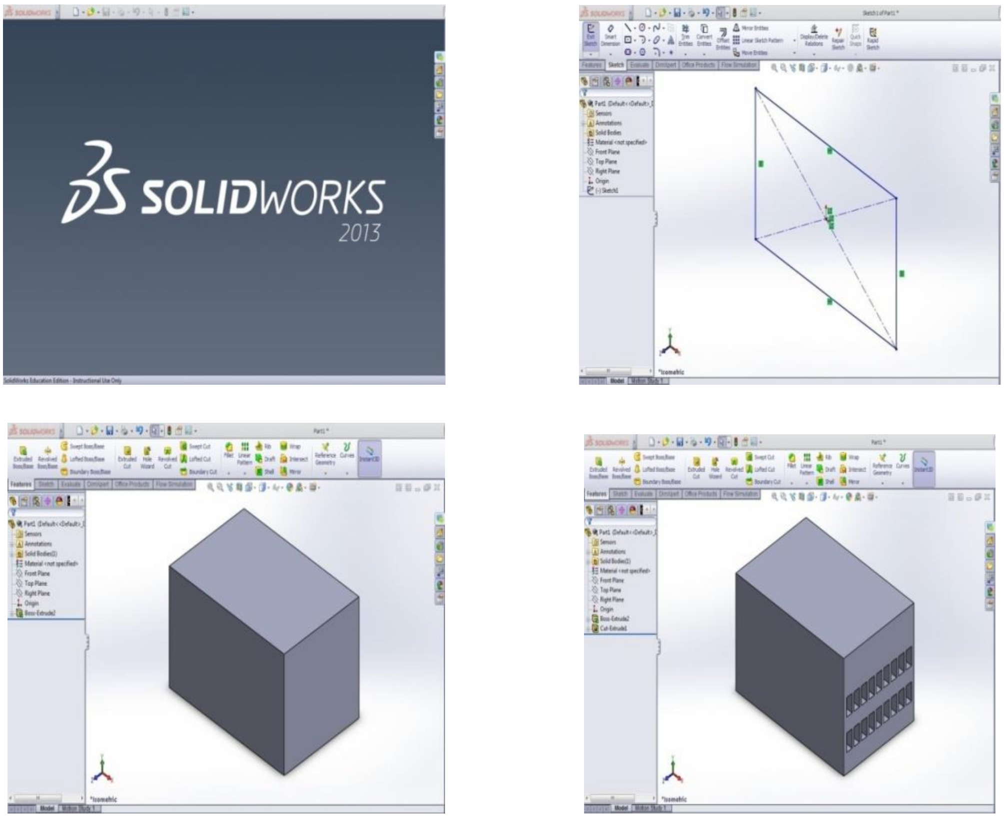


**Fig.5** Snapshots of Microchannel Stacked Rectangular Heat Sink in Solid Works.

## 5.3 Creating Lids:

Lids are created to all the channel openings of the heat sink in order to develop the computational fluid domain in the flow simulation process for defining the fluid flow channel in the heat sink.

