CFD ANALYSIS OF INCLINED JET MICRO CHANNEL IMPINGMENT HEAT SINK

In partial fulfillment of the requirement of the degree of

MASTERS OF TECHNOLOGY
IN
THERMAL ENGINEERING

Submitted by:
RAHUL KUMAR
(**Roll No. : 2K14/THE/16**)

Under the guidance of
Mr. MD. ZUNAID
Asst. Professor

DEPARTMENT OF MECHANICAL ENGINEERING
DELHI TECHNOLOGICAL UNIVERSITY

(2014-2016)

CERTIFICATE

DELHI TECHNOLOGICAL UNIVERSITY

(Formerly DELHI COLLEGE OF ENGINEERING)

Date:-__________

This is to certify that report entitled **"CFD Analysis of inclined jet micro channel impingement heat sink"** by **MR. RAHUL KUMAR is** the requirement of the partial fulfillment for the award of Degree of **Master of Technology (M.Tech)** in **Thermal Engineering** at **Delhi Technological University, New Delhi.** This work was completed under my supervision and guidance. He has completed his work with utmost sincerity and diligence. The work embodied in this project has not been submitted for the award of any other degree to the best of my knowledge.

Mr. MD. ZUNAID
Asst. Professor
Department of Mechanical Engineering
DELHI TECHNOLOGICAL UNIVERSITY, DELHI

ACKNOWLEDGEMENT

First of all, I would like to express my gratitude to God for giving me ideas and strengths to make my dreams true and accomplish this thesis. I ask God that this thesis and the published works to be a useful guide to help other researchers towards a green world that is utilized by sustainable energy systems.

To achieve success in any work, guidance plays an important role. It makes us put right amount of energy in the right direction and at right time to obtain the desired result. Express my sincere gratitude to my guide, **Mr. MD. ZUNAID,** Asst. Professor, Mechanical Engineering Department for giving valuable guidance during the course of this work, for his ever encouraging and timely moral support. His enormous knowledge always helped me unconditionally to solve various problems.

I am greatly thankful to **Dr. R.S. MISHRA**, Professor and Head, Mechanical Engineering Department, Delhi Technological University, for his encouragement and inspiration for execution of the this work. I express my feelings of thanks to the entire faculty and staff, Department of Mechanical Engineering, Delhi Technological University, Delhi for their help, inspiration and moral support, which went a long way in the successful completion of my report work.

RAHUL KUMAR

(2K14/THE/16)

ABSTRACT

In this study inclined jet micro channel impingement heat sink used for electronic cooling application several particles and staggered inclined micro-jet configuration will be used for study. The effectiveness of inclined micro jet configuration will be analyzing at various flow rates for the maximum temperature-rise and pressure drop. The shape optimization of a micro-channel heat sink with a grooved structure has been performed using a multi objective evolutionary algorithm.

In the study inclined jet micro channel impingement focus on electronic cooling in micro level. In the study of inclined jet impingements focus on 3D micro channel for fluid flow ansys. The jets used have been placed at an inclination of 30^0,45^0,60^0 from the top of the surface .during this process copper has been used as the solid and water has been used as a fluid. The optimum results were obtained by using the jets at an inclination of 45^0. This study simulated six basic materials in inclined jet micro channel at 45^0 at water as a fluid. During study simulation carried out on copper, steel, nickel, silicon, titanium, and aluminum. After the simulation copper show maximum temperature drop through the metal. During the simulation difference in temperature and heat transfer coefficient define maximum heat transfer through the micro channel.

In the study using four fluids in micro channel such as water, DIUF- water. HF-7200 and performance fluid (PF- 5060). During analyzing we have obtained optimum result through micro channel. Performance fluid gave maximum temperature drop through micro channel. During the simulation Nusselt number (Nu) show with respect to temperature drop. In the study of incline jet micro channel impingement heat sink all conclusion are made with respect to the fluid flow through in it. Flow define the characteristics of fluid such laminar or turbulent. During the iteration we have justified the solid material such as copper maximum heat transfer from the micro channel. Performance fluid provided maximum temperature drop through micro channel. It is a compactible study through the micro channel. For the best result study on Nano fluid used in micro channel. In the simulations we used Nano fluid mixture of water and Cuo in the volume fraction. During the simulation volume fraction of Cuo and water is 0.1%, 0.5%, and 1.5%. During the simulation we have found maximum temperature drop for 0.5% of Cuo and water mixture.

CONTENT

LIST OF TABLES

LIST OF FIGURES

LIST OF ABBREVIATIONS

A	Base area, m^2
A_n	Area of nozzle, m^2
C_p	Specific Heat Constant Pressure, J/kg-K
D	Nozzle diameter, m
ρ	Density of fluid, kg/ m^3
V	Volume,m^3
H_c	Height of the Channel, m
H	Heat transfer coefficient, W/m^2 K
K	Thermal conductivity, W/mK
l_x	Length of the heat sink, mm
l_y	Width of the heat sink, mm
l_z	Height of the heat sink, mm
n	Number of jets
q	Heat flux, W/m^2
R_{th}	Thermal resistance
T_s	Temperature of solid, K
T_f	Temperature of the fluid, K
ΔT	Temperature rise, K
v	Velocity of fluid, m/sec
T_1	Temperature of fluid at inlet, K
T_2	Maximum Temperature, K
Tm	Mean Temperature, K
Re	Reynolds number

W_c	Channel width, mm
W_j	Jets width, mm
x,y,z	Cartesian coordinates

CHAPTER 1

INTRODUCTION

The branch of fluid dynamics that provides a cost effective means of simulating real flows by the numerical solutions of the governing equations is known as Computational Fluid Dynamics (CFD.).The Navier–Stokes equations, one of the governing equations for Newtonian fluid dynamics have been known for over 100 years. Still today the development of reduced form of these equations a major and active field of research, particularly the turbulent closure problem of Reynolds averaged Navier-Stokes equations. But in case of non-Newtonian fluid dynamics, two phase flows and chemically reacting flows, theoretical development is at a lesser advanced stage.

In this method i.e. Computational technique, the governing partial differential equations replaced by a system of algebraic equations that can be solved very easily by the help of computers. One of the major reasons for the rise and emergence of CFD is the steady enhancement in computer power since 1960s.It allowed testing of situations which are very tough or impossible to be measured experimentally and are not amenable to analytic solution.

Experimental methods play a major role in analyzing and validating the limits of different approximations to the governing equations, mainly rig tests and wind tunnel thus providing a cost effective alternative to full scale testing. The analytic solutions cannot be achieved for most practical applications due to the extreme complexity of flow governing equations.

In the computational fluid dynamic problem we used different code in authorized ansys. In this condition the CFD conclude problem to problem as per data given. The method constraint in the alternative testing scale. In using the thermal design problem CFD provided the simulation on the various flow technique as per the problem.

CFD base on different problem on different situation on grid data it noted that fluid flow condition in micro channel. It is noted the segment of analytic solution describe the condition of fluid flow condition.

1.1 APPLICATIONS OF CFD:-

1.1.1 **Aerospace:** - Various methods of CFD are used in many aerospace applications for the role of predicting component performance and also as an integral part of the design cycle. The applications are many like flow around an aircraft etc.

1.1.2 **Automotive: -** In the field of automotive applications CFD is used for modeling the cars aerodynamics to reduce drag and optimize the down force under different operating conditions and also in other fields like auxiliary systems, engine components etc.

1.1.3 **Biomedical: -** CFD is used to design and simulate the blood flow in the heart vessels, inhalers, flow in heart assist devices and other equipment like drug and anesthetic devices from university of buffalo.

1.1.4 **Chemical Engineering: -** Applications of CFD in the field of chemical engineering are vast and surplus such as petrochemicals, pollution control, fertilizers, food processing, waste treatment recycle etc.

1.1.5 **Power Generation: -**In the field of power sector CFD finds applications in the analysis of economizer, super heaters, pulverized coal combustion, low NOx burner design and in other areas to improve performance and efficiency of the plant.

1.1.6 **Electronic systems: -** The demand for quieter, smaller, more powerful, safer and reliable electronic equipment and various thermal challenges faced by manufactures highlights the increasing demand for easy to use, accurate and cheap computational design tools to overcome complex thermal problems related to their cooling. It is the thermal analysis of electronic system by thermal management solution(TMC)

1.2 Steps of CFD-

- Divide the fluid volume (surface) up into manageable chunks (gridding)
- Equation to be simplify at required condition
- boundary condition must be set
- Initialization of grid values
- Simplify the equation through the step grid at the required set point.

1.3 Advantages :

In CFD process there are following advantage carried out. It dominate the various step process but it quiet effective to generation 3d volume. The generation process take less time as time take manually operated proses.

- In CFD process carried out great time reduction and reduction in cost.
- There is a possibility analyze different problem which are very difficult and dangerous.
- CFD technique offer analyze the problem in capacity of limit in over its limit.
- Practical unlimited in level of detail.
- For analyze the result plot various graph for validation the result in deferent purpose use.
- For the good result simplification has been quite simple in mode of process.
- Several incomplete modal to describe the turbulence, multiphase and other difficulties problem.
- Untrained user has tendency pc output true always.

1.4 Disadvantages :

- **.** Easy to excess and law investment lead to over trading.
- Initially cost provided through the set up more as par data.

1.5 CFD code :

CFD code use in different category in various applications. The use of these code we can apply in the generation tool which an associate in cfd analysis.

• **CFD commercial code**: STAR CD, FLUIENT, CFX, CFDESIGN, FLUIDYN etc.
• **CFD research code:** COOLFLUID, CFDSHIP IOWA etc.
• **CFD public code:** WINPIPED, HYDRO etc**.**
• Other CFD code use in generation software such as GEMBIT and grid visualization such as ADINA-AUI CFX-Post, COMSO, ENSIGHT, FIELDVIEW, Hyper View ETC.

1.6 CFD PROCESS-

• The purpose of CFD use with respect to the code applies in various applications such as separated the bubbly flow through domain.
• In the application we seen the massive flow, uniform flow and unsteady flow. In the proses which provide the conjugate configuration rate in the code assign.
• With respect to the code we can apply various other application such that multiphase fluid. Marine, biomedical etc.
• The flow tendency in various manner judges by the code applies on it. It is configure the step in apply the code.
• It leads to apply the code as per the step required in generation process. During the process configuration said the next step to lead. If there drawback than post process to apply.

STEPS-

1. Layout the geometry
2. Apply the physics
3. Generate mesh
4. Solution
5. Report
6. Feedback

1.7 BACKGROUND INFORMATION

The science and practice of achieving approximate numerical solution using digital computers is known as Computational Mechanics. When this approach is applied in the field of problems concerned with thermal and fluid problems, it is generally termed as **Computational Fluid Dynamics (CFD)**. Thus CFD is primarily a branch of continuum mechanics that deals with numerical simulation of heat transfer and fluid flow problems.

The basic fundamental of almost all CFD problems are Naiver–Stokes equations, which defined many single-phase (gas or liquid, but not both) fluid flows. The Euler equations are achieved by simplifying these equations by removing terms defining viscous action. The full potential equations can be obtained by doing further simplification by eliminating terms describing vorticity. For small perturbations in subsonic and supersonic flows equations could be linearized to yield the linearized potential equations.

Historically, a method was first developed to solve the linearization potential equations. Two-dimensional (2D) methods were developed using flow over a cylinder by taking air foil as reference selection. Even though they failed, these calculations set the basis for modern CFD. The computer power available paced development of three-dimensional methods. Los Alamos National Lab can be considered as the place where the first work using computers to model fluid flow governed by the Navier-Stokes equation. Francis H. Harlow, who is considered as one of the pioneers of CFD led this group and they developed various types of numerical methods like Particle-in-cell method, Fluid-in-cell method, Vorticity stream function method, Marker-and-cell method in order simulate transient two-dimensional fluid flows. The initially treatment of strongly contorting incompressible flows in the world were method of the vorticity-stream-function for 2D, transient, incompressible flow by Fromm's. Today different codes are used in the development of many submarines, helicopters, surface ship and more recently wind turbines. Many codes have also been used for modelling such things as high speed trains, racing yachts and also in jet impingement and micro-channel cooling technology. Now days in to achieve greater high heat flux removal and thereby reducing the high temperature and pressure gradient along channel flow, a new field of research is trying to harness the two cooling technologies (micro-channel and jet impingement cooling).

The characteristics of fluid flow are investigated by modelling single-phase fluid flow and heat transfer in a micro-channel jet impingement with the help of CFD. Numerical model has been used to analyse the three dimensional micro-channel jet impingement fluid flows by assuming the flow to be fully developed flow. Different parameters such as Reynolds number, channel geometry, substrate material, and working fluid effects on the performance of the configuration were analysed and results thus obtained were used to compare existing literatures.

Optimum values of some of the important parameters used during the study were observed and further steps were also taken to improve the model. Reduction of flow analysis complexity is also done by applying the fully developed flow assumption. Most recently in order to achieve higher temperature uniformity and improved in electronics items of hot spot management, the application of micro jet impingements on substrates was also proposed. The value of characteristics equation more precise the optimum condition in simulation of given data. It is the synthesized the various dimension on the factor of micro channel design. The function can be excluded the flow parameter in micro channel which the flow condition satisfied.

1.8 OBJECTIVES

The recent work has main objectives as mentioned below:

- At the required modal of jet impingement in micro channel and heat transfer.
- To measure fluid flow characteristics and the heat transfer.
- To study effect on micro channel in such a way that jet configuration angle, heat transfer, Nusselt number and substrate the material on performance of cooling during fluid flow.
- To improve a cooling characteristics apply the optimum characteristics during studies.

CHAPTER 2

LITERATURE REVIEW

In this chapter look at the review of previous research paper that focused in the areas of micro channel and heat sink concentrated jet in different condition. At the experiment carried out in manage heat transfer and solve numerically of electronics system. In the study of micro channel focused on heat transfer or how to cooling channel by apply fluid at required properties. The performance of heat flow and mean temperature on micro channel at required flow fluid.

In this chapter brief define the works against the research on the field. At that condition all research has been done in past year described the different manner. As the purpose use of this modal in different category field for the fluid flow definite condition. In this segment layout define in cross manner which no interrupt the fluid flow condition.

As per the given data we defined the category of reference research of experiment in various manner, it dominate the flow jet impingement in use different stages. As per the study various research carried on that condition given as different condition

Also the heat exchanger performance was affected by heat flow parameters using various kinds of technologies such as jet impingement and micro-channel heat sink. We can review this work in the following two parts:

- Micro-channel
- Jet impingement
- Nano fluid using in micro-channel

2.1 MICRO-CHANNEL

In the last decade, micro-channel system has been mainly used cooling devises vinous mode of cooling system as electronic system. These systems are more effective as heat transfer due to temperature drop. As micro channel demand in electronic device mostly as per its advantage and disadvantage [1]; as per the study of micro channel dimension specified at required parameter in mm [2,3].

In study of heat transfer reported the parameter of geometry such as with, length and height to micro channel. As heat transfer conduct the geometry function to flow fluid [4-10]. In the difficulties of machining of micro channel of the system calculating the 3-d properties towards the wall of micro channel and simulation has adopted. It predicts the pressure drop and temperature along the channel. The combination of velocity and temperature value change by change the properties of fluid flow to the micro channel [11, 12].

