COMPUTATIONAL FLUID DYNAMICS (CFD) ANALYSIS OF MICROCHANNEL HEAT SINK USING NANOFLUID

DISSERTATION

Submitted in Partial Fulfillment of the

Requirement for Award of the Degree

Of

MASTER OF TECHNOLOGY

In

MECHANICAL ENGINEERING

By

AMIT AGNIHOTRI

11002822

Under the Guidance of

Mr. AASHISH SHARMA

DEPARTMENT OF MECHANICAL ENGINEERING

LOVELY PROFESSIONAL UNIVERSITY

PHAGWARA, PUNJAB (INDIA) -144402

2015

Lovely Professional University Jalandhar, Punjab

CERTIFICATE

I hereby certify that the work which is being presented in the Dissertation entitled **"Computational fluid dynamics (CFD) analysis of microchannel heat sink using nanofluid"** in partial fulfillment of the requirement for the award of degree of **Master of Technology** and submitted in Department of Mechanical Engineering, Lovely Professional University, Punjab is an authentic record of my own work carried out during period of Dissertation under the supervision of **Mr. Aashish Sharma**, **Assistant Professor**, Department of Mechanical Engineering, Lovely Professional University, Punjab.

The matter presented in this dissertation has not been submitted by me anywhere for the award of any other degree or to any other institute. .

Date: **Amit Agnihotri**

This is to certify that the above statement made by the candidate is correct to best of my knowledge.

Date: **Mr. Aashish Sharma**

Supervisor

The M-Tech Dissertation examination of Amit Agnihotri, has been held on

Signature of Examiner

ACKNOWLEDGEMENT

I would like to express my sincere gratitude to Assistant Professor **Mr. Aashish Sharma** Department of Mechanical Engineering, Lovely Professional University Jalandhar for suggesting the topic for my thesis report and for his always ready guidance throughout the course of my dissertation II report. I am highly indebted to him for his creative ideas and criticism at time to time during the entire course of dissertation II.

I am also extremely thankful to **Dr. Rajiv Kumar Sharma** HOS of Mechanical Engineering Department, Lovely Professional University Jalandhar for valuable suggestions and encouragement.

ABSTRACT

A numerical simulation of three dimensional heat transfer in a silicon based microchannel heat sink has been conducted using nanofluid (TiO_2-H_2O) by SIMPLE method. Model of microchannel consists of trapezoidal channel. Dimensions of trapezoidal microchannel are 10 mm length, 280 μm channel top width, 225 μm channel bottom width, 431 μm channel hypotenuse and 430 μm channel height. Influence of properties of nanofluid on the heat transfer is investigated. Different parameters like heat transfer coefficient, Nusselt number, heat flux, outlet temperature are studied for different pressure drop. Pumping power depends upon pressure difference. So power consumption can be optimized by this study. Result shows that heat transfer coefficient is high in comparison to the water as a coolant in microchannel heat sink. Because of boundary layer, variation of Nusselt number decreases along the flow direction.

TABLE OF CONTENTS

ACKNOWLEDGEMENT	**i**
ABSTRACT	**ii**
LIST OF FIGURES	**vi**
LIST OF TABLES	**ix**
NOMENCLATURE	**x**
Chapter 1: INTRODUCTION	**1-9**
1.1 General	1
1.2 Causes of Heat Generation in Computer System	2
1.3 Different Cooling Methods for Computer System	3
1.3.1 Air Cooling	3
1.3.2 Liquid Immersion Cooling	4
1.3.3 Hybrid Air-Water Cooling	6
1.3.4 Refrigerant Cooling	6
1.3.5 Heat Sink	6
1.3.6 Microchannel Heat Sink	7
1.3.7 Working Fluid in MCHS	9
Chapter 2: TERMINOLOGY	**10**
Chapter 3: SCOPE OF STUDY	**11**

Chapter 4: LITERATURE REVIEW **12-19**

4.1 Experimental Study 12

4.2 Numerical Study 13

4.3 Other Study 18

Chapter 5: OBJECTIVE OF THE STUDY **20**

Chapter 6: RESEARCH METHODOLOGY **21-31**

6.1 Introduction 21

6.2 Mesh Generated In ICEM CFD 14.0 24

6.3 Boundary Conditions and Governing Equations 25

6.3.1 Boundary Conditions 26

6.4 Properties of Nanofluid and Sink Material 28

6.5 FLUENT as Solver 30

Chapter 7: RESULTS AND DISCUSSION **32-48**

7.1 Validation of Study 32

7.1.1 Convergence Graph 32

7.1.2 Grid Independence Test 32

7.1.3 Validation of Results with Numerical Study 33

7.2 Temperature Contours 34

7.3 Pressure Contours 43

7.4 Velocity Vector 44

7.5 Heat transfer coefficient 44

7.6 Nusselt Number 45

7.7 Variation of Heat Transfer Coefficient with Heat Flux 47

7.8 Variation of Nusselt number with Heat Flux 48

Chapter 8: CONCLUSION AND FUTURE SCOPE **49-50**

REFERENCES	**51-54**
APPENDIX I	**55-63**

LIST OF FIGURES

S.No	Title	Page No.
1.1	Cooling Fan	3
1.2	A liquid cooled CPU in a clear case	4
1.3	A pump for a liquid cooled PC	5
1.4	A heat sink uses lots of surface area to transfer heat from electronic components to air	7
1.5	Microchannel heat sink with manifolds	8
6.1	Isometric view of complete microchannel heat sink created In PRO-E Wildfire 4.0	21
6.2	Various Dimensions of MCHS (in mm)	22
6.3	Isometric view of single microchannel in heat sink created in PRO-E Wildfire 4.0	23
6.4	Geometry imported in ICEM CFD 14.0	24
6.5	Mesh generated in ICEM CFD 14.0	25
6.6	Solid view of Mesh	25
7.1	Convergence Graph	32
7.2	Validation of results by numerical study in terms of heat transfer coefficient	33
7.3	Temperature distribution along the channel	34
7.4	Temperature distribution at inlet	35
7.5	Temperature distribution at outlet under pressure 50000 Pa and 900000 W/m^2 heat flux	35

7.6	Temperature distribution at outlet under pressure 10000 Pa and 900000 W/m^2 heat flux	35
7.7	Temperature distribution at outlet under pressure 20000 Pa and 900000 W/m^2 heat flux	36
7.8	Temperature distribution at outlet under pressure 35000 Pa and 900000 W/m^2 heat flux	37
7.9	Temperature distribution at outlet under pressure 65000 Pa and 900000 W/m^2 heat flux	37
7.10	Temperature of fluid inside the channel under pressure 50000 Pa and 900000 W/m^2 heat flux	38
7.11	Temperature of fluid inside the channel under pressure 10000 Pa and 900000 W/m^2 heat flux	38
7.12	Temperature of fluid inside the channel under pressure 35000 Pa and 900000 W/m^2 heat flux	38
7.13	Temperature of fluid inside the channel under pressure 20000 Pa and 900000 W/m^2 heat flux	39
7.14	Temperature of fluid inside the channel under pressure 65000 Pa and 500000 W/m^2 heat flux	39
7.15	Temperature of fluid inside the channel under pressure 50000 Pa and 500000 W/m^2 heat flux	39
7.16	Temperature of fluid inside the channel under pressure 50000 Pa and 1500000 W/m^2 heat flux	40
7.17	Temperature of channel in pressure 50000 Pa and 900000 W/m^2 heat flux	40

7.18	Temperature of channel in pressure 10000 Pa and 900000 W/m^2 heat flux	41
7.19	Temperature of channel in pressure 35000 Pa and 900000 W/m^2 heat flux	41
7.20	Temperature of channel in pressure 20000 Pa and 900000 W/m^2 heat flux	42
7.21	Temperature of channel in pressure 65000 Pa and 900000 W/m^2 heat flux	42
7.22	Temperature of channel in pressure 50000 Pa and 500000 W/m^2 heat flux	43
7.23	Temperature of channel in pressure 50000 Pa and 1500000 W/m^2 heat flux	43
7.24	Pressure distribution inside the channel	44
7.25	Velocity Vectors	45
7.26	Variation of heat transfer coefficient in direction of flow	45
7.27	Variation of heat transfer coefficient for nanofluid and water	46
7.28	Variation of Nusselt number in direction of flow	47
7.29	Variation of Nusselt number for nanofluid and water	48
7.30	Variation of heat transfer coefficient along the direction of flow	49
7.31	Variation of Nusselt number along the direction of flow	49

LIST OF TABLES

S.No	Title	Page No.
1.1	Different Causes of Failure of Electronic Chip	2
6.1	Dimensions of Trapezoidal Channel	23
6.2	Properties of Silicon	28
6.3	Properties of Water	30
6.4	Properties of TiO_2-H_2O	30
7.1	Grid Independence Test	33
7.2	Outlet temperature at different pressure drop	44

NOMENCLATURE

CFD	Computational Fluid Dynamics
IC	Integrated circuit
MCHS	Microchannel Heat Sink
CPU	Central Processing Unit
TIM	Thermal Interface Material
TiO_2	Titanium Dioxide
H_2O	Water
TGMCHS	Trapezoidal Grooved Microchannel Heat Sink
RAM	Random Access Memory
SIMPLE	Semi Implicit Method for Pressure Linked Equation
Al_2O_3	Aluminum Oxide
SiO_2	Silicon dioxide
CuO	Copper oxide

Chapter 1 INTRODUCTION

1.1 General

Cooling of electronic chip is essential for the proper functioning of electronic device, in which it is used. At present time it has become major concern to electronic packaging engineers. Most of the electronic devices face problem of improper cooling of electronic chip, which results failure of device. Failures are caused due to temperature rise because of accumulation of heat. Electronic gadgets have become ultra-compact. With invention of integrated circuit, trend has shifted toward higher packaging density. In past few decades an exponential increment occurred in density of circuit.

With the rapid development of the information technology (IT) industry, the heat flux in integrated circuit (IC) chips cooled by air has almost reached its limit about 1000000 W/m^2. Even in some applications, in high technologies, require heat fluxes well beyond such a limitation. So for the further development of IT industry a better cooling method has become a bottleneck problem. Even after the replacement of air cooling by liquid cooling, results are not totally satisfactory.

For development of compact and efficient thermal management technology for advanced electronic devices, cooling devices have to be in light-weight, small in size and of high performance. Microchannel heat sink can play an important role in this requirement. Microchannels are very fine channels of the width of a normal human hair and are widely used for electronic cooling purposes. In a MCHS, multiple microchannels are stacked together, which can increase the total contact surface area for heat transfer enhancement. MCHS reduces the total pressure drop by dividing the flow among many channels. Liquid or gas is used as a coolant to flow through these microchannels. The large surface area of MCHS enables the coolant to take away large amounts of energy per unit time per unit area while maintaining a considerably low device temperature. Using these MCHS, heat fluxes can be dissipated at relatively low surface temperatures.

This chapter deals with the causes of heat generation in computer system, different cooling methods for computer system like air cooling, liquid immersion cooling hybrid air-water cooling and refrigerant cooling. After it heat sink is described along with microchannel heat sink. Different working fluid which are used in microchannel heat sink are also discussed in last section of first chapter.

1.2 Causes of Heat Generation in Computer System

Following table shows the causes of failure of electronic chip and percentage corresponding to the cause-

Cause of Failure	Percentage Of Failure
Vibration	20%
Temperature	55%
Dust	6%
Humidity	19%

Table 1.1 Different Causes of Failure of Electronic Chip

From table it is very clear that failures due high temperature is very high. It is due to the heat generation. Heat generated by CPU is function of frequency and voltage upon which it operate. It also depends upon the design and construction of CPU. During operation, the temperature of components will rise until the thermal equilibrium is achieved. It means temperature gradient between the computer's components and their surroundings is such that the rate at which heat is lost to the surroundings is equal to the rate at which heat is being produced by the electronic component.

Dust blocks the flow of air. It acts as a thermal insulator. Thus it effect's the performance of cooling fan and also of heat sink. Vibration and humidity also contribute in heat generation and ultimately failure of system.

1.3 Different Cooling Methods for Computer System

Before going in detail of microchannel heat sink, it is important to know the various types of techniques used for cooling of integrated circuit. It may be chipset, central processing unit (CPU) in computer industry and data centers. The techniques used are listed below-

1.3.1 Air Cooling

In this method air is moved around or to computer enclosures by some means. Generally fans are commonly used for accomplishing that task. The term computer fan usually refers to fans attached to computer enclosures, but may also be intended to signify any other computer fan, such as a CPU fan**.** Different fans of varying sizes are available as from 40 to 240 mm. In desktops one or more fans are used for heat dissipation. It is recommended by most manufacturers that to bring fresh and cool air in from the bottom front of the case, and exhausting warm air from the top rear.

Air cooling is also used in laptops. In these, fan forces the air through small port. So fan can be choked. It will cause overheating. Ultimately failure of component may occur.

In data centres, number of computers is very high. There is a risk of overheating of various components if proper cooling method is not applied. There are a number of flat severs. Air enters from the front of the rack and comes out from the back. Generally in such application heating ventilation air conditioning is applied.

Fig 1.1 Cooling Fan

Source: http://www.extremetech.com/computing/107147-intels-liquid-cpu-cooler-is-water-worth-the-cost

1.3.2 Liquid Immersion Cooling

No fan is used in this cooling method. Components of computers to be cooled are submerged into the thermally conductive liquid. Liquid must have low electrical conductivity. It must not interfere with working of computer's components. Parts susceptible to electromagnetic interference must be insulated. Dielectric liquid should be preferred. In this method, evaporation can create a problem. Refilling of liquid should be done regularly. Sometimes liquid seeps in damaged components. Finally, it will cause failure of component. The liquids used in the cooling of electronic equipment must meet several requirements, depending on the specific application.

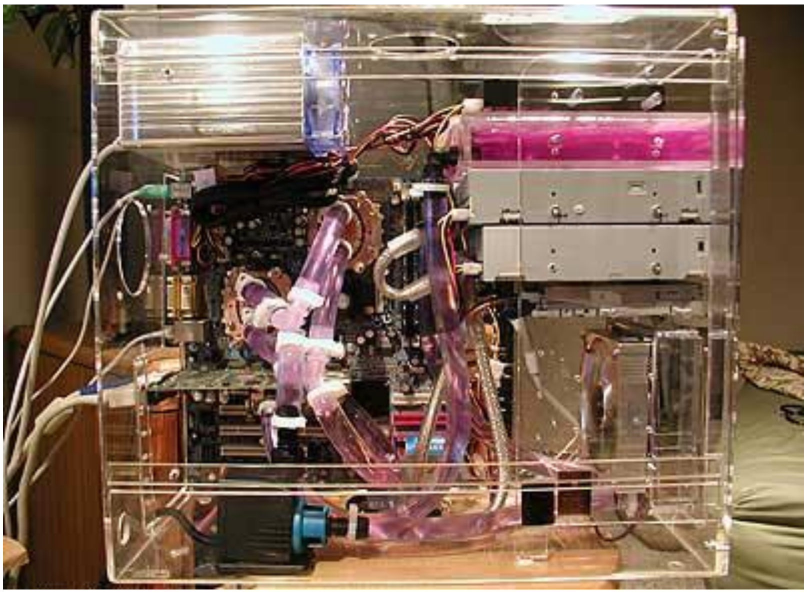


Fig 1.2 A liquid cooled CPU in a clear case

Source: http://computer.howstuffworks.com/liquid-cooled-pc.htm

Desirable characteristics of cooling liquids include –

- High Thermal Conductivity – It will yield high heat transfer coefficients.
- High Specific Heat –It requires smaller mass flow rate.
- Low Viscosity – It causes a smaller pressure drop and thus requires a smaller pump.
- High Surface Tension- Leakage problems are reduced by it.
- Chemical Inertness – It should not react with surfaces with which it comes into contact.
- Chemical Stability - It should not decompose under prolonged use.

- Nontoxic- It should be safe for personnel to handle.
- Low Freezing And High Boiling Points – It extends the useful temperature range.
- Low Cost- Different fluids may be selected in different applications because of the different priorities set in the selection process.

Pump is used in this system for feeding liquid. Some liquid cooling pumps are submersible, and can be placed directly inside the coolant reservoir. Pump is shown in following figure –

Fig 1.3 A pump for a liquid cooled PC

Source: http://computer.howstuffworks.com/liquid-cooled-pc3.htm

1.3.3 Hybrid Air-Water Cooling

In this type of system, a water-cooled heat exchanger extracts heat from heated air stream and reduces temperature. This cooling system consists of a heat exchanger of air-to-water finned tube type. Forced type convection with air was used to cool the module located on board. High temperature air entered into a heat exchanger before entering into next module. Up to fifty percent of heat is transferred to cooling water by high temperature air. Liquid-to-air type cooling system is a variation of this type cooling system. In this type cooling, there is a sealed loop. Through this coolant is circulated. There is an electronic module through which cold plate is attached to dissipate heat. Heat is carried away by liquid to heat exchanger of air cooled type. Further it is transferred to atmospheric air.

1.3.4 Refrigerant Cooling

In this type of cooling system, chemical coolants are used. These systems work like very small refrigerator. A block is added on processor, liquid runs over the block and transfers heat away from the chip and outside the case. This cooling system also removes the need of noisy case cooling fans. Cost of refrigerant cooling system is very high.

1.3.5 Heat Sink

It is a block of metal which is attached to the part which needs cooling. In CPU of personnel computers, it is clamped over the chip. A thermal conductive material is used in between block of material and component upon which it is clamped. It is called thermal interface material (TIM). Generally thermal grease or thermally conductive pad is in between the both. Microprocessors and power handling semiconductors are examples of electronics that need a heat sink to reduce their temperature through increased thermal mass and heat dissipations. Heat sinks have application in heat engines, refrigeration and cooling of electronic devices like microprocessors etc.

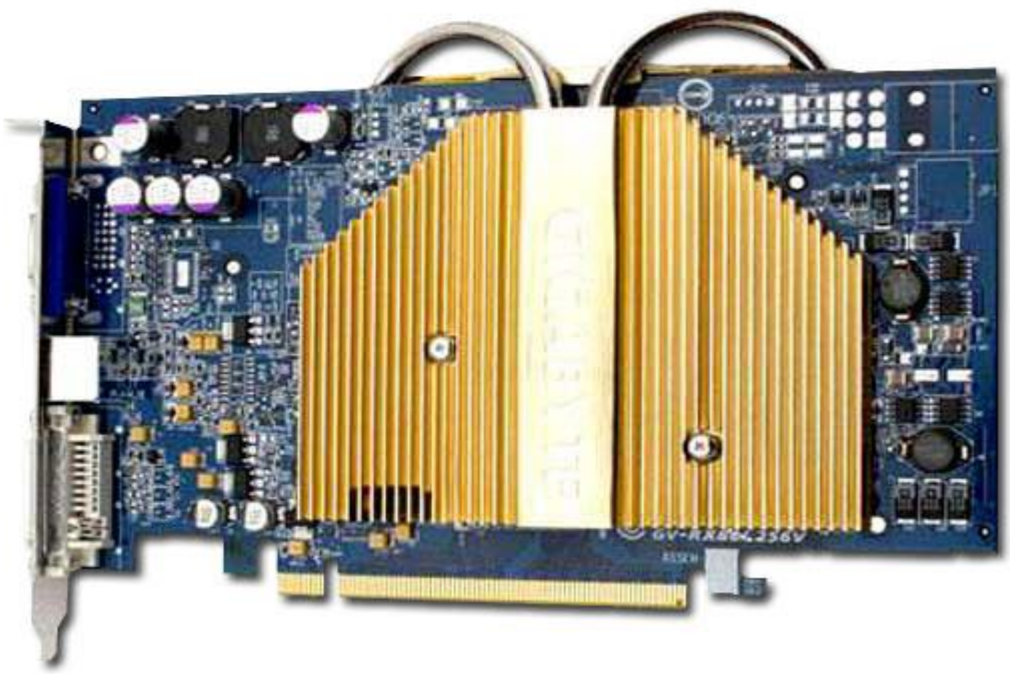


Fig 1.4 A heat sink uses lots of surface area to transfer heat from electronic components to air Source: http://computer.howstuffworks.com/liquid-cooled-pc1.htm

These are made of different materials. Figure shows the thermal conductivities of different material used for heat sink-

1.3.6 Microchannel Heat Sink

Microchannel heat sinks were introduced in the early 1980s to be used as a means of cooling integrated circuits. The implementation of manifold microchannel heat sink, cooling micro heat pipes, pool boiling, multiphase flow, liquid metal heat sink and microchannel heat sink are also proposed for the cooling solution in microelectronics. Heat is removed fifty times more effectively than conventional systems by microchannel heat sink. Thermal stress is induced on the chips by one layer type microchannel heat sink due to high temperature. High temperatures can be avoided by high pressure drop. Because high pressure drop causes rapid flow of coolant in channel. It will require a pumping system which will be large and noisy. Multi-layered MCHS are strong improvement on previous designs of one layered MCHS. There is current flow mechanism in multi-layered microchannel heat sink. Thermal performance of these MCHS was analysed. To optimize the design parameters a procedure was suggested. Although system of power supply in multilayer MCHS is simpler than one layered MCHS yet temperature rise was less on base surface.

In comparison to one layered MCHS, required pressure drop is significantly less in multi-layered MCHS. It has been shown that the thermal resistance is low up to 0.03 ^{0}C/W for MCHS. Thermal resistance is significantly lower in multi layered MCHS than conventional heat sinks. Dimension of micron scale in microchannels increases heat transfer rate substantially in comparison to conventional heat sinks. It is advantageous for electronics cooling.

