



Organization of Islamic Cooperation

# **Simulation and comparison of heat flow through rectangular and cylindrical fins by ANSYS**

A thesis submitted to the department of MECHANICAL AND CHEMICAL ENGINEERING (MCE), ISLAMIC UNIVERSITY OF TECHNOLOGY (IUT)

IN THE PARTIAL FULLFILMENT OF REQUIREMENT FOR THE DEGREE OF BACHELOR IN SCIENCE IN MECHANICAL & CHEMICAL ENGINEERING

Prepared by:

**Md. Anisur Rahman (091422)**

**Nazran Zubaer (091426)**

**Ratul Rizvan (091447)**

Supervised by:

**Dr. Mohammad Ali Jinnah**

Associate Professor, Department of Mechanical and Chemical Engineering (MCE),

Islamic University of Technology (IUT)

The Organization of Islamic Cooperation.

# CANDIDATES' DECLARATION

It is hereby declared that this thesis or any part of it has not been submitted elsewhere for the award of any degree or diploma.

Signature of the Candidate

---

**Md. Anisur Rahman**

Student No.: 091422, Session: 2012-13,  
Department of Mechanical & Chemical  
Engineering,  
IUT, OIC.

Signature of the Candidate

---

**Nazran Zubaer**

Student No.: 091426, Session: 2012-13  
Department of Mechanical & Chemical  
Engineering,  
IUT, OIC.

Signature of the Candidate

---

**Ratul Rizvan**

Student No.: 091447, Session: 2012-13,  
Department of Mechanical & Chemical Engineering,  
IUT, OIC.

Signature of the Supervisor

---

**Dr. Mohammad Ali Jinnah,**

Associate Professor,  
Department of Mechanical & Chemical Engineering,  
Islamic University of Technology (IUT), OIC.

## ACKNOWLEDGEMENT

We express our heartiest gratefulness to Almighty Allah for His blessings, which enables us to complete this thesis successfully. It is our pleasure to express our gratitude to our project supervisor **Dr. Mohammad Ali Jinnah**, Department of Mechanical & Chemical Engineering, **Islamic University of Technology (IUT)** for his constant support, guidance, helpful suggestion and supervision at all stages of the research work. He had dedicated his valuable time, even at odd hours, which had ensured completion of this project. We also express the gratefulness to **Prof. Dr. Md. Abdur Razzaq Akhanda**, Head of the department, Department of Mechanical & Chemical Engineering, IUT for providing us all sorts of support. Special thanks to our beloved parents.

We are highly indebted to the above mentioned persons and without their encouragement; it would have been impossible to complete this dissertation.

# CONTENTS

Page

Acknowledgement \_\_\_\_\_ 3

## Chapter One: Theory

1.1 Introduction \_\_\_\_\_ 10

1.2 Fins \_\_\_\_\_ 10

1.3 Rectangular fins \_\_\_\_\_ 11

1.4 Cylindrical fins \_\_\_\_\_ 12

1.5 ANSYS \_\_\_\_\_ 13-14

1.5.1 ANSYS Mechanical Flo \_\_\_\_\_ 13

1.5.2 ANSYS CFD \_\_\_\_\_ 14

1.5.3 ANSYS CFD Flo \_\_\_\_\_ 14

1.6 Mesh \_\_\_\_\_ 14

1.7 Loads \_\_\_\_\_ 15

1.8 Nodal Temperature \_\_\_\_\_ 15

1.9 Heat Flow \_\_\_\_\_ 15

1.10 Condenser \_\_\_\_\_ 16

## Chapter Two: Background

2.1 Literature review \_\_\_\_\_ 17

2.2 Previous Works \_\_\_\_\_ 17

2.2 (A) Paper Review 1 \_\_\_\_\_ 18-20

2.2 (B) Paper Review 2 \_\_\_\_\_ 20-22

## Chapter Three: Thesis Work

3.1 Objectives	23
3.2 Model	23-25
3.2.1 Rectangular Fin	23-24
3.2.2 Cylindrical Fin	24-26
3.3 Boundary Conditions	26
3.4 Solver and Solving Equations	26-27

## Chapter Four: Methodology

4.1 Rectangular Fin	28-35
4.1.1 Building Model	28-33
4.1.2 Defining Loads and solve	33-35
4.2 Cylindrical Fins	36-42
4.2.1 Building Model	36-39
4.2.2 Defining Loads and solve	39-42

## Chapter Five: Result

5.1 Available Solutions	43
5.2 Rectangular Fins	44-52
5.3 Cylindrical Fins	53-61

## Chapter Six: Result Analysis and Comparison

6.1 Available Results	62
6.2 Nodal Temperature Analysis	62
6.3 Distance vs. temperature graph	63-64

6.4 Distance vs. Thermal Flux graph _____	64-65
6.5 Distance vs. thermal gradient graph _____	66-67
6.6 Heat Flow Rate _____	67-68

## Chapter Seven: Conclusion

7.1 Decisions _____	69
7.2 Future Recommendations _____	69
References _____	70

## List of Figures

## Page

Fig 1: Rectangular Fin _____	11
Fig 2: ANSYS model of Rectangular Fin _____	12
Fig 3: Cylindrical Fins _____	12
Fig 4: Simple meshing of Rectangular Fin _____	18
Fig 5: Temperature v/s distance graph _____	19
Fig 6: Contour Plot of the Rectangular Fin _____	20
Fig 7: Temperature v/s distance graph for Aluminium _____	21
Fig 8: Temperature v/s distance graph for mild steel _____	21
Fig 9: Temperature v/s distance graph for stainless steel _____	21
Fig 10: Temperature v/s distance graph for 6.5 mm perforation _____	21
Fig 11: Temperature difference and distribution along the length of solid perforated fin _____	21
Fig 12: 3-D model of rectangular fin with dimension _____	24
Fig 13: 3-D model of cylindrical fin with dimension _____	25
Fig 14: Nodal Temperature for Rectangular fin _____	44

<b>Fig 15: Heat Flow for Rectangular fin</b>	<b>44</b>
<b>Fig 16: Thermal Gradient Vector Plot for Rectangular fin</b>	<b>45</b>
<b>Fig 17: Thermal Gradient Vector Sum for Rectangular fin</b>	<b>46</b>
<b>Fig 18: Thermal Flux Vector Plot for Rectangular fin</b>	<b>46</b>
<b>Fig 19: Thermal Flux Vector Sum for Rectangular fin</b>	<b>47</b>
<b>Fig 20: Volumetric Element Solution for Rectangular fin</b>	<b>48</b>
<b>Fig 21: Temperature vs. Distance graph for Rectangular fin</b>	<b>49</b>
<b>Fig 22: Distance vs. Thermal Flux graph for Rectangular fin</b>	<b>50</b>
<b>Fig 23: Distance vs. Thermal Gradient graph for Rectangular fin</b>	<b>51</b>
<b>Fig 24: Thermal flux vs. Temperature graph for Rectangular fin</b>	<b>51</b>
<b>Fig 25: Thermal Gradient vs. Temperature Graph for Rectangular fin</b>	<b>52</b>
<b>Fig 26: Thermal gradient vs. Thermal flux Graph for Rectangular fin</b>	<b>52</b>
<b>Fig 27: Nodal Temperature for cylindrical fin</b>	<b>53</b>
<b>Fig 28: Heat Flow for cylindrical fin</b>	<b>53</b>
<b>Fig 29: Thermal Gradient Vector Sum for cylindrical fin</b>	<b>54</b>
<b>Fig 30: Thermal Flux Vector Plot for cylindrical fin</b>	<b>55</b>
<b>Fig 31: Thermal Flux Vector Sum for cylindrical fin</b>	<b>56</b>
<b>Fig 32: Volumetric Element Solution for cylindrical fin</b>	<b>57</b>
<b>Fig 33: Temperature vs. Distance graph for cylindrical fin</b>	<b>58</b>
<b>Fig 34: Distance vs. Thermal Flux graph for cylindrical fin</b>	<b>58</b>
<b>Fig 35: Distance vs. Thermal Gradient graph for cylindrical fin</b>	<b>59</b>
<b>Fig 36: Thermal flux vs. Temperature graph for cylindrical fin</b>	<b>60</b>

<b>Fig 37: Thermal Gradient vs. Temperature Graph for cylindrical fin</b>	<b>60</b>
<b>Fig 38: Thermal gradient vs. Thermal flux Graph for cylindrical fin</b>	<b>61</b>
<b>Fig 39: Heat flow rate chart for Rectangular fin</b>	<b>67</b>
<b>Fig 40: Heat flow rate chart for cylindrical fin</b>	<b>68</b>



# ABSTRACT

Heat generation has been a common phenomenon in different mechanical machines. Heat dissipation system thus has become more and more important for better performance and smooth running of the machine. The dissipation has to be not only efficient but also economical and shorter in size. Fins are extended surface which increases the surface area of the machine. This project focuses on two types of fins e.g. rectangular and cylindrical. The heat flow is simulated by the help of simulation software ANSYS 14.0. The rectangular fin is simulated then verified with the experimental result. Then cylindrical fin is simulated and results are compared with the rectangular fin results. Thus a generalized methodology is established and can be used to simulate and compare other types of fins.

# CHAPTER ONE

## THEORY:

---

### 1.1 INTRODUCTION

Heat generation is a common phenomenon in different machines and cycles. This heat must be dissipated or removed by some arrangement. The thermal efficiency to some extent depends on the successful removal of heat.

Fins are considered as a simple and common form of heat transferring arrangement. It is used in different machines like refrigerator, condenser, evaporator, heat engines even in airplanes too.

The knowledge of heat flow characteristics through may help to determine the type, size, area and no of fins to be used in particular case. Simulation is an effective way to explain heat flow characteristics. Successful simulation will produce an ideal condition or future reference of fins in different machines in this case condenser.

As simulation software ANSYS is selected. ANSYS is huge simulation software which contains different sub section containing Workbench, CFD, CFM, CFD Flo, ICEM CFD, Fluent etc. In this case ANSYS Mechanical Flo 14.0 has been used.

Fins are of different types. Rectangular and cylindrical or pin type dominates in case of usage. So an experiment based simulation for rectangular fins and a theoretical based simulation is done. The theoretical approach is based on the size and dimension of rectangular fins. With the reference of the results achieved from the rectangular fin simulation, another simulation for the cylindrical fin is done. Then performance, effectiveness and efficiency are compared. By this procedure the right type of fin can be selected for any other purpose.

### 1.2 FINS

Fins are the extended surface attached to different heat producing machines to exchange heat with the atmosphere and hence create a cooling effect in the machine. The amount of conduction, convection, or radiation of an object determines the amount of heat it transfers. Increasing the temperature difference between the object and the environment, increasing the convection heat transfer coefficient, or increasing the surface area of the object increases the heat transfer. Sometimes it is not

economical or it is not feasible to change the first two options. Adding a fin to an object, however, increases the surface area and can sometimes be an economical solution to heat transfer problems.

There are different types of fins used in different purposes. Different names have been given to fins according to their purpose of use and shape. Some of them are:

1. Rectangular
2. Cylindrical/ Pin type
3. Annular
4. Grid

But in this case focus has been paid particularly to rectangular and cylindrical fins.

### 1.3 Rectangular Fins

Fins those are rectangular in shape can be treated as rectangular fins. It has base and tip in it. Normally heated end is considered as base and the tip is the conductive or convective end. Normal rectangular fin may look like the following figure:

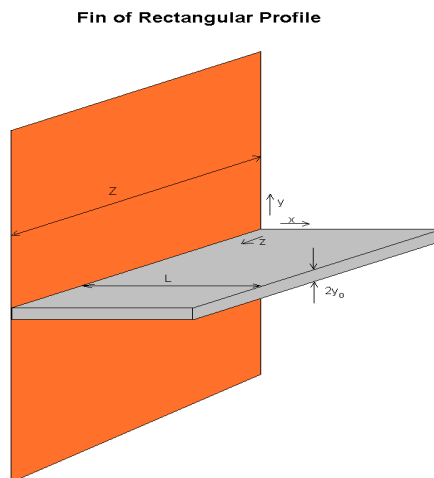


Fig: 1

But in this case, due to the presence of condenser tube the base is assumed to be a half circle. The radius of the half circle will be equal to the outer radius of the condenser tube. The following figure is the actual figure that has been used in simulation.

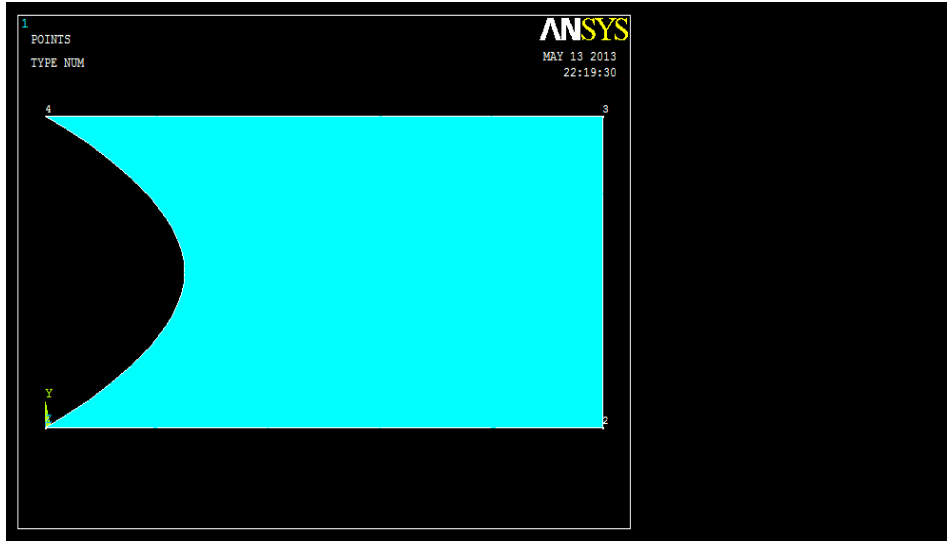


Fig: 2

## 1.4 Cylindrical Fins

Fins those are cylindrical in shape can be treated as cylindrical (or pin) fins. It also has base and tip in it. Normally heated end is considered as base and the tip is the conductive or convective end. Normal cylindrical fin may look like the following figure:

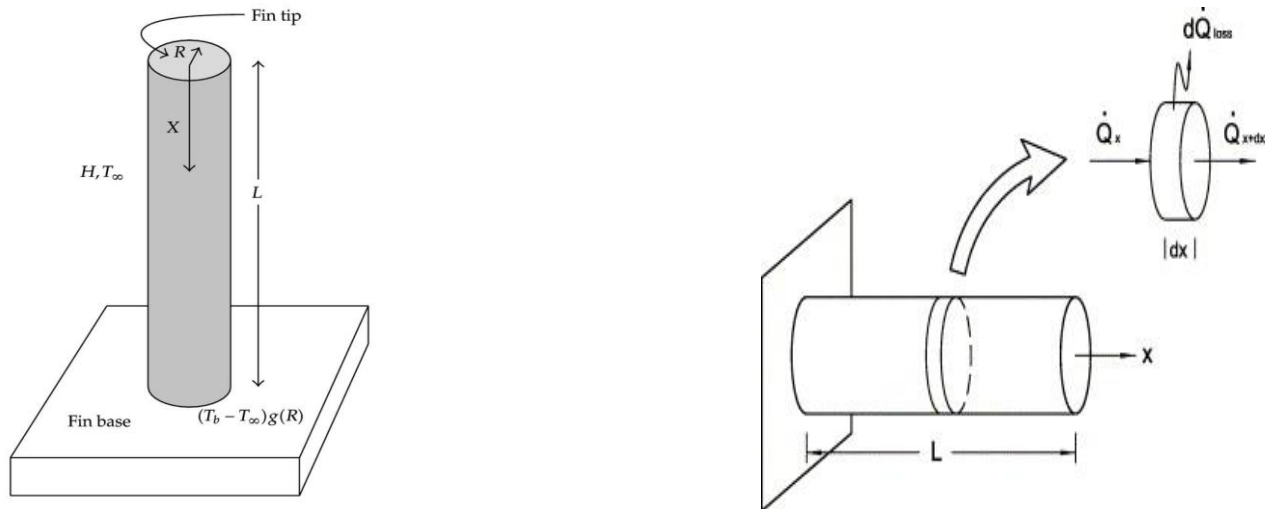


Fig: 3

## **1.5 ANSYS**

### **1.5.1 ANSYS Mechanical Flo**

ANSYS is simulation software used widely for different purposes. It has many subsections as stated before. In example: Fluent, ICEM CFD, Workbench, CFM, CFD Flo etc. Normally in case of heat flow simulation, CFD based software are used worldwide. In case of CFD simulation it is necessary to consider the fluid flow and fluid characteristics, but to concentrate specially on fins a simple CFD software named ANSYS Mechanical FLO is available. In this thesis, this software is used. As ANSYS 14.0 version is the latest software available right now so the result may be considered as the most accurate possible.

ANSYS Mechanical Flo-14.0 normally is 2-d software with high performance and easier user interface. As fins have very short width, so it does not have a huge effect which should be considered. So it was easy to neglect that axis containing width. In case of rectangular fins 2-d design is very easily done. But in case of cylindrical fins, the axis is considered, and the calculation is done according to that.

The general purpose fluid analysis tools are the renowned ANSYS CFX and ANSYS FLUENT products, now also available together in the ANSYS CFD bundle. ANSYS CFD have access to an unprecedented array of fluid flow physics models, allows analyzing equipment with a great deal of confidence. ANSYS CFD technology is highly-scalable, allowing for efficient parallel calculations on thousands of processing cores. ANSYS CFD also includes the full-featured ANSYS CFD-Post fluid flow post-processing tool. This can be used for advanced quantitative analysis and high-quality visualizations. When ANSYS CFD is used in combination with ANSYS Mechanical it is eminently suitable to solve complex fluid-structure interaction problems.

### **1.5.2 ANSYS CFD**

Computational fluid dynamics (CFD) is an engineering method in which flow fields and other physics are calculated in detail for an application of interest. ANSYS uses a multidisciplinary approach to simulation in which fluid flow models integrate seamlessly with other types of physics simulation technologies. The CFD or fluids simulation results can be used to illustrate how a product or process operates, to troubleshoot problems, to optimize performance and to design new products.

ANSYS CFD provides access to both the well-known ANSYS FLUENT and ANSYS CFX products. Also available separately, these are the main general-purpose fluids simulation products offered by ANSYS. These two core fluids simulation solvers represent more than 1,000 person-years of research and development. This effort translates into the key benefits of fluids

ANSYS which enables product design and analysis in a virtual environment has revolutionized fluid dynamics by automating the solution, even for problems that are numerically large. By identifying physical forces and flow characteristics that are sometimes impossible to measure or gain insight into, CFD solutions from ANSYS can help a company dramatically improve time to market, slash development costs and fulfill critical product promises.

### **1.5.3 ANSYS CFD Flo**

ANSYS CFD-Flo is integrated into the unified ANSYS Workbench platform. This easy-to-use platform provides you access to bi-directional parametric CAD connections, powerful geometry and meshing tools, an automated project level update mechanism, pervasive parameter management, and integrated optimization tools. Key benefits include:

To quickly prepare product/process geometry for flow analysis without tedious rework  
To avoid duplication through a common data model that is persistently shared across physics beyond basic fluid flow  
To easily define a series of parametric variations in geometry, mesh, physics and post processing, and to automatically get new CFD results for that series after a single mouse click  
To improve product/process quality by increasing the understanding of variability and design sensitivity

## **1.6 Mesh**

Meshing is an important tool in simulation process. It divides the whole heat conducting surface into small divisions. The whole heat transfer equation is applied into each of these divisions. So the software solves the equation for numerous times. Mesh size can be selected by the user. Some predetermined meshing characteristics are also available in the software such as prism, triangle, tetra and quad mesh. In this case, quad mesh with global attributes is selected.

## 1.7 Loads

Loads are very important in case of heat transfer and flow. In this particular simulation, the loads are the temperature of the fin at the base. The spline curved base (rectangular fin) contains the temperature load. The initial temperature was given at that curve. The base of the cylinder is a circular surface. One end of the cylinder is considered as base and the temperature input is given there.

In both cases, the initial temperature is 52 ° C or 325 K.

Loads also contain the convective heat transfer coefficient applied on the surfaces except the base. The convective heat transfer coefficient is taken as  $40 \text{ Wm}^{-2}\text{K}^{-1}$  and the bulk temperature is 30° C.

## 1.8 Nodal Temperature

A Probe in ANSYS can return a maximum or minimum result on associated geometry; however it does not produce an average value of the result. Using ANSYS will allow outputting the temperature distribution in an extremely simple and accurate way. This simulation divides the whole fin surface into different temperature region.

## 1.9 Heat Flow

To analyze heat flow through fins different prospective and figures are available in ANSYS. For more visual clarification different approaches such as heat flow vector sum, heat flow vector plot, and temperature and distance graph are available. The heat flow vector sum figure shows the amount of heat transferred to the surface of the fin. According to that the fin surface has been divided into few regions. The heat flow vector plot fig shows the direction of the heat that is dissipated from the fin surface. To get more precise view of the direction of heat flow from the fin surface zoom option can be applied.

## 1.10 Condenser

Condenser is a device for reducing a gas or vapor to a liquid. In distilling operations, the device in which the vapor is transformed to a liquid state is called a condenser. All condensers operate by removing heat from the gas or vapor; once sufficient heat is eliminated, liquefaction occurs. For some applications, all that is necessary is to pass the gas through a long tube to permit heat to escape into the surrounding air. A heat-conductive metal is commonly used to transport the vapor. A condenser's efficiency is often enhanced by attaching fins to the tubing to accelerate heat removal. Commonly, such condensers employ fans to force air through the fins and carry the heat away. In many cases, large condensers for industrial applications use water or some other liquid in place of air to achieve heat removal.



# CHAPTER TWO

## BACKGROUND

---

### 2.1 Literature Review

Due to the high demand for lightweight, compact and economical fins, the optimization of the size is of great importance. Therefore, fins must be designed to achieve maximum heat removal with minimum material expenditure, taken into account, and also with the ease of manufacturing the fin shape. Other studies have introduced shape modifications by cutting some materials from fins to make cavities, holes, slots, grooves or channels through the fin body to increase the heat transfer area and/or the heat transfer coefficient. One popular heat transfer augmentation technique involves the use of rough interrupted surfaces of different configurations. The surface roughness or interruption aims at promoting surface turbulence that is mainly intended to increase the heat transfer coefficient rather than the surface area. In reference, another experimental study on the forced heat transfer enhancement over a flat surface equipped with square cross-sectional perforated pin fins in a rectangular channel was conducted. The experimental results show that the use of square pin fins with the circular perforations lead to the heat transfer and friction loss characteristics of a horizontal rectangular channel with hollow rectangular profile fins attached over one of its heated surfaces. The study showed a significant enhancement of the heat transfer due to the hollow fins.

### 2.2 Previous Works

The heat flow characteristics were analyzed before in various occasions. For referral few papers can be taken.

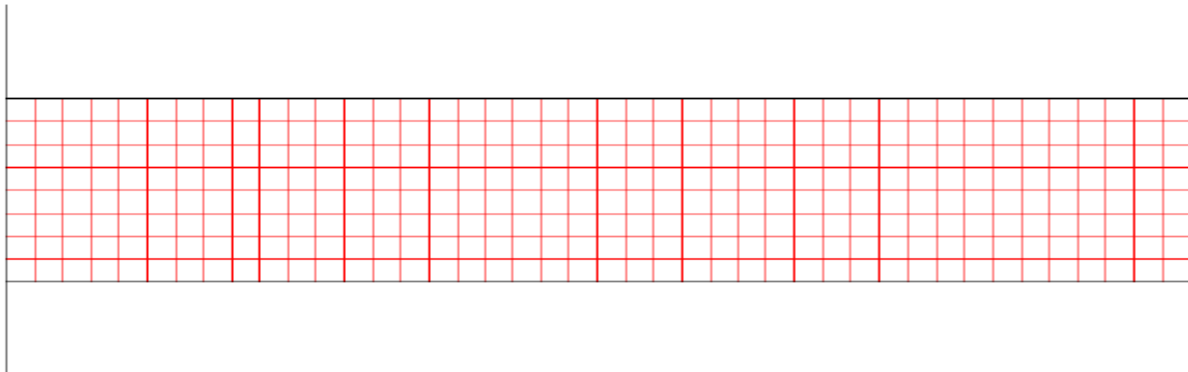
## 2.2 (A): Paper Review 1

In “Heat Transfer through fins “, Fluent Inc. (Flow Lab 1-2), April 12, 2007, rectangular and triangular fins were analyzed and thus efficiency and effectiveness were measured. The solution was solved by FLUENT.

**Mesh:** Mesh was done by considering different refinement factors:

Mesh Density	Refinement Factor
Fine	1
Medium	1.5
Coarse	2

The mesh was done in the rectangular fin. The whole surface area was divided into small grids.



**Fig: 4**

### Graph Plot:

An X-Y plot was then plotted. The graph represents the temperature distribution along the length of the fin. The Paper suggests that a higher slope can be observed near

the base of the fin due to the maximum temperature difference between fin surface and the surrounding medium at the base.

The graph is presented below:

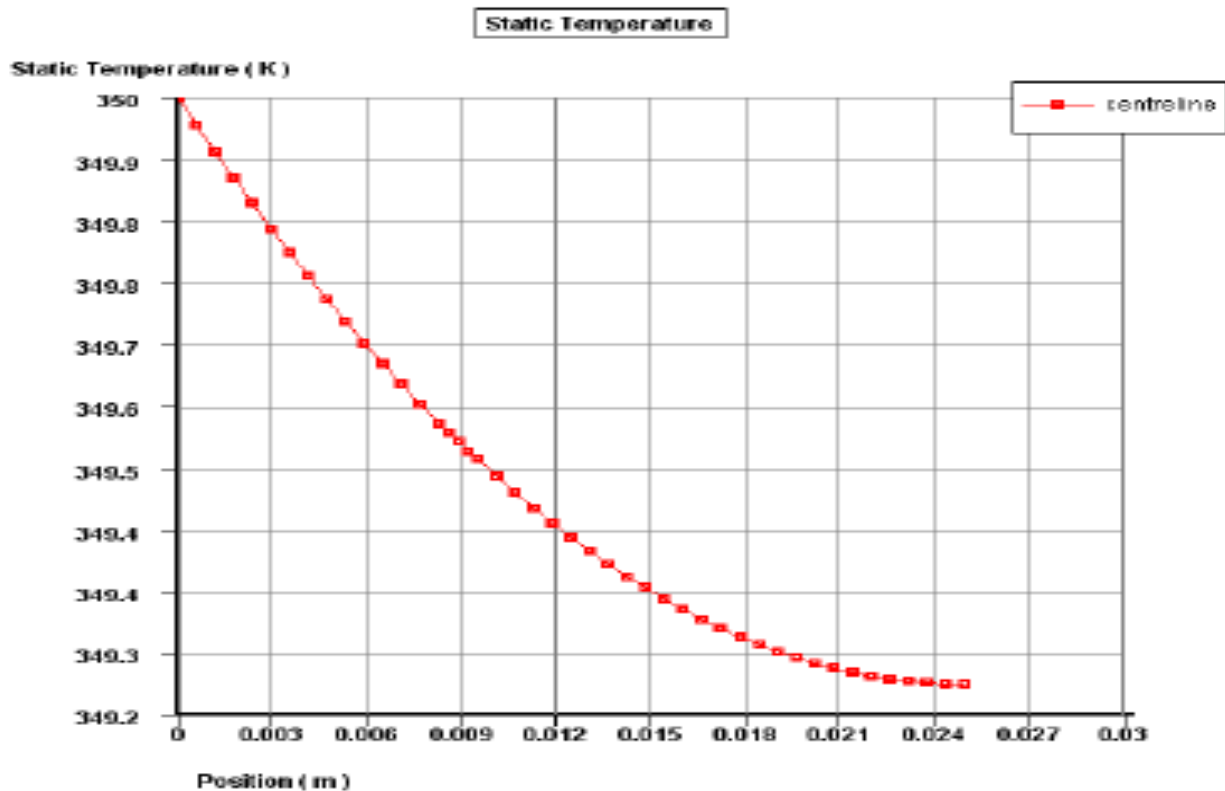
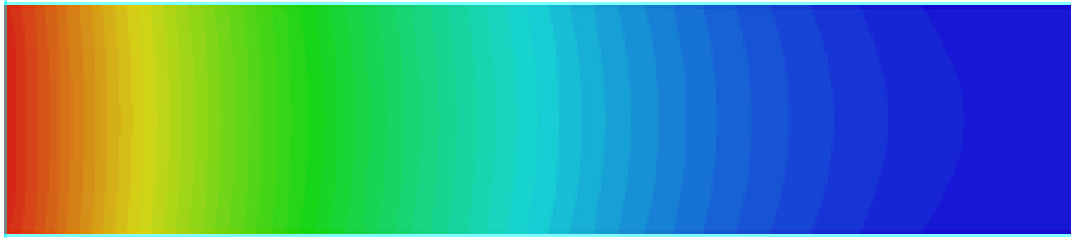


Fig: 5

### Contour Plot:

Then a contour plot was shown for the simplified representation of the temperature regions. The colors in the fin show different temperature regions starting from the base of the fin towards the tip of it.



