DESIGN AND ANALYSIS OF MICROCHANNEL HEAT SINK FOR CONCENTRATOR PHOTOVOLTAICS

A Project Report Submitted

In Partial Fulfillment of the Requirements

for the Degree of

Bachelor Of Technology

In

Mechanical Engineering

By

ABHISHEK PRATAP SINGH	2018051010
ABHISHEK KANNOJIA	2018051006
ABHISHEK NISHAD	2018051009
ABHYUDAY PRATAP SINGH	2018051014
ARSHIT SINGH	2018051031
SHIVANSH TRIPATHI	2018051100

Under the Supervision of

Prof. S.C. Jayswal

DEPARTMENT OF MECHANICAL ENGINEERING

Madan Mohan Malaviya University of Technology, Gorakhpur (U.P.), INDIA

June,2022

CERTIFICATE

This is to certify that project entitled "***Design and Analysis of Microchannel Heat Sink for Concentrator Photovoltaics***" has been submitted to the Department of Mechanical Engineering for the fulfillment of the requirement for the award degree of Bachelor of Technology from Madan Mohan Malaviya University of Technology, Gorakhpur under my supervision. The project embodies results of original work and studies carried out by students themselves and the contents of the project report do not form the basis for the award of any other degree to the candidate or to anybody else.

Students' Name

Abhishek Pratap Singh (2018051010)

Abhishek Kannojia (2018051006)

Abhishek Nishad (2018051009)

Abhyuday Pratap Singh (2018051014)

Arshit Singh (2018051031)

Shivansh Tripathi (2018051100)

Prof. S.C. Jayswal
Mechanical Engineering Department
M.M.M.U.T. Gorakhpur

Date: ____________________

CANDIDATES' DECLARATION

We declare that this written submission represents our work and ideas in our own words and where others' ideas or words have been included, we have adequately cited and referenced the original sources. We also declare that we have adhered to all principles of academic honesty and integrity and have not misrepresented or fabricated or falsified any idea/data/fact/source in our submission. We understand that any violation of the above will be cause for disciplinary action by the University and can also evoke penal action from the sources which have thus not been properly cited or from whom proper permission has not been taken when needed.

Abhishek Pratap Singh (2018051010) ……………………

Abhishek Kannojia (2018051006) ………………………..

Abhishek Nishad (2018051009) …………………………..

Abhyuday Pratap Singh (2018051014) …………………..

Arshit Singh (2018051031) ……………….…………...

Shivansh Tripathi (2018051100) …………………………

APPROVAL SHEET

The project report entitled "***Design and Analysis of Microchannel Heat Sink for Concentrator Photovoltaics***" by Abhishek Pratap Singh (2018051010), Abhishek Kannojia (2018051006), Abhishek Nishad (2018051009), Abhyuday Pratap Singh (2018051014), Arshit Singh (2018051031), Shivansh Tripathi (2018051100) is approved for the degree of Bachelor of Technology in Mechanical Engineering.

Examiner

____

____

____

Supervisor

____

Prof. S.C. Jayswal

Mechanical Engineering Department

M.M.M.U.T. Gorakhpur

Head of Department

____

Prof. Jeeoot Singh

Mechanical Engineering Department

M.M.M.U.T. Gorakhpur

Date _____________________

Place_____________________

ACKNOWLEDGEMENT

It gives us immense pleasure to make our B.Tech. project report on the topic "***Design and Analysis of Microchannel Heat Sink for Concentrator Photovoltaics***". The excellent teamwork of several months and the collaborative effect of Mechanical Engineering Department of our University has been the prime secret behind its present shape.

Above all, comes the name of our scholarly guide **Prof. S.C. Jayswal**, Mechanical Engineering Department, we would like to express our deep sense of gratitude towards Sir, whose supervision, encouragement, and support never allowed us to deviate from our objective.

We would also like to thank our project coordinator **Sri S.K. Yadav** for his constant support.

Lastly, we would like to thank our parents, friends and all those who in some manner or the other helped in the accomplishment of this project.

LIST OF FIGURES

LIST OF TABLES

ABSTRACT

Microchannel Heat Sink is basically an object that absorbs and takes away the heat from high heat flux region in micro devices thereby not allowing the temperature of concerned region to go above desired level.

The world is moving towards miniaturization. The specifications which a bulky desktop used to have 10 years back is being offered by a small smartphone. With this attempt to pack as much features as possible in smallest device possible, comes the need for cooling of electronic components.

The cooling of electronic components is essential as it enhances the performance and increases its reliability. Many techniques have been used in the past to cool down the surface of electronic components. One of the most promising techniques has been incorporating microchannels.

Many researchers have worked on finding optimum design of microchannels so as to get maximum thermal performance out of it. In this project, numerical analysis using ANSYS Fluent has been done on radial straight and radial curved microchannel based heat sinks. The performance of both the heat sinks are compared.

Concentrator Photo Voltaic (CPV) has been emerging as a promising technology in the field of renewable energy. It has been established that efficiency of CPV module drops with increase in temperature. A CAD model of CPV module with suggested heat sink is also prepared.

Keywords:

Heat sink

Microchannel

Radial microchannel

Curved microchannel

Concentrator Photovoltaic

TABLE OF CONTENTS

CHAPTER- 1

INTRODUCTION

1.1 Heat Sink

Heat sink is basically an object that absorbs and takes away the heat from high heat flux region thereby not allowing the temperature of concerned region to go above desired level. It is generally a metallic substrate the takes away the heat and dissipates in the environment. More recently forced fluid flow-based techniques have given better thermal performance.

1.2 Requirements of Heat Sink

The figure [Ref.1] below shows the rate at which complexity of microchips, with more and more parts being added, is increasing over the years.

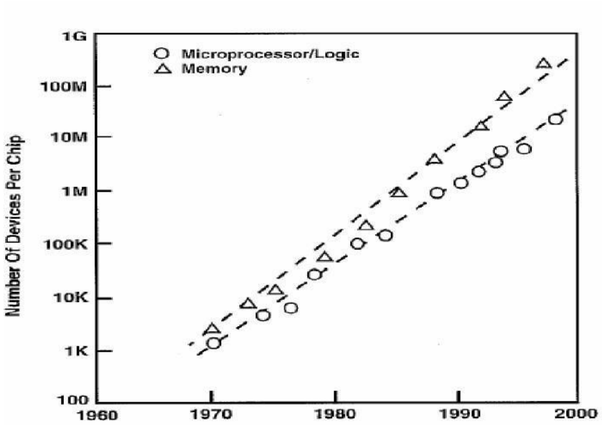


Fig.1: Complexity in circuit over the years

Due to such huge number of components, of the order of 105 to 108, being added on microchips a heat flux of 100 W/cm^2 is generated. It becomes imperative to remove heat so as to ensure their reliability and efficiency.

The pie-chart [Ref.10] below describes that 55% of the electronic component's failure occurs due to high temperature related issues.

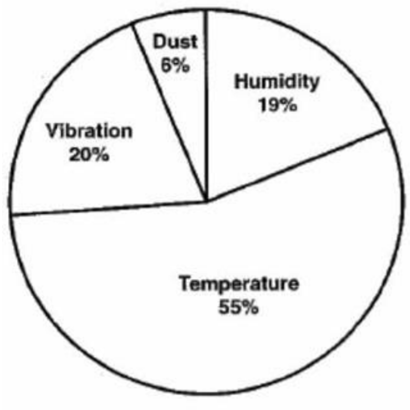


Fig.2: Causes of electronic component failure

Apart from reliability, another aspect in which heat generation has adverse impact on electronic component is the efficiency. The figure [Ref.10] below depicts the variation in the efficiency of CPV module with respect to temperature. It shows that there is a significant decrease in the efficiency of the module as the cell temperature increases.

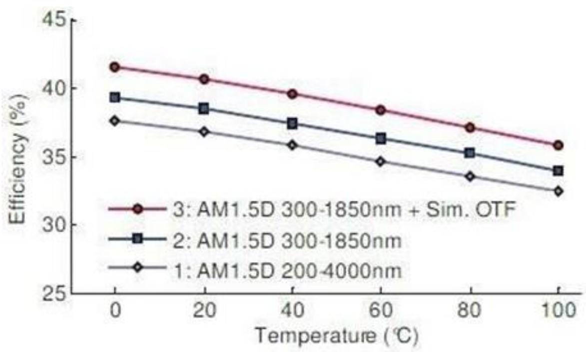


Fig.3: Variation of efficiency of CPV wrt temperature

1.3 Types of Heat Sink

The heat sinks are broadly classified as active heat sinks and passive heat sinks. The major difference has been listed out in the table below.

Active heat sinks	Passive heat sinks
• Involves usage of external power	• No external power required
• Has fans and pumping systems	• Doesn't have mechanical parts
• Ex. Forced air system, Microchannel	• Ex. Heat pipes, PCM

Table.1: classification of heat sinks

1.4 Current Cooling Systems used to cool Electronic Modules

In the bid of achieving of high-performance in microelectronics the thermal management of high heat fluxes has been a critical barrier. Although the average heat flux in components is around the range of 100 to 300 W/cm^2, however the maximum heat flux at certain locations may be of the order or even be greater than 1 kW/cm^2. So, for handling such heat fluxes at much lower operating temperatures, there is requirement of integrated cooling methods which are capable of removing heat from close to source. These includes:-

1. Microjet Impingement.
2. Spray cooling.
3. Micro-channel Heat Sink

Micro-jet Impingement:

It depends mainly on the usage of a high-velocity liquid jet that ejects out from a nozzle. This is done so as to reduce the thicknesses of the thermal boundary layer. Consequently, the average heat transfer coefficient of convection on the surface increases.

Drawbacks in using micro-jet impingement: If we see from the point of view of fluid recovery in the open systems and the uniformity in temperature, we can find out that Microjet impingement has got various drawbacks. Therefore, complex design models incorporating many microjet ports are required. This causes the designs to specific to narrow range of parameters and also the designs are not robust enough.

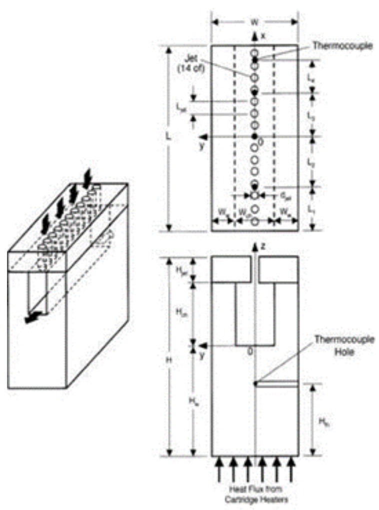


Fig.4: Micro-jet impingement schematic

Spray Cooling:

This method depends upon a collection of liquid drops instead of a continuous liquid jet striking the heated source. This provides improved surface coverage and uniform distribution of temperature. Instead of depending solely on convective heat transfer the liquid drops also promote evaporative cooling. The latent heat translates into lower surface-temperature requirements associated with the phase change. This is an added advantage over the microjet impingement cooling. Spray cooling is one of the best methods in the bid to cooling for the components subjected to heat flux exceeding $1kW/cm^2$.

Drawbacks in using spray cooling: Drawbacks is that it involves complex pumping mechanisms to develop drops and spread them. This is one of the major hurdles for this technology to be fully used to its potential.

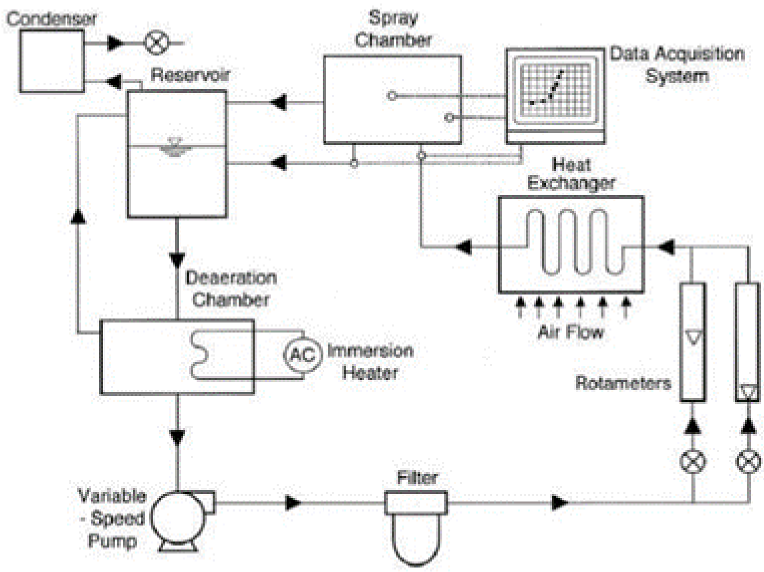


Fig.5: Spray Cooling Schematic

1.5 Microchannel Heat Sink

Liquid coolant microchannel based heat sinks has emerged as one of the technologies to be most promising one in area where high heat flux removal is required. The average heat transfer coefficient can be increase to high value by incorporating very small hydraulic diameter channels. Since microchannels have very small cross section the flow is laminar and is essentially constant Nusselt number flow. Consequently, the average heat transfer coefficient has an inversely varying relation with the hydraulic diameter. This implies that as we will go on decreasing the channel hydraulic diameter the average coefficient for heat transfer for convection will go on increasing.

Pumping a liquid through a large number of small hydraulic diameter channels in comparison to a small number of large hydraulic diameter channels increments the level of transfer of heat from the hot surfaces such as microchips into the fluid flowing. So, the concept basically is to have a number of small cross section channels. Consequently, the heat sink with greater number of smaller hydraulic diameter channels gives improved performance concerned to heat transfer.

The science behind the popularity of microchannels as heat sink refers to the formula of the Nusselt (Nu) number which incorporates the average coefficient of heat transfer as:

$$\boldsymbol{h} = \boldsymbol{K} * \boldsymbol{Nu} / \boldsymbol{D}$$

Here K refers to the property of thermal conductivity of the coolant fluid and D refers to the hydraulic diameter of the channel cross section. We can consider flow inside the channel of very small hydraulic diameter to be laminar and one that is fully developed. Consequently, considering classical approach to channel flow, the (Nu) number is assumed to be constant. The small magnitude D of microchannels being in the denominator should significantly enhance the average coefficient of heat transfer. For a fully developed laminar (smooth) flow in a square channel with constant wall temperature or constant wall heat flux, the Nusselt number is a constant.

One of the major reasons for increased use of microchannel based heat sinks is the compactness it provides and high surface area to volume ratio.

In past different coolants, channel material, flow conditions and microchannel shapes were investigated to improve the heat transfer capabilities of microchannel heat sinks.