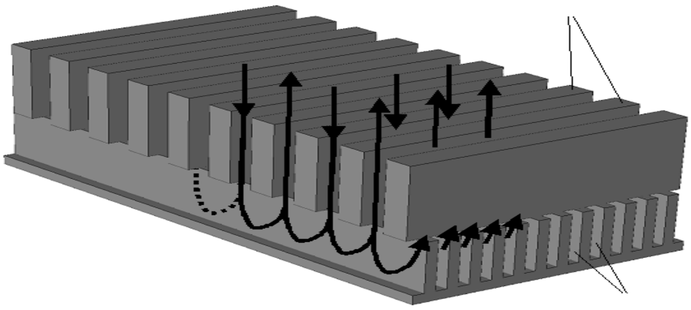


Fig 1.5 Microchannel heat sink with manifolds

Source: www.uk.comsol.com/model/download/.../microchannel_heatsink

Microchannel heat sinks have attracted special attention because of their ability size. Less coolant is required in it. An extensive research has already been done on microchannels. Still researchers are working in this field in present. Electronic packaging engineers got some relief by the introduction of nanofluids in the field of cooling of electronic devices. To increase the efficiency and the performance of systems, nanofluids can be used. Suspension of nanosized particles of metals, oxides, carbides or carbon nanotubes in base fluid is known as nanofluid. Thermal conductivity of the nanofluids is more so these exhibit better results.

1.3.7 Working Fluid in Microchannel Heat Sink

With the rapid development of the information technology (IT) industry, the heat flux in integrated circuit (IC) chips cooled by air has almost reached its limit about 100 W/cm2. So water is used as a coolant. Nanofluids are also tested by researchers. The MCHS wall temperature decreases with the increase of particle volume fraction under the extreme heat flux of 1000 W/m2 as compared with lower heat flux. The heat transfer coefficient at φ=5% of Al_2O_3–H_2O nanofluid flow could not enhance or performing almost the same result as pure water compared with lower particle volume fractions. The presence of the nanoparticles in water flowing through MCHS appears to give only a slight rise in the friction factor values. Thus the friction factor increases as the percentage of nanofluids particle volume fraction increases. Besides these, the hot water cooling enables efficient recovery of heat dissipated by the even hotter chip by using hot water recovered from a secondary application.

Heat Sink

Heat sink is a metal object which is placed over an electronic chip. Heat is transferred from one side of heat sink which is dissipated from another surface.

Microchannel Heat Sink

Microchannel heat sink has a number of channels through which coolant flows. Heat is carried away by this coolant from hot device.

Nanofluid

Suspension of nanosized particles of metals, oxides, carbides or carbon nanotubes in base fluid is known as nanofluid. Different base fluids are being used like glycerin, water, oil, ethylene glycol. One of the most important property of nanofluids is their high thermal conductivity. Advantages of nanofluid include enhanced heat transfer. It will lead to lighter and smaller heat exchanger. Disadvantages of nanofluid include channel clogging, sedimentation of particles, costly, toxicity, erosion.

Computational Fluid Dynamics

Science of predicting fluid flow, heat and mass transfer, and other related phenomenon by solving mathematical equations which govern these processes using numerical methods is known as Computational Fluid Dynamics.

ICEM CFD and FLUENT

ICEM CFD is a complete pre-processing software. Geometry can be created in it or imported. Mesh can be created in ICEM. This mesh cab be exported for several solvers like FLUENT, CFX etc.

FLUENT is a solver. User applies different operating, boundary conditions for a mesh imported in FLUENT. FLUENT solves the problem according to applied conditions.

Various nanofluids have developed at present. Microchannels show different characteristics for different nanofluids. So it is important to investigate for suitable combination of nanofluid and geometry of channel, which provides better performance of microchannel heat sink. It is suitable to investigate this using computational fluid dynamics (CFD). CFD is the science of predicting fluid flow, heat transfer, mass transfer, chemical reactions and related phenomenon by solving mathematical equations which govern these processes using numerical methods.

Present research deals with fluid flow and heat transfer analysis of microchannel heat sink. Channel is of trapezoidal shape. Nanofluid is used as a coolant in microchannel. Nanofluid TiO_2-H_2O is taken as a coolant. Effect of this nanofluid on the performance of microchannel heat sink is studied. Pro E, ICEM CFD 14.0 and FLUENT 14.0 softwares are used in this research. Heat transfer coefficient will be calculated. Nusselt number will also be calculated. Water is taken as a base fluid for nanofluid (TiO_2-H_2O). For volume fraction (ϕ = 2%), properties of nanofluid (TiO_2-H_2O) are calculated using correlations available from literature review. Silicon is taken as material of heat sink. This study will also cover a comparison of performance of microchannel heat sink having water as coolant and nanofluid (TiO_2-H_2O) as coolant. Geometry of cooling model is created in Pro E Wildfire 4.0. It is imported in ICEM CFD 14.0, in which mesh is created. This mesh will be read by FLUENT 14.0. Results will be obtained for different pressure drop and amount of heat flux at the bottom of single microchannel heat sink by applying different boundary conditions, operating conditions and other conditions.

A substantial amount of work has been done in the area of microchannel heat sink for cooling of electronic chip. Studies are performed on heat transfer rate through channel, thermal resistance, Nusselt number, friction factor, entropy generation and some other parameter also. Literature review is classified in following three categories-

4.1 Experimental Study

Tuckerman and Pease [1] performed research on microchannel heat sink in 1981. It was concluded that heat transfer rate would increase by decreasing liquid cooling channel dimensions. In it, for dissipating the heat, silicon microchannels were used with water as working fluid. They had a channel width of approximately 60 μm and a parameterized channel height varying between 287 μm and 376 μm.

G. Hetsroni et al. [2] has performed investigation of a heat sink for cooling of electronic devices by using a dielectric liquid as a coolant. The temperature on the hot surface maintained in the range 323–333 K by cooling fluid in this study. It was concluded that the entire behaviour of temperature variations corresponds to that of pressure variations. The maximum pressure variations did not differ significantly from the pressure drop across the channels.

Piyanut Nitiapiruk et al. [3] investigated the performance of a microchannel heat sink using TiO_2/water nanofluid experimentally. Width, height and length of microchannel were 500, 800 and 40 mm. There were 40 channels in microchannel heat sink. Effects of different thermal and physical properties on the Nusselt number and friction factor were investigated by using three different sets of thermophysical models. These were based on theoretical and experimental relations. It was concluded that the use of the model which is based on experimental data is very important to estimate the friction factor on using different models. Prediction of Nusselt number did not depend upon thermal conductivity much.

H.Y. Wu, Ping Cheng [4] experimentally investigated the laminar convective heat transfer and pressure drop of water in 13 different trapezoidal silicon microchannels. It was determined that the Nusselt number and apparent friction constant depend greatly on different geometric parameters. Nusselt number and friction constant increased with the increase of surface roughness and surface hydrophilic property. These increments became more obvious at higher Reynolds numbers. It was also showed experimentally that the Nusselt number would increase almost linearly with the Reynolds number at low Reynolds numbers (Re < 100), but increases slowly at a Reynolds number greater than 100.

4.2 Numerical Study

Harpole and Eninger [5] suggested mainfold microchannel heat sinks first time. There were many inlets and outlets in manifold MCHS in comparison to the conventional microchannel heat sinks along the length of the microchannels. Coolant flows from the inlet port in to the manifolds and forms different streams. Every stream flowed through a short part of microchannels. The pressure drop was reduced by a factor equal to the number of manifold inlet/outlet compared with conventional microchannel heat-sink.

Li et al. [6] numerically simulated a forced convection heat transfer with rectangular micro channel. Width, depth and hydraulic diameter of microchannel 57 μm, 180 μm and 86 μm respectively, fabricated along entire length. The effect of the geometric parameters of the channel and thermo physical properties of the fluid on the flow and the heat transfer, were investigated using temperature dependent thermo physical property method. Result showed that these liquid's properties could significantly influence both the flow heat transfer in the micro channel heat sink. The result indicated that the variations in the way the Nusselt number is defined may results in different conclusions even using the same experimental data.

Weilin Qu, Issam Mudawar [7] performed numerical investigation the three-dimensional fluid flow and heat transfer in a rectangular micro-channel heat sink numerically using water as the cooling fluid. A numerical code based on the finite difference method and the SIMPLE algorithm was developed to solve the governing equations. It was found that the temperature rise along the flow direction in the solid and fluid regions can be approximated as linear. The highest temperature was spotted at the heated base surface of the heat sink just above the channel outlet. Heat flux and Nusselt number near the channel inlet had higher values. These parameters varied around the periphery and zero at corners.

Chander Shekhar Sharma [8] presented a detailed thermo-hydrodynamic analysis of a hot water cooled manifold microchannel heat sink for electronic chip cooling. The hot water cooling enabled efficient recovery of heat dissipated by the even hotter chip by using hot water recovered from a secondary application. This analysis showed that entropy generation due to heat transfer would dominate the net entropy generation in the heat sink for both conditions. Although entropy generation due to viscous dissipation increased significantly with increased Reynolds number, it still would contribute less than a third to the total entropy generated at high Reynolds numbers. Component of entropy generation was reduced significantly due to use of hot water, second law efficiency was increased.

X.L et al [9] performed numerical study of laminar heat transfer and pressure drop characteristics in a water-cooled minichannel heat sink. The results indicated that heat transfer performance would increase by a narrow, deep and thin channel with a relatively high but acceptable pressure drop. A configuration of heat sink was approximately optimized which could cool a chip having 256 W/cm^2 heat flux of at 0.205 W pumping power.

J. Koo, C. Kleinstreuer [10] simulated and analyzed liquid nanofluid flow in microchannels by considering two types of nanofluids, i.e., CuO nanospheres at lower volume concentrations in water or ethylene glycol. Based on this study, high-Prandtl number carrier fluids was recommended for microchannel heat sink. High volume concentrations nanoparticles of about 4% with increased thermal conductivities and dielectric constants very close to that of the carrier fluid, microchannels with high aspect ratios were also recommended for the microchannel heat sink.

Manay et al. [11] numerically investigated the pressure drop and laminar convection heat transfer characteristics of nanofluids in microchannel heat sink with square duct. The water based nanofluids created with Al_2O_3 and CuO particles in four different volume fractions of 0%, 0.5%, 1%, 1.5% and 2% are used to analyze their effects on heat transfer. The results were determined in terms of pressure drop, Nusselt number and heat transfer coefficient. Analysis showed that the nanofluids would enhance heat transfer while the Reynolds number and the volume fractions were increasing. The best overall enhancement was obtained at φ=2% and Re=100 for CuO-water nanofluid.

Seok Pil Jang, Stephen U.S. Choi [12] investigated the cooling performance of a microchannel heat sink with nanoparticles – fluid suspensions (''nanofluids'') numerically. A theoretical model of thermal conductivity of nanofluids was that account for the fundamental role of Brownian motion, they investigated the temperature contours and thermal resistance of a microchannel heat sink with nanofluids such as 6 nm copper- in-water and 2 nm diamond-in-water. The results showed that the cooling performance of a microchannel heat sink with water-based nanofluids containing 1%, vol. 2 nm diamond at the fixed 2.25W pumping power, was enhanced by about 10% in comparison of microchannel heat sink with water as coolant. Thermal resistance and the temperature difference both between the heated wall and the coolant were reduced on using nanofluid in microchannel heat sink.

H.A. Mohammed et al. [13] investigated the effect of using nanofluids on heat transfer and fluid flow characteristics in rectangular shaped microchannel heat sink (MCHS) numerically for Reynolds number range of 100–1000. In this study, the MCHS performance was examined using alumina–water (Al_2O_3-H_2O) nanofluid as coolant with volume fraction ranged from 1% to 5%. The results revealed that when the volume fraction of nanoparticles was increased under the highest heat flux, the heat transfer coefficient and wall shear stress both would increase while the thermal resistance of the MCHS would decrease. Only a slight increase in the pressure drop across the MCHS was found compared with the pure water-cooled MCHS.

Bhanu Pratap Singh et al. [14] did CFD modelling to know the effect of number of channel with symmetrical geometry. Two models having rectangular and trapezoidal cross section were chosen. It was found that performance of symmetrical channel had a good agreement when compared with the single channel microchannel heat sink.

Harry Garg et al. [15] used COMSOL multi physics module in integration with Fluid Flow & Conjugate Heat transfer Module. Investigations had been conducted to better understand & establish the fluid flow & heat transfer characteristics in different micro channel heat sinks.
Different parameters like heat transfer rate, heat transfer coefficient and temperature at outlet were studied with different flow rates and an optimum best range of flow rate for all the fluids separately by considering maximum heat transfer rate, minimum base temperature and less pumping power had been considered.

Navin Raja Kuppusamy et al. [16] studied the thermal and flow fields in a trapezoidal grooved microchannel heat sink (TGMCHS) using nanofluids. The governing and energy equations were solved using the finite volume method. The effect of the geometrical parameters on the thermal performance of TGMCHS was examined. At different Reynolds numbers, the effects of different nanoparticle types, volume fraction, particle diameter and base fluid were also studied. It was found that the increment of the maximum width 'a' and reduction of the minimum width 'b' of the trapezoidal groove gave the maximum thermal performance. It was shown that Al_2O_3-H_2O had the highest thermal performance with 4% volume fraction and 25 nm diameter.

Mostafa Mirzaei and Maziar Dehghan [17] investigated laminar flow and heat transfer of Al_2O_3-water nanofluid under constant heat flux numerically. Heat transfer was enhanced. An increase in friction factor had been also obtained by the use of nanofluid. Heat transfer enhancement was more evident by the use of variable properties. Effects of temperature variation on nanofluid heat transfer were greater than the pure water.

Ping Li et al. [18] investigated flow and heat transfer characteristics of Al_2O_3–water nanofluids in the microchannel with dimple and protrusion surface are investigated for the first time in order to improve the efficiency of compact heat exchanger and microchannel heat sinks. They studied the influence of nanoparticle volume fraction ϕ (0–3%), and geometrical parameters of microchannel on flow and heat transfer characteristics in detail. Three similar microchannel structures were adopted in their work, which were smooth, dimpled and dimpled plus protrusioned microchannel. The results showed that the relative Fanning friction factor f/f_0, Nusselt number Nu/Nu_0 and thermal performance TP would increase with the increase of U for nanofluids.

H.A Mohammed et al. [19] performed numerical investigations to investigate the laminar flow and heat transfer characteristics of trapezoidal microchannel heat sink using different types of base nanofluids and various substrate materials on MCHS performance. Four types of base fluids including water, ethylene glycol (EG), oil, and glycerin with 2% volume fraction of diamond nanoparticle were considered in this study. It was found that the best uniformities in heat transfer coefficient and temperature among the four mixture flows could be obtained using nanofluid which had glycerin as base fluid.

Roy W. Knight et al.[20] presented the equations governing the fluid dynamics and combined conduction/convection heat transfer in a heat sink in dimensionless form for both laminar and turbulent flow. This study was useful to determine heat sink's dimensions that displayed the lowest thermal resistance between the hottest portion of the heat sink and the coming fluid. Results from the this method were applied to heat sinks to study effects of their restrictions regarding the nature of the flow (laminar or turbulent), the ratio of fin thickness to channel width, or the aspect ratio of the fluid channel. Results indicated that when the pressure drop through the channels was small, laminar solutions yield lower thermal resistance was obtained by laminar solutions in comparison to turbulent solution, when pressure drop was small through channel.

Mohammad Kalteh et al. [21] examined study the laminar convective heat transfer of an alumina-water nanofluid flow inside a wide rectangular microchannel heat sink numerically and experimentally. A microchannel was made using a silicon wafer with glass layers for experimental study. A two-phase Eulerian-Eulerian method using the finite volume approach was adopted for the numerical study. The two-phase results showed that the velocity and temperature difference between the phases were very small and negligible. The average Nusselt number increased with an increase in Reynolds number and volume concentration as well as with a decrease in the nanoparticle size.

Saeed Zeinali Heris et al. [22] numerically investigated laminar flow-forced convective heat transfer of Al_2O_3-water nanofluid in a triangular duct under constant wall temperature condition. In it the effects of different parameters like diameter of nanoparticles, concentration, and Reynolds number on the enhancement of heat transfer was studied. Numerical results represented an enhancement of heat transfer of fluid associated with changing to the suspension of nanometer sized particles in the triangular duct. The results of that model indicated that the nanofluid Nusselt number would increase with increasing concentration of nanoparticles and decreasing diameter.

Deewakar Sharma [23] analyzed the double layer configuration for rectangular ducts. In this study performance of trapezoidal shape double layer microchannel heat sink was investigated and compared to rectangular double layer heat sink of same flow area. The performance was evaluated on the basis of maximum temperature attained at the heated surface as well as minimum temperature variations. Finally the best performing configuration was compared with double layer rectangular heat sink. Analysis proved that among various trapezoidal configurations, the one with larger side face to face was most suitable. Comparative study with rectangular system showed that performance of trapezoidal double layer heat sink was superior in both aspects.

Yue-Tzu Yang et al. [24] performed numerical study of trapezoidal MCHS using CuO-H_2O as coolant. Particle volume fraction and volumetric flow rate were studied. It was shown that thermal resistance would decrease with increasing particle volume fraction and volumetric flow rate.

H.A. Mohammed et al. [25] numerically investigated Al_2O_3-H_2O, Ag-H_2O, CuO-H_2O, diamond-water, SiO_2-H_2O and TiO_2-H_2O for triangular shaped microchannel heat sink. Heat transfer coefficient was minimum for Al_2O_3-H_2O but it had highest temperature. Heat transfer coefficient was maximum for diamond-water. Highest pressure drop and highest wall shear stress occurred in case of SiO_2- H_2O.

4.3 Other Study

Deewakar Sharma et al. [26] analyzed the performance of trapezoidal and rectangular microchannels and compared for two different coolants, liquid gallium and water. Performance was compared on the basis of maximum temperature at hot surface. For water as coolant the performance of rectangular type was found to be superior in terms of flow rate and pump power. For liquid gallium as a coolant, microchannels having A-type cross-section was found to be performing better followed by V-type then rectangular type at same flow rate. While comparing on the basis of pump power the rectangular type of geometry was more suitable than trapezoidal.

Weilin Qu and Issam Mudawar [27] explored several issues important to the thermal design of single-phase and two-phase micro-channel heat sinks. First half portion of study dealt with single-phase heat transfer in rectangular micro-channels. Experimental results were compared with predictions based on both numerical as well as fin model. The best agreement between predictions and experimental results was accomplished with numerical simulation, a few of the fin models were found to provide fairly accurate predictions.

C.Y. Zhao, T.J. Lu [28] performed analytical and numerical study on the heat transfer characteristics of forced convection across a microchannel heat sink. The porous medium model and the fin approach, these two approaches were used. In the porous medium approach, Darcy equation (modified) was used for the fluid and the two-equation model was used for heat transfer between the solid and fluid phases. Effect of channel aspect ratio and effective thermal conductivity ratio on the overall Nusselt number of the heat sink were determined in this study. Both approaches showed that the overall Nusselt number (Nu) would increase as aspect ratio would increase and decrease with increasing thermal conductivity.

Ahmed Mohammed Adham et al. [29] reviewed the available studies regarding non-circular microchannels heat sinks with emphasis on rectangular microchannels. Different parameters were reviewed in this study. The review showed that studies from 1981 to 1999 were largely performed using experimental or analytical approaches while more recent studies from 2000 to the end of 2012 performed numerically. It was also found laminar was the prevailing flow condition as out of the 69 articles reviewed, 54 employed laminar flows. It was found that the use of liquid coolants was preferable over gaseous coolants. Recent researches in nanofluids are providing alternative coolants.

Shakuntala Ojha [30] investigated rectangular microchannel heat sink using water as a coolant. Heat transfer coefficient, Nusselt number were studied for different pressure drop and heat flux in this study. Semi implicit method was used for fully developed laminar flow in microchannel heat sink. Maximum temperature was found at bottom of channel outlet. Maximum heat transfer coefficient and Nusselt number was found maximum at inlet region.

Through this extensive literature survey it is found that CFD analysis of fluid flow and heat transfer through microchannel heat sink having trapezoidal microchannels using nanofluid TiO_2-H_2O has not been performed yet. It is also clear from literature review that water was used previously in microchannels as a coolant. But due to heat dissipation limitations of water, other coolants are introduced. In recent times, nanofluids are developed. It is important to check performance of microchannel heat sink using nanofluid as coolant.

So in this thesis, cfd analysis of fluid flow and heat transfer through microchannel heat sink having trapezoidal channels is investigated by using TiO_2 nanofluid which has not been done yet. ANSYS ICEM CFD 14.0 and FLUENT 14.0 softwares are used. Objective is to perform following –

- Simulating single MCHS for various pressure drop.
- Determining pressure, velocity and temperature profiles for constant heat flux on the bottom of the sink.
- Determining temperature distribution along the channel.
- To determine heat transfer coefficient and Nusselt number for various pressure drop at constant heat flux and different heat flux.
- To make a comparison between performance of microchannel heat sink using nanofluid and water.

Results will be validated with the previous results available from literature review.

6.1 Introduction

An isometric view of microchannel heat sink having trapezoidal channel is shown in the figure 6.1. Various dimensions of this microchannel heat sink are shown in the figure 6.2. A constant heat flux is applied at the bottom of surface. Here heat transfer is a conjugate problem which combines conduction and convection. Heat is transferred by conduction through solid and dissipated away by cooling fluid by convection.