**Fig: --6**

**Comparison:**

Then fin efficiency and effectiveness were measured and compared.

Shape	Fin efficiency (%)		Fin effectiveness	
	FlowLab	Theory	FlowLab	Theory
Rectangular	95.99	96.14	39.36	39.418
Triangular	94.446	94.82	37.79	37.97

Comparing the results, the theoretical and experimental results are very close and there is a very little difference in efficiency and effectiveness between the two different shaped fins.

**2.2 (B): Paper Review 2**

In “**Finite Element Analysis and Experimental Study of Convective Heat Transfer Augmentation from Horizontal Rectangular Fin by Triangular Perforations**” written by **Kumbhar D.G., Dr. N.K. Sane, Chawan S.T.** has shown an analysis by ANSYS 9.0 where temperature variations differed with the presence and absence of perforations.

The main jest of this paper gives the methodology of explaining the fin characteristics by ANSYS. The temperature v/s distance graph always has been the main correspondence of the heat transfer rate.

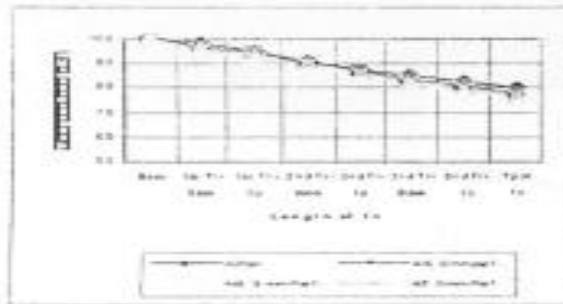


Fig.10: Temperature v/s distance graph for Aluminum fins with and without perforations

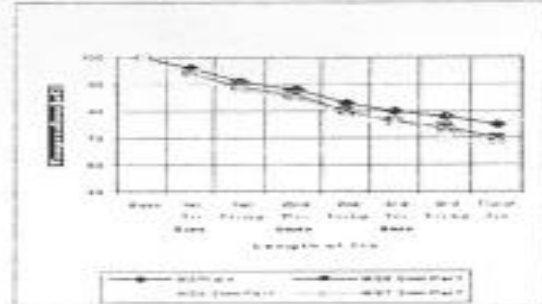


Fig 11: Temperature v/s distance graph for Mild Steel fins with and without perforations

Fig: 7

Fig: 8

Then temperature v/s distance graph was plotted with respect to different materials and dimension of the perforation.

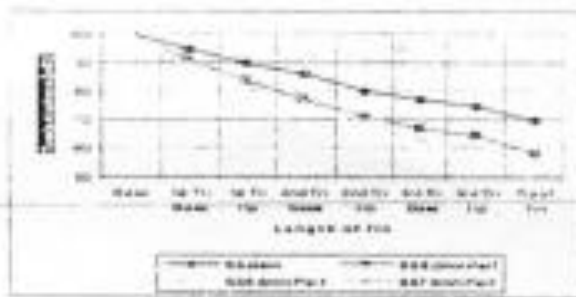


Fig.12. Temperature v/s distance graph for Stainless Steel fins with and without perforations

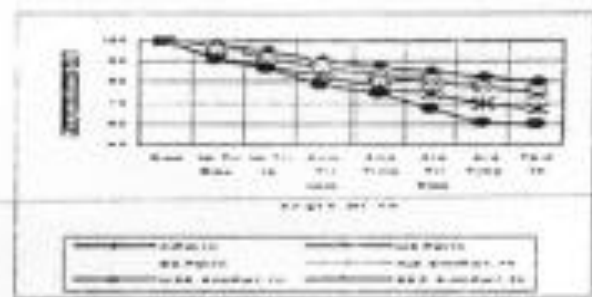


Fig.13. Comparison graph of different material with 8.5mm perforations and without perforations

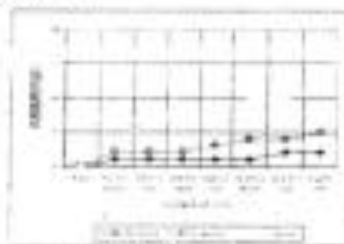


Fig 14-a

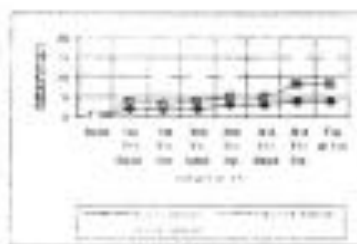


Fig 14-b

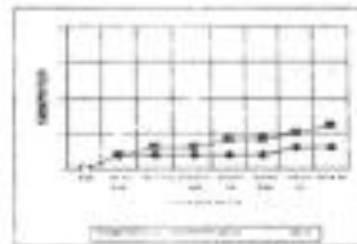


Fig 14-c

Fig 14 Temperature difference and distribution along the length of solid perforated fins

Fig: 9, 10, 11

The graphs plotted here were generated automatically by ANSYS. The command was to take a reference line in the surface of the fin then taking the temperature differences along that line. The comparison becomes easier when multiple graphs are generated for same fin and the graph characteristics differences are visible.

## **2.3 Summary of Previous Works**

The area of this project is relatively new. So the previous works are not that much of an exact help. Hence by the help of these papers a common methodology can be established and followed. The first paper is helpful more to analyze the fin characteristics and introduction of fin analyzing. The second one is helpful for the way of analyzing the fin with ANSYS. The material properties have to be taken as per type of fin to be used. The boundary conditions are very important in this case. The boundary conditions are assumed in some cases ideal and in some cases experimental values available in different lab experiment results.

The meshing of the model plays a vital role in the analysis of the fin. The mesh size and type must meet the requirement of complete structure. In case of rectangular fin the mesh size was very easy to choose. The first paper shows that the meshing is only square. As the structure is very simple so the mesh size did not hamper the result or analysis. The refinement factor is the degree of refinement needed in case of any complex area. The irregular shapes need more refinement than simplified structure.

# CHAPTER THREE

## THESIS WORK

---

### 3.1 Objectives

- ✓ Simulation of rectangular and cylindrical fin.
- ✓ Comparison of experimental and simulation data in case of rectangular fin.
- ✓ Establish a methodology of simulation of rectangular fin.
- ✓ Establish a process of choosing the best suited fin size and shape for any machine.
- ✓ Using the rectangular fin simulation methodology to simulate the cylindrical fin.
- ✓ Simulate the cylindrical fin and compare it with the rectangular fin results.
- ✓ To use the simulation process for other types of fins.
- ✓ To avoid any further experiment to determine the fin shape and size.

### 3.2 Model

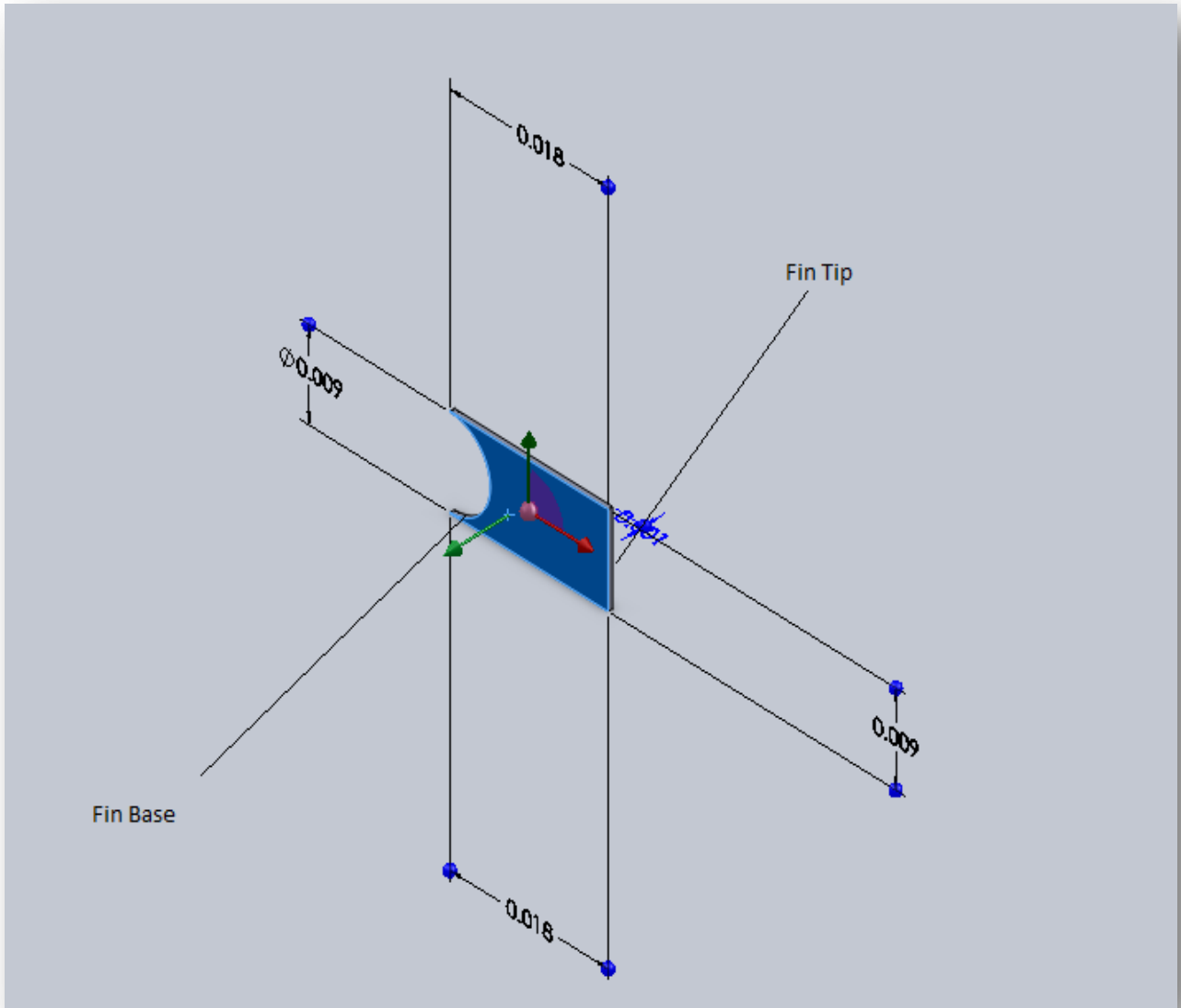
#### 3.2.1 Rectangular fin

Dimension: Length – 0.0177 m

Height—0.009 m

Diameter of the condenser pipe or Radius of the spline curve—0.009m

The rectangular fins are mounted on the condenser pipe vertically. The base of the fin hence makes a curvature as per the radius of the condenser pipe. So the fin base area must be calculated taking considering this condition. All the dimensions are achieved by measuring the fin size with help of slide calipers. The machine is universal air condition testing bench in the laboratory.



**Fig- 12:** 3-D model of rectangular fin with dimension

### 3.2.2 Cylindrical fin

The dimension of the cylindrical fin is achieved by calculation. As the cylindrical part of the simulation is entirely a theoretical approach so for the simplicity of calculation the area of the cylindrical fin is taken equal to the area of the rectangular fin. The area of the rectangular fin is,



$$\begin{aligned}
A &= (\text{height} \times \text{length}) - \text{half circle area} \\
&= (0.009 \times 0.0177) - \left(\frac{1}{2} \pi \times 0.0045^2\right) \\
&= 1.593 \times 10^{-4} - 3.18 \times 10^{-5} \\
&= 1.275 \times 10^{-4} \text{ m}^2
\end{aligned}$$

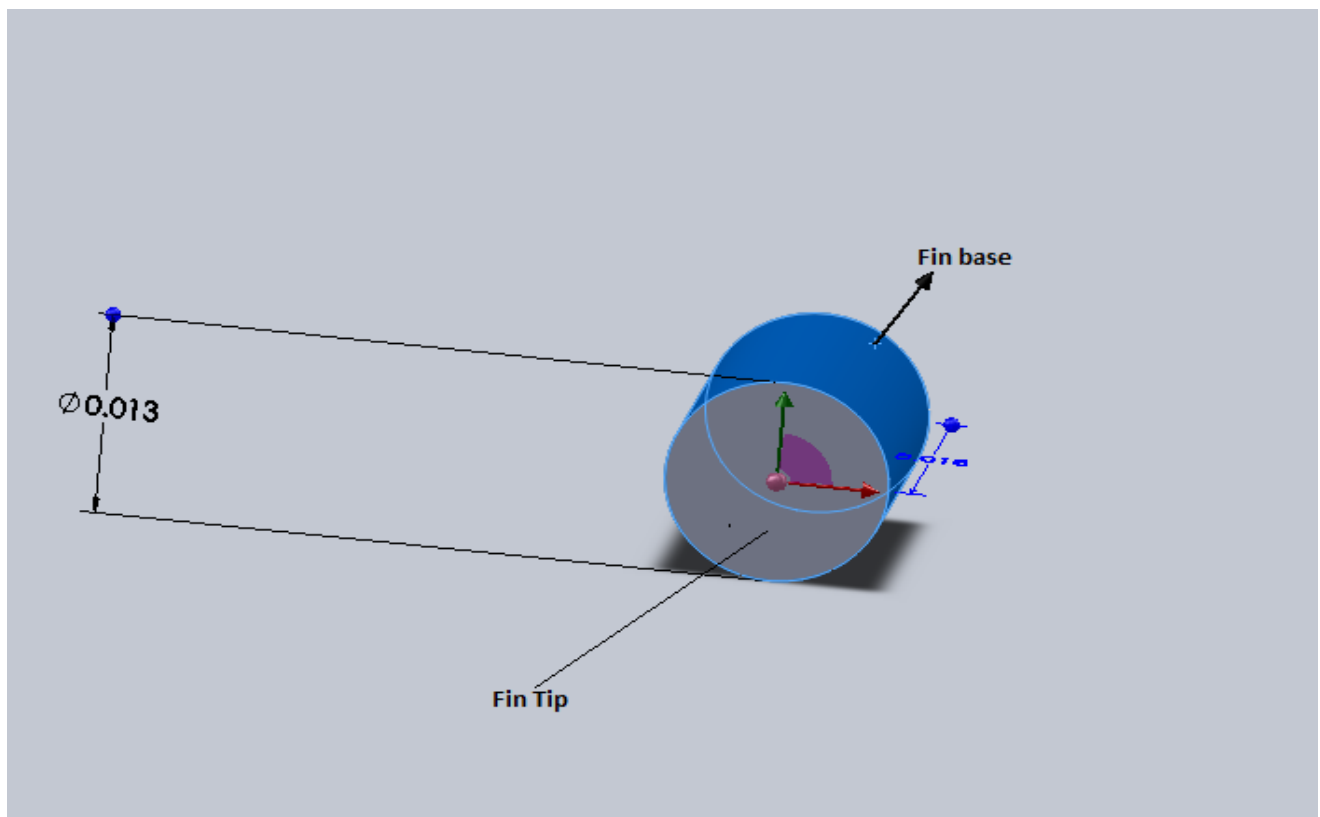
Taking this area equal to the cross sectional area of the cylindrical fin. Keeping the length of the cylinder equal to the length of rectangular fin, the radius is measured.

$$\text{Cross sectional area of the cylinder} = 1.275 \times 10^{-4} \text{ m}^2$$

$$\text{Or, } \pi r^2 = 1.275 \times 10^{-4} \text{ m}^2$$

$$\text{Or, } r = 6.37 \times 10^{-3} \text{ m}$$

So the diameter of the cylinder is 0.013 m. The model is then designed.



**Fig 13:** 3-D model of cylindrical fin with dimension

### 3.3 Boundary Conditions

- ✓ Steady state
- ✓ Uniform cross sectional area
- ✓ No internal heat generation in fin
- ✓ Constant material properties
- ✓ One dimensional conduction
- ✓ Uniform convection across the surface area
- ✓ For this particular condenser the fins are made of **ALUMINIUM**.
- ✓ The inlet (or fin base) temperature: **52° C/325K**
- ✓ The **specific heat** of Aluminium: **910 J/Kg K**
- ✓ The **convective heat transfer coefficient** of air: **40 W/m<sup>2</sup> K**
- ✓ **Thermal Conductivity** of aluminium: **205 W/m-K**
- ✓ **Density** of aluminium: **2700 kg/m<sup>3</sup>**

### 3.4 Solver and Solving Equations

- **Simplified Case:**

$$\frac{d^2T}{dx^2} = - \left( \frac{1}{A_c} \frac{dA_c}{dx} \right) \frac{dT}{dx} + \left( \frac{1}{A_c} \frac{h}{k} \frac{dA_s}{dx} \right) (T - T_\infty)$$

#### Nomenclature:

$A_c$  = Cross sectional area of differential element

$A_s$  = Surface area of differential element

$h$  = Convection coefficient

$k$  = Fourier Constant

$T$  = Temperature of the base

$T_\infty$  = Temperature of the atmosphere

- **Convective tip:**

$$\dot{Q}_{\text{convecting tip}} = \sqrt{hpkA_c}(T_b - T_\infty) \tanh \left( \sqrt{\frac{hp}{kA_c}} L_c \right)$$

**Nomenclature:**

h = Convection coefficient

p = Perimeter

k = Fourier Constant

$T_b$  = Temperature at base

$T_\infty$  = Temperature of atmosphere

$A_c$  = Cross sectional area of differential element

$L_c$  = Effective length =  $L + A_c/2$

L = Actual length of fin

# CHAPTER FOUR

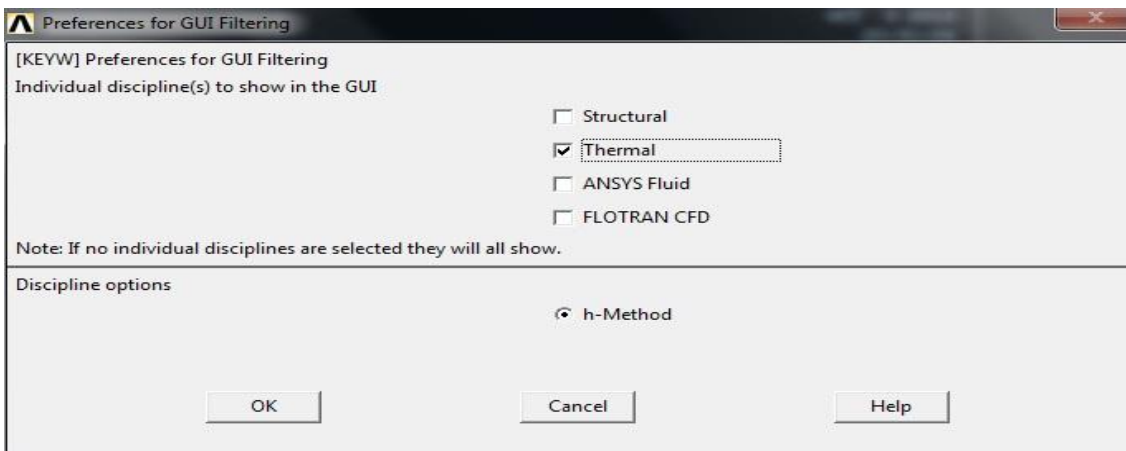
## METHODOLOGY

---

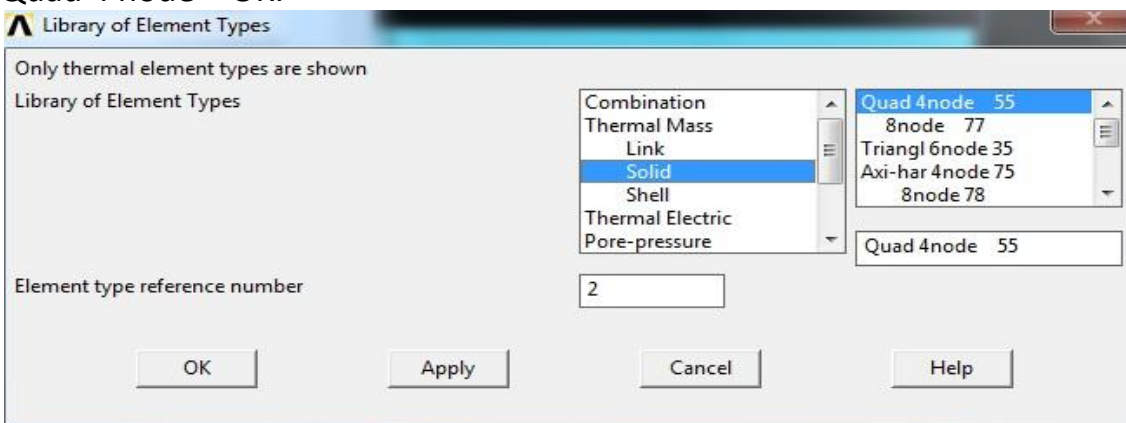
### 4.1 Rectangular Fins

#### 4.1.1 Building Model:

1. First of all the preferences was selected for the simulation. In this case it was thermal.



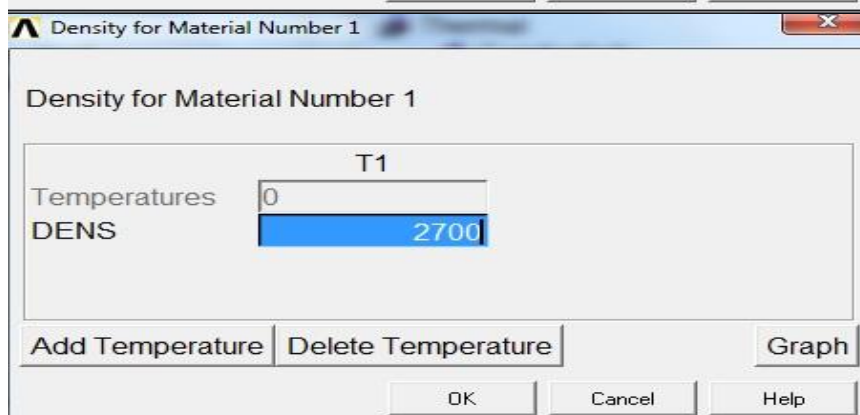
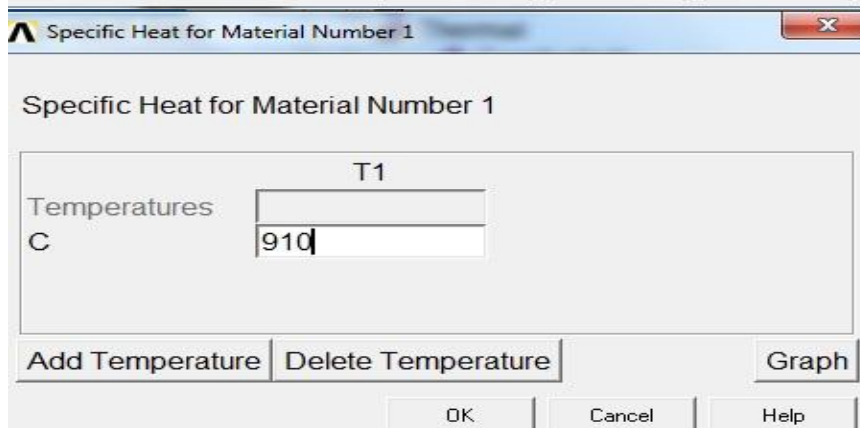
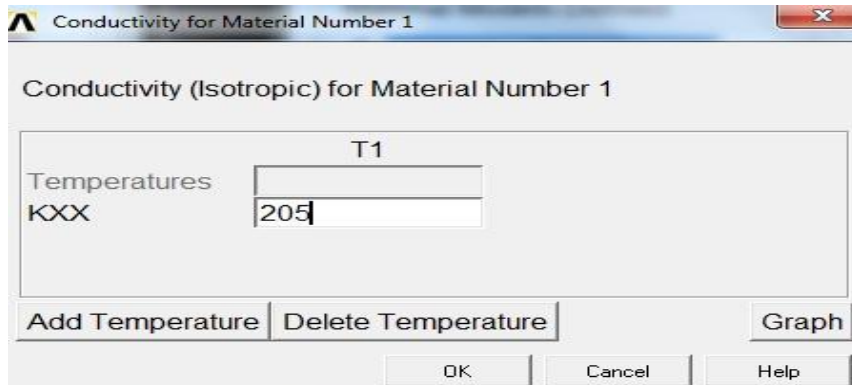
2. Then the proper element type was selected for designing purpose by following order-  
Preprocessor >Element type >Add/Edit/Delete > (Add) >Thermal Mass >Solid > Quad 4 node > OK.



3. As experiments were conducted on Aluminium fins, so proper material properties were given by following order-  
Preprocessor > Material properties > Material models > Thermal > Enter Conductivity > Isotropic > KXX = 205> OK

Preprocessor > Material properties > Material models > Thermal > Enter Specific Heat > C = 910 >OK

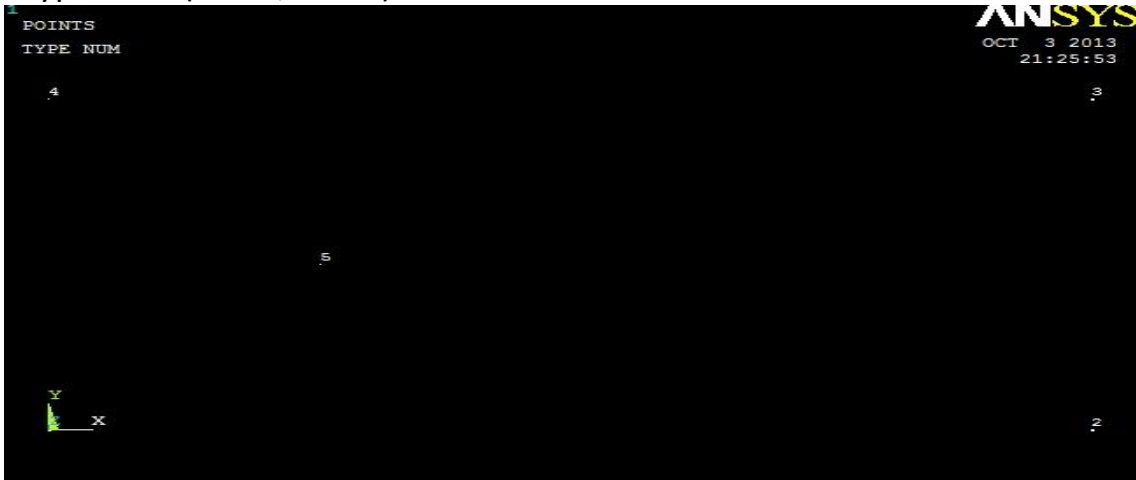
Preprocessor > Material properties > Material models > Thermal > DENS >2700 > OK



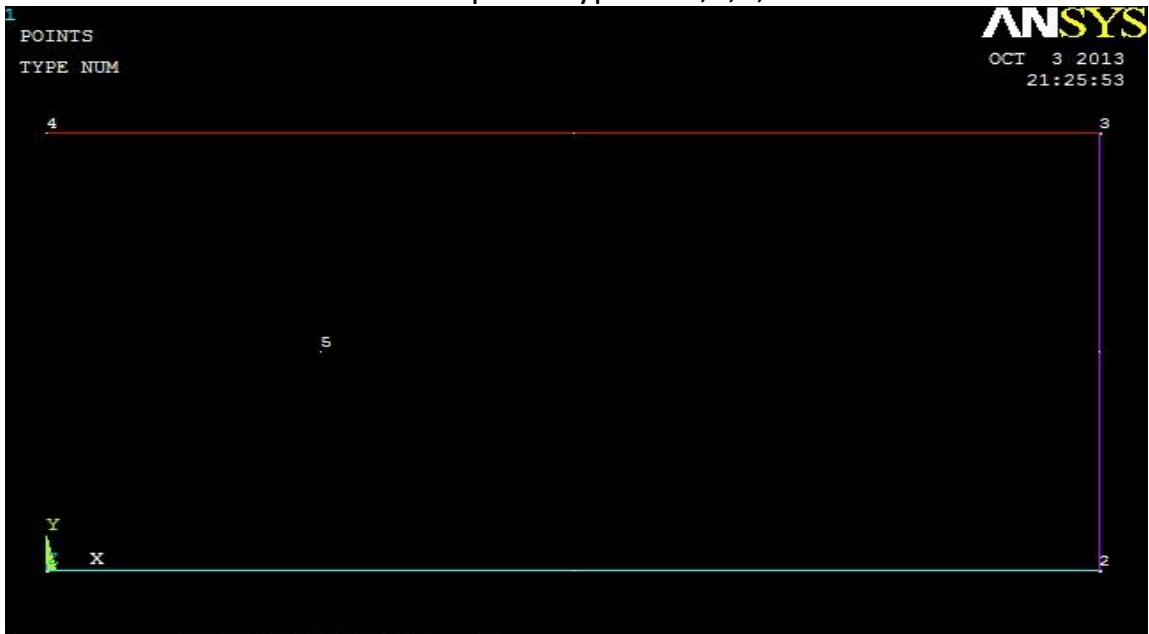
4. Then for creating the model different coordinates of keypoints were put (Preprocessor >Modeling > Create >Keypoints> In Active CS). The coordinates were-  
Keypoint 1- (0, 0)  
Keypoint 2- (.0177, 0)  
Keypoint 3- (.0177, .0092)

Keypoint 4- (0, .0092)

Keypoint 5- (.0046, .0046).