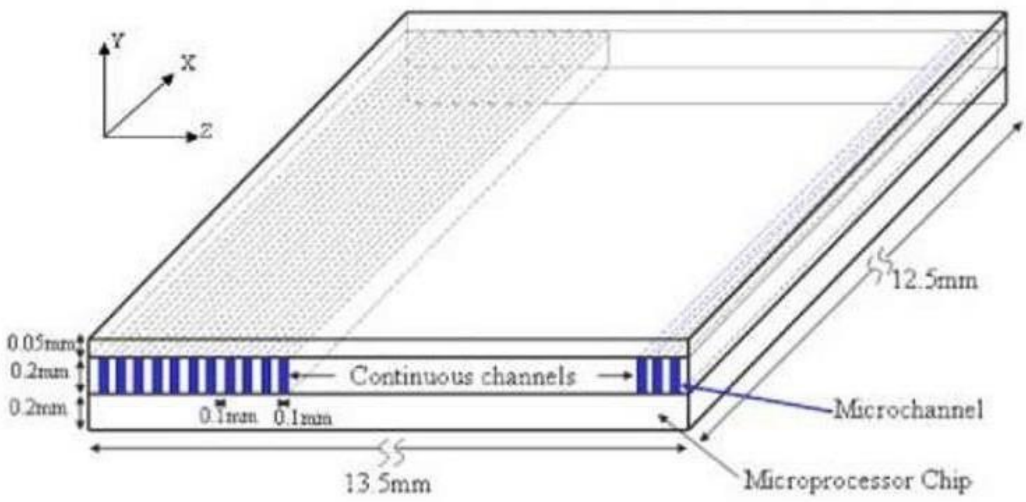


Fig.6: Microchannel heat sink in microchip

Microchannel Geometry

To improve the overall heat transfer in the microchannel, various geometries such as rectangular, circular, trapezoidal, diamond and hexagonal shapes were researched with the primary motivation of increasing the effective area of heat transfer and thus reducing convective thermal resistance.

Heat augmenters such as ribs and cavities were used to increase surface area. Also attempts were made with flow path changes as zigzag, curves, steps and waves in rectangular microchannels. The augmenters increased thermal performance at the cost of more pumping power requirements. Amongst various other geometries, rectangular ones were found to be most stable and machinable with better thermal performance and a preferred choice for researchers.

Fluid/Coolant Used

Liquid coolants such as water, methanol, refrigerants and nanofluids with gaseous coolants such as air, helium, argon and ammonia are also being researched for better dissipation of heat. However, water and air were the most used due to availability and low cost. Liquid coolants are found to provide better required heat transfer with respect to gaseous coolants. Nanofluids is one of the major research areas as a coolant nowadays.

Structural Material

Materials such as Copper, Aluminium, Silicon, Stainless Steels, Glass and Bronze as the channel material were the areas of major research. However, the effect of the channel material was only found significant in case of higher channel heights and non- influential on overall heat dissipation in case of lower channel heights. This is the reason why silicon is preferred over copper and aluminium for heat sinks due to its lesser weight which is suitable for cooling applications in electronics.

Flow Conditions

Dominant count of researches were focused on laminar flow instead of the turbulent flows in the channel due to the short length of microchannel to create turbulence and general Reynolds Number being in range of 500-2300. However, some researches have suggested turbulent flows better than laminar but with higher pumping power requirements.

Microchannel plays a crucial role in electronics cooling, medical instruments, laser equipments, automotive industries and aerospace technology. It has shown immense potential for cooling by their high surface to volume ratio and usage by nanotubes, grooves, pins which can further enhance the performance. In this project, work has been done on geometry aspect. More specifically radial heat sinks with straight and curved microchannels are analyzed using simulation tools.

CHAPTER- 2

LITERATURE REVIEW

In last 40 years microchannel has received much attention from the researchers. This was first realized by **Tuckerman and Pease** [1] by experimenting on microscale flow through straight rectangular channels and consequently how it improved heat transfer characteristics (Fig. 7)

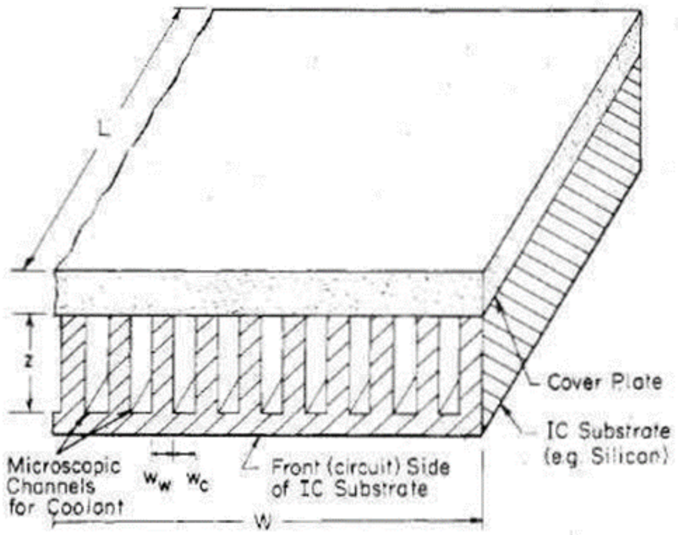


Fig.7: Rectangular microchannel used by Pease, Tuckerman

Guo and Cheng [2] experimentally numerically studied the fluid flow and thermal heat transfer related performance of curved microchannels. They compared 4 curved microchannels with same cross section but different radius of curvatures. They plotted the variation of pressure drop with respect to Reynolds number. They reported that pressure drop increases with increase in Reynolds number for the same channel. At the same time pressure drop is more for channel with smaller radius if curvature. The increased pressure drop is attributed to increased skin friction coefficient at the outside wall of curved channel. The fluid velocity at the outer wall increases due to centrifugal force generated by the curvature.

Xu and Guo [3] also performed thermal analysis in continuation to the work done above. They applied heat flux on Silicon based curved microchannel heat sink. They also plotted the variation of Nusselt (Nu) with varying Reynolds (Re) number for all the channels. They found out that there is significant improvement in Nu values for channels with lesser radius of curvature. They reported that curved channels outperform straight microchannels in terms of heat transfer enhancement. The enhancement is attributed to formation of secondary flows generated due to centrifugal force. The enhancement increases with increase in curvature at the expense of increase in pumping power. They found that the simulation results were in strong agreement with experimental ones. They also pointed out that at higher Reynolds number the increase in Nu values is more significant compared to the increase in pressure drop. Thus increase in heat transfer more than compensates for the penalty of increased pressure drop.

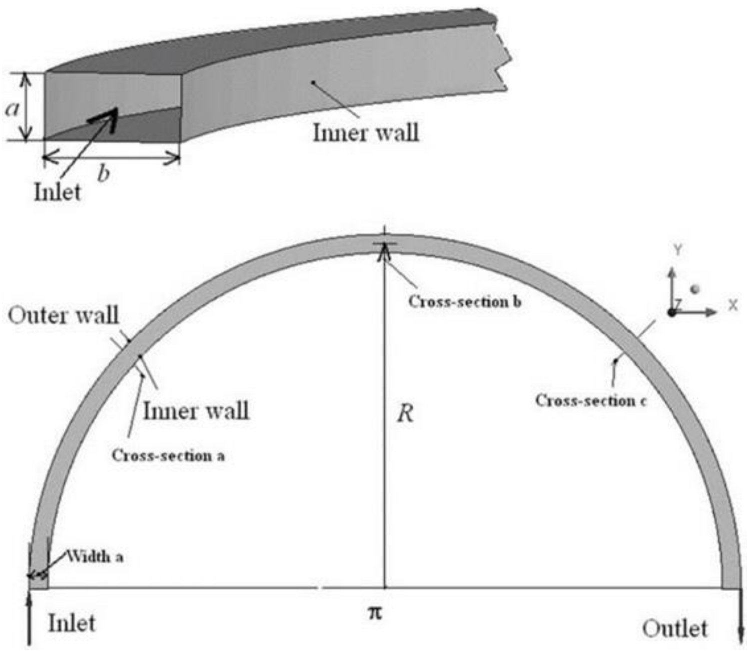


Fig.8: Curved microchannel by Guo and Cheng

Duryodhan and Abhimanyu [4] experimentally and numerically studied the performance of heated diverging and heated converging microchannels. They used deionized water as the fluid medium. They reported that converging channels gives better thermal performance that straight ones. One of the significant reasons for that is increase in the fluid velocity along the length. This increases the Reynolds number along the flow direction consequently giving better Nu values. They also developed correlations to calculate pressured drop and Nu values for both diverging and converging channels. They also reported that diverging sections require lesser pressure difference and hence lesser pumping power. Most importantly they reported that the dimensionless ratio of increase in Nusselt number to that of pumping power is better for both diverging and converging sections as compared to straight ones.

As a result, it is found that converging and diverging microchannels portray better thermo- hydraulic performance compared to constant cross section area microchannel. Consequently, these types of microchannels could be of more potential benefit in the domains of electronics, biological, chemical, biological cooling purposes involved with single phase liquid coolant.

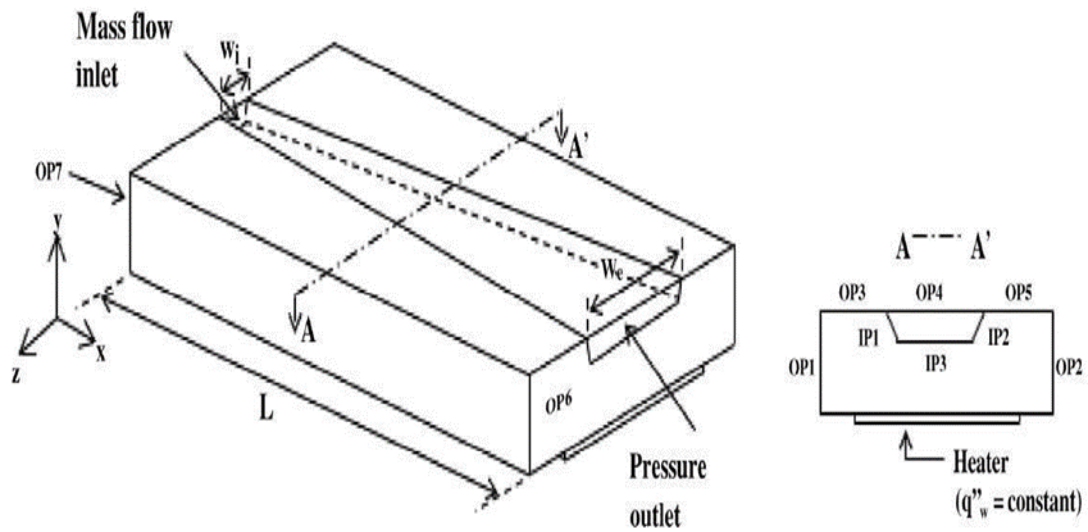


Fig.9: Converging and diverging section studied by Duryodhan

P.S. Lee and Teo [5] numerically studied heat transfer and thermal performance in sinusoidally varying wavy channels using ANSYS Fluent package. For this they took laminar flow with coolant as water and heat flux to be applied in 3-dimensional wavy microchannels having uniform rectangle cross section. Analysis and studies were made by simulation. Constant heat flux conditions were applied on the wall. Under conjugate simulation a silicon heat sink was also designed and fluid domain was defined separately. This was done so as to ensure more practical case. The fluid flow was studied and reasons were pointed out as to why wavy channel performs better. Also, it is suggested in the study to modify the relative wavy (2A/L) amplitude (wavy amplitude/distance between crests) of the microchannel in the flow direction for realistic applications. This ensured that there was no compromise as far as compactness of microchannel is concerned. They suggested that the relative (2A/L) value should be increased in flow direction to compensate for the deterioration observed in conventional straight channels. This causes an increase in heat transfer performance in the flow direction and promotes uniform temperature distribution. The relative (2A/L) value may also be modified as per need and demand to be at region with high heat flux (as shown in figure c) for removing heat from hotspots.

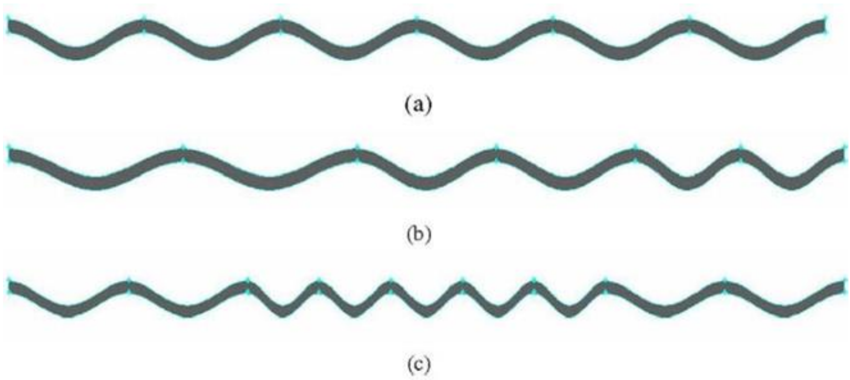


Fig.10: Wavy studied by P.S Lee. (a) constant ϒ (b) increasing ϒ (c) varying ϒ

Xi, Yu, Xie and Gao [6] realized this and the geometry was improved by Xi, Yu, Xie and Gao (Youmin Xi, 2010), (Fig 10) who worked on single phase flow and heat transfer in swirl microchannel, and proposed that the heat transfer performance increased by an average of 50% compared to that of straight and curved microchannel. But this also had a drawback of huge increase in pumping power and much increase in the temperature of the sink towards the outlet side.

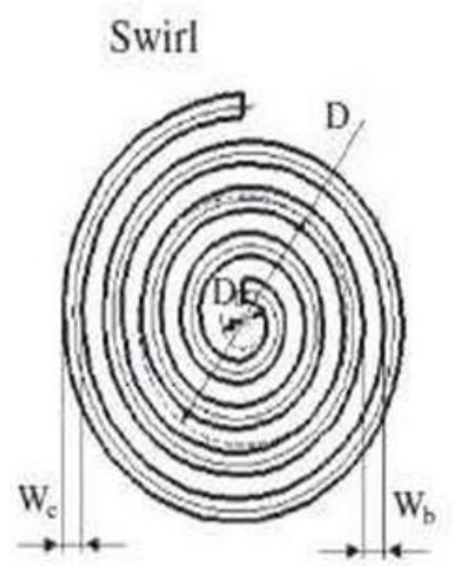


Fig.11: Swirl microchannel studied by Xie and Gao

Hassan and Muwanga [7] studied the performance of radial microchannel heat sink over conventional parallel channels. They reported that radial microchannels theoretically reduced the pressure drop by half by cutting the fluid path by half. Radial channels are also more feasible when it comes to incorporating varying cross section channels.

The major fallback of straight microchannels is a higher pressure drop. With a radial array of channels and a central inlet, the area cooled by the working fluid is nearly cut in half. Because of this design, theoretically the pressure drop over the length of the channels is halved. There is another problem associated with the use of conventional parallel channel heat sink of non-uniform temperature distribution. Heat sink temperature rises to high values at the exit side of the channel. This problem is solved by the use of radial microchannel based heat sink as it provides more uniform temperature distribution.

One important benefit of using radial heat sinks is that it becomes easier and natural to incorporate varying cross section channels. If fluid enters at center it becomes diverging and converging if fluid enters at periphery. As already pointed out above that diverging and converging microchannels give better thermo-hydraulic performance compared to straight counterparts.