2.2 USE OF MICRO CHANNEL

Tuckerman and Pease (1981) was studied the fluid flow for heat transfer as a scope of Ph.D. study. The publication title on "High Performance Heat Sinking in VLSI" was first study on micro channel for heat transfer. Their good work on the micro level have innovative research to focus on the field as micro channel and fluid flow in the channel has recognized such a high performance fluid heat transfer through the micro channel

As the study of micro channel in different category which assist the "microchannel" as heat transfer body from the substrates fluid. The scope of micro channels the debate in this topic between researchers in the field. Mehendale et al. (2000) the following used as study research factor in different "*D*", being the smallest.

The micro channel describes the flow field in different category which can be operated the substrate fluid dynamics. As the channel describe in flow field in category of micro channel.

- Micro channels-$1\mu m < D < 100\mu m$
- Mini channel-$100\mu m < D < 1\ mm$
- Compact Passages-$1\ mm < D < 6\ mm$
- Conventional Passages-$6\ mm < D$

Easiest classification was introduced by Obot (2003) which was totally based on the hydraulic diameter on behalf of dimension of smallest channel. Obot describe channels of hydraulic diameter range under 1 mm ($D_h < 1mm$) as micro channels, which was also adopted by some other researchers such as Bahrami and Jovanovich (2006), Bahrami et al. (2006) and Bayraktar and Pidugu (2006).

For the thesis this definition very importance as heat transfers through micro channel. Higher volumetric transfer of heat densities need improved manufacturing techniques and lead to more complicated manifold designs (Kandlikar et al. 2006). Many researchers describe the various applications on electronic circuit in field flow heat transfer as compact parameter.

Micro channel transfer the heat through the body constraint in two ways. First, the small amount fluid flow in to the refrigerant side of micro channel. Second, the flat tube which is oriented in flat position that reduce flat side , which leads to be either increased airflow or reduced fan power.

- Conventional channel- Dc＞3 mm
- Mini channels -3 mm Dc ＞ 200micro mm
- Micro channel - 200mocro mm Dc ＞micro mm
- Nano channels - Dc；0.1micro mm

Chemical reaction carried out in a micro reactor with typical lateral dimension below 1.0 mm. micro channel heat exchanger act as reactor when temperature of the reactor need to maintain. . Micro reactors used these devices are commonly regular flow reactors rather than batch reactors, with the regular in the micro channel as confined the good and best performance of fluid as possible.

2.3 JET IMPINGEMENT ON EXPERIMENT

Micro-channel and jet impingement give use full means for convective heat transfer coefficient due to large transfer of heat and mass transfer rate. Micro channel is mostly used in industrial applications like as tempering of material, annealing, plastic cooling and paper fabric dry process [13-15]. Due to the geometrical condition heat transfer of multiple jet substantially from single jets [16, 17]. The spacing between the jets is greater effect on the amount of heat transfer.

The complexity as comparing result, a multiple jets geometry dilution a large no. of militating factors [18-20]. For the comparing the result another development correlation as in Sung and Mudawar [20]. Reynolds number (Re), nozzle to impingement surface dimension, H/D and angle of impingement are parameter to determine jet configuration [23, 24].

Sung & Mudawar and others have carried out various experimental and numerical simulation on micro-channel and jet-impingement system reduce the value of pressure drop & temperature gradient in micro-channel as application on device of electronic, jet impingement in micro channel in jet condition zone thermal boundary layer development during fluid flow. In 2005, Sung and Mudawar [25] noted that results obtained from the 3- dimensional characteristics of heat transfer using water , DIUF water , PF-5060, HF-7200 as working fluid were improve the result the individual technologies.

As the research by Sung and Mudawar [21] posit that performance of the hybrid configuration in which Fluid flow for cooling is carried by jet impingement micro channel, compared to the effect on micro-channel flow. They divided test surface into part or grid system that dominate jet impingement and micro channel flow while applying the appropriate critical heat flux on each part. By doing this they was measure the critical heat flux data in hybrid micro channel with an absolute error of about 8.42%.

Sung and Mudawar [20], numerically and experimentally simulated results investigated on new hybrid cooling system proposed high heat flux management in the micro channel. As th velocity increasing the jets penetrated in micro channel flow in between heated the surface and decreases wall temperature.

They divided the surface into various zone at different process and at a time heat transfer each zone.

The study Sung and Medawar developed a new correlation, the results obtained from study approximate error of 6.037%. In 2007, Sung and Mudawar [16] further studied a single-phase cooling performance of the hybrid cooling system as a series of jets that provide cooling into each channel and each row of a micro-channel heat sink. This was symmetrical flow in each micro-channel, and the coolant provided through in each end of the micro-channel. Three (3) jet patterns consider in this case were: decreasing-jet-size, increasing-jet-size and equal-jet-size.

As the decreasing the jet-size pattern yields the maximum temperature difference through the micro channel, whereas equal-jet-size pattern provided uniformity in wall temperature. Also, increasing jet size complex yield pattern as more wall temperature gradient, fluid flow due to the imping of jet near the channel outlet. Barrau et al [28] noticed and experimentally studies that proposed hybrid system for high flux thermal management in a power devices. As they tested the geometry characteristics and its cooling performances, also to check the capacity to provide parameter.

Thermocouple was used for measured temperature distribution in the channel. Since the temperature decrease in the direction of fluid flow seen along the channel. The hybrid cooling has capacity to enhance the temperature of the channel to cool.

The various literature review, either experimentally or numerically simulation has no investigation taken place to improve design of hybrid cooling system. At present study micro channel via modeled on using ansys15.0 used for investigation of modeling of system. The data provided in to the modeling to control of the meshing on the system. It is conclude that better result come from in practical purpose use the simulation as refine the mesh.

Due to the recommendation of all performance of fluid in between the flow characteristics, the effect of Reynolds no., overall performance of fluid and thermal resistance investigated. The height of channel and width of channel used as a geometrical parameter. Substrate the material such as copper, aluminum, silicon, steal, titanium, nickel used in micro channel. At the fluid flow condition for type fluid has been used such as water, DIUF water, PF-5060, HF-7200. At the required property of fluid or increase the heat transfer of fluid Nano fluid used in channel in different fraction of volume. The requirement solid domain and fluid domain in micro channel heat flux used in the segment 250000w/m^2.

2.4 Nano fluid with micro channel

Now a days Nano fluid is a very effective for the heat transfer from the micro channel. At present several Nano fluids organized in many applications. Such as maximum heat transfer from the fluid taken with use its property. Nano fluid made by the oxide, carbide, and carbon particle in Nano size for the heat transfer through the micro channel. Nano fluid driving in to the micro channel as a conjugate manner. During the Nano fluid heat transfer through the substrate the material in uniform flow. At the stagnant condition both fluid and Nano particle defined heat transfer. Nano fluid containing Nano size in a variant flow. At that condition the flow making suspension colloidal in base of fluid.

We used copper oxide as Nano particle in volume fraction. We used these Nano fluids in to micro channel at a required Reynolds number. Mass flow rate in to the channel 6.67×10^{-6} Kg/s flow with various characteristics. The channel at a required characteristics Nano fluid separates the temperature drop through. The volume fraction with Nano fluid 0.1%, 0.5%, 1.5% respectively. We used required mass rate flow in fluid flow through into micro channel. As convention flow heat transfer occurs required data define. In heat transfer between solid domain and fluid domain can be recognized conjugate manner. In such a way Nano fluid converge the heat transfer.

In the find out maximum heat transfer through the micro channel, the various chapters include the parameter under the heat load and volume fraction of Nano particle in different segment.

It is necessary to evolution of Nano fluid by the important factor as diameter of Nano particle [34]. The necessary of Nano fluid in maximum heat transfer during the different category described in various flow, in this condition describe the volume fraction with molar mass of the Nano fluid. The condition of the various flow conditions in micro channel, the feature of the Nano fluid in CUO and water, Al2O3 and water at category of fluid flow [33].

CHAPTER 3

METHODOLOGY/ MODEL DESCRIPTION

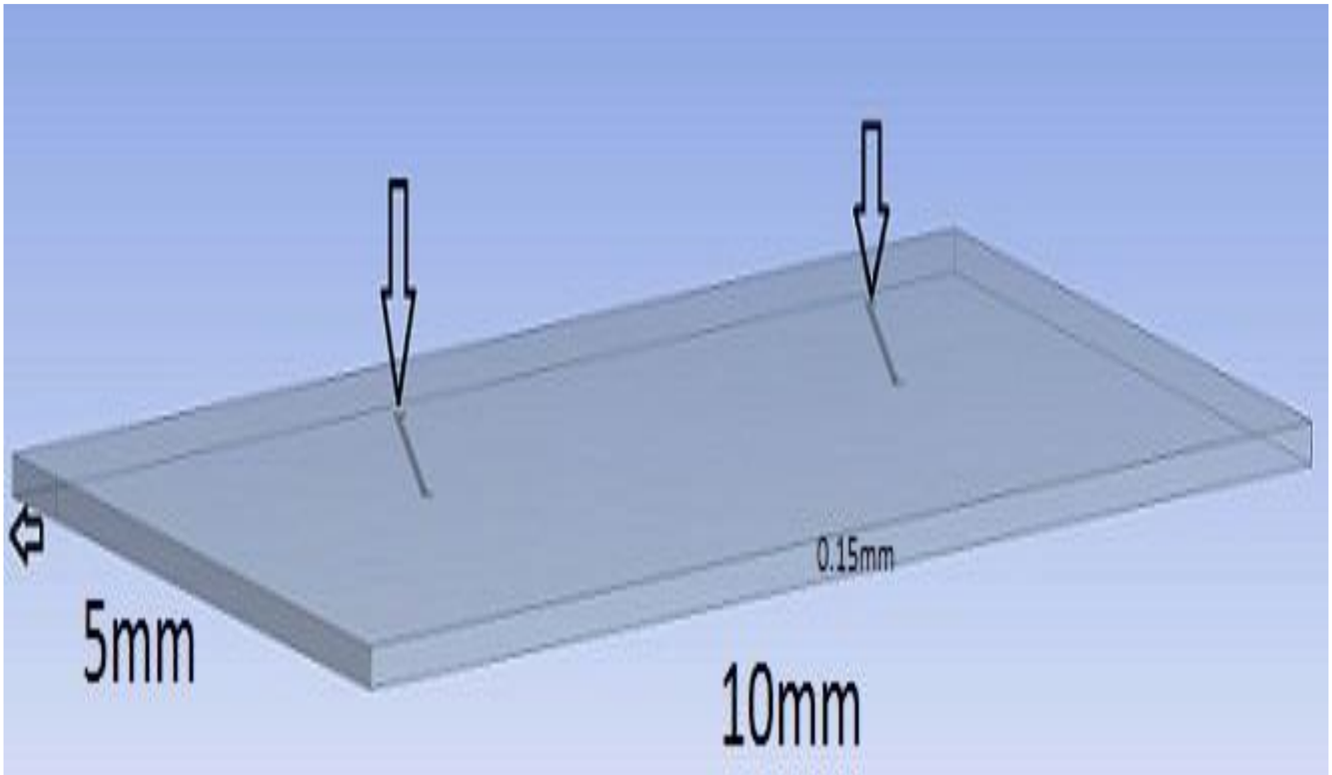
3.1 MODEL DESCRIPTION

A multiple inclined jet impingement heat sink show in below fig. no.1 designed for cooling backside of electronic microchip. The nozzles were fabricated such a way that provide incline fluid to substrate the fluid domain and solid domain. The working fluid has taken in fluid domain fig. no. 1(a) in the micro channel. The working fluid strike such a way that fluid strike inside the solid domain which can be seen in fig. no.1 (b).

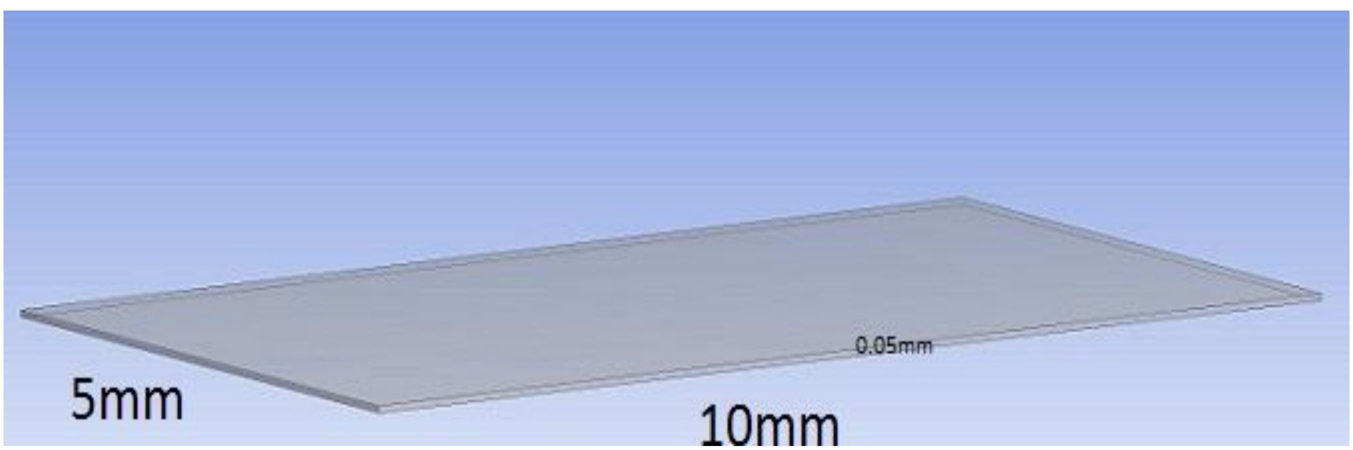
In the study the substrate part in which nozzle part not examine, during the fluid flow substrate base part including all domain. During the substrate all domain conservative and joined with the epoxy design structure. Jets and outlet channel desired for the fluid flow in the channel.

The micro channel dimension classified in various solid and fluid domain 10mm, 5mm, 0.15mm.The depth of fluid domain which interact different material is 0.15mm. It is part define in various flow process in substrate material. The thickness of material substrate base metal is 0.05mm. During the fluid flow pass in to the solid domain obtained better performance. On of the other one half heat sink taken as symmetry of the channel about the side plane. As per the modal nozzle used inclined from the base of the upper surface of the fluid. At that condition the fluid impinging through the nozzle in very frequent manner. The hybrid channel described in the form of the channel dimension.

The fluid flow in micro channel in different segment which define the various step through in ansys15.0. Combined solid domain and fluid domain can be seen in fig.2.which is show the various heat transfer factor.