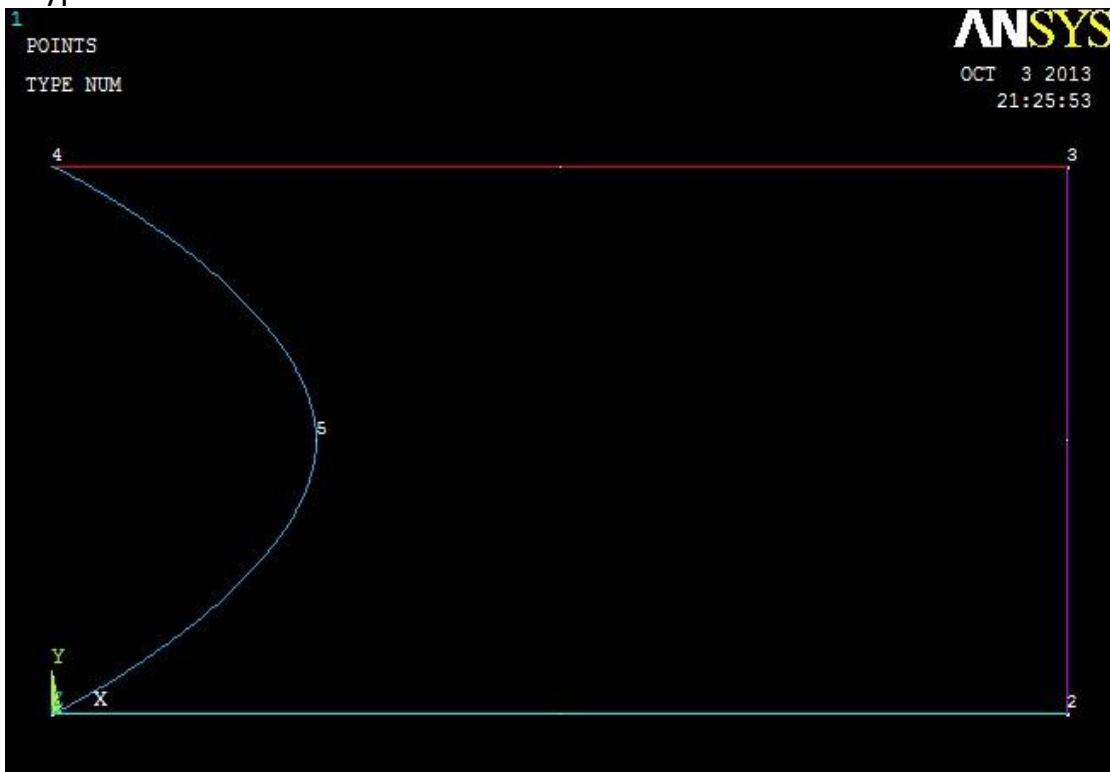


5. Now lines were made by connecting one keypoint with another.  
(Preprocessor >Modeling >Create>Lines >Straight Lines> select keypoints> ok) 3 lines were formed with the help of Keypoint 1,2,3,4.

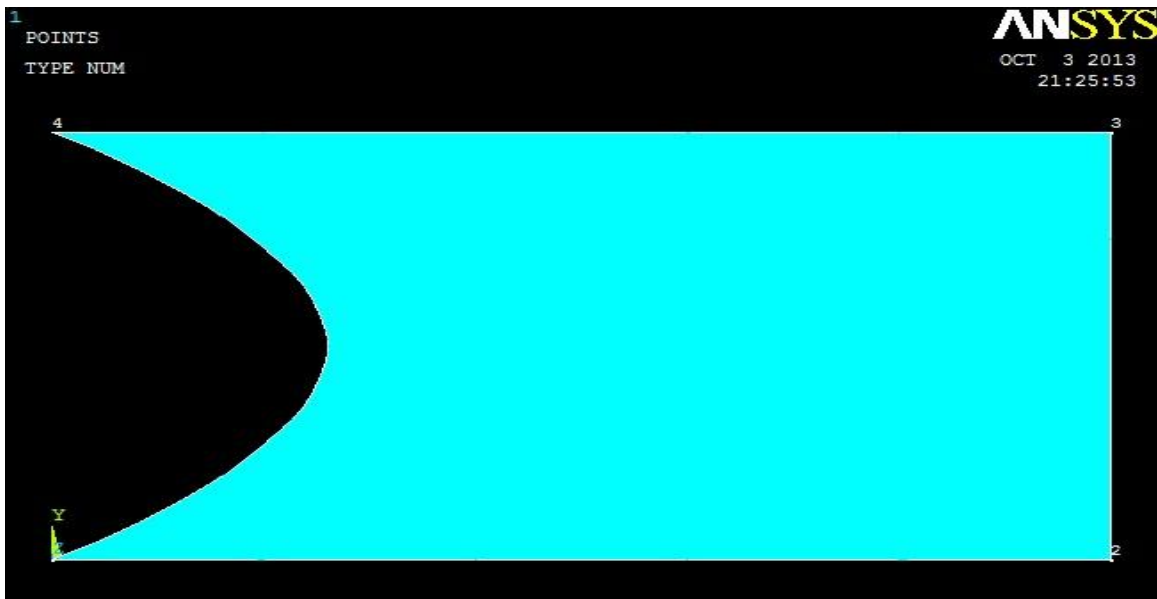


6. Next a spline was created with the help of keypoint1, 4& 5. The steps were -  
Preprocessor >Modeling >Create>Lines > Spline through Keypoints> select

Keypoints > ok.

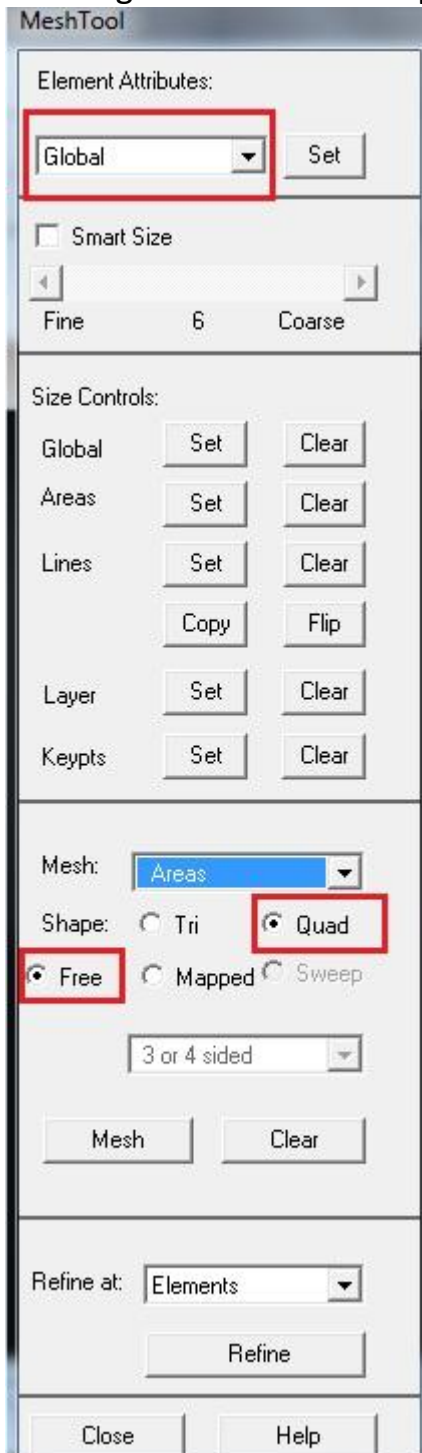


7. Then the full structure was converted into an area. The steps were- Preprocessor >Modeling >Create>Areas >Arbitrary >By Lines > Select all lines>ok.



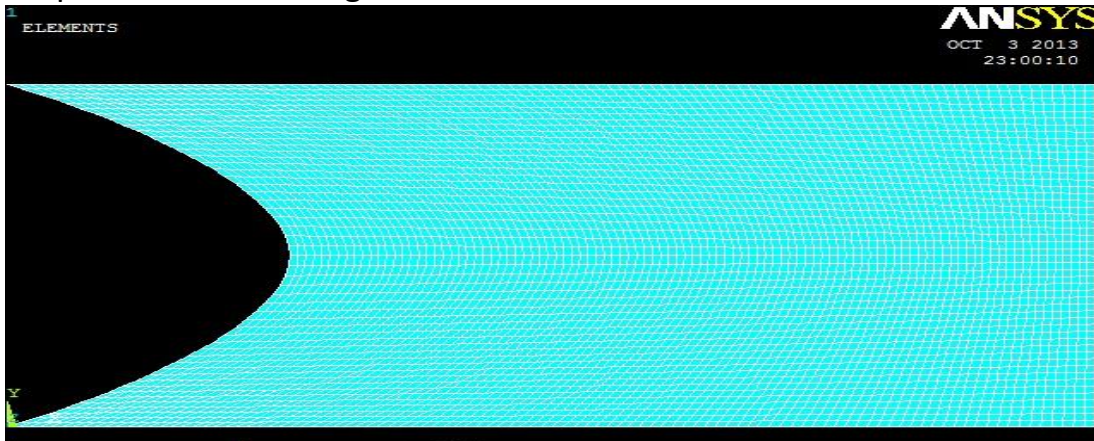
8. Now for meshing, first the length of the fin was divided into 100 no of element divisions and the width of the fin was divided into 50 no of element divisions. Steps-Preprocessor >Meshing >Mesh Tool >Size controls-Lines-set.

9. Global mesh attributes and quad free meshing options were selected for meshing in the mesh tool options.





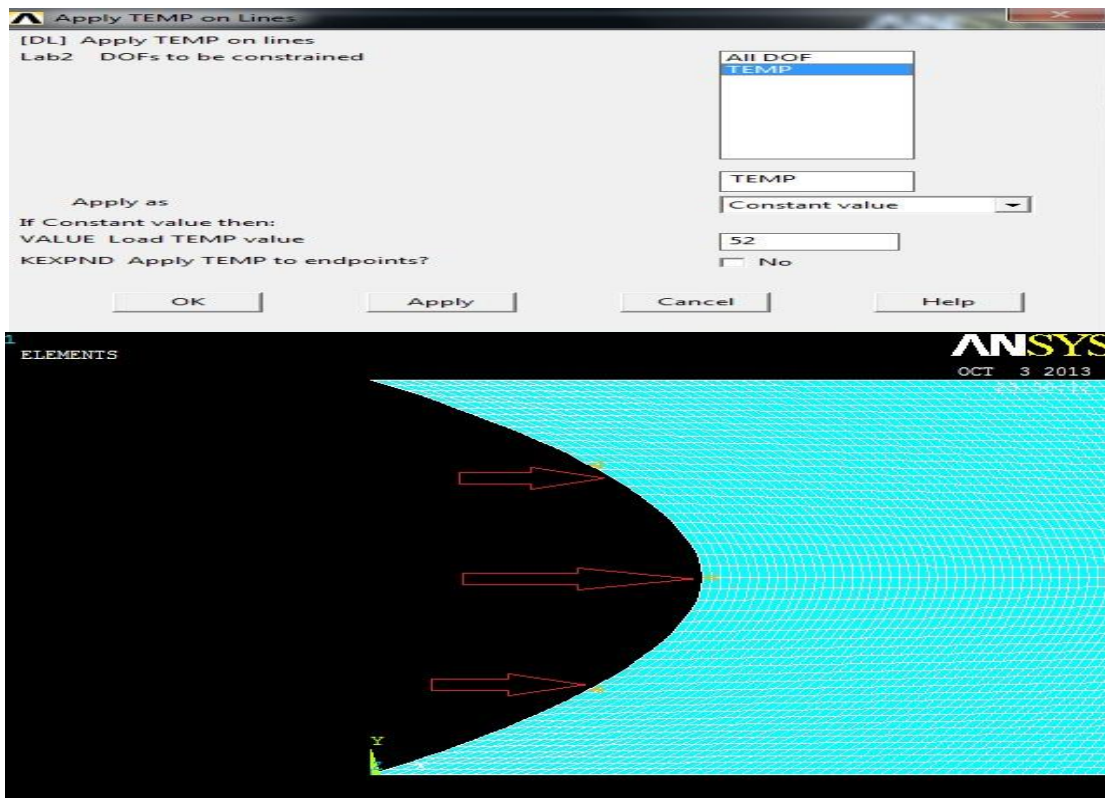
10. Then the meshing was done by selecting the fin area. Steps- Preprocessor > Meshing > Mesh Tool > Mesh > Pick the whole area > ok.



#### **4.1.2 Defining Loads & solve:**

11. In the next stage loads were defined on the fin. For giving fin base temperature the spline curve was selected and the initial base temperature (52°C or 325K) was given. steps were-

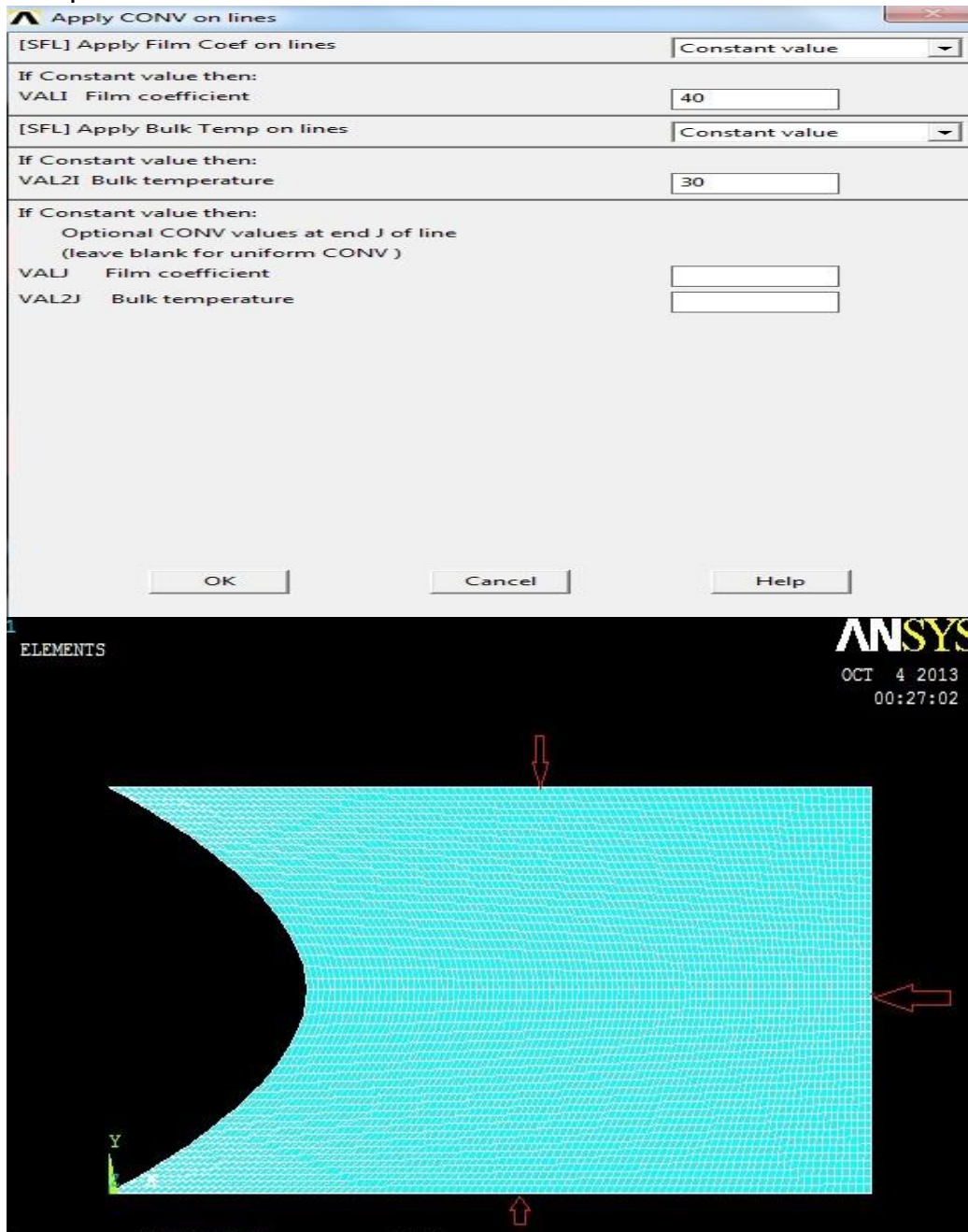
Preferences > Preprocessor > Loads > Thermal > Temperature > On Lines > Select the spline curve > ok > Load TEMP value = 325 > o



12. Heat was transferred by convection through all planes of the fin. So convective heat transfer coefficient & ambient temperature was given.

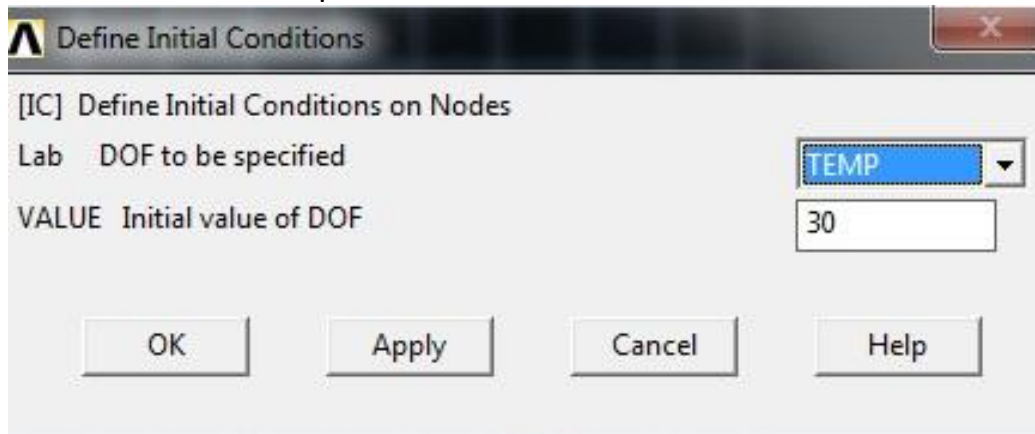
Steps were-

Preferences > Preprocessor > Loads > Define Loads > Thermal > Convection > on lines > select all planes except base > ok > Film Coefficient = 40, Bulk Temperature = 30 > ok.

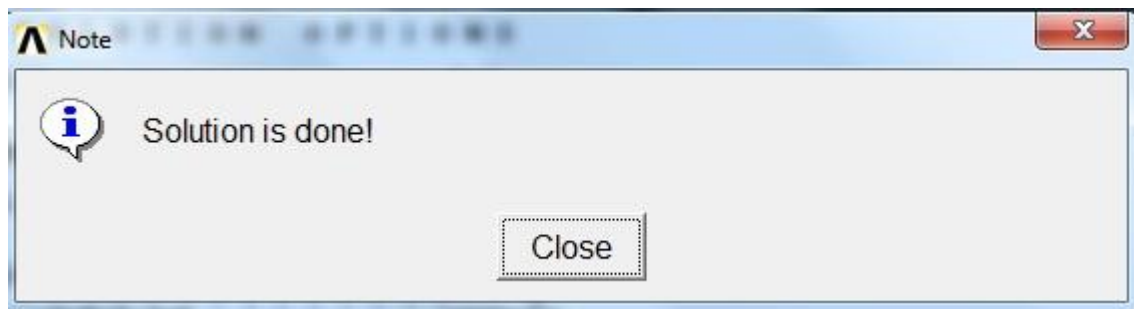


13. In the next stage the initial condition was defined for fins. Steps were-

Preferences> Preprocessor > Loads >Define Loads > Initial Condit'n>Define > Pick All >DOF to be specified=TEMP > Initial value of DOF=30 > OK.



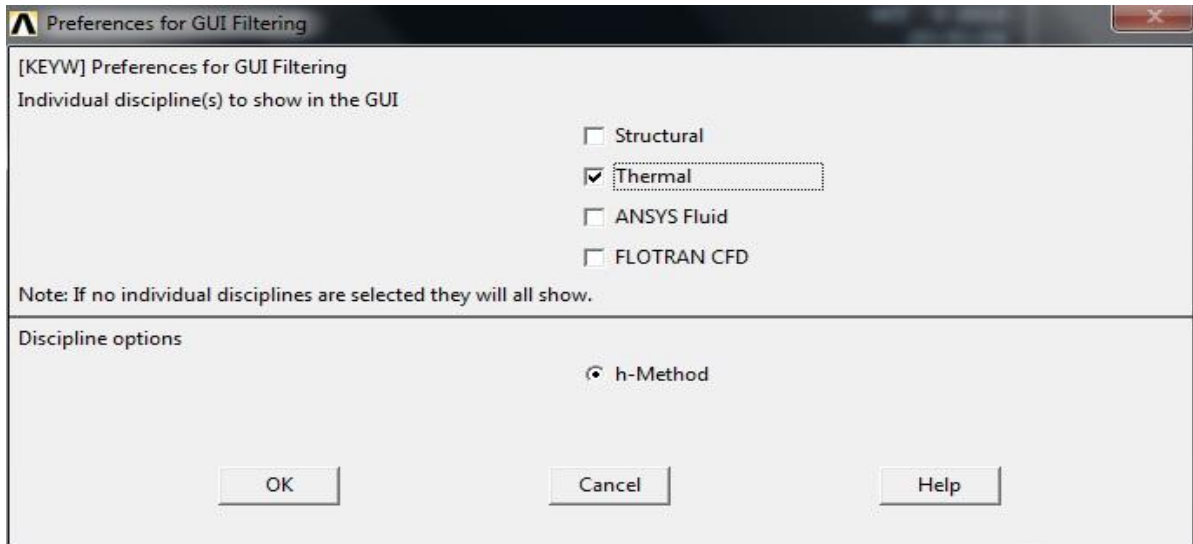
14. In the end with the help of ANSYS solver the solution was done. Steps- Preferences >Solution > Solve > Current LS > OK.



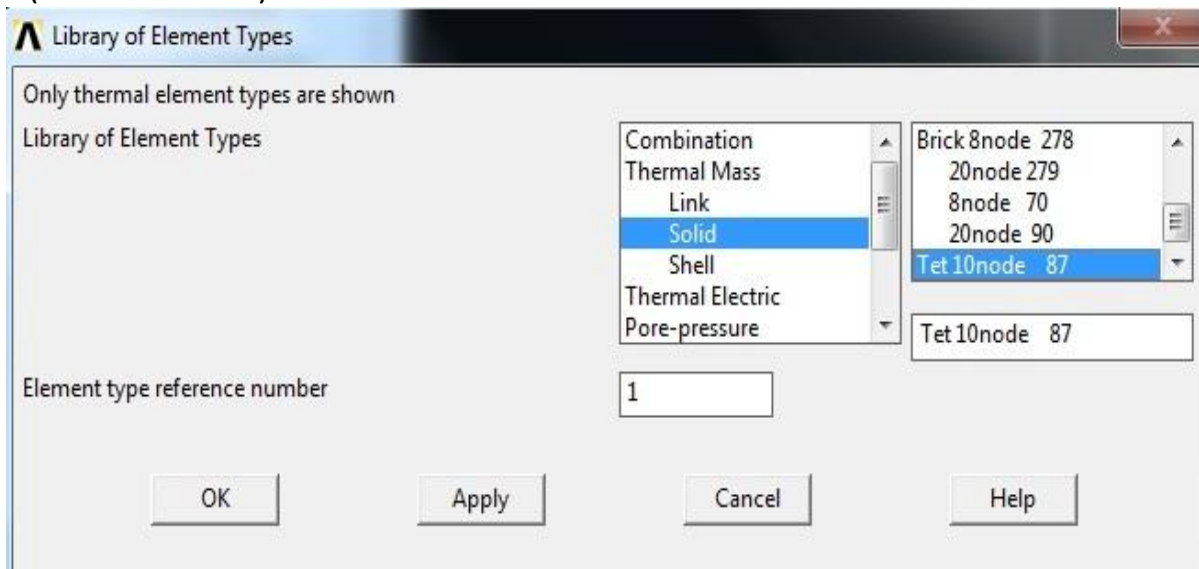
## 4.2 Cylindrical Fins

### 4.2.1 Building Model:

1. First of all the preferences was selected for the simulation. In this case it was thermal.



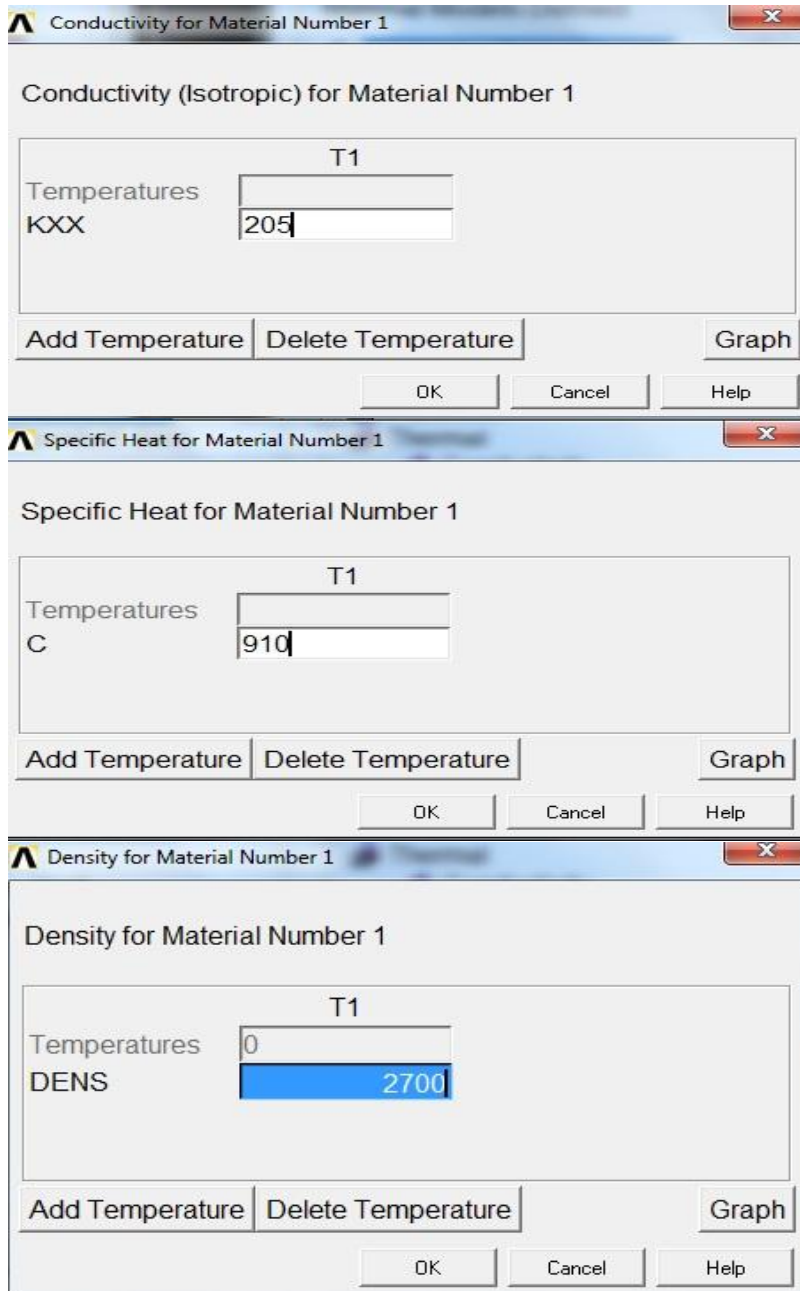
2. Then the proper element type was selected for designing purpose by following order-  
Preprocessor >Element type > Add/Edit/Delete > (Add) > Thermal Mass >Solid > (Tet 10node 87) > OK.