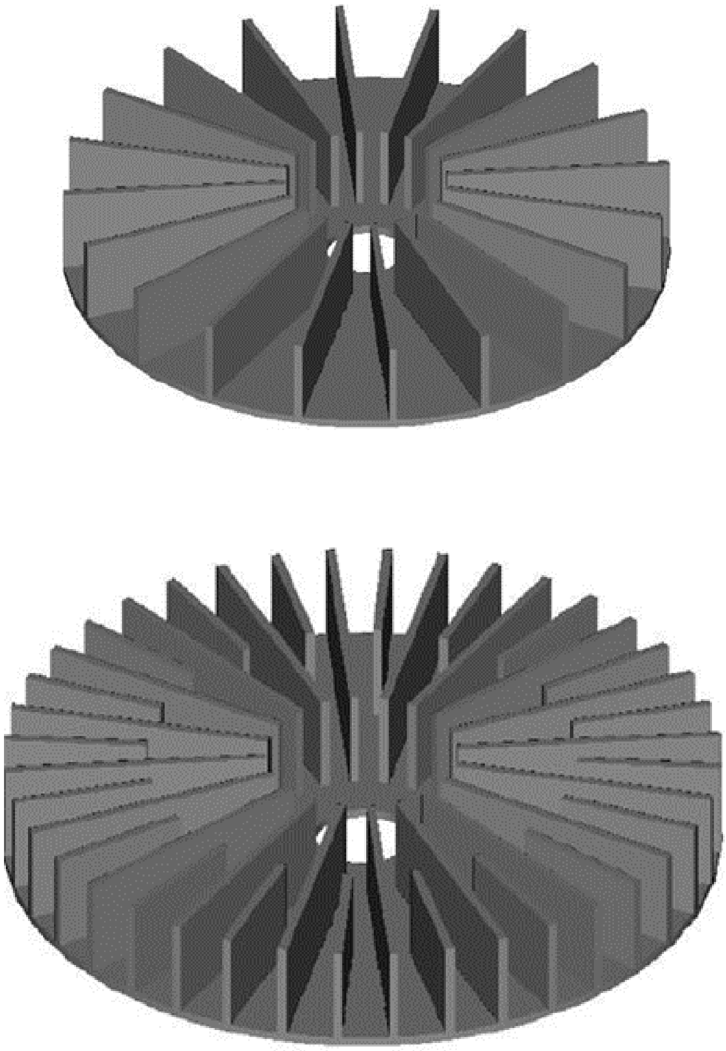


Fig.12: Radial microchannel based heat sink

Although there have been many studies on various geometries for the case of parallel channels, very few studies have been done on radial heat sink. In this project we have proposed and numerically analyzed a new radial curved microchannel based heat sink in an attempt to combine the advantages of curved channel and radial sink using ANSYS Fluent package.

CHAPTER- 3

ANALYSIS

3.1 Fluid Dynamics

In engineering, fluid dynamics is one of the most challenging areas of study and research. It is a sub disciple of the fluid mechanics, and describes any fluid flow– gases and liquids. Aerodynamics, hydrodynamics are some of the disciples that are included in the branch of fluid dynamics. In the real world, we are surrounded by fluid from all the directions and fluid dynamics helps us in providing a systematic structure, to construct, describe and model any fluid flow. Any fluid flow problem can be solved using three basic equations:

1. Continuity equation or mass conservation
2. Navier Stokes equations or the momentum equation
3. Energy equation or conservation of Energy

Any problem can be analyzed in three basic ways

1. Analytical Approach
2. Experimental approach
3. Numerical approach

Each of the above ways has its own merits and demerits. The analytical approach uses different laws/theories that are associated with the equation and provides us an exact solution but this approach can be applied to only selective problems and these problems are formulated in an idealized manner in order to provide an exact solution. Hence this approach cannot be used for majority of real-life fluid flow problems.

The experimental approach justifies its name, by doing a variety of experiments, any phenomena or problem is solved. It does provide an edge over the previous approach as this one is the most reliable and depicts the real-life problems. But these experiments can be very expensive and has a higher probability of human error, in order to avoid the error, the expense increases.

Numerical approach is an approach to solve any fluid problem using the various numerical techniques. It doesn't provide a proper and exact answer but it provides us a rough idea of the solution. It is the least expensive, encounters least human errors and also provides near to exact solutions.

3.2 <u>CFD</u>

CFD or computational fluid dynamics is branch of fluid dynamics that uses a set of numerical techniques and applies it to fluid flow problems to obtain an approximate solution. CFD may not be classified as science as it just combines 2 different disciplines into one, numerical analysis and fluid flow or heat transfer. Even though the solutions provided by CFD are not exact yet this method is used for every fluid problem where experimental and analytical approach may not function properly. The major governing equation of CFD is Navier Stokes equation. There are two approaches that can be applied to solve equations using CFD:

1. Writing your own CFD code : for solving a particular problem , a CFD code can be written down and then solved in MATLAB to provide the solution
2. Using commercial CFD softwares : ANSYS, CFX, COMSOL are some of this software's which take inputs from the user and then solve it.

3.3 <u>ANSYS Fluent</u>

ANSYS Fluent is one of the engineering simulation software that is very popular and powerful in solving basic fluid models. The ANSYS Fluent solver incorporates finite volume method by dividing the domain into many discrete control volumes. All the general conservation equation discussed earlier is solved on these sets of control volumes. We will use ANSYS Fluent in our project to solve the engineering problems.

To solve any problem using ANSYS Fluent, there are some necessary steps to be followed

1. Pre analysis
2. Geometry
3. Mesh

4. Physical setup
5. Numerical solution
6. Verification and validation

Rather than going into depth of each and every step, we would explain the steps we took into the analysis of our fluid flow problem.

1. Geometry

The geometry has been designed in CREO and then transferred to the ANSYS workbench as a step file and then the analysis is preceded.

2. Meshing

The quality of mesh cannot be determined by performing visual inspection. It is determined by effectively by looking at various statistics, such as maximum skewness, number of nodes and elements. For perfecting the meshing of the geometries, a minimum size of 0.05 µm is chosen through the sizing method.

3. Physical Setup

It is done in solver analysis. Various inputs like boundary conditions, physics involved etc. are provided in order to depict the real solution we expect.

The double precision is marked in order to improve the accuracy of our solution to expect least errors. Out of the two solver techniques available: pressure based and density bases, we worked on the pressure based. Pressure based solver is used because the fluids density remains constant throughout. It handles Mach number in the range of 0 to 4. In order to solve problems involving high Mach numbers we can switch to density-based solver but as the Mach number in our solver is restricted under the range hence pressure based solver is used.

The model used is viscous laminar, as we expect our fluid problem to be under the laminar range and the energy equation is also switched on. The fluid used is water liquid. Condition for constant inlet velocity is applied throughout every model and a constant heat flux.

4. Solution method

We are provided with solution methods like SIMPLE, SIMPLEC, PISO AND COUPLED. The SIMPLE algorithm is one of the most straightforward and easiest algorithms used in various CFD procedures. An improved version is SIMPLEC which uses the pressure corrections also. The SIMPLEC algorithm corrects the velocity field which results in correcting the pressure fields, unlike the SIMPLE algorithm. The number of calculation involved in SIMPLEC model is 30% more yet it provides a reduced computer time by 50%. We have used SIMPLEC model in order to get a better calculation and reduced time. The PISO and coupled models can also be used in case we obtain divergence in the SIMPLEC model. But we obtained convergence in all our cases hence the SIMPLEC model was used in our solution method.

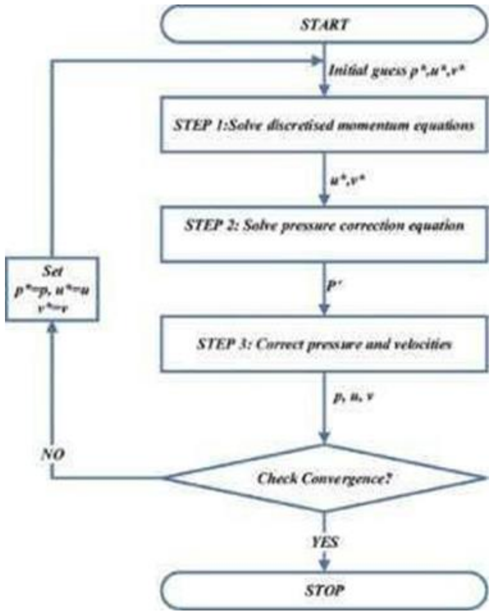


Fig.13: Flowchart for simulation in ANSYS

5. Hybrid Initialization and Iteration

Hybrid initialization generally makes guesstimates and thus often provides accurate results for complex geometries in lesser iterations. We provided 100 iterations for each model. And the solver setting ends.

6. Results

Various results of different temperature distribution and velocity is then obtained from the models and discussed further in the report.

7. Convergence Criteria

The convergence criteria for the residuals for continuity, x velocity, y velocity and energy are taken to be 10^{-6} which gives very accurate and consistent results.

CHAPTER- 4

VALIDATION

4.1 Fluid flow in Curved Microchannel

Paper title: “Experimental and numerical study on the flow characteristics in curved rectangular microchannels.”

Authors: “Jiann Cherng Chu, Jyh Tong Teng, Ralph Greif.”

The dimensions of the microchannel are tabled below.

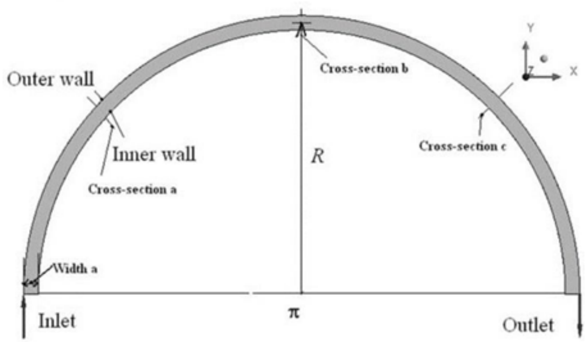


Fig.14: Schematic of curved channel

Channel No.	Width, w ($x10^{-6}$m)	Height, h ($x10^{-6}$m)	R (10^{-6}m)	Dh ($x10^{-6}$m)	Aspect Ratio, s	curvature ratio, l
C1	200	200	5000	200	1	0.04

Table.2: dimensions of curved MC

Boundary Conditions:

1. Stationary wall condition on the other 3 faces.
2. Constant inlet velocity as per the desired mass flow rate.
3. Atmospheric pressure outlet.
4. Convergence criteria in x,y,z velocity, continuity to be 10^{-6}.
5. Hybrid initialization from inlet.

Mesh Report

Table3. Mesh Information for curved MC

Domain	Nodes	Elements
cu_substrat_cu	2595450	2357646
water__2_water__1	1116060	1008882
All Domains	3711510	3366528

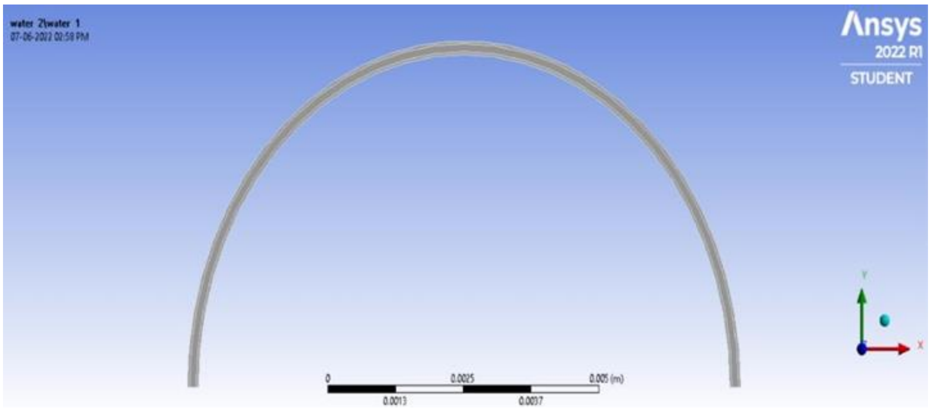


Fig.15: Geometry of Curved Microchannel in design modeler

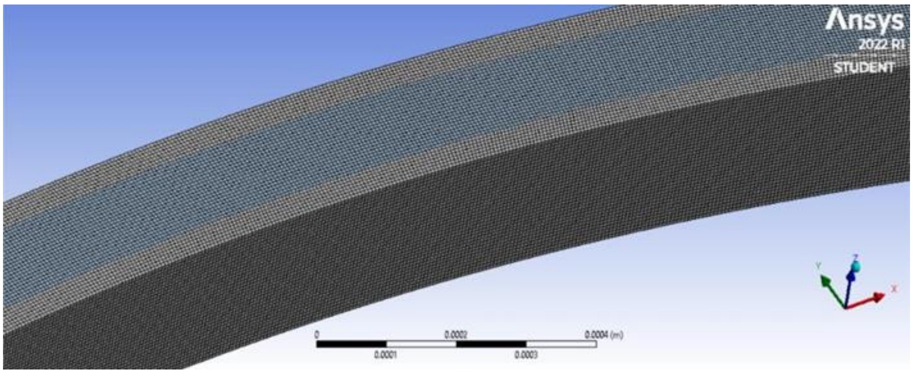


Fig.16: Mesh quality on bends of curved MC

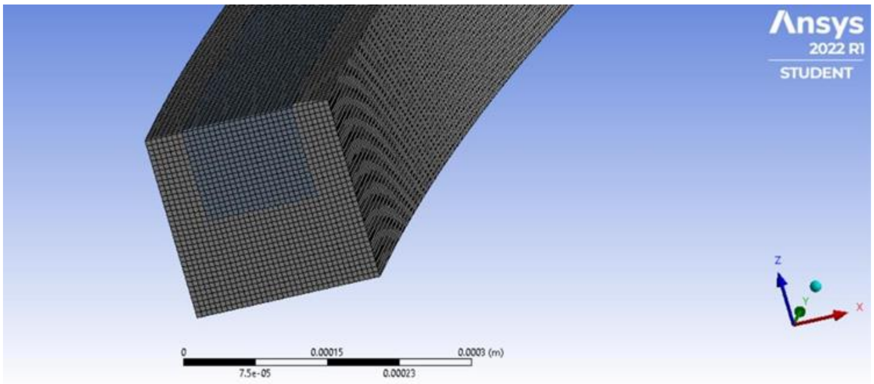


Fig.17: Mesh quality at inlet of curved MC

Physics Report

Table 4. Domain Physics for curved MC

Domain - cu_substrat_cu	
Type	FLUID
Location	cu_substrat_cu
Domain - water__2_water__1	
Type	FLUID
Location	water__2_water__1

Table 5. Boundary Physics for curved MC

Domain	**Boundaries**	
cu_substrat_cu	**Boundary - contact_region src**	
	Type	INTERFACE
	Location	contact_region-src
	Boundary - heat_source_wall	
	Type	WALL
	Location	heat_source_wall
	Boundary - wall cu_substrat_cu	
	Type	WALL
	Location	wall-cu_substrat_cu

water__2_water 1	**Boundary - inlet**	
	Type	VELOCITY-INLET
	Location	inlet
	Boundary - moving_wal	
	Type	WALL
	Location	moving_wal
	Boundary - outlet	
	Type	PRESSURE-OUTLET
	Location	outlet
	Boundary - stationary_wall contact_region trg	
	Type	INTERFACE
	Location	stationary_wall-contact_region-trg