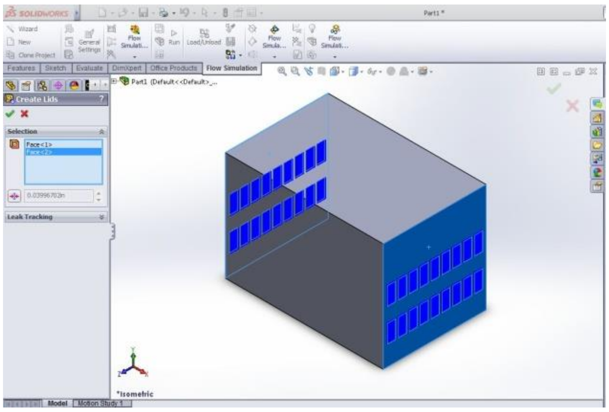

**Fig.6** Lids Creation for Microchannels

## 5.4 Developing the Flow Simulation Wizard:

For generating flow simulations wizard the followings steps are to be done,

i. In generating flow simulation wizard first define the project name for which the simulation process is conducting.
ii. Define the unit system in terms of SI system.
iii. Mention the analysis type as,
   a. Internal fluid flow,
   b. Heat conduction in solids and
   c. Gravity along Y-axis as -9.81 m/sec$^2$
iv. Define the type of fluid flowing through microchannels and its properties,
   a. Liquid : water,
   b. Density : 1000 kg/m$^3$
   c. Flow type : laminar and turbulent
   d. Cavitations : nil
v. Define the wall condition as adiabatic wall,
vi. Define initial conditions as,
   a. Pressure : 101325 pa
   b. Temperature : 293 K
   c. Velocity along X-direction : 0 m/s
   d. Velocity along Y-direction : 0 m/s
   e. Velocity along Z-direction : 0 m/s
vii. Finish the wizard with fine resolution.

## 5.6 Defining Boundary Conditions

Define the boundary conditions for the heat conduction in solid as well as the inlet and outlet conditions of fluid flow through microchannel heat sink as follows,

*a.* ***Inlet Boundary Conditions:*** Select all the lid faces through which the fluid enters into the microchannels and define the mass flow rate normal to face of 1.0000e-005 kg/s
*b.* ***Outlet Boundary Conditions:*** Select all the lid faces through which the fluid exits from the microchannels and define it as Static pressure: 101325.00 Pa.
*c.* ***Heat flux:*** Select the bottom face of the heat sink to define the Surface heat generation rate of 7500000.000 W/m^2

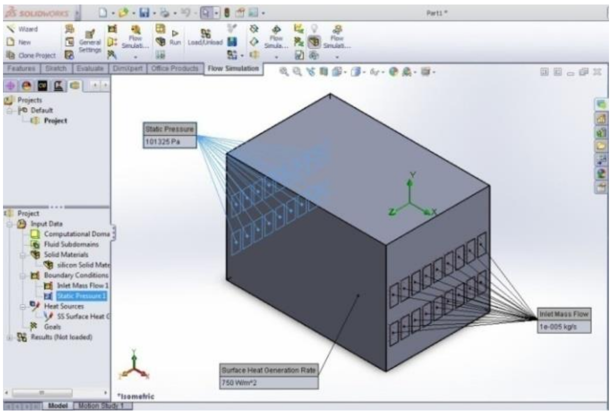


**Fig.7** Boundary Conditions for Microchannel Heat sink

## VI. CFD Analysis Result

The performance of various cross-sections of microchannel heat sinks like rectangular, triangular, pentagonal and circular channels is evaluated. The results are compared for parallel and counter flow arrangement of each followed by comparison among each configuration. The following figures shows the pressure and temperature trajectories of different configuration microchannels.