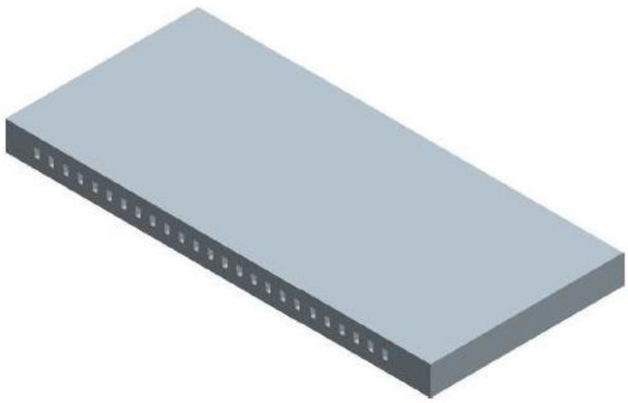


Fig 6.1 Isometric View of Complete Microchannel Heat Sink Created In PRO-E Wildfire 4.0

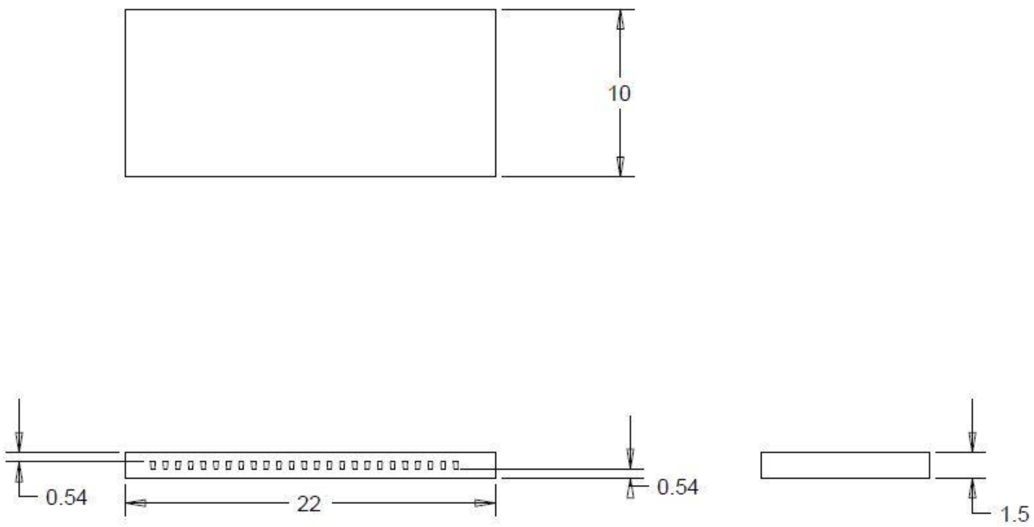


Fig 6.2 Various Dimensions Of MCHS (in mm)

Single microchannel heat sink is shown in the figure 5.3 below. Various dimensions are listed in the table 5.1. A constant heat flux is supplied at the bottom surface of heat sink at y = 0. Top surface of sink which is at y = h is well insulated. Hydraulic diameter for trapezoidal channel is calculated using formula given in equation 5.12.

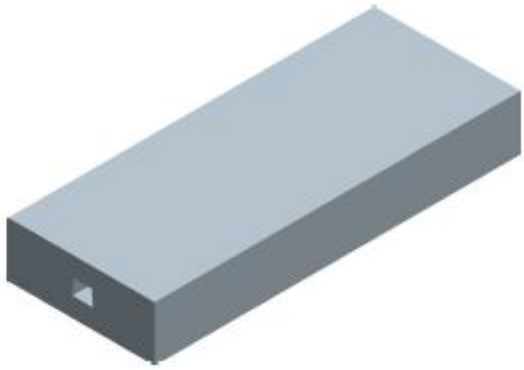


Fig 6.3 Isometric View of Single Microchannel in Heat Sink Created In PRO-E Wildfire 4.0

Table 6.1 Dimensions of Trapezoidal Channel

a	b	c	h	L_c	S	D_h
280	225	431	430	10000	500	318

Where all dimensions are in (μm)

In table,

a = channel top width for trapezoidal cross-section MCHS

b = channel bottom width for trapezoidal cross-section MCHS

c = channel hypotenuse for trapezoidal cross-section MCHS

h = channel height for trapezoidal cross-section MCHS

L_c = channel length

S = distance between two micro channels

D_h = hydraulic diameter

t = 1.8 mm (distance from left face/right face of sink to left/right face of channel)

S_b = 0.54 mm (distance from base of sink to bottom of channel)

W = width of the sink = 4 mm

H = height of sink = 1.5 mm

6.2 Mesh Generated In ICEM CFD 14.0

ICEM CFD 14.0 is a meshing software which is used to create mesh for single microchannel heat sink. After creating geometry in PRO E Wildfire 4.0, IGES file is imported in ICEM CFD 14.0. Geometry imported in ICEM CFD 14.0 is shown in figure given below.

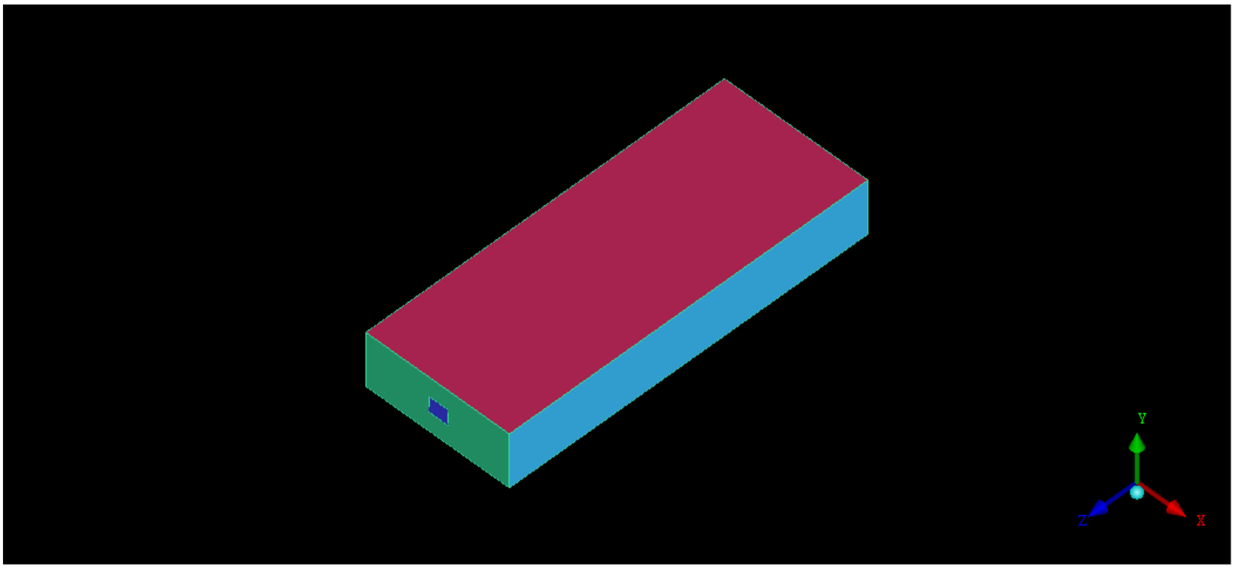


Fig 6.4 Geometry Imported In ICEM CFD 14.0

After importing geometry file in IGES format, a volume mesh is created for it. Trapezoidal microchannel is made fluid zone and rest is made solid. Different faces are assigned names as inlet, outlet, channel, heat flux wall, insulated top, insulated right, insulated left. The step wise procedure is given in appendix I.

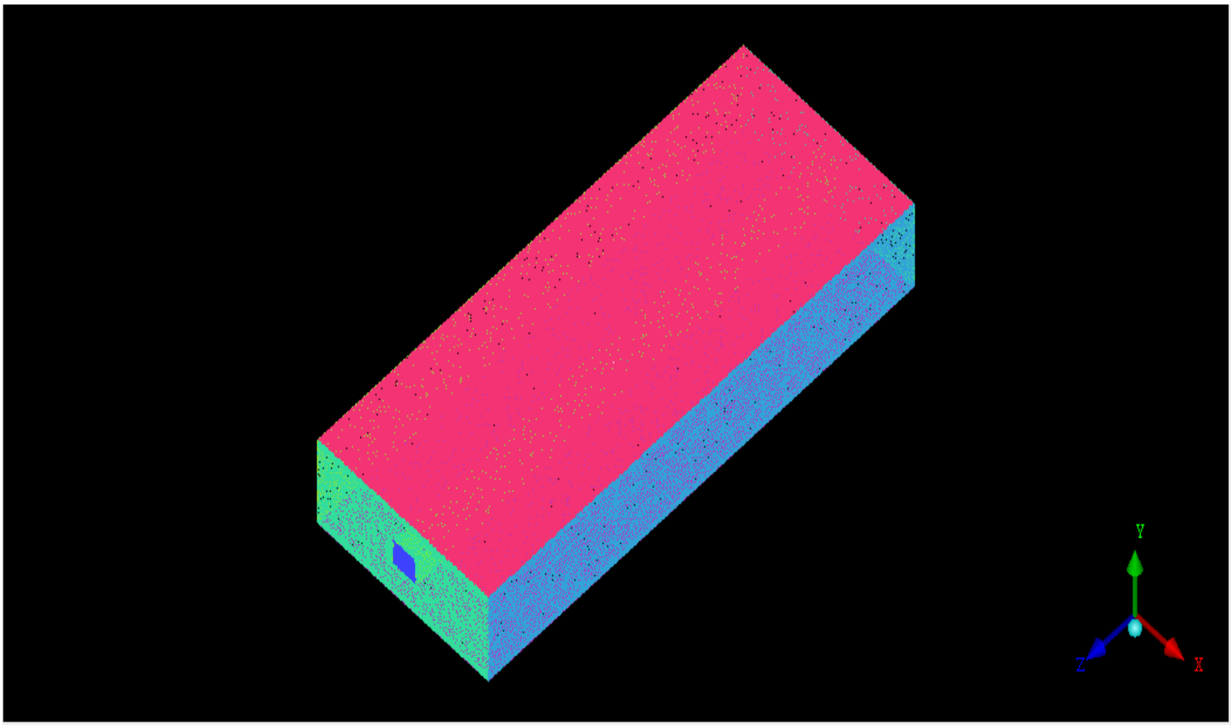


Fig 6.5 Mesh generated in ICEM CFD 14.0

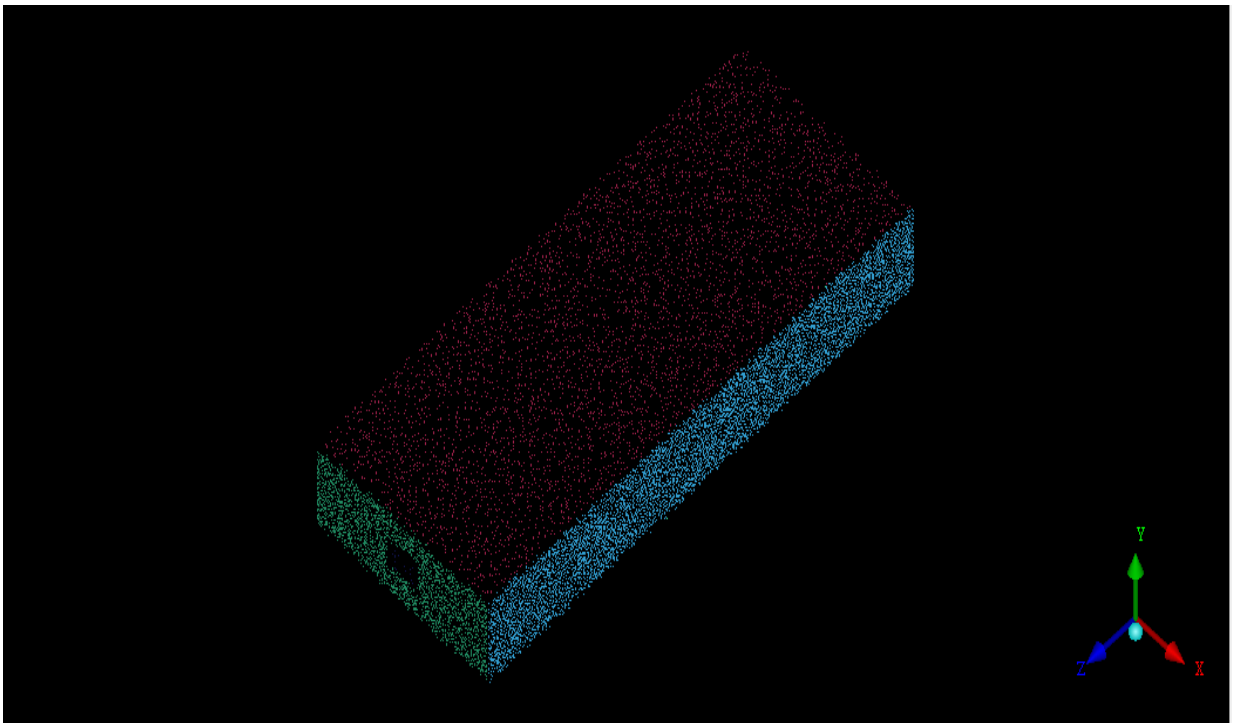


Fig 6.6 Solid view of Mesh

Mesh generated contains 1856699 total elements and 326858 total nodes. This mesh is exported to FLUENT 14.0.

6.3 Boundary Conditions and Governing Equations

Following assumptions are made for applying Navier Stokes Equation and energy equation to the model –

- Flow is laminar and steady
- Solid and fluid properties are constant
- Heat Flux at the bottom surface is uniform
- Fluid is incompressible
- Radiation heat transfer is negligible

For a flow which is fully developed continuity, momentum and energy equations are given as below,

Continuity Equation

$$\frac{\partial \mathrm{u}}{\partial \mathrm{x}}+\frac{\partial \mathrm{v}}{\partial \mathrm{y}}+\frac{\partial \mathrm{w}}{\partial \mathrm{z}}=0 \qquad (6.1)$$

Momentum Equation

X-momentum equation

$$\rho\left(u\frac{\partial u}{\partial x}+v\frac{\partial u}{\partial y}+w\frac{\partial u}{\partial z}\right)=-\frac{\partial p}{\partial x}+\mu\left(\frac{\partial^2 u}{\partial x^2}+\frac{\partial^2 u}{\partial y^2}+\frac{\partial^2 u}{\partial z^2}\right) \qquad (6.2)$$

Y-momentum equation

$$\rho\left(u\frac{\partial v}{\partial x}+v\frac{\partial v}{\partial y}+w\frac{\partial v}{\partial z}\right)=-\frac{\partial p}{\partial y}+\mu\left(\frac{\partial^2 v}{\partial x^2}+\frac{\partial^2 v}{\partial y^2}+\frac{\partial^2 v}{\partial z^2}\right) \qquad (6.3)$$

Z-momentum equation

$$\rho\left(u\frac{\partial w}{\partial x}+v\frac{\partial w}{\partial y}+w\frac{\partial w}{\partial z}\right)=-\frac{\partial p}{\partial z}+\mu\left(\frac{\partial^2 w}{\partial x^2}+\frac{\partial^2 w}{\partial y^2}+\frac{\partial^2 w}{\partial z^2}\right) \quad (6.4)$$

Energy Equation

$$u\frac{\partial T}{\partial x}+v\frac{\partial T}{\partial y}+w\frac{\partial T}{\partial z}=\frac{1}{\alpha}\left(\frac{\partial^2 T}{\partial x^2}+\frac{\partial^2 T}{\partial y^2}+\frac{\partial^2 T}{\partial z^2}\right) \quad (6.5)$$

6.3.1 Boundary Conditions

Following boundary conditions are specified for the single microchannel heat sink.

Pressure at inlet,

for, $t < x < t + b, S_b < y < S_b + h$ and

At $z = 0,\ p_1 = p_{in}$ (6.7)

Velocity in y and z direction is zero, so

$$\text{v} = 0, \text{w} = 0 \quad (6.8)$$

Pressure at outlet,

for, $t < x < t + b, S_b < y < S_b + h$ and

At $z{=}L_c,\ p_1 = p_{out}$ (6.9)

At the bottom surface, constant heat flux is applied,

At $y = 0, 0 < z < L_c$ and $0<x<W$,

$$q'' = -k_s \frac{\partial T}{\partial y} \tag{6.10}$$

Inlet Temperature,

At $z = 0, t < x < t + b$ and $S_b<y<S_b + h$,

$$T = T_{in} \tag{6.11}$$

Hydraulic diameter,

$$D_h = \frac{4A}{P} = \frac{2(a + b)h}{(a + b + 2c)} \tag{6.12}$$

6.4 Properties of Nanofluid and Sink Material

Here silicon is taken as material of the sink. Properties of this are listed in following table-

Table 6.2 Properties of Silicon

Material	Density(ρ_s) (kg/m^3)	Thermal Conductivity k_s (w/m-k)	Specific Heat c_{ps} (J/kg-k)
Silicon	2330	148	712

Properties of nanofluid (TiO_2- H_2O) are calculated using correlations [31]. Properties of nanofluid are listed in the table.

For density of nanofluid,

$$\rho_{nf} = (1-\varphi)\rho_{bf} + \varphi\rho_p \tag{6.13}$$

Where,

$\rho_{nf} =$ density of nanofluid

$\varphi =$ particle volume fraction

$\rho_{bf} =$ density of base fluid

$\rho_p =$ density of particle

For thermal conductivity of nanofluid,

$$k_{nf} = \frac{k_p + 2k_{bf} + 2(k_p - k_{bf})\varphi}{k_p + 2k_{bf} - (k_p - k_{bf})\varphi} k_{bf} \tag{6.14}$$

Where,

$k_{nf} =$ thermal conductivity of nanofluid

$k_p =$ thermal conductivity of particle

$k_{bf} =$ thermal conductivity of basefluid

For viscosity of nanofluid,

$$\mu_{nf} = \mu_{bf}(1 + 2.5\varphi) \tag{6.15}$$

Where,

$\mu_{nf} =$ viscosity of nanofluid

$\mu_{bf} =$ viscosity of basefluid

For heat capacity of nanofluid,

$$(\rho c)_{nf} = (1-\varphi)(\rho c)_{bf} + \varphi(\rho c)_p \tag{6.16}$$

Where,

$(\rho c)_{nf}$ = heat capacity of nanofluid

$(\rho c)_{nf}$ = heat capacity of basefluid

$(\rho c)_p$ = heat capacity of particle

In nanofluid (TiO_2- H_2O), water (H_2O) is base fluid. Properties of the water are listed in following table-

Table 6.3 Properties of Water

Fluid	(ρ_f) Kg/m^3	(k_f) W/m-K	(μ_f) kg/m-s	(c_p) J/kg-K
Water	998.2	0.6	0.001003	4182

After applying above values of base fluid's properties in formulae following properties for nanofluid (TiO_2-H_2O) are obtained-

Table 6.4 Properties of TiO_2-H_2O

Fluid	(ρ_{nf}) Kg/m^3	(k_{nf}) W/m-K	(μ_{nf}) kg/m-s	(c_p) J/kg-K
TiO_2-H_2O	1063.236	0.64364	0.00105315	3902.53

6.5 FLUENT as Solver

Fluent will be used as solver for this research. Three popular solvers are available in ANSYS FLUENT. These are given below-

- Pressure Based Segregated Solver (PBSS)
- Pressure Based Coupled Solver (PBCS)
- Density Based Coupled Solver (DBCS)

Among above three, first one PBSS will be used for following reasons –

- It is suitable for incompressible fluid as assumption is made.
- It solves three momentum equations, continuity and energy equations. It update velocity also.
- Pressure based segregated solver is applicable to wide variety of problems.
- It requires less memory or lower RAM requirements.
- This solver is also quick than other solvers.

As pressure based segregated solver is chosen for pressure velocity coupling, default algorithm in FLUENT, semi implicit method for pressure linked equation (SIMPLE) will be used. It is most robust algorithm among others available in FLUENT like SIMPLEC, PISO and coupled.

7.1 Validation of Study

Present study is validated by convergence graph and grid independence test. Previous numerical study is also used to validate the results of current study.

7.1.1 Convergence Graph

Fig.7.1 shows convergence graph. On solving problem in FLUENT, iterations are converged after 45 iterations.

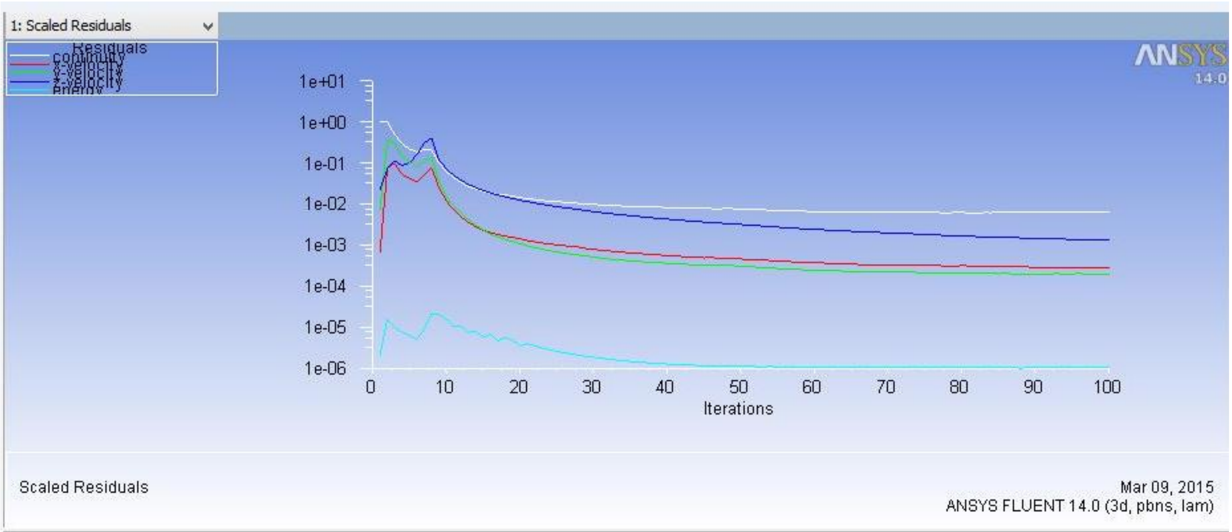


Fig. 7.1 Convergence Graph

7.1.2 Grid Independence Test

Grid independence test is done to validate the results. Three mesh are generated to ensure the accuracy of final results. Results are evaluated by outlet temperature. These three mesh contains 1.7 million, 0.7 million and 0.4 million cells respectively. These are summarized in table. Variation in outlet temperature is very less for these coarse, fine and very fine mesh created for solving the problem in FLUENT. Difference between temperature for fine and very fine mesh is very less. So mesh containing 1.7 million cells is most suitable for performing simulation of trapezoidal single channel microchannel heat sink using solver FLUENT.