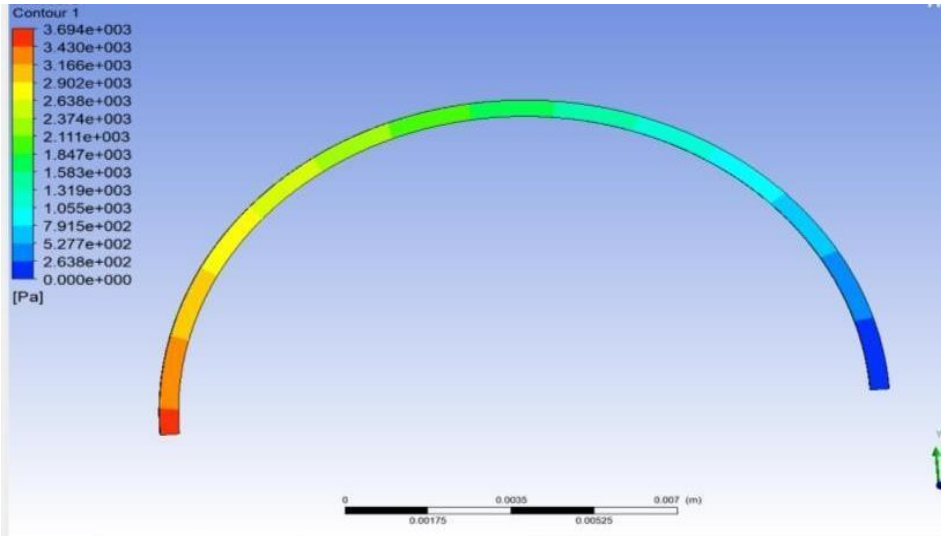
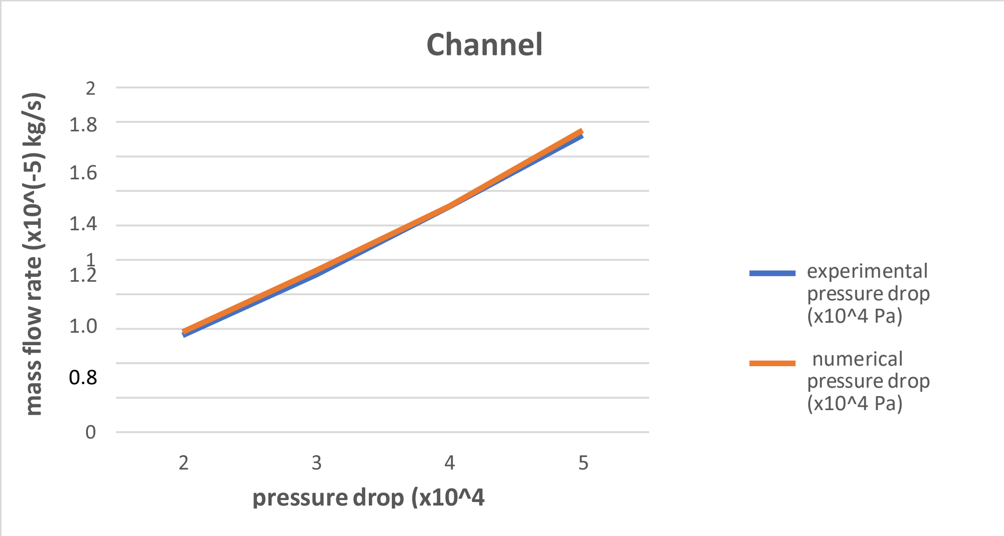


Fig.18: Pressure variation in curved MC

Channel	Mass flow rate ($x10^{-5}$ kg/s)	Experimental Pressure drop ($x10^{4}$ Pa)	Numerical pressure drop ($x10^{4}$ Pa)	% Error
C1	2	0.5618	0.5813	-3.47099
C1	3	0.9127	0.938	-2.772
C1	4	1.3089	1.3095	-0.04584

Table 6: Pressure drop data for curved MC C1



Graph 1: Pressure drop validation curve for C1

4.2 Heat transfer in Curved Microchannel

Paper title: "Experimental and numerical study on the flow characteristics in curved rectangular microchannels,"
Authors: "Jiann Cherng Chu, Jyh Tong Teng, Ralph Greif."

The dimensions of the microchannel are tabled below.

Channel No.	Width, w ($x10^{-6}$m)	Height, h ($x10^{-6}$m)	R (10^{-6}m)	Dh ($x10^{-6}$m)	Aspect Ratio, s	curvature ratio, l
C1	200	200	5000	200	1	0.04

Table.7: Dimension of curved heat sink

Boundary Conditions:

1. Constant heat flux on the bottom wall of 100 W/cm^2.
2. Stationary wall condition on the other 3 faces.
3. Constant inlet velocity as per the desired mass flow rate.
4. Atmospheric pressure outlet.
5. Convergence criteria in x,y,z velocity, continuity, energy to be 10^{-6}.

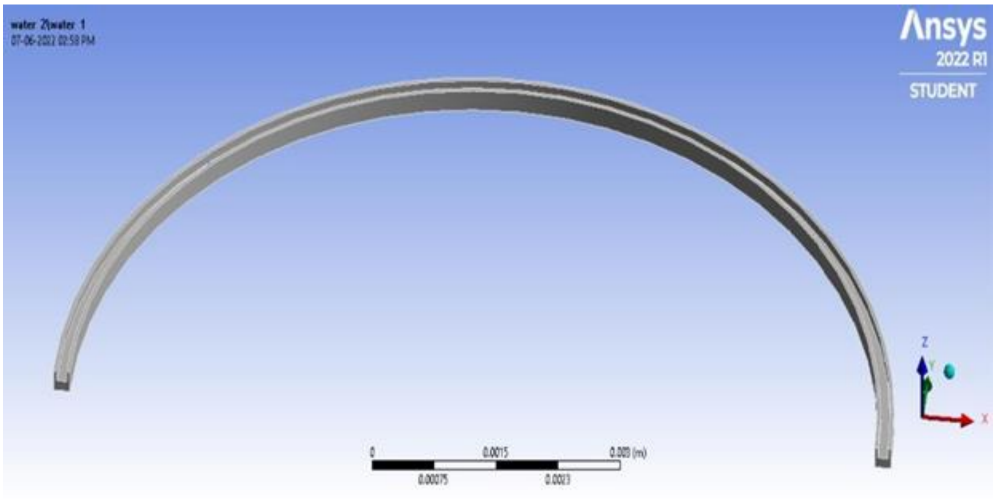


Fig.19: Cu substrate for curved MC

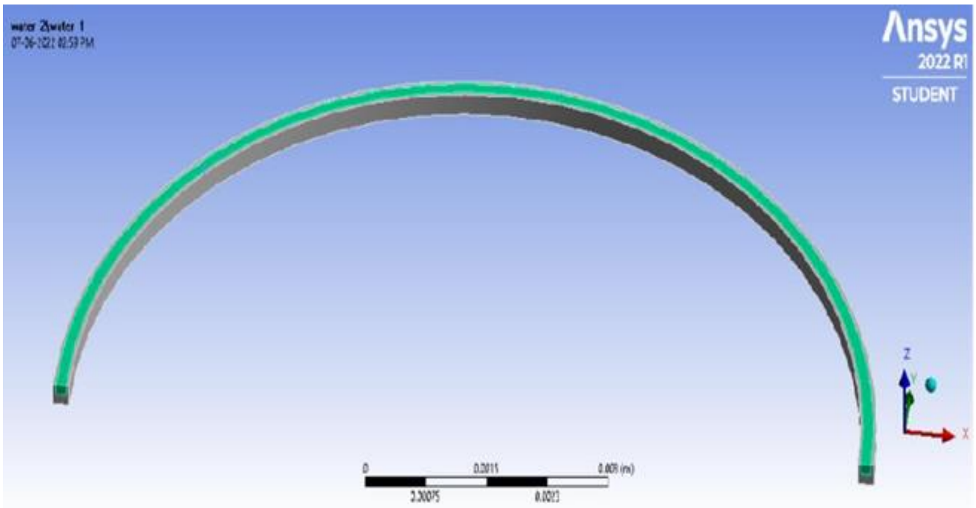


Fig.20: Cu substrate with water domain

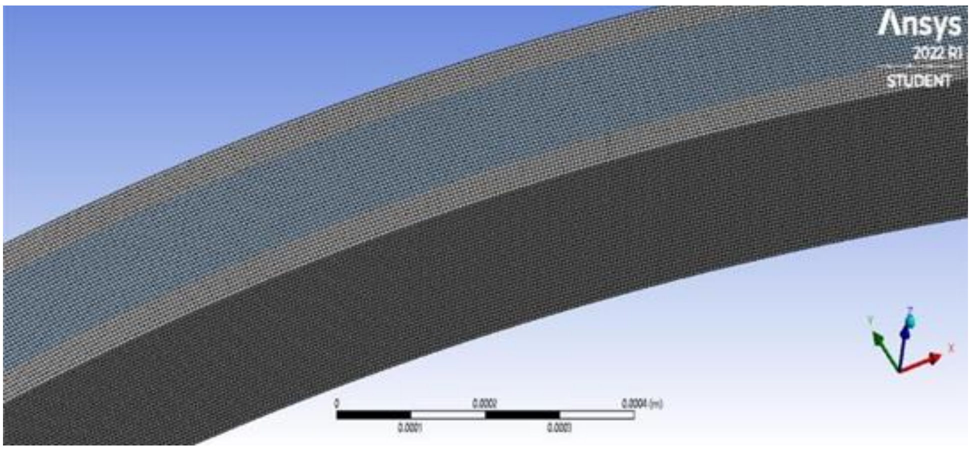


Fig.21: Mesh quality at bend for curved heat sink

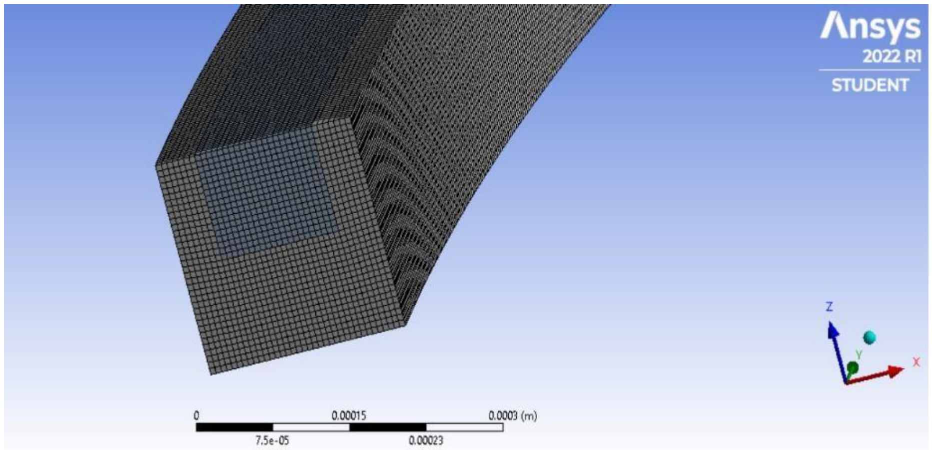


Fig.22: Mesh quality at inlet for curved heat sink

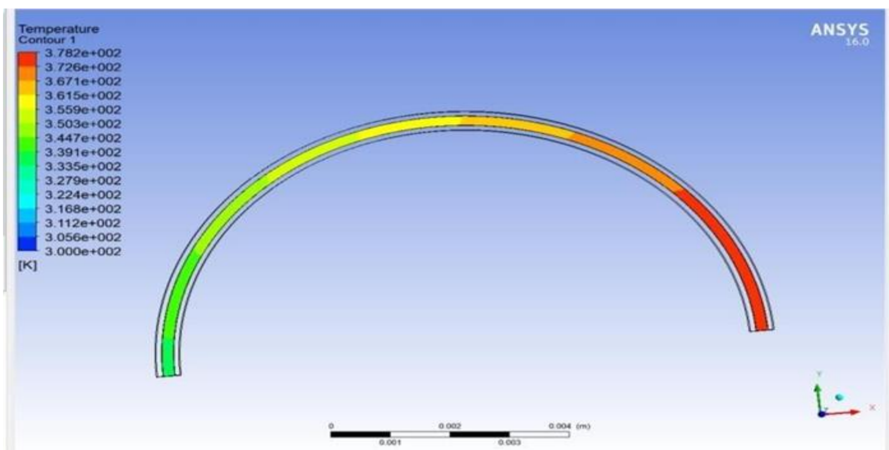
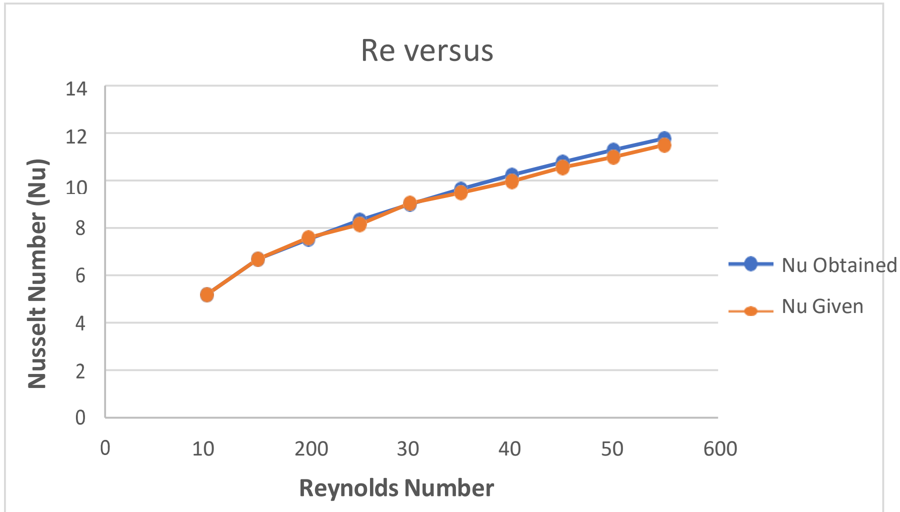


Fig.23: Temperature variation for curved heat sink

Re	Nu Obtained	Nu Given in paper	%error
99.52143569	5.189678535	5.188658	-0.01967
149.5409093	6.687582625	6.68276	-0.07217
199.0428714	7.517038777	7.58621	0.911802
249.201675	8.320937923	8.15172	-2.07586
298.5643071	8.999512001	9.03448	0.38705
348.3250249	9.630995934	9.48621	-1.52628
398.0857428	10.21906487	9.96552	-2.54422
447.8464606	10.76923129	10.5483	-2.09447

Table.8: Nusselt number values for curved MC heat sink

Graph 3: Nusselt number validation curve for curved MC heat sink

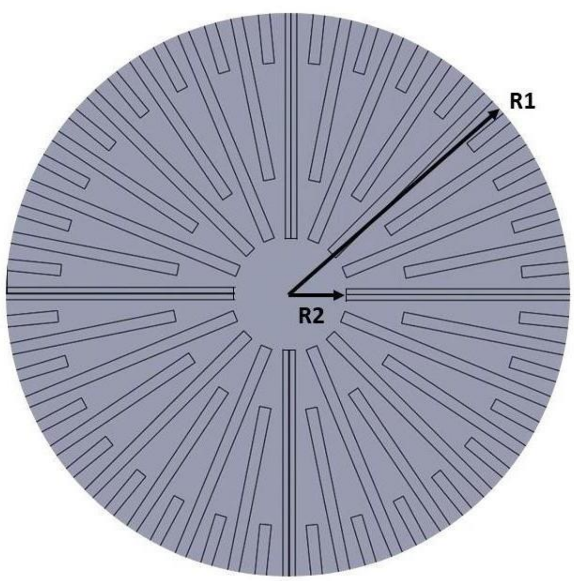


CHAPTER- 5

RADIAL HEAT SINKS

5.1 Radial Straight Microchannel Heat Sink

Radial straight microchannel based heat sink consists number of straight radially diverging microchannels. The inlet of the fluid is at the center and fluid flows radially outwards. At the periphery extra fins are added to increase the surface area for heat dissipation. The design of radial straight microchannel based heat sink is shown below.