3. As experiments were conducted on Aluminium fins, so proper material properties were selected. Steps –  
Preprocessor > Material properties > Material models > Thermal > Enter Conductivity > Isotropic > KXX = 205 > OK

Preprocessor > Material properties > Material models > Thermal > Enter Specific Heat > C = 910 >OK

Preprocessor > Material properties > Material models > Thermal > DENS >2700 >OK.

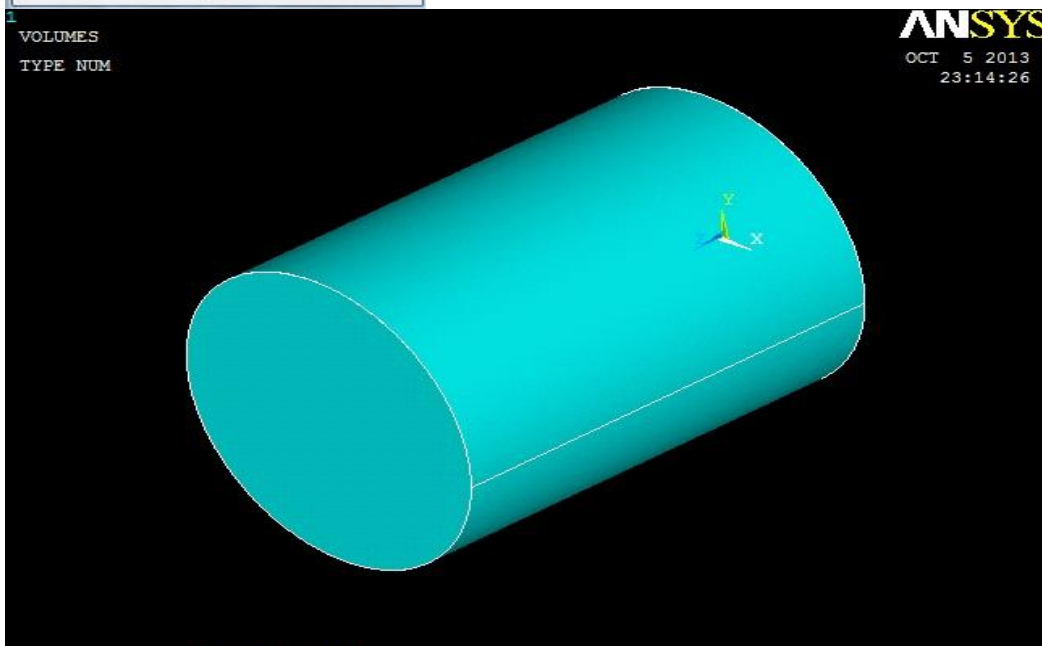
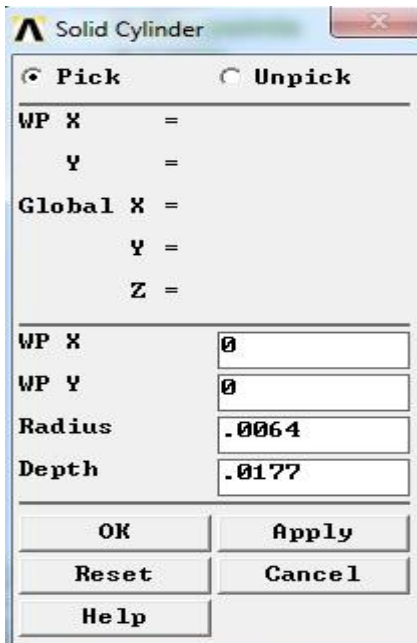


4. For creating cylinder following steps were taken-
- Preprocessor > Modeling > Create > Volumes > Cylinder > Solid cylinder > WP X = 0,
  - WP Y = 0,
  - Radius = 0.0064,

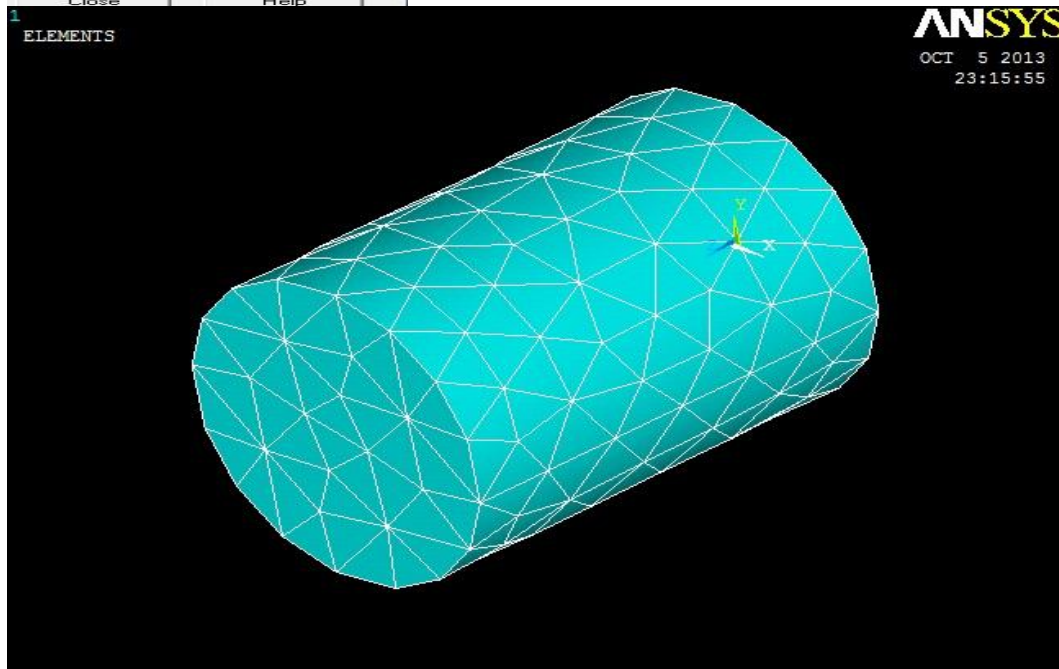
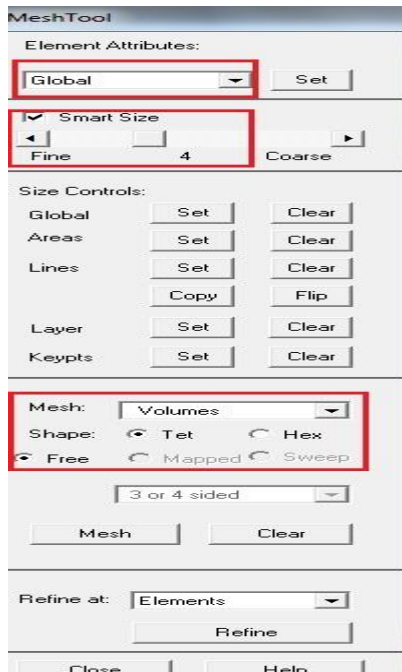


Depth = 0.0177 > ok.

A solid cylinder was created whose cross sectional area was equal to the cross sectional area of the rectangular fin.

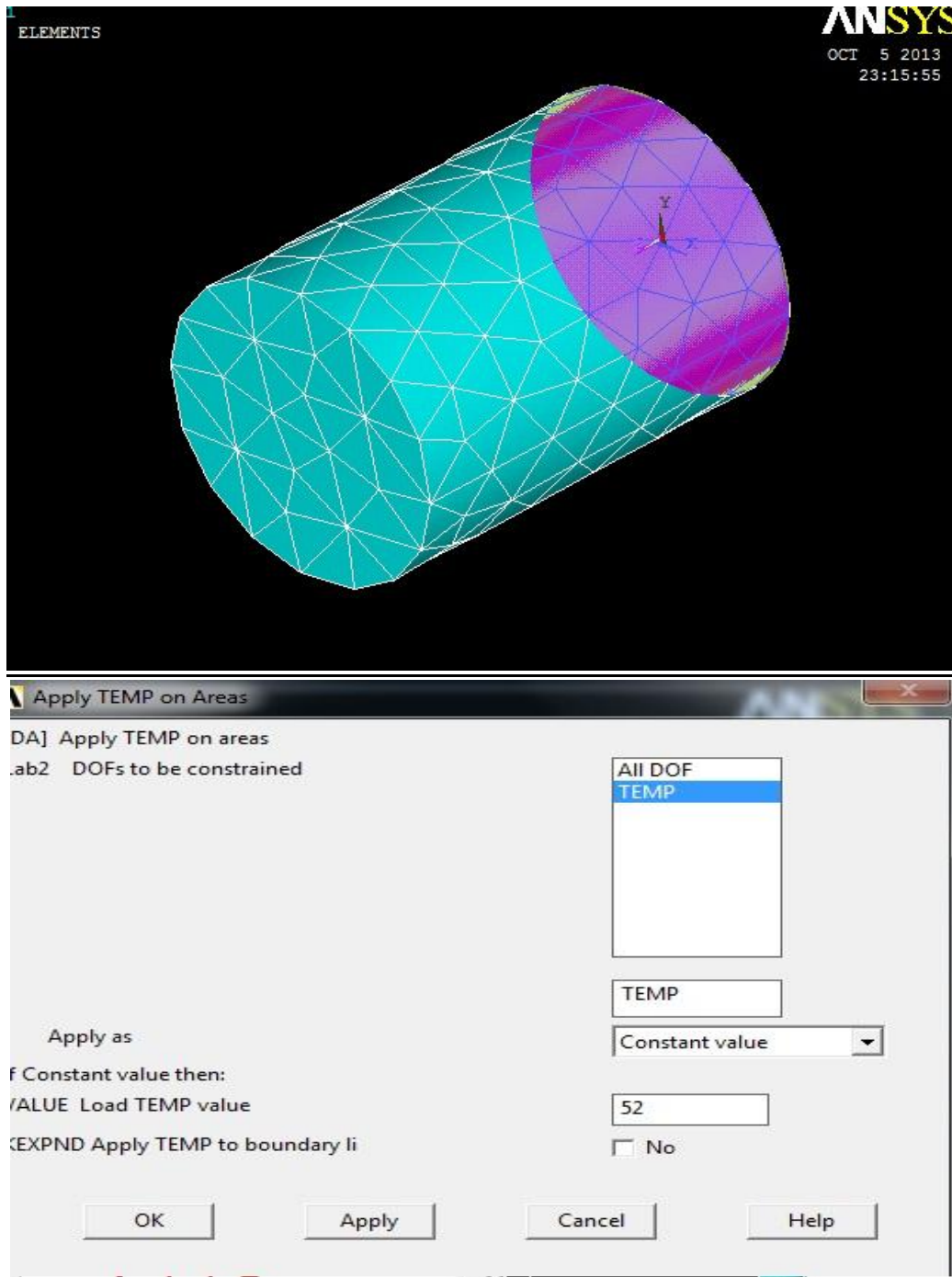


5. Now for meshing the smart size option was turned on from mesh tool options and smart size 4 was chosen. Then using free volumetric tetra mesh option the mesh operation was done. Steps-  
Preprocessor > Meshing > Mesh tool > Turn on smart size to 4 > Mesh > Volumes > Mesh > (Pick the body) > OK.



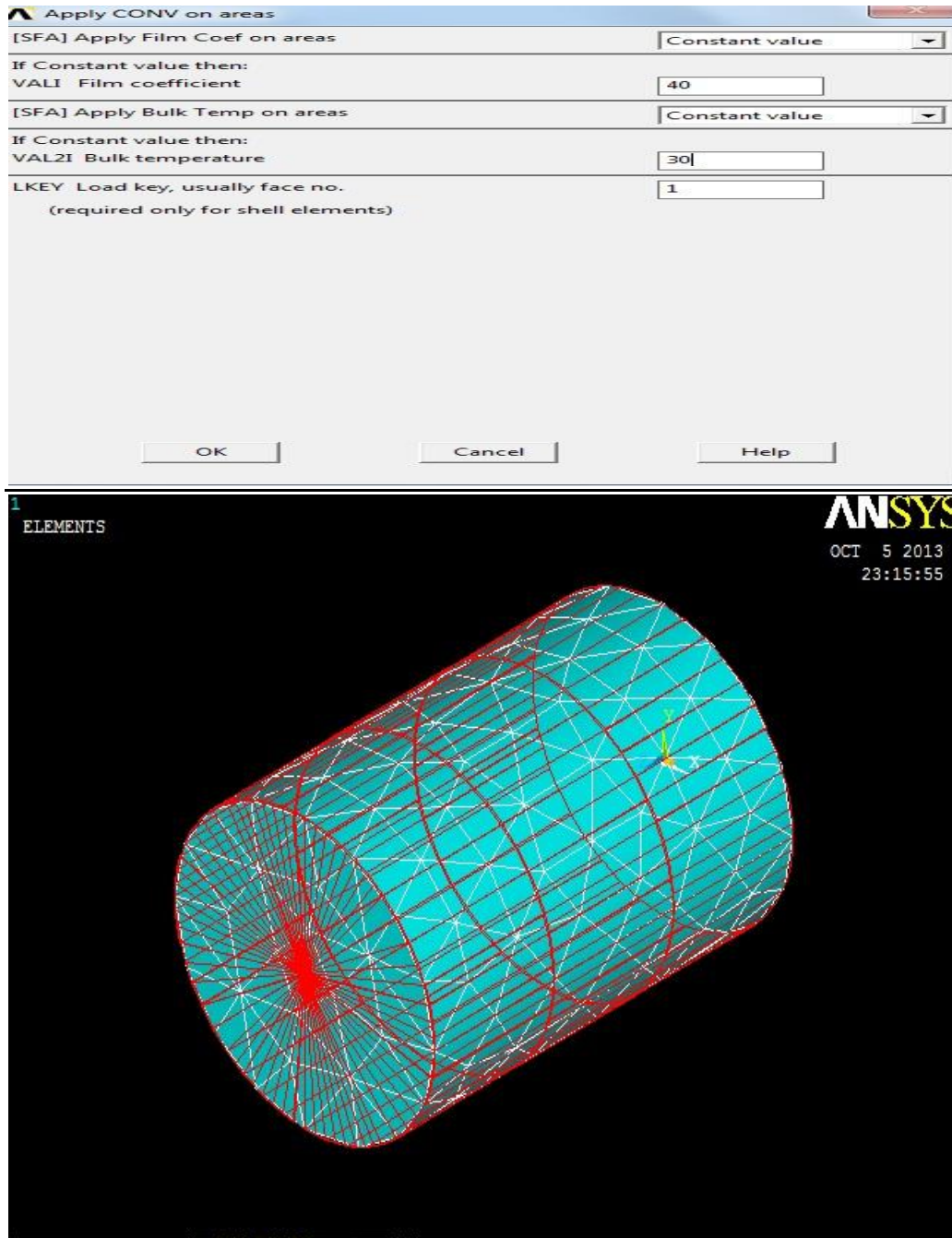
#### **4.2.2 Defining loads and solves:**

6. Now for giving thermal temperature the fin base was selected and the initial base temperature was given. Steps-  
 Preferences> Preprocessor > Loads > Thermal > Temperature >On Areas >Pick the area >OK > DOF to be constrained=TEMP >Load TEMP value=325 > OK.

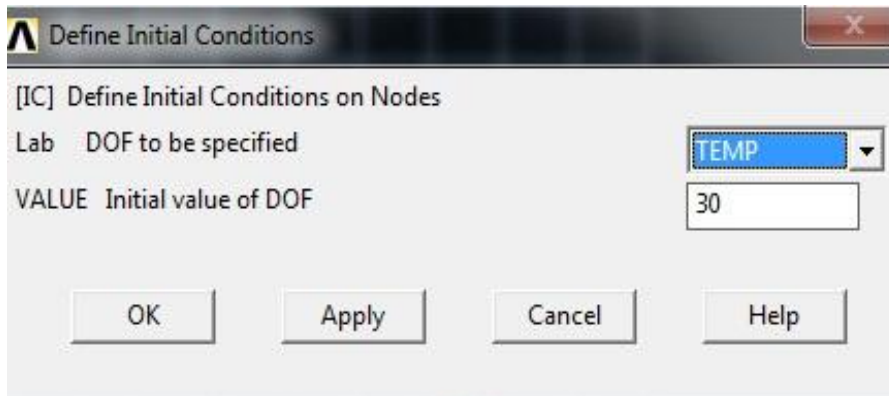


- Heat was transferred by convection by all areas except fin base area. So choosing appropriate areas convective heat transfer coefficient and the ambient temperature was given. Steps-  
 Preferences> Preprocessor > Loads > Define Loads > Thermal > Convection > On areas > Pick all areas except fin base > OK > Film coefficient=40 > Bulk Temperature=30 > OK.

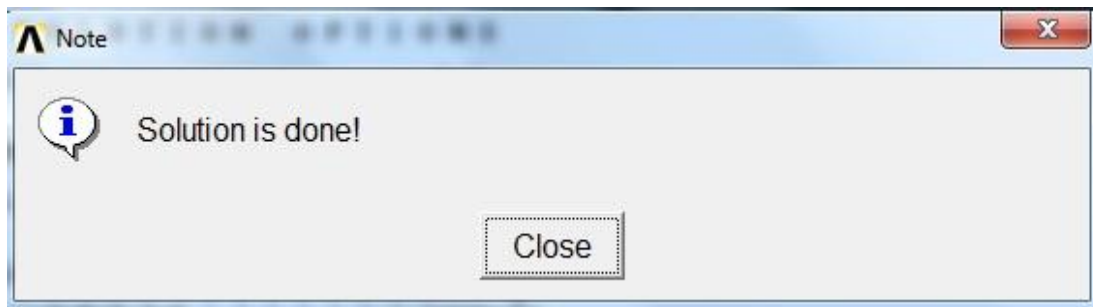




8. In the next stage the initial condition was defined for fins. Steps were- Preferences> Preprocessor > Loads >Define Loads > Initial Condit'n>Define > Pick All >DOF to be specified=TEMP > Initial value of DOF=30 > OK. It was the initial temperature of the fin that's why entire fin elements were selected.



9. At the last stage with the help of ANSYS solver the solution for the operation was done. Steps-  
Preferences > Solution > Solve > Current LS > OK.



# CHAPTER FIVE

## RESULTS

---

### 5.1 Available Solutions

Result is analyzed in different solution manners. Here in this thesis few solutions are considered, they are:

1. Nodal Temperature analysis (gives the temperature at different points of the surface)
2. Heat Flow Analysis (gives the amount of heat flow with respect to distance)
3. Thermal Gradient Vector Plot (gives the thermal gradient with respect to distance)
4. Thermal Gradient Vector Sum (gives the thermal gradient sum with respect to distance)
5. Thermal Flux Vector Plot (gives thermal flux plot characteristics)
6. Thermal Flux Vector Sum (gives thermal flux sum characteristics)
7. Volumetric Element Solution (gives the temperature difference summation according to volume)
8. Temperature Vs. Distance graph (a graph plotted for temperature with respect to distance)
9. Distance Vs. Thermal Flux graph ( a graph plotted for thermal flux with respect to distance)
10. Distance Vs. Thermal Gradient graph (a graph plotted for thermal gradient with respect to distance)
11. Thermal flux vs. Temperature graph
12. Thermal Gradient vs. Temperature Graph
13. Thermal gradient vs. Thermal flux Graph

## 5.2 Rectangular Fin:

### 5.2.1 Nodal temperature:

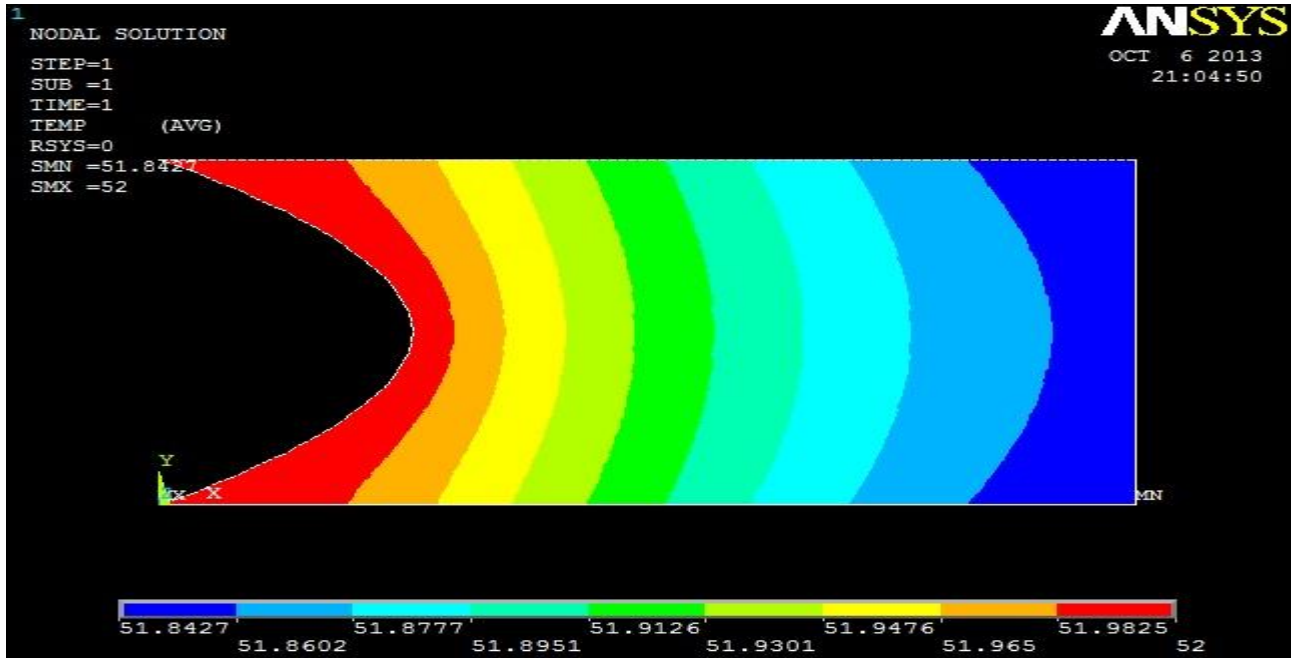


Fig: 14

### 5.2.2 Heat Flow

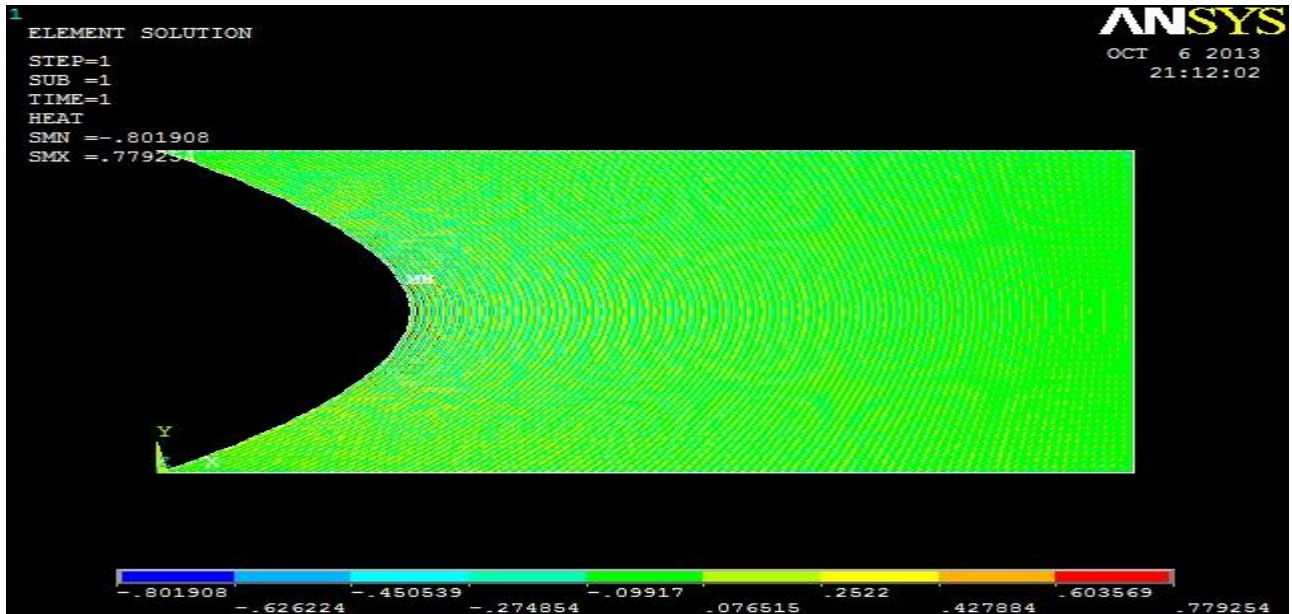


Fig: 15



### 5.2.3 Thermal Gradient Vector Plot

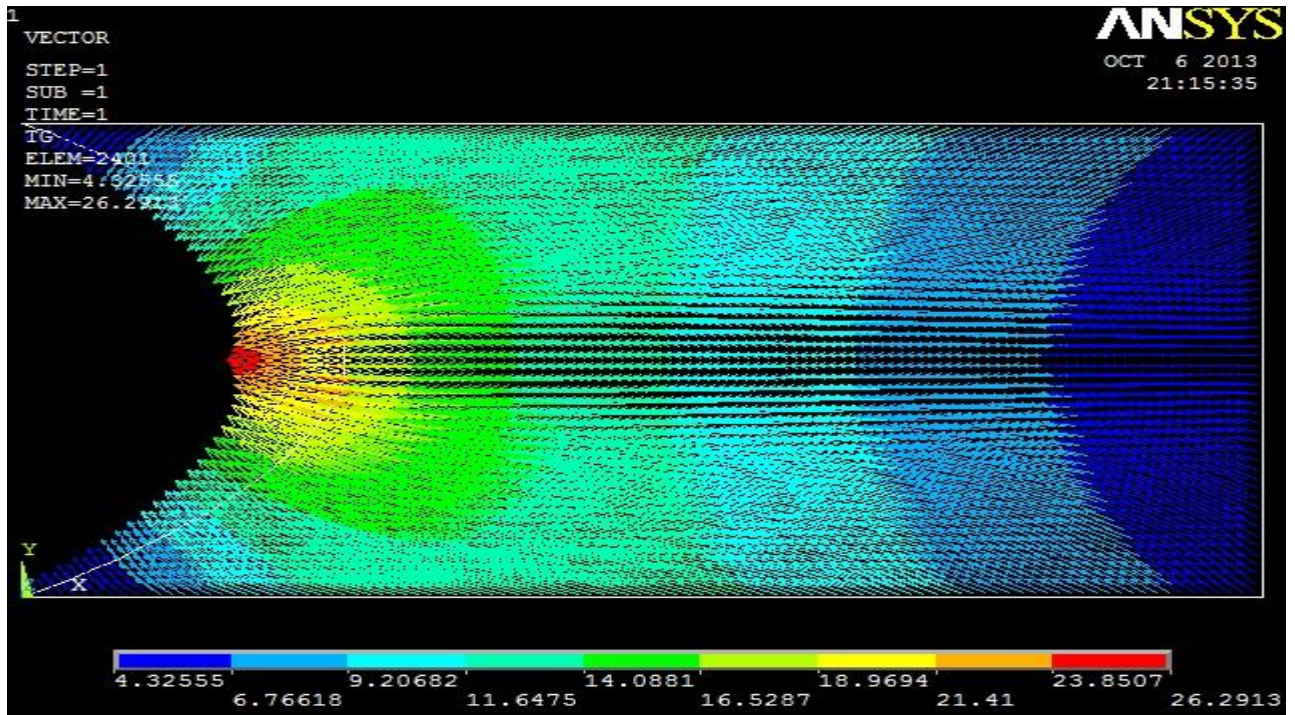
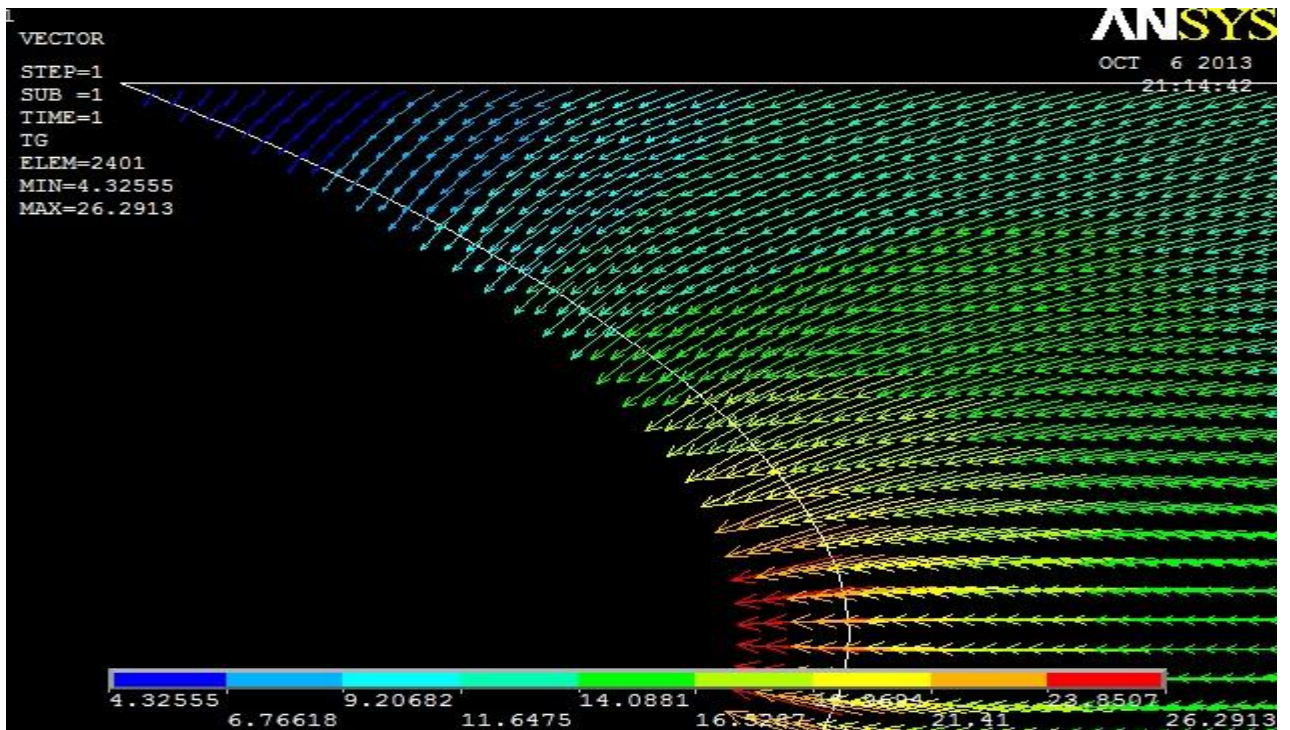