Table 7.1 Grid Independence Test

Mesh	Number Of Cells	Outlet Temperature
Mesh 1	1721028	378
Mesh 2	701391	377.8
Mesh 3	445884	377.6

7.1.3 Validation of Results with Numerical Study

Results of current study are validated by a previous numerical study in terms of heat transfer coefficient. This previous study was done by J. Li et al. [6] who performed numerical simulation of rectangular microchannel heat sink using water as coolant in channel. Results are matching with previous study as there is increase in heat transfer coefficient due to use of nanofluid. Fig.7.2 shows the variation of heat transfer coefficient in present study and previous numerical study by J. Li et al.

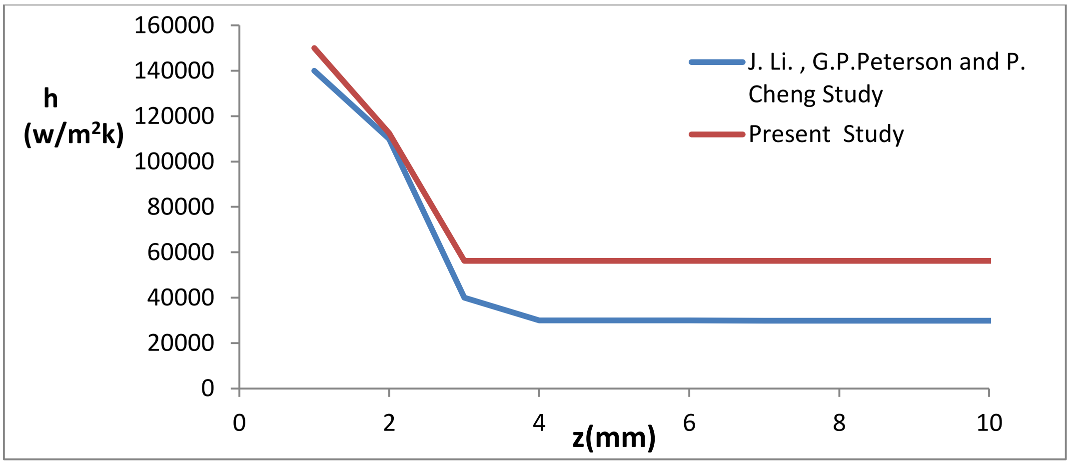


Fig. 7.2 Validation of results by numerical study in terms of heat transfer coefficient

7.2 Temperature Contours

Fluid entered at 293K and its temperature was increased along the flow direction. It is shown in Fig. 7.3. Temperature was increased to 375K.Temperature contour at inlet and outlet is shown in Fig. 7.4. And Fig. 7.5.

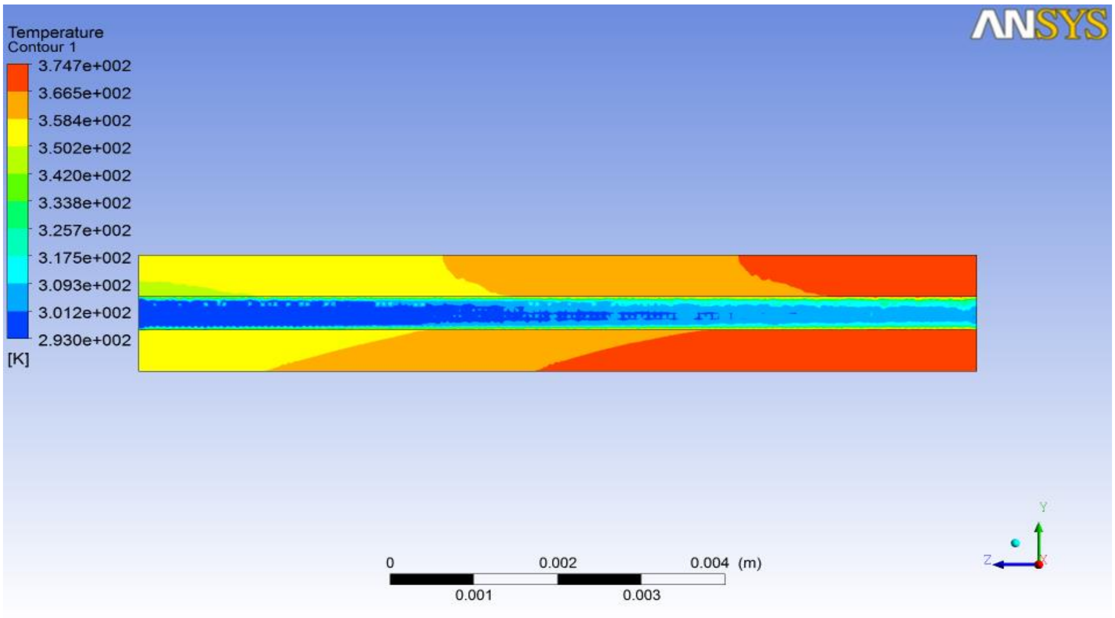


Fig. 7.3. Temperature distribution along the channel

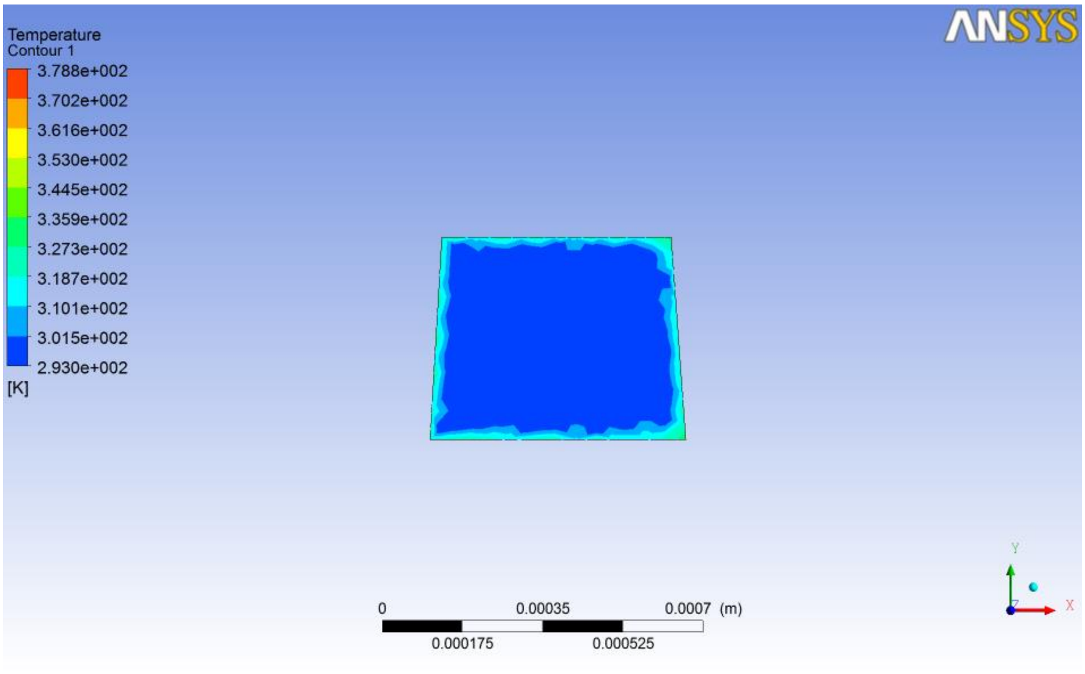


Fig. 7.4. Temperature distribution at inlet

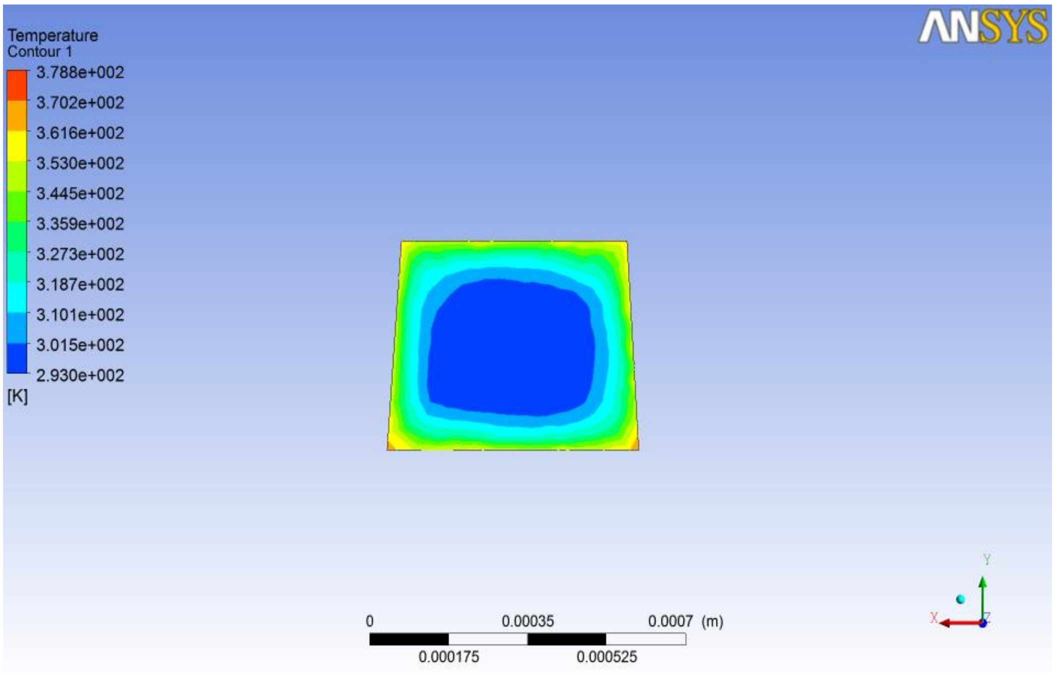


Fig. 7.5. Temperature distribution at outlet under pressure 50000 Pa and 900000 W/m^2 heat flux

Following figures (from Fig. 7.6. to show the temperature contours at inlet and outlet for different pressure drop and heat flux.

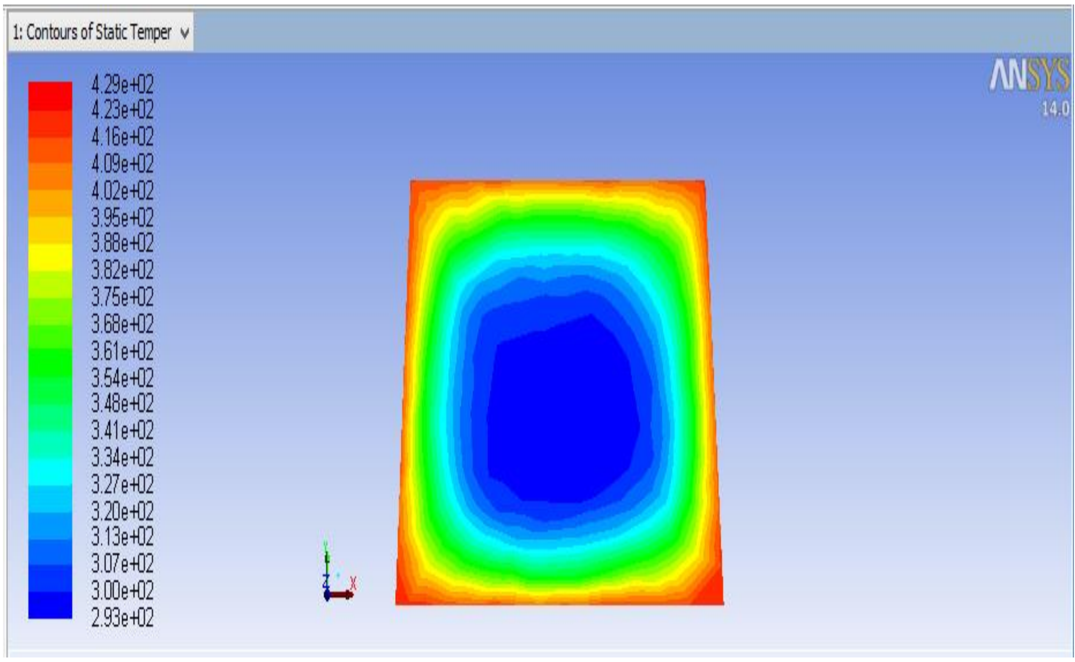


Fig. 7.6. Temperature distribution at outlet under pressure 10000 Pa and 900000 W/m^2 heat flux

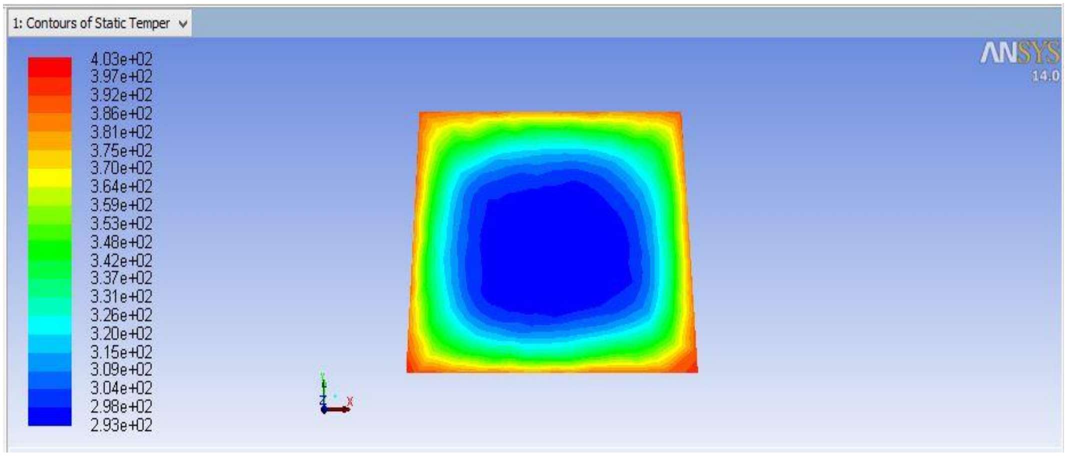


Fig. 7.7. Temperature distribution at outlet under pressure 20000 Pa and 900000 W/m^2 heat flux

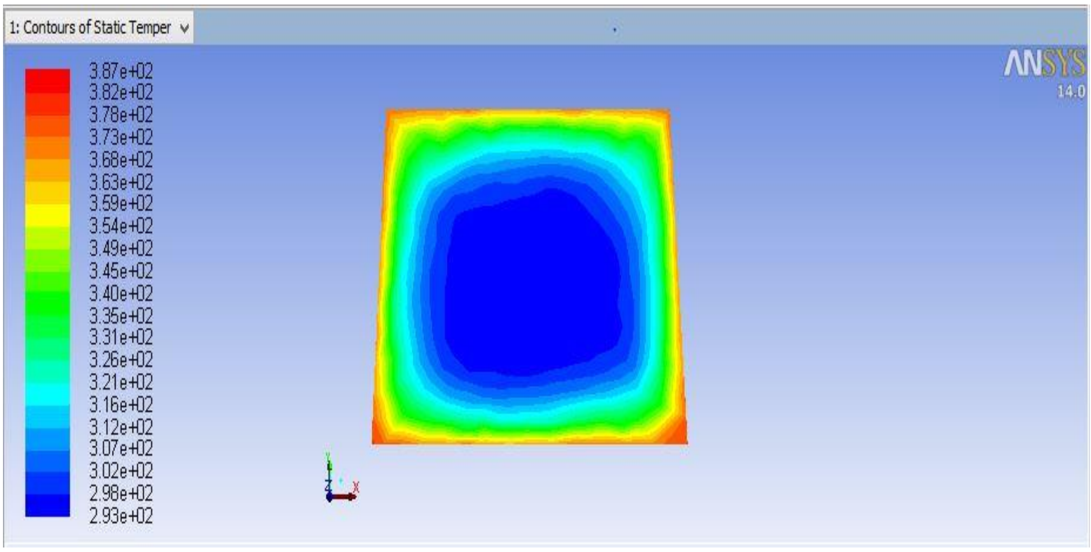


Fig. 7.8. Temperature distribution at outlet under pressure 35000 Pa and 900000 W/m^2 heat flux

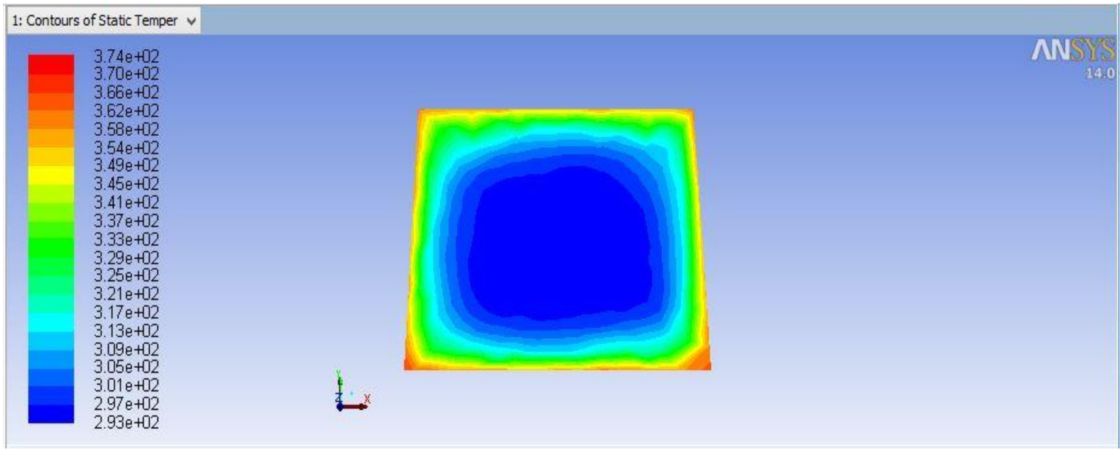


Fig. 7.9. Temperature distribution at outlet under pressure 65000 Pa and 900000 W/m^2 heat flux

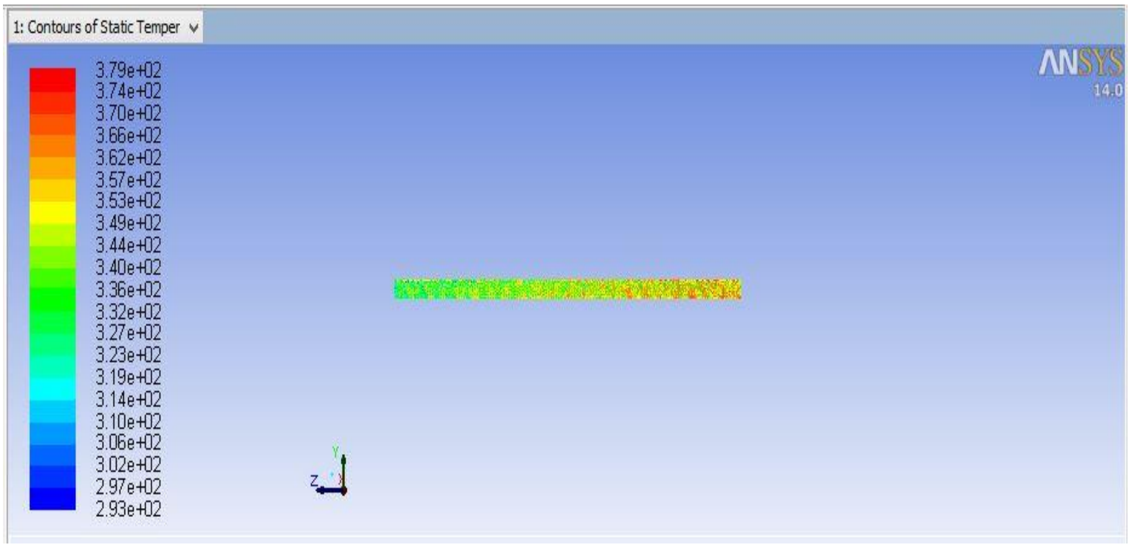


Fig. 7.10. Temperature of fluid inside the channel under pressure 50000 Pa and 900000 W/m^2 heat flux

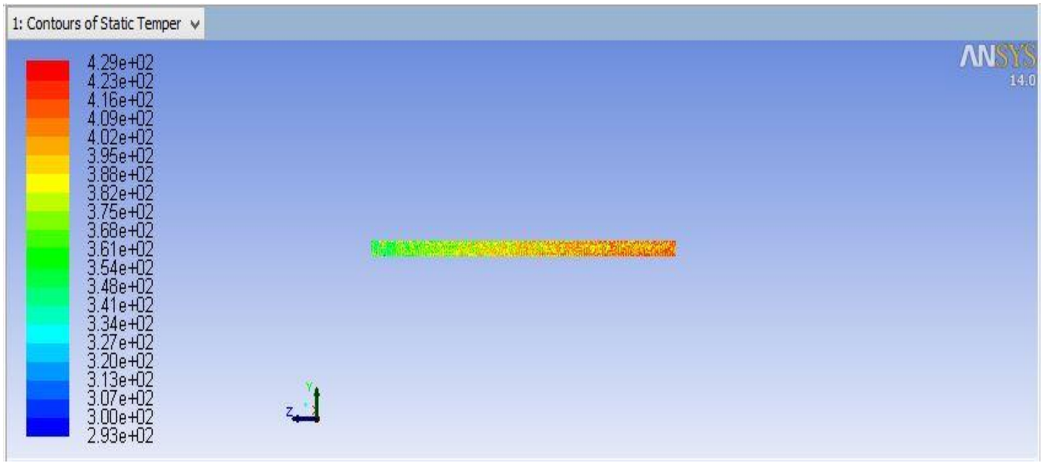


Fig. 7.11. Temperature of fluid inside the channel under pressure 10000 Pa and 900000 W/m^2 heat flux