R2= 12.7 mm
R1= 28.7 mm
Channel height= 6 mm
Channel width= 1 mm
Footprint area= 831 mm^2

Fig.24: Design of radial straight MC heat sink

For the ease of simulation and reducing the computation time only 1/16th of the entire circular domain is considered. The domain selected for analysis is shown below.

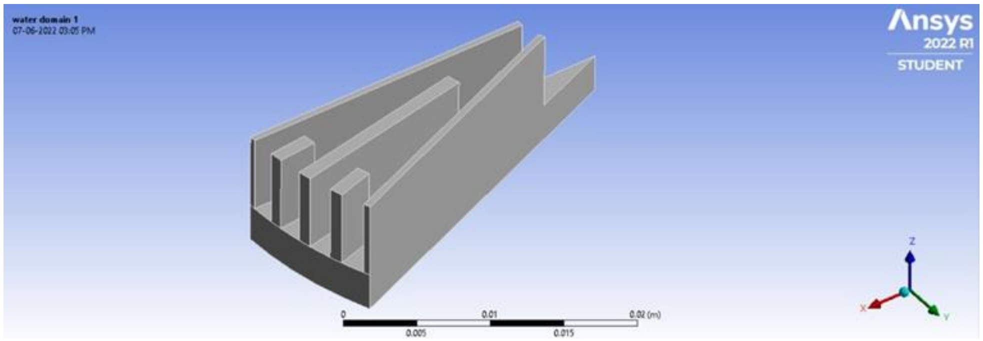


Fig.25: Design of 1/16th radial straight MC heat sink

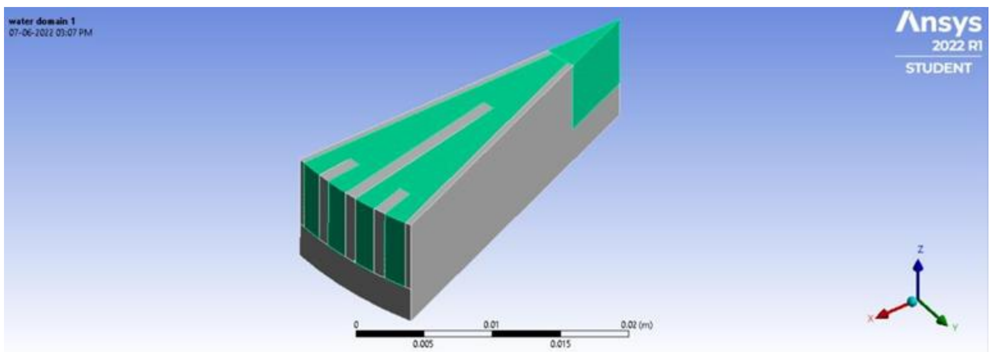


Fig.26: Design of 1/16th radial straight MC heat sink with water domain

<u>Boundary Conditions</u>:

1. Heat sink material: Copper
2. Fluid medium: water
3. Constant heat flux on the bottom wall of heat sink 100 W/cm^2.
4. Constant inlet velocity as per the desired mass flow rate.
5. Atmospheric pressure outlet.
6. Convergence criteria in x,y,z velocity, continuity, energy to be 10^{-6}.
7. Hybrid initialization from inlet.
8. Conjugate analysis is done with silicon substrate.
9. Periodic conditions on the outer right and left wall.

Mesh Report

Table.9. Mesh Information for radial straight MC

Domain	Nodes	Elements
partbody_pad.2	14820	66566
water_domain2	7137	5448
water_domain_1	728	480
All Domains	22685	72494

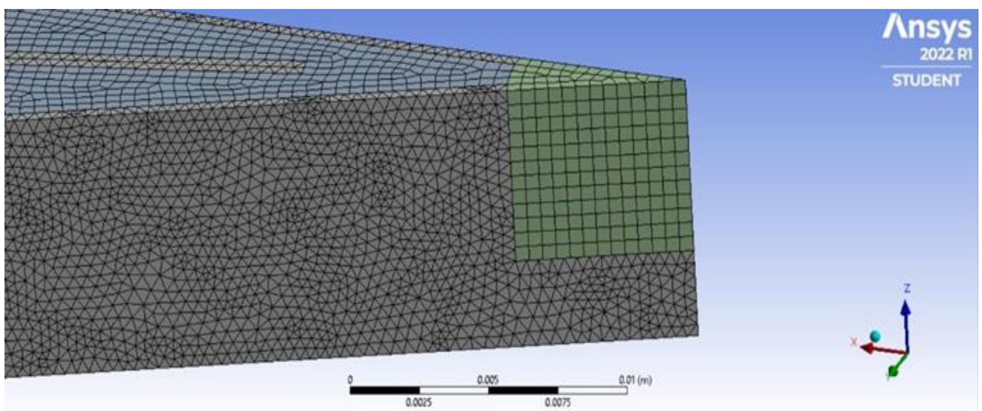


Fig.27: Mesh quality for 1/16th radial straight MC heat sink with water domain

Physics Report

Table.10. Domain Physics for radial straight MC

Domain - partbody_pad.2	
Type	FLUID
Location	partbody_pad.2
Domain - water_domain2	
Type	FLUID

Location	water_domain2
Domain - water_domain_1	
Type	FLUID
Location	water_domain_1

Table.11. Boundary Physics for radial straight MC

Domain	Boundaries	
partbody_pad.2	**Boundary - contact_region contact_region_2 contact_region_2 contact_region_3 src partbody_pad.2**	
	Type	INTERFACE
	Location	contact_region-contact_region_2-contact_region_2-contact_region_3-src-partbody_pad.2
	Boundary - heat_source_wall	
	Type	WALL
	Location	heat_source_wall
	Boundary - wall partbody_pad.2	
	Type	WALL
	Location	wall-partbody_pad.2
water_domain2	**Boundary - contact_region contact_region_2 contact_region_2 contact_region_3 src water_domain2**	
	Type	INTERFACE
	Location	contact_region-contact_region_2 contact_region_2-contact_region_3-src-water_domain2
	Boundary – contact_region contact_region_2 contact_region_2 contact_region_3 trg water_domain2	
	Type	INTERFACE
	Location	contact_region-contact_region_2-contact_region_2-contact_region_3-trg-water_domain2

	Boundary – moving_wall	
	Type	WALL
	Location	moving_wall
	Boundary – outlet	
	Type	PRESSURE-OUTLET
	Location	outlet
water_domain_1	**Boundary – contact_region contact_region_2 contact_region_2 contact_region_3 trg water_domain_1**	
	Type	INTERFACE
	Location	contact_region-contact_region_2-contact_region_2-contact_region_3-trg-water_domain_1
	Boundary – inlet	
	Type	VELOCITY-INLET
	Location	inlet
	Boundary – wall water_domain_1	
	Type	WALL
	Location	Wall-water_domain_1

Simulation results for Radial Straight Heat Sink:

Inlet Vel (m/s)	Tin (K)	Tout (K)	Tmean (K)	Twall (K)	A total (m^2)	InletA (m^2)	Inlet Perimeter (m)
0.15	300	330.92	315.46	402.53	0.000541	5.0766E-06	0.01215
0.2	300	323.40	311.70	388.07	0.000541	5.0766E-06	0.01215
0.25	300	319.07	309.53	376.35	0.000541	5.0766E-06	0.01215

Table.12: Simulation results for radial straight MC

Nusselt Number Calculation:

Sensible Heat Gain by Cooling water (q) = $\rho \times C \times Q \times (T_{out} - T_{in})$

Average Heat Transfer Coefficient (h) = (q) / $\{A_{tot} \times (T_{wall} - T_{mean})\}$

Nusselt Number (Nu) = $(h \times D_h) / K_f$

ρ- Density of water = 1000 kg/m^3

C- Specific heat capacity of water = 4182 J/kg-K

Q- Discharge rate of water = Inlet Area × Inlet Velocity (m^3/s)

D_h- Characteristic Dimension of Channel = 4 × Inlet Area / Inlet Perimeter

K_f- Thermal Conductivity of water at T_{mean}

(At 315.4602 K, Kf = 0.634 W/m-K)

(At 311.7039 K, Kf = 0.627 W/m-K)

(At 309.5396 K, Kf = 0.625 W/m-K)

The Nusselt number and Pressure drop values for radial straight microchannels are:

Inlet velocity (m/s)	Twall (K)	Pressure drop (kPa)	Nusselt No (Nu)
0.15	402.53	11.2589	5.9111
0.2	388.07	18.9872	6.8327
0.25	376.35	28.9638	7.9733

Table.13: Simulation results for radial straight heat sink

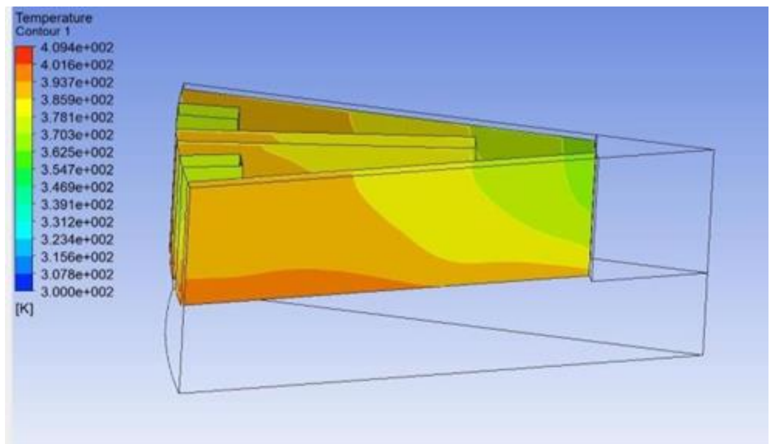


Fig.28: Temperature variation at side wall for 1/16th radial straight MC heat sink with water domain

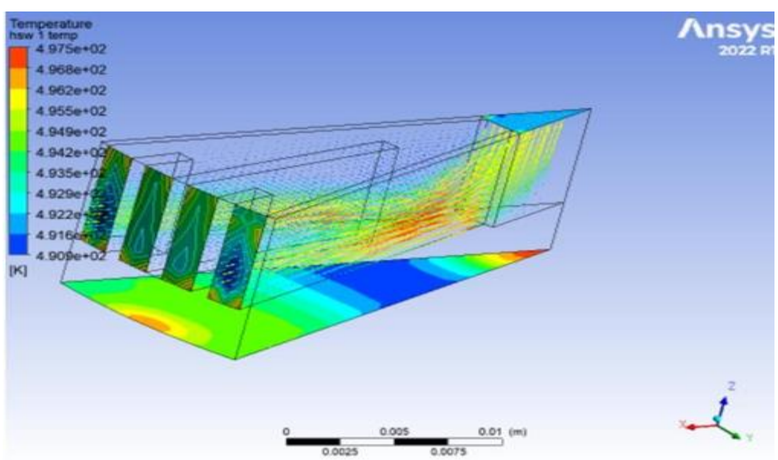
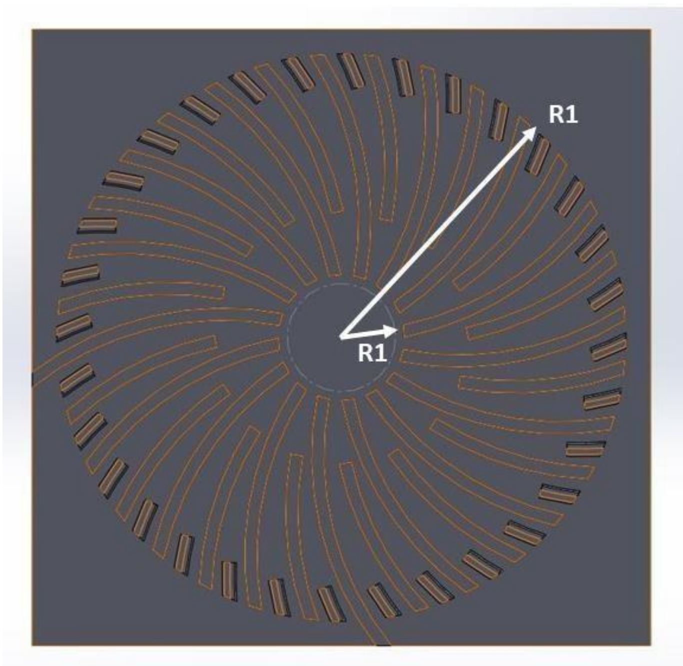


Fig.29: Temperature variation at bottom surface for 1/8th radial straight MC heat sink with water domain

5.2 <u>Radial Curved Microchannel Heat Sink</u>

Radial straight microchannel based heat sink consists number of straight radially diverging microchannels. The inlet of the fluid is at the center and fluid flows radially outwards. At the periphery extra fins are added to increase the surface area for heat dissipation. This is an attempt to combine the benefits of radial heat sink and curved microchannel. The design of radial straight microchannel based heat sink is shown below.