(a) Fluid domain



(b) Solid domain

Fig. 1 Geometry of inclined jet micro channel (a) fluid domain and (b) solid domain

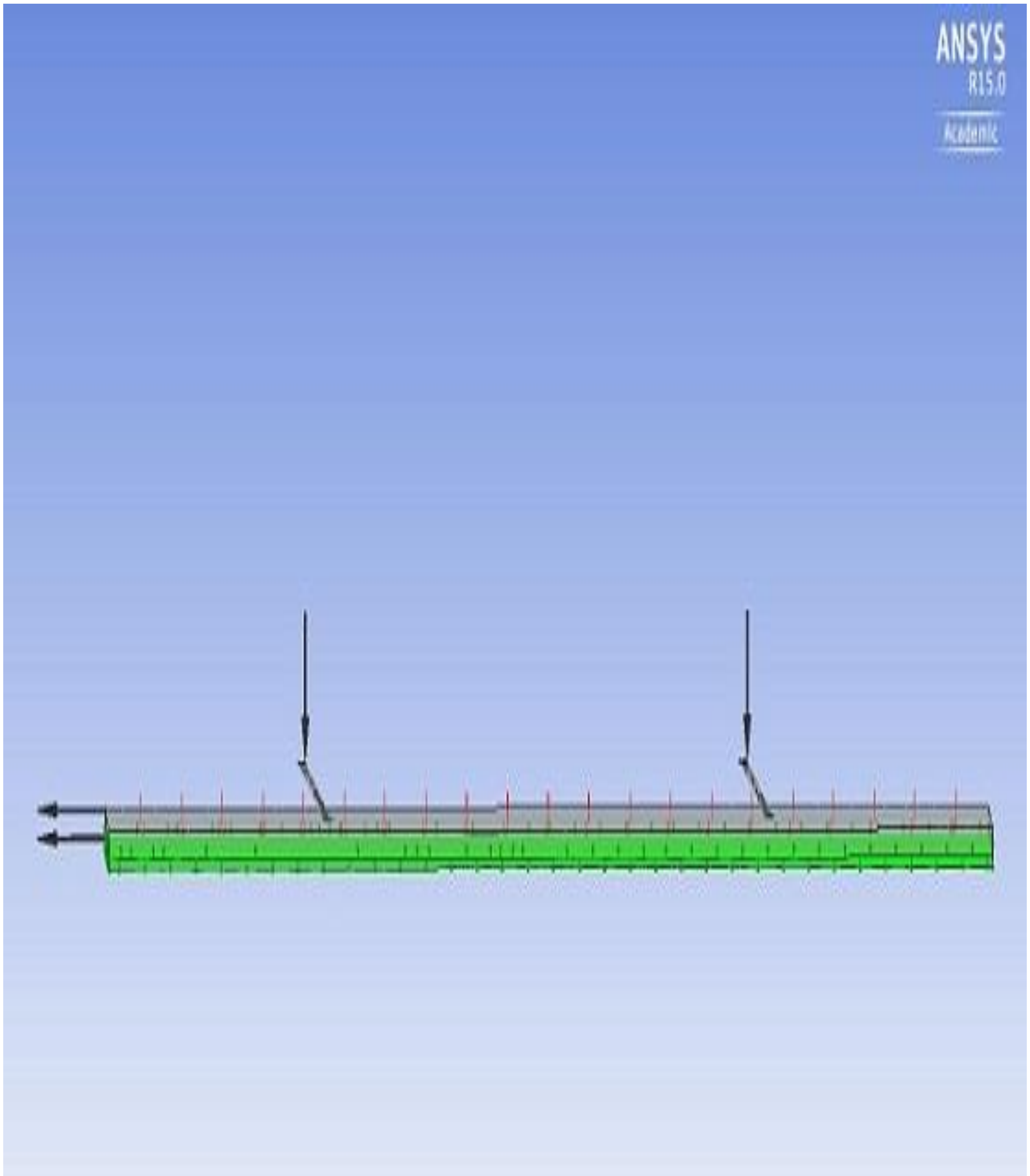


Fig. 2 Combined geometry of fluid domain and solid domain

3.2 NUMERICAL SCHEME AND BOUNDARY CONDITIONS

In the analysis which was performed on the fluid flow domain and heat transfer through the model, different types of fluids such as water, DIUF water, HF-7200,PF-5060 were used in the micro channel in order to provide a cooling effect on the micro channels. The equations used during the process include governing equation through the channel as fluid flow, conservation of mass, momentum and energy equation in flow field.

MASS CONSERVATION EQUATION

$$(\partial(\rho u))/\partial x=0 \quad (1)$$

MOVEMENTUM CONSERVATION EQUATION

$$(\partial(\rho uv))/\partial x= -\partial\rho/\partial x +\partial/\partial x\ (\mu\partial u/\partial x\ ) \quad (2)$$

ENERGY CONSERVATION EQUATION

$$(\partial(\rho C_p\ u\ T))/\partial x=\partial y/\partial x\ (k\partial T/\partial x\ ) \quad (3)$$

Simulation provides the flow field in ansys15. It was accomplish during various fluid flow through in it. The simulation define simple hexahedral mesh generated specified computational fluid domain. During iteration solid and fluid domain are classified in a conjugated manner.

It recommended hexahedral mesh in particular computational domain during fluid flow. For study heat sink optimized specified parameter in a channel fluid wall in fluid domain except solid fluid domain interface at bottom of channel kept adiabatic.

In a channel uniform heat flux provided as solid towards the cooling such as applied the no slip condition(v=0). During the flow in internal wall of nozzle. Throughout the condition nozzle applied incline from the top surface of the channel.

HEAT FLUX EQUATION:

$$Q/A = h\ (T2-T1)$$

$$q = h\ (T2-T1) \qquad (4)$$

NUSSELT NO:

$$Nu = hD/k \qquad (5)$$

$$h = q/(T2-T1)$$

CHRACTERSTICS DIAMETRE:

$$D = 4A/P \qquad (6)$$

$$A = H*B$$

$$P = 2(H+B)$$

ENERGY DUSSSIPETE FROM THE FLUID:

$$Q = M\ Cp\ (T2-T1) \qquad (7)$$

$$M = \rho V$$

3.3 ASSUMPTIONS

Following assumption are made due to complexity of 3d heat sink -

1. fully developed flow through micro channel and steady state condition

2. Water uniformly flows in fluid domain.

3. No slip condition in domain separate and adiabatic condition

4. Temperature assumed to be constant in interface fluid and solid domain

5. Assumed Nano fluid as volume fraction with water.

CHAPTER 4
RESULTS AND DISCUSSIONS

4.1 SUBSTRATE MICRO CHANNEL

We used different jet angle in simple hexagonal microchannel. We have used 30^0, 45^0, and 60^0 from the top surface of the body. We have also observed in one fix fluid to driving inside the microchannel. We used water as coolant in microchannel for maximum heat transfer. Whereas heat flux given to based metal at value 250000 W/m^2. We have also signifies nusselt number for heat transfer coefficient at required temperature drop in micro channel.

In the used inclined micro channel in different position of jet. In this segment we have used cooper as a solid metal through micro channel. The property of copper in micro channel seen in table 4.1, as per the data given in the table we have used cooper as base metal and water as fluid in to micro channel

We have seen the variation in the temperature in micro channel at base plate, also signifies the heat transfer coefficient. Due to the variation in temperature in microchannel signifies the maximum heat drop from the base metal.

We used water as a fluid and copper as base metal in three meshes incline jet micro channel. We know that cooper has high conductivity metal for heat transfer and water as a natural cooling liquid.

During iteration we have found maximum temperature drop in three inclined jet impingement in micro channel with copper base metal in different angle. The jet imping at 45^0 give maximum temperature drop (fig. 4.1). After simulation maximum temperature drop measure maximum heat transfer.

After the iteration we have calculated the maximum temperature drop in microchannel. In 45^0 jet impingement gave maximum temperature drop and heat transfer through base metal.

In the table 4.2 define deferent temperature drop with respect to same base metal and same fluid through the microchannel. At the required temperature define maximum heat transfer flow through the system.

Now we have computed best angle through the microchannel in between best heat transfer. After simulation we used 45^0 jets through micro channel for further simulation. We used different material as a solid domain in microchannel.

We used copper, nickel, aluminum, titanium, steel, silicon for furthers simulation. We used water as a coolant in between simulation. Different material show variable conductivity. Due to the variable conductivity in decreasing order maximum temperature drop obtained.

At the fluid flow through micro channel we have simulated the result at same heat flux through the base metal, the variation of temperature drop seen contour. In between we have seen that copper largest conductivity and its show largest temperature drop in micro channel (fig. 4.2). We have also signifies heat transfer coefficient in variable zone. The variation in the heat transfer coefficient in different stages show heat transfer through micro channel.

Table 4.1 Property of copper metal:

Property of copper			
Material	Density (kg/m^3)	C_p (J/kg-K)	K(W/m-K)
Copper	8933	385	401

Table 4.2 Result of simulation of inclined jet at different angles

Result of simulation of different inclination of jet in micro channel with copper as base metal			
Result of simulation	Jet impingement at 30^0	Jet impingement at 45^0	Jet impingement at 60^0
Mean temp. (K)	403.07	407.16	362.25
Temp. diff. data (K)	214.31	222.50	132.69
Heat transfer coefficient (h) W/m^2 K	1165.52	1123.52	1888.048

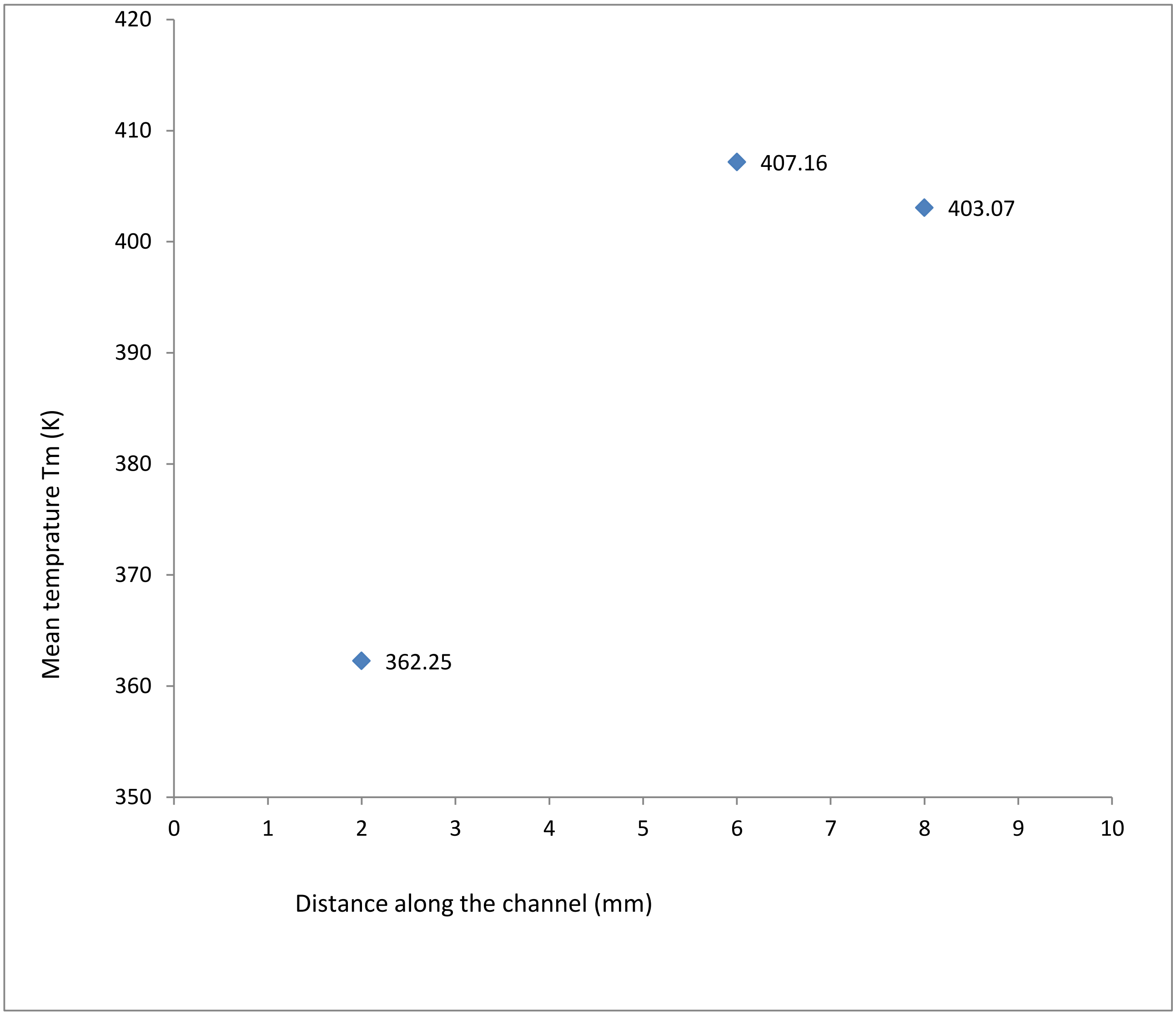
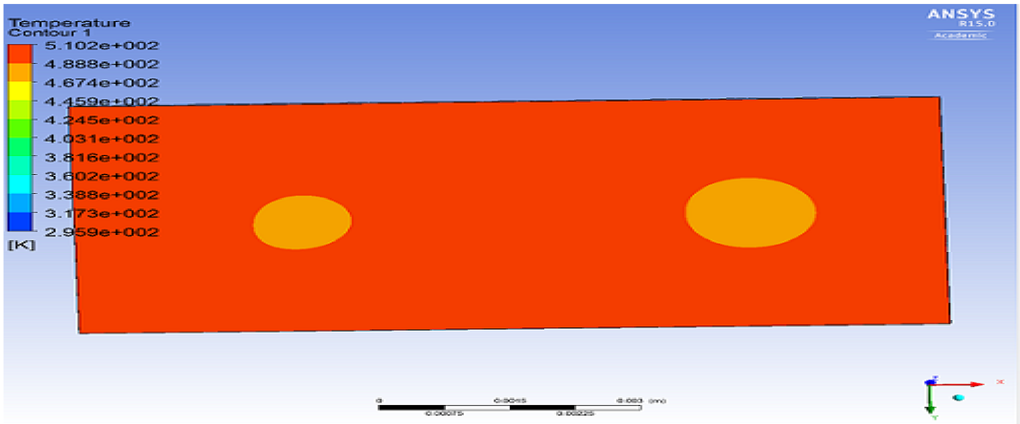
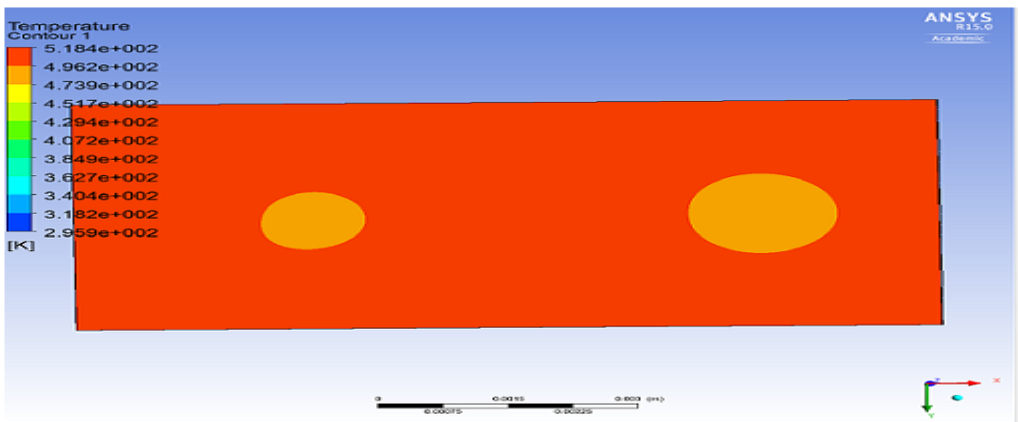