Fig: 16

Zoomed



## 5.2.4 Thermal Gradient Vector Sum

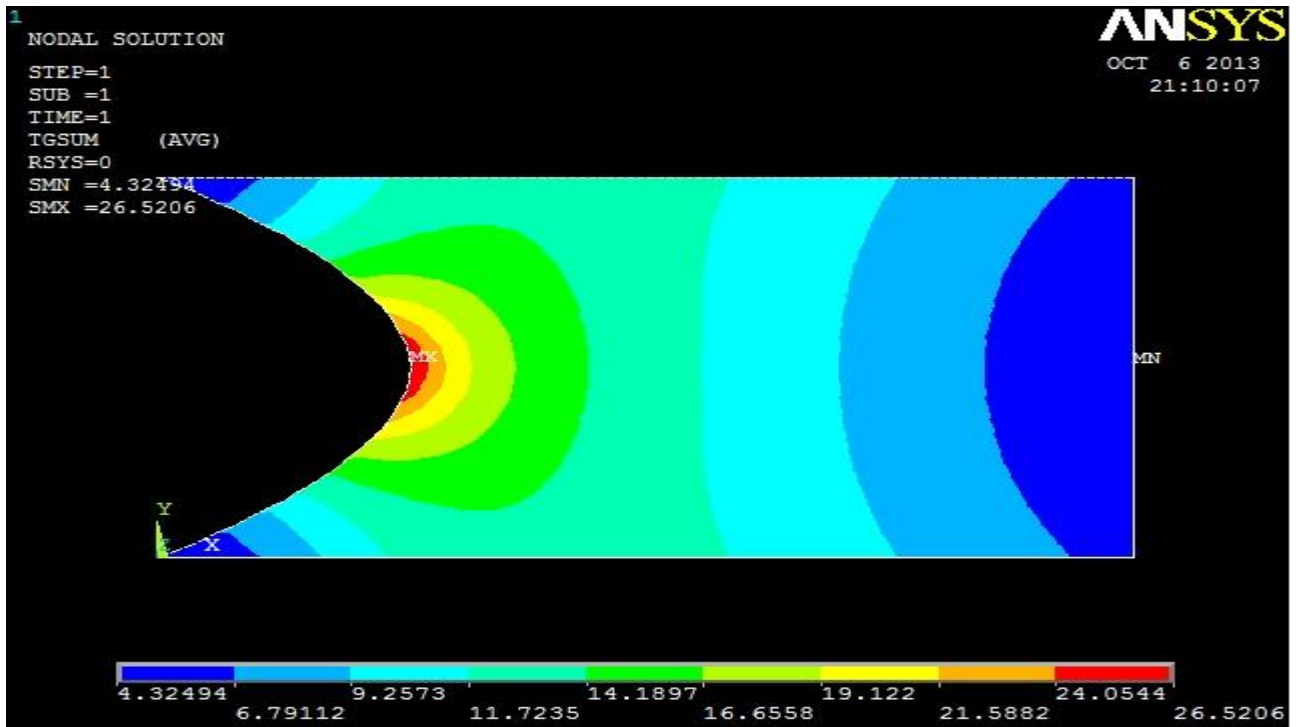


Fig: 17

## 5.2.5 Thermal Flux Vector Plot

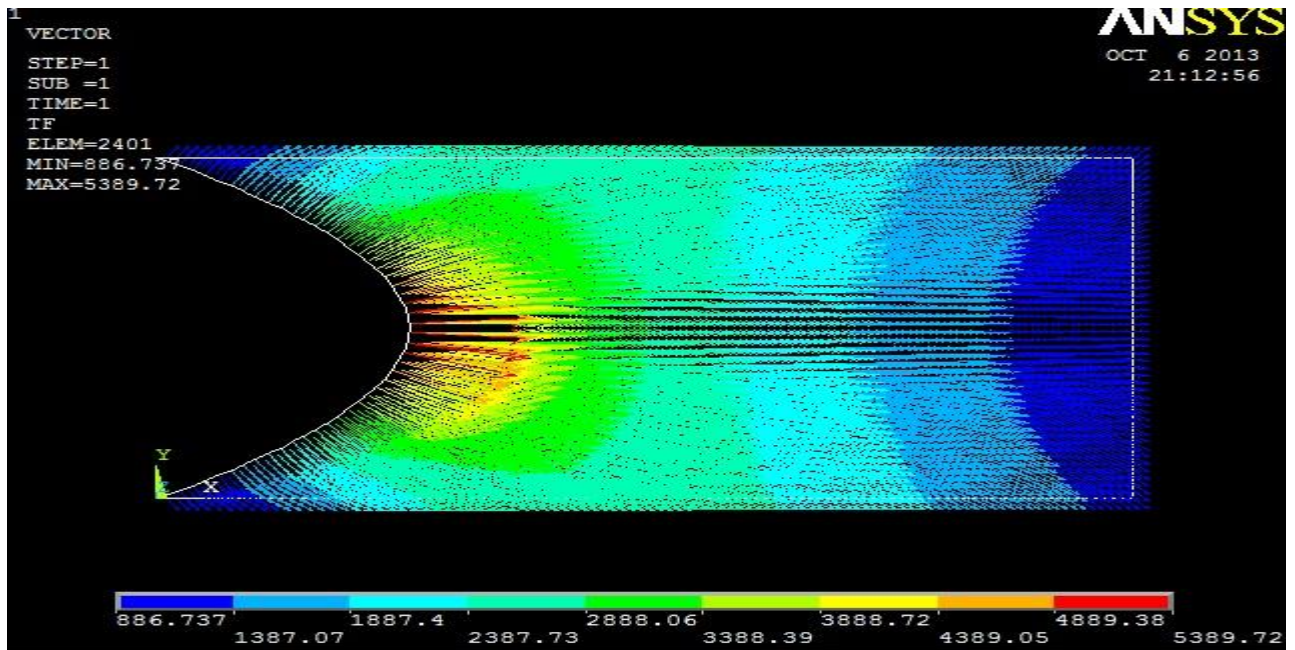
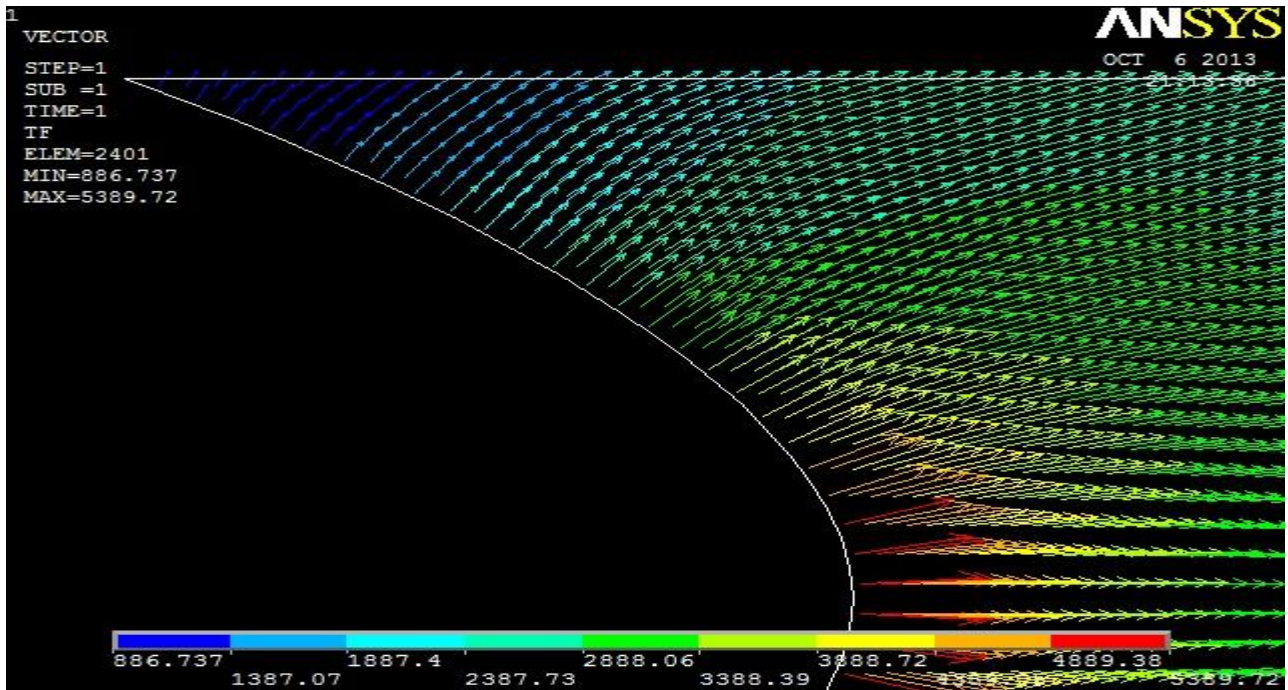


Fig: 18



## Zoomed



## 5.2.6 Thermal Flux Vector Sum

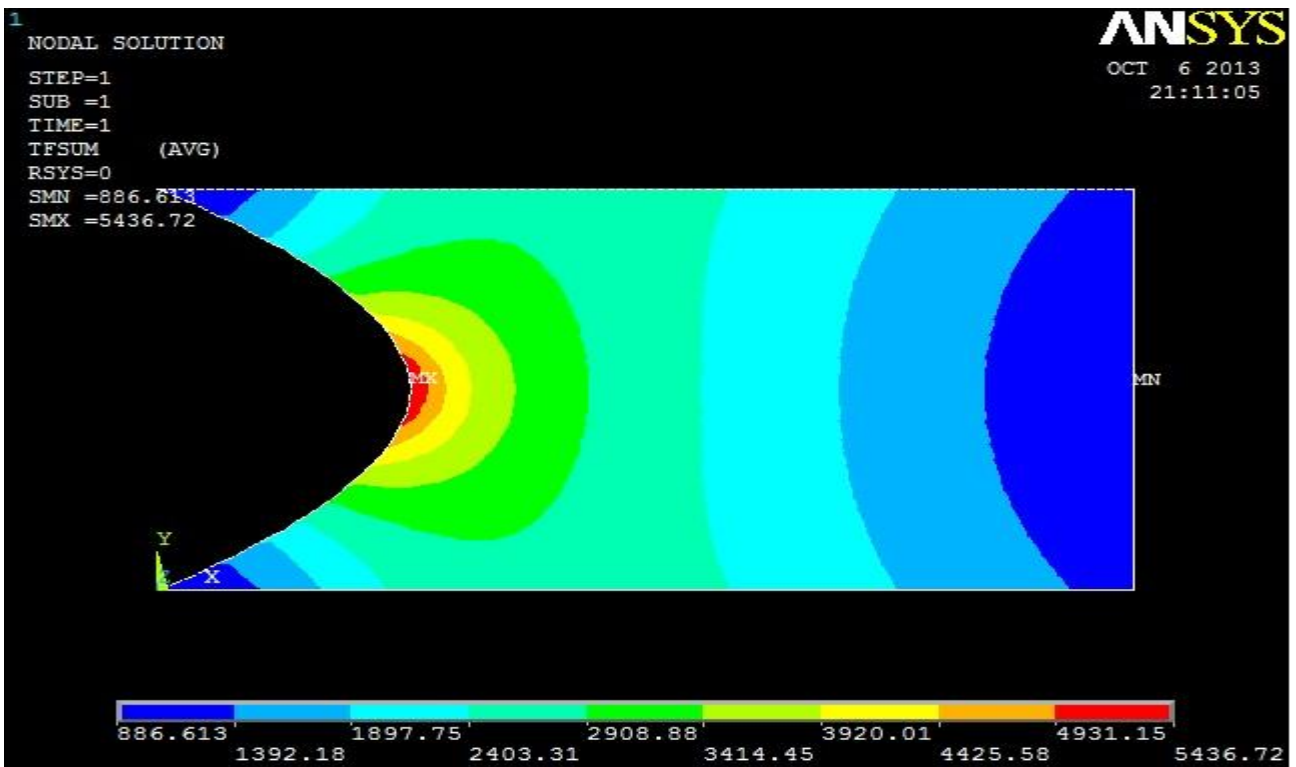


Fig: 19

## 5.2.7 Volumetric Element Solution

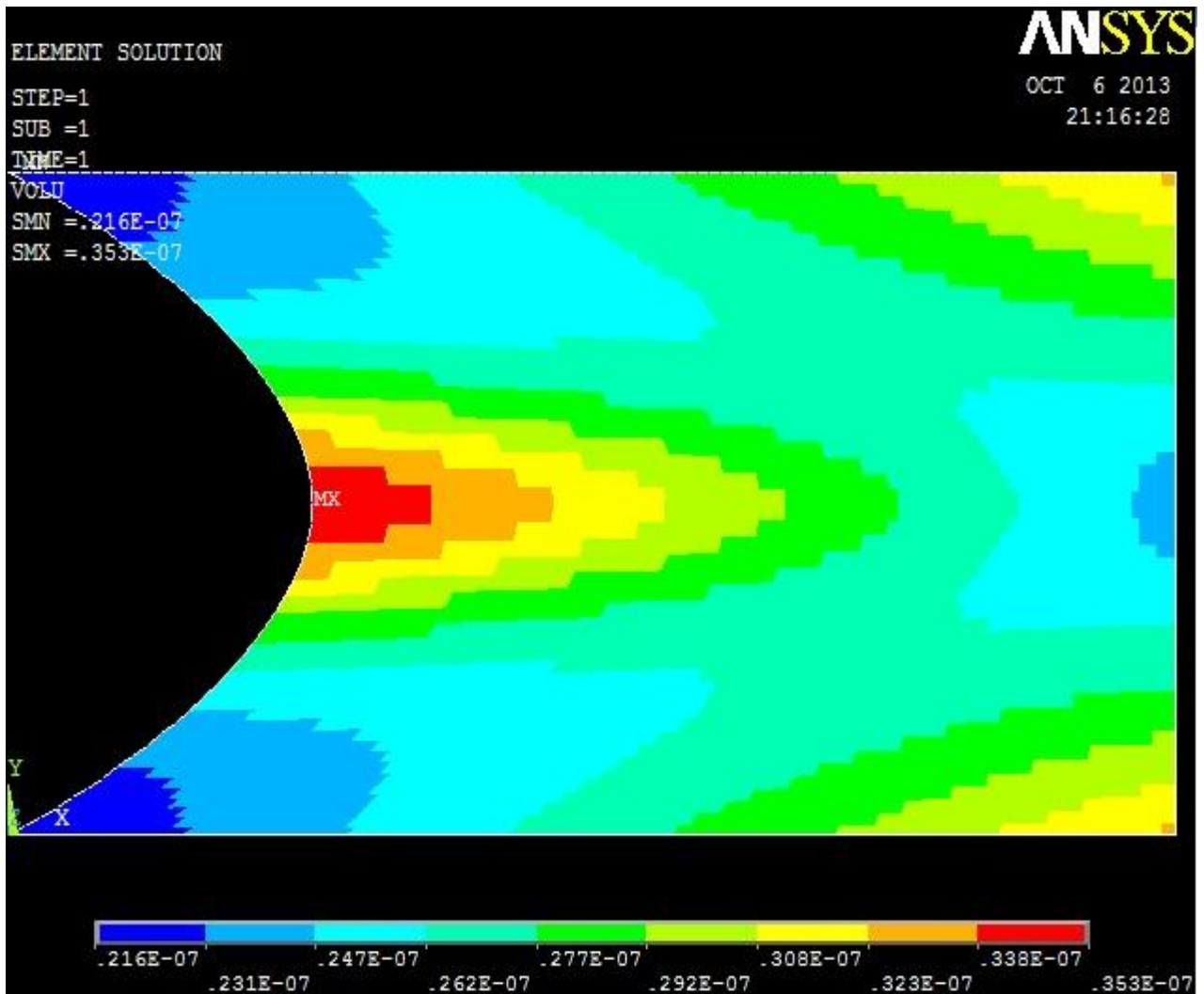
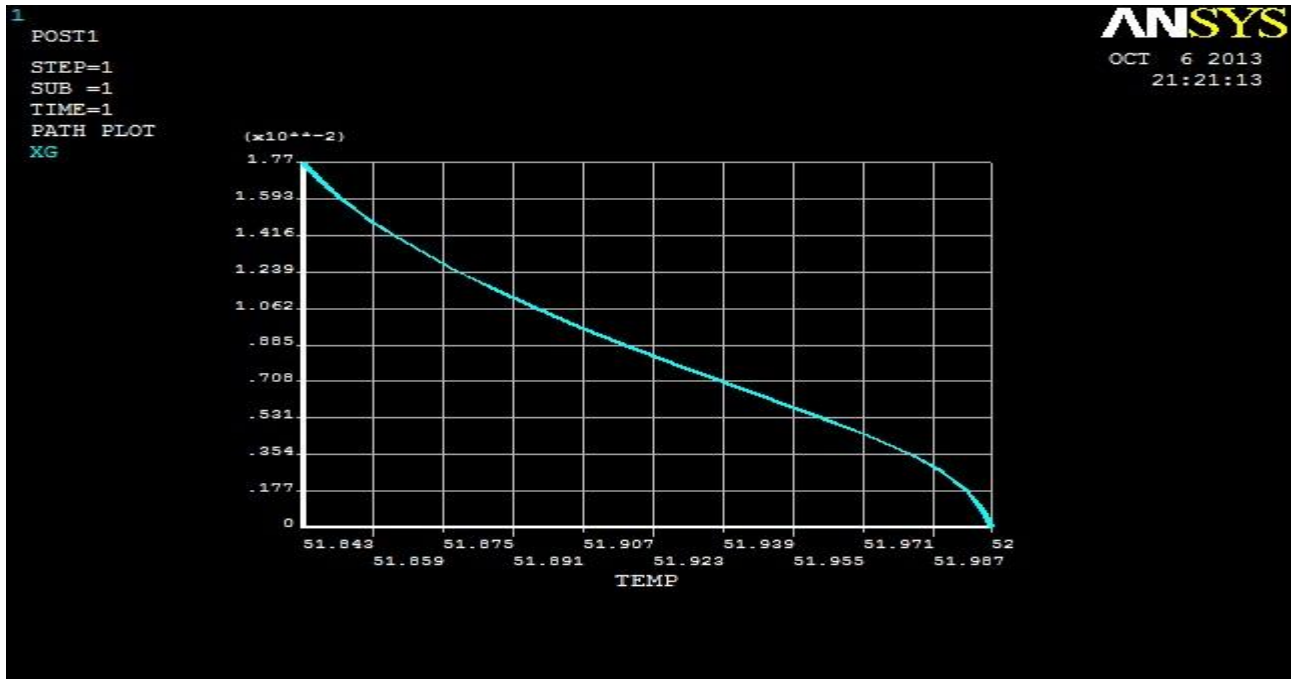