R2= 12.7 mm

R1= 28.7 mm

Channel height= 6 mm

Channel width= 1 mm

Footprint area= 831 mm^2

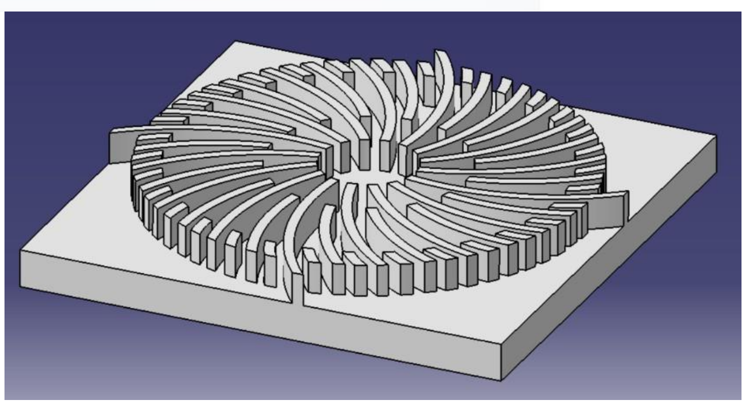


Fig.30: Design of radial curved MC heat sink

Mesh Report

Table 14. Mesh Information for FFF 1

Domain	Nodes	Elements
cu_pad.3	7893	35941
wd1_pocket.1	2400	1620
wd2_pad.5	560	360
All Domains	10853	37921

For the ease of simulation and reducing the computation time only 1/16th of the entire circular domain is considered. The domain selected for analysis is shown below.

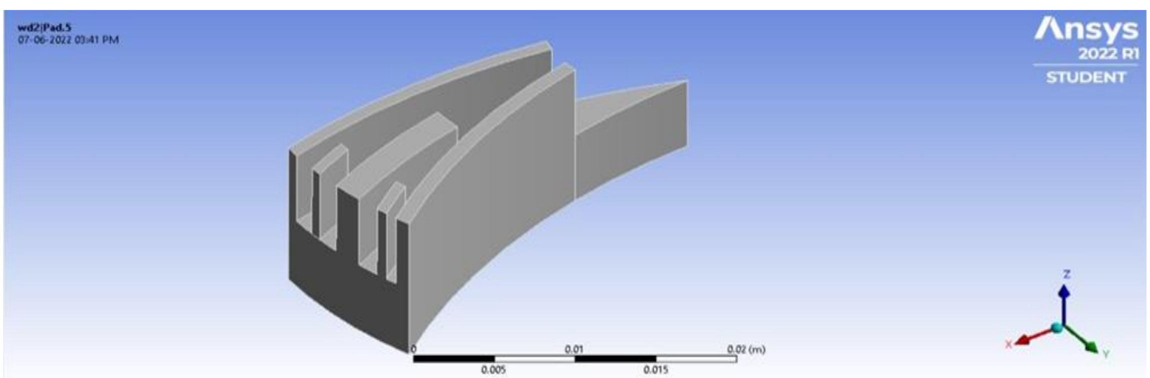


Fig.31: Design of 1/16th radial curved MC heat sink

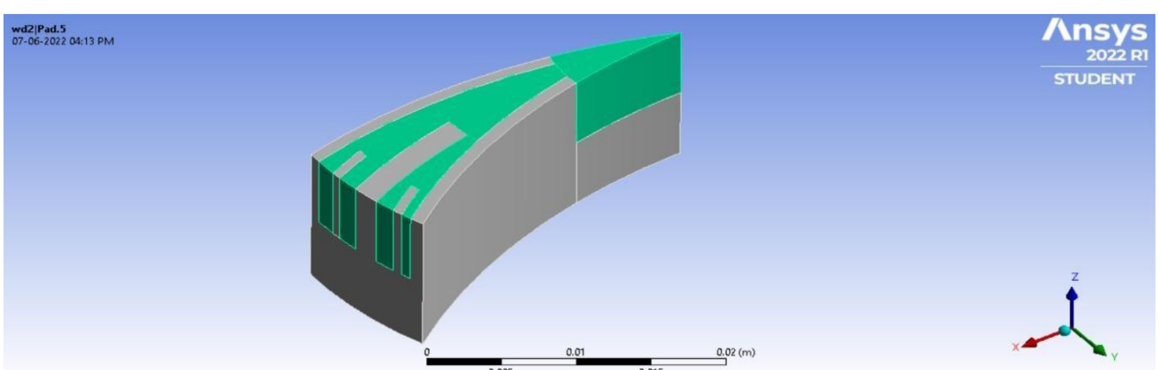


Fig.32: Design of 1/16th radial curved MC heat sink with water domain

Boundary Conditions:

1. Heat sink material: Copper
2. Fluid medium: water
3. Constant heat flux on the bottom wall of heat sink 100 W/cm^2.
4. Constant inlet velocity as per the desired mass flow rate.
5. Atmospheric pressure outlet.
6. Convergence criteria in x,y,z velocity, continuity, energy to be 10^{-6}.
7. Hybrid initialization from inlet.
8. Conjugate analysis is done with silicon substrate.
9. Periodic conditions on the outer right and left wall.

Physics Report

Table.15. Domain Physics for radial curved MC

Domain - cu_pad.3	
Type	FLUID
Location	cu_pad.3
Domain - wd1_pocket.1	
Type	FLUID
Location	wd1_pocket.1
Domain - wd2_pad.5	
Type	FLUID
Location	wd2_pad.5

Table.16. Boundary Physics for radial curved MC

Domain	**Boundaries**	
cu_pad.3	**Boundary - contact_region contact_region_2 src**	
	Type	INTERFACE

	Location	contact_region-contact_region_2-src
	Boundary - hsw	
	Type	WALL
	Location	hsw
	Boundary - lw	
	Type	WALL
	Location	lw
	Boundary - rw	
	Type	WALL
	Location	rw
	Boundary - wall cu_pad.3	
	Type	WALL
	Location	wall-cu_pad.3
wd1_pocket.1	**Boundary - contact_region contact_region_2 trg wd1_pocket.1**	
	Type	INTERFACE
	Location	contact_region-contact_region_2-trg-wd1_pocket.1
	Boundary - contact_region_3 src	
	Type	INTERFACE
	Location	contact_region_3-src
	Boundary - movingwall2	
	Type	WALL
	Location	movingwall2
	Boundary - outlet	
	Type	PRESSURE-OUTLET
	Location	outlet
wd2_pad.5	**Boundary - contact_region contact_region_2 trg wd2_pad.5**	
	Type	INTERFACE

	Location	contact_region-contact_region_2-trg-wd2_pad.5
	Boundary - contact_region_3 trg	
	Type	INTERFACE
	Location	contact_region_3-trg
	Boundary - inlet	
	Type	VELOCITY-INLET
	Location	inlet
	Boundary - movingwall1 contact_region contact_region_2 trg	
	Type	INTERFACE
	Location	movingwall1-contact_region-contact_region_2-trg
	Boundary - wall wd2_pad.5	
	Type	WALL
	Location	wall-wd2_pad.5

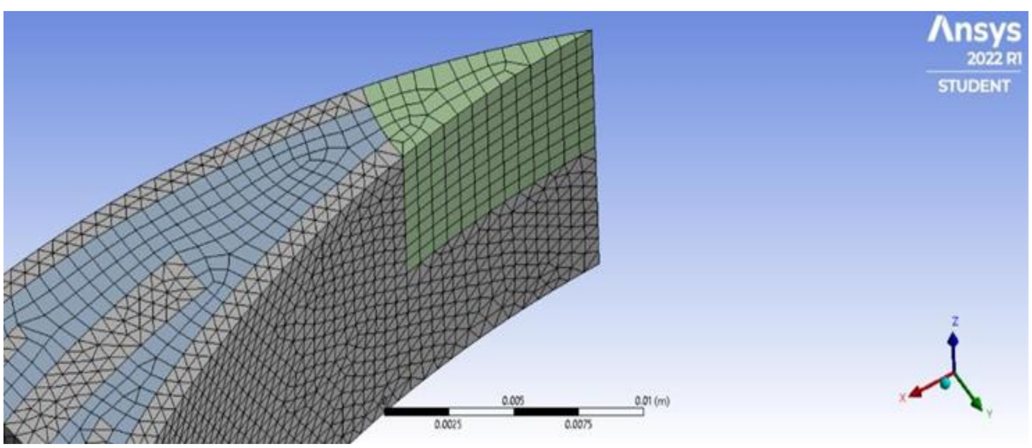


Fig.33: Mesh quality of 1/16th radial curved MC heat sink with water

Simulation results for Radial Curved Heat Sink:

Inlet Vel (m/s)	Tin (K)	Tout (K)	Tmean (K)	Twall (K)	A total (m^2)	Inlet A (m^2)	Inlet Perimeter (m)
0.15	300	329.36	314.68	391.30	0.000553467	5.0766E-06	0.01217
0.2	300	323.40	311.70	379.55	0.000553467	5.0766E-06	0.01217
0.25	300	319.36	309.68	369.72	0.000553467	5.0766E-06	0.01217

Table.17.: Simulation results for radial curved MC

Nusselt Number Calculation:

Sensible Heat Gain by Cooling water (q) = $\rho \times C \times Q \times (T_{out} - T_{in})$

Average Heat Transfer Coefficient (h) = $(q) / \{A_{tot} \times (T_{wall} - T_{mean})\}$

Nusselt Number (Nu) = $(h \times D_h) / K_f$

ρ- Density of water = 1000 kg/m^3

C- Specific heat capacity of water = 4182 J/kg-K

Q- Discharge rate of water = Inlet Area × Inlet Velocity (m^3/s)

D_h- Characteristic Dimension of Channel = 4 × Inlet Area / Inlet Perimeter

K_f- Thermal Conductivity of water at T_{mean}

The Nu and Pressure drop values for Radial Curved Microchannels are:

Inlet velocity (m/s)	Twall (K)	Pressure drop (kPa)	Nusselt No (Nu)
0.15	391.30	12.2987	6.2873
0.2	379.55	20.3838	7.4991
0.25	369.72	30.9183	8.7997

Table.18: Simulation results for radial curved heat sink MC

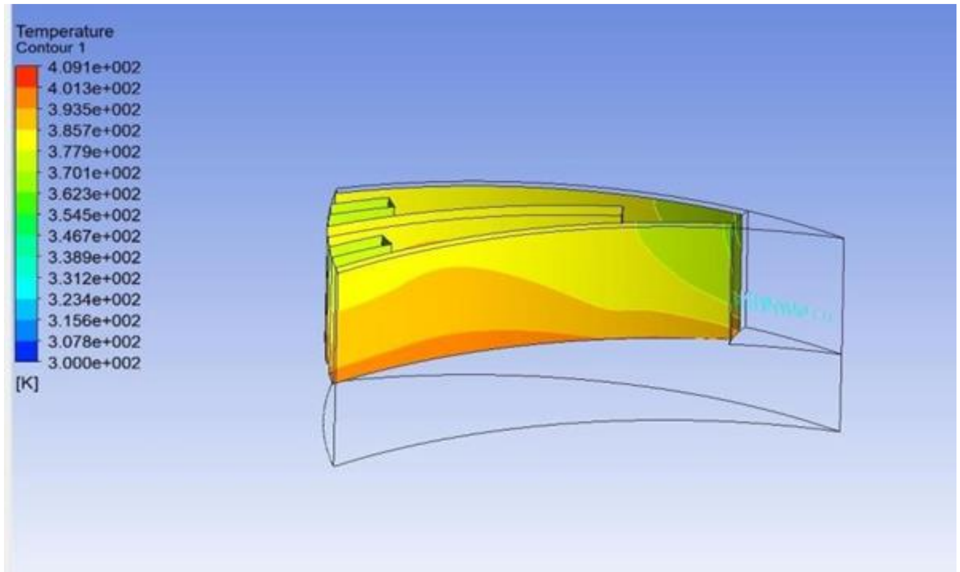


Fig.34: Temperature variation at side wall for 1/16th radial curved MC heat sink with water domain

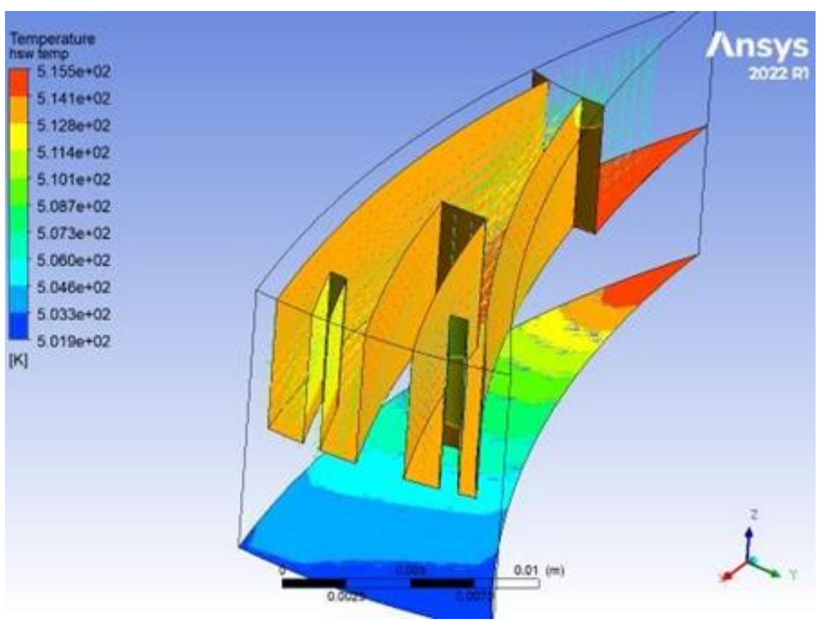


Fig35: Temperature variation at base for 1/16th radial curved MC with water domain

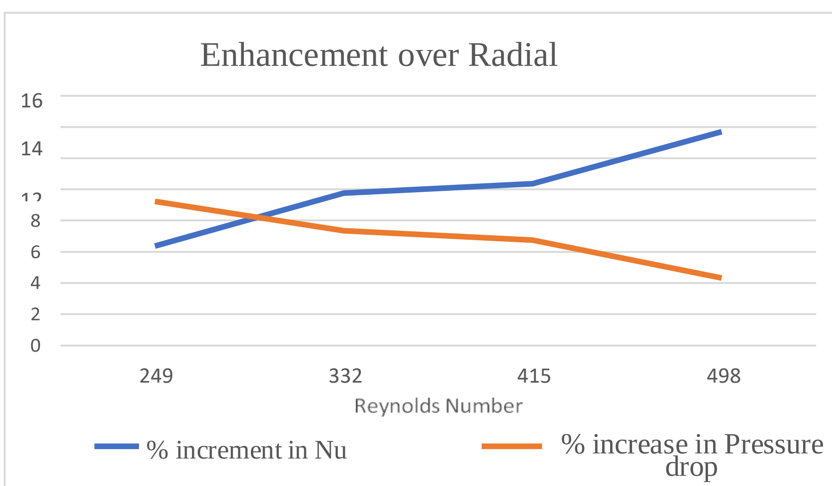
CHAPTER- 6

RESULTS AND CONCLUSIONS ON RADIAL STRAIGHT AND RADIAL CURVED MICROCHANNEL HEAT SINKS

Tabulated below is the data comparing Nusselt number values and pressure drop values of radial straight and radial curved microchannel based heat sinks.