### 6.1 Parallel Flow Rectangular Stacked Microchannel Heat Sink

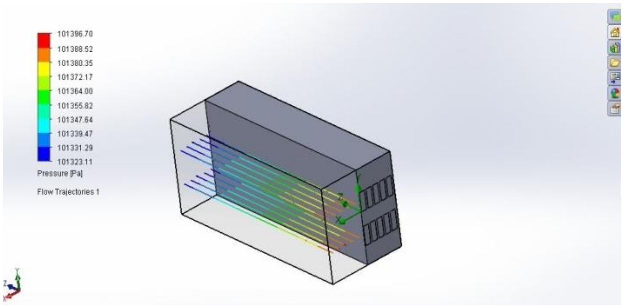


**Fig.8** Pressure Trajectories of parallel flow rectangular microchannel

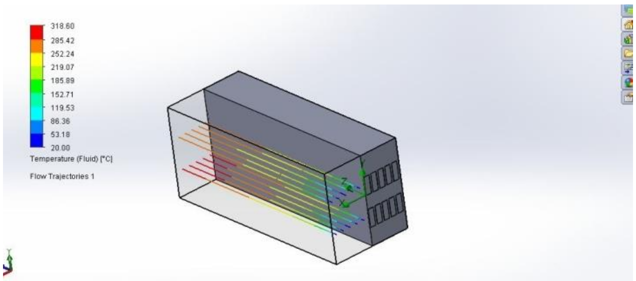


**Fig.9** Temperature Trajectories of parallel flow rectangular microchannel

### 6.2 Counter Flow Rectangular Stacked Microchannel Heat Sink

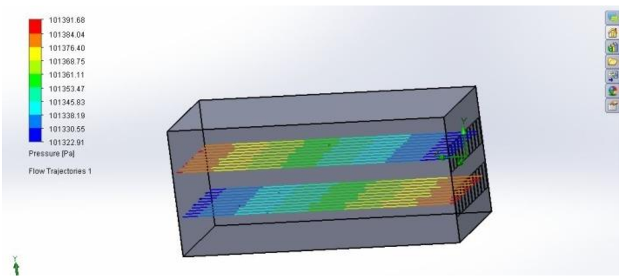


**Fig.10** Pressure Trajectories of counter flow rectangular microchannel

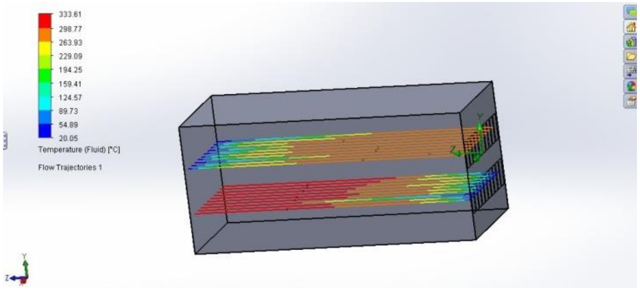


**Fig.11**Temperature Trajectories of counter flow rectangular microchannel

## 6.3 Results Obtained from Simulation

Table 2 shows the results obtained from flow simulation of different cross-sectional microchannels like rise in the temperature of fluid, static pressure drop cross the microchannels and volumetric heat transfer coefficient from parallel and counter flow. From the results we can observe that, the temperature of fluid coming out from rectangular microchannel in both parallel (318.6$^0$C) and counter flows (33.61$^0$C) are more when compared to other cross-sectional microchannel heat sinks.

**Table.2**Results Obtained from Simulation

| S. No. | Microchannel Used | Quantity | Parallel Flow | | Counter Flow | |
|---|---|---|---|---|---|---|
| | | | Min | Max | Min | Max |
| *1* | ***Triangular*** | Temperature of fluid (°c) | 20 | 33.24 | 20 | 48.94 |
| | | Temperature of Solid (°c) | 25 | 48.81 | 25 | 56.44 |
| | | Static Pressure (Pa) | 101324.69 | 102295.97 | 101322.80 | 102202.68 |
| | | Volumetric heat transfer coefficient (W/cm$^3$.$^0$c) | 0.477 | | 0.503 | |
| *2* | ***Rectangular*** | Temperature of fluid (°c) | 20 | 45.6 | 20 | 60.61 |
| | | Temperature of Solid (°c) | 25 | 53.42 | 25 | 68.39 |
| | | Static Pressure (Pa) | 101323.18 | 101397.95 | 101322.91 | 101391.68 |
| | | Volumetric heat transfer coefficient (W/cm$^3$.$^0$c) | 0.599 | | 0.64 | |
| *3* | ***Circular*** | Temperature of fluid (°c) | 20 | 42.53 | 20 | 43.41 |
| | | Temperature of Solid (°c) | 25 | 50.50 | 25 | 51.46 |
| | | Static Pressure (Pa) | 101324.35 | 101829.05 | 101321.21 | 101829.66 |
| | | Volumetric heat transfer coefficient (W/cm$^3$.$^0$c) | 0.217 | | 0.215 | |

From the numerical study on different microchannel heat sinks (rectangular, triangular, pentagonal and circular) by parallel and counter flow study we conclude that rectangular microchannel heat sink will give better volumetric heat transfer coefficient 0.64 W/cm3K in counter flow when compared to other cross-sectional microchannel heat sinks.

## 6.4 VALIDATION OF CFD:

Engineering design optimization involving multiple conflicting objectives produces a set of solutions called the Pareto optimal solution set. Generating Pareto optimal solutions is the focus of multi objective optimization as employed here (Deb, 2001).