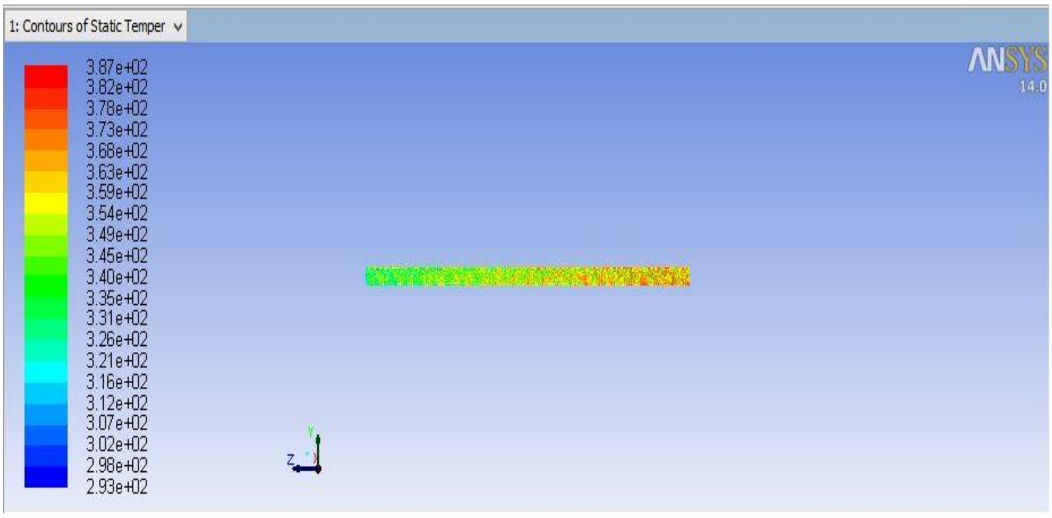


Fig. 7.12. Temperature of fluid inside the channel under pressure 35000 Pa and 900000 W/m^2 heat flux

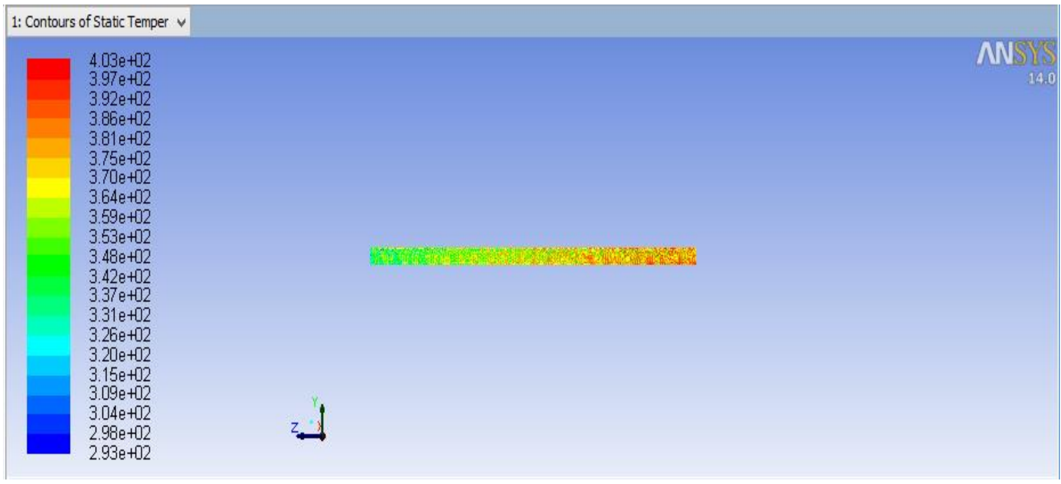


Fig. 7.13. Temperature of fluid inside the channel under pressure 20000 Pa and 900000 W/m^2 heat flux

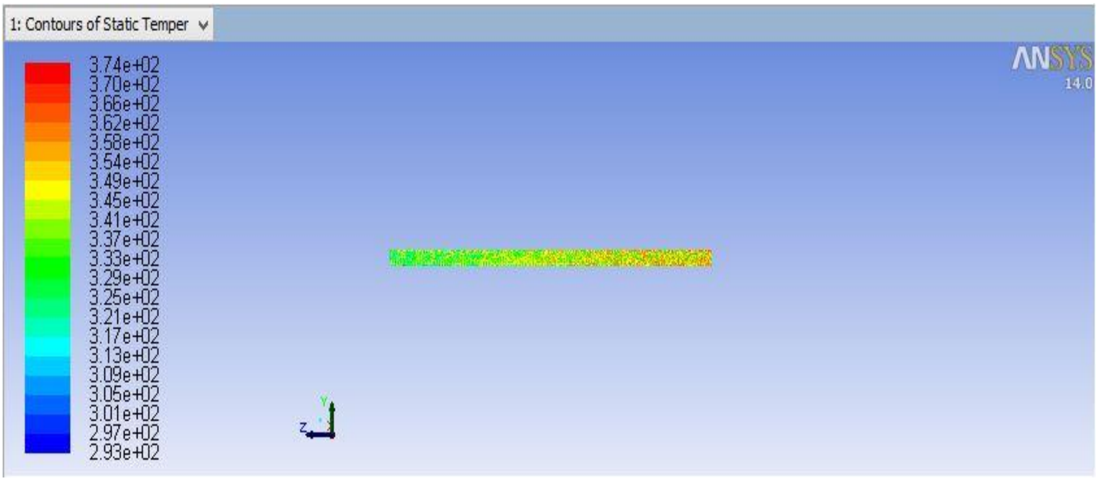


Fig. 7.14. Temperature of fluid inside the channel under pressure 65000 Pa and 500000 W/m^2 heat flux

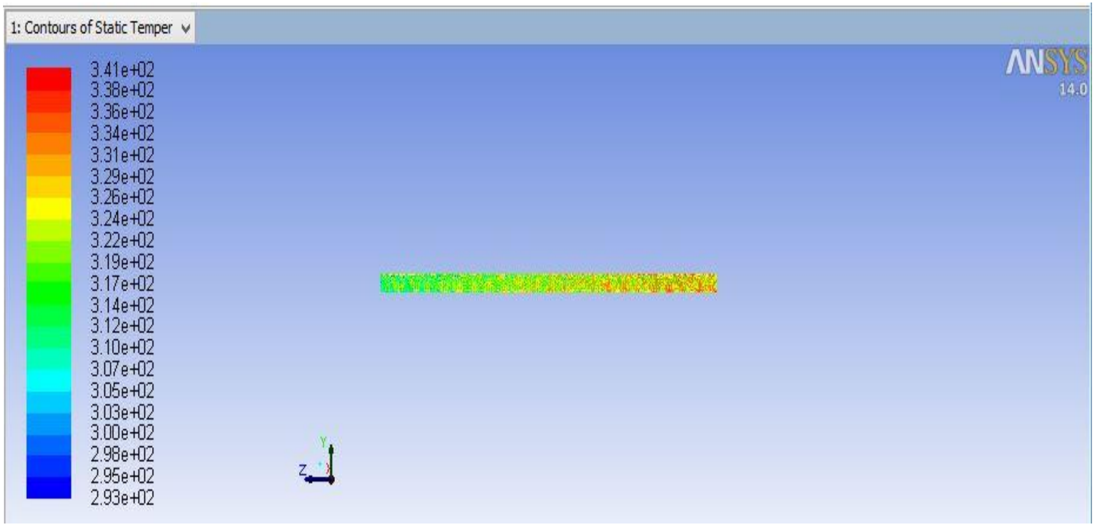


Fig. 7.15. Temperature of fluid inside the channel under pressure 50000 Pa and 500000 W/m^2 heat flux

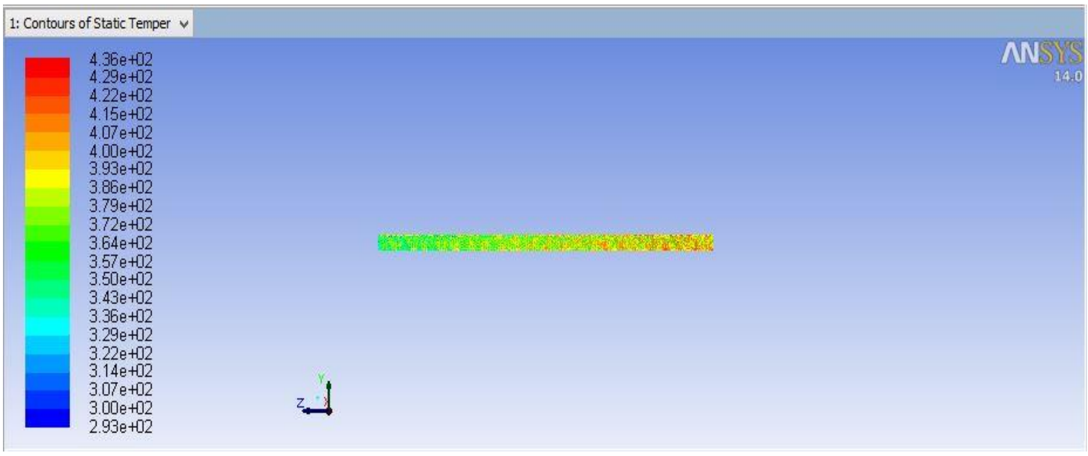
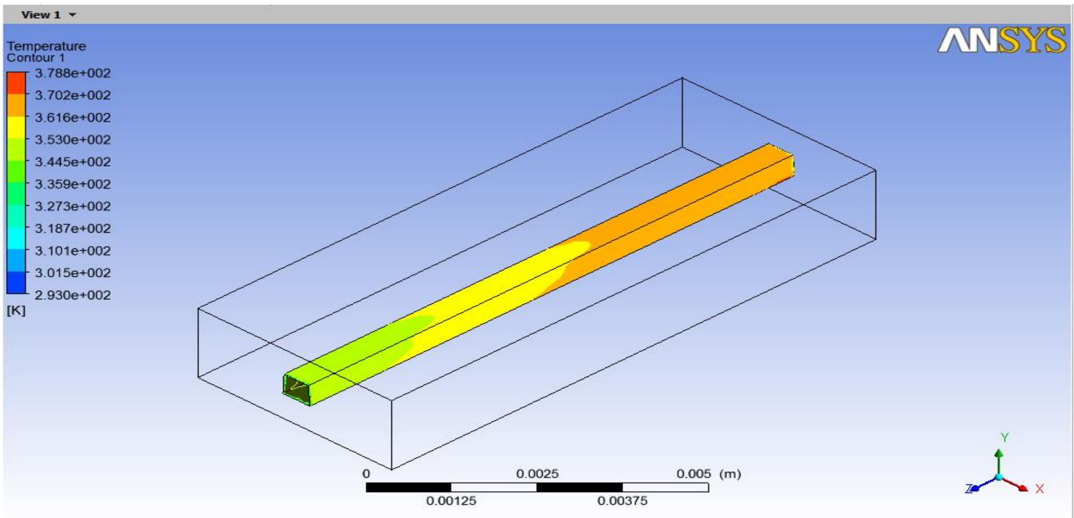


Fig. 7.16. Temperature of fluid inside the channel under pressure 50000 Pa and 1500000 W/m^2 heat flux

Following figures show temperature contours of channel walls for different pressure drop and heat flux.



.

Fig. 7.17. Temperature of channel in pressure 50000 Pa and 900000 W/m^2 heat flux

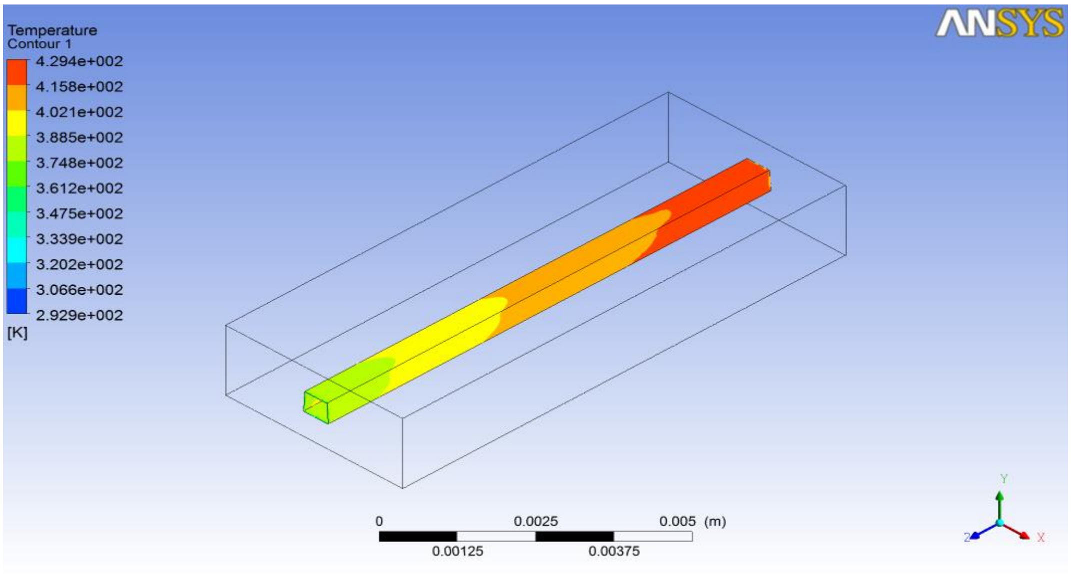


Fig. 7.18. Temperature of channel in pressure 10000 Pa and 900000 W/m^2 heat flux

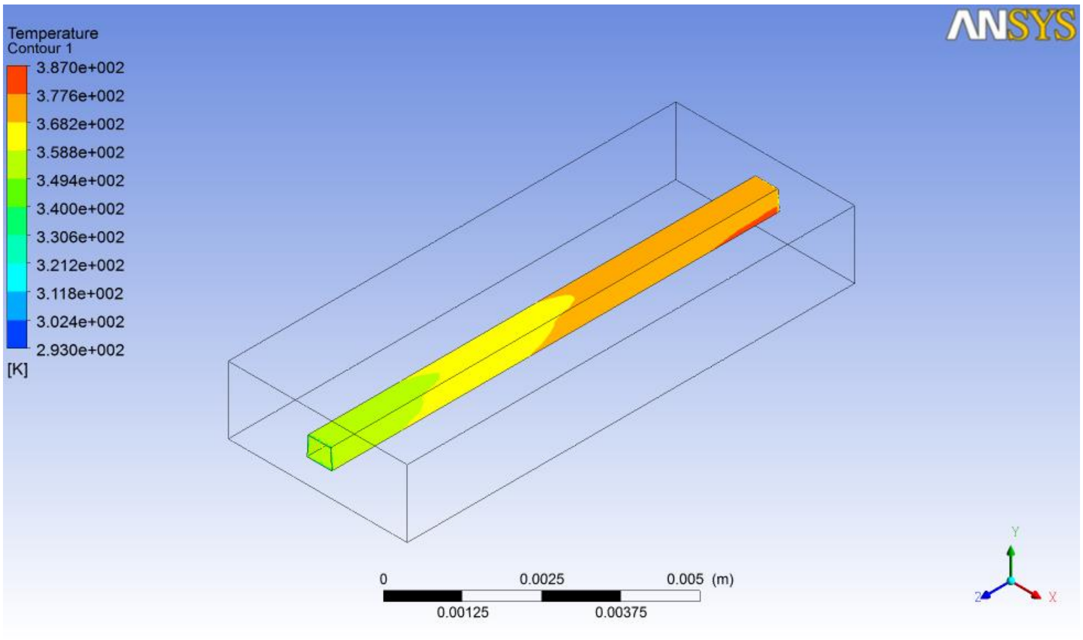


Fig. 7.19. Temperature of channel in pressure 35000 Pa and 900000 W/m^2 heat flux

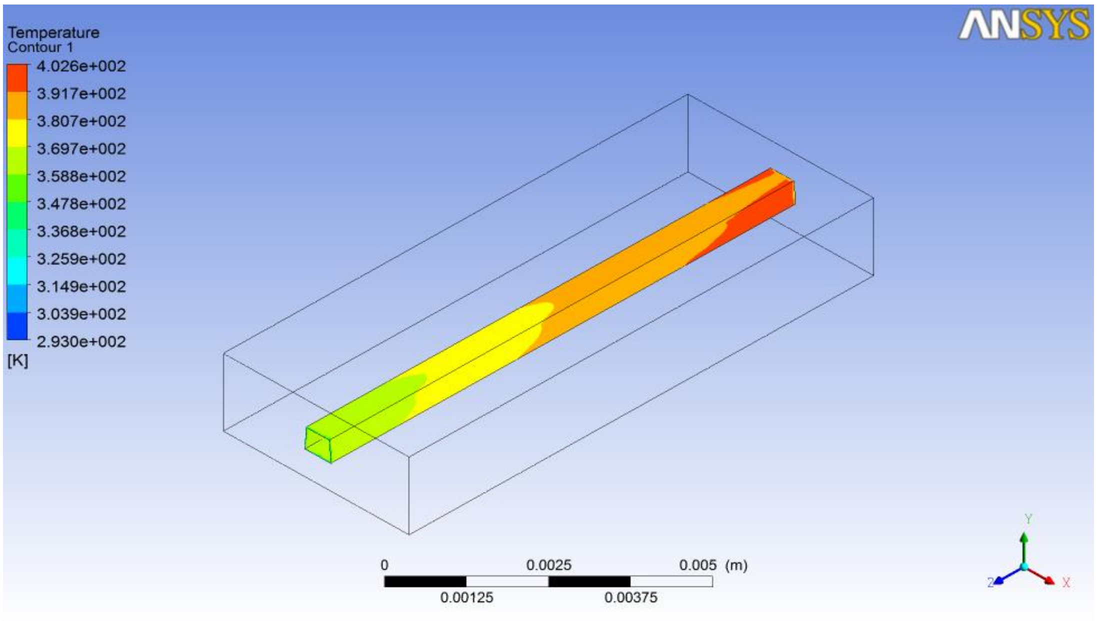


Fig. 7.20. Temperature of channel in pressure 20000 Pa and 900000 W/m^2 heat flux

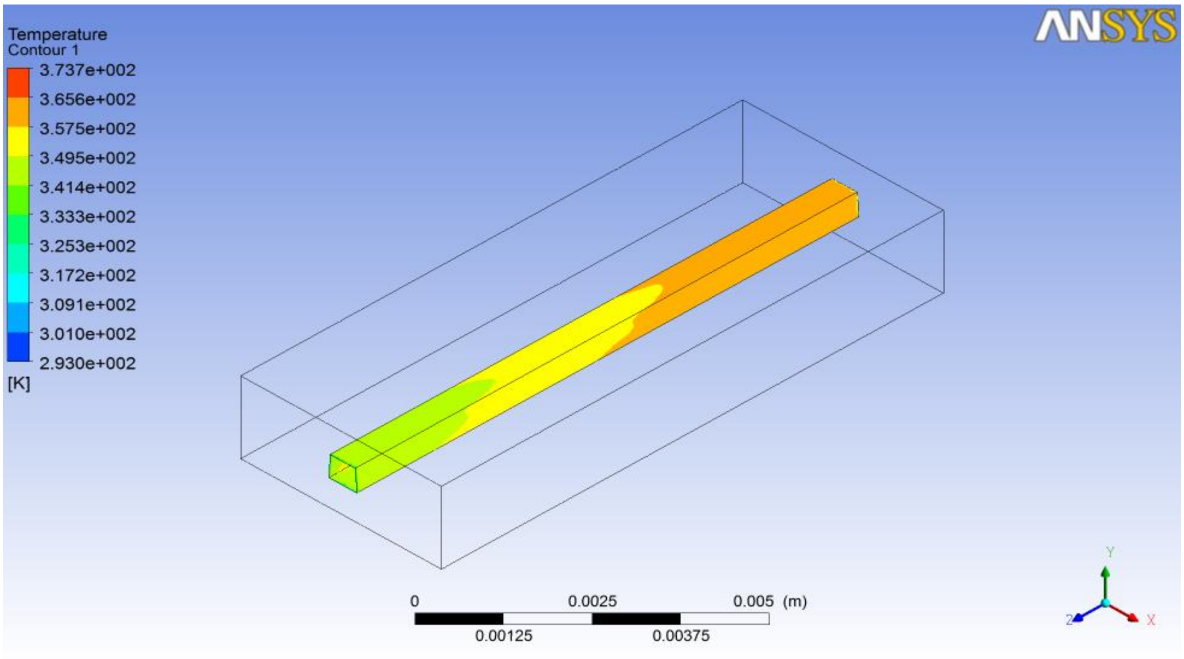


Fig. 7.21. Temperature of channel in pressure 65000 Pa and 900000 W/m^2 heat flux