Re	Nu Straight	Nu Curved	% Increment In Nu	Pressure Drop Straight (kPa)	Pressure Drop Curved (kPa)	% increase in Pressure drop
249	5.9111	6.2873	6.36	11.2589	12.2987	9.23
332	6.8327	7.4991	9.75	18.9872	20.3838	7.35
415	7.9733	8.7997	10.36	28.9638	30.9183	6.74

Table.19: Comparison between radial straight and radial curved heat sinks



Graph 4: Increase in Nu values and pressure drop in radial curved heat sink wrt radial straight heat sink

The above graph clearly states that except for very low Reynolds number the enhancement in Nusselt number is more than the pressure drop penalty. Hence, radial curved microchannel based heat sink gives better thermo-hydraulic performance compared to radial straight microchannel based heat sink.

To determine the overall thermal and flow performance of microchannel heat sink Liu and Zhang [9] suggested use of a non dimensional parameter known as thermal enhancement formulated as

$$\eta = k.\,\mathrm{Nu} \,/\, f^{1/3}$$

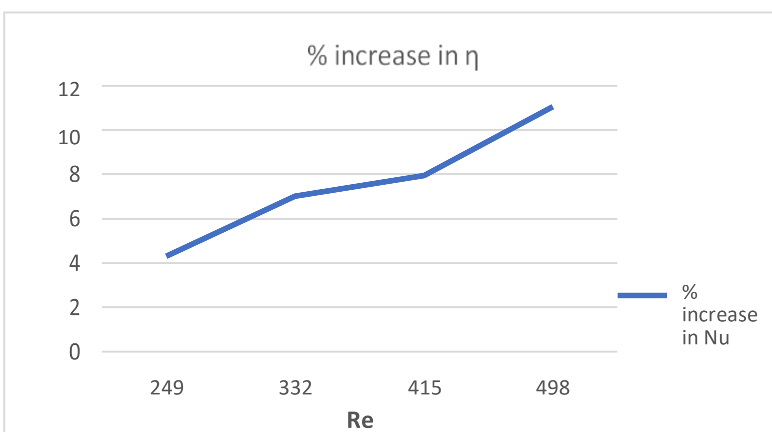
$$k = 0.873$$

Where k can be taken as constant for radial straight and radial curved heat sinks.

The results are tabulated below.

Re	**Nu Straight**	**Nu Curved**	**f for straight**	**f for curved**	**η for straight**	**η for curved**	**% increase in η**
249	5.9111	6.2873	0.899	0.954	6.122483	6.385769	4.3003141
332	6.8327	7.4991	0.668	0.721	7.805777	8.353924	7.02232462
415	7.9733	8.7997	0.575	0.615	9.570788	10.33092	7.94220915

Table.20: Comparison of η between radial straight and radial curved heat sinks



Graph 5: Increase in η with respect to Re

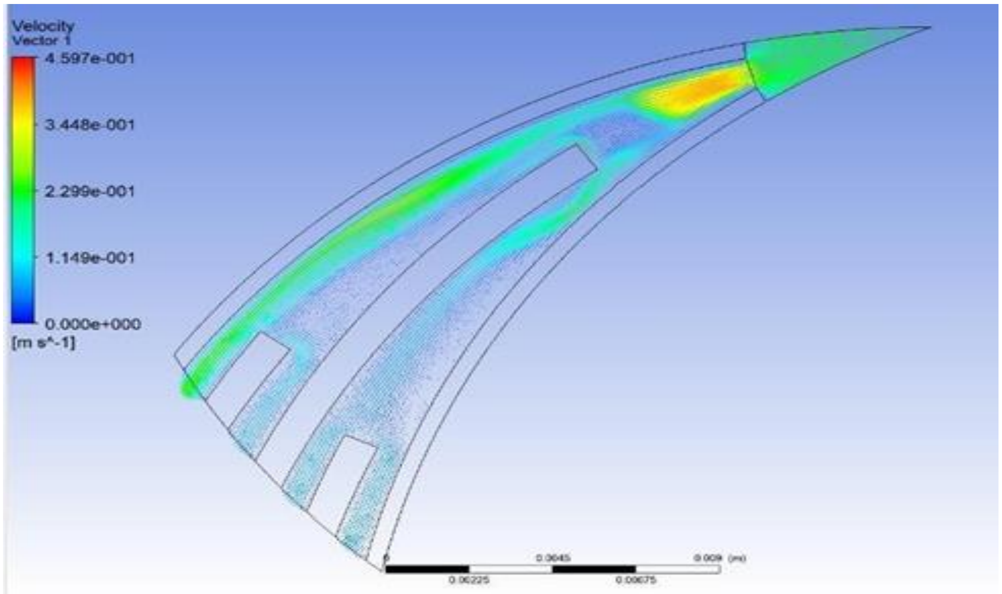


Fig.36: Velocity vector at mid plane in the fluid domain for radial curved heat sink

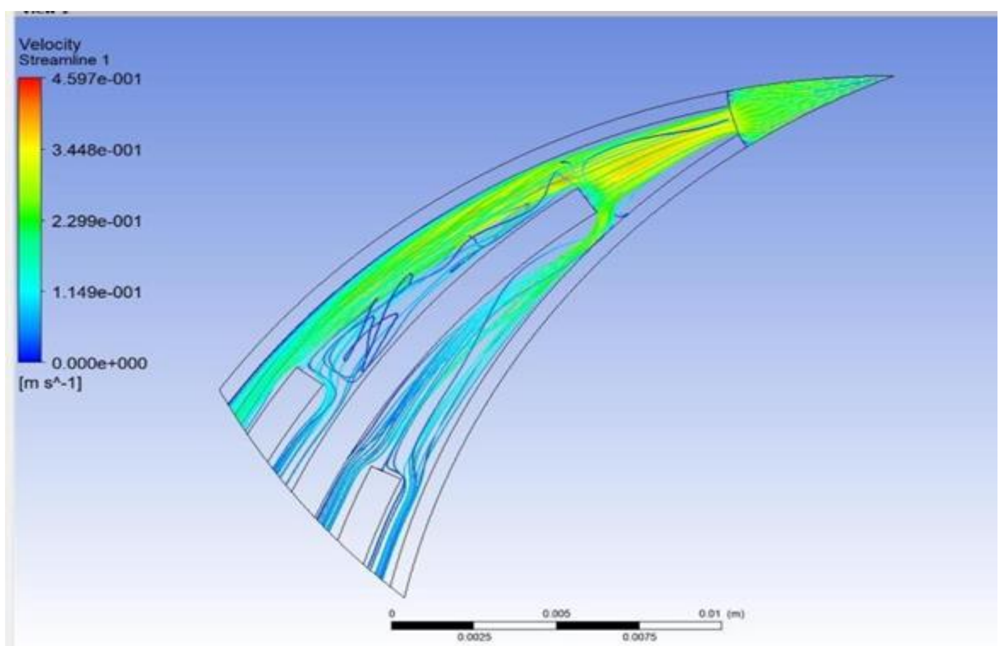


Fig.37: Streamlines for Radial Curved heat sink

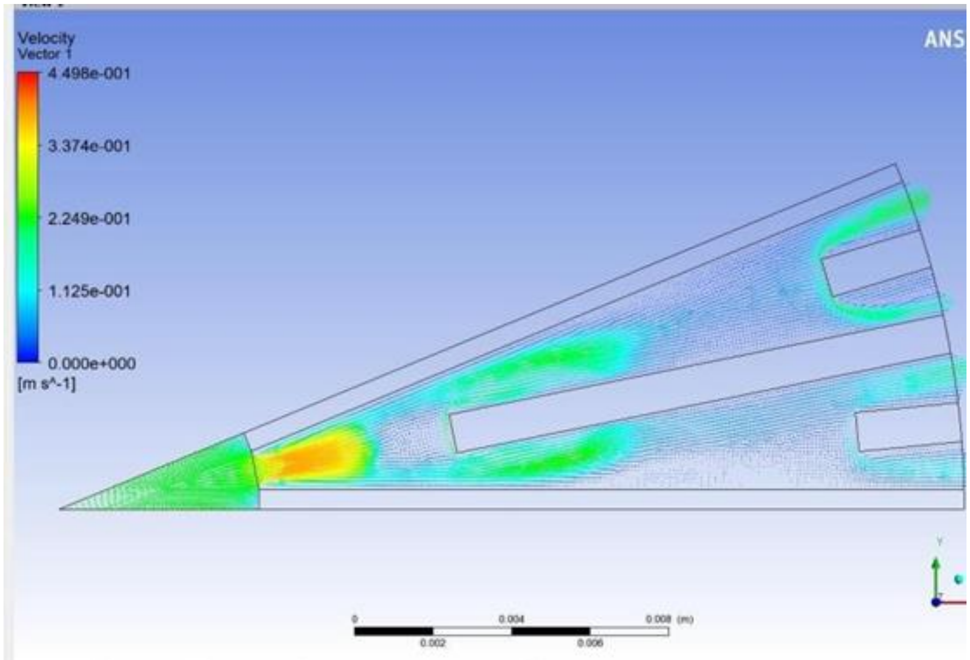


Fig.38: Velocity vector at mid plane in the fluid domain for radial straight heat sink

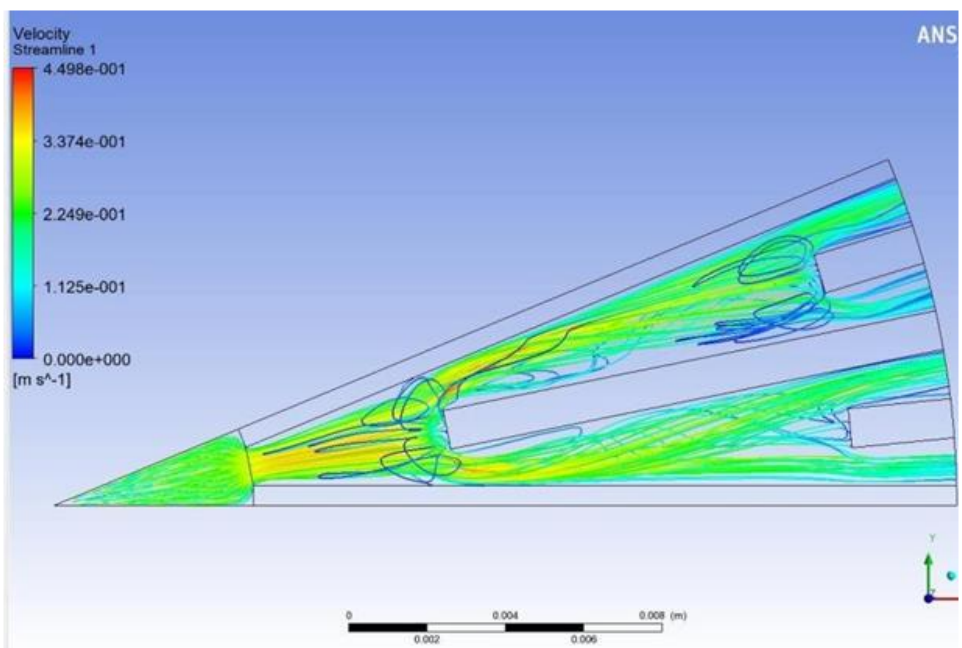


Fig.39: Streamlines for radial straight heat sink

Conclusion :

The graph 5 clearly states that radial curved microchannel based heat sink gives better overall performance and the enhancement in performance increases with increase in Reynolds number.

The post-processing images (Fig 36, 37, 38, 39) show that in case of curved channels the flow gets changed due to centrifugal force arising because of curvature. The flow tries to get concentrated at the outer wall. This causes pressure difference and leads to formation of secondary flows. It is also clear from the streamlines that there is better mixing of fluid elements in case of radial curved channels. This promotes better heat transfer. Because of increased velocity near the walls there isincrease in the skin friction coefficient too. This is the main reason for increase in pressure drop. But as mentioned above the penalty of pressure drop is compensated by even more enhancement in Nusselt number values. Hence it can be concluded that radial curved microchannel based heat sinks perform better than radial straight microchannel based heat sink.

Scope of further work:

There is a lot of scope of work in this field of microchannel heat sinks. Various other geometries of microchannels can also be studied like spiral, wavy, zig-zag. Research work can also be done on various fluid coolents that can be used in different geometries of microchannels. Studies can also be done on diiferent materials that can be used for manufacturing the microchannels.

CHAPTER- 7

CPV AND DEVELOPMENT OF 3D CAD MODEL OF CPV WITH RADIAL HEAT SINK

7.1 Introduction to CPV

Solar photovoltaic is a commercial technology to convert solar energy from sun to electrical energy. However lower conversion efficiency and higher cost are reasons behind Concentrator Photovoltaic (CPVs) as an emerging technology. The conversion efficiencies are as high as 40% with lower costs however heating of cells prove to be a major concern due to higher concentration ratios. The temperature increase causes decrease in efficiency and lifespan for cells. Various attempts are currently being made for effective cooling of CPVs by keeping factors such as reliability, lower cell temperatures, easier operation and efficiency as a priority.

The concentration ratio (CR) is referred to as ‘suns’ which can be classified as-

Low CPV Systems- (1-40) suns

Medium CPV Systems- (40-300) suns

High CPV Systems- (300-2000) suns

Increase in CR causes increase in cell average temperature resulting in decrease of conversion efficiency. This necessitates the need of cooling of CPVs by using effective methods. Low and medium CPV systems may not need active cooling methods however higher CPV systems generate higher temperatures that are necessary to be cooled by active methods.

The CPVs consist of Fresnel lens or small reflecting mirrors which are lower in cost and are used to focus the sunlight from 200-1000 suns of C.R. This ultimately decrease the amount of materials used in normal PV cells by 200 to 1000 suns. It is also found that multi junction cells are also much more efficient with respect to the single cells. It is also found that for same illumination of sunlight on PV cell and CPV of similar material then CPV are more efficient only due to the concentration.

Nowadays, the commercially available CPVs utilize upto 200-300 suns. However, the future demands indicate utilization of CPVs with even 1000 suns of C.Rs which the necessitates the presence of trackers(rotating about one-axis or two axis and hence called one axis and two axis trackers) to keep the sun focused on the solar cell.

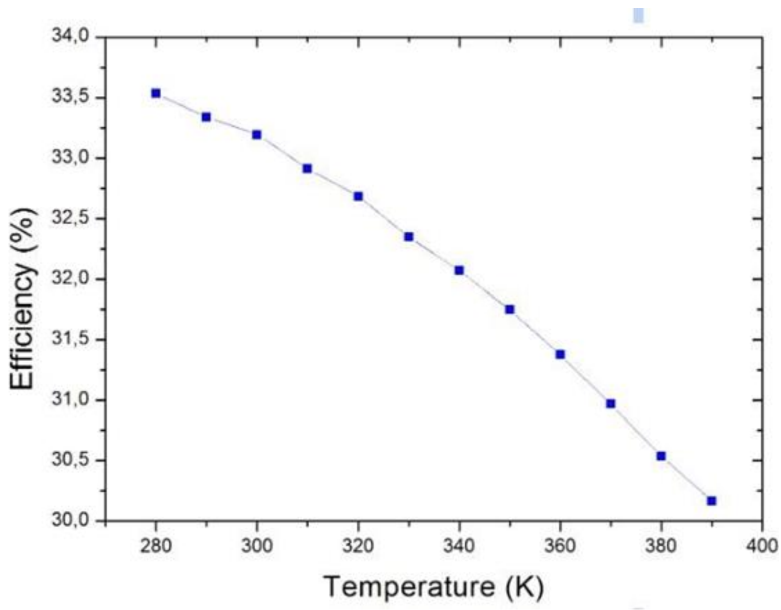
The average temperature of the CPV plays a crucial role in deciding the efficiency of CPV and it has to be kept lower than the melting point of die material and other attachment material present in the cell. Various studies in correlations between conversion efficiency and operating temperature have shown an average decrease of 0.45% in efficiency with per degree Celsius increase in operating temperature.

