

FLOW ANALYSIS ON AN ATMOSPHERIC RE-ENTRY CAPSULE

Dissertation submitted

in partial fulfilment of requirements

for the award of degree of

**Bachelor of
Technology in
Mechanical Engineering**

by

Saroj Khan (Regd. No: 18135A0344)

Shaik Thanveer Basha (Regd. No: 17131A03J1)

Sobha Srujana Patri (Regd. No: 17131A03J5)

Under the guidance of

Sri Y PRASAD REDDY

Assistant Professor, Department of Mechanical Engineering



Department of Mechanical Engineering

Gayatri Vidya Parishad College of Engineering (Autonomous)

Visakhapatnam-530 048

CERTIFICATE

This is to certify that the project titled **FLOW ANALYSIS ON AN ATMOSPHERIC RE-ENTRY CAPSULE** is a bonafide record of the work done by

Saroj Khan (Regd. No. 18135A0344)

Shaik Thanveer Basha (Regd. No. 17131A03J1)

Sobha Srujana Patri (Regd. No. 17131A03J5)

in partial fulfillment of the requirements for the award of degree of

Bachelor of Technology in Mechanical Engineering

of the Gayatri Vidya Parishad College of Engineering (Autonomous) affiliated to Jawaharlal Nehru Technological University, Kakinada during the year 2017-2021.

(Name & Signature of the Supervisor)

(Name & Signature of the HoD)

Project Viva-Voce held on _____

External Examiner

ABSTRACT

Atmospheric re-entry refers to the movement of human made objects as they enter the atmosphere of a planet from outer space. Re-entry modules are blunt-bodies designed to withstand high heating loads experienced during entry into the atmosphere. Here conduct an external flow analysis on atmospheric re-entering vehicle called Apollo AS-202 developed by NASA. Computational fluid dynamics is used to obtain the flow field that develops around re-entry capsules. To evaluate the heat flux variation, velocity profile, temperature variation and pressure distribution at various locations of the capsules are presented. By specifying the appropriate boundary conditions, one can modify the speed and of the re-entry vehicle. It accounts for changes in temperature, density, and pressure of the surrounding atmosphere, and even includes viscous effects and shock waves. The analysis is carried out for turbulent flow and standard flow properties available for re-entry capsules in the literature using Navier-Stokes solver for different Mach numbers.

KEY WORDS: Atmospheric re-entry, Aerodynamic Heating, Angle of attack, CFD, Heat fluxes, Hypersonic Flow, Re-entry vehicles, Thermal protection system.

ACKNOWLEDGEMENT

It is with a sense of great respect and gratitude that we express my sincere thanks to , **Y Prasad Reddy** ,Assistant Professor, Department of Mechanical Engineering, Gayatri Vidya Parishad College of Engineering (Autonomous) for his inspiring guidance, supervision and encouragement towards the successful completion of my project work.

We take this opportunity to thank **Dr. B. Govinda Rao**, Professor and Head of the Department of Mechanical Engineering, Gayatri Vidya Parishad College of Engineering(A) for permitting us to pursue the project.

We express our sincere thanks to **Dr. A. Bala Koteswara Rao**, Principal of Gayatri Vidya Parishad College of Engineering(A) for granting permission for providing all the necessary resources for completing this project.

We wish to express our appreciation and heartfelt thanks to our parents who supported us towards our goal and we would like to thank our friends, who helped and inspired us in odd and even hours for successful completion of our project work.

We would like to convey special thanks to all those who helped either directly or indirectly for the completion of our project work.

Saroj Khan (Roll No. 18135A0344)

Shaik Thanveer Basha (Roll No. 17131A03J1)

Sobha Srujana Patri (Roll No. 17131A03J5)

CONTENTS

	Page No.
<i>Certificate</i>	i
<i>Abstract</i>	ii
<i>Acknowledgement</i>	iv
<i>Contents</i>	v
<i>List of figures</i>	vii
<i>List of Tables</i>	ix
<i>List of graphs</i>	ix
Chapter-1 INTRODUCTION	
1.1 Introduction	1
1.2 Need of Space Capsule	2
1.3 Need for CFD Analysis	3
1.4 Aerodynamic Heating	3
1.4.1 Protection from Aerodynamic Heating	4
1.5 Methodology	4
1.6 AS-202 Flight Data	4
Chapter-2 LITERATURE REVIEW	6
Chapter-3 PROCEDURE	
3.1 Modeling	8
3.2 Steps involved to create this model	8
3.3 ANSYS Procedure	13

Chapter -4 RESULTS AND DISCUSSIONS

4.1. Pressure Variation	23
4.1.1 At 54.6 km Altitude	23
4.1.2 At 70.0 km Altitude	24
4.1.3 At 77.2 km Altitude	25
4.2 Mach Number Variation	27
4.2.1 At 54.6 km Altitude	27
4.2.2 At 70.0 km Altitude	28
4.2.3 At 77.2 km Altitude	29
4.3 TEMPERATURE VARIATION	31
4.3.1 At 54.6 km Altitude	31
4.3.2 At 70.0 km Altitude	32
4.3.3 At 77.2 km Altitude	33

Chapter-5 CONCLUSION

5.1 Conclusions	36
5.2 Scope for Future Study	36

REFERENCES	37
-------------------	-----------

LIST OF FIGURES

Fig 1.1 Apollo Reentry Capsule	3
Fig 1.2 Capsule Dimensions	5
Fig 3.1 Basic sketch of capsule and modify the capsule as per dimensions.	12
Fig 3.2 Create solid model of capsule	13
Fig 3.3 Enclosure around the capsule	14
Fig 3.4 Subtract solid from fluid domain using Boolean operation	15
Fig 3.5 Meshing Operation	16
Fig 3.6 Sectional view of meshing	17
Fig 3.7 Create named selections to different boundaries	17
Fig 3.8 Insert domains	18
Fig 3.9 Assigning materials	18
Fig 3.10 RNG K- ϵ model was taken in turbulent model	19
Fig 3.11 Inlet boundary conditions	19
Fig 3.12 Outlet boundary conditions	20
Fig 3.13 Assigning min and max. Iterations	20
Fig 3.14 Considering pressure interpolation type (linear-linear)	21

Fig 4.1.1	Pressure variations at 54.6km altitude	23
Fig 4.1.2	Pressure variations at 70.0km altitude	24
Fig 4.1.3	Pressure variations at 77.2km altitude	25
Fig 4.2.1	Mach numbers variations at 54.6 km altitude	27
Fig 4.2.2	Mach numbers variations at 70km altitude	28
Fig 4.2.3	Mach numbers variations at 77.2km altitude	29
Fig 4.3.1	Temperatures variations at 54.6 km altitude	31
Fig 4.3.2	Temperatures variations at 70 km altitude	32
Fig 4.3.3	Temperatures variations at 77.2 km altitude	33

LIST OF TABLES

Table 1.1	Considering values of Altitude and Velocity	5
Table 1.2	Variation of pressure, temperature and Mach number at different altitudes	35

LIST OF GRAPHS

Fig 4.1.4	Pressure between different altitudes	26
Fig 4.2.4	Velocity between different altitudes	30
Fig 4.3.4	Mach number between different altitudes and pressure	34
Fig 4.4.4	Temperature between different altitudes	35

CHAPTER 1

INTRODUCTION

1.1 INTRODUCTION

The general objective of the European Space Agency FLPP program (Future Launchers Preparatory Programmed, [1]) is placing Europe inside the worldwide strategic area of atmospheric reentry for future international transportation, exploration and scientific projects. Several studies on experimental vehicle concepts and improvements of critical reentry technologies have been undertaken in recent years by ESA (ARD), France (Pre-X), Germany (Phoenix) and Italy (USV), in order to consolidate their worldwide position in this strategic field. The aero thermodynamic studies can support and address both aero shape consolidation and mission analysis, whose goal is to minimize the heat fluxes to the windward part of the vehicle (nose cap) and control surfaces (flaps). Body-flap efficiency prediction and environment characterization have been carried out in recent past with both thermo-chemical equilibrium and non-equilibrium flow assumption. A preliminary benchmarking phase has been performed in order to assess the numerical strategy to be adopted for the scheduled activity concerning both laminar and transitional simulations to be performed with different flow field hypotheses (chemical equilibrium and non-equilibrium), and also accounting for the effects of the angle of attack, the angle of sideslip, the Mach number (i.e. total enthalpy) and the symmetric and asymmetric flap deflection.

The numerical code used to carry out the aerothermodynamics analysis of the IXV vehicle is the CIRA code H3NS that solves the Reynolds Averaged Navier Stokes equations in a density-based finite volume approach with a cell centered, Flux Difference Splitting second order ENO-like upwind scheme for the convective terms. The need for a safer access to space dictates the review of operational capabilities and hence of design approach for manned reentry vehicles of next generation. Research has shown that reentry vehicle designs with high L/D could be designed to take

advantage of aerodynamic lift during reentry. Higher L/D is desirable because it increases the area from which a re-entering vehicle can be recovered (e.g. reentry window). Keeping constant the temperature of the nose stagnation in the radiation equilibrium conditions, restricting the g peak experienced to less than a tenth above normal ground-level values and, finally, with a wider than usual “reentry window” that would permit landing at any one of the many choices of airfields. The results, provided in this paper, consider a Mars entry scenario compliant with an approach to the red planet both by elliptic and hyperbolic orbit. These results may be used to provide numerical data for understanding requirements for the human exploration of Mars.

From the point of view of approach strategies, the different values of velocity at entry interface (given the entry angle) will characterize the MBS design by means of mechanical loads (i.e. pressure and acceleration), thermal loads (i.e. heat flux peak and integrated heat load), and landing dispersion. The flow around the capsules was computed by the DSMC method. The number of collisions was calculated by the majorant frequency technique. The collisions of molecules were computed using the variable hard sphere (VHS) model. The internal degrees of freedom were taken into account by the Larsen – Borgnakke model. The SMILE software system was used for computations.

A number of papers have been already written about EXPERT with different aims, from the evaluation of the aerodynamic behavior to the description of tests and experiments to be made during the re-entry. Preliminary computations of aerothermo-dynamic data base at high altitudes were provided by approximate engineering methods or bridging formulae. The aim of the present work is making an additional analysis and, hopefully, a better characterization of the aerothermo-dynamic data base in rare field regime.

1.2. NEED OF SPACE CAPSULE

A **space capsule** is an often manned spacecraft which has a simple shape for the main section, without any wings or other features to create lift during atmospheric reentry. Capsules have been used in most of the manned space programs to date, including the world's first manned spacecraft Vostok and Mercury, as well as in later Soviet Voskhod, Soyuz, Zond/L1, L3, TKS, US Gemini, Apollo Command Module, Chinese Shenzhou and US, Russian and Indian manned spacecraft

currently being developed. A capsule is the specified form for the Orion Multi-Purpose Crew Vehicle.



Fig 1.1 Apollo Reentry Capsule

1.3 NEED FOR CFD ANALYSIS

- When a capsule reenters an atmospheric environment, a strong shock wave is formed in front of it. Behind the shock wave, a shock layer with very high temperature appears where a high enthalpy fluid flows around a capsule, resulting in a severe heating environment.
- Moreover, in an environment where the capsule velocity exceeds 8 km/s such as a super-orbit re-entry, there appear complicated phenomena accompanied by the radiation and/or the influence of turbulence.
- The computed results are utilized to determine whether the aero-thermodynamic loads exceed the allowable values. If the loads are exceeding, then an optimized design is required to account for these loads. Moreover, if it is not exceeding, further analysis is done on the other components to ensure their reliability.

1.4 AERODYNAMIC HEATING

Atmospheric re-entry vehicles are subjected to aerodynamic heating during re-entry phase of their operation. Aerodynamic heating is the heating of a solid body produced by the passage of fluid

over the body. It is a form of forced convection in that the flow field is created by forces beyond those associated with the thermal processes. This process generates heat and consequently all external surfaces of the vehicle are heated. Due to aerodynamic heating external surfaces of the re-entry vehicle gets heated. Thermal Protection Systems are necessary in order to protect the internal structure of the vehicle from the elevated heat fluxes occurring on the external surfaces. The design of a Thermal Protection System is based on the principle that the energy released by the aerodynamic heating must be absorbed or rejected by the Thermal Protection System.

1.4.1 PROTECTION FOR CAPSULE FROM AERODYNAMIC HEATING

- Thermal Protection Systems are necessary in order to protect the internal structure of the vehicle from the elevated heat fluxes occurring on the external surfaces.
- The design of a Thermal Protection System is based on the principle that the energy released by the aerodynamic heating must be absorbed or rejected by the Thermal Protection System.

1.5 METHODOLOGY

- Model of the capsule and fluid flow domain are created in design modeler of ANSYS workbench as per the dimensions.
- Meshing is done using ANSYS ICEM mesh module
- CFD analysis is done by using ANSYS CFD code CFX taking high velocity aerodynamic heating model by setting the environmental process and design parameters.
- Post processing the solution to get the results.

1.6 AS-202 Flight Data

The flight data used for assessment/comparison of heat flux data on the capsule were taken from the AS-202 flight test which was performed as part of the Apollo program. Once the Apollo entry vehicle design was determined, two flight tests of the actual Command Module (AS-201 and AS-202) were conducted at super orbital entry velocities resulting from suborbital boosted trajectories

with an intentional skip maneuver. Although AS-201 did not carry an on board inertial measurement unit (IMU), one was carried during the AS-202 flight, which enabled a reconstruction of the flight trajectory and vehicle orientation as a function of time.

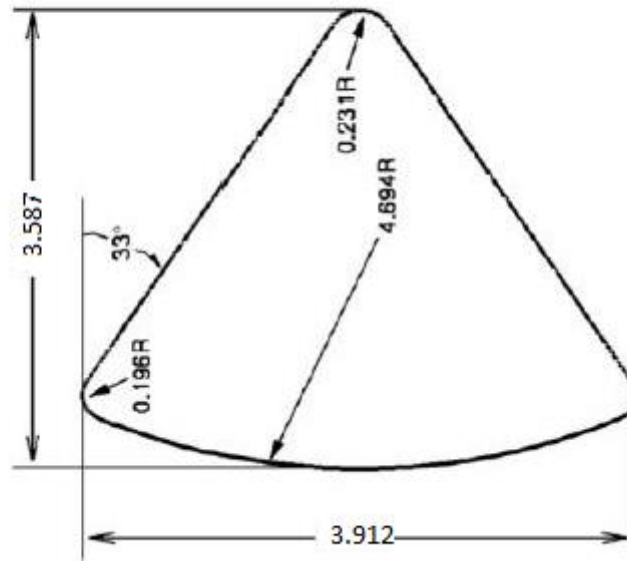


Fig 1.2 Capsule Dimensions

ANSYS FLUENT SETUP is opened to define the problem for the flow analysis with inlet velocities 6.49, 7.92, 5.09 km/s at the altitude is 77.2, 70.0 and 54.6 respectively. Inlet temperature(245.5k) was considered constant at all altitudes since it is negligible when compared to generated temperatures at capsule-air interface, the static pressure is $P = 130.5\text{Pa}$.