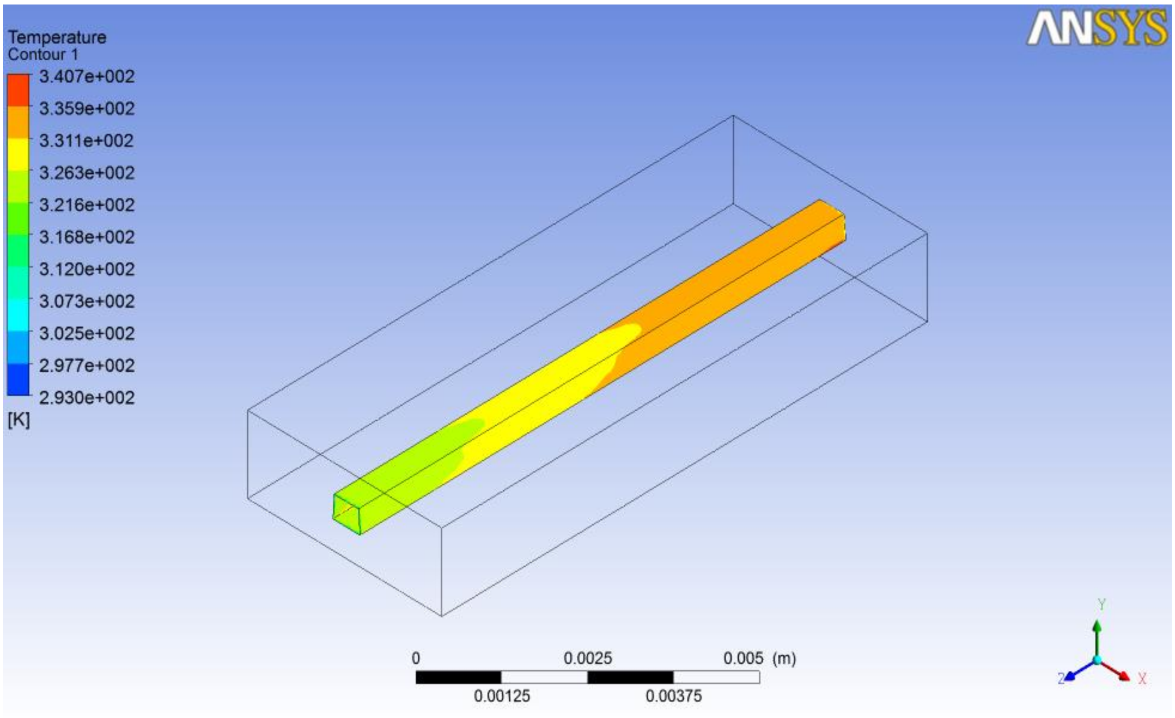


Fig. 7.22. Temperature of channel in pressure 50000 Pa and 500000 W/m^2 heat flux

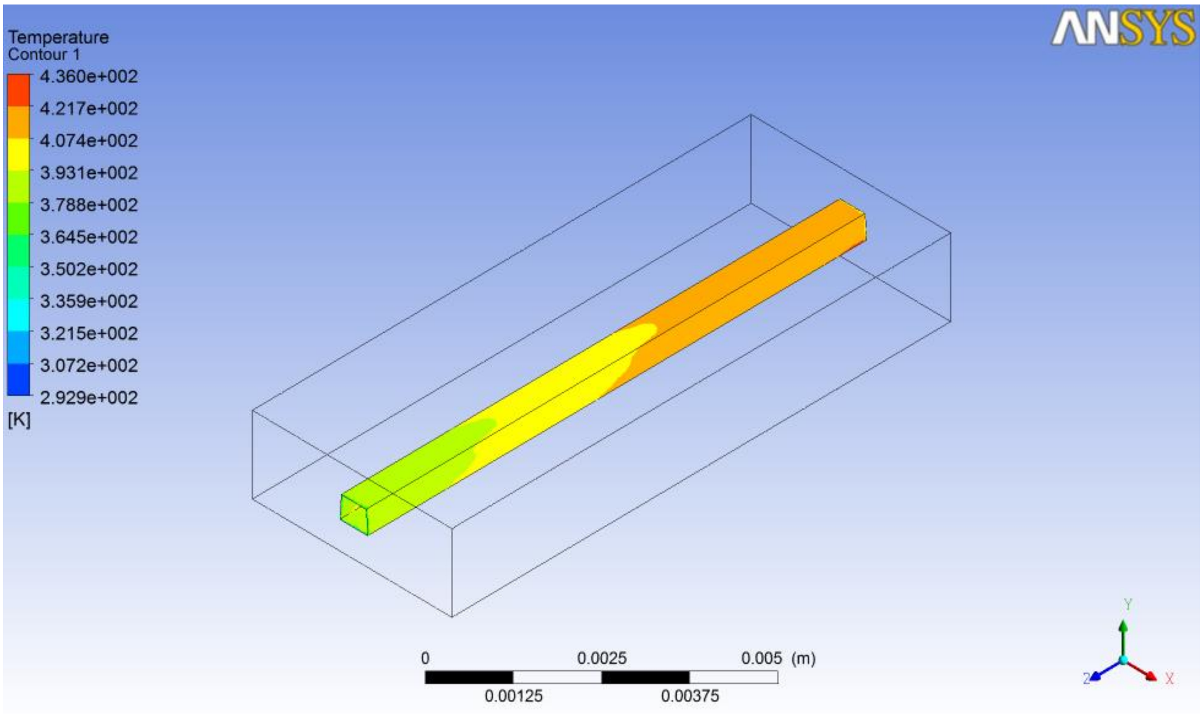


Fig. 7.23. Temperature of channel in pressure 50000 Pa and 1500000 W/m^2 heat flux

7.3 Pressure Contours

At 293K and 50000 Pa, fluid entered in channel. Fluid came out at outlet at atmospheric pressure. Pressure contour inside the in y-z plane is shown in Fig. 7.24. Heat flux of 900000 W/m^2 is applied at bottom surface of heat sink.

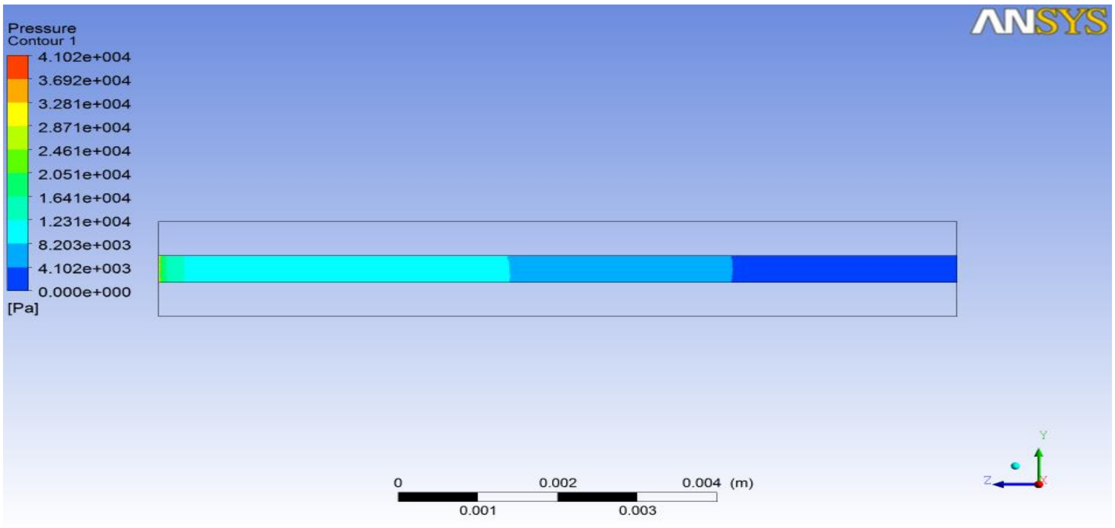


Fig. 7.24. Pressure distribution inside the channel

Table 7.2 Outlet temperature at different pressure drop

Pressure drop (KPa)	T_{in} (K)	T_{out} (K)
10	293	429
20	293	402
35	293	387
50	293	378
65	293	374

7.4 Velocity Vector

Fig. 7.25 shows the velocity vectors at inlet and outlet of microchannel heat sink. Maximum velocity is at inlet.

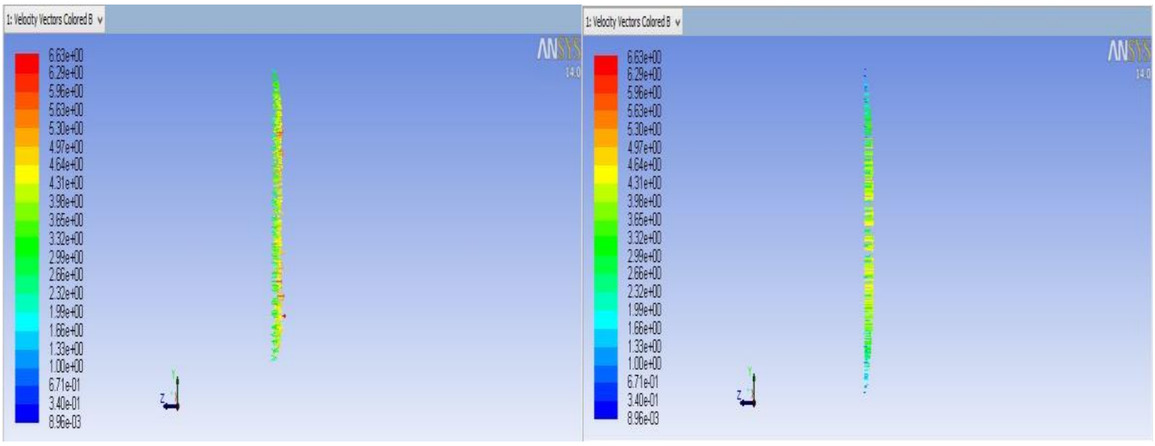


Fig. 7.25 Velocity Vectors at inlet and outlet

7.5 Heat transfer coefficient

Heat transfer coefficient is calculated. Variation of heat transfer coefficient with flow direction is shown in Fig. 7.26. From figure it is clear that value of heat transfer coefficient is very high at inlet. Because of increasing boundary layer heat transfer coefficient decreases in direction of flow. It is calculated using following expression:

$$q'' = h\Delta T$$

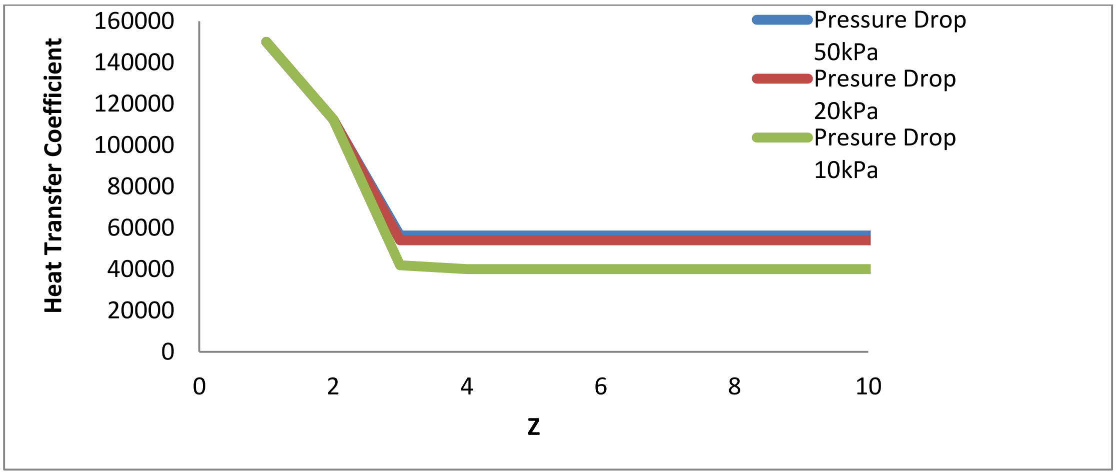


Fig.7.26. Variation of heat transfer coefficient in direction of flow

Fig. 7.27 shows the variation of heat transfer coefficient along the direction of flow for water and nanofluid as a coolant in trapezoidal microchannel heat sink. Heat transfer coefficient is high for nanofluid as coolant in comparison of water as a coolant in trapezoidal microchannel heat sink.

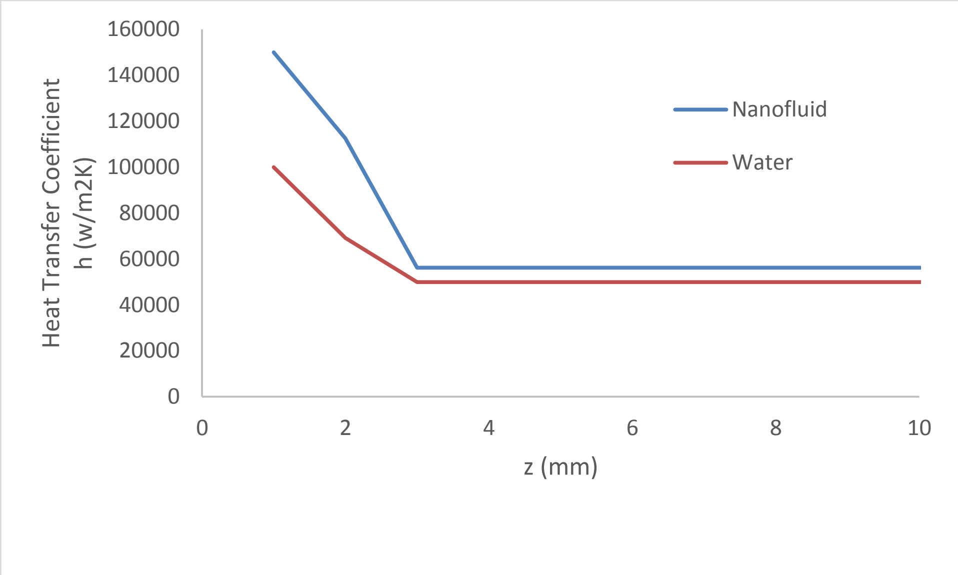


Fig.7.27. Variation of heat transfer coefficient for nanofluid and water

7.6 Nusselt Number

Variation of Nusselt number along the flow direction for different pressure drop is shown in Fig. 7.28. Nusselt number is defined as:

$$Nu = \frac{hL}{k}$$

Where, L= hydraulic diameter of trapezoidal channel and k is thermal conductivity of nanofluid.

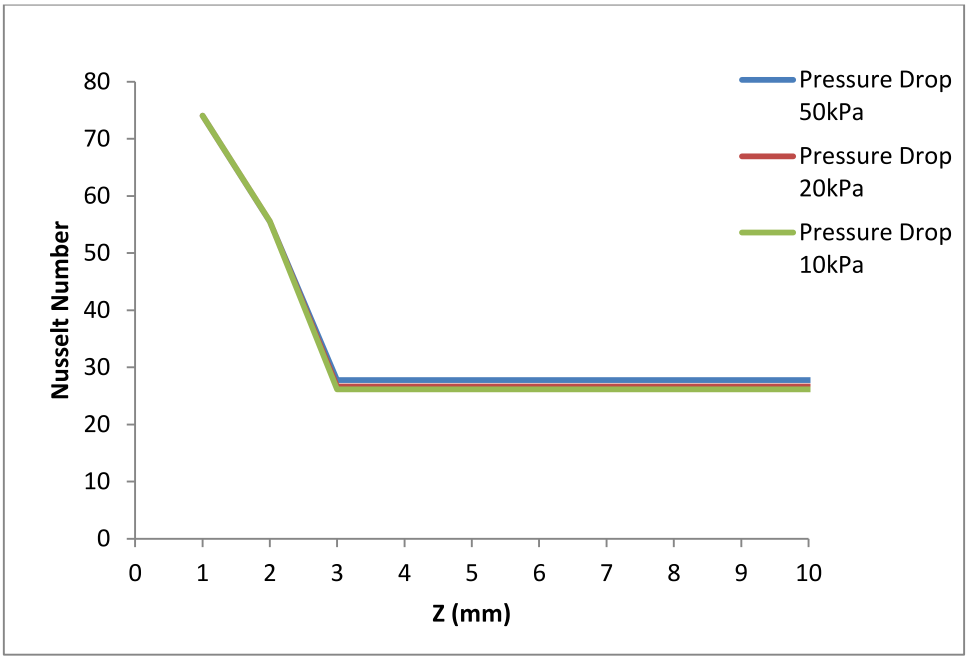


Fig. 7.28. Variation of Nusselt number in direction of flow

Value of Nusselt number for 50000 Pa is obtained which is 74. Because of boundary layer formation, average Nusselt number is decreasing along the direction of fluid flow. At entrance, values of average Nusselt numbers for 20000 Pa and 10000 Pa are approximately same. Nusselt number decreases sharply for each value of pressure drop in channel, after some distance from inlet Nusselt number does not change. It remains constant.

Fig.7.29. shows the variation of Nusselt number in channel for nanofluid and water as a coolant in microchannel heat sink. Value of Nusselt number is high in case when coolant is nanofluid in comparison of water as coolant in trapezoidal microchannel heat sink.

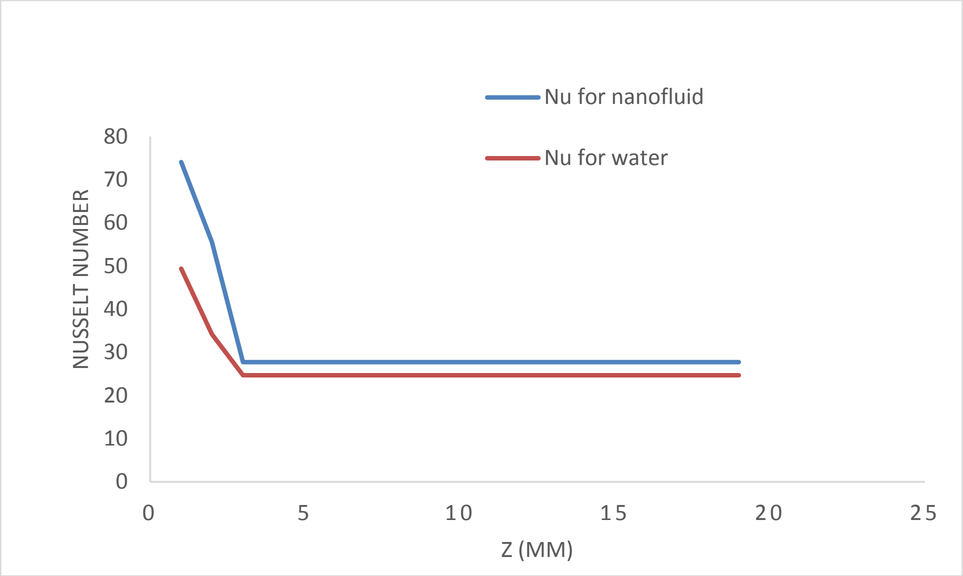


Fig. 7.29. Variation of Nusselt number for nanofluid and water

7.7 Variation of Heat Transfer Coefficient with Heat Flux

Single channel trapezoidal microchannel heat sink is also simulated by applying different heat flux at the bottom surface of heat sink. Heat transfer coefficient variation is checked along the length of channel. Fig. shows the variation of heat transfer coefficient along the direction of flow for different heat fluxes.

Fig.7.30 shows the variation of heat transfer coefficient along the direction of flow with different amount of heat fluxes. High heat flux causes increase in heat transfer coefficient along the direction of flow. Due to development of boundary layer and increase in thickness of boundary layer in inlet region, heat transfer coefficient's value is high in this region.

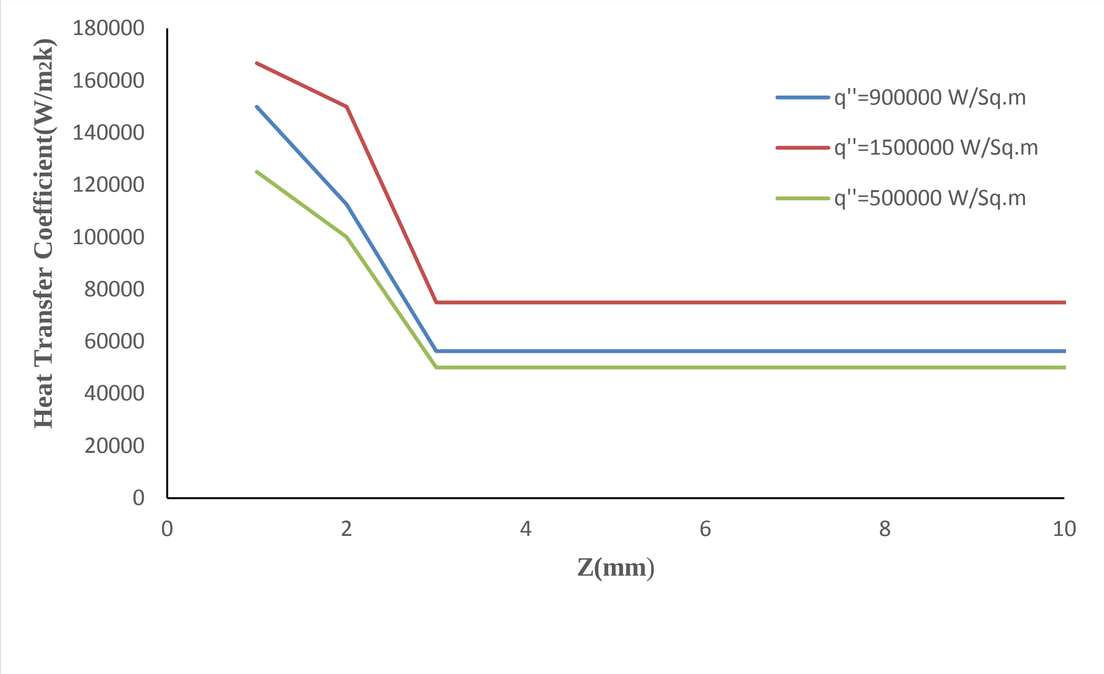


Fig. 7.30. Variation of heat transfer coefficient along the direction of flow

7.8 Variation of Nusselt number with Heat Flux

Fig.7.31 shows the variation of Nusselt number along the direction of flow with different amount of heat fluxes. High heat flux causes increase in Nusselt number along the direction of flow. Due to development of boundary layer and increase in thickness of boundary layer in inlet region, Nusselt number's value is high in this region.