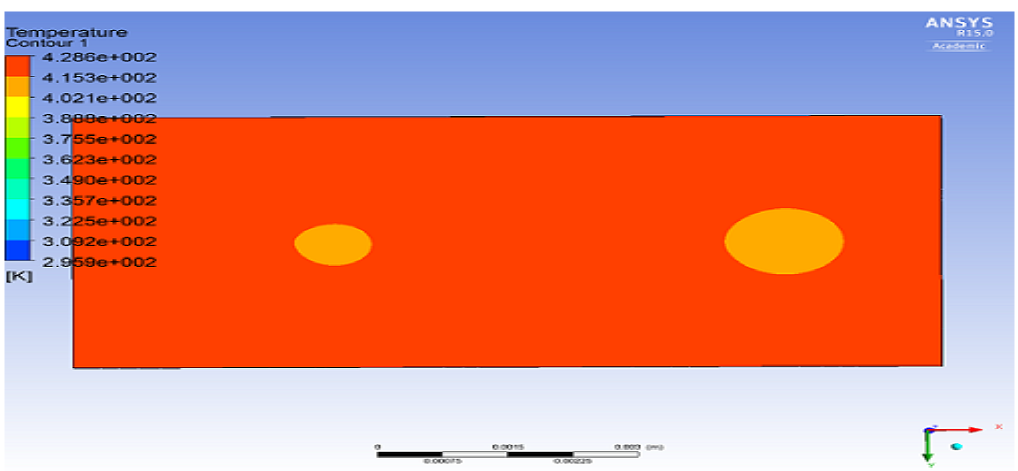
Fig. 4.1 Mean temperatures along length of the micro channel with copper as base metal



(a) Temperature distribution of inclined jet at 30^0



(b) Temperature distribution of inclined jet at 45^0



(c) Temperature distribution of inclined jet at 60^0

Fig. 4.2 (a), (b) and (c) temperature distribution contour of inclined jet in micro channel at various angles

4.2 SUBSTRATE MATARIAL

We have used different type of material in a microchannel. In use of material in staged manner water as a coolant in fluid flow ansys15. During fluid flow as water define different condition via different material. Due to high conductivity of material in decreasing manner define in below chart. Due to high conductivity liberate maximum heat transfer as maximum temperature drop in microchannel. In fluid flow during the iteration we used CFX modeling. In the CFX modeling we can approach to maximum heat transfer.

We have used the different category of material for suited the best result as heat transfer. In this condition copper, aluminum, steel, titanium, silicon, and nickel material used for simulation the property of material and result of simulation defined in table 4.3. For the best heat transfer we have classified the material in crosswise manner.

As the deferent material used in micro channel, the combination effect of mean temperature along the length of micro channel defined in (fig. 4.3). It is defined the variation of temperature along the channel at the required dimension of micro channel. The variation of temperature along the distance classified the variation of heat transfer through the micro channel (fig. 4.4). Temperature difference defined maximum heat transfer through the micro channel in substrate the materials. The distribution of temperature through the channel seen in (fig. 4.5). In this cooper contour defined maximum temperature drop and maximum heat transfer.

In a different segment copper define maximum temperature drop. It dominates maximum heat transfer from base metal in microchannel. The jet impingement provided in microchannel at the above the surface. It is a segment where fluid passes. After the analysis copper has high heat transfer from base metal. Due to high heat transfer of copper we can use to check best fluid in flow in microchannel. It suited that if heat transfers from the microchannel it better to use maximum conductivity metal.

Table 4.3 Result after simulation of different materials:

Material data				Results after Simulations		
Material	Density (Kg/m^3)	C_p (J/Kg-K)	K(W/m-K)	Mean temp. (K)	Temp. diff. data (K)	Heat transfer coefficient. (h) $W/m^2 K$
Copper	8933	385	401	407.161	295.904-518.413	1123.5248
Aluminum	2702	903	237	374.774	295.94-453.597	1585.8114
Steel	7854	434	60.5	376.1345	295.965-465.30	1476.3285
Silicon	2329	700	149	376.6865	295.954-457.419	11904.76
Titanium	4500	520	21	383.7275	295.967-471.44	1424.3309
Nickel	8908	445	90	378.886	295.962-461.81	1507.407

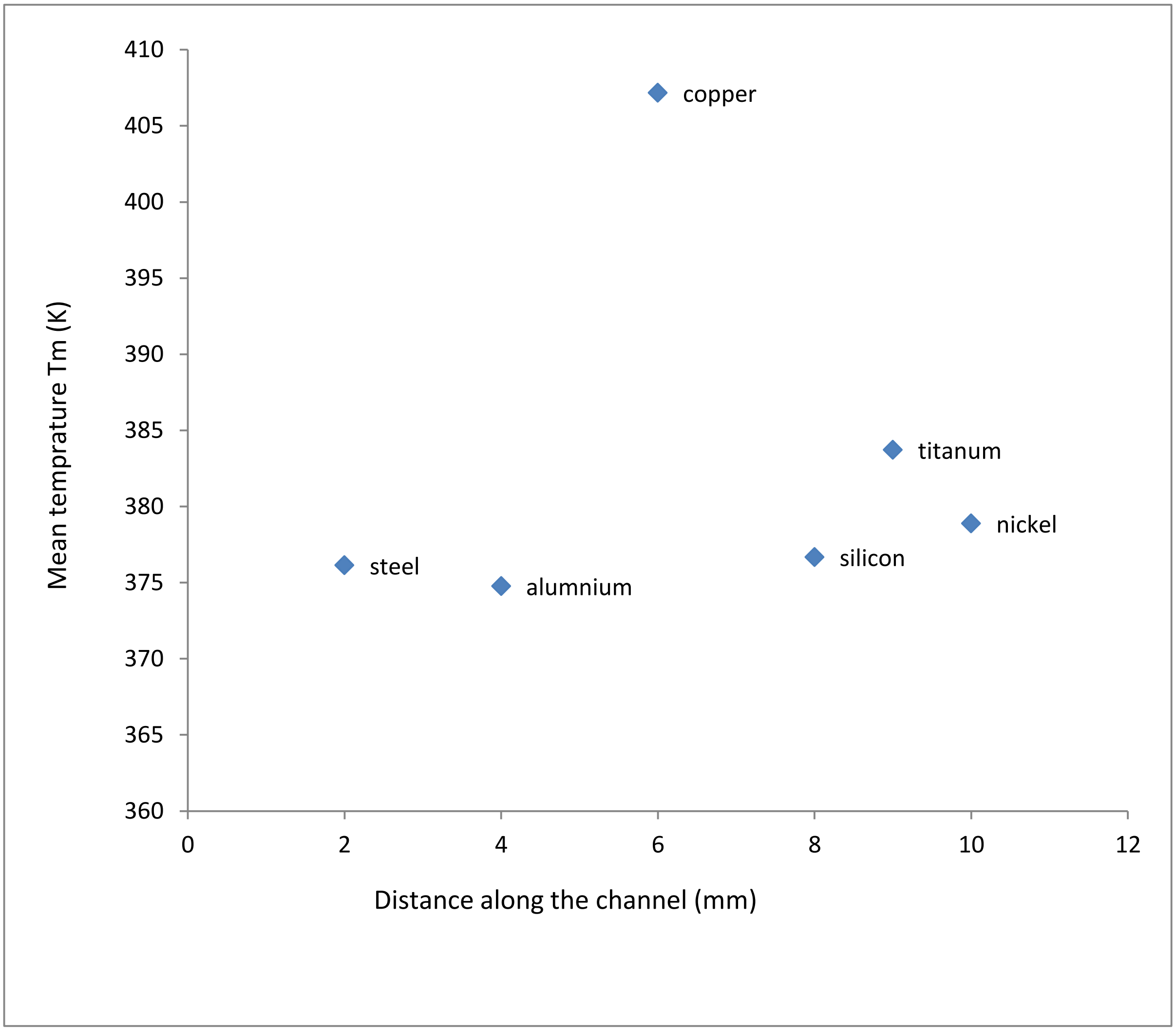


Fig. 4.3 Mean temperatures along length of the micro channel for different materials

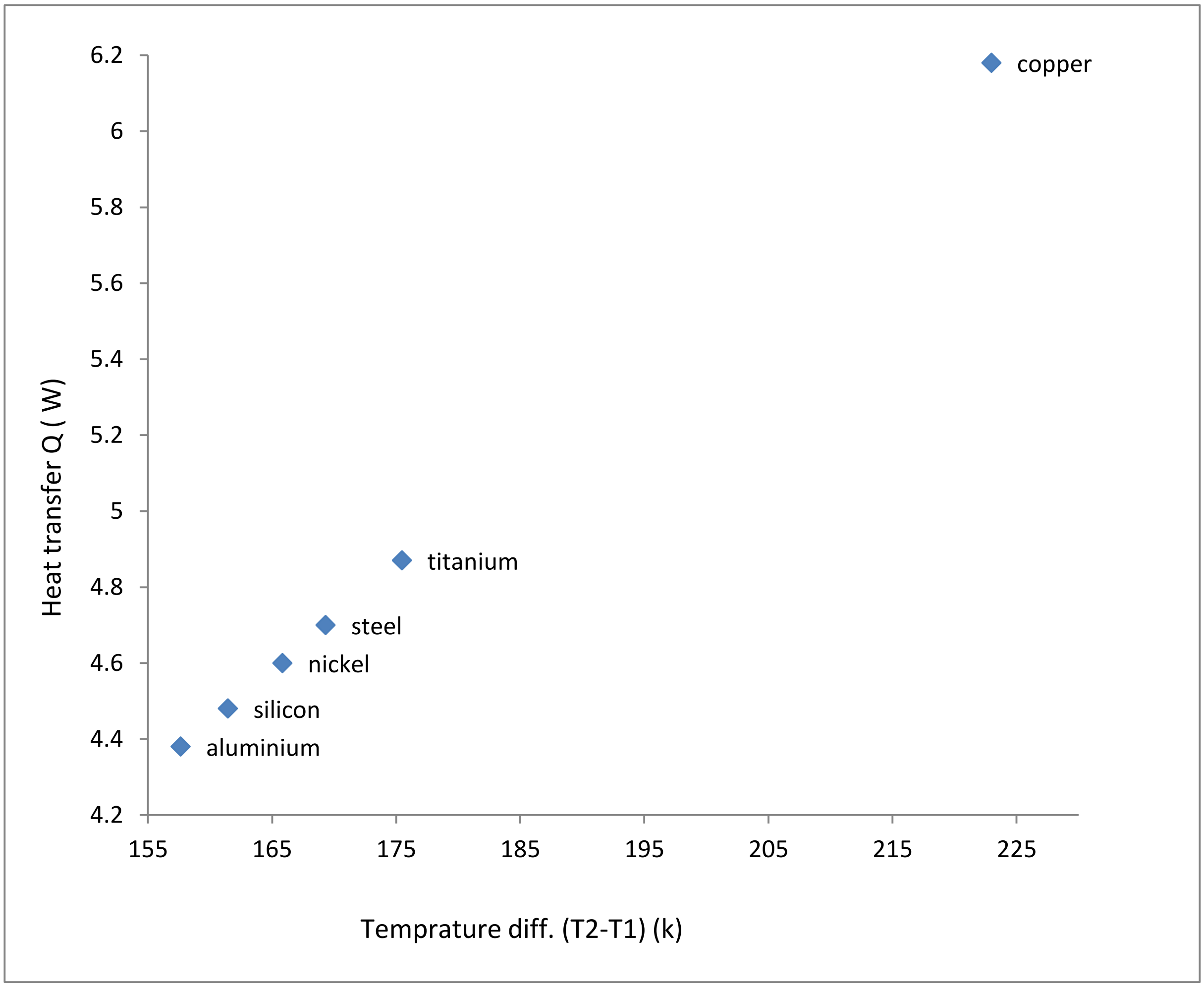
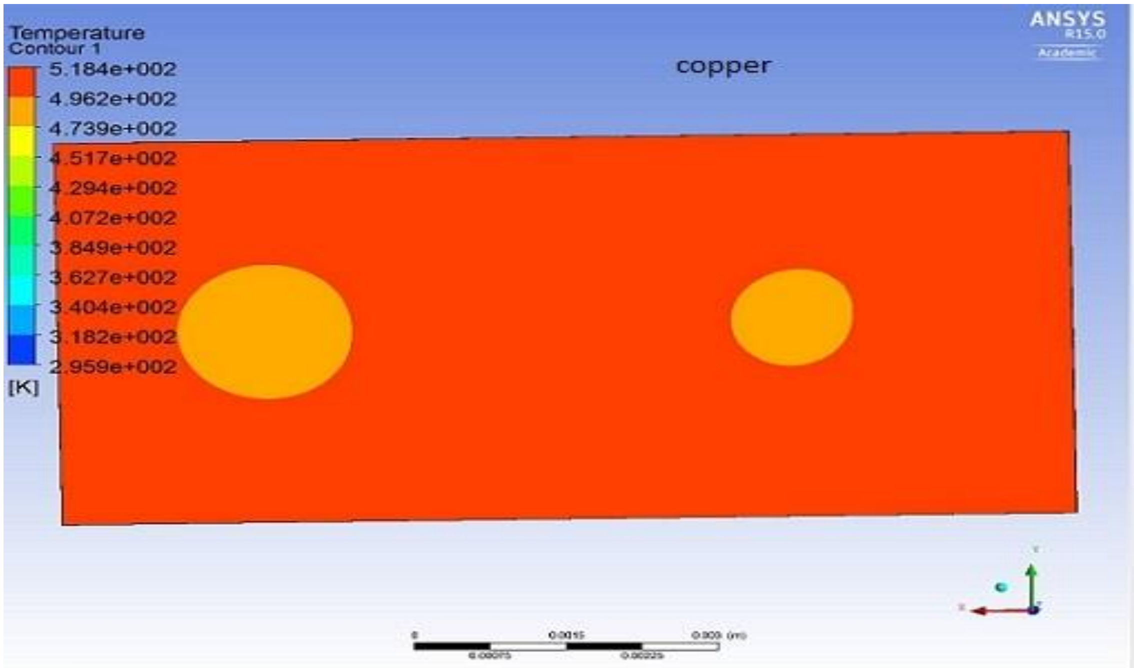
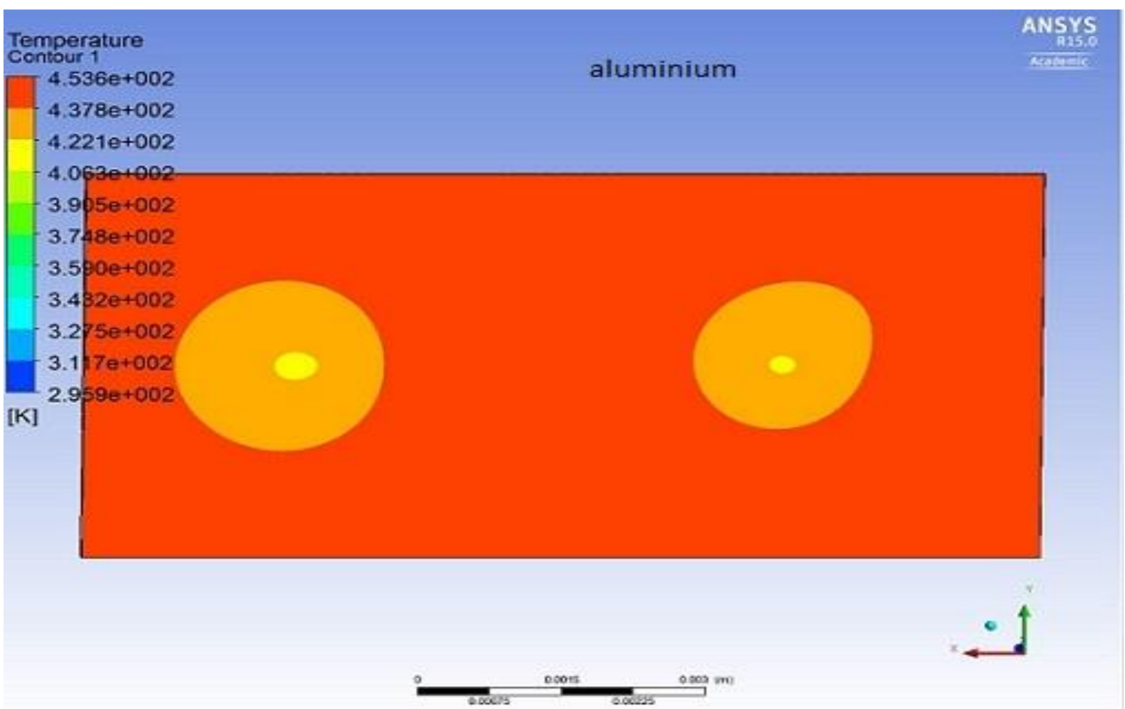