Case	Altitude (km)	Velocity (km/s)
1	77.2	6.49
2	70	7.92
3	54.6	5.09

Table 1.1 Considering the values of Altitude and Velocity

CHAPTER 2

LITERATURE REVIEW

Ernest R Hillje

In this work, a Hypersonic Inflatable Aerodynamic Decelerator (HIAD) concept is studied from the aerothermodynamics point of view using computational fluid dynamics. Using a commercial CFD software for solving Navier-Stokes equations, the heat flux of and the aerodynamic forces acting on the reentry vehicle with and without inflatable aerodynamic decelerator are calculated. Results have shown considerable effect of inflatable aerodynamic decelerator on the aerothermodynamic parameters of reentry capsule. Trajectory Points And Free stream Conditions from which we have selected the points like Altitude and Velocity. Entry flight aerodynamics from Apollo Mission AS-202.

KrishnenduSinha

Hypersonic flows are characterized by high Mach number and high total enthalpy. An elevated temperature often results in thermo-chemical reactions in the gas, which play a major role in aerothermodynamic characterization of high-speed aerospace vehicles. Hypersonic flows in propulsion components are usually turbulent, resulting in additional effects. Computational simulation of such flows, therefore, need to account for a range of physical phenomena. Further, the numerical challenges involved in resolving strong gradients and discontinuities add to the complexity of computational fluid dynamics (CFD) simulation. In this article, physical modeling and numerical methodology-related issues involved in hypersonic flow simulation are highlighted. State-of-the-art CFD challenges are discussed in the context of two prominent applications the flow in a scramjet inlet and the flow field around a re-entry capsule

Dr.B.Balkrishna

The main goal of the work described in this paper was to setup a procedure for modeling a thermal production system for hypersonic reentry vehicle by the author. A Multiphysics framework has been setup for the simulation of hypersonic reentry vehicle using commercial codes CFD and FEA with user defined programming CFD (FLUENT) and the material thermal and structural response code (ANSYS) are loosely coupled to achieve the solution. The evaluation of the aero thermodynamics analysis of reentry trajectory. CFD results are presented to show the flow field around a capsule in hypersonic flow. FEA results are presented to show the stress around a capsule in hypersonic flow. CFD analysis represents a key technology within planetary entry vehicle design. Safe landing of vehicles re-entering from space requires, in fact, an accurate understanding of all physical phenomena that take place in the flow field past the hypersonic vehicle to assess its aerodynamics and aerothermodynamics performance. These parameters are of primary relevance for the design of reentry trajectory and vehicle thermal protection system; for the latter, in particular, it is pointed out to role catalyticity on the vehicle thermal load. In this framework, a possible earth-entry scenario for the proposed capsule type vehicle is reported and analyzed.

ROY N MATHEWS

In his work he conducted an external flow analysis on atmospheric re-entering vehicle called Apollo AS-202 which was developed under the NASA and thus provided with standard Schematic drawing of the outer mould line of AS-202 capsule which we will be using to create the 3D structural geometrical model for our flow analysis around the surface of the capsule to generate simulation depicting the flow of various parameters around the capsule inside the fluid domain.

.

CHAPTER 3

PROCEDURE

3.1 MODELING

The Apollo Command Module essentially consisted of a spherical section fore body and a 330 conical after body. The CM capsule was a 330 half-angle cone with the blunt after heat shield formed from a segment of a sphere of radius 4.694 m. A toroidal section with radius of 0.196 m provided the transition between the conical and spherical sections. The maximum capsule diameter of 3.91 m occurred in the toroidal section. To account for the fact that air flows around the launch vehicle, the area surrounding the re-entry vehicle model is meshed, rather than the re-entry vehicle itself.

The geometry of the capsule is quite complex and the solid modeling is carried by ANSYS DESIGN MODELER modeling tools. The dimensions taken for the capsule as for the figures 1. The model of the Capsule is modified slightly to do the flow analysis. Here the analysis the flow over the capsule so requires the flow domain for the flow analysis. Therefore for flow analysis, a flow domain is created as for the dimensions required. Here creating the rectangular shape domain with 2L on both sides 4L on bottom and 5L on top of the capsule. Where L is the length of the Capsule. Then subtract the main body form the flow domain. Before starting the mesh need to create the boundary layer around the capsule body.

3 dimensional tetrahedral mesh model was created in ANSYS ICEM CFD using different mesh control techniques like element size=100mm, inflated for 20 layers around the interface with growth rate 1.

3.2 THE STEPS INVOLVED TO CREATE THIS MODEL ARE AS FOLLOWS

Click on Geometry →select the xy-plane →select the line command to sketch capsule →select the system of units in mm

The geometry of the capsule is quite complex and the solid modeling is carried by ANSYS DESIGN MODELER modeling tools. The dimensions taken for the capsule for the figures 1. The model of the Capsule is modified slightly to do the flow analysis. Here the vertical length of the capsule is $L=3587\text{mm}$, the breadth of the capsule $b=3912\text{mm}$, the radius of two corners is $R1=0.196$ and the noise radius of capsule $R2=0.231$.

Here the analysis the flow over the capsule so requires the flow domain for the flow analysis. Therefore for flow analysis, a flow domain is created as for the dimensions required. Here creating the rectangular shape domain with $2L$ on both sides $4L$ on bottom and $5L$ on top of the capsule.

- $L=3587\text{mm}$
- $2L=3587*2=7174\text{mm}$
- $4L=3587*4=14348\text{mm}$
- $5L=3587*5=17935\text{mm}$

where L is the length of the Capsule. Then subtract the main body from the flow domain. Before starting the mesh need to create the boundary layer around the capsule body by using Boolean option .therefore fluid domain and capsule bodies parameters before substrate

- Volume of the 2 bodies $V=1.1601\text{e}+0.13\text{mm}^3$,
- Surface area $A=3.2909\text{e}+0.09\text{mm}^2$,
- Total no faces selected =14,
- No. of edges =18,
- No .of vertices=8.

From the analysis system the fluid flow (CFX) operation after subtract. In this individual fluid domain body is consider as fluid flow and the parameters is

- Volume $V = 1.586\text{e}+0.13\text{mm}^3$
- Surface area $=3.25\text{e}+0.09\text{mm}^2$
- No of faces =10
- No of edges =15
- No of Vertices=8

According to capsule parameters

- Volume = $1.456 \times 10^{10} \text{mm}^3$
- Surface area = $3.3904 \times 10^7 \text{mm}^2$
- No. of faces = 4
- No. of edges = 3
- No of vertices = 0

- 1) Then subtract the main body from the flow domain. Before starting the mesh need to create the boundary layer around the capsule body.
- 2) If a Sizing control is used on a body with hard edge sizing and the source and target faces contain hard divisions which are not the same for each respective edge. therefore we assign 8 edges bottom and top of the domain and divided into 120 divisions each one another with hard behavior, the corners of fluid domain has 4 edges is divided into 240 divisions each corner with hard behavior.
- 3) According to capsule face sizing consists of 4 faces .the element size = 0.15m and the capsule behavior consider soft.
- 4) ANSYS Workbench Meshing has several options and tools to aid in the generation of high quality inflation layers. Options such as previewing the inflation layer before it is applied to the final mesh, saves valuable time during the meshing process.
- 5) 3 dimensional tetrahedral mesh model was created in ANSYS ICEM CFD using different mesh control techniques like element size=100mm, inflated for 20 layers around the interface with growth rate 1.5 to refine the mesh with minimum orthogonal quality 0.816 and aspect ratio 4.73E-1.
- 6) Select the faces and click on right and assign name section like outlet, inlet, symmetric, capsule.
- 7) From the analysis system consider the fluid flow (CFX) module →insert→domain.
- 8) Default-domain the location is assigned B19 (capsule),the domain type fluid domain, the reference pressure is 1[atm], material type is air ideal gas, the heat transfer obtained by total energy, the turbulence flow option is RNG-K-EPSILON with high speed (compressible) wall.

- 9) The capsule boundary type condition is wall and the location is capsule.
- 10) As inlet flow regime supersonic, mass and momentum with normal speed/pressure therefore the relative static pressure = 0[pa].
- 11) The normal speed 7920 m/s, the turbulence flow as medium-intensity= 5%, heat transfer consider as static temperature =245.5k.
- 12) As outlet flow regime supersonic, mass and momentum with average static pressure, relative pressure = 0[pa] pressure.
- 13) Symmetric boundary type is wall location as symmetric, mass and momentum is no slip wall, wall roughness equal to smooth wall the transfer take place adiabatic.
- 14) Solver consists with high resolution, first order turbulence, convergence control by min. iterations=1, max. Iterations =1000.
- 15) The fluid time scale control by auto time scale,
- 16) Length scale option is conservative
- 17) Time scale factor =1.0
- 18) Expert parameter memory control to run the solution topology estimate factor value= 1.1
- 19) After the mesh of the capsule ANSYS FLUENT SETUP is opened to define the problem for the flow analysis with inlet velocities 6.49, 7.92, 5.09 km/s at the altitude is 77.2, 70.0 and 54.6 respectively. Inlet temperature(245.5k) was considered constant at all altitudes since it is negligible when compared to generated temperatures at capsule-air interface, the static pressure is $P = 130.5\text{Pa}$. After importing of the mesh file in to the FLUENT, mesh was checked for the closely accurate solution(considering PC configuration). The model type solver pressure based with viscous model as RNG K- ϵ . Ideal-gas law to determine the air density, while Sutherland's law was used to calculate the air viscosity. The operating pressure was set to 0 Pa, to decrease the chance of numerical error due to the low pressures resulting from the solution. For this problem we have taken the pressure outlet to check the mach number variations.

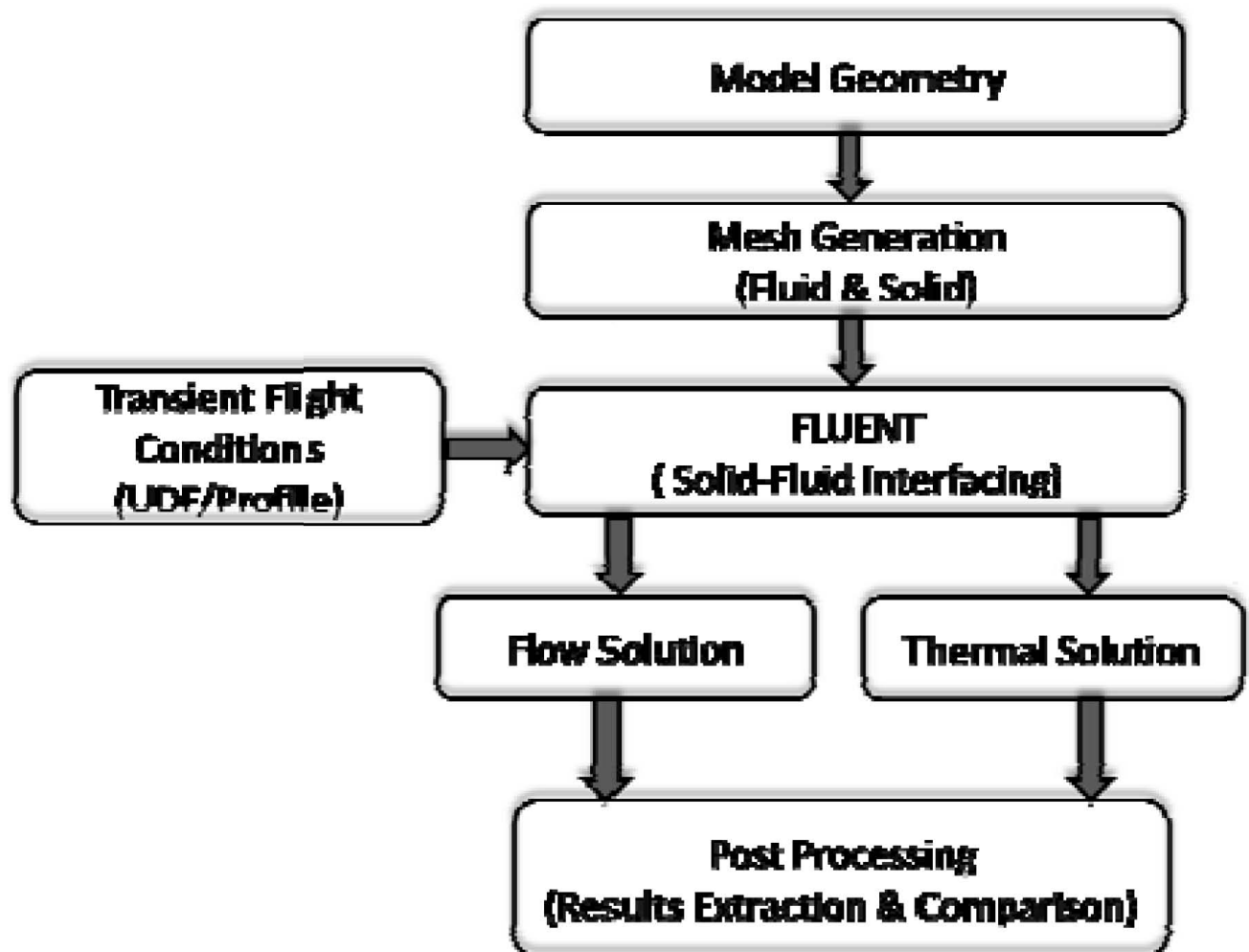


Fig. 3.1 Flow Chart for Transient Aero-thermal Analysis

3.3 ANSYS PROCEDURE

1. Open design modeler in ANSYS workbench toolbox and go to sketcher tools to develop basic sketch of capsule and modify the capsule as per dimensions.