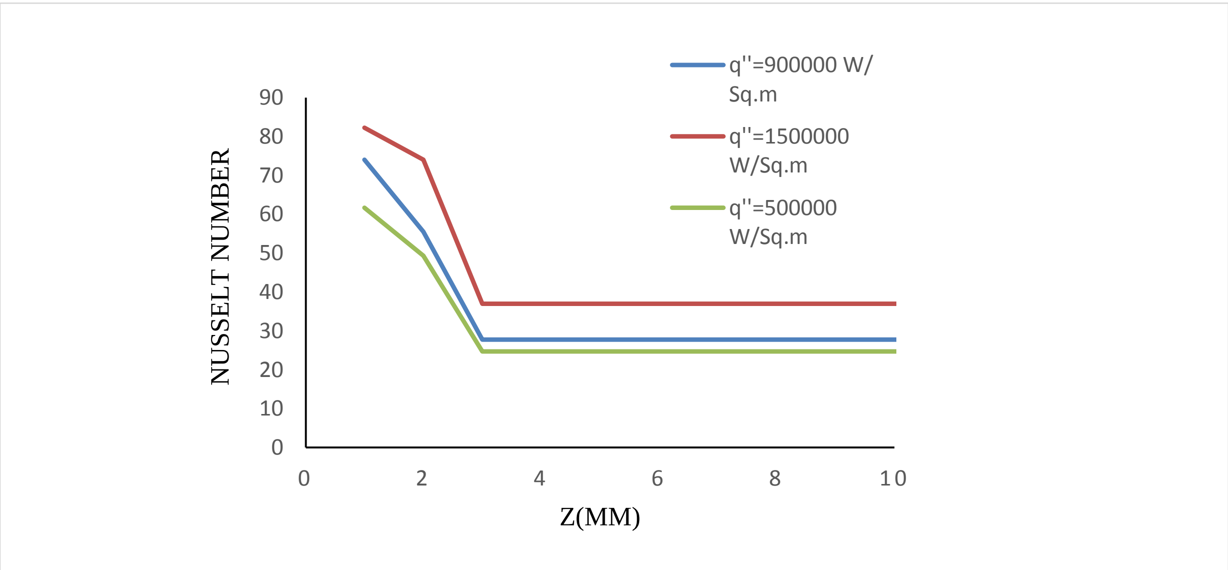


Fig. 7.31 Variation of Nusselt number along the direction of flow

8.1 Conclusion

In this study, numerical simulations on two dimensional fluid flow and three dimension heat transfer are performed. Conjugate heat transfer problem is solved to investigate performance of microchannel heat sink. Effect of nanofluid (TiO_2-H_2O) on different parameters is studied. Some assumptions are also made to solve this conjugate heat transfer problem, which involves conduction as well as convection. On the basis of presented results in previous chapter following conclusions can be described:

- Because of low fluid velocity, maximum temperature occurs below the channel outlet. Heat flux is concentrated at that particular region due to fluid's low velocity.
- Outlet temperature is maximum for 10000 Pa pressure drop and 900000 W/m^2 heat flux. On increasing the heat flux, outlet temperature increases because it increases temperature of fluid due to more heat transfer by convection. Outlet temperature is maximum for 1500000 W/m^2 heat flux. Velocity of fluid is low for low pressure drop. So during low pressure drop temperature of fluid rises more. Velocity of fluid is high for high pressure drop. So during high pressure drop temperature of fluid rises less.
- Comparison of heat transfer coefficient is done for water and TiO_2-H_2O.
- In comparison to water as coolant, heat transfer coefficient is increased on using nanofluid.
- Nusselt number is high for nanofluid as coolant in comparison to the water as coolant.
- Nusselt number and heat transfer coefficient are maximum at inlet region and decreases along the direction of flow. Thickness of boundary layer is very less in inlet region so Nusselt number and heat transfer coefficient both are very high in this region. But with the development of boundary layer along the direction fluid flow, thickness of boundary layer increases. So both these parameters decrease along the direction of fluid flow. Nusselt number is high on using nanofluid (TiO_2-H_2O).

8.2 Future Scope

If problems associated with nanofluids are solved, nanofluids definitely will be used as coolants in microchannel heat sink.

- As fluid's temperature remains very high few times so performance of microchannel heat sink can be studied using multiphase analysis.
- As nanofluids are suspension of nanoparticles of oxides in base fluid, two phase model can also be considered for microchannel heat sink.

REFERENCES

[1] Tuckerman, D.B. and Pease, R.F., (1981), “High performance heat sinking for VLSI”, IEEE Electronic Devices Letters, Vol. 2, pp. 126-129.

[2] G. Hetsroni , A. Mosyak, Z. Segal , G. Ziskind, “A uniform temperature heat sink for cooling of electronic devices”, International Journal of Heat and Mass Transfer 45 (2002) 3275–3286

[3] Piyanut Nitiapiruk, Omid Mahian , Ahmet Selim Dalkilic , Somchai Wongwises, “Performance characteristics of a microchannel heat sink using TiO2/water nanofluid and different thermophysical models”, International Communications in Heat and Mass Transfer 47 (2013) 98–104

[4] H.Y. Wu, Ping Cheng, “An experimental study of convective heat transfer in silicon microchannels with different surface conditions”, International Journal of Heat and Mass Transfer 46 (2003) 2547–2556

[5] Harpole, G.M. and Eninger, J.E., (1991), “Microchannel heat exchanger optimization”, IEEE Proceedings of the Seventh Semi-Therm Symposium, Vol. 361, pp. 59-63.

[6] J. Li, G.P. Peterson, P. Cheng (2004),” Three-dimensional analysis of heat transfer in a micro-heat sink with single phase flow”, International Journal of Heat and Mass Transfer 47 (2004) 4215–4231.

[7] Weilin Qu, Issam Mudawar, “Analysis of three-dimensional heat transfer in micro-channel heat sinks”, International Journal of Heat and Mass Transfer 45 (2002) 3973–3985.

[8] Chander Shekhar Sharma , Manish K. Tiwari , Bruno Michel ,Dimos Poulikakos, “Thermo fluidics and energetics of a manifold microchannel heat sink for electronics with recovered hot water as working fluid”, International Journal of Heat and Mass Transfer 58 (2013) 135–151

[9] X.L. Xie, Z.J. Liu, Y.L. He, W.Q. Tao, “Numerical study of laminar heat transfer and pressure drop characteristics in a water-cooled minichannel heat sink”, Applied Thermal Engineering xxx (2008) xxx–xxx

[10] J. Koo, C. Kleinstreuer, “Laminar nanofluid flow in microheat-sinks”, International Journal of Heat and Mass Transfer 48 (2005) 2652–2661

[11] Manay E., Sahin B., Yilmaz M., Gelis K., “Thermal Performance Analysis Of Nanofluids In Microchannel Heat Sink”, World Academy of Science, Engineering and Technology Vol:6 2012-07-26

[12] Seok Pil Jang, Stephen U.S. Choi, “Cooling performance of a microchannel heat sink with nanofluids, Applied Thermal Engineering 26 (2006) 2457–2463.

[13] H.A. Mohammed, P. Gunnasegaran, N.H. Shuaib, “Heat transfer in rectangular microchannels heat sink using nanofluids”, International Communications in Heat and Mass Transfer 37 (2010) 1496–1503.

[14] Bhanu Pratap Singh, Eshan Gupta, Dr.Arun K Lall, Harry Garg, “Computational Based Study to Analyse The Effect of Number of Channel and Cross Sectional Area in Micro channel Heat Sink”, International Journal of Applied Engineering Research, ISSN 0973-4562 Vol.7 No.11 (2012).

[15] Harry Garg, Vipender S.Negi, Nidhi, Arun K Lall, “Numerical Study Of Microscale Heat Sinks Using Different Shapes & Fluids”, Excerpt from the Proceedings of the 2013 COMSOL Conference in Bangalore.

[16] Navin Raja Kuppusamy, H.A. Mohammed, C.W. Lim, “Numerical investigation of trapezoidal grooved microchannel heat sink using nanofluids”, Thermochimica Acta573 (2013), 39-56.

[17] Mostafa Mirzaei, Maziar Dehghan, “Investigation of flow and heat transfer of nanofluid in microchannel with variable property approach”, Heat Mass Transfer (2013) 49:1803–1811.

[18] Ping Li, "Heat transfer and flow analysis of Al_2O_3–water nanofluids in microchannel with dimple and protrusion", International Journal of Heat and Mass Transfer, Volume 73, June2014, Pages456–467.

[19] H.A. Mohammed, P. Gunnasegaran, N.H. Shuaib," Influence of various base nanofluids and substrate materials on heat transfer in trapezoidal microchannel heat sinks", International Communications in Heat and Mass Transfer Volume 38, Issue 2, February 2011, Pages 194-201.

[20] Roy W. Knight, Donald J. Hall, John S. Goodling, and Richard C. Jaeger, "Heat Sink Optimization with Application to Microchannels", IEEE Transactions on Components, Hybrids, and Manufacturing Technology, vol. 15, no. 5, October 1992.

[21] Mohammad Kalteh , Abbas Abbassi , Majid Saffar-Avval , Arjan Frijns , Anton Darhuber , Jens Harting, "Experimental and numerical investigation of nanofluid forced convection inside a wide microchannel heat sink", Applied Thermal Engineering 36 (2012) 260-268.

[22] Saeed Zeinali Heris, Seyyed Hossein Noie, Elham Talaii, Javad Sargolzaei, "Numerical investigation of Al2O3/water nanofluid laminar convective heat transfer through triangular ducts", Nanoscale Research Letters 2011, 6:179

[23] Deewakar Sharmaa, Parbar Pratham Singha, Harry Garg, "Comparative Study of Rectangular and Trapezoidal Microchannels Using Water and Liquid Metal", Procedia Engineering 51 (2013) 791 – 796.

[24] Yue-Tzu Yang, Kuo-Teng Tsai, Yi-HsienWang, Shih-Han Lin, Numerical study of microchannel heat sink performance using nanofluids, International Communications in Heat and Mass Transfer 57, 2014,27–35.

[25] H.A. Mohammed, P. Gunnasegaran, N.H. Shuaib, The impact of various nanofluid types on triangular microchannels heat sink cooling performance, International Communications in Heat and Mass Transfer 38, 2011, 767–773.

[26] Deewakar Sharma, Parbar Pratham Singh & Harry Garg, "Numerical Analysis of Trapezoidal Shape Double Layer Microchannel Heat Sink", International Journal of Mechanical and Industrial Engineering (IJMIE) ISSN No. 2231-6477, Vol-3, Iss-1, 2013.

[27] Weilin Qu and Issam Mudawar, "Thermal Design Methodology for High-Heat-Flux Single-Phase and Two-Phase Micro-Channel Heat Sink", IEEE Transactions on Components and Packaging Technologies, VOL. 26, NO. 3, September 2003.

[28] C.Y. Zhao, T.J. Lu, "Analysis of microchannel heat sinks for electronics cooling", International Journal of Heat and Mass Transfer 45 (2002) 4857–4869.

[29] Ahmed Mohammed Adham, Normah Mohd Ghazali, Robiah Ahmed, "Thermal and hydrodynamic analysis of microchannel heat sinks: A review", Renewable and Sustainable Energy Reviews 21 (2013) 614-622.

[30] Shakuntala Ojha, CFD analysis on forced convection cooling of electronic chips, National Institute of Technology, India, 2009.

[31] P. Gunnasegaran, N.H. Shuaib, H.A. Mohammed, M.F. Abdul Jalal and E. Sandhita, "Heat Transfer Enhancement in Microchannel Heat Sink Using Nanofluids".

APPENDIX I

CFD analysis of single microchannel heat sink, involves following steps-

(1) Pre-Processing

(2) Solver

(3) Post-Processing

Preprocessing involves geometry creation and mesh of geometry. Here geometry of single microchannel heat sink is created in Pro E Wildfire 4.0 and exported in ICEM CFD 14.0 in IGES format. Solver performs numerical computations. Third and last step is post processing which involves report generation and understanding flow with different plots and contours.

Steps in Pro E Wildfire 4.0 for Creating Single microchannel Heat Sink-

After opening the Pro E Wilfire4.0,

File > Set Working Directory > Ok

New > Part > mmns_ part_solid > Ok

Select Front Plane > Sketch rectangle of dimensions 4×1.5 mm^2 > Extrude 10mm > ok

Select Front Face > Sketch > Draw trapezoidal channel of dimensions specified in table > Extrude 10mm > Cut > ok

File > Save A Copy in IGES format.

Steps in ICEM CFD 14.0 for Meshing –

After opening ICEM CFD 14.0,

File > Import Geometry > STEP/IGES > Select Geometry File > Open > Apply

Assigning names to different parts,

Right click on create part > Part Name (Channel) > Select entities comprises channel > apply, similarly inlet, outlet, insulated top, insulated left, insulated right, heat flux name are assigned to different surfaces in the geometry.

Creating Body- Fluid body and solid body are defined by following below steps-

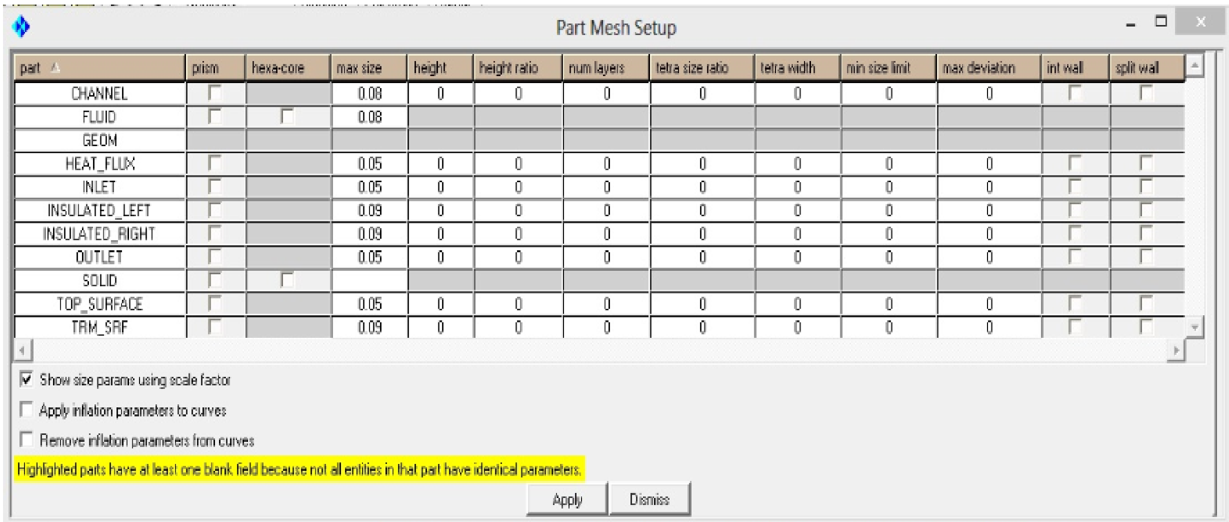
Create Body > Part Name > Fluid > Select two diagonally opposite corners of channel > apply again, Part name > Solid > select two diagonally opposite corners of sink > apply

Repair Geometry > Tolerance > 0.005 > Apply

Creating Volume Mesh –

Mesh > Global Mesh Setup > Max element >10 >apply >volume meshing parameters > Mesh type >Tetra/mixed > apply

Part mesh Setup > Following values are assigned for different parts > apply

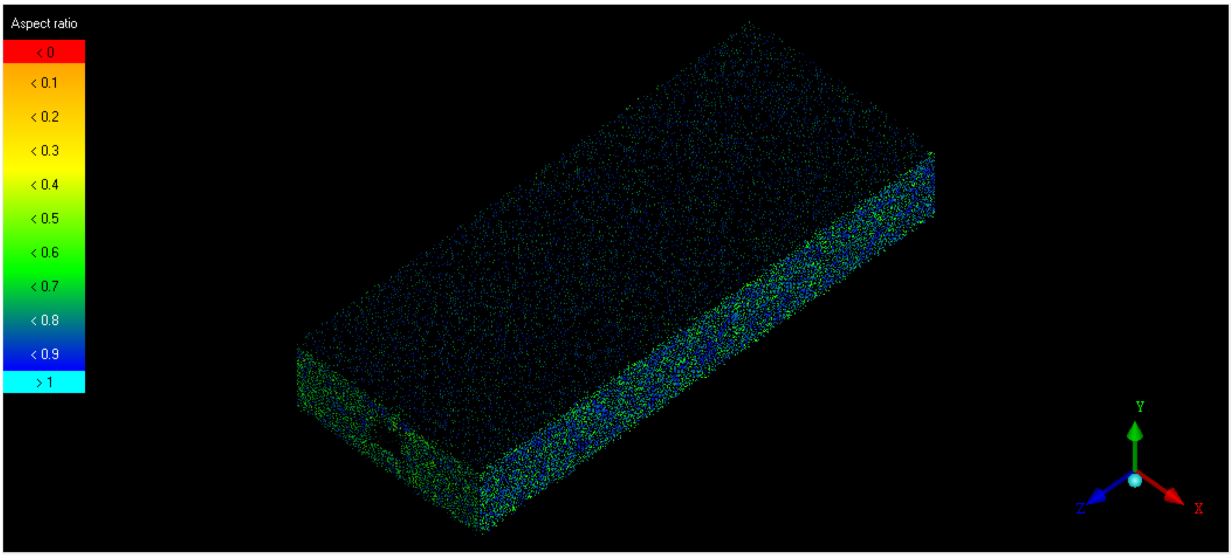


part	prism	hexa-core	max size	height	height ratio	num layers	tetra size ratio	tetra width	min size limit	max deviation	int wall	split wall
CHANNEL			0.08	0	0	0	0	0	0	0		
FLUID			0.08									
GEOM												
HEAT_FLUX			0.05	0	0	0	0	0	0	0		
INLET			0.05	0	0	0	0	0	0	0		
INSULATED_LEFT			0.09	0	0	0	0	0	0	0		
INSULATED_RIGHT			0.09	0	0	0	0	0	0	0		
OUTLET			0.05	0	0	0	0	0	0	0		
SOLID												
TOP_SURFACE			0.05	0	0	0	0	0	0	0		
TRM_SRF			0.09	0	0	0	0	0	0	0		

Compute Mesh > Volume Mesh >Mesh Type > Tetra/Mixed > Mesh Method > Robust (octree) > Select Geometry > From File > Select tin file > Open > Compute