### 6.5 Micro channel Optimization using MOGA

The objectives of this study are to minimize both the maximum temperature and the volume per unit heat (which means increasing the heat density per unit volume). The constraints are the pressure drop and limitation on both maximum temperature and temperature differences across the fluid (i.e., temperature difference between outlet and inlet of the channel). MOGA was used to obtain optimum solutions. The problem formulation is as follows:

Minimize : $T_{max}$
Minimize : V/Q
Subject to : $34.5\ \text{kPa} \leq \Delta P \leq 400\ \text{kPa}$
$2\ \text{K} \leq \Delta T_f \leq 14\ \text{K}$
$T_{max} \leq 600\ \text{K}$
$q = 750\ \text{W/cm}^2$

**Table.3**Design Variables Limits:

| **Design Variable** | **Lower Limit** | **Upper Limit** |
|---|---|---|
| $H_{ch}/H_o$ | 1 | 2.5 |
| $W_{ch}/H_o$ | 1 | 75 |
| $L_{ch}/H_o$ | 300 | 80012.8 |
| $T_{side}/H_o$ | $0.1 \times W_{ch}/H_o$ | $0.5 \times W_{ch}/H_o$ |
| $T_{top}/H_o$ | $0.1\times H_{ch}/H_o$ | $0.5\times H_{ch}/H_o$ |
| $T_{bottom}/H_o$ | $0.1\times H_{ch}/H_o$ | $0.5\times H_{ch}/H_o$ |
| $v$ | 0.2 m/s | 12 m/s |

But from the optimization we get the total thermal resistance as,

$$R_{th} = \frac{T_{max}-T_f}{q}$$

According to "Closed-form correlation for thermal optimization of microchannels", Int. J. Heat Mass Transfer by D.K. Kim, S.J. Kim, the total thermal resistance of microchannel will be,

$$R_{pr} = \frac{1}{3}\frac{(1+\beta_{ch})w_{ch}\cdot\alpha_{ch}}{k_s\beta_{ch}LW} + \frac{17}{140}\frac{(1+\beta_{ch})w_{ch}}{k_f\alpha_{ch}LW} + \frac{\sqrt{12\,\mu_f w_{ch}(1+\beta_{ch})L}}{\rho_f C_{pf}\sqrt{w_{ch}\alpha_{ch}WP}} + \frac{t}{K_s}\frac{1}{LW}$$

### 6.6 Validation for Rectangular Microchannel Heat Sink:

**Table.4** validation for Rectangular Microchannel Heat Sink

| **Heat sink** | | **Water Coolant** | | **Microchannel** | |
|---|---|---|---|---|---|
| L | 10 mm | $k_f$ | 0.60 W/m K | $\alpha_{ch}$ | 180 µm |
| W | 8.6 mm | $cp_f$ | 4179 J/kg K | $\beta_{ch}$ | 10 mm |
| t | 7 mm | $q_f$ | 997.1 kg/m$^3$ | $w_{ch}$ | 56 µm |
| $k_s$ | 140 W/m K | | | | |
| $q_s$ | 2330 kg/m$^3$ | | | | |
| Cp | 714.0 J/(kg*K) | | | | |
| Melting temperature | 726.85 °C | | | | |

By comparing the thermal resistances with different input heat flux, we get:

**Table.6** Comparing Different Input Heat Flux

| Design of channel | Heat input (W/cm$^2$) | Thermal resistance | |
|---|---|---|---|
| | | $R_{th}$ | $R_{pr}$ |
| Rectangular | 750 | 0.151 | 0.1413 |
| Triangular | 450 | 0.112 | 0.108 |

So from the above results we conclude that the results obtained from simulation are approximately equal to theoretical results for rectangular microchannel.

# VII. CONCLUSIONS

The fluid flow and heat transfer processes in a rectangular, triangular, pentagonal and circular micro-channel heat sink with parallel and counter flow were analyzed by using "SOLID WORKS FLOW SIMULATION" and obtained numerically base temperature of the heat sink, temperature rise in fluid, pressure drop across micro channels and volumetric heat transfer coefficients.
Based on the analysis of the different cross-sectional stacked microchannel heat sink behaviour the following conclusions can be drawn

- From the results during the process of simulation it was observed that the base temperature of the heat sink is having a minimum temperature of 25$^0$C to maximum of 68.39$^0$C for rectangular microchannel heat sink when compared to other cross-sectional heat sinks.
- From the numerical study on different micro-channel heat sinks (rectangular, triangular, pentagonal and circular) by parallel and counter flow study we conclude that rectangular microchannel heat sink will give better volumetric heat transfer coefficient 0.64 W/cm$^3$K in counter flow when compared to other cross-sectional microchannel heat sinks.
- From the results we can observe that, the temperature of fluid coming out from rectangular microchannel in both parallel (45.6$^0$C) and counter flows (60.61$^0$C) are more when compared to other cross-sectional microchannel heat sinks.
- The heat transfer and fluid flow models used in the present study have been developed and simulated using "SOLID WORKS FLOW SIMULATION" process, however to validate the results we calculate the thermal resistances and are compared against experimental results.