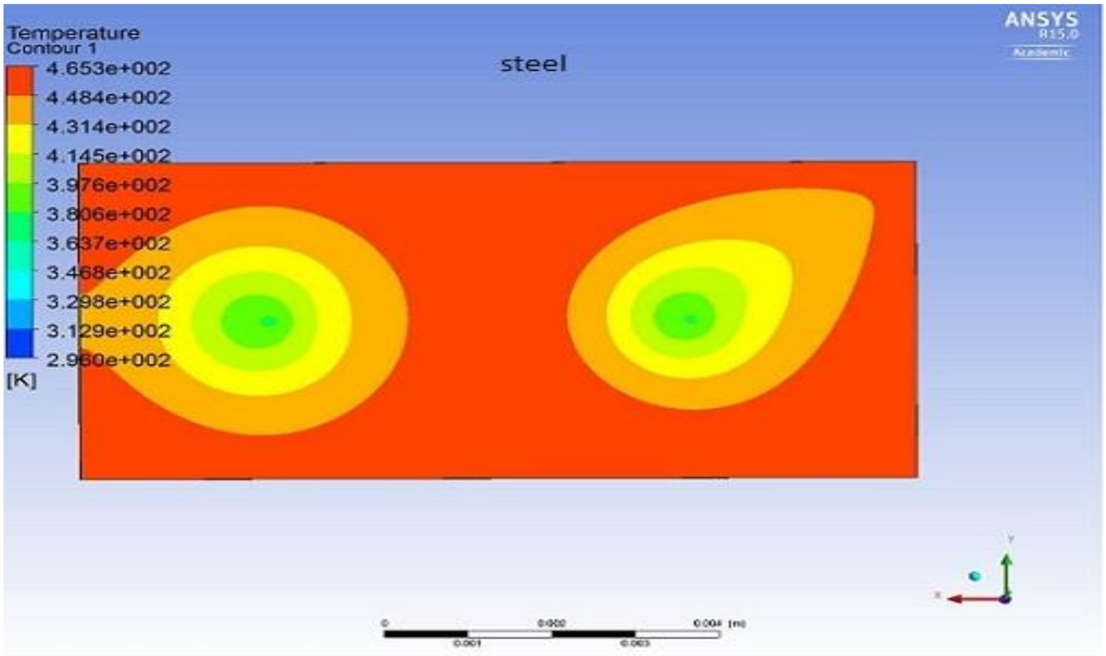
Fig. 4.4 Temperature difference and heat transfer of different materials



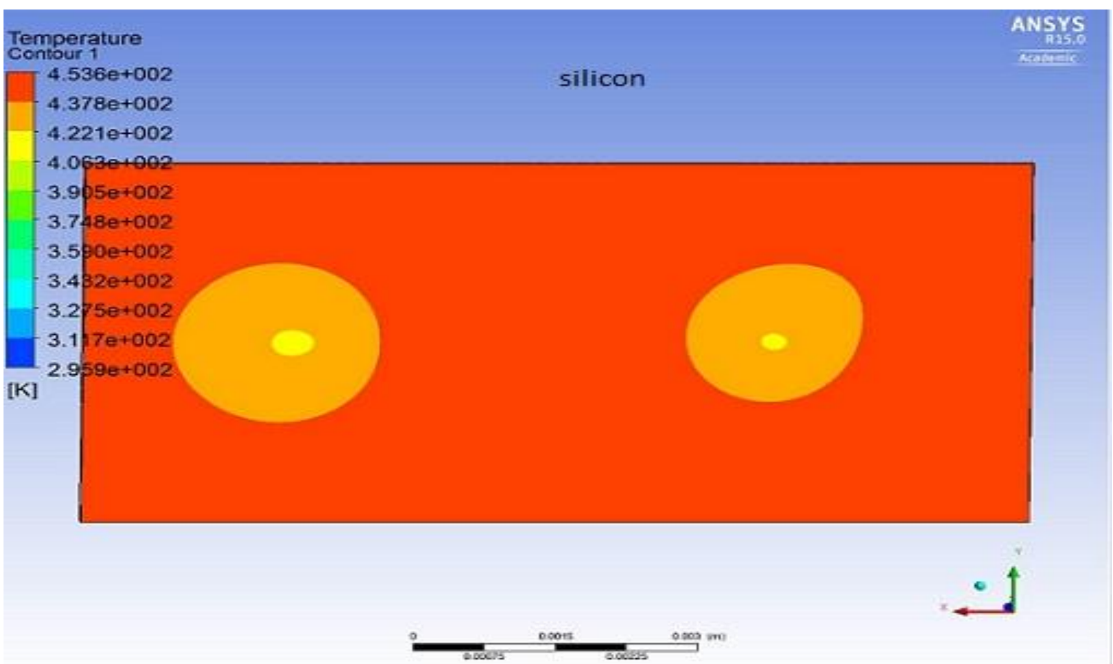
(a) Temperature distribution contour of inclined jet micro channel with copper as solid metal



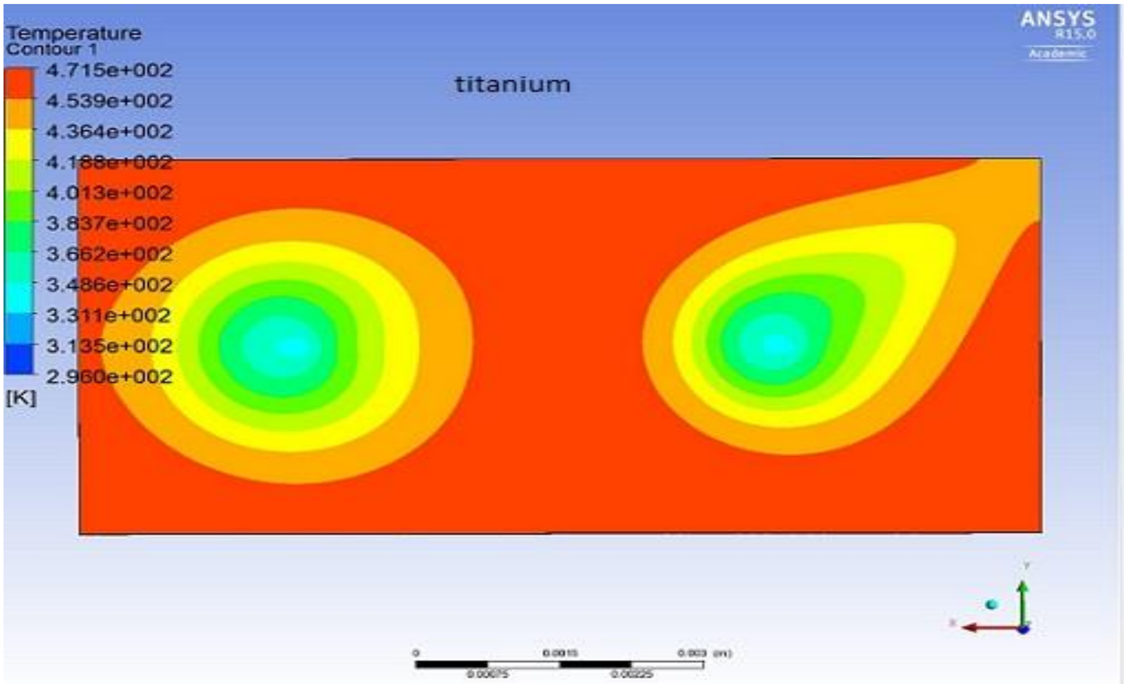
(b) Temperature distribution contour of inclined jet micro channel with aluminium as solid metal



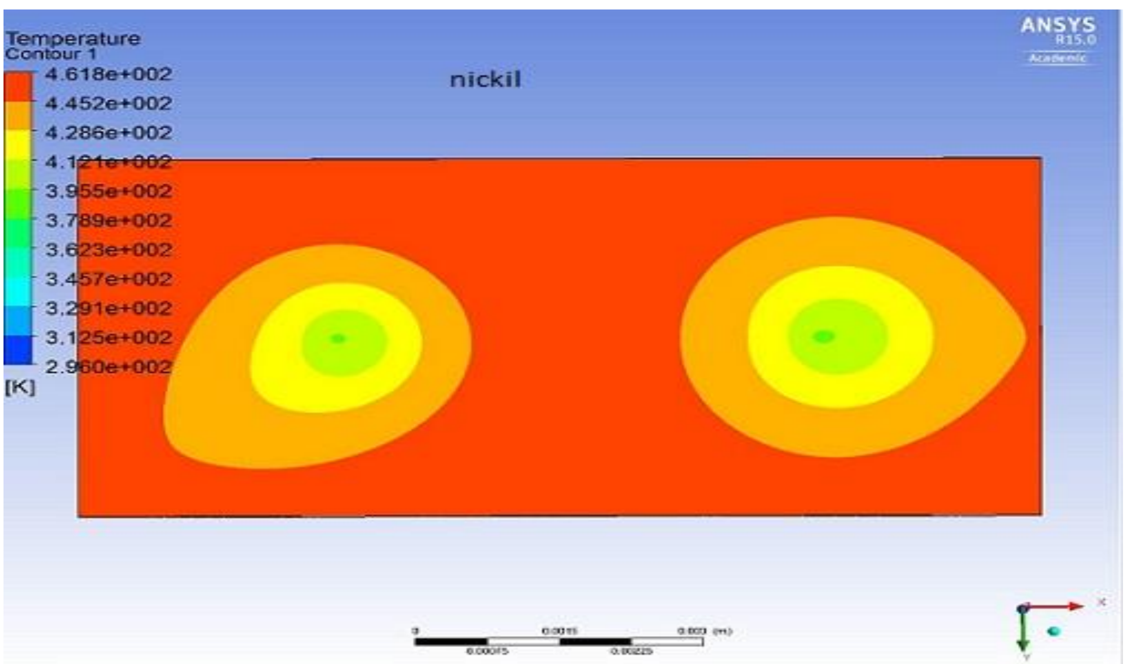
(c) Temperature distribution contour of inclined jet micro channel with steel as solid metal



(d) Temperature distribution contour of inclined in jet micro channel with silicon as solid metal



(e) Temperature distribution contour of inclined jet micro channel with titanium as solid metal



(f) Temperature distribution contour of inclined jet micro channel with nickel as solid metal

Fig. 4.5 (a), (b), (c), (d), (e) and (f) temperature distribution contour of inclined jet micro channel for different materials

4.3 SUBSTRATE FLUID

We used different type material through the micro channel. During fluid flow we consider best fluid suited through micro channel. During the maximum heat transfer through micro channel we consider fix heat flux at base metal. We provided 250000 W/m^2K heat flux to base metal. We used copper as a base metal because it obtains maximum temperature difference through micro channel.

We used DIUF water, water, HF-7200, PF-5060 such type material in micro channel. This fluid provided into the nozzle at top surface of the micro channel. Nozzle sprays that fluid into the micro channel in laminar flow. The flow in micro channel at required heat flux, the temperature corresponding to fluid defined in table 4.4.

We used fluid as DIUF water through micro channel. It provided the temperature different 147.4 K through base metal. In further simulation we used water through micro channel. It provides the temperature different 223 K through base metal. In further simulation we used HF- 7200 through micro channel. It provides the temperature different 594.7 K through base metal. We used PF-5060 through micro channel. It provides the temperature different 594.7 K through base metal.

At the required material property and diversified the variation of fluid show the mean temperature along the length of channel (fig 4.6). At the required heat transfer through the micro channel classified the Reynolds number for enhance property of fluid. The (fig. 4.7) shows variation of nusselt number at required temperature. At the required flows the variation of temperature under heat transfer through the micro channel on contour in (fig 4.8).

In the simulation we used different type fluid through micro channel, we have computed temperature difference and we have computed best fluid for maximum heat transfer. In the use performance fluid define maximum temperature drop through micro channel. PF-5060 is a performance fluid and it dominate enhance the performance of heat transfer through the micro channel.