Check Quality of Mesh –

Right click on mesh > Color by quality >



Save Mesh > ok

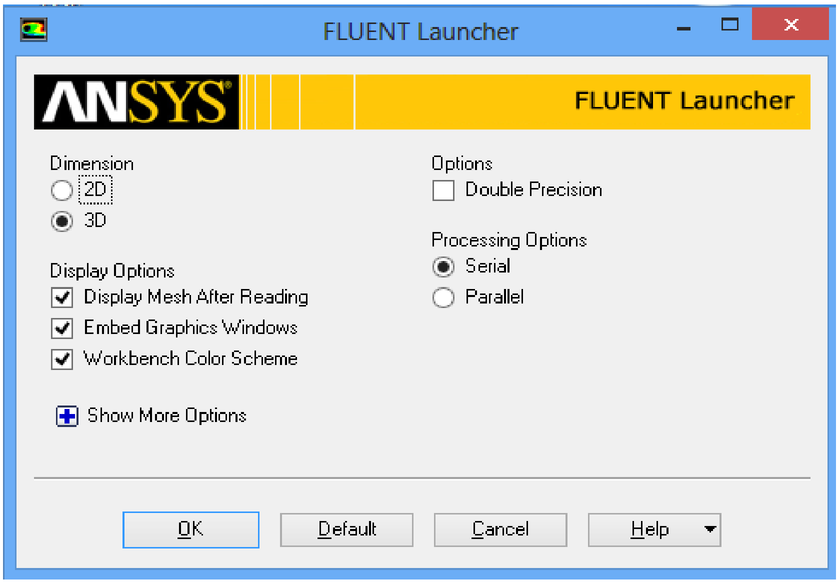
Writing input file for FLUENT 14.0-

Output > Output Solver > Fluent_V6 > Ok,

Write input > Save as > File name > nastran > save

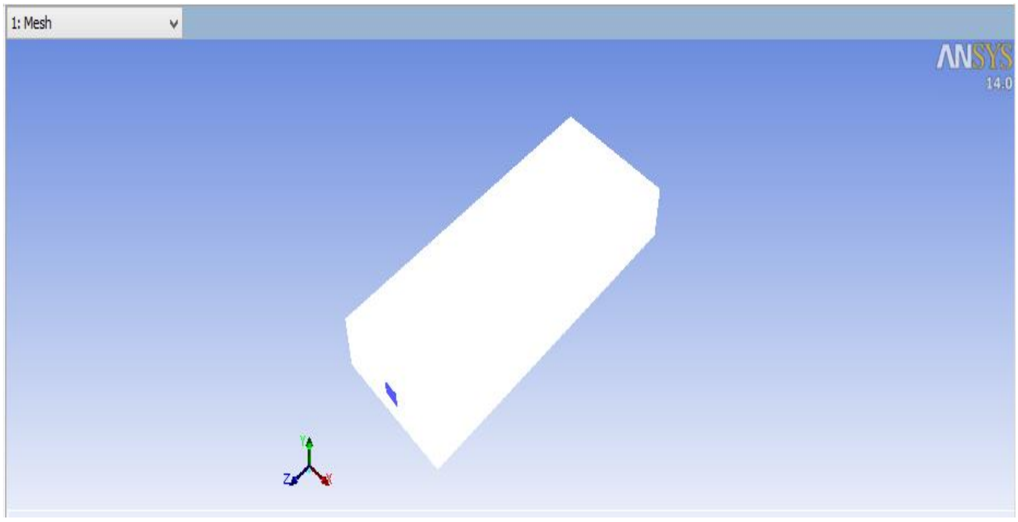
Solver

Launch ANSYS FLUENT 14.0 as given below,



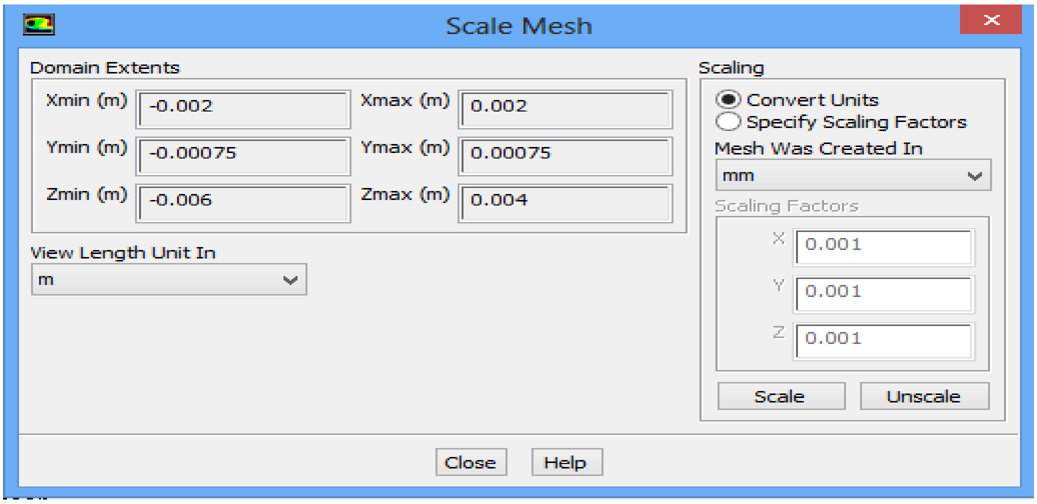
Click on ok

File > read > mesh…select fluent (MESH file) > ok

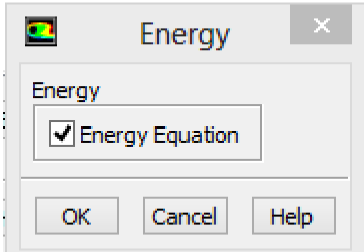


Mesh > check

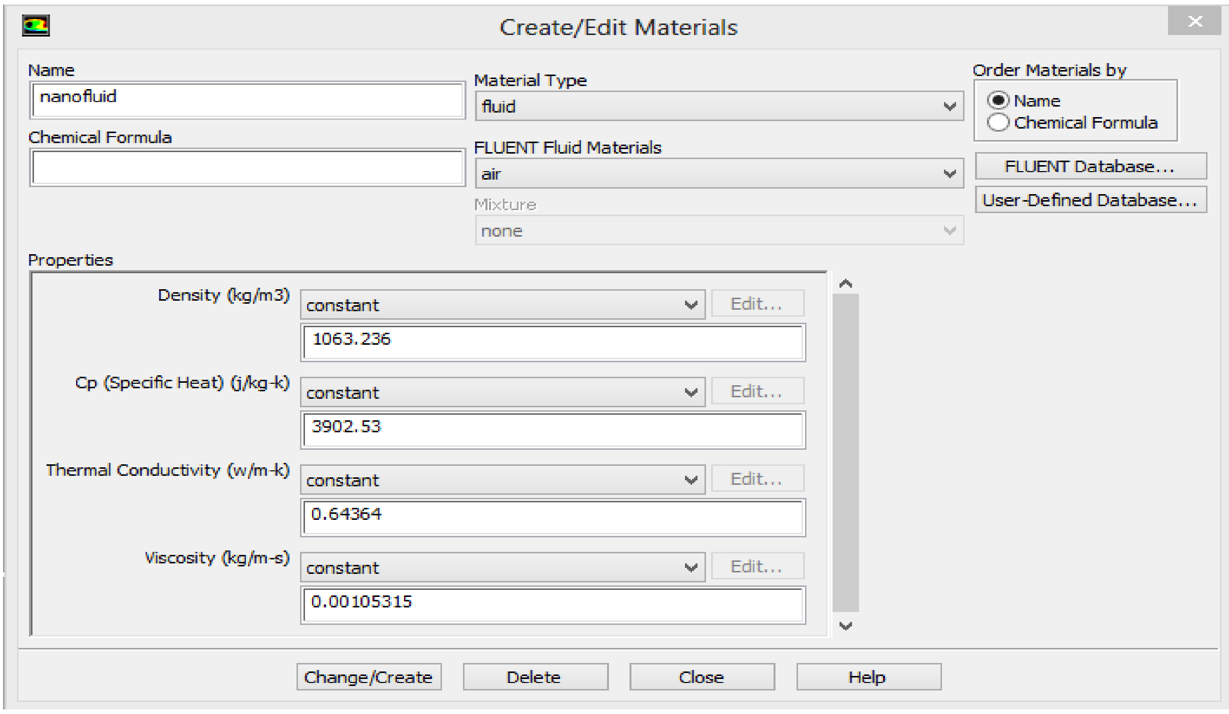
Scale > mesh was created in > mm > scale > close



Models > energy > on

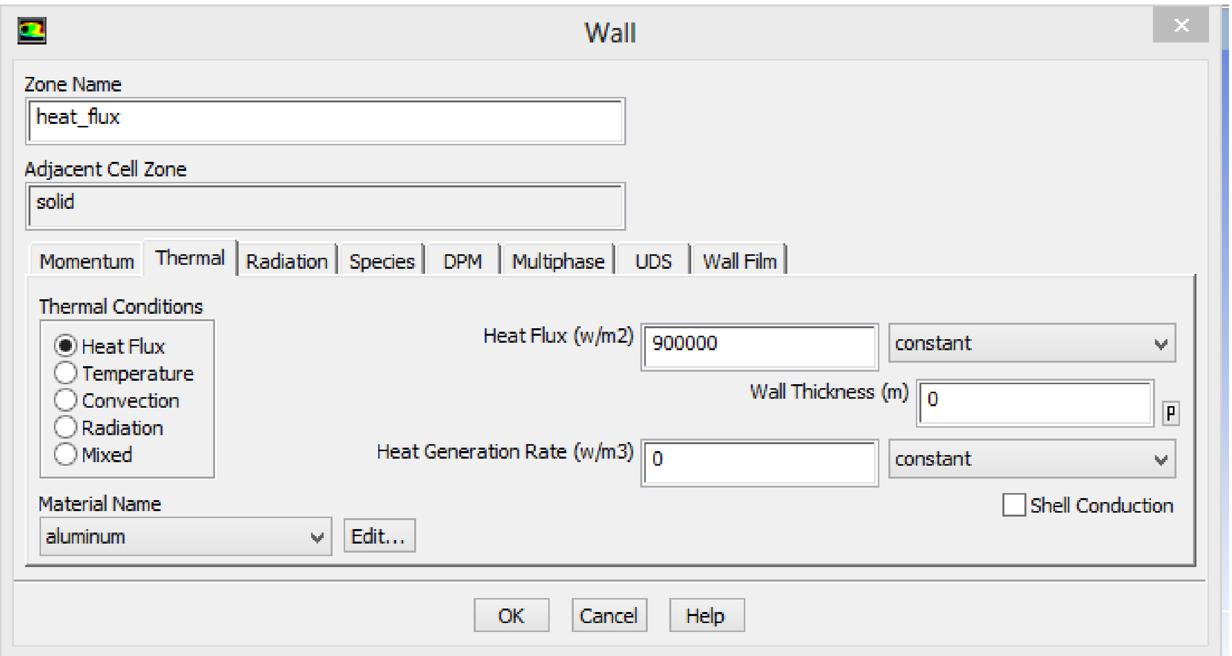


Materials > fluid > enter properties of nanofluid

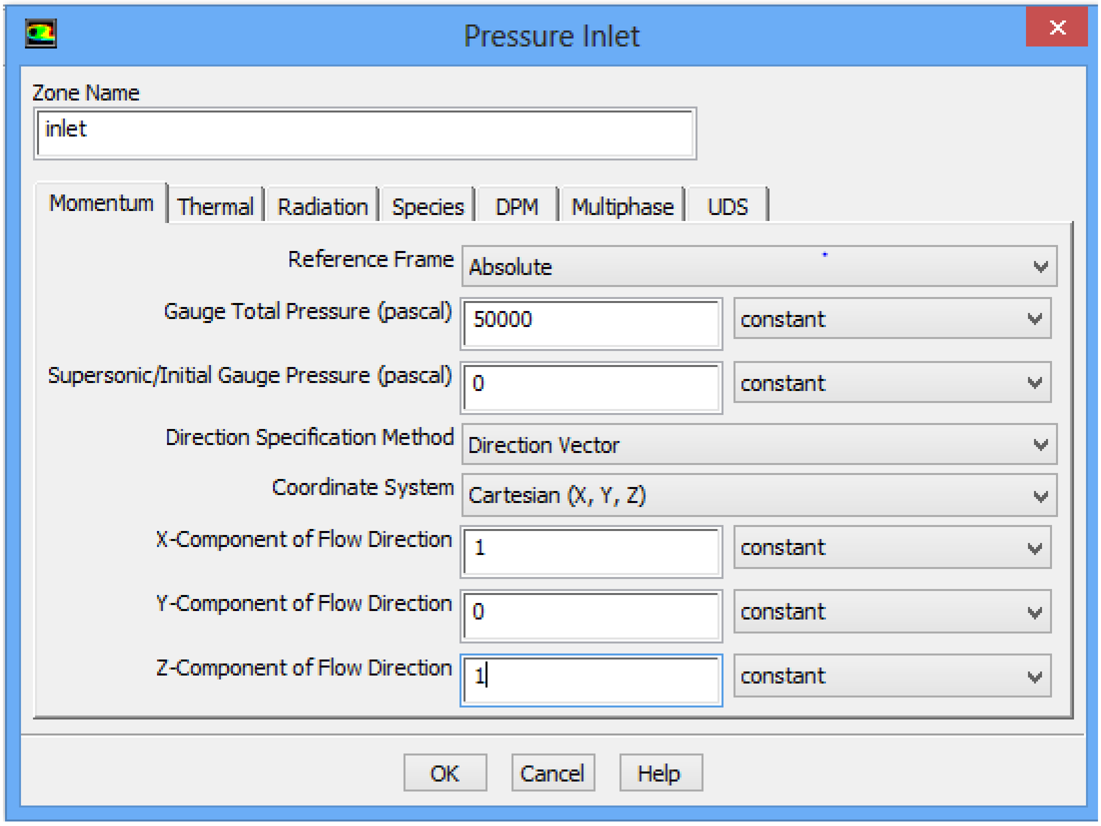


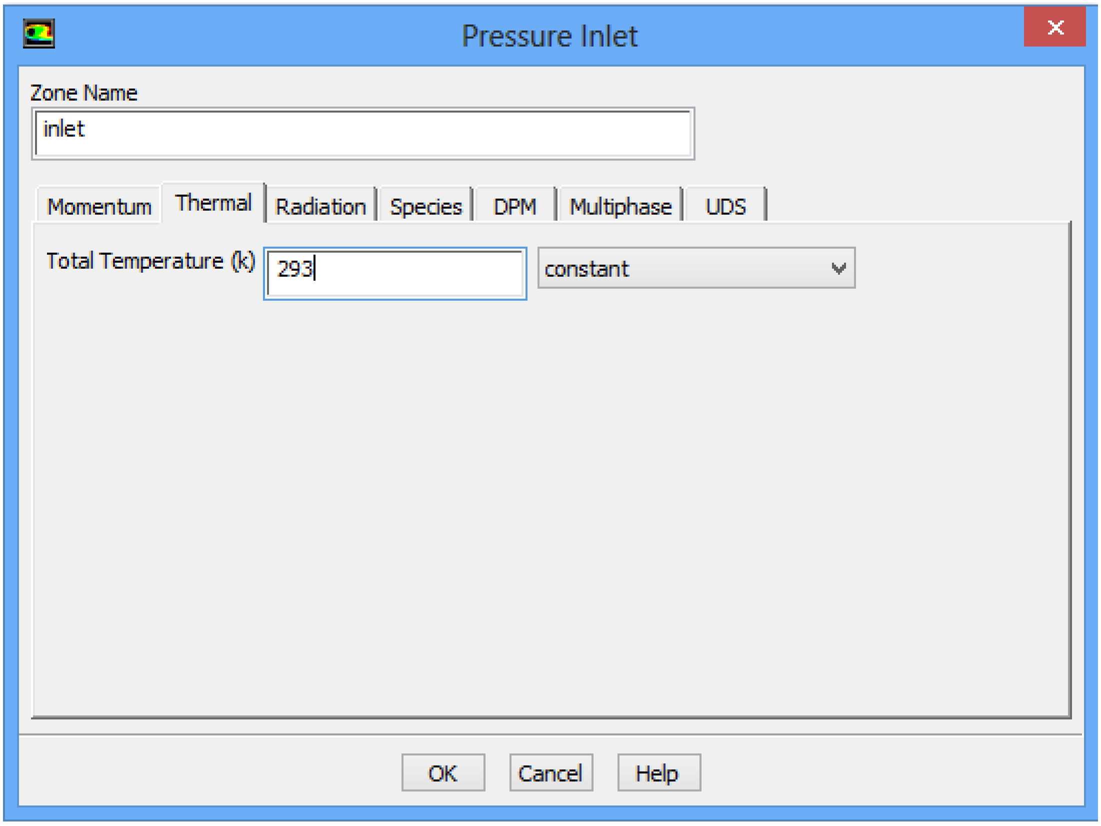
Materials > solid > enter properties of solid

Boundary conditions > heat flux

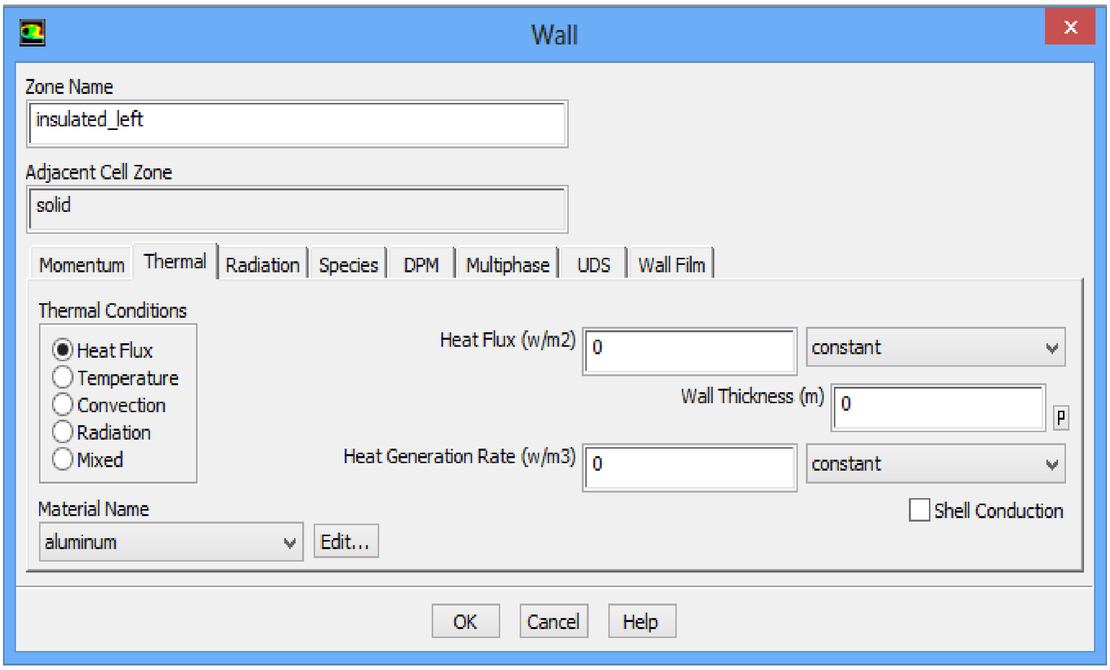


Boundary conditions > inlet > edit

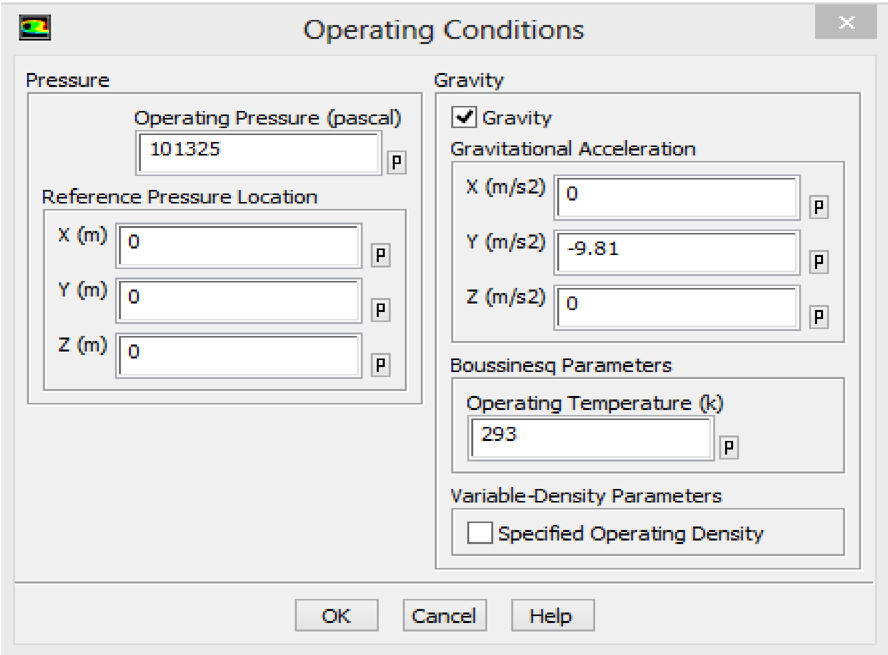




Walls of sink are insulated.

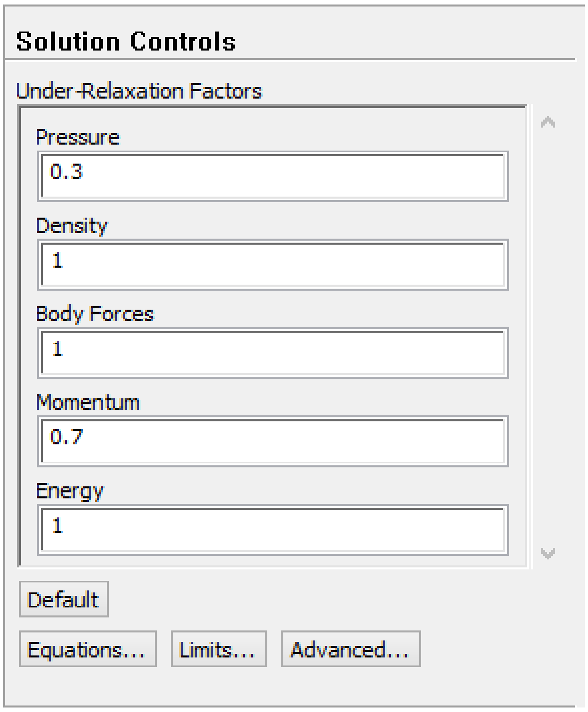


Define > operating conditions

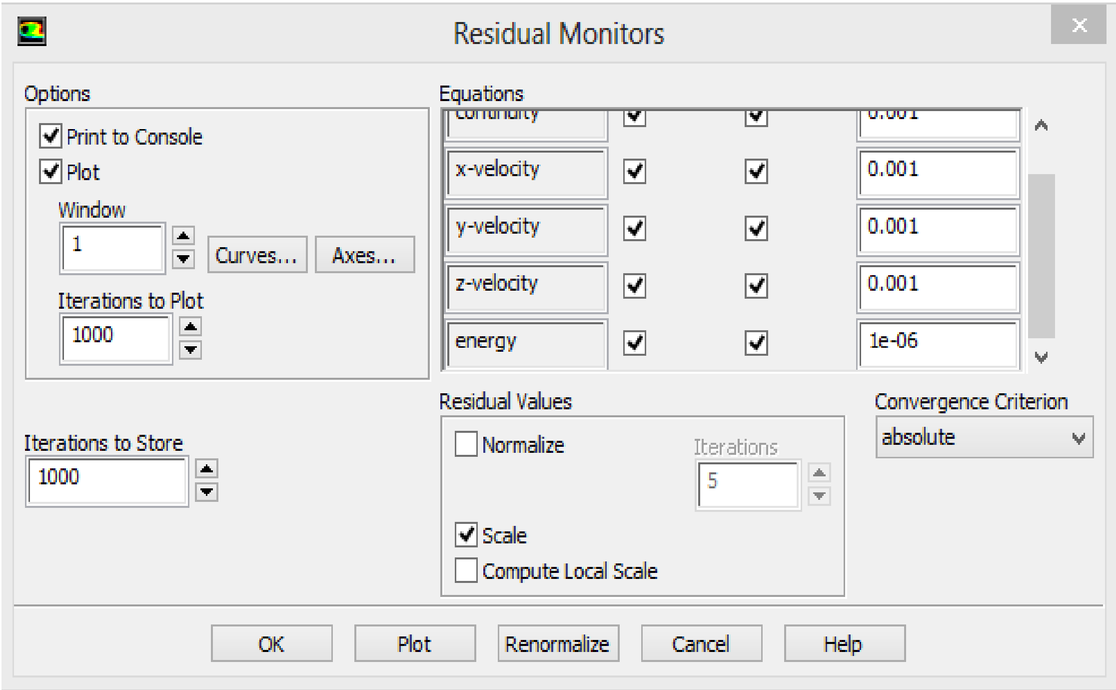


Solution methods > pressure, momentum, energy > second order upwind

Solution controls



Monitors > Residuals- Print, Plot



Solution initialization > standard initialization >compute from > heat flux

Run calculation > number of iterations > 100 > calculate