## ACKNOWLEDGEMENT

The author would like to thank all the persons who helped in the completion of his experimental work. Also thanks are extended to ASR College of Engineering, Tanuku, and, INDIA for support through the execution of the experimental work.

## REFERENCES

[1]. Hetsroni, G., Mosyak, A. and Segal, Z., (2001), "Non-uniform temperature distribution in electronic devices cooled by flow in parallel microchannels", IEEE Transactions onComponents and Packaging Technologies, Vol. 24, pp. 16-23.
[2]. Hetsroni, G., Mosyak, A., Segal, Z. and Pogrebnyak, E., (2003), "Two-phase flow patterns in parallel microchannels", International Journal of Multiphase Flow, Vol. 11, pp. 353-358.
[3]. Bau, H.H., (1998), "Optimization of conduits' shape in micro heat exchangers", International Journal of Heat and Mass Transfer, Vol. 41, pp. 2717-2723.
[4]. Zhao, C.Y. and Lu, T.J., (2002), "Analysis of microchannel heat sinks for electronics cooling", International Journal of Heat and Mass Transfer, Vol. 45, pp. 4857-4869.
[5]. Harpole, G.M. and Eninger, J.E., (1991), "Microchannel heat exchanger optimization", IEEE Proceedings of the Seventh Semi-Thermo Symposium, Vol. 361, pp, 59-63.
[6]. Copeland, D., Behnia, D. and Nakayama, W., (1996), "Manifold microchannel heat sinks: Isothermal analysis", IEEE Proceedings of the Fifth Intersociety Conference onThermal Phenomena in Electronic Systems, pp. 251-257.

[7]. Ng, E.Y.K. and Poh, S.T., (1999), "Investigative study of manifold microchannel heat sinks for electronic cooling design", Journal of Electronics Manufacturing, Vol. 9, pp. 155-166.
[8]. Hung, T.C., Wangi S.K. and Tsai, F.P., (1997), "Simulation of passively enhanced conjugate heat transfer across an array of volumetric heat sources", International Journalfor Numerical Methods in Engineering, Vol. 13, pp. 855-866.
[9]. Li, J., Peterson, G.P. and Cheng, P., (2004), "Three-dimensional analysis of heat Transfer in a micro-heat sink with single phase flow", International Journal of Heat and MassTransfer, Vol. 47, pp. 4215-4231.
[10]. Masud, Bebnia,,Wataru, Nakayama. and Jeffrey, Wan., (1998), "CFD simulations of heat transfer from a heated module in an air Stream: comparison with experiments and a parametric study", IEEE Intersociety Conference on Thermal Phenomena, Vol. 54,pp.539-544.
[11]. Qu, W. and Mudawar, I., (2002), "Analysis of three-dimensional heat transfer in microchannel heat sinks", International Journal of Heat and Mass Transfer, Vol. 45, pp. 3973-3985.
[12]. Ryu, J.H., Choi, D.H. and Kim, S.J., (2003), "Three-dimensional numerical optimization of a manifold microchannel heat sink", International Journal of Heat and Mass Transfer, Vol. 46, pp. 1553-1562.

## AUTHORS BIOGRAPHY

**Bella Jaya Chandana** was born in Polavaram, Andhrapradesh, India, in 1991. He receive the bachelor's degree in mechanical engineering from the university of Jawaharlal Nehru technological university, Kakinada, in 2013. He is currently pursuing the master degree in thermal engineering from the university of Jawaharlal Nehru technological university, Kakinada. His research interests include Design of Stacked Microchannel Heat Sinks Using Computational Fluid Dynamic.