Fig: 20

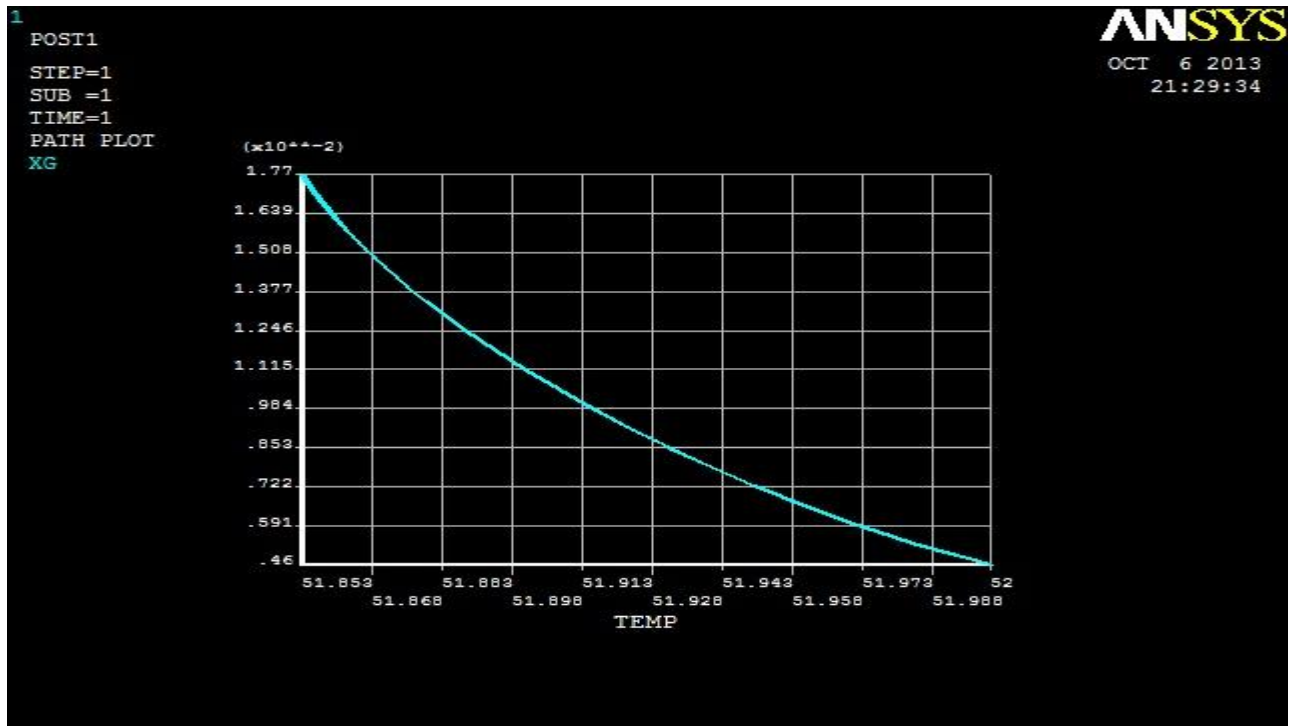


## 5.2.8 Temperature vs. Distance graph (Fig: 21)

Distance taken from the (0, 0) point of the fin



Distance taken from the middle point



## 5.2.9 Distance vs. Thermal Flux graph

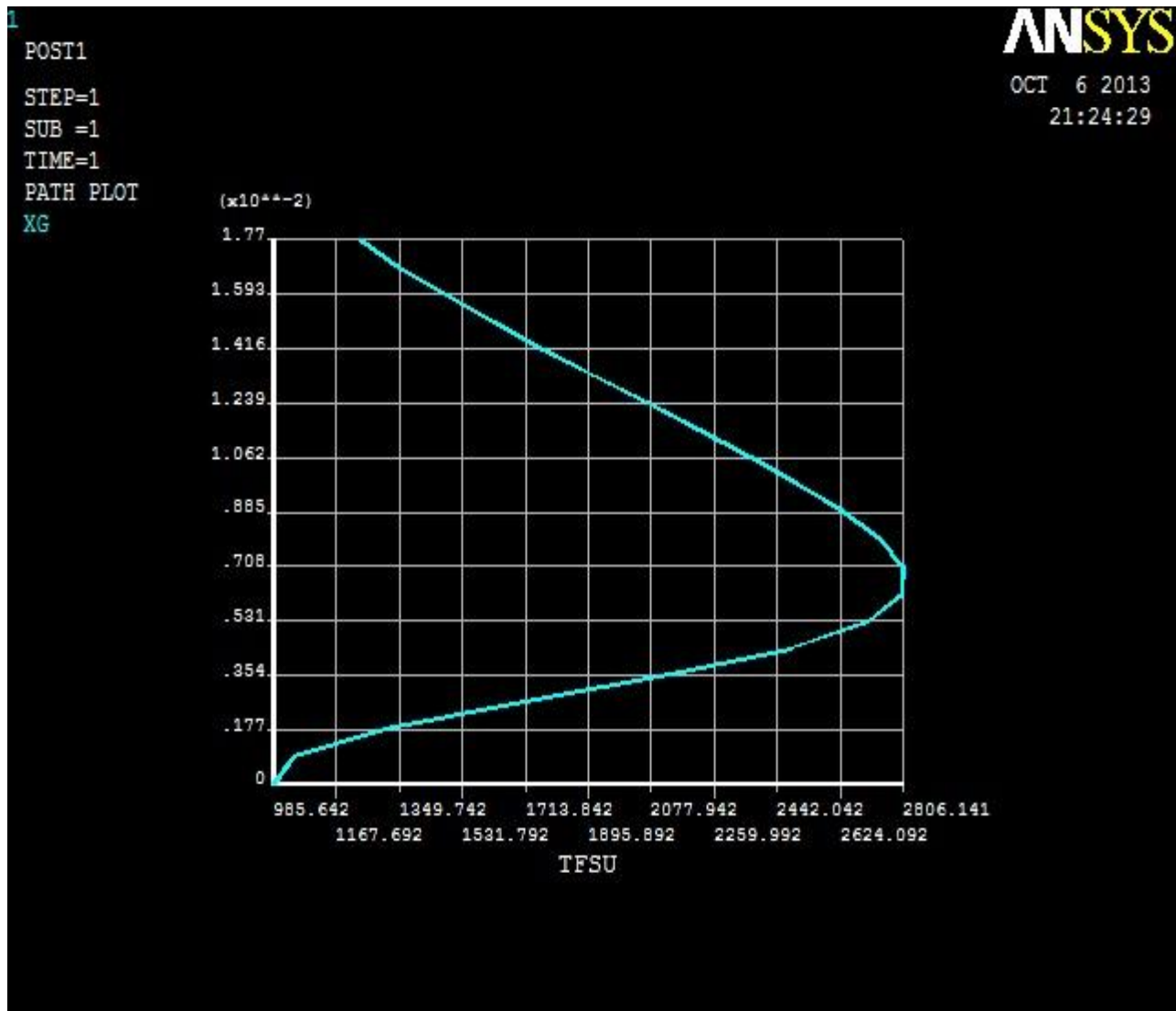


Fig: 22

### 5.2.10 Distance vs. Thermal Gradient graph

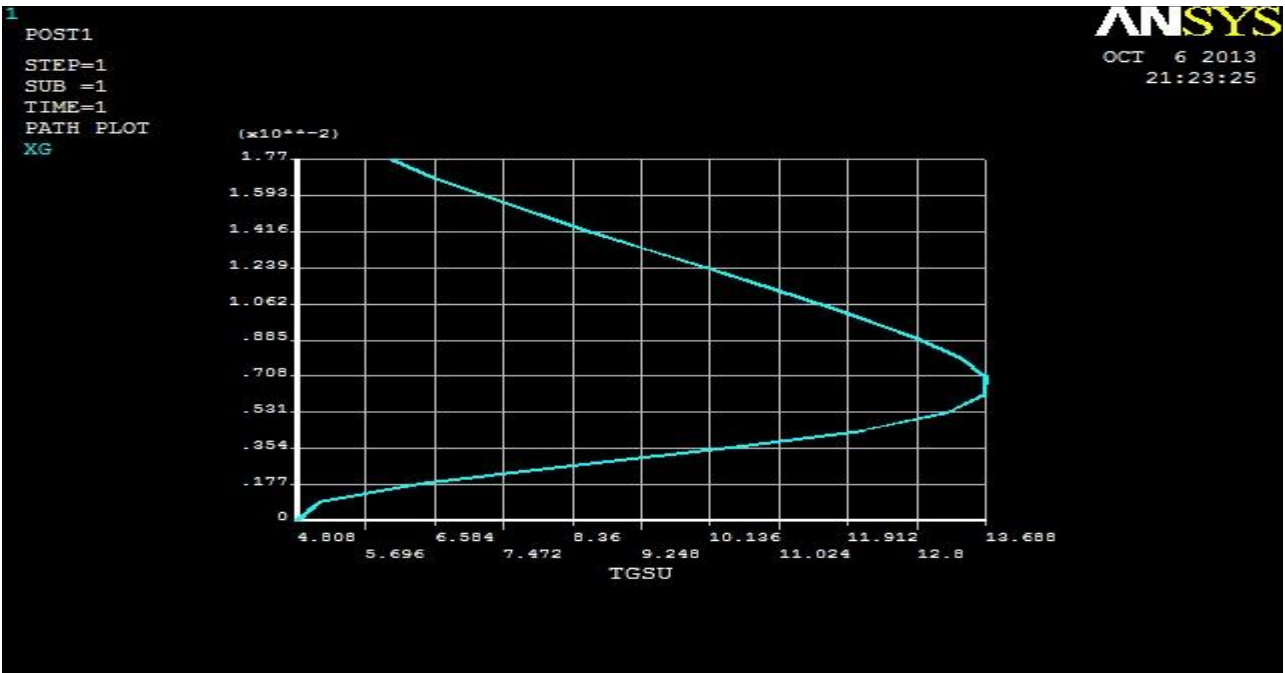


Fig: 23

### 5.2.11 Thermal flux vs. Temperature graph

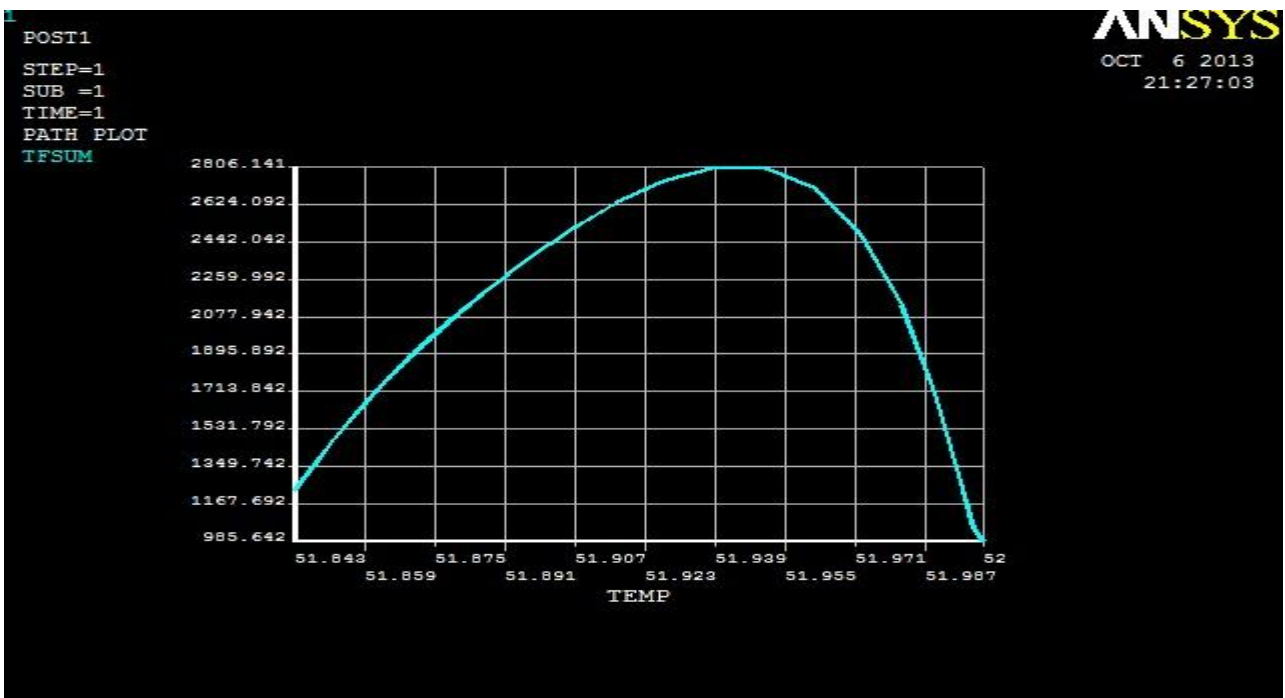


Fig: 24

## 5.2.12 Thermal Gradient vs. Temperature Graph

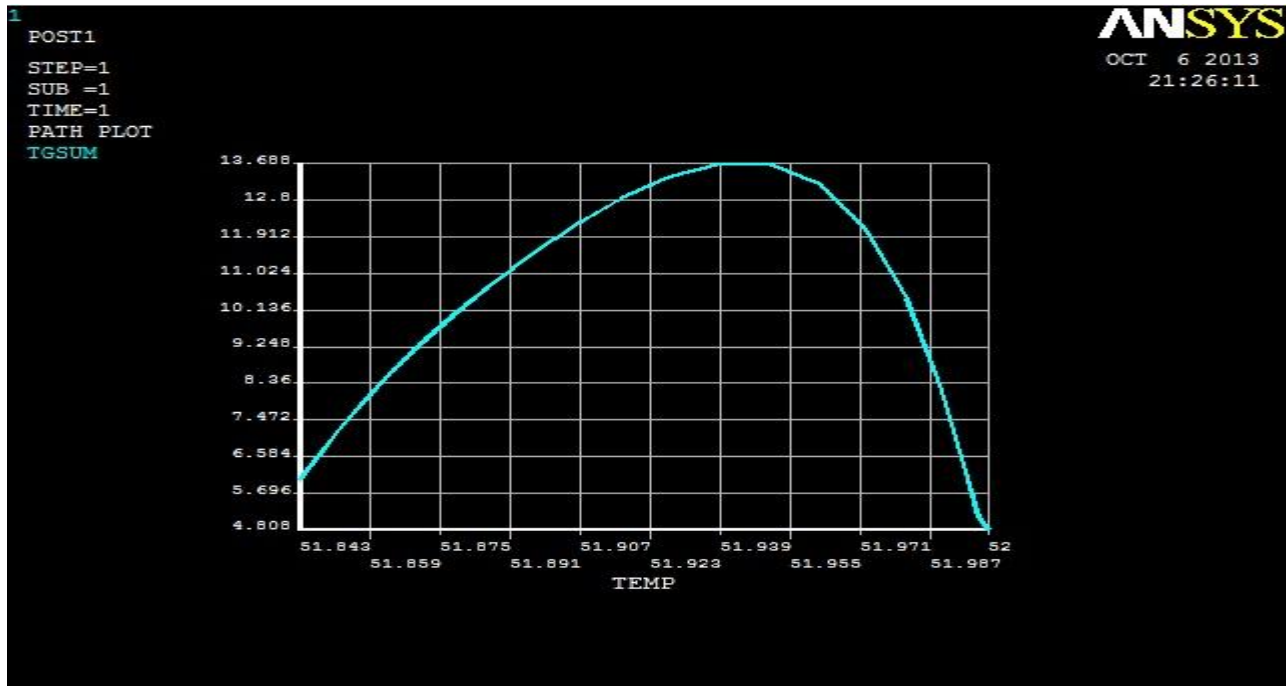


Fig: 25

## 5.2.13 Thermal gradient vs. Thermal flux Graph

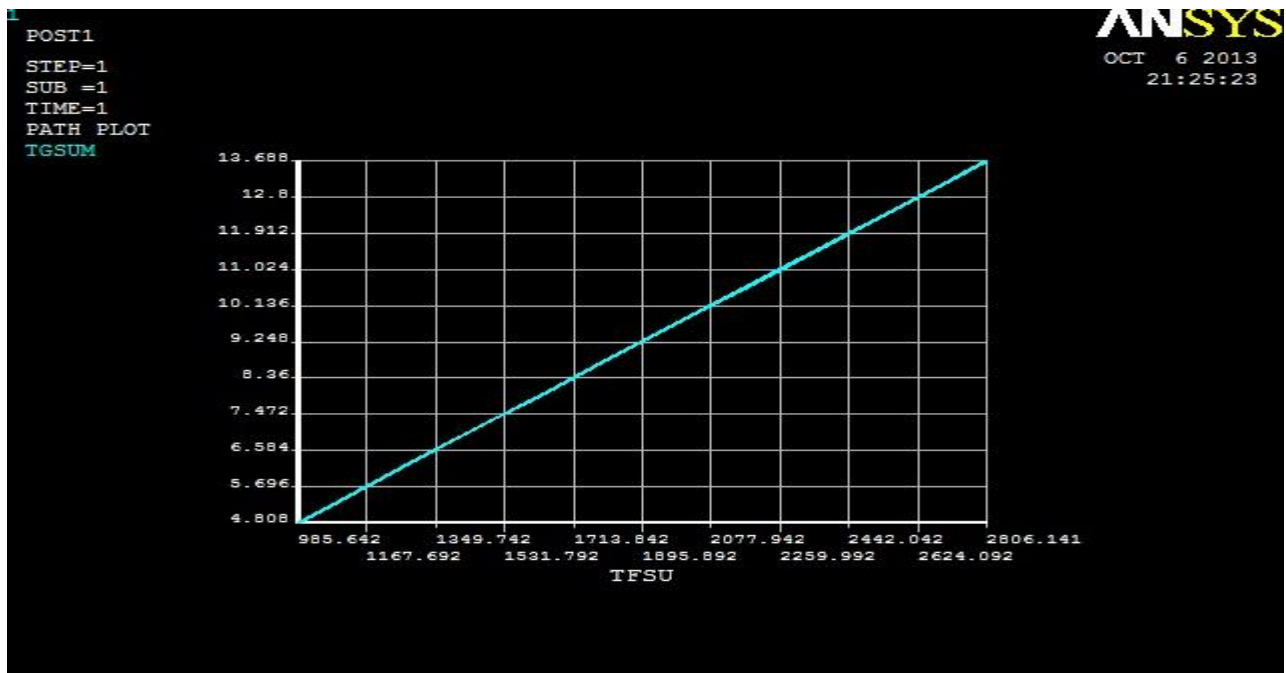


Fig: 26

### 5.3 Cylindrical/Pin Fin:

#### 5.3.1 Nodal temperature:

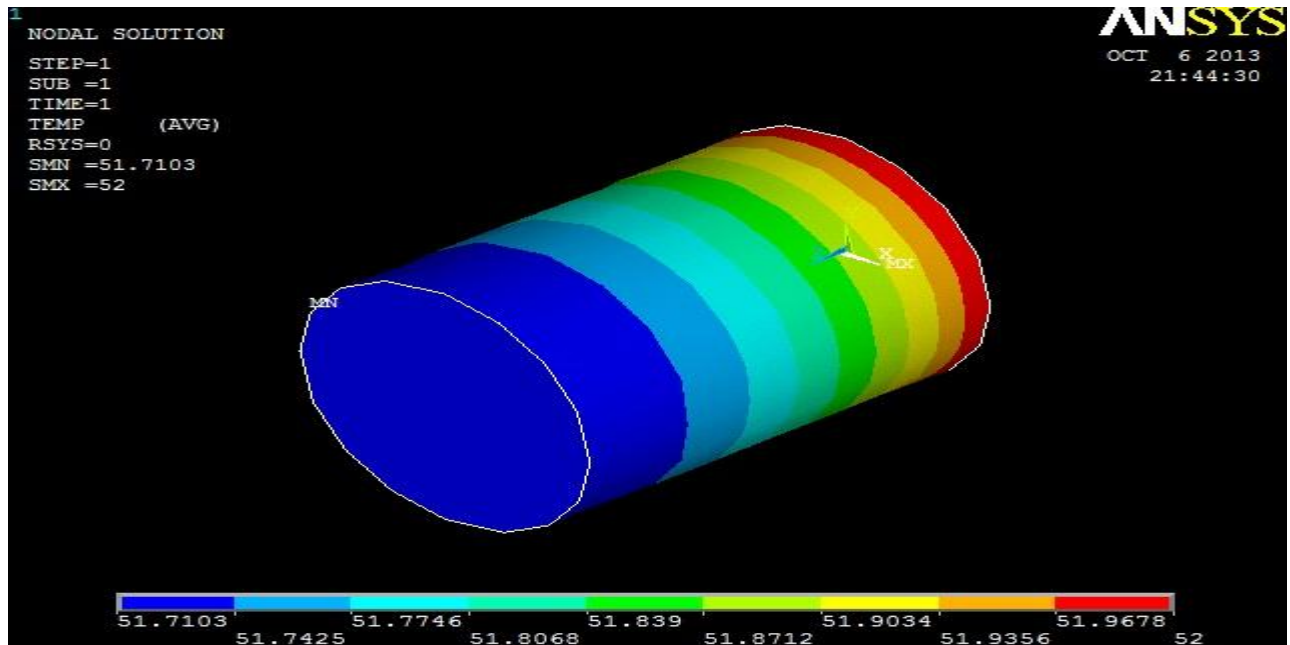


Fig: 27

#### 5.3.2 Heat Flow

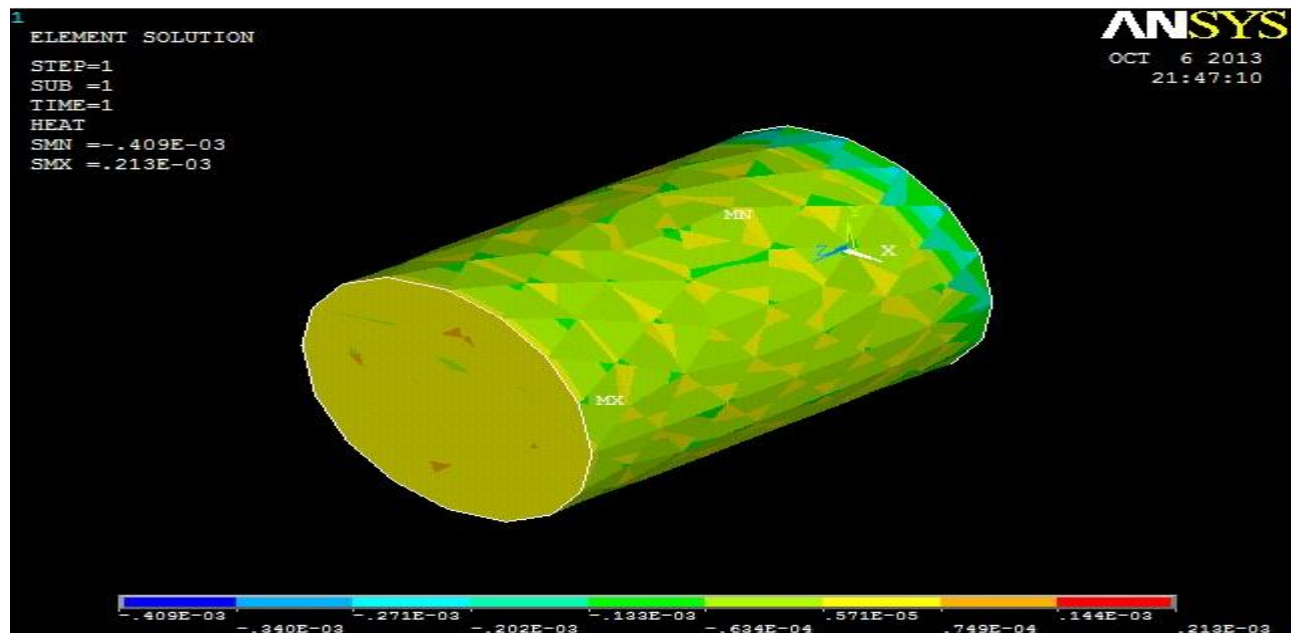


Fig: 28

### 5.3.3 Thermal Gradient Vector Plot

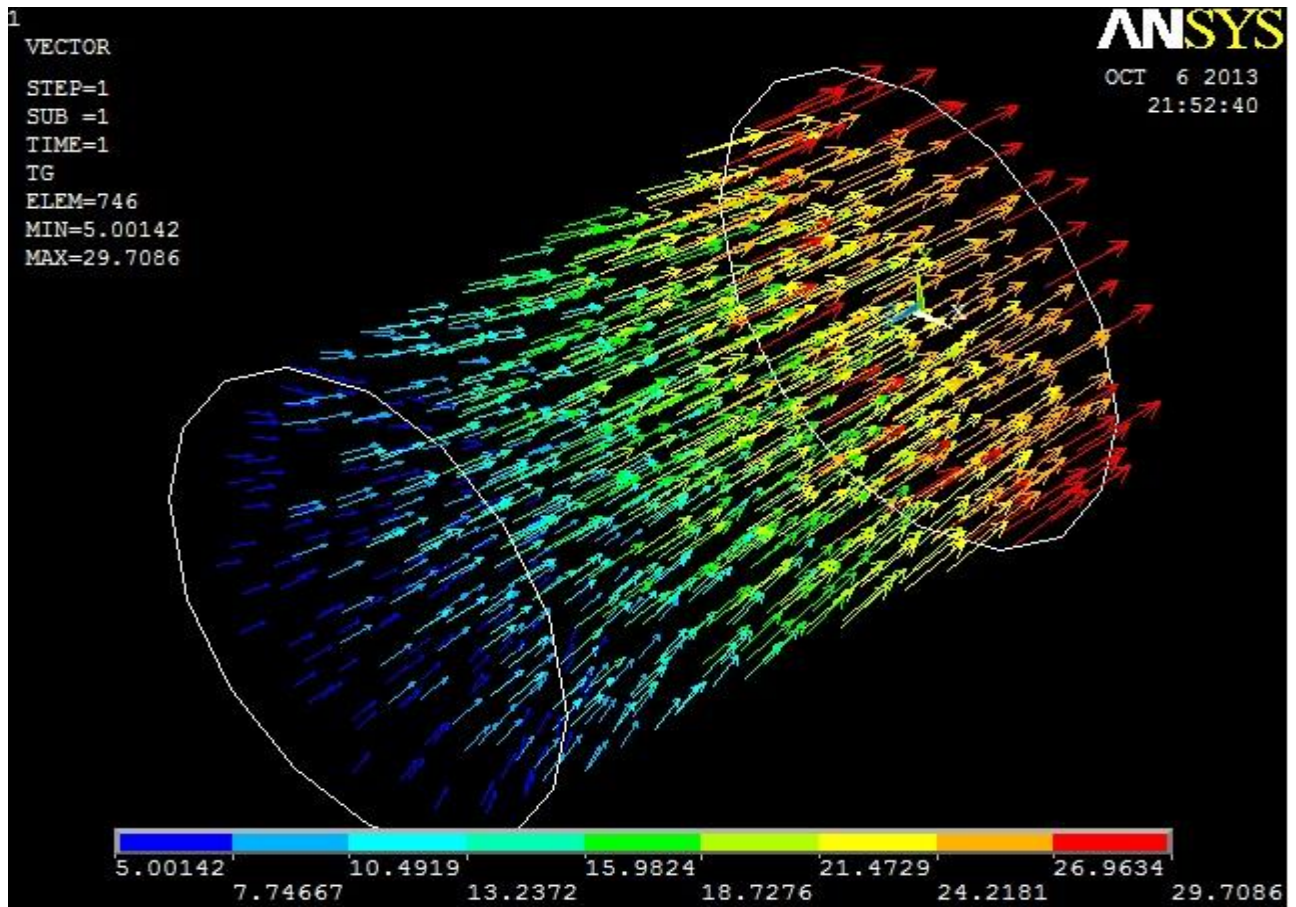


Fig: 29



### 5.3.4 Thermal Gradient Vector Sum

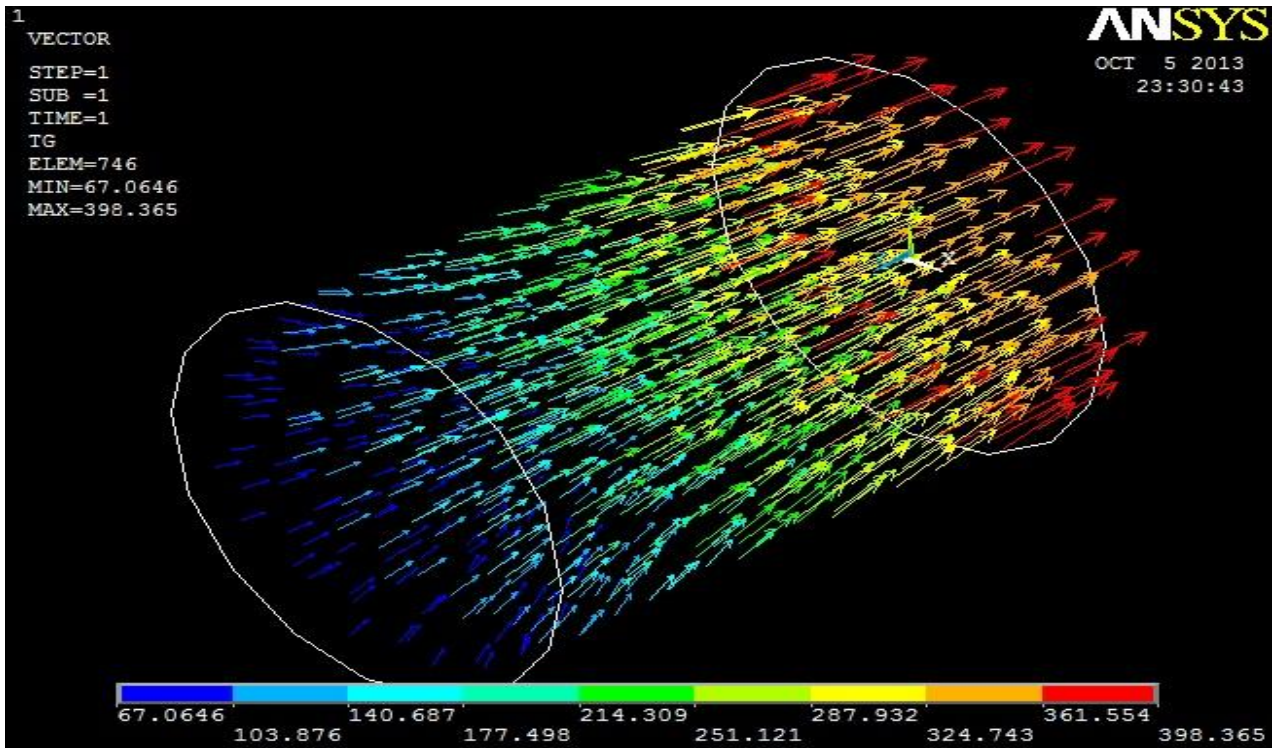
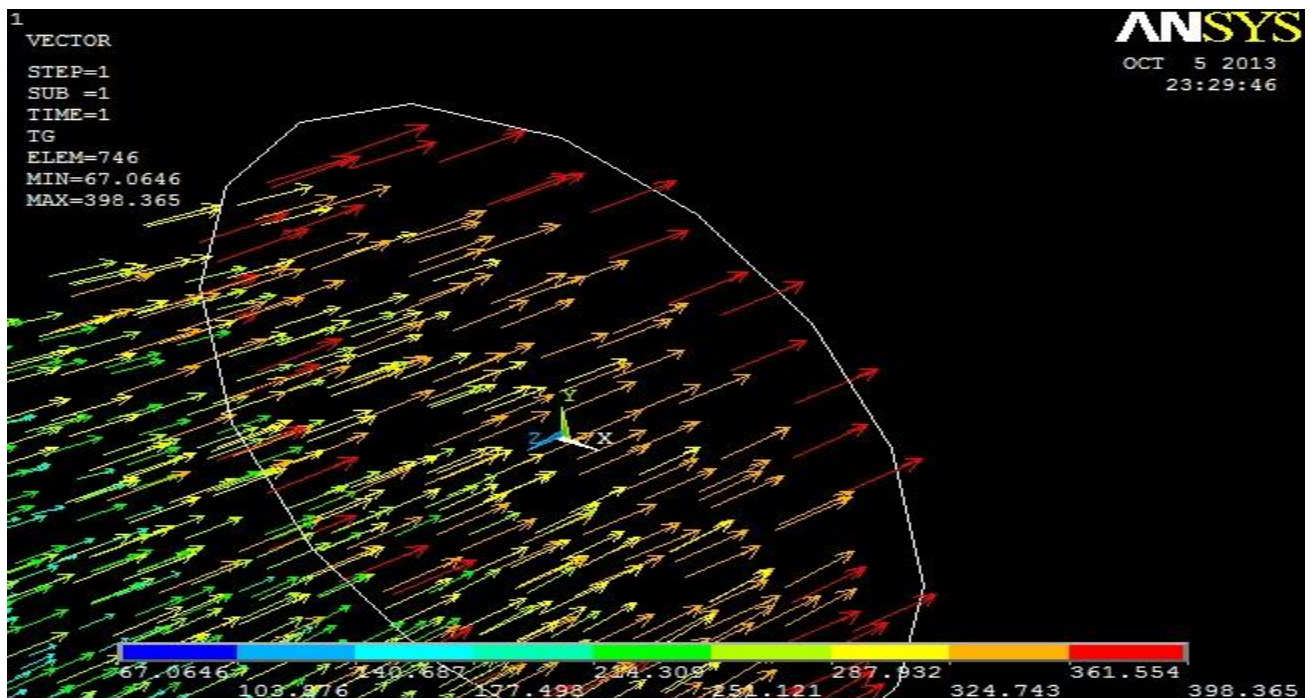