Table 4.4 Simulation result of different fluids

Properties					Results after Simulations		
Fluid	ρ(kg/m^3)	C_p(J/kg –K)	K(W/m-K)	μ(kg/m-s) x 10^{-5}	Mean temp. (K)	Temp.diff. data (K)	Nusselt number (Nu)
DIUF water	990	4190	.582	100.2	369.6	147.4	10.33
Water	998	4182	0.6	100.3	407.15	223	9.07
HF-7100	1538.3	1133	.074	76.99	592.42	594.7	34.14
PF-5052	1776	1014	.065	97.77	1332.9	2080.2	40.17

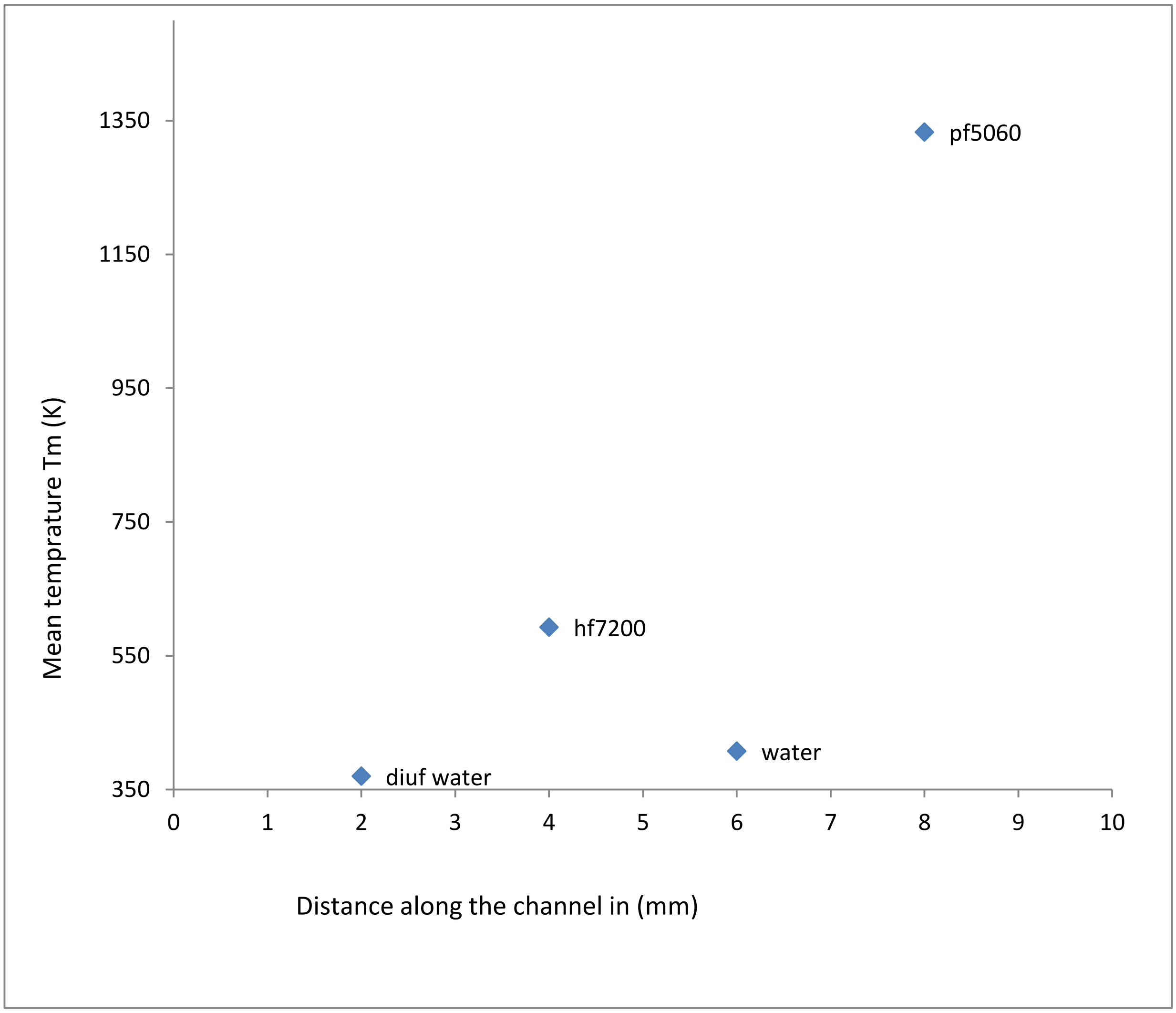


Fig. 4.6 Mean temperatures along length of the micro channel for different fluids

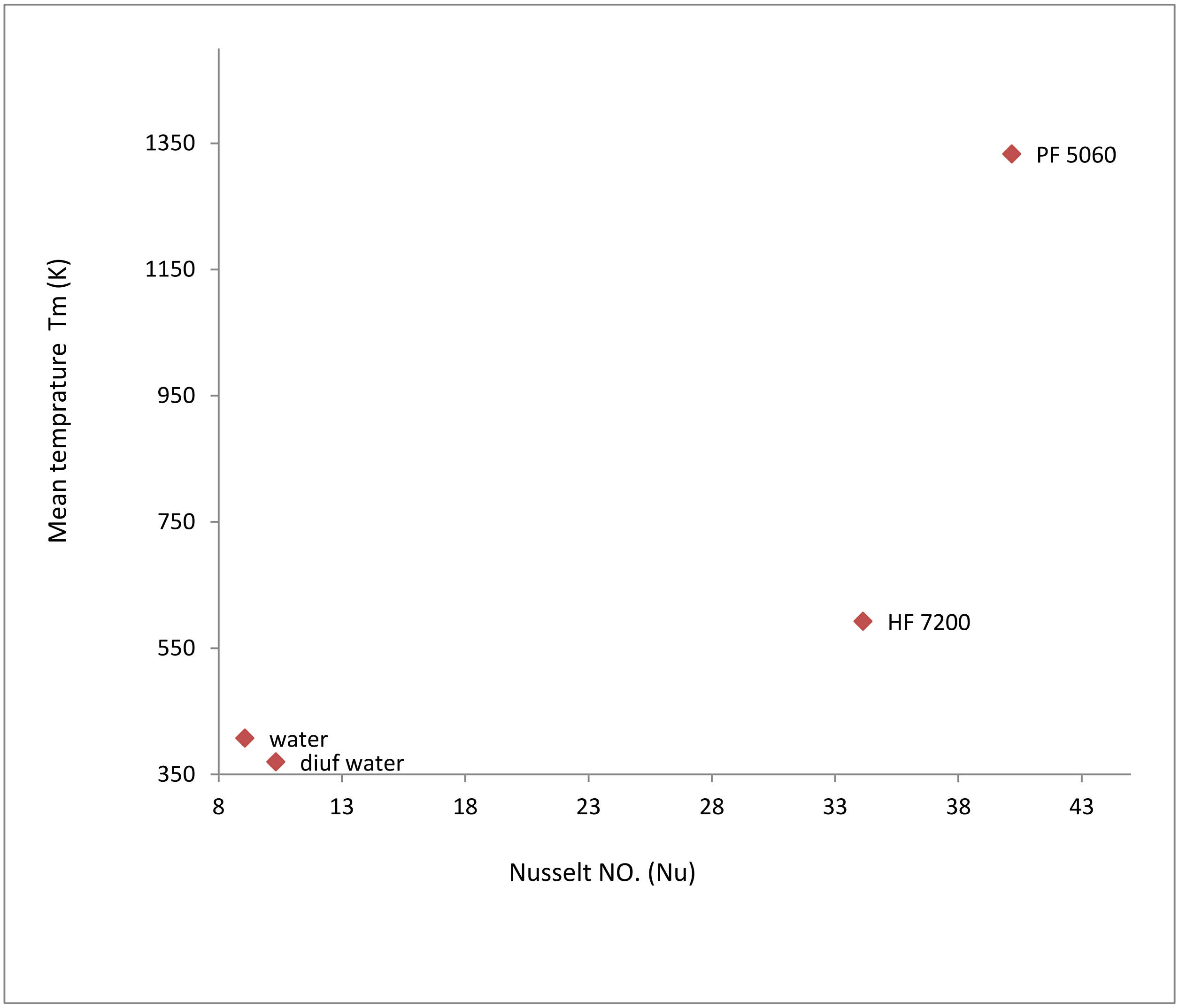
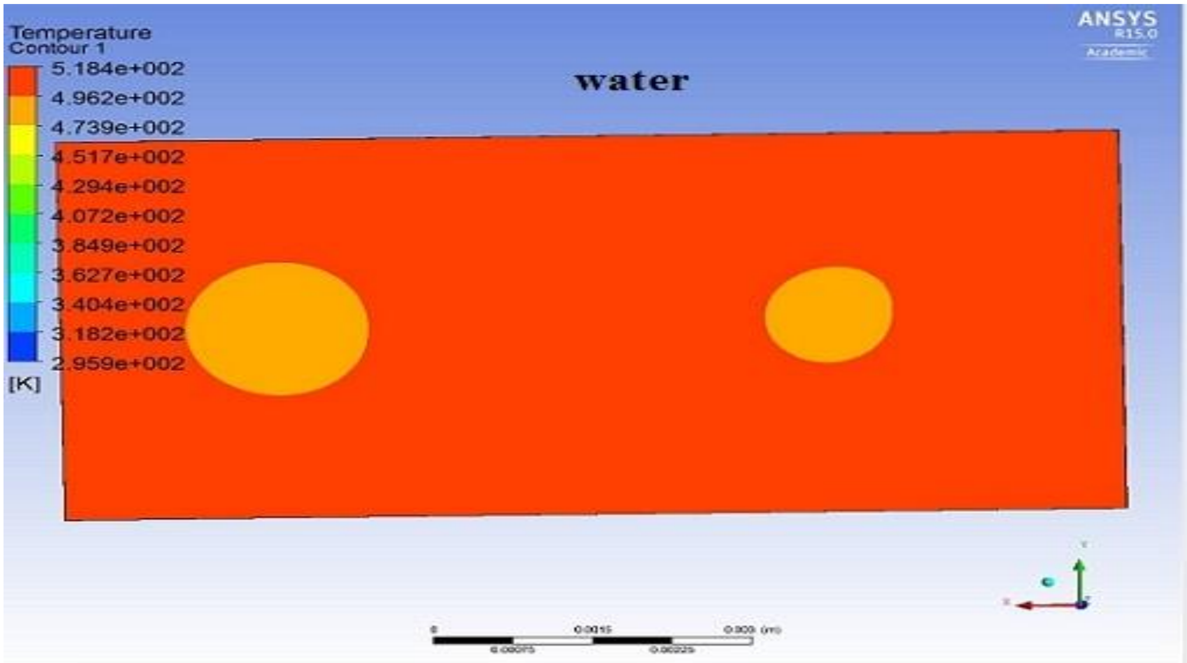
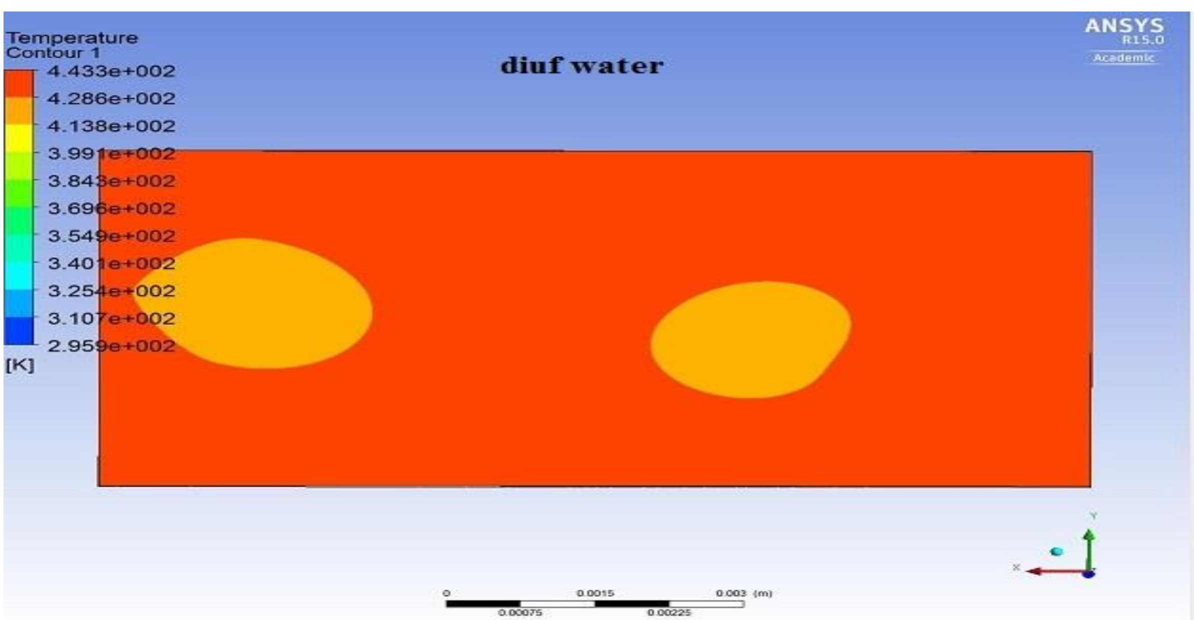


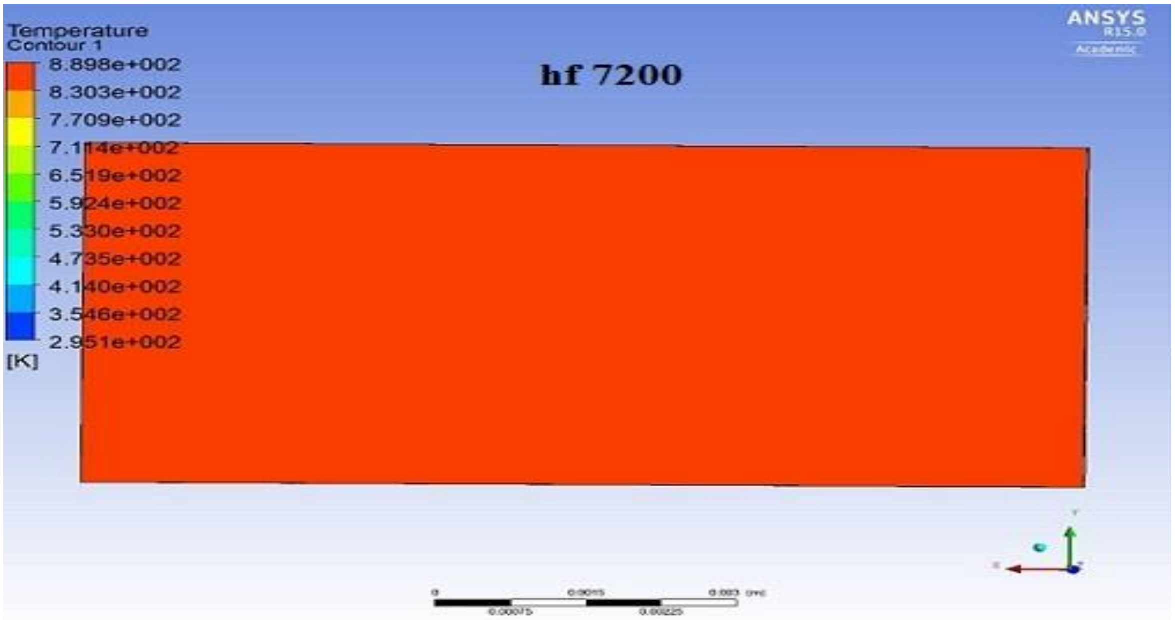
Fig. 4.7 Mean temperature and Nusselt number of different fluids



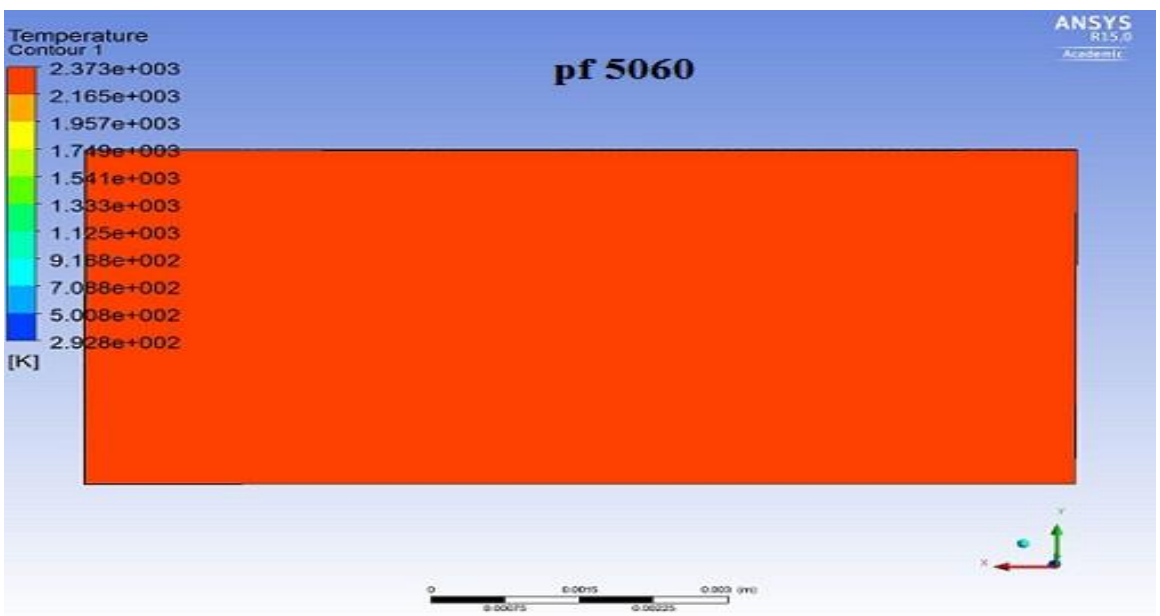
(a) Temperature distribution contour of inclined jet micro channel with water as fluid



(b) Temperature distribution contour of inclined jet in micro channel with DIUF water as fluid



(c) Temperature distribution contour of inclined jet micro channel with hf-7200 as fluid



(d) Temperature distribution contour of inclined jet micro channel with pf-5060 as fluid

Fig. 4.8 (a), (b), (c) and (d) temperature distribution contour of inclined jet micro channel for different fluids

4.4 SUBSTRATE NANO FLUID

For the performance fluid we used Nano fluid as fluid flow through the micro channel. The Nano fluid water mixed with Nano particle copper oxide (Cuo) in fraction of different volume. We used copper oxide (Cuo) 0.1%, 0.5% and 1.5% in fraction volume. During the simulation Nano fluid impingement through in micro channel we have obtained better result from water fluid in micro channel. We used copper as base metal in micro channel. Due to mixed Nano particle in water we justified our result.