At low Concentration ratios for the CPVs, generally single silicon cells are used. However, there is also option of utilizing highly efficient non silicon cells which are currently being used for space applications. These are also known as multi junction III-V cells primarily because the elements used in these cells are from the third and fifth columns from the periodic table namely gallium and arsenide. However, these are expensive for large PV panels but can be used in future for CPVs due to the reduction in area.

Multi-junction Solar Cells

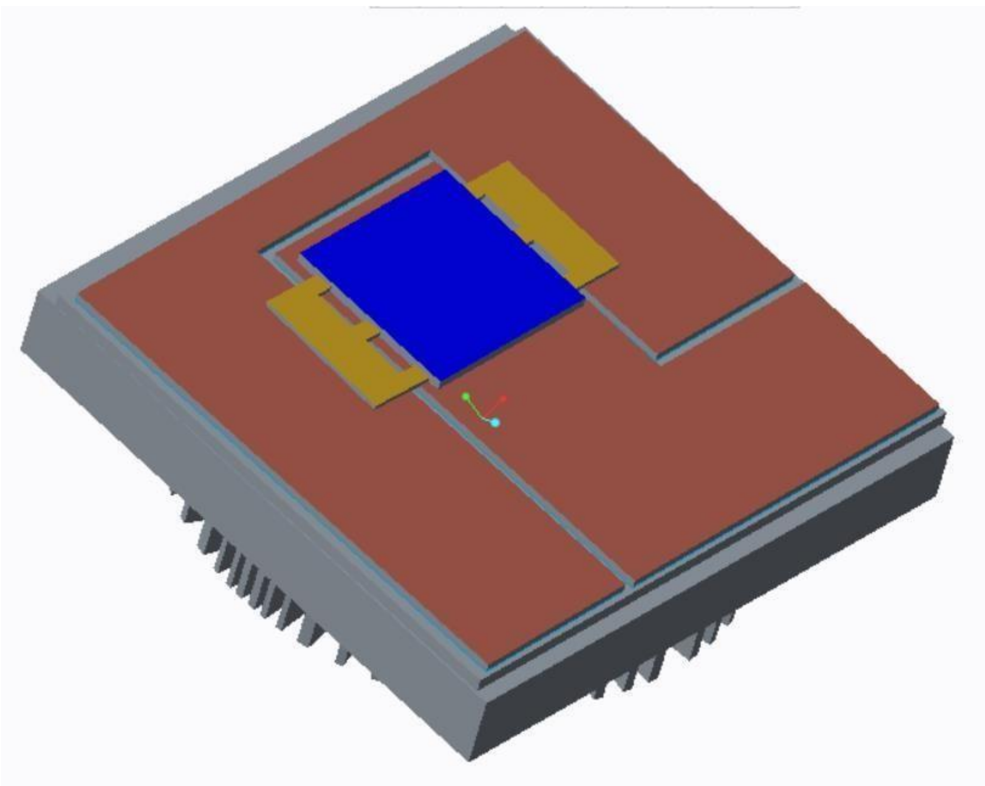
Multi-junction (MJ) solar cells are solar cells with multiple p–n junctions made of different semiconductor materials. Each material's p-n junction will produce electric current in response to different wavelengths of light. The use of multiple semiconducting materials allows the absorbance of a broader range of wavelengths, improving the cell's sunlight to electrical energy conversion efficiency.

Multi-junction solar cells are the combination of cells compiled layer by layer, and each layer absorbs different wavelength to maximize the spectrum of absorption and also to increase conversion efficiency of sunlight to electricity. In current market, normal CPVs are not being commercially manufactured and instead High concentrator photovoltaics are seen as future of solar technology due to its higher efficiency which is feasible with respect to the cost incurred in the manufacturing process. Also, the presence of 4 junctions can improve the efficiency of CPV to 46% with respect to the 34% in case of the 3 junction CPV.



Graph 5: Variation of efficiency of CPV module with temperature

7.2 <u>CAD Model of CPV with Microchannel Heat Sink</u>



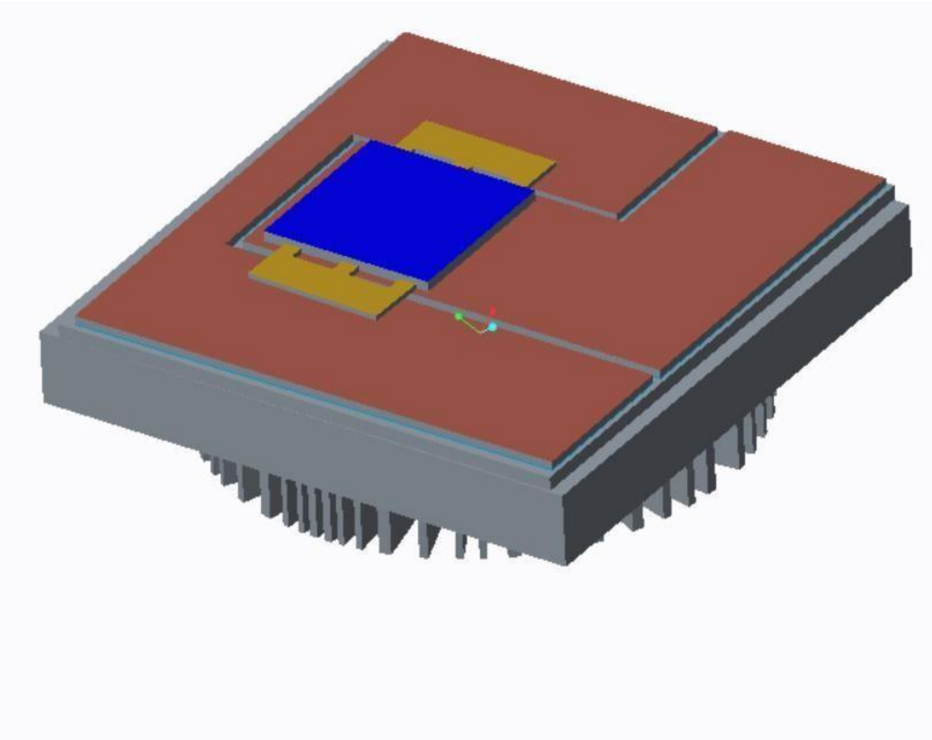


Fig.40: 3D CAD model of CPV module with heat sink

7.3 <u>System Model Description</u>

1. <u>Solar Cell</u>: The plate is made up of semiconductor material like silicon or germanium which absorbs the photo energy from the sunlight and converts it into electrical energy. The larger surface area produces higher output. It is the most significant and the costliest component of the PV system. Conventional PV operates at one sun concentration while CPV operates at about 200-300 suns concentration.

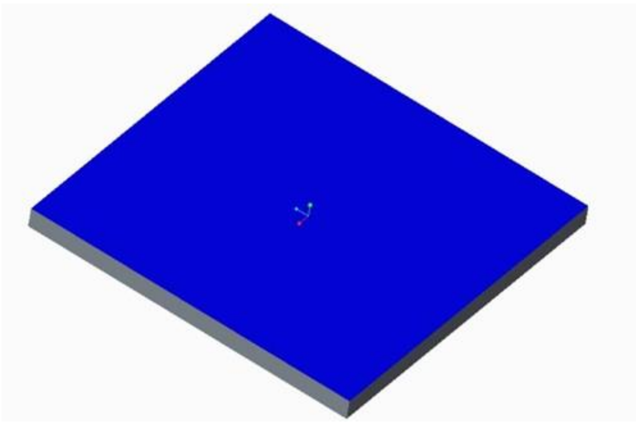


Fig.41: 3D CAD model of solar cell

2. Interconnector: It connects the solar plate to the copper plate and allows the flow of energy from cell to the copper plate which further gives output power. The solar cell is sandwiched between two interconnectors which hold it in position. Also interconnector reduces the power loss by channelizing the energy.

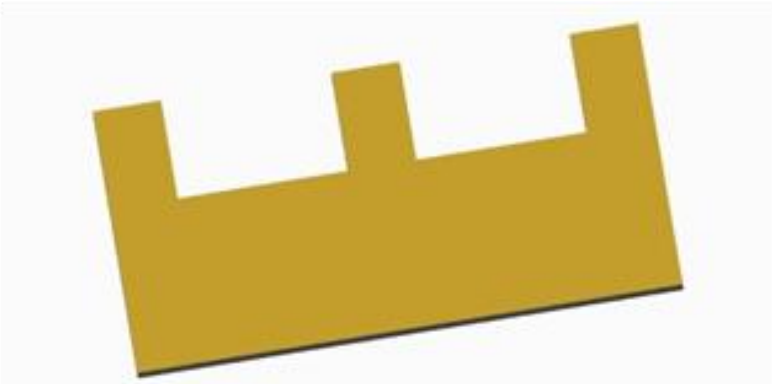


Fig.42: 3D CAD Model of Interconnector

3. Copper Layer: Two copper layers are placed in a manner for optimum thermal conduction. It extracts the excess heat from the cell and pass it to the heat sink for heat removal.

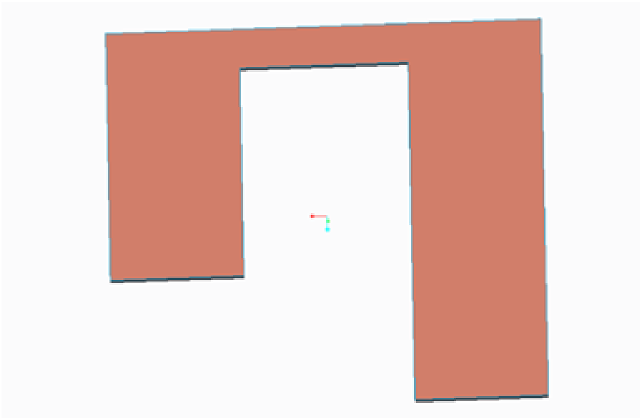


Fig.43: 3D CAD model of copper layer

4. Ceramic Layer: This layer is placed under copper plate. On the other side of layer, heat sink is attached. It is the main contributor to thermal resistance of the stack and to the thermal misalignment with the cell material. It provides base support to the stack.

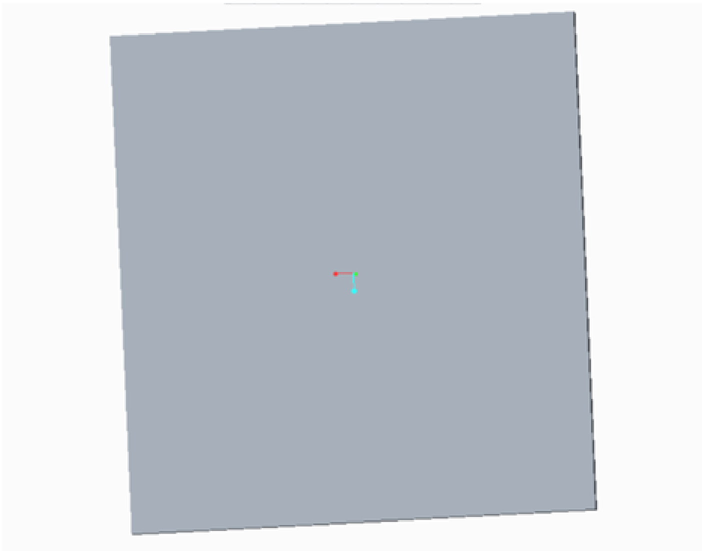


Fig.44: 3D CAD model of ceramic layer

5. <u>Radial Microchannel</u>: The microchannel is attached to the other side of the ceramic layer and facilitates extensive heat removal. It can be of various types:- multi-layer microchannel, radial straight microchannel, radial curved microchannel. The fluid enters in axial direction and flows out through the radial channels.

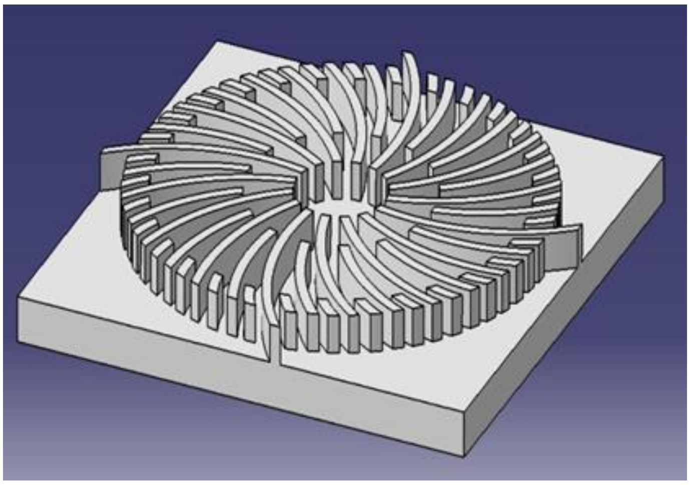


Fig.45: 3D CAD model of heat sink

REFERENCES

[1] D.B. Tuckerman, R.F.W. Pease, High-performance heat-sinking for VLSI, IEEE Electr. Dev. L. 2 (5) (1981) 126–129.

[2] Chu, J. C., Teng, J. T. & Greif, R., Experimental and numerical study on the flow characteristics in curved rectangular microchannels, Applied Thermal Engineering Vol. 30, pp. 1558-1566 (2010).

[3] J.F. Guo, M.T. Xu, L. Cheng, Second law analysis of curved rectangular channel, Int. J. Therm. Sci. 50 (2011) 760–768.

[4] 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.

[5] V.S. Duryodhan, A. Singh, S.G. Singh, A. Agrawal, Convective heat transfer in diverging and converging microchannels, Int. J. Heat Mass Transf. 80 (2015) 424–438.

[6] 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.

[7] Youmin Xi, J. Y. (2010). Single-phase flow and heat transfer in swirl microchannels. Experimental Thermal and Fluid Science 34, 1309–1315.

[8] Muwanga, R., Hassan, I., and Ghorab, A., 2008, "Numerical investigation of a radial microchannel heat exchanger with varying cross-sectional channels," Journal of Thermophysics and Heat Transfer, 22, pp. 321-332.

[9] Xiaogang Liu, Meng Zhang, Zhongyi Wang, Juhui Chen, 2020, “Numerical Analysis of Fluid Flow and Heat Transfer in Micro-Channel Heat Sinks with Double-Layered Complex Structure”

[10] Numerical simulation of copper heat spreader used in Electronics chip cooling Swapnil M. Patel, Mehul P. Bambhania. PG Student, Department of Mechanical Engineering, Faculty of Technology & Engineering, MSU-Baroda, India.