Fig: 30

### Zoomed



### 5.3.6 Thermal Flux Vector Sum

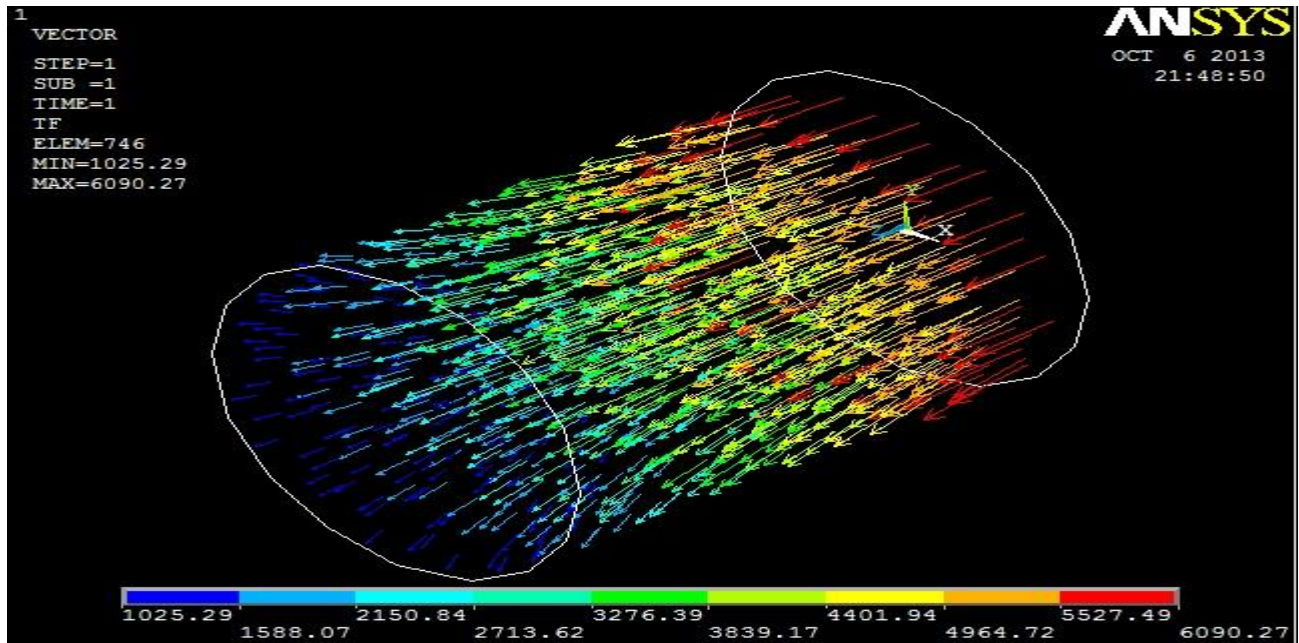
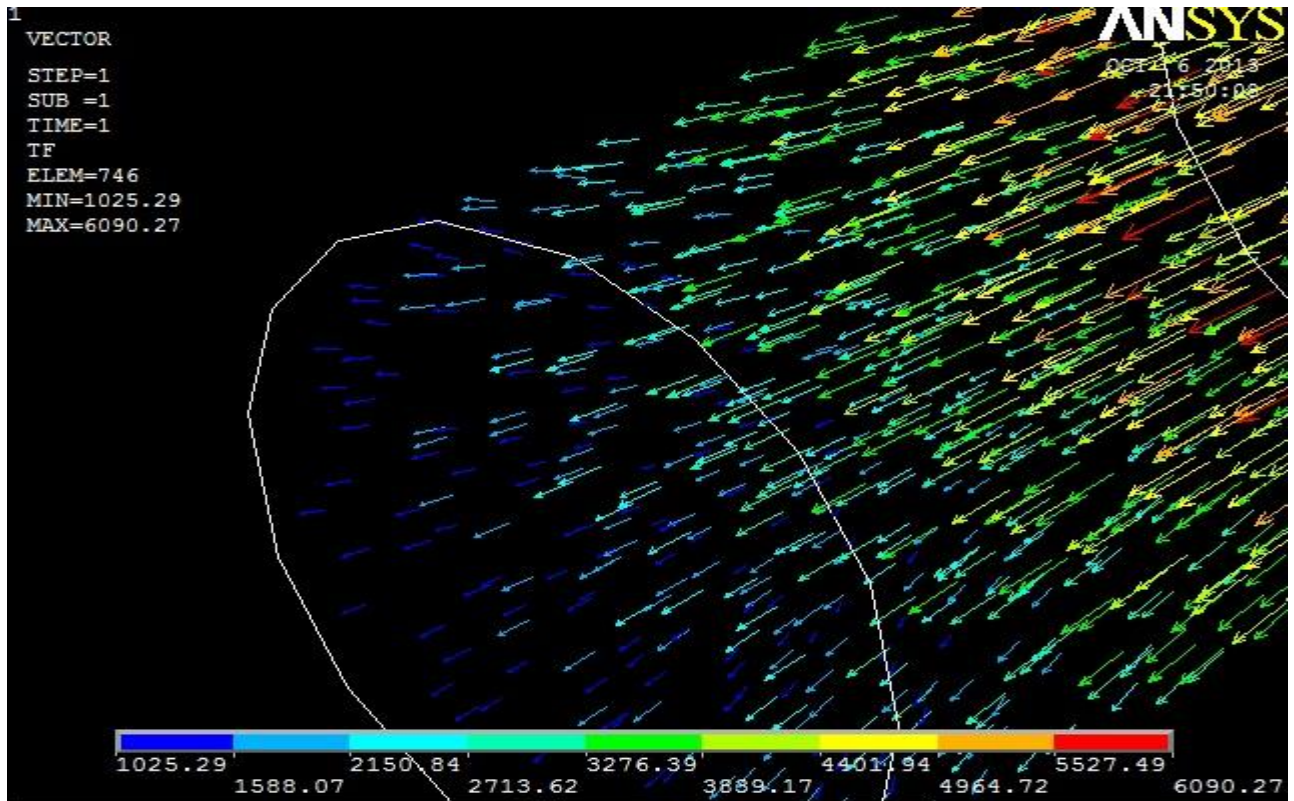


Fig: 31

### Thermal Flux Vector Sum ZOOM





### 5.3.7 Volumetric Element Solution

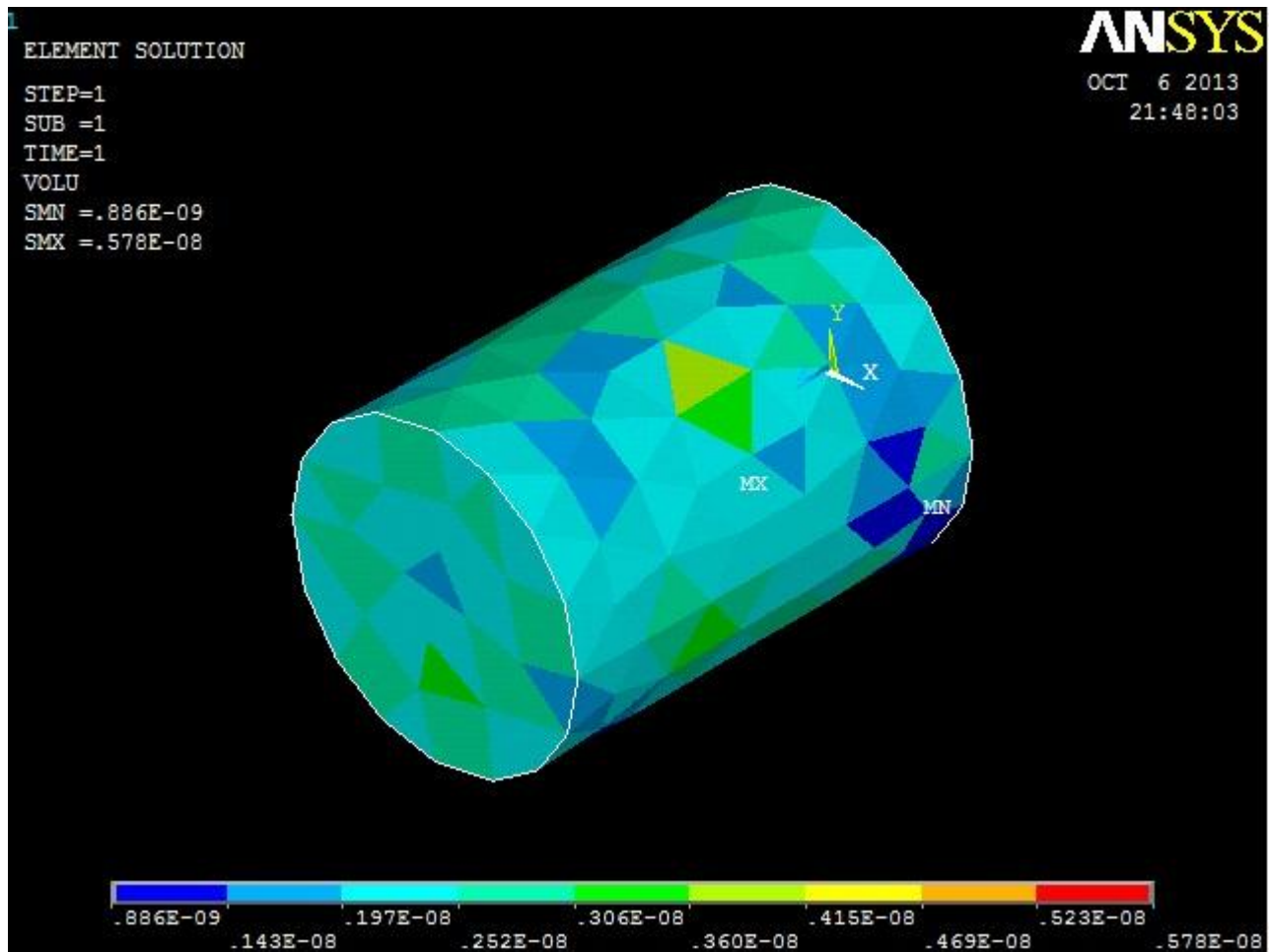


Fig: 32

### 5.3.8 Temperature vs. Distance graph

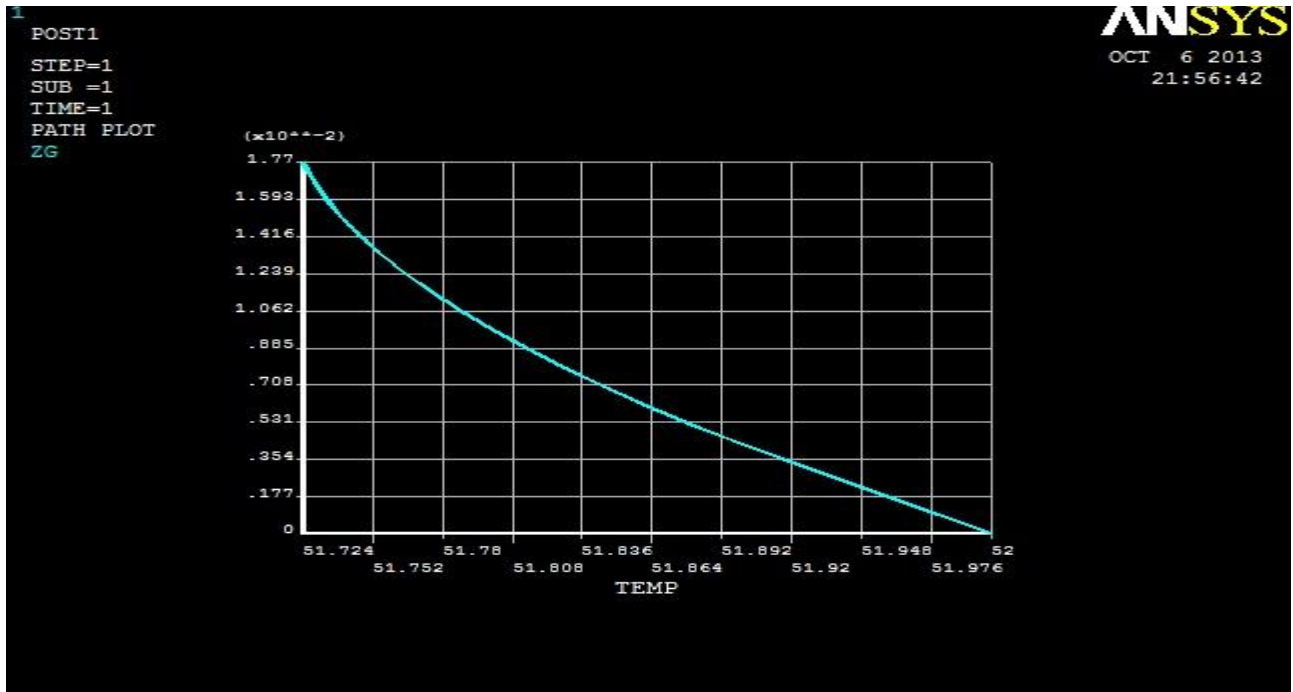


Fig: 33

### 5.3.9 Distance vs. Thermal Flux graph

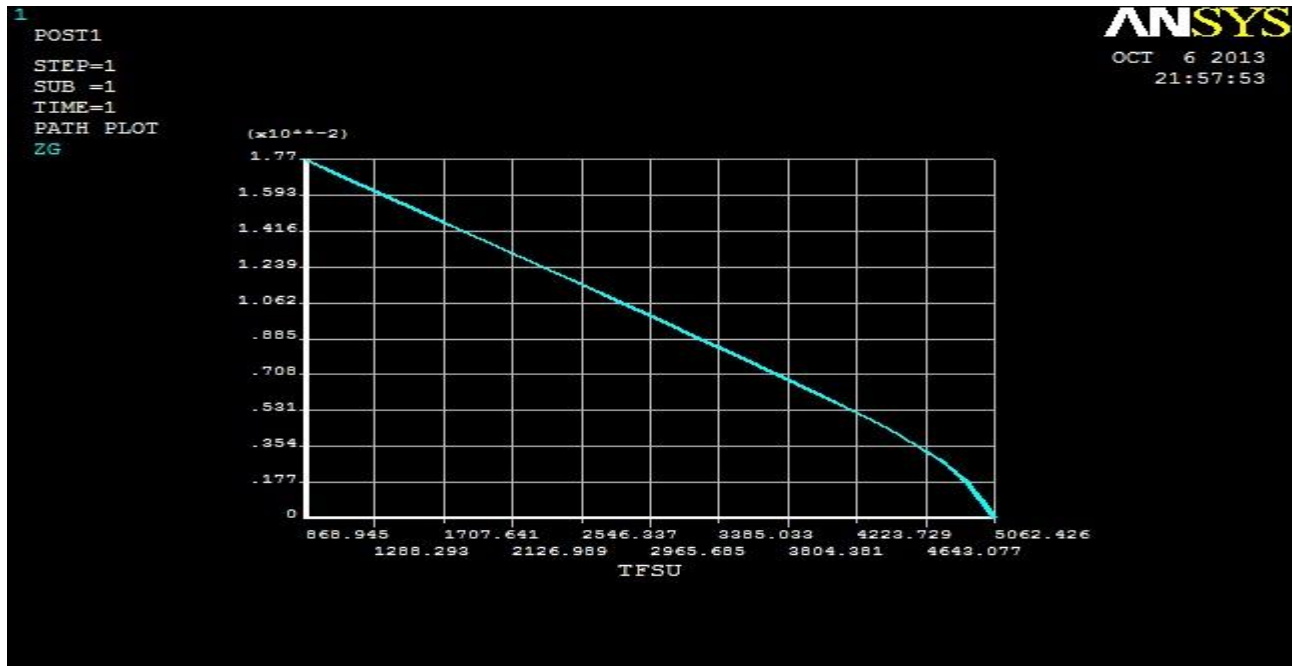


Fig: 34

### 5.3.10 Distance vs. Thermal Gradient graph

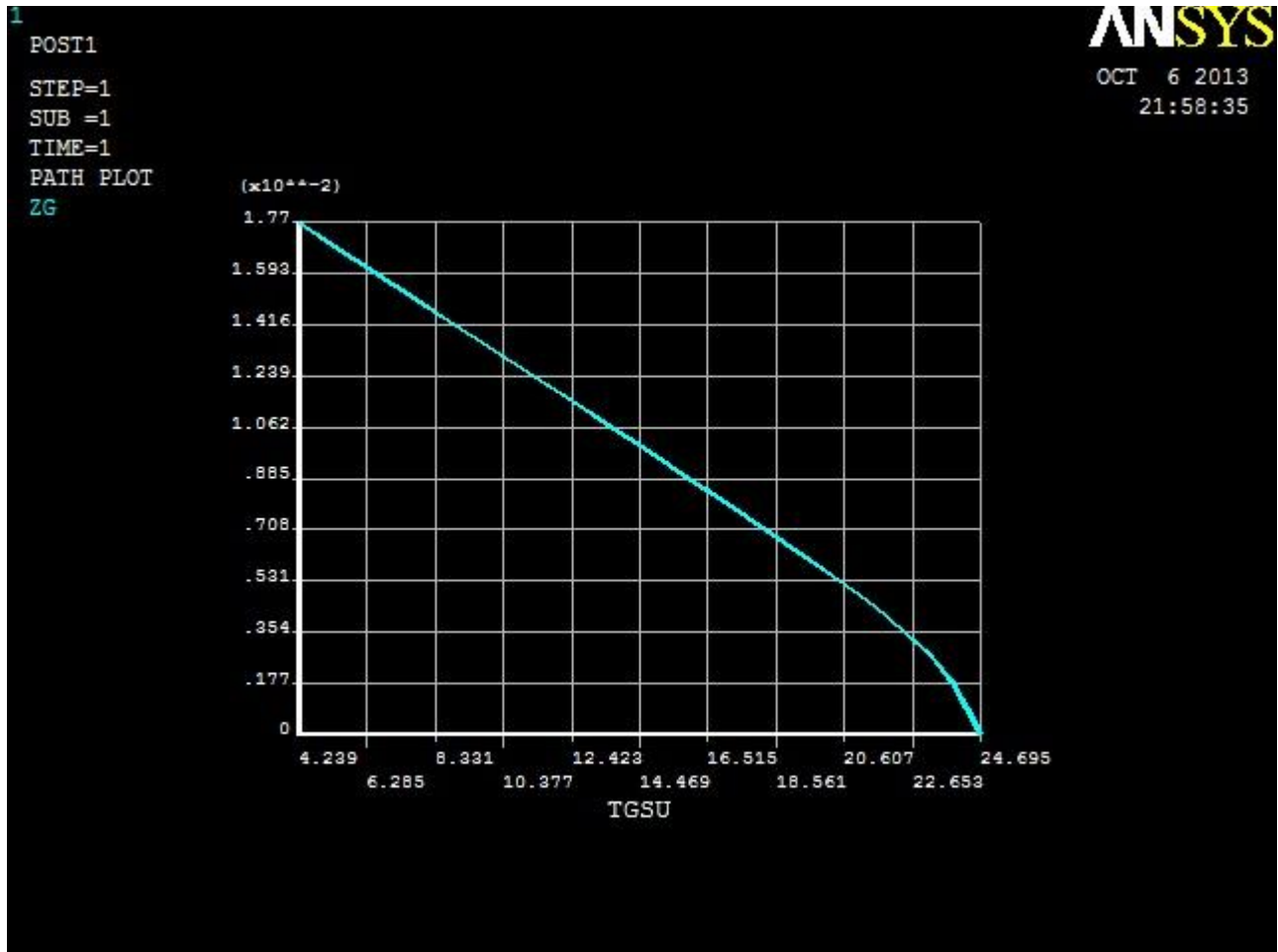


Fig: 35

### 5.3.11 Thermal flux vs. Temperature graph

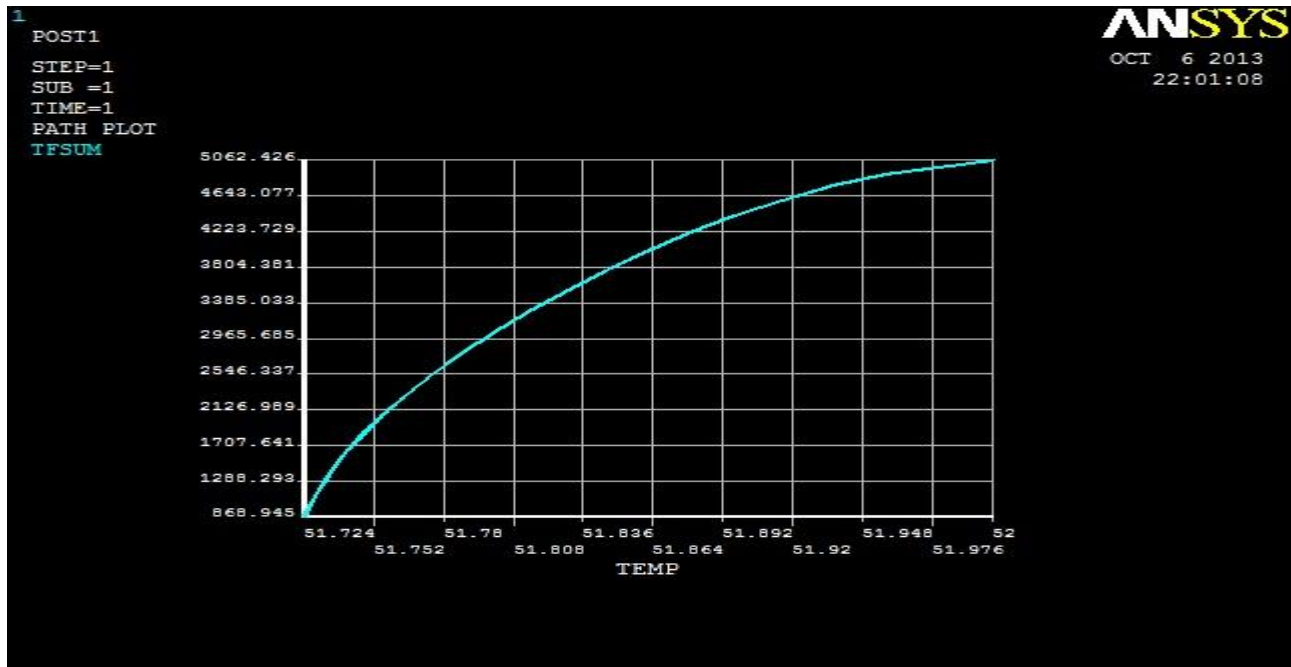


Fig: 36

### 5.3.12 Thermal Gradient vs. Temperature Graph

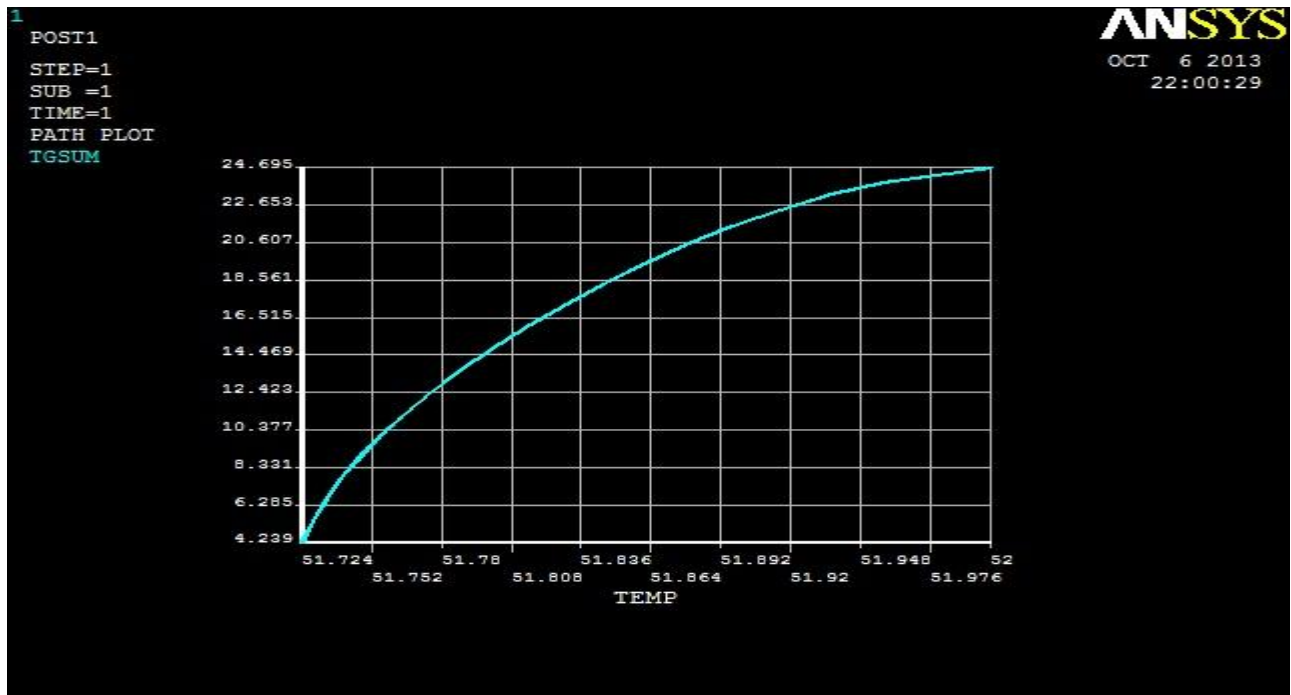


Fig: 37

### 5.3.13 Thermal gradient vs. Thermal flux Graph

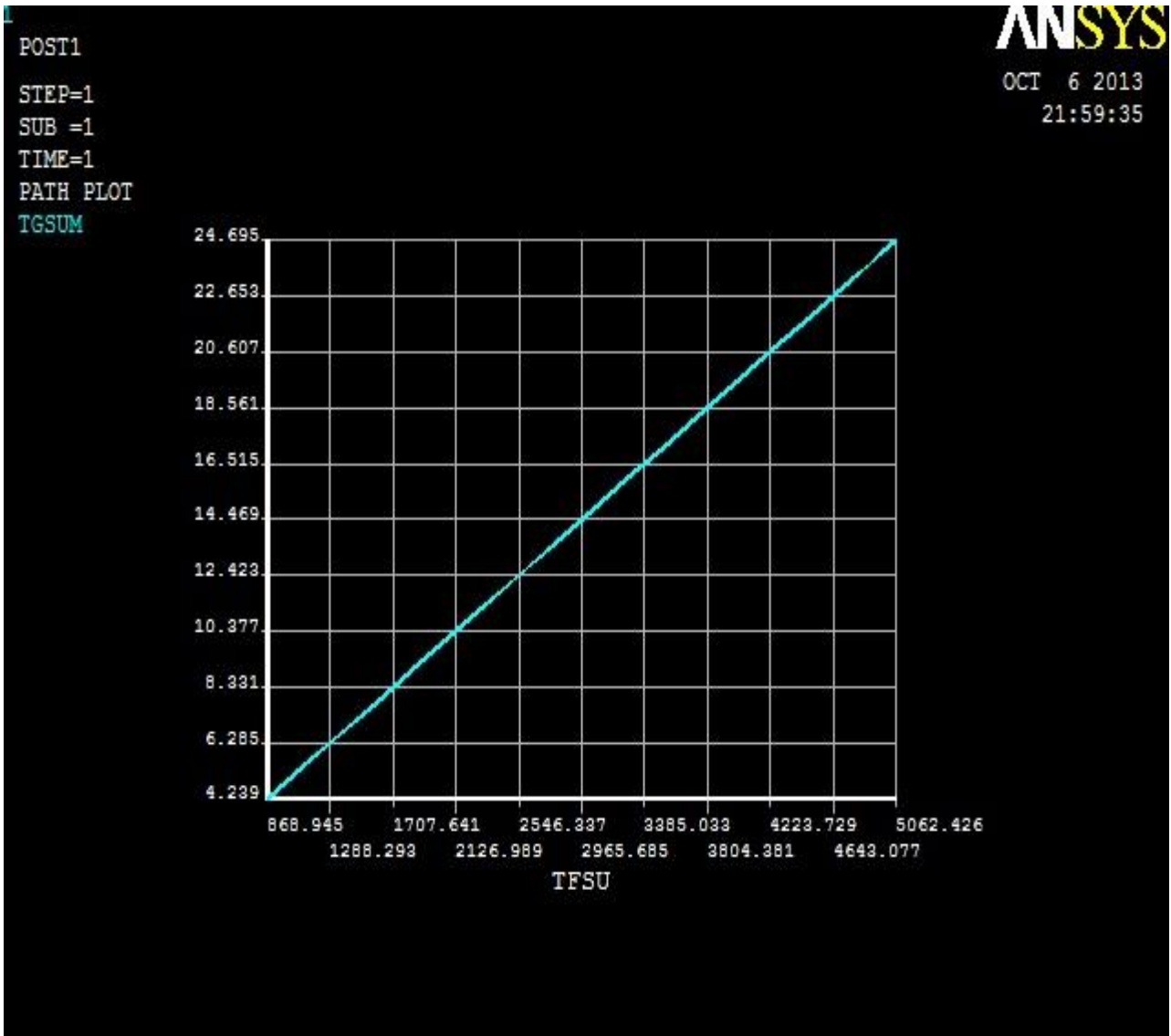


Fig: 38

# CHAPTER SIX

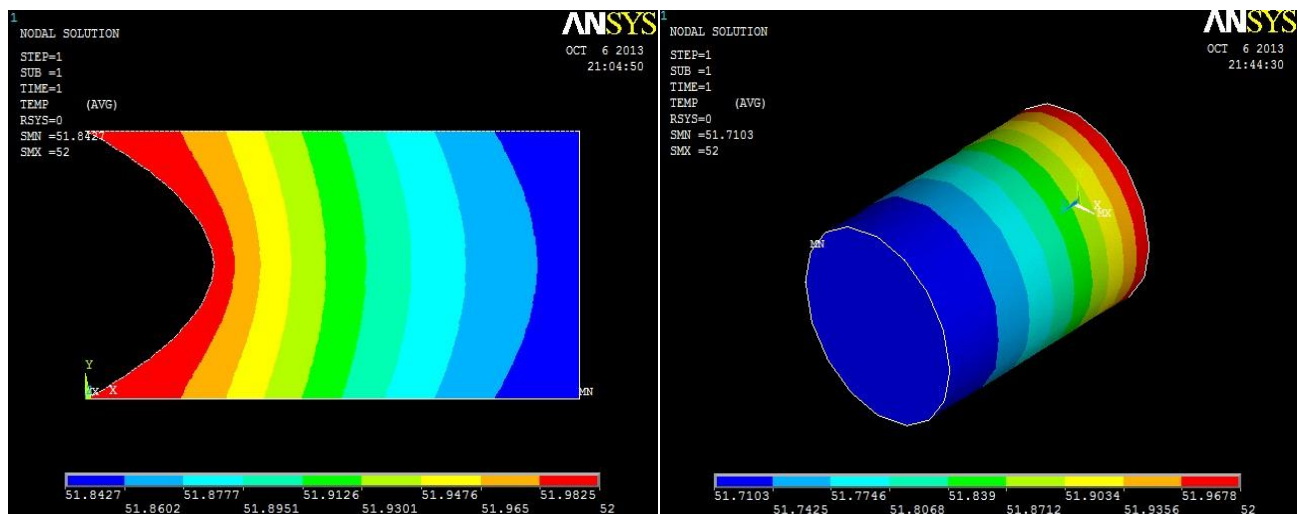
## RESULT ANALYSIS AND COMPARISON

### 6.1 Available Results

The simulation process yields different sets of results as shown in Result chapter. For comparison and analysis few results has been chosen here. These are:

1. Nodal Temperature
2. Distance vs. temperature graph
3. Distance vs. thermal flux graph
4. Distance vs. thermal gradient graph
5. Heat Flow rate

### 6.2 Nodal Temperature Analysis:

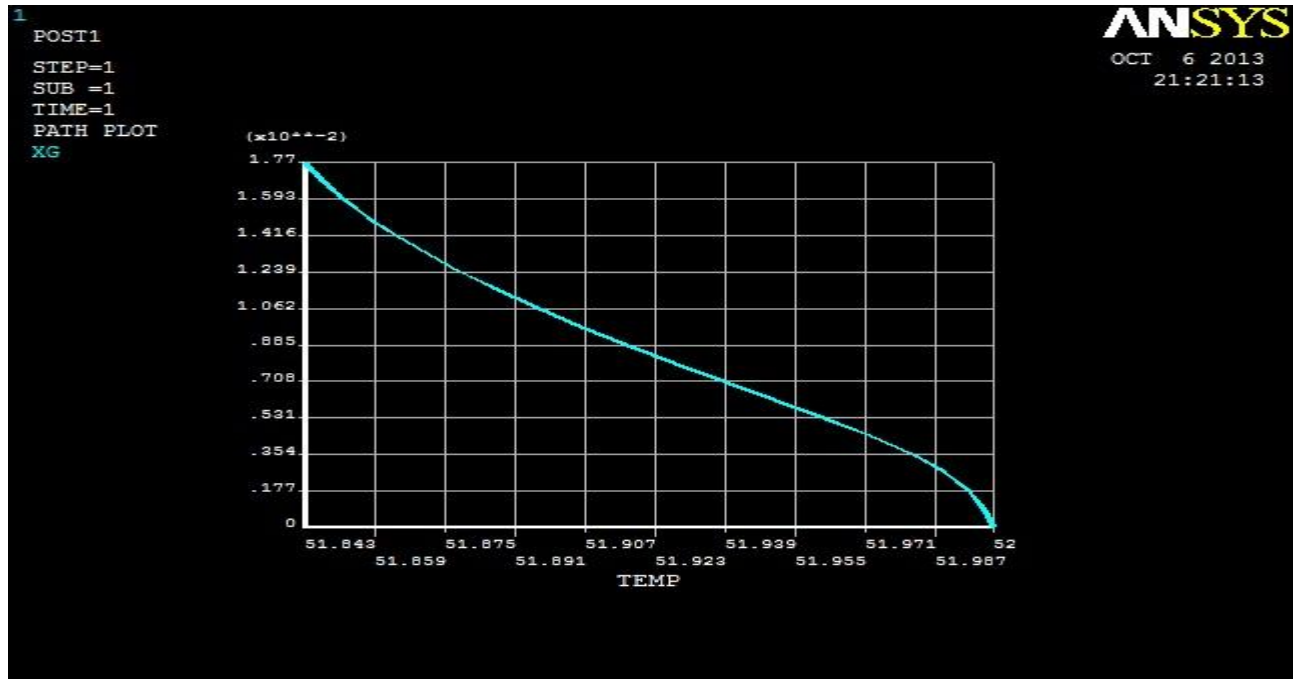


Rectangular Fin

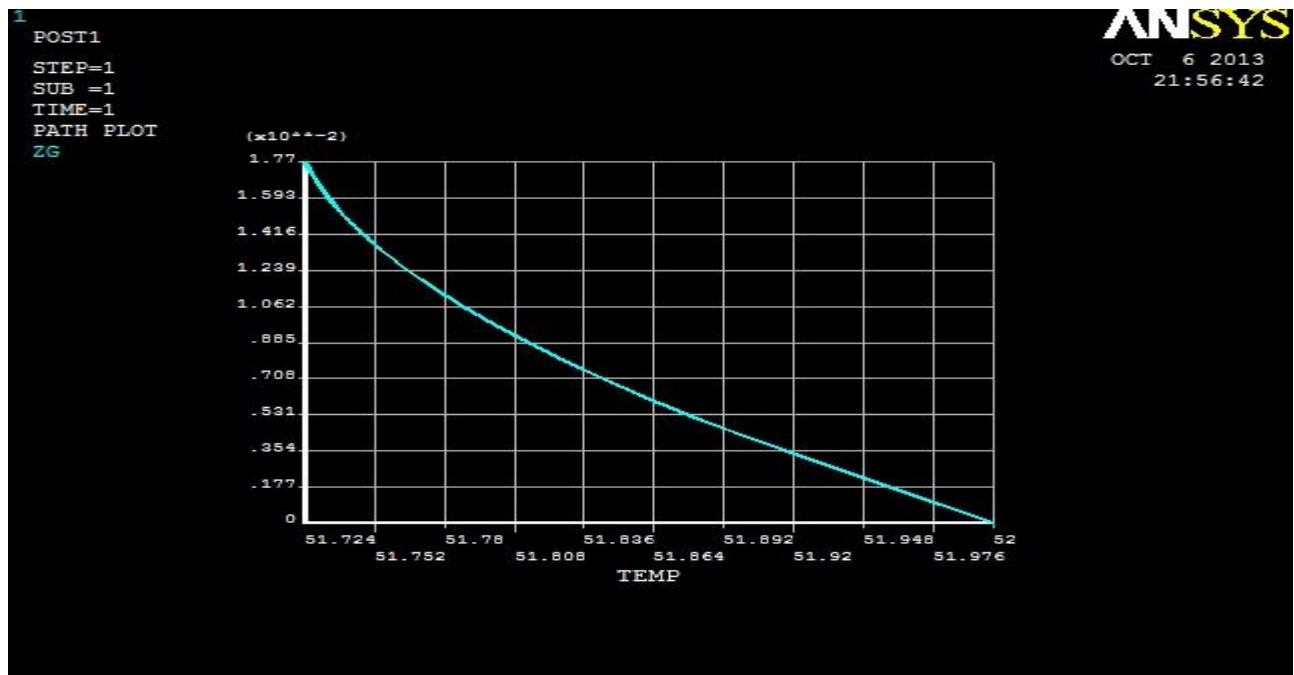
Cylindrical Fin

The base temperature of both the fins is same as the initial temperature  $52^{\circ}\text{C}$  or  $325\text{ K}$ . As the nodal temperature shows the temperature regions at different distance it can be easily seen the tip temperature is  $322.891\text{ K}$  in rectangular fin where cylindrical fin's tip temperature is  $321.115\text{ K}$ . So the cylindrical fin has more cooling effect on the heat transfer than the rectangular fin.

### 6.3 Distance vs. temperature graph:



Rectangular Fin



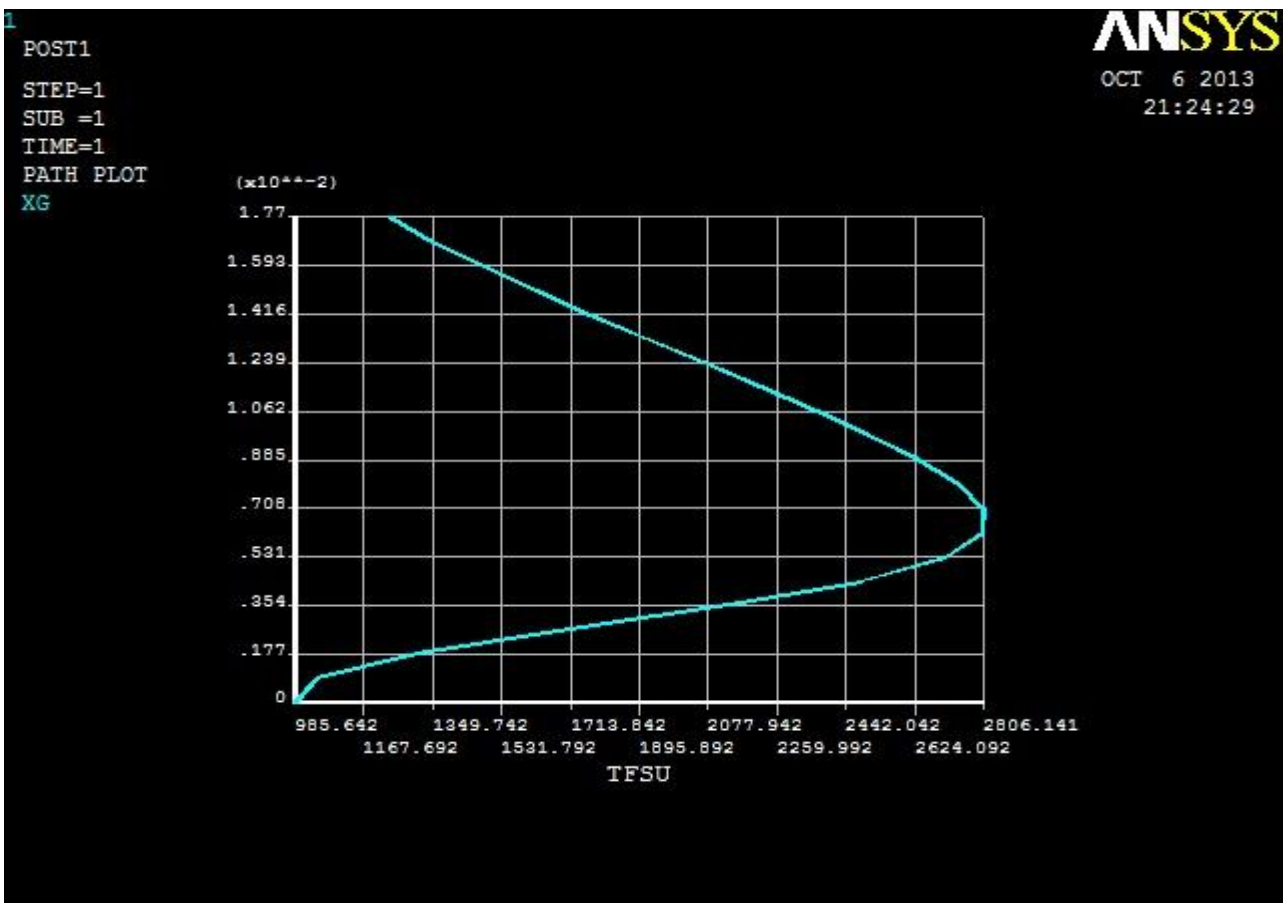
Cylindrical Fin

The software has divided the whole length of the fin into 10 equal lengths. Then it has generated a graph automatically by keeping distance in Y axis and temperature in X axis. As the rectangular fin has a half circle shaped base, so there are two possible graphs. One taken from the center of the circle to the tip and another one is from the starting point (0, 0) to the tip. Cylindrical fin does not have that sort of irregularity in shape.

For comparison the graph chosen are: starting point to tip and the cylinder graph.

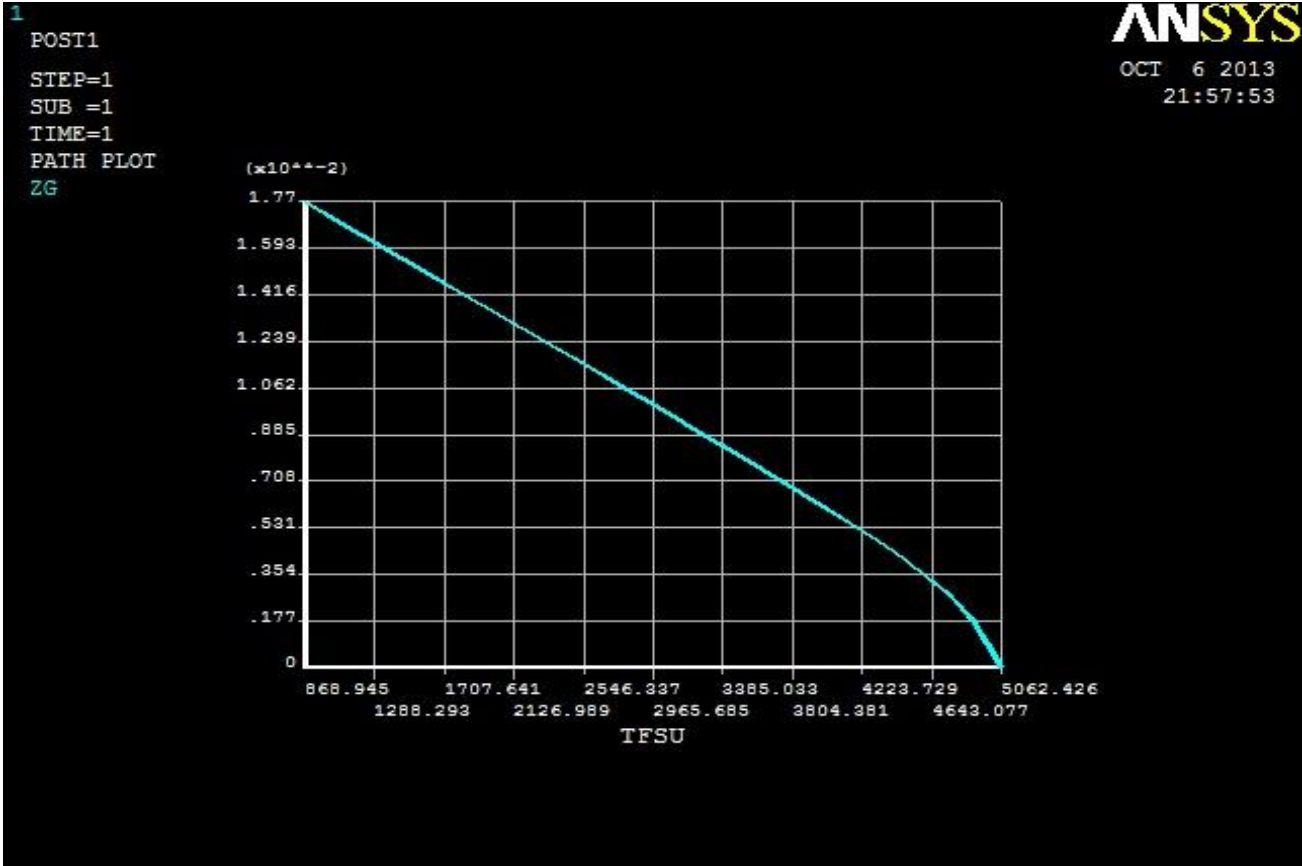
From the graph it is clearly seen that the rectangular fin transfers more heat than the cylindrical fin.

### 6.4 Distance vs. Thermal Flux graph:



Rectangular fin

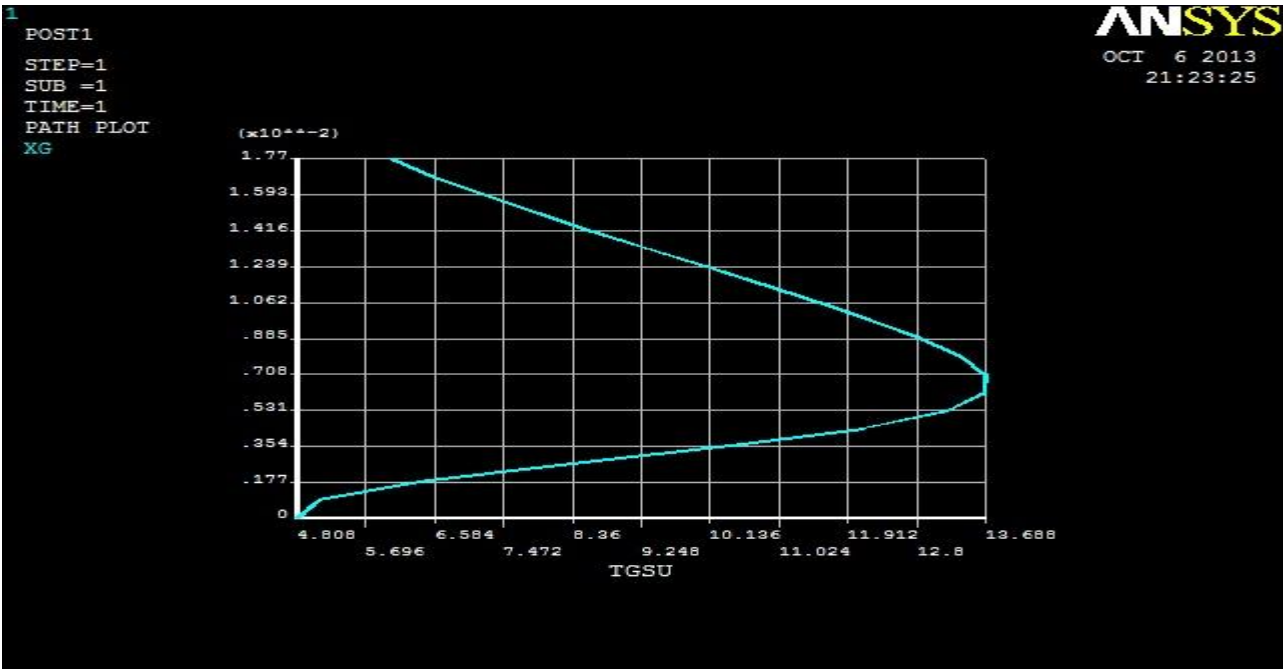




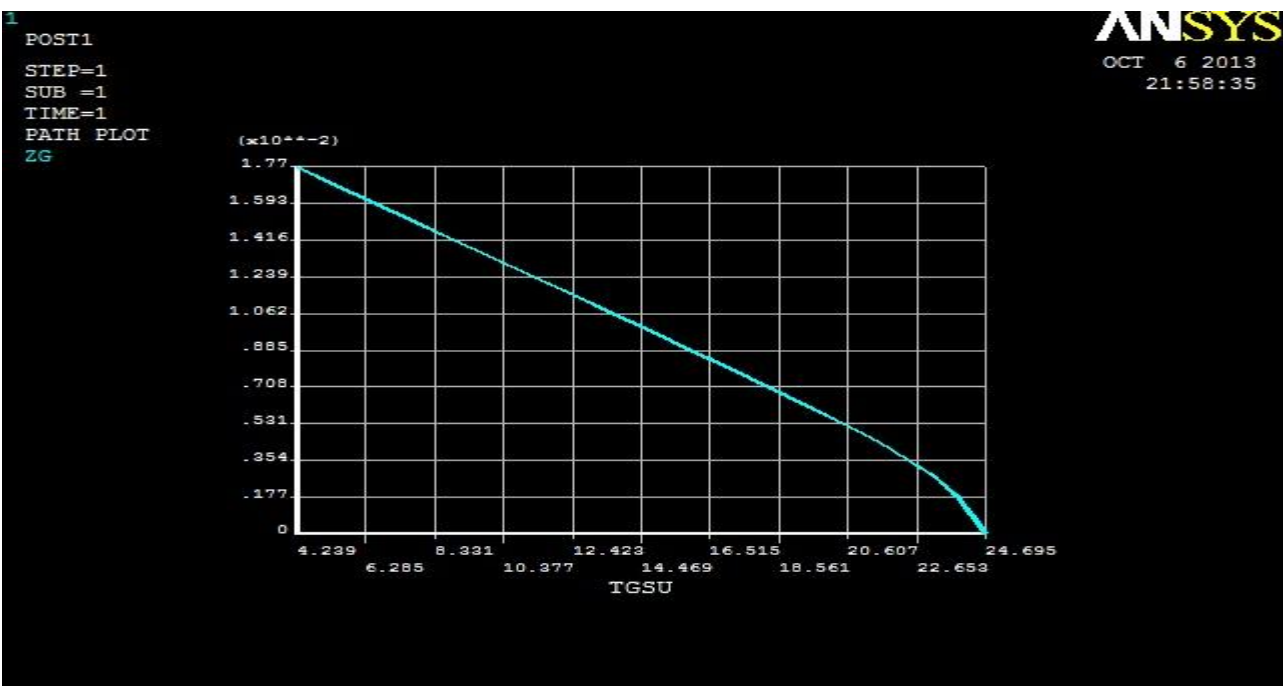
Cylindrical fin

Thermal flux is the amount of heat transferred along the area perpendicular to the direction of heat flow at unit time. The graphs show that the amount of heat flux through the rectangular fin is greater than the cylindrical fin.

## 6.5 Distance vs. thermal gradient graph:



Rectangular fin

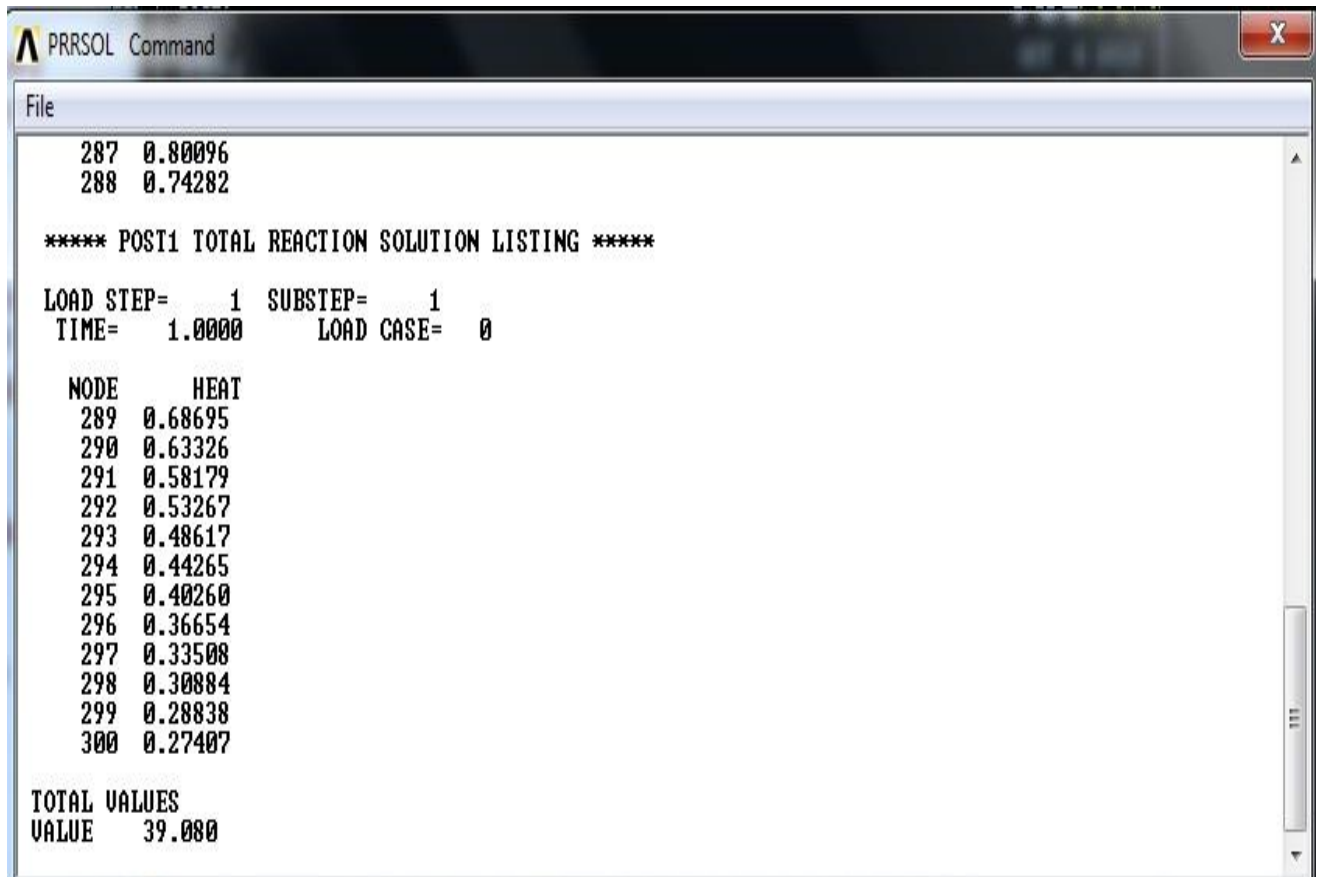


Cylindrical fin

Thermal gradient is the physical quantity that describes in which direction and at what rate the temperature changes the most rapidly around a particular location. Unit is

$\text{K m}^{-1}$ . From the graphs shown above it is very much clear that the rectangular fin is giving more as rectangular fin is giving 87.994 K/m whereas cylindrical fin is giving 56.828 K/m.

## **6.6 Heat Flow Rate:**



```
PRRSOL Command
File
287 0.80096
288 0.74282

**** POST1 TOTAL REACTION SOLUTION LISTING ****
LOAD STEP= 1 SUBSTEP= 1
TIME= 1.0000 LOAD CASE= 0

NODE HEAT
289 0.68695
290 0.63326
291 0.58179
292 0.53267
293 0.48617
294 0.44265
295 0.40260
296 0.36654
297 0.33508
298 0.30884
299 0.28838
300 0.27407

TOTAL VALUES
VALUE 39.080
```

**Fig 39:** Rectangular fin

```
PRRSOL Command
File
***** POST1 TOTAL REACTION SOLUTION LISTING *****
LOAD STEP= 1 SUBSTEP= 1
TIME= 1.0000 LOAD CASE= 0

NODE    HEAT
133  0.13139E-01
134  0.13288E-01
135  0.13065E-01
136  0.42446E-04
137 -0.27934E-03
138 -0.55361E-04
139 -0.28448E-03
140  0.43699E-04
141 -0.29002E-03
142 -0.85232E-04
143 -0.37317E-03
144 -0.14164E-04
145 -0.24262E-04
146 -0.22524E-04
147 -0.12974E-04

TOTAL VALUES
VALUE  0.73292
```

**Fig 40:** Cylindrical fin

The heat flow rate chart shows that rectangular fin has the heat flow rate is 39 W/m and cylindrical fin has heat flow rate is 0.73292 W/m.

# CHAPTER SEVEN

## CONCLUSION

---

### 7.1 Decisions

- **The above simulations, results and comparisons conclusion can be drawn that the rectangular fin is better heat dissipating surface than the cylindrical fin.**
- **The methodology of this simulation is established and can be used to simulate other types of fins.**
- **Temperature variations along the distance are the main reference of heat flow.**
- **By this successful simulation it is easy to select the types and sizes of fin for any machine working in any condition.**

### 7.2 Future Recommendations

- The project work here is very limited as it was first step to provide the analysis of fin characteristics.
- More advanced and developed approach can be made to analyze other types of fins.
- The project focused only on the heat flow through the fin, but a better simulation can be made taking the fluid flow through the condenser pipe.
- The thesis work was limited in 2-D form by using ANSYS Mechanical CFD Flo-14.0, but for 3-D simulation and better result ICEM CFD may be used.
- It is expected that the thesis work will continue in the future under the same supervisor and more advanced approach will be achieved.

# References

---

1. Papadopoulos, K.D., Guzman-Garcia, A. G., and Bailey, R. V., "The response of straight and circular fins to fluid temperature changes," Int. Comm. Heat Mass Transfer, Vol. 17, 1990, pp.587-595.
2. Aziz, A. and Kraus, A. D., "optimum design of radiating and convecting – radiating fins," Heat Transfer Engineering, Vol.17, 1996, pp.19-29
3. Su, R. J. and Hwang, J. J., "Transient analysis of two-dimensional cylindrical pin fin with tip convective effects," Heat Transfer Engineering, 1999, Vol.20, pp.57 -63.
4. Ma S. W., Behbahani A. I. and Tsuei Y. G., Two-dimensional Rectangular Fin with Variable Heat Transfer Coefficient, Int. J. Heat.
5. S.P.Chauhan and Dr. S. Singh, Steady and Unsteady State Thermal Analysis of Fins Using Cfd, International Journal of Innovative Research and Studies, vol 2 issue 5, ISSN 2 319-9725, pp.519-526
6. Heat Transfer through fins, Fluent Inc. (Flow Lab 1-2), April 12, 2007
7. Finite Element Analysis and Experimental Study of Convective Heat Transfer Augmentation from Horizontal Rectangular Fin by Triangular Perforations" written by Kumbhar D.G., Dr. N.K. Sane, Chawan S.T.
8. [www.wikipedia.com](http://www.wikipedia.com)
9. [www.buzzle.com](http://www.buzzle.com)
10. [www.wiki.answers.com](http://www.wiki.answers.com)
11. [www.math.about.com](http://www.math.about.com)