We used Nano fluid (water+0.1%Cuo) in micro channel we obtained temperature difference 223.4 K through micro channel. On further simulation we used Nano fluid (water+0.5%Cuo) in micro channel we obtained temperature difference 227 K through micro channel On further simulation we used Nano fluid (water+1.5%Cuo) in micro channel we obtained temperature difference 414.4 K through micro channel in table 4.5. It has concluded that the Nano fluid gave beat heat transfer through the metal. It also characterized impingement data under various fluid flows in micro channel. We have obtained characteristics dimension in fluid flow. For the simulation of inclined jet micro channel used the Nano fluid in different category of volume fraction of fluid. In this category we have defined the Nano fluid in fraction of volume. The volume fraction in use of Nano fluid defined the maximum heat transfer at temperature drop. It consist the nusselt no. at required heat transfer at (fig 4.9).

As the mean temperature increase through the micro channel defied the nusslet number increase. During the Nano fluid flow in micro channel defined the maximum heat transfer at the volume fraction of Nano fluid. As increasing volume fraction decreased the heat transfer through micro channel (fig 4.10). After all simulation we have obtained temperature difference in increase order which conclude that Nano fluid give best heat transfer through the metal while using in microchannel.

As temperature increase in micro channel heat transfer increase, due to this effect at required mean temperature dominate the Nusselt number in increase manner. It applied that maximum heat transfer through micro channel at given temperature difference. The distribution of temperature drop at required micro channel (fig 4.11).

Table 4.5 Simulation result of different Nano fluids:

Nano fluid properties					Simulations		
Fluid	Density (kg/m^3)	Specific heat (J/kg –K)	Thermal cond.(W /m-K)	vicocity(kg/m-s)	Mean temp. (K)	Temp.diff. data (K)	Nusselt number (NU)
Water	998	4182	0.667	0.00089	407.15	223	9.07
Water+0.1% cuo	1002	4160.6	0.6068	0.000892	407.6	223.4	9.96
Water+0.5% cuo	1024	4078.5	0.6139	0.000901	409.4	227	9.86
Water+1.5% cuo	1077	3895.2	0.6019	0.000923	413.65	235.5	9.90

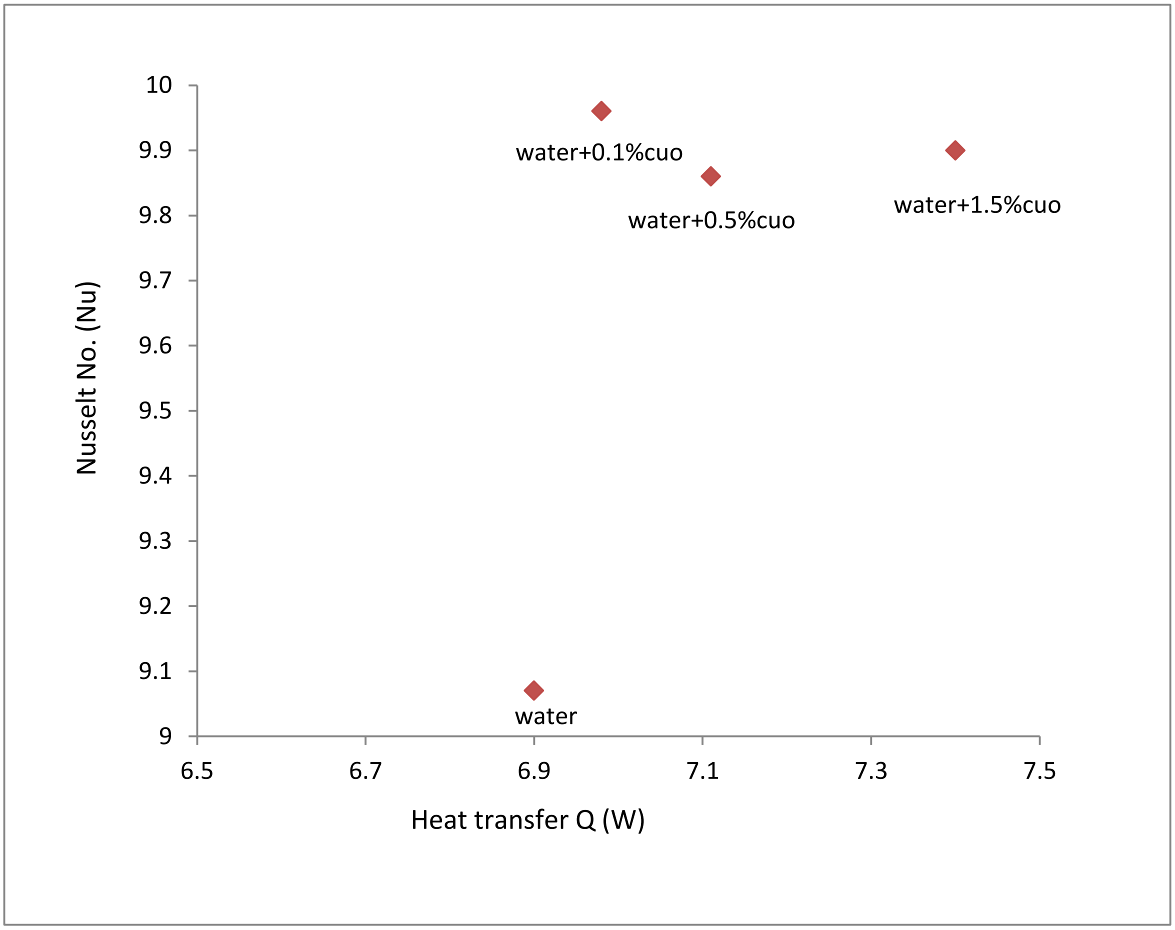


Fig. 4.9 Variation of Nusselt no. with heat transfer for water and different Nano fluids

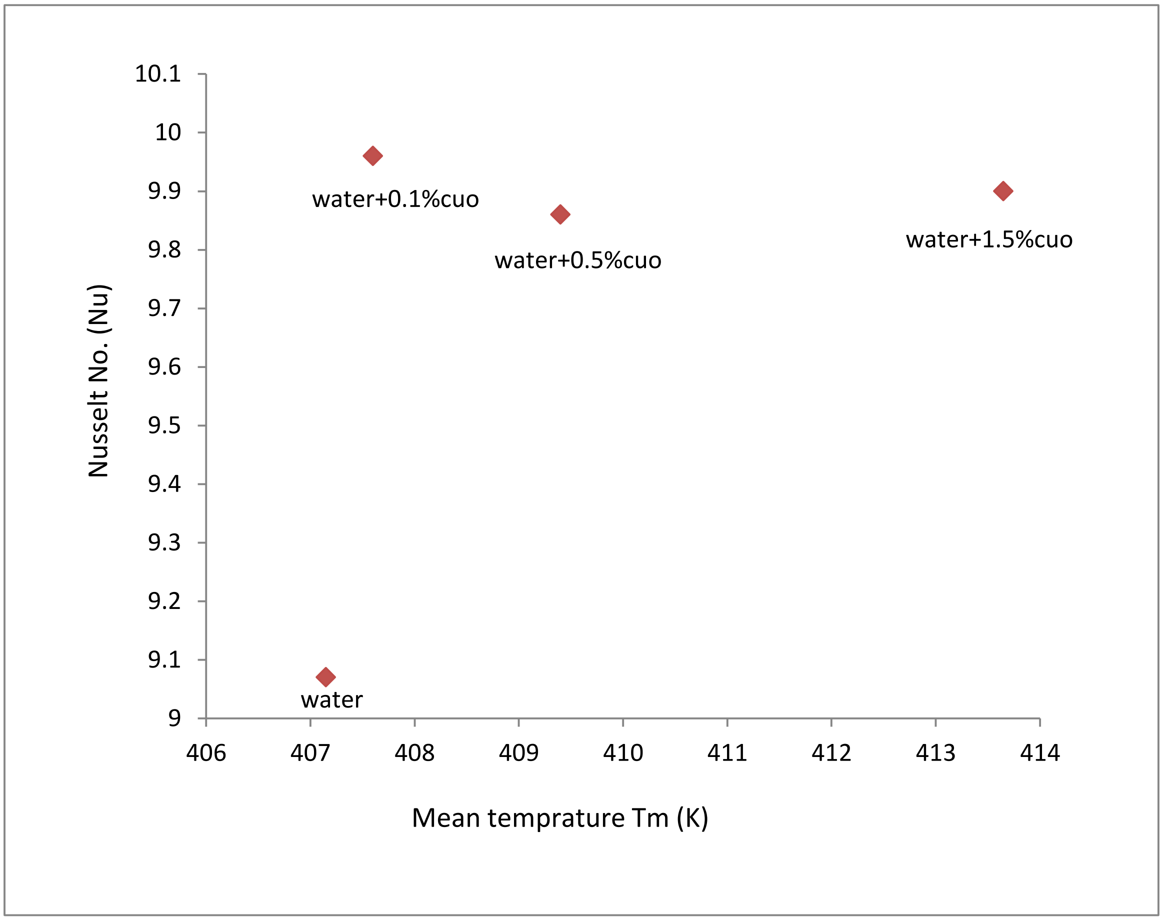
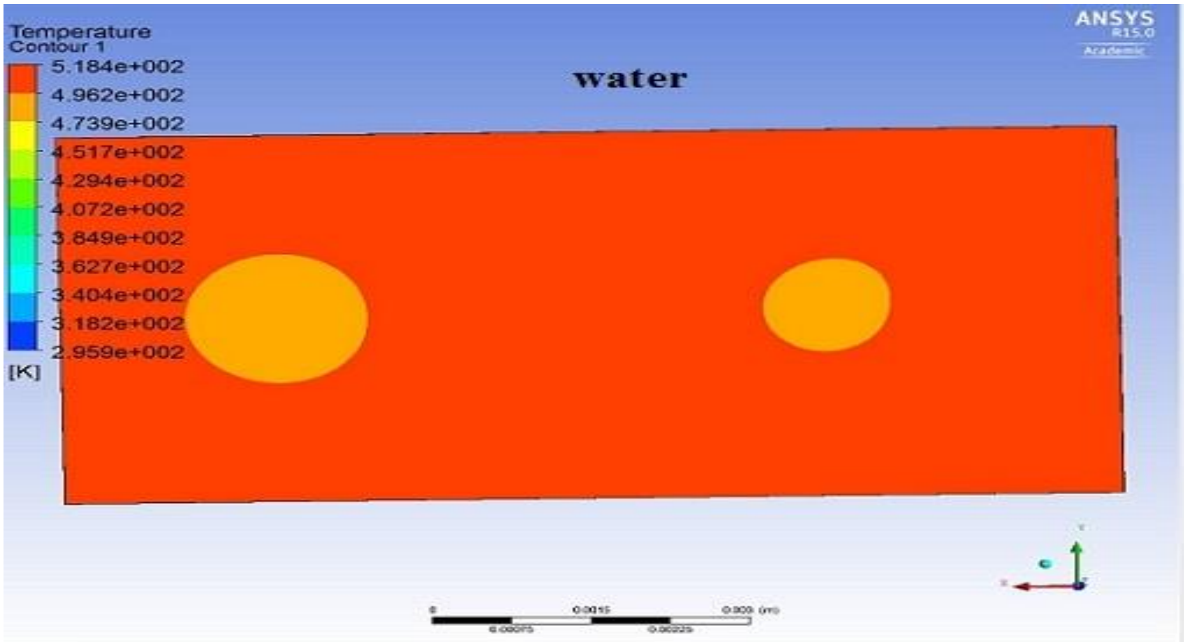
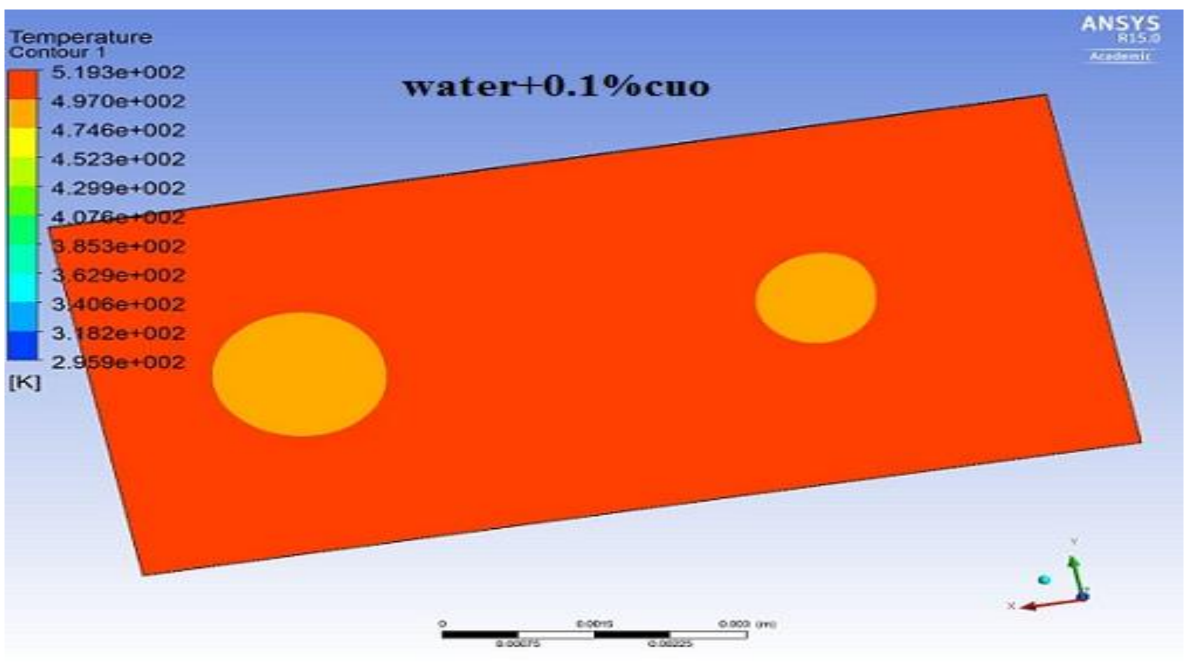