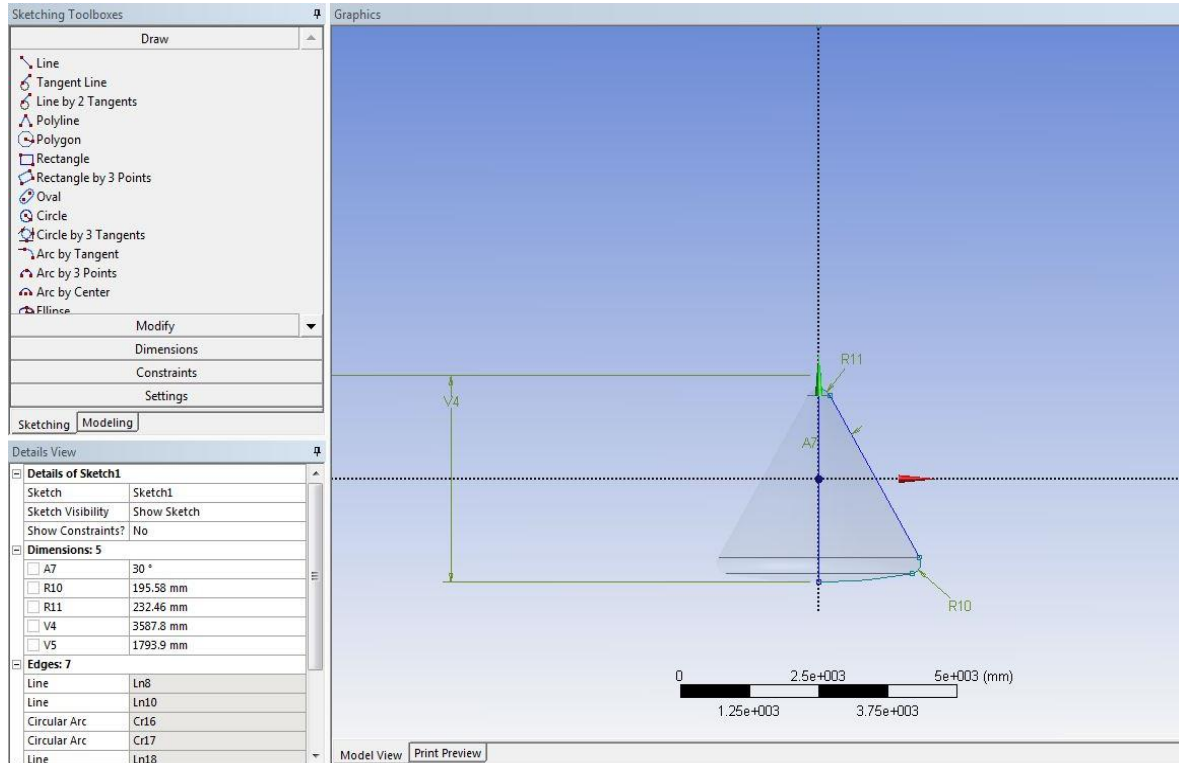


Fig.3.2 Basic sketch of capsule and modify the capsule as per dimensions.

2. After finishing required modifications go to REVOLVE option to create solid model of capsule.

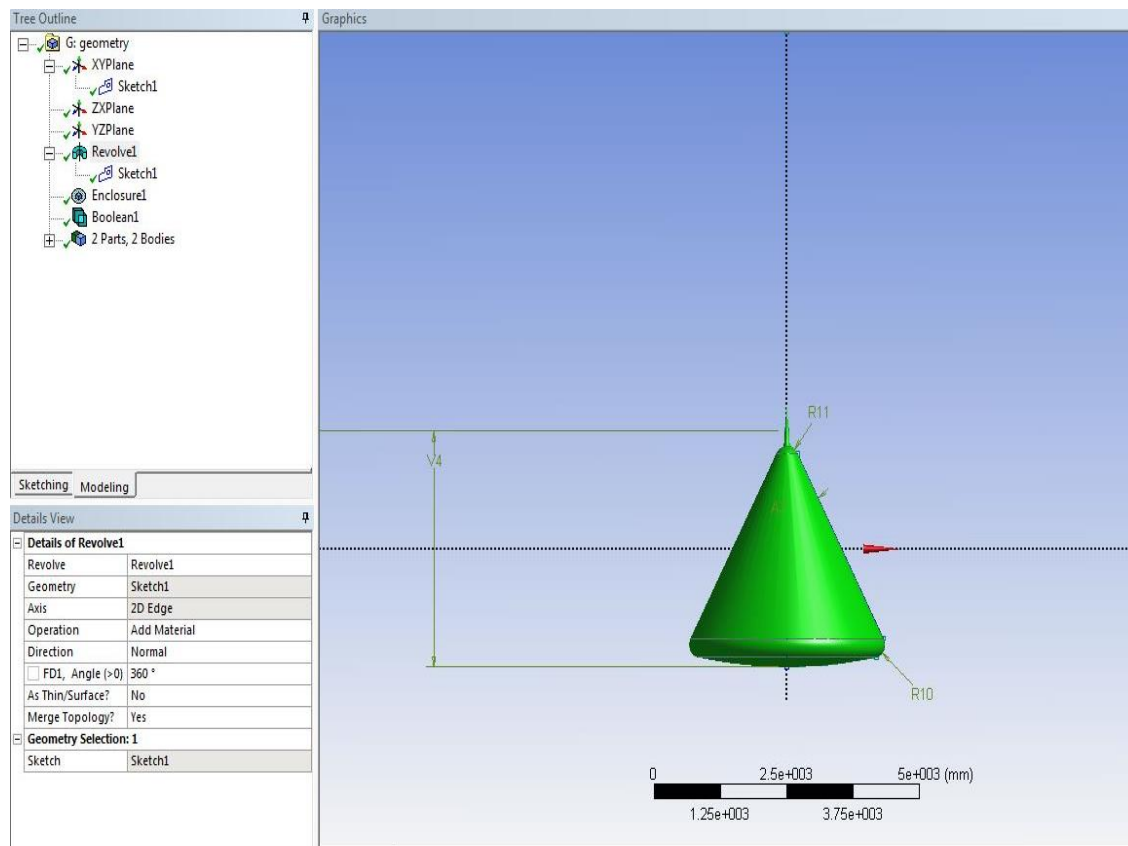


Fig.3.3 Solid model of capsule

3. Create an enclosure around the capsule to create fluid flow domain around the capsule.

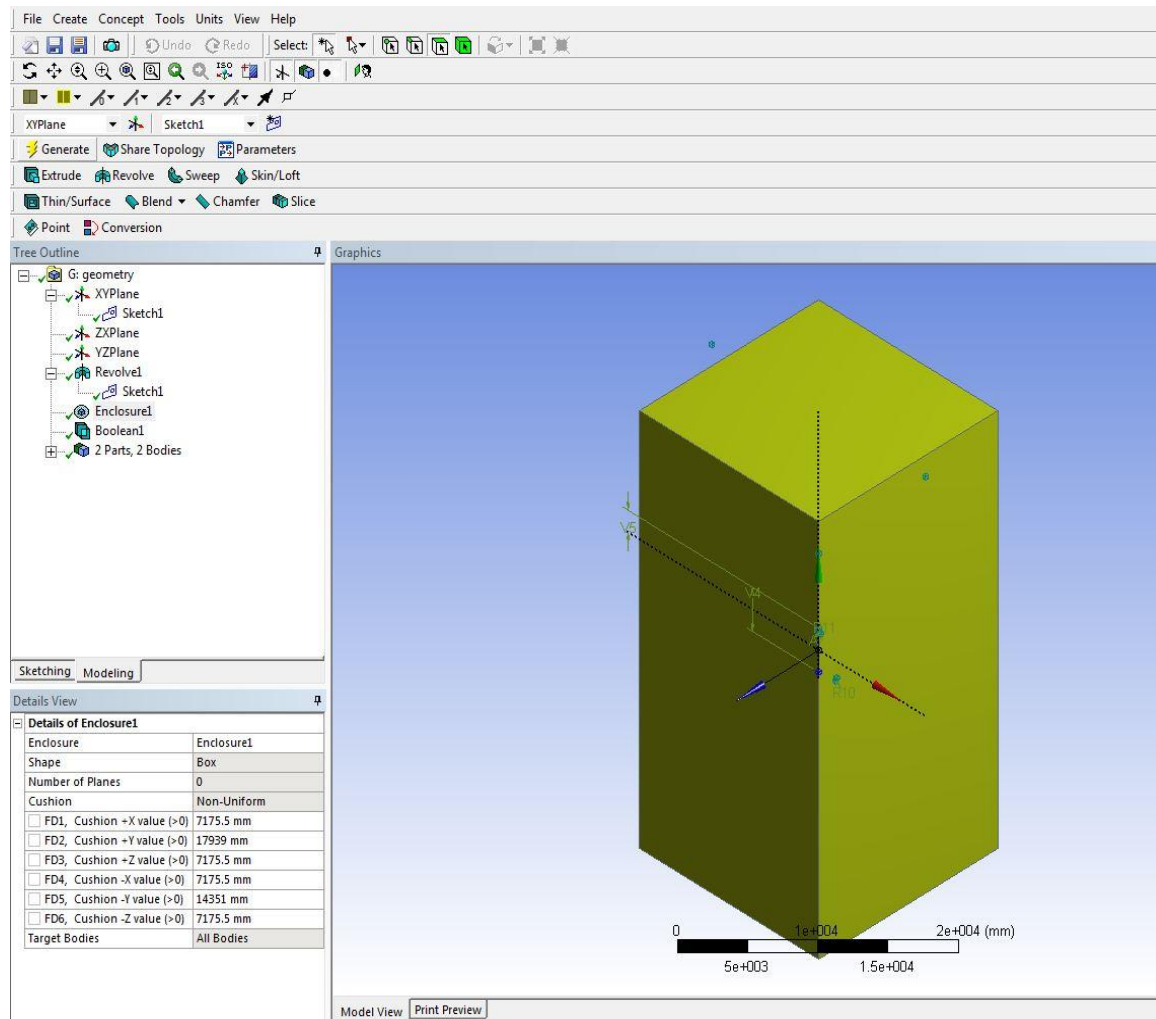


Fig.3.4 Enclosure around the capsule

4. . Subtract solid from fluid domain using Boolean operation to create separate domains for fluid flow and capsule(solid domain)

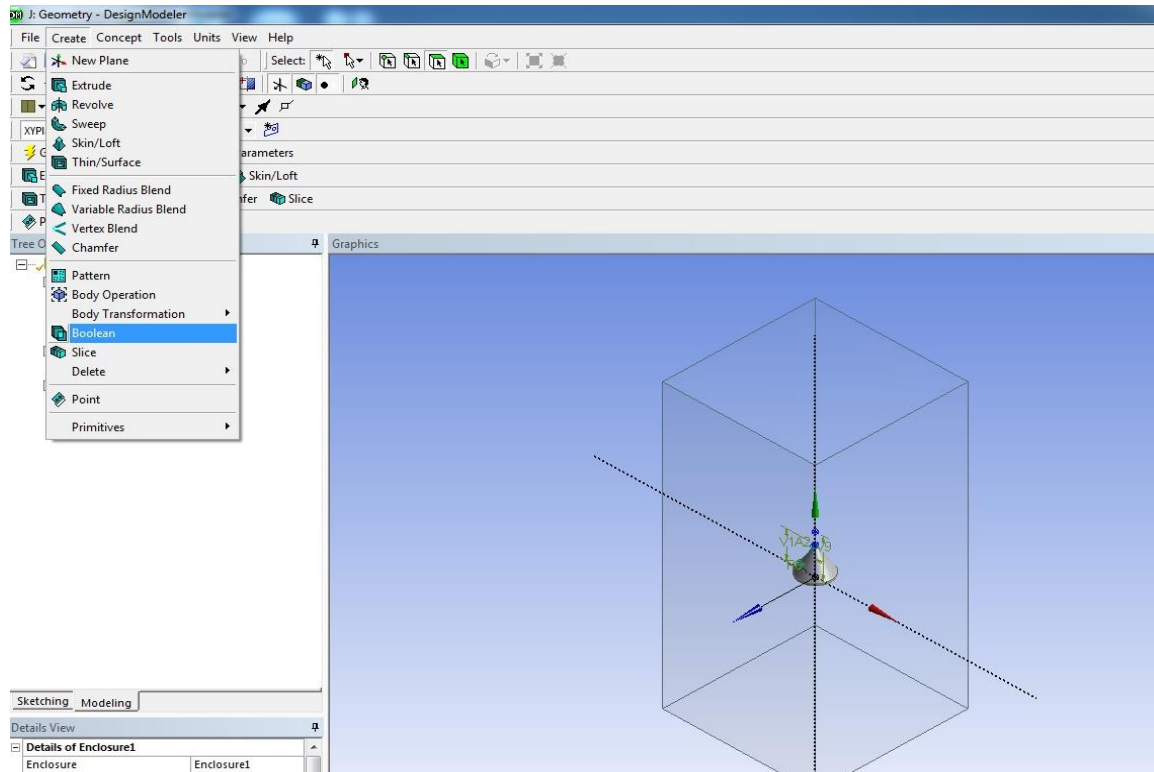


Fig.3.5 Subtract solid from fluid domain using Boolean operation

5. Open mesher to create mesh for the model and different mesh control options like edge sizing, body sizing, face sizing and inflation were used to create and refine the mesh to improve the quality of mesh that is required for CFD model.

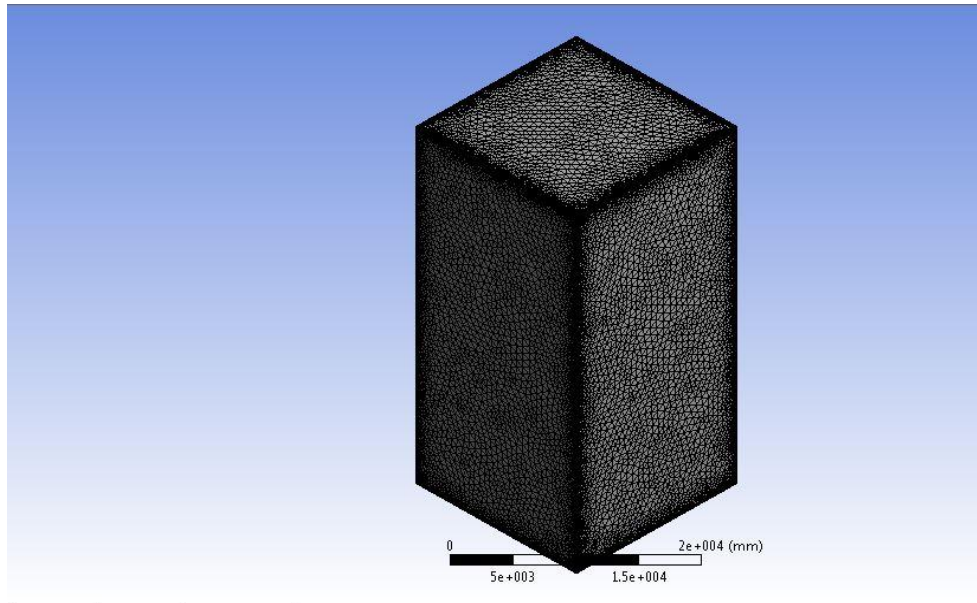


Fig.3.6 Meshing Operation

Sectional view of mesh to show inflated layers around the fluid- capsule interface.

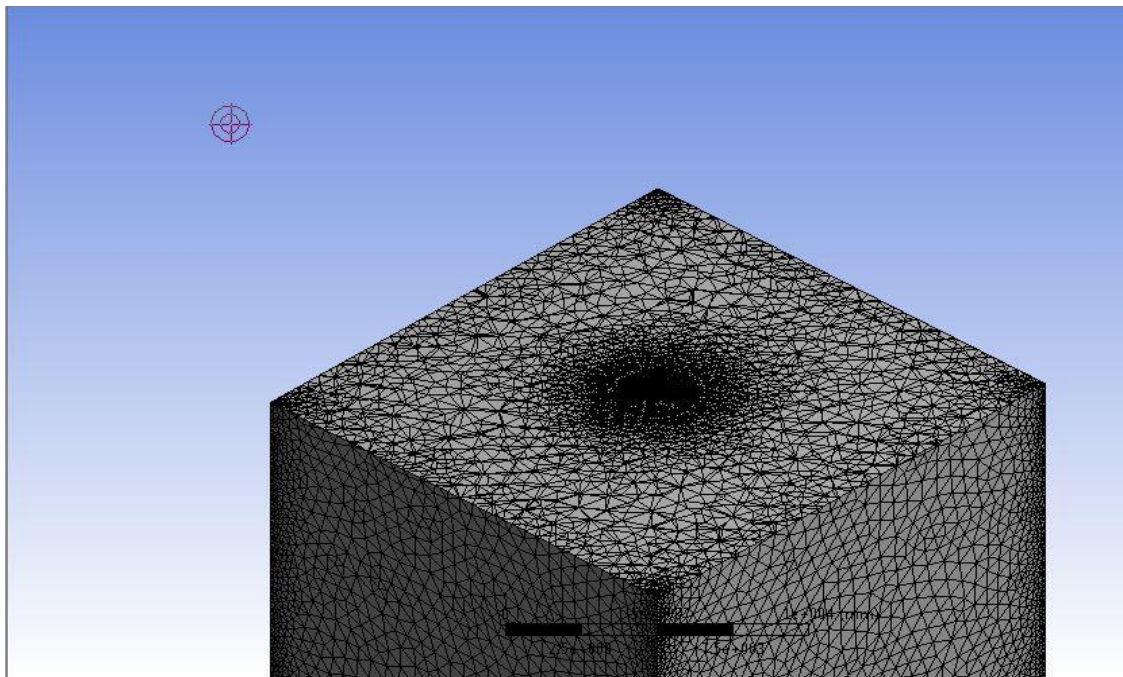


Fig.3.7 Sectional view of meshing

6. Create named selections to different boundaries for inlet, outlet, symmetries and fluid capsule interface

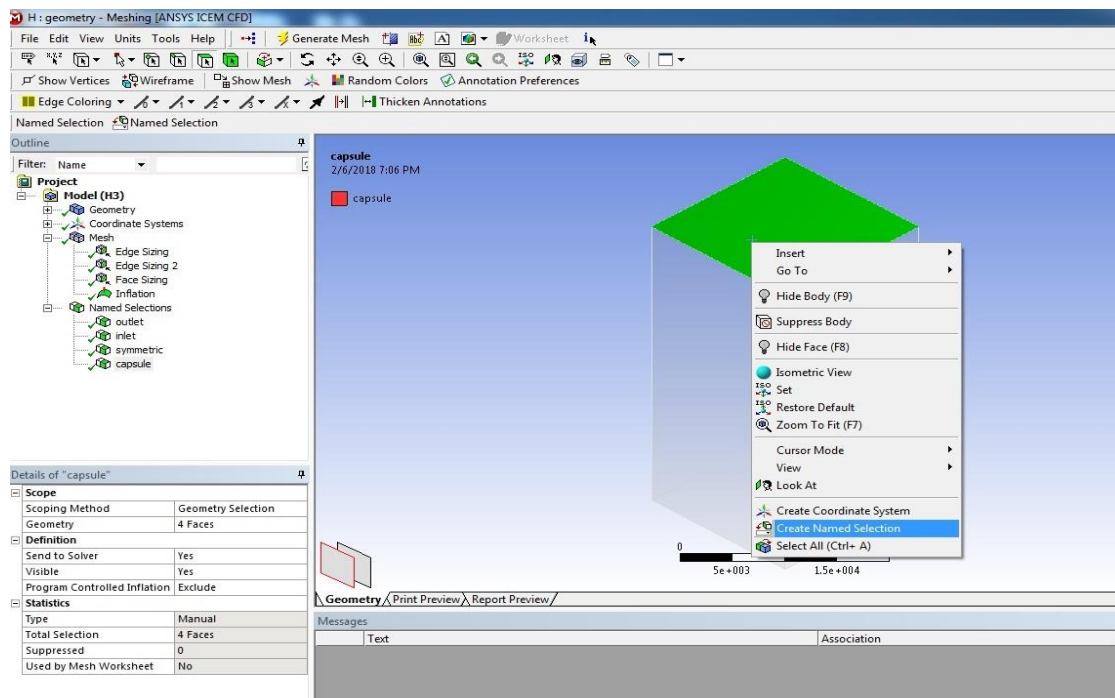


Fig.3.8 Create named selections to different boundaries

7. Open cfx-pre to define the computational problem and define the domains in CFX

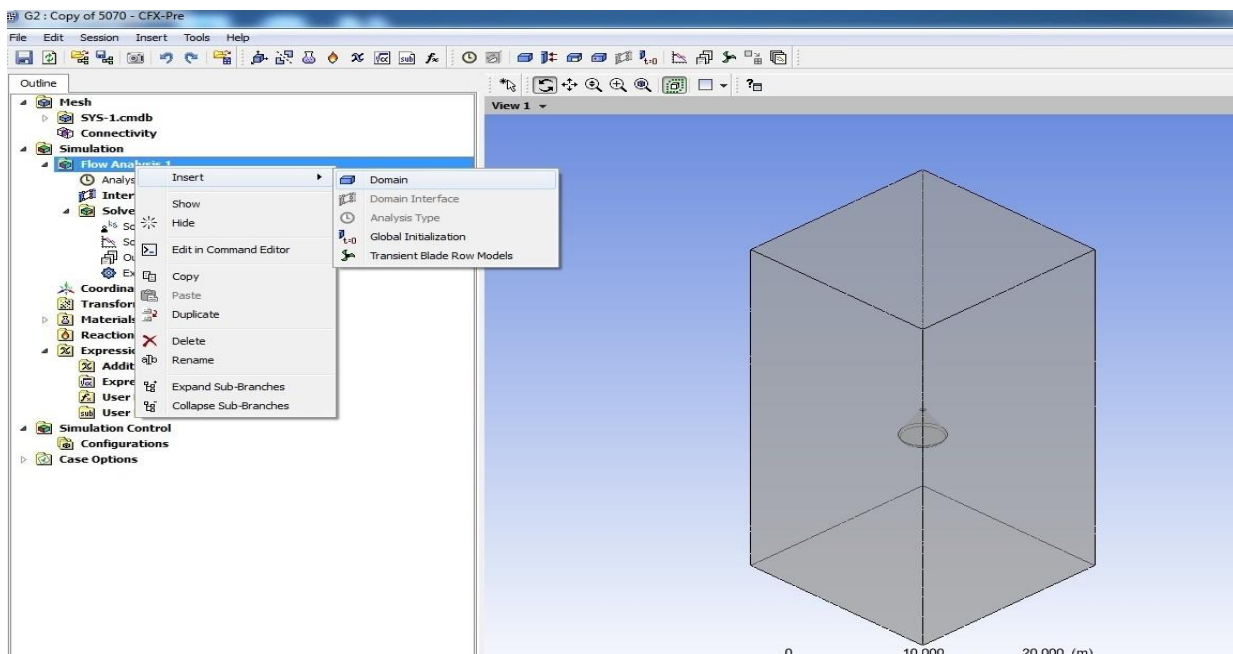


Fig.3.9 Insert domains

8. Assign materials for domains

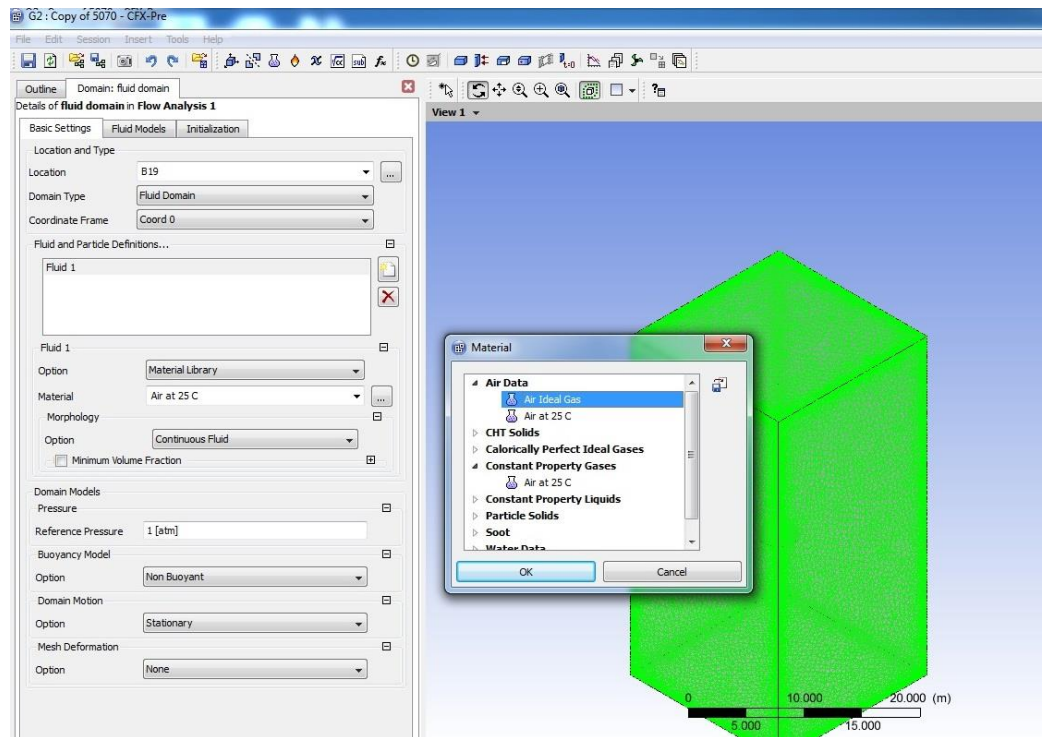


Fig.3.10 Assigning materials

- 9 Total energy was considered in energy option and RNG K- ϵ model was taken in turbulent model .

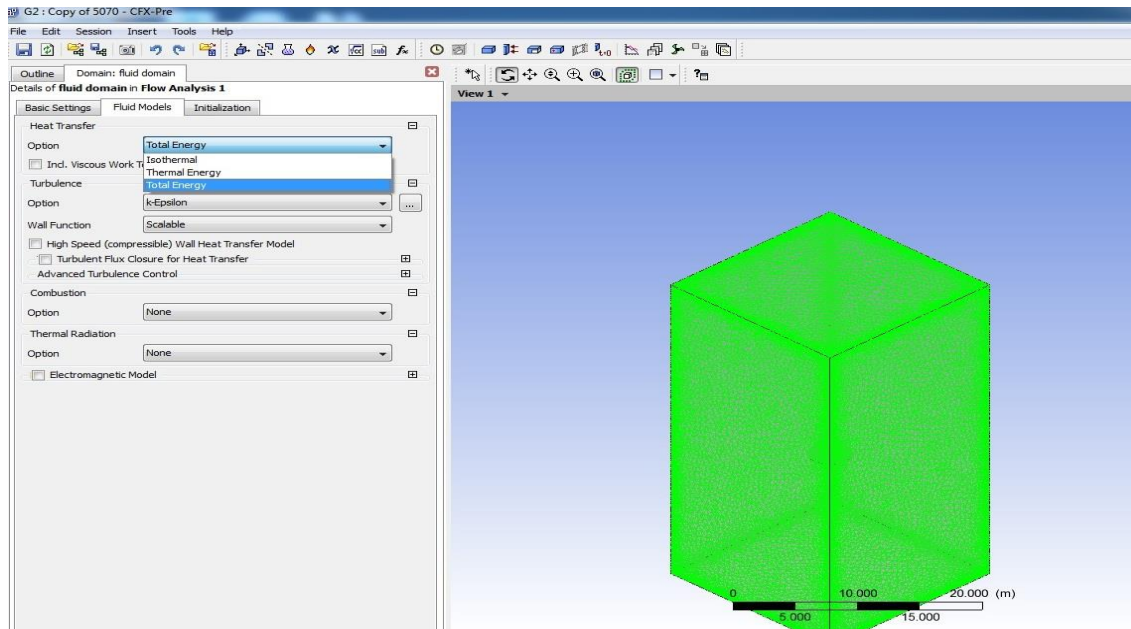


Fig.3.11 RNG K- ϵ model was taken in turbulent model

10 Define boundary conditions for inlet, outlet, symmetry and capsule-air interface

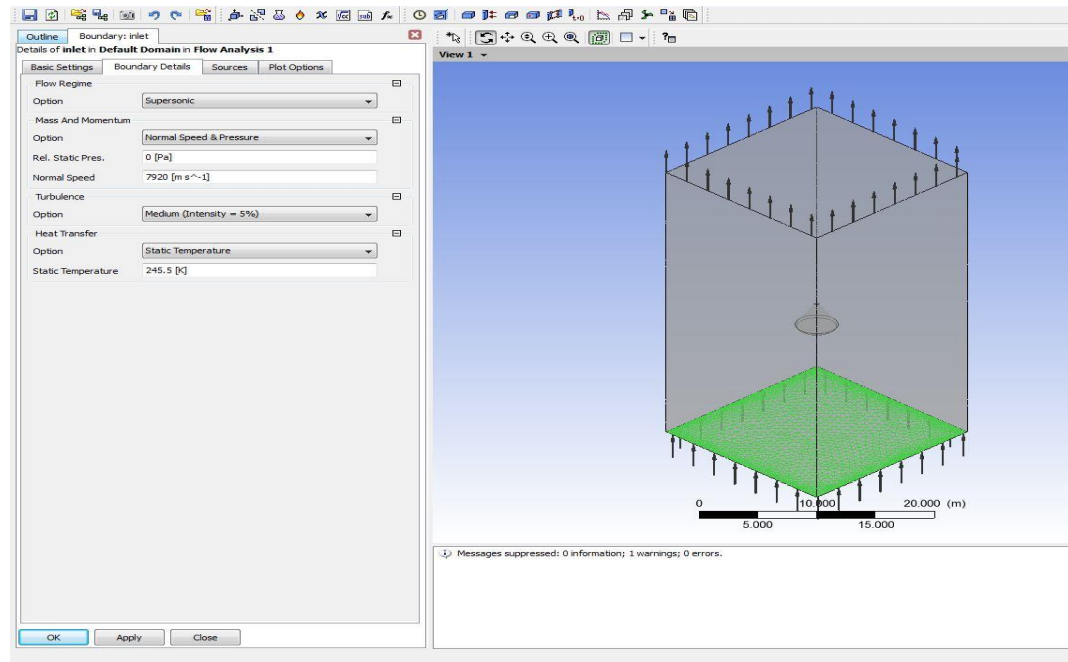


Fig.3.12 Inlet boundary conditions

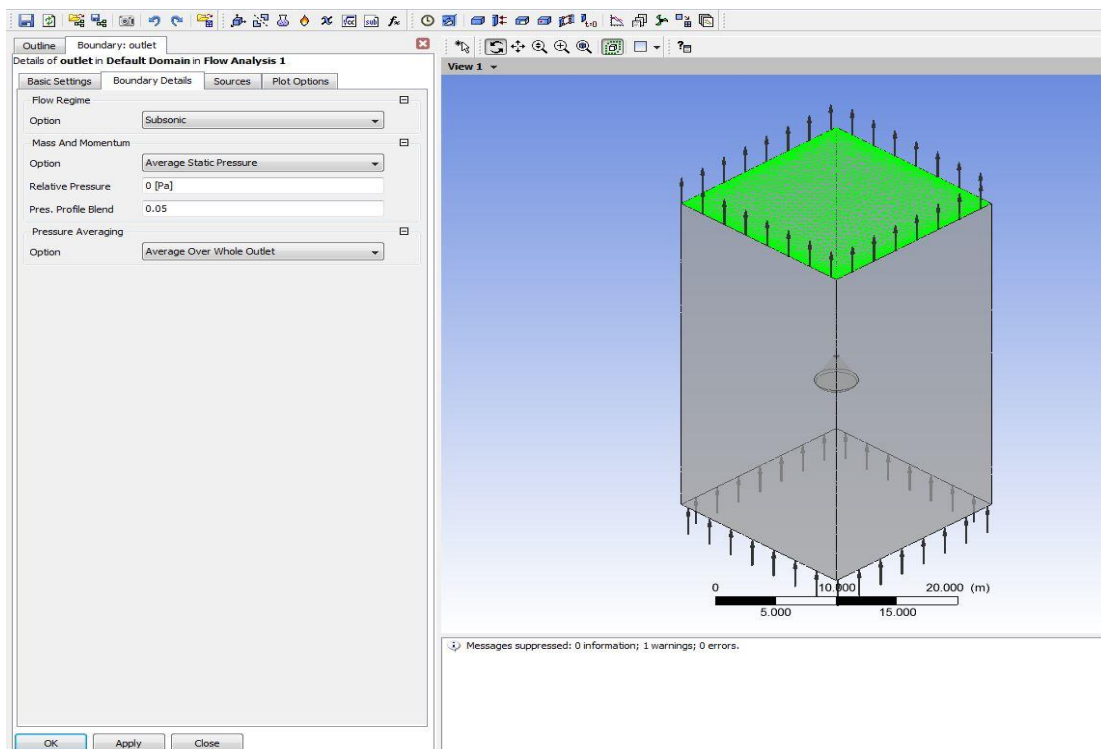


Fig.3.13 Outlet boundary conditions

11 Solver options:

11.1 Basic settings:

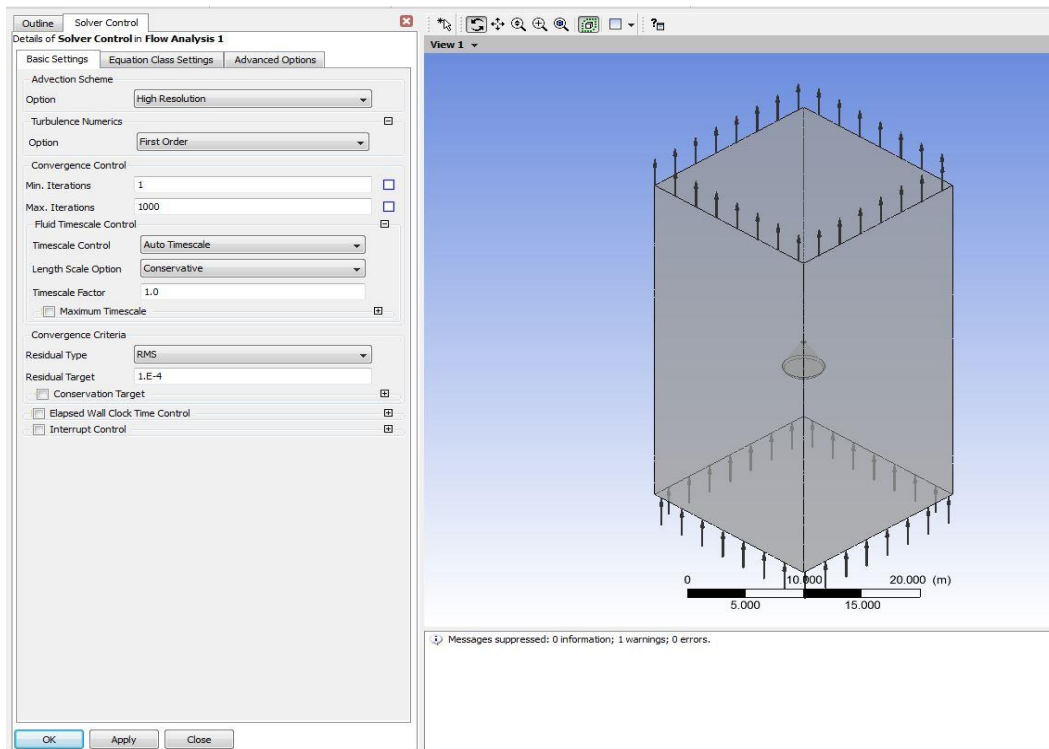


Fig.3.14 Assigning min. and max. Iterations

11.2 Advanced settings:

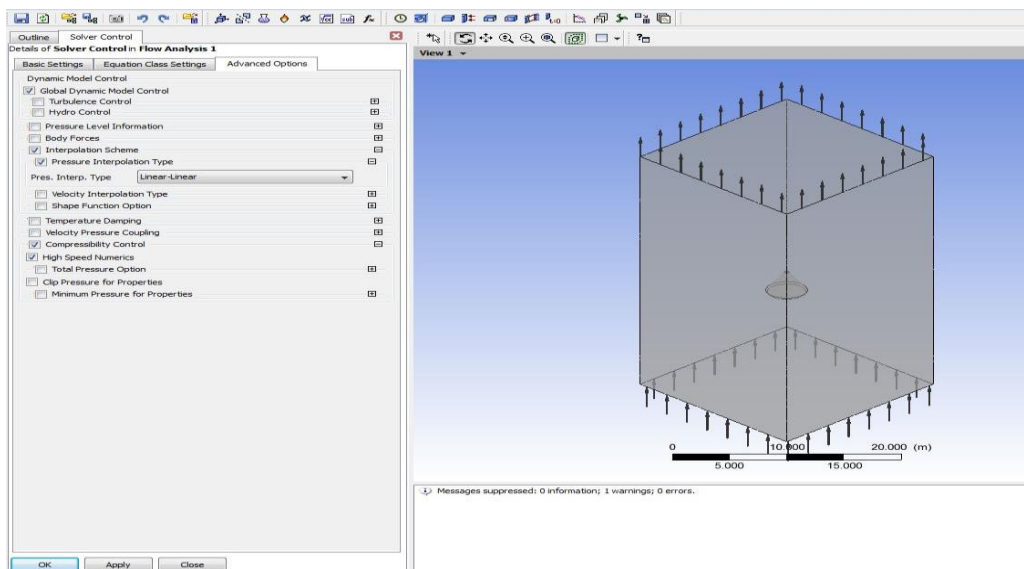


Fig.3.15 Considering pressure interpolation type (linear-linear)

CHAPTER 4

RESULTS AND DISCUSSIONS

The flow field around the Apollo-shaped body is initialized to free-stream values all over the domain. As the simulation progresses the bow shock and the boundary layer on the vehicle are formed, followed by flow separation on the after body. As the separation bubble forms on the windward side. A large recirculation bubble is formed on the leeward side and the shear layer enclosing the separation bubble coalesces at the neck, where the recompression shock is formed. While the re-entry vehicle enters into the atmosphere, a bow shock is created at the base of the Vehicle.

The pressure measurements on the conical section generally agreed with the wind-tunnel predictions. The conical pressure measurements were low during maximum heating. Maximum pressure at fore body and its value is 7.789E7 Pa. Minimum pressure is just above zero Pascal. It shows the severe pressure drag at the two edges of the module base. High static pressure is created in the base of the reentry vehicle as illustrated in Fig Since, the pressure is high while re-entering in to the atmosphere due to the strong bow shock created. This bow shock will increase drag force acting on the re-entry vehicle and has the capability to decelerate the vehicle to low Mach numbers. The maximum static pressure is created at the far field of the re-entry vehicle because of the progressing bow shocks marching downstream of the vehicle. The increase in pressure is visualized exactly using the static pressure contour for 0° angle of attack

4.1. Pressure variation:

4.1.1 At 54.6km altitude

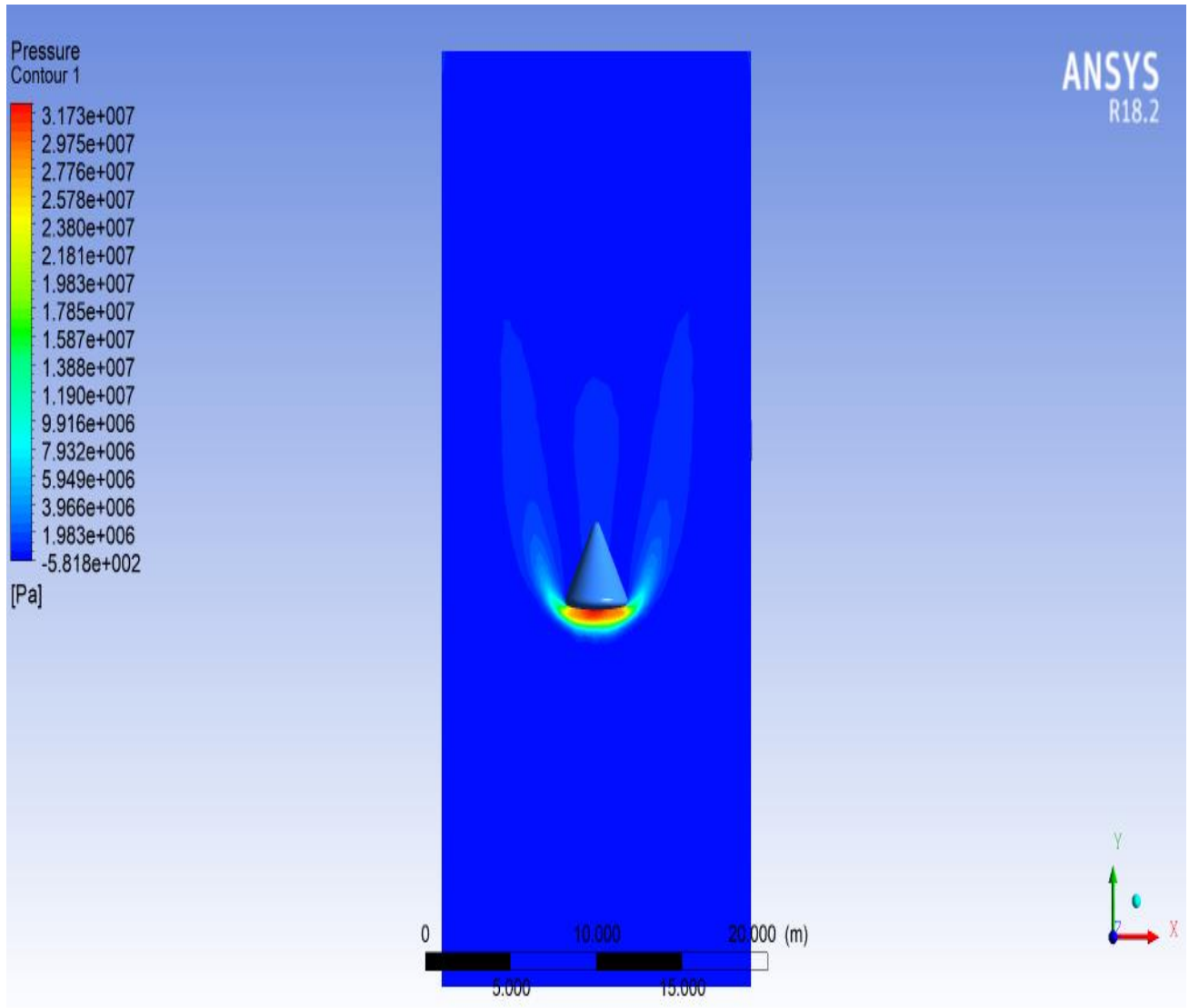


Fig.4.1.1 Pressure variations at 54.6km altitude

4.1.2 At 70.0 km altitude

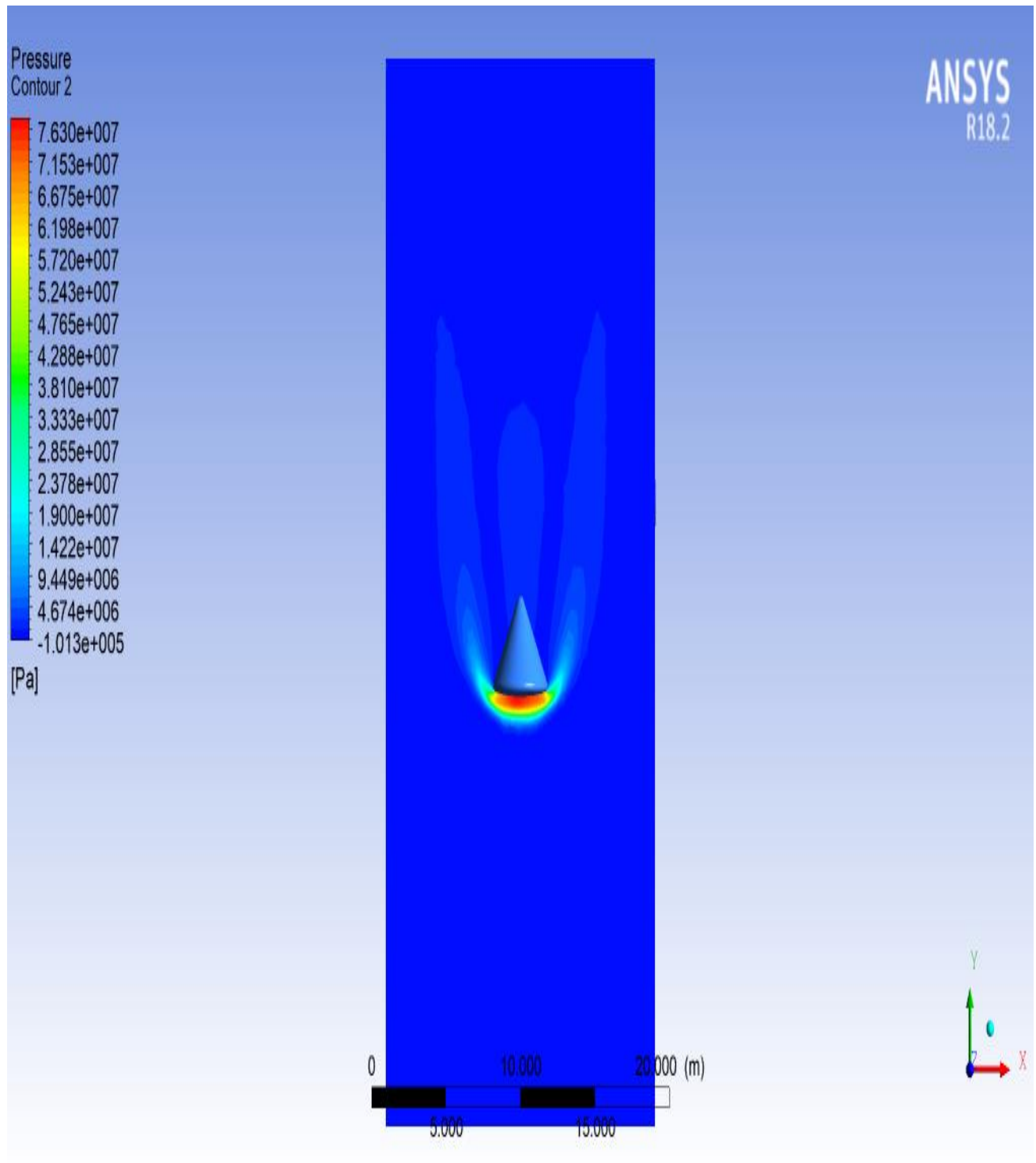


Fig.4.1.2 Pressure variations at 70.0km altitude

4.1.3 at 77.2 km altitude

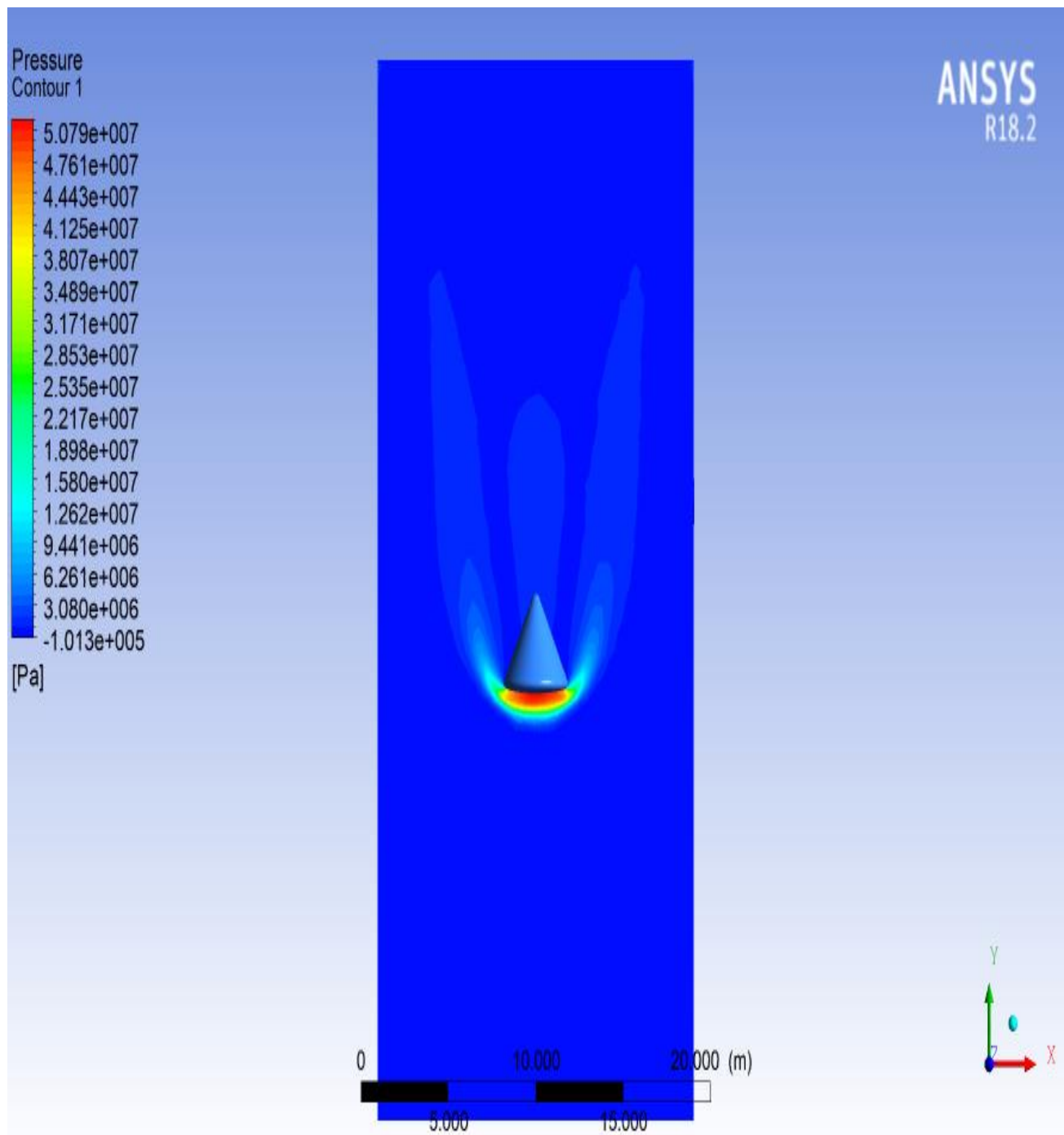


Fig.4.1.3 Pressure variations at 77.2km altitude

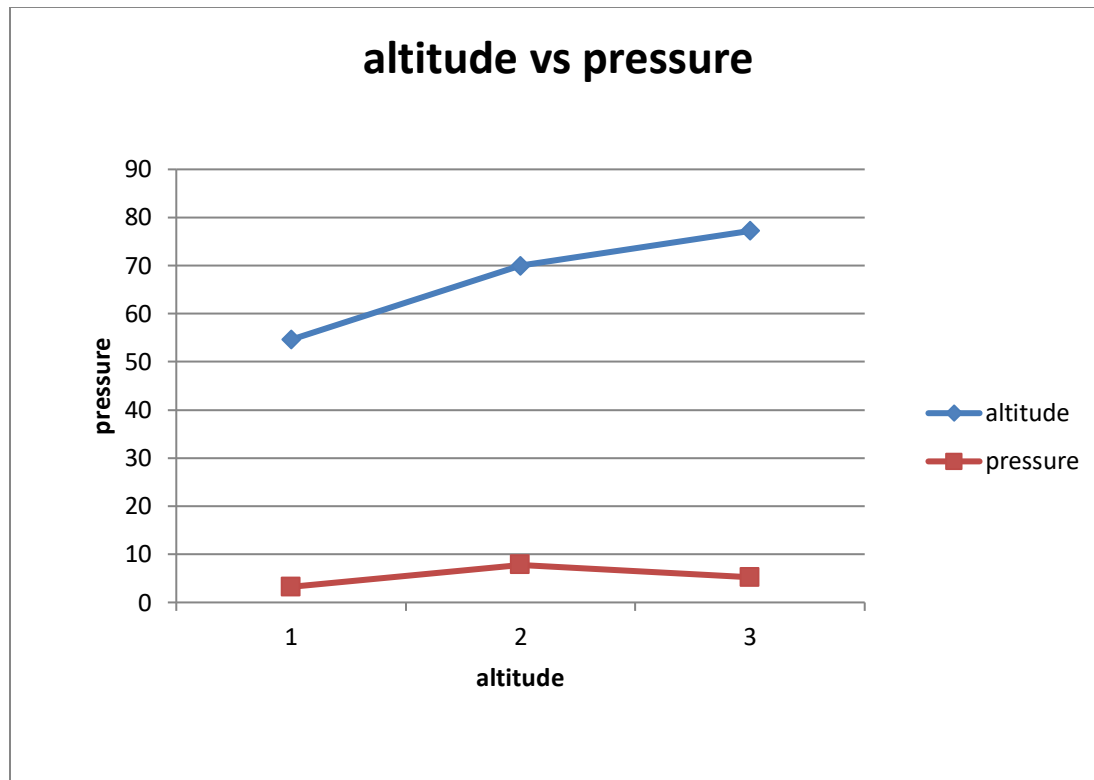


Fig.4.1.4 Pressure between different altitudes

Pressure variation in fluid flow at different altitudes is shown in the above graph. Pressure is high at 70km (7.92km/s velocity) when compared to other two altitudes since the velocity is high and impact of fluid flow on capsule is very high.

4.2 Mach number variation

Fig. shows Mach number distribution of two dimensional model at 54.6 km altitude. The maximum Mach number is produced at the base of the re-entry vehicle and it is lowest amount at the edges.

4.2.1 At 54.6 Km Altitude

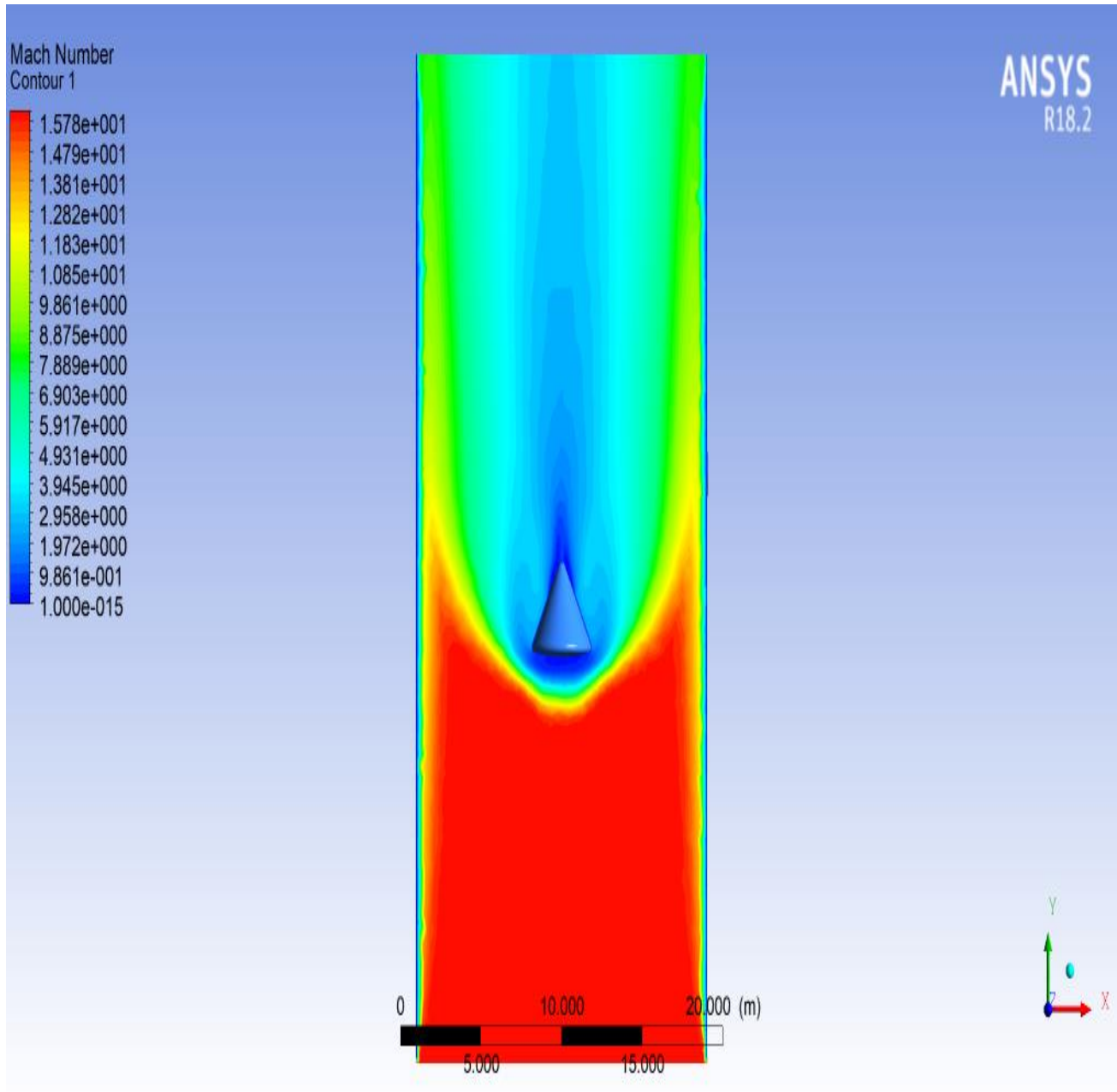


Fig. 4.2.1 Mach numbers variations at 54.6 km altitude

4.2.2 At 70.0 km altitude

Fig. shows Mach number distribution of two dimensional model at 70km altitude. The maximum Mach number is produced at the base of the re-entry vehicle and it is lowest amount at the edges

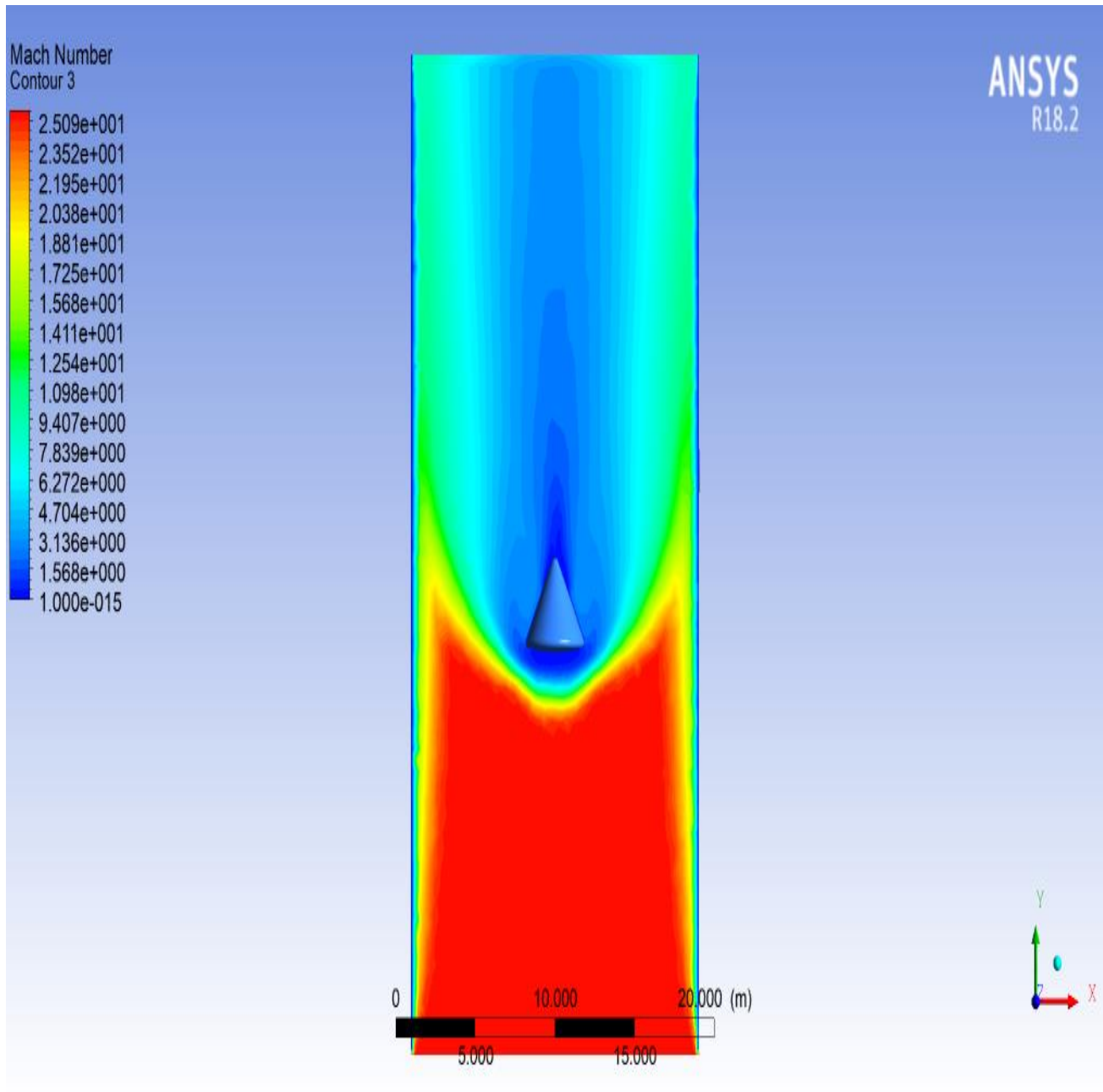


Fig. 4.2.2 Mach numbers variations at 70km altitude

4.2.3 At 77.2 km altitude

Fig. shows Mach number distribution of two dimensional model at 77.2 km altitude. The maximum Mach number is produced at the base of the re-entry vehicle and it is lowest amount at the edges.

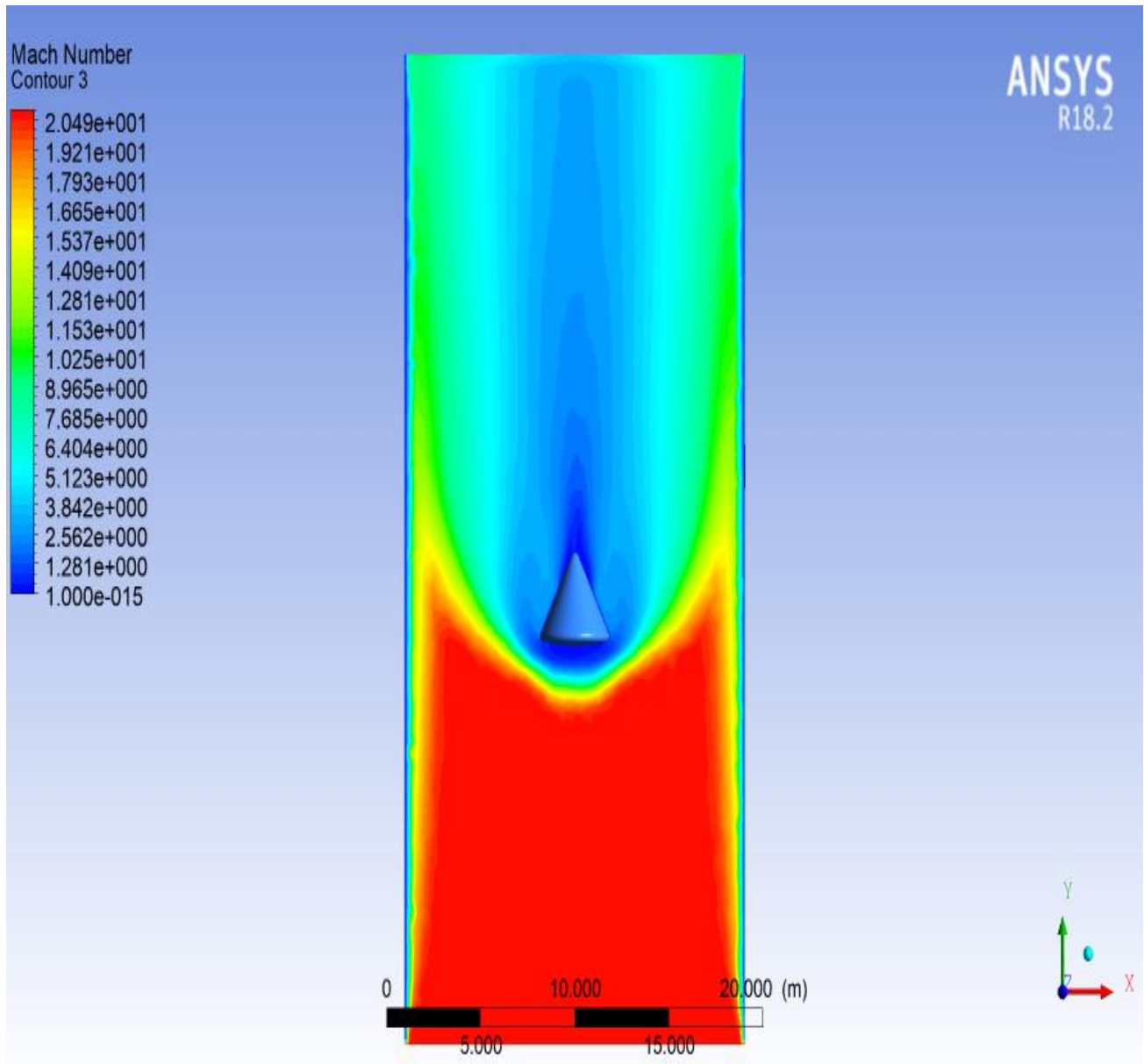


Fig.4.2.3 Mach numbers variations at 77.2km altitude

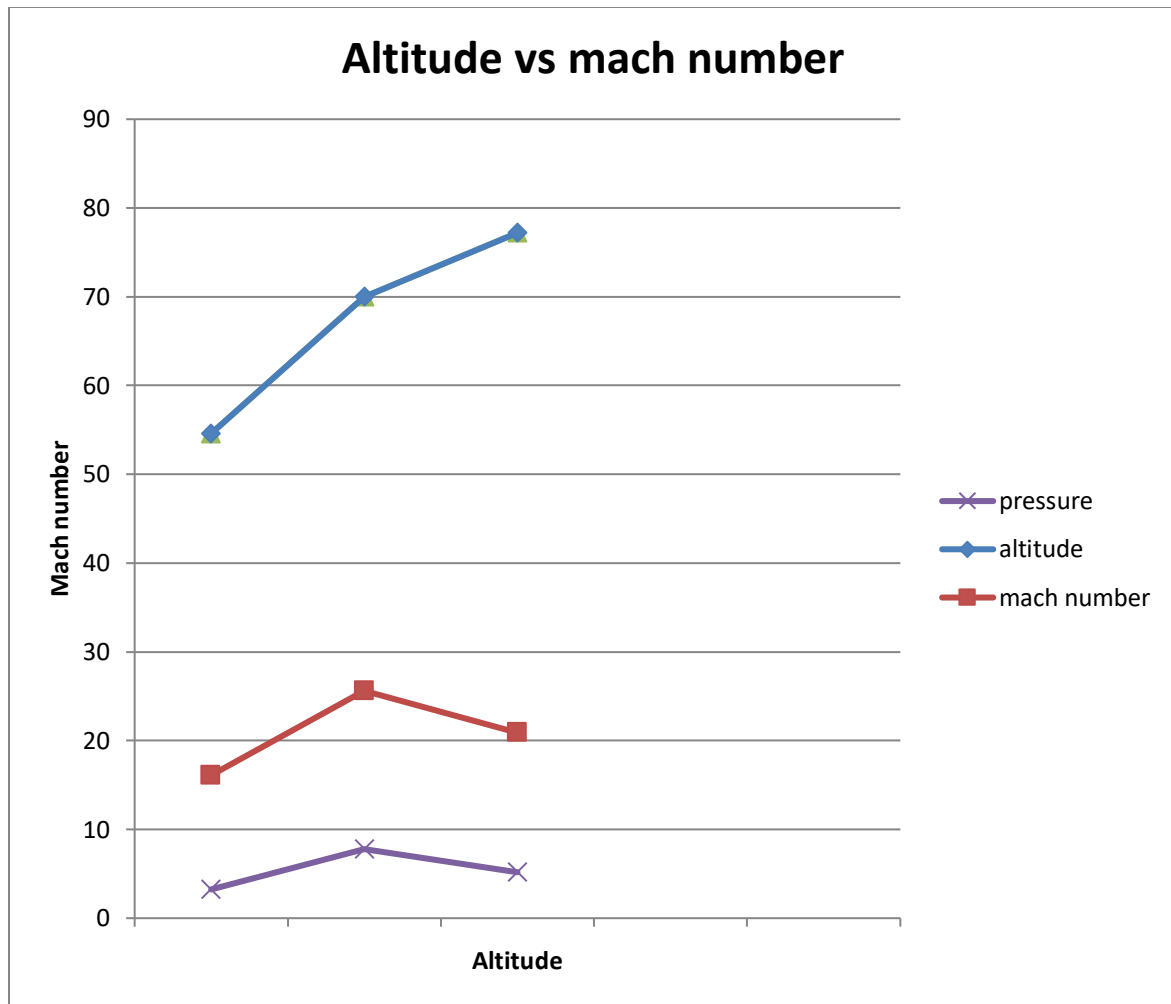


Fig.4.2.4 Mach number between different altitudes and pressure

Since Mach number is the ratio of velocity of object in a medium to the velocity of sound in that medium at same conditions, Mach number is high in case of capsule at 70.0km as the velocity is high compared to other two cases.

4.3 Temperature variation

The fig. shows the simulation of the temperature contours over the capsule. Here we can see, the temperature is maximum at the heat shield and it is also observed that the potential as well as kinetic energy decreases. So according to the law of conservation, if some energy function decreases so in order to be conserved some other energy should be increasing. Here the kinetic and potential energy is decreasing and it is dissipating in the form of heat energy. Maximum temperature was at fore body section and its value $3.147\text{E}4$ K. Minimum temperature value 245.5K.

4.3.1 At 54.6 km altitude

Fig. shows temperature distribution of two dimensional model at 54.6 km altitude. The maximum temperature is produced at the base of the re-entry vehicle and it is lowest amount at the edges.

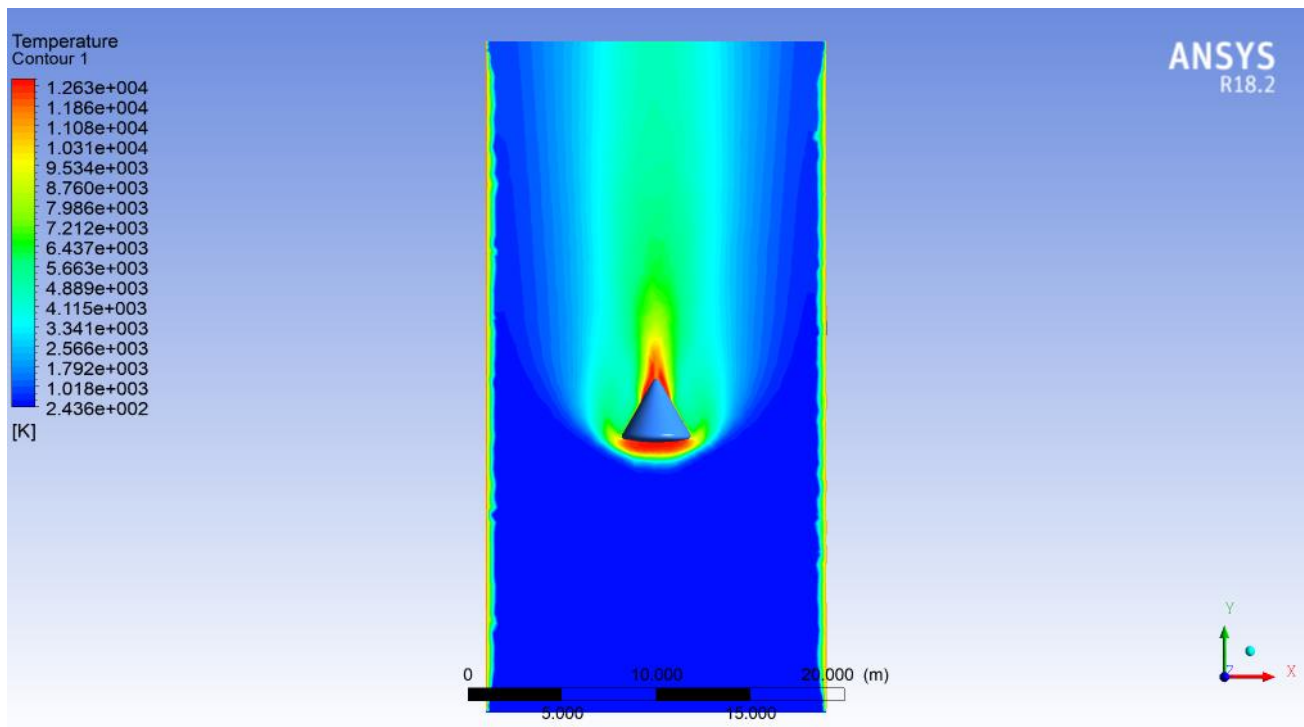


Fig. 4.3.1 Temperatures variations at 54.6 km altitude

4.3.2 At 70 km altitude

Fig. shows temperature distribution of two dimensional model at 70km altitude. The maximum temperature is produced at the base of the re-entry vehicle and it is lowest amount at the edges.

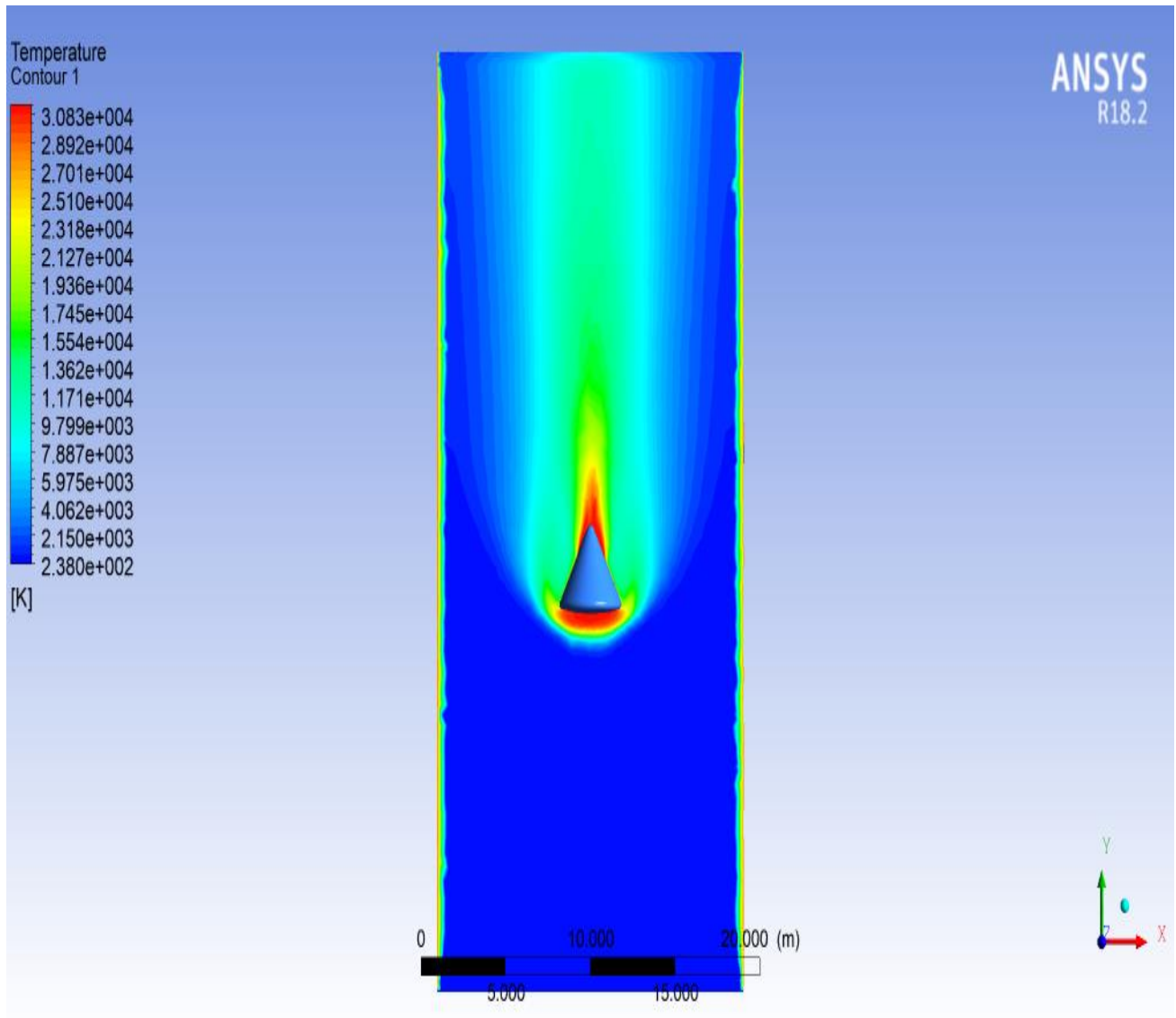


Fig. 4.3.2 Temperatures variations at 70 km altitude

4.3.3 At 77.2 km altitude

Fig. shows temperature distribution of two dimensional model at 77.2 km altitude. The maximum temperature is produced at the base of the re-entry vehicle and it is lowest amount at the edges.

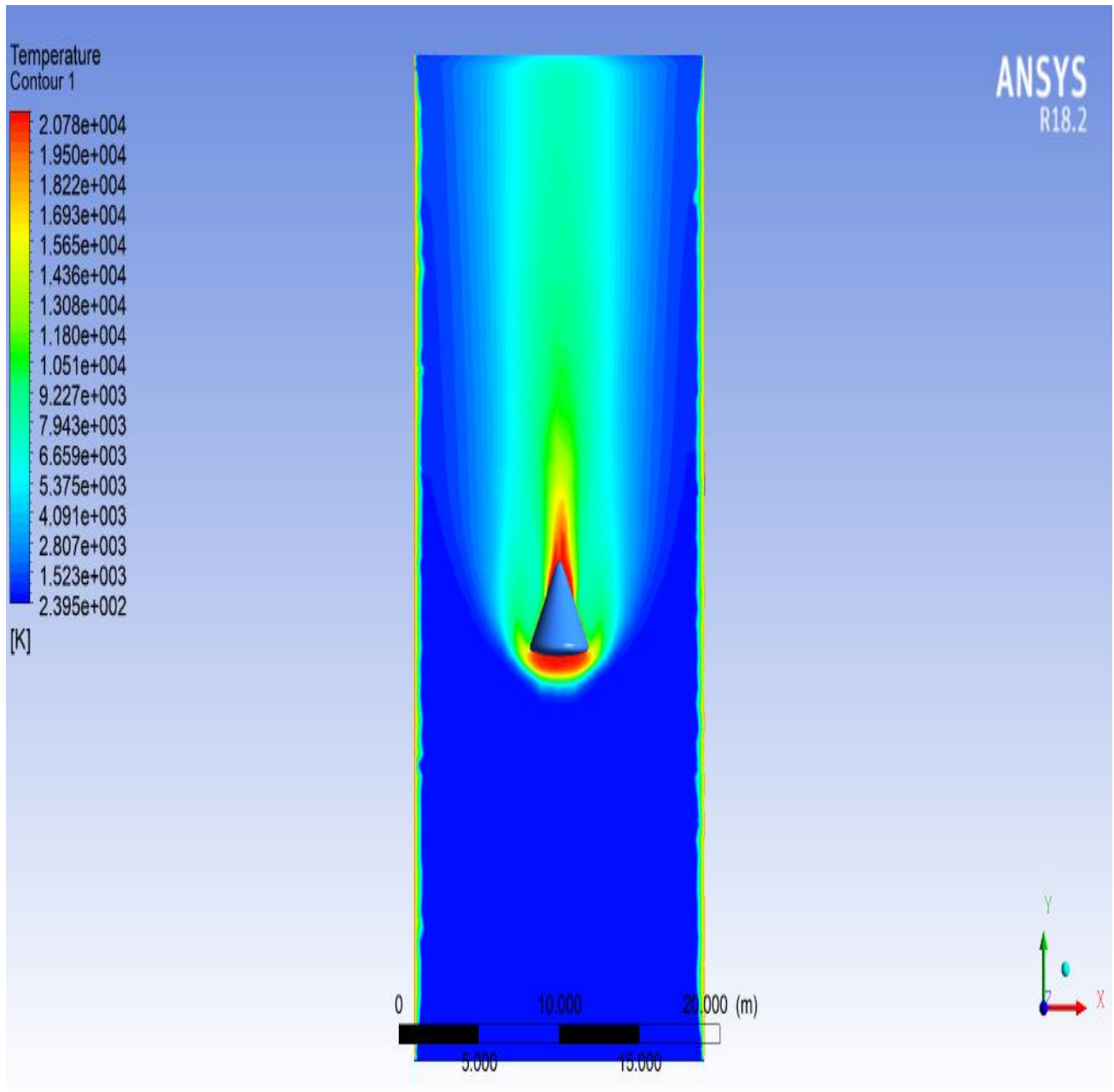


Fig. 4.3.3 Temperatures variations at 77.2 km altitude

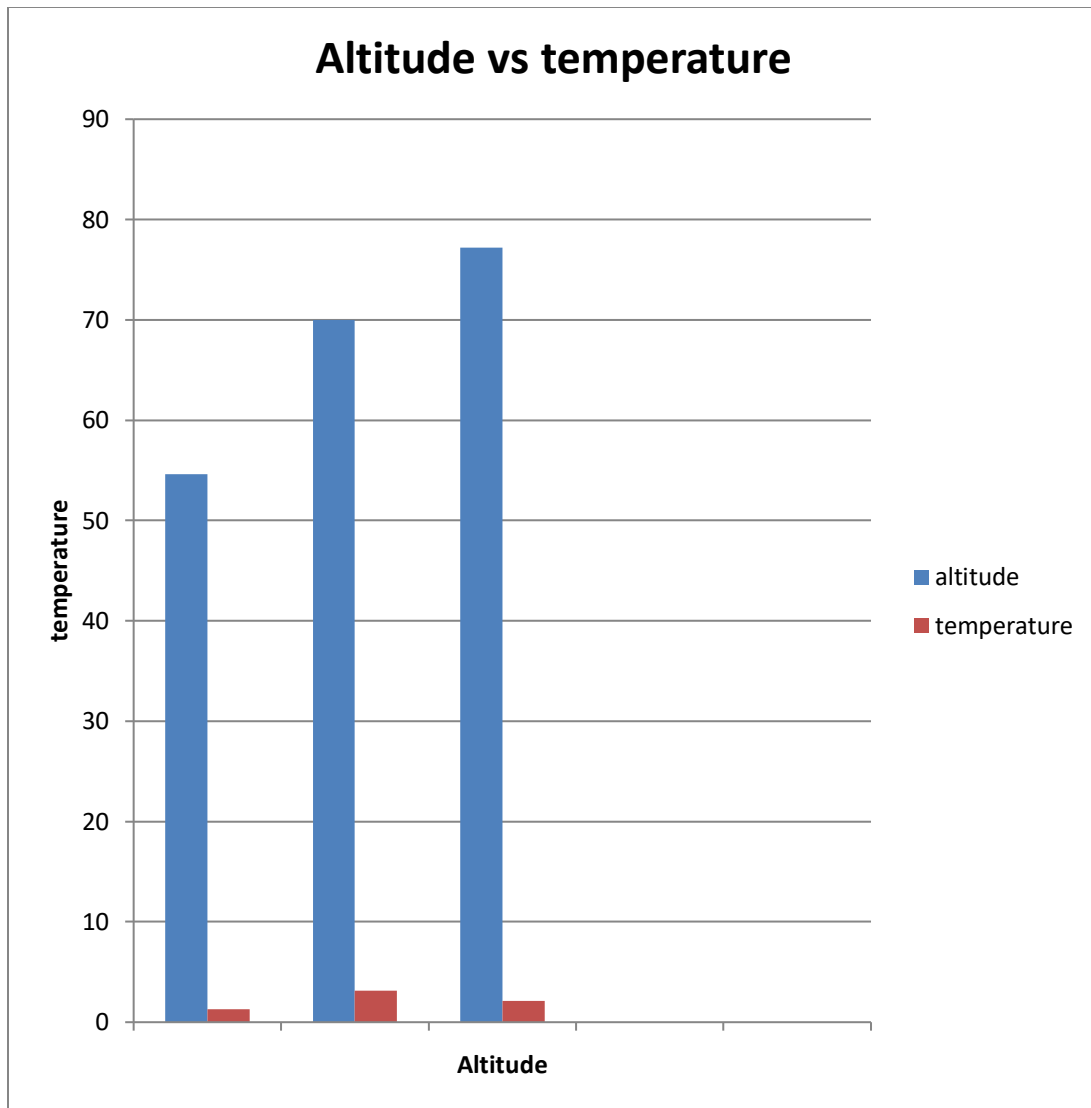


Fig. 4.3.4 Temperature between different altitudes

Above graph shows the variation of maximum temperature generated at the capsule wall. Maximum temperature is high when capsule is at 70.0km altitude because the wall shear is more at higher velocity. Temperature generated is proportional to wall shear.

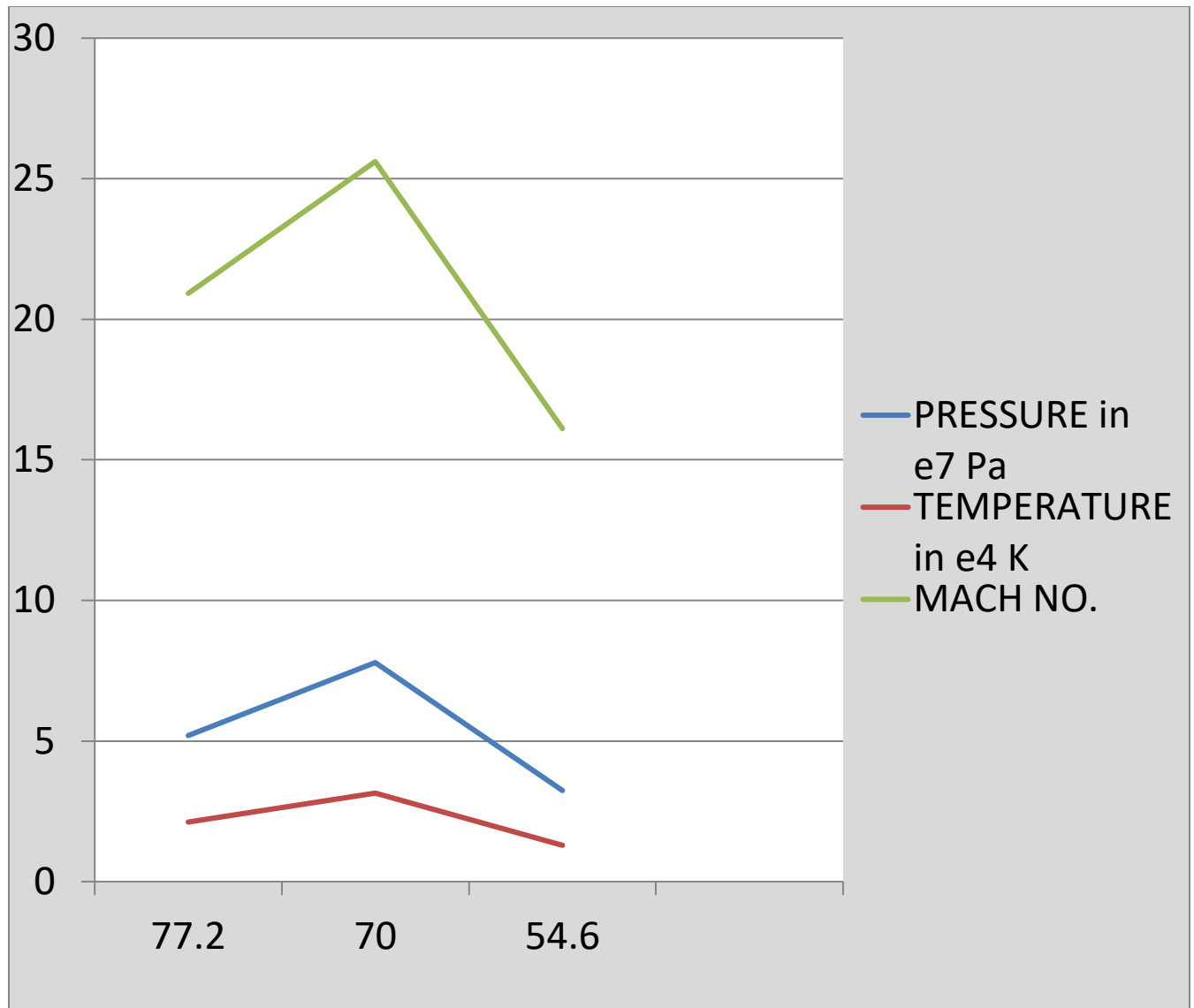


Fig. 4.4 e presents maximum pressure, temperature and mach number obtained at different altitudes

Altitude	Pressure	Temperature	Mach number
54.6	3.239E7	1.289E4	16.11
70	7.789E7	3.147E4	25.61
77.2	5.185E7	2.121E4	20.92

Table 1.2 Variation of pressure, temperature and mach number at different altitudes

CHAPTER 5

CONCLUSION

5.1 Conclusion

- Pressure, Temperature and Mach number Distribution at different altitudes is found during re-entry of space capsule in atmosphere.
- The parameters increase from 77.2 km which comes under thermosphere where air density is very low but from 70km to 54.2km the parameters decrease because at 70km mesosphere starts and around 55km stratosphere starts, which results in drastic increase of air density. This causes retardation of capsule, decreasing it's parameters.

5.2 Scope for Future Work

- Our project will help for material selection of capsule based on parameters such as pressure, temperature which we found

REFERENCES

1. Dr. Roy N Mathews, “*Hypersonic Flow Analysis on an Atmospheric Re-Entry Module*”, IJERGS, September-October, 2015, Volume 3, Issue 5.
2. Krishnendu Sinha, “*Computational Fluid Dynamics in Hypersonic Aerothermodynamics*”, Defence Science Journal, November 2010, Vol. 60, No. 6.
3. Ernest R Hillje, “*Entry flight aerodynamics from Apollo Mission AS-202*”, NASA TN D-4185, October 1967.
4. Dr.B.Balakrishna, “ *Flow Analysis of an Atmosphere Reentry Vehicle*”, IJERD, August 2012, Volume 3, Issue 4.