An Aerodynamic Optimization of Supersonic Flow Over The Nose Section of Missiles.

B.Kaleeswaran [1], S.Ranjith Kumar [2], Jeniwer Bimro.N [3]
1. M.Tech (CFD) Research Scholar, UPES-Dehradun
2. Designspecialist, CAD Centre, Chennai
3. M.Tech (ES) Research Scholar, UPES

Abstract

This abstract tells about the aerodynamic study over 2D supersonic nose cone models of missiles. First a Spherical nose cone model was tested with a Mach speed of 3 and then with the same Mach speed another Spherical model with a parabolic nose cavity was tested. Both the models were designed in GAMBIT and analysed in FLUENT. Various temperature, pressure, velocity contour and pressure plots were taken and studied as results. From the results it was proven that the new curvature model shows good thermal resistance to high temperature than the existing spherical model. This study would be useful in the design of nose section of the missiles.

Keywords: Supersonic, Missiles, Gambit, Fluent, Mach Speed.

1. Introduction.

A Missile produces thrust by the combustion of the solid/liquid propellants it carries. The main purpose of missiles is to carry deadly payloads and to strike the enemy positions. The payload that the missile carries is usually positioned in the nose area. Increase in the temperature of the nose area can cause the missile to endorse itself to many thermal unequilibrium conditions. This may leads to explosion or some chemical changes in the payload mixture. The above case suites well if the case is a supersonic one. Thus, in this paper two models one existing and one proposed models were analysed in supersonic condition of Mach number 3.The designs were tested in 0 angle of attack (suitable for cruise condition).The main purpose of this paper is to show that the parabolic nose cavity model of the missiles shows less temperature effects (resistances) when compared to the existing spherical nose cone model. Also the results obtained were plotted and contour graphs were drawn to visualize the results.

2. Nose cone Shapes.

Missile's/Rocket's nose cone usually is of following shapes/types;
1. Spherical.
2. Cylindrical.
3. Ogive shape.
4. Wedge shaped.
5. Hemispherical shape.
The spherical nose cone shape is mostly used widely in missile design. Some examples are Patriot missiles, SCUD missiles, and AGNI missiles.

3. Design Specifications.

Design 1: Sphere cone.
Design 2: Sphere cone with a curvature at the nose (nose cavity)
A 2D model was designed as the rocket/missile nose cones possess the symmetric shape.

Design1:
The spherical model (2D) is of 1 unit length was designed in GAMBIT (it doesn't have any units, only FLUENT has). The spherical curve is of radius of 0.268 unit .A pressure farfield of 1.5 along X axisand 1.3 along Y axis were drawn to capture the shock formation.

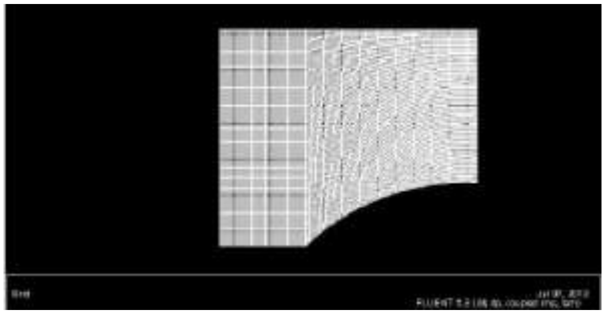

Figure: 3.1.Grid contour of the 2D spherical nose.

Design2:
The Curvature model (2D) is of 1 unit in length and was designed in GAMBIT. The spherical curve is of 0.268 unit radius. The nose curvature is of 0.1 unit of radius. A farfield of same dimension of design1 was designed.

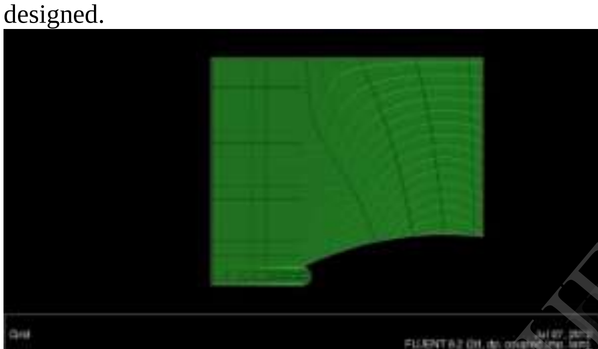

Figure: 3.2.Grid contour of 2D spherical nose with a cavity.

Both the models were designed, grid generated and meshed in GAMBIT. An interval size of good quality mesh of 0.01 was assigned to edges. Similarly, a mesh interval size of 0.01 was assigned to faces too.

```
Grid Size

Level    Cells    Faces    Nodes    Partitions
    0    17900    36076    18177             1

 1 cell zone, 4 face zones.
```

Figure: 3.3.Cell, node, faces counts in nose cavity model.

4. Literature survey:

A literature survey of many papers were analysed and some abstracts from it are presented here;
A study on Re-entry capsule at Mach 3 by Siva Prasad et.al [1] is taken into consideration. The paper is on study of Re-entry capsules at supersonic and hypersonic speeds. The US FIREII and OREX capsules were analysed. The primary design is that if the model is spherical shape it will increase the aerodynamic drag (good during the atmospheric Re-entry) and a short body. A turbulence model of KE was chosen in it. This paper shows the Mach number value before and after shock formulations.

A study of hypersonic capsule into Titan's atmosphere by Karthik Sundarraj et.al [2] .This paper focuses on the curvature model of the nose during entry into Titan's atmosphere at about Mach 19.

A study on flow analysis over aero disc at Mach 6 was carried out by Mehta et.al [3].Aero spike disc greatly reduces aerodynamic drag on the blunt body. Shock polar is obtained using the vector plots. A complete study on the reattachment shock, separation layer, and drag reduction was done. High speed flow past a blunt body creates bow shock. Spikes can be used to reduce aerodynamic drag. Bow shock creates high pressure region which inturn causes high wave drag. A region of recirculation can reduces aerodynamic drag to a great much. After the spike region, the shock again reattaches. Shock polar graph was plotted based on L/D ratio to detect drag reduction. Schlieren shadow graph was also used to study the same effect in IITK.

5. Boundary Conditions.

The boundary condition of the design model includes;

S.no	Boundary names	Edges taken
1.	Pressure Farfield	All edges except sphere.
2.	Wall	Curved edge.

Table: 5.1.Boundary Conditions.

The mesh file was saved and then exported as 2D mesh.

6. Initial Conditions.

The mesh files were read using FLUENT [4] commercial software. The grid check of the model was done.
A **density based solver** was chosen. **Energy equation** was taken into consideration. An **Inviscid**

flow condition was chosen. Thus, the problem was solved under **Ideal gas condition** in materials dialogue box with cp value 0f 1.006 J/kg-K with a viscosity of 1.789*10^-05 Kg/m-s and with a thermal conductivity value of 0.0204 W/m-K.In the boundary conditions set farfield as **pressure farfield**with **Mach number of 3** and gauge pressure of 1 atm (101325 Pa).Set wall conditions to sphere curve.
Similarly, initialize the condition in the initialization tab with the exact specifications to pressure farfieldas Mach 3, **Isothermal** static temperature **300K,** gauge pressure **1 atm**.Solve the residuals with **convergence value of 10^-6** and **courant number** (determines the stability of the flow analysis) of **0.1** with **second order upwind criteria**. Iterate the solution to interval iteration counts of about 1000 initially. After it gets converged, increase it to about 10,000.

The grid arrangement of the computational domain consists of 2D quadrilateral cells. This 2D structured mesh supports stretching and high rate of skew factor. This arrangement can align with the flow easily and can provide better accurate results.

6.1. Reynolds number effect.

In this Supersonic case Velocity is high and hence Reynolds number associated with the flow is also high. When the Reynolds number is large, the viscous force associated with the flow can be neglected. This, is because the viscous forces are small when compared to the inertial forces. A region of shock front occurs and region can be considered to be in viscid. That's the reason for choosing the inviscid solver model.
Short summary of above details;

S.no	Condition names(steps)
1.	Density based solver
2	Energy equation
3	In viscid flow model
4	Ideal gas flow

Table: 6.1.Conditions applied.

7. Equations Used.

The models were solved in inviscid conditions. So, the Continuity, Momentum, Energy equations in inviscid Conditions were written as follows;

1. Energy Equation.

$$\left(\frac{D}{Dt}\right)(e+V^2/2) = \rho q - \partial\left(\frac{UP}{\partial x}\right) - \partial\left(\frac{VP}{\partial y}\right) - \partial\left(\frac{WP}{\partial Z}\right) + \rho f V \ldots\ldots.(\mathbf{7.1})$$

2. Momentum Equation.

Along X axis;

$$\frac{\rho Du}{Dt} = -\left(\frac{\partial p}{\partial x}\right) + \rho f \ldots\ldots\ldots\ldots\ldots\ldots\ldots\ldots(\mathbf{7.2})$$

Along Y axis;

$$\frac{\rho Dv}{Dt} = -\left(\frac{\partial p}{\partial y}\right) + \rho f y \ldots\ldots\ldots\ldots\ldots\ldots\ldots(\mathbf{7.3})$$

Along Z axis;

$$\frac{\rho Dw}{Dt} = -\left(\frac{\partial p}{\partial z}\right) + \rho f z \ldots\ldots\ldots\ldots\ldots..(\mathbf{7.4})$$

3. Continuity Equation.

$$\frac{D\rho}{Dt} + \rho\nabla.V = 0 \ldots\ldots\ldots\ldots\ldots\ldots\ldots\ldots.\ldots.(\mathbf{7.5})$$

These equations were solved with upto 20,000 run iterations to meet the convergence criteria.

8. Physical Interpretation.

In this section various obtained contour plots of the results were shown and the aerodynamic variations were discussed**,**
1. Pressure contours.
2. Velocity contours
3. Temperature contours.
4. XY plots.

8.1. Pressure Contours.

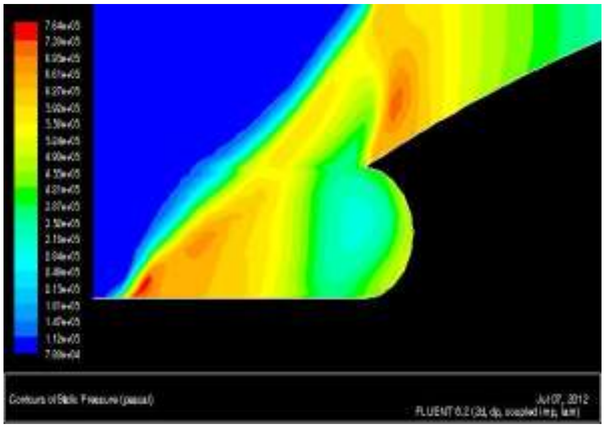

Figure 8.1.Pressure contour over a nose cavity

The pressure contour over the nose curvature spherical model was shown in the above figure. From the diagram it is clear that region of high pressure is created before the nose section and not in the nose section. Thus, by ideal gas law pressure is directly proportional to temperature. From this as pressure in the nose section reduces temperature in that region also reduces. This is a good factor for storing the chemical payloads in it. Because as less pressure in that region doesn't causes any chemical changes in its proportion.

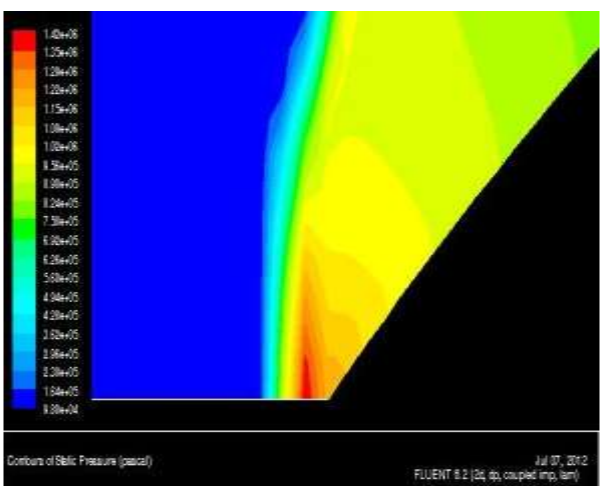

Figure: 8.2.Pressure contour over a spherical nose.

The pressure contour of the spherical nose depicts that a region of high pressure is present just before the nose section and region of mild high pressure in the nose region. This may leads to high temperature effects in the present existing model of the nose cone model of the missiles.

8.2. Velocity Contours.