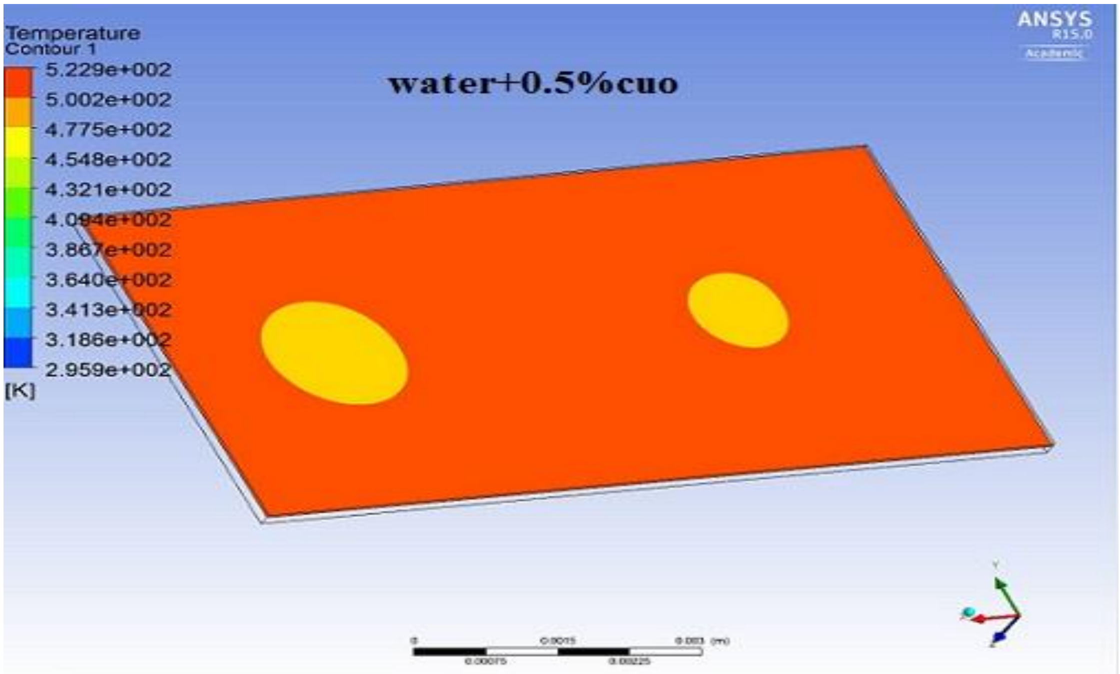
Fig. 4.10 Variation of Nusselt no. with mean temperature for water and different Nano fluids



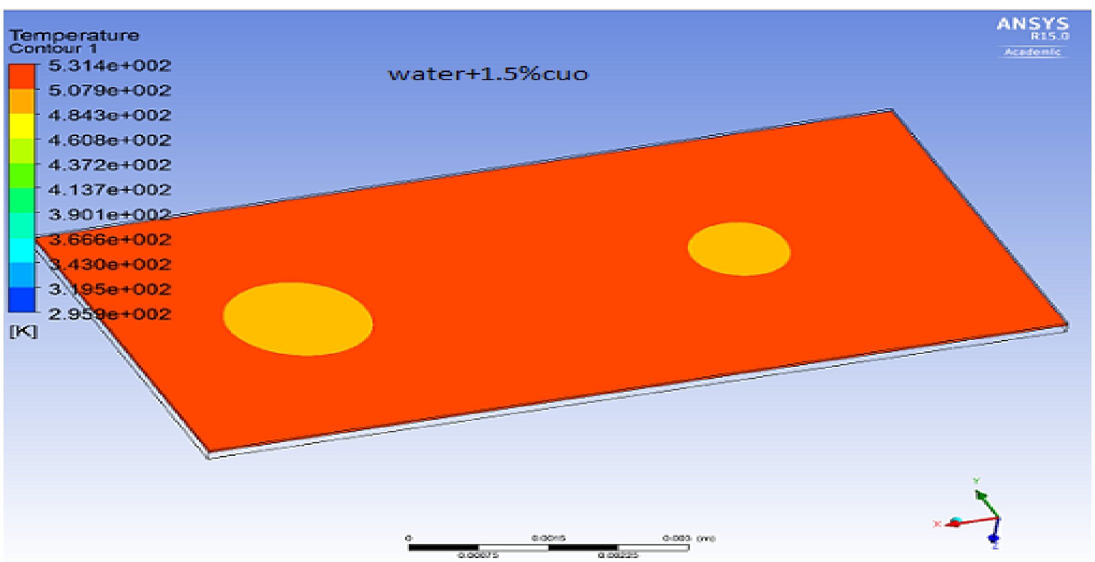
(a) Temperature distribution contour of inclined jet micro channel with water as fluid



(b) Temperature distribution contour of inclined jet micro channel with water+0.1% Cuo as Nano fluid



(c) Temperature distribution contour of inclined jet micro channel with water+0.5% Cuo as Nano fluid



(d) Temperature distribution contour of inclined jet micro channel with water+1.5 Cuo as Nano fluid

Fig. 4.11 (a), (b), (c) and (d) temperature distribution contour of inclined jet micro channel at different Nano fluids

CHAPTER 5

CONCLUSIONS AND RECOMMENDATIONS

CONCLUSIONS

After all inceptions we obtained some conclusion from simulation. It has been defined in different category above the chart. There are chance to conclude various flow through micro channel in single pass flow.

- In a single pass incline nozzle jet impingement in micro channel cooling device has been modeled and numerically solved.

- During different nozzle angle into micro channel has been Coted the best angle suited with highest temperature at the outlet of the system. After result it is suited that 45^0 best angle for heat transfer. For the heat transfer it is necessary to maximum temperature drop.

- During the study of substrate the material maximum temperature drop has been achieved in cooper with required water flow through it. It is necessary to highest conductivity of material liberate maximum temperature drop. It achieved maximum average temperature at 407.161 K.

- During the study of different material in substrate solid achieved best property of material in transferring the fluid. As per the result copper has maximum transfer of heat at required Nusselt number.

- As per material to propel different fluid in domain of fluid we have achieved the best heat transfer through the material in different result. As per the result water and DIUF water has law average temperature from the HF-7200 and PF-5060.
- For the maximum heat transfer from the material Nano fluid flow through micro channel. As per the result Nano fluid defined maximum average temperature through the system. In used of Nano fluids in micro channel we have obtained differentiate result. In the reference maximum heat transfer obtained with respect to volume fraction through the channel.

5.2 RECOMMENDATIONS FOR FUTURE STUDIES

In the future study from result obtained, I would recommended following studies.

- Contact of jet configuration.
- Jet hybrid system position.
- Modification of nozzle cross section area.

REFERENCES

[1] Ho, C.M. and Tai, Y.C., 1998. Micro-electro-mechanical-systems (MEMS) and fluid flows. Annual Review of Fluid Mechanics, 30(1), pp.579-612.

[2] Qu, W. and Mudawar, I., 2002. Analysis of three-dimensional heat transfer in micro-channel heat sinks. International Journal of heat and mass transfer,45(19), pp.3973-3985.

[3] 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 Mass Transfer, 47(19), pp.4215-4231.

[4] Gromoll, B., 1998. Micro cooling systems for high density packaging. Revue générale de thermique, 37(9), pp.781-787.

[5] Tuckerman, D.B. and Pease, R.F.W., 1981. High-performance heat sinking for VLSI. IEEE Electron device letters, 2(5), pp.126-129.

[6] Gamrat, G., Favre-Marinet, M. and Asendrych, D., 2005. Conduction and entrance effects on laminar liquid flow and heat transfer in rectangular microchannels. International Journal of Heat and Mass Transfer, 48(14), pp.2943-2954.

[7] Owhaib, W. and Palm, B., 2004. Experimental investigation of single-phase convective heat transfer in circular microchannels. Experimental Thermal and Fluid Science, 28(2), pp.105-110.

[8] Kandlikar, S.G., Joshi, S. and Tian, S., 2001. Effect of channel roughness on heat transfer and fluid flow characteristics at low Reynolds numbers in small diameter tubes. Atmosphere, 4(7).

[9] Yao, J., Patel, M.K., Yao, Y. and Mason, P.J., 2006, September. Numerical simulation of heat transfer in rectangular microchannel. In ECCOMAS CFD 2006: Proceedings of the European Conference on Computational Fluid Dynamics, Egmond aan Zee, The Netherlands, September 5-8, 2006. Delft University of Technology; European Community on Computational Methods in Applied Sciences (ECCOMAS).

[10] Qu, W. and Mudawar, I., 2002. Experimental and numerical study of pressure drop and heat transfer in a single-phase micro-channel heat sink. International Journal of Heat and Mass Transfer, 45(12), pp.2549-2565.

[11] Peng, X.F. and Peterson, G.P., 1996. Convective heat transfer and flow friction for water flow in microchannel structures. International journal of heat and mass transfer, 39(12), pp.2599-2608.

[12] Fedorov, A.G. and Viskanta, R., 2000. Three-dimensional conjugate heat transfer in the micro-channel heat sink for electronic packaging. International Journal of Heat and Mass Transfer, 43(3), pp.399-415.

[13] Glynn, C., O'Donovan, T. and Murray, D.B., 2005. Jet impingement cooling. In Proceedings of the 9th UK National Heat Transfer Conference, Manchester, England, September (pp. 5-6).

[14] Sung, M.K. and Mudawar, I., 2006. Correlation of critical heat flux in hybrid jet impingement/micro-channel cooling scheme. International Journal of Heat and mass transfer, 49(15), pp.2663-2672.

[15] Sung, M.K. and Mudawar, I., 2008. Single-phase hybrid micro-channel/micro-jet impingement cooling. International Journal of Heat and Mass Transfer,51(17), pp.4342-4352.

[16] Sung, M.K. and Mudawar, I., 2008. Effects of jet pattern on single-phase cooling performance of hybrid micro-channel/micro-circular-jet-impingement thermal management scheme. International Journal of Heat and Mass Transfer, 51(19), pp.4614-4627.

[17] Goldstein, R., Ibele, W.E., Patankar, S.V., Simon, T.W., Kuehn, T.H., Strykowski, P.J., Tamma, K.K., Heberlein, J.V.R., Davidson, J.H., Bischof, J. and Kulacki, F.A., 2010. Heat transfer—a review of 2005 literature.International Journal of Heat and Mass Transfer, 53(21), pp.4397-4447.

[18] Naphon, P. and Wongwises, S., 2010. Investigation on the jet liquid impingement heat transfer for the central processing unit of personal computers. International Communications in Heat and Mass Transfer, 37(7), pp.822-826.

[19] O'Donovan, T.S. and Murray, D.B., 2006. Effect of vortices on jet impingement Heat Transfer. In International Heat Transfer Conference 13. Begel House Inc..

[20] Sung, M.K. and Mudawar, I., 2006. Correlation of critical heat flux in hybrid jet impingement/micro-channel cooling scheme. International Journal of Heat and mass transfer, 49(15), pp.2663-2672.

[21] Sung, M.K. and Mudawar, I., 2006. Experimental and numerical investigation of single-phase heat transfer using a hybrid jet-impingement/micro-channel cooling scheme. International journal of heat and mass transfer, 49(3), pp.682-694.

[22] Barrau, J., Chemisana, D., Rosell, J., Tadrist, L. and Ibañez, M., 2010. An experimental study of a new hybrid jet impingement/micro-channel cooling scheme. Applied Thermal Engineering, 30(14), pp.2058-2066.

[23] Wang, E.N., Zhang, L., Jiang, L., Koo, J.M., Maveety, J.G., Sanchez, E.A., Goodson, K.E. and Kenny, T.W., 2004. Micromachined jets for liquid impingement cooling of VLSI chips. Journal of Microelectromechanical systems, 13(5), pp.833-842.

[24] Lee, D.Y. and Vafai, K., 1999. Comparative analysis of jet impingement and microchannel cooling for high heat flux applications. International Journal of Heat and Mass Transfer, 42(9), pp.1555-1568.

[25] Sung, M.K. and Mudawar, I., 2006. Experimental and numerical investigation of single phase heat transfer using a hybrid jet-impingement/micro-channel cooling scheme. International journal of heat and mass transfer, 49(3), pp.682-694.

[26] Yao, J., Patel, M.K., Yao, Y. and Mason, P.J., 2006, September. Numerical simulation of heat transfer in rectangular micro channel. In ECCOMAS CFD 2006: Proceedings of the European Conference on Computational Fluid Dynamics, Egmond aan Zee, The Netherlands, September 5-8, 2006. Delft University of Technology; European Community on Computational Methods in Applied Sciences (ECCOMAS).

[27] Lee, P.S., Garimella, S.V. and Liu, D., 2005. Investigation of heat transfer in rectangular microchannels. International Journal of Heat and Mass Transfer,48(9), pp.1688-1704.

[28] Barrau, J., Chemisana, D., Rosell, J., Tadrist, L. and Ibañez, M., 2010. An experimental study of a new hybrid jet impingement/micro-channel cooling scheme. Applied Thermal Engineering, 30(14), pp.2058-2066.

[29] CFX-Solver, A.N.S.Y.S., 2006. Theory Guide. Release ll.

[30] Raw, M., 1996. Robustness of coupled algebraic multi grid for the Navier-Stokes equations. AIAA paper, 96, p.0297.

[31] Incropera FP, DeWitt DP 2002 Fundamentals of heat and mass transfer. Wiley, New York

[32] Herwig, H. and Mahulikar, S.P., 2006. Variable property effects in single-phase incompressible flows through micro channels. International Journal of Thermal Sciences, 45(10), pp.977-981.

[33] Manay, E., Sahin, B., Yilmaz, M. and Gelis, K., 2012. Thermal performance analysis of Nano fluids in micro channel heat sinks. World Academy of Science, Engineering and Technology, 67.

[34] Murthy, M.S.R. and Gupta, A.V.S.S.K.S., 2012, November. Flow And Heat Transfer Characteristics Over Naca0018 Aero foil. In International Journal of Engineering Research and Technology (Vol. 1, No. 9 (November-2012)). ESRSA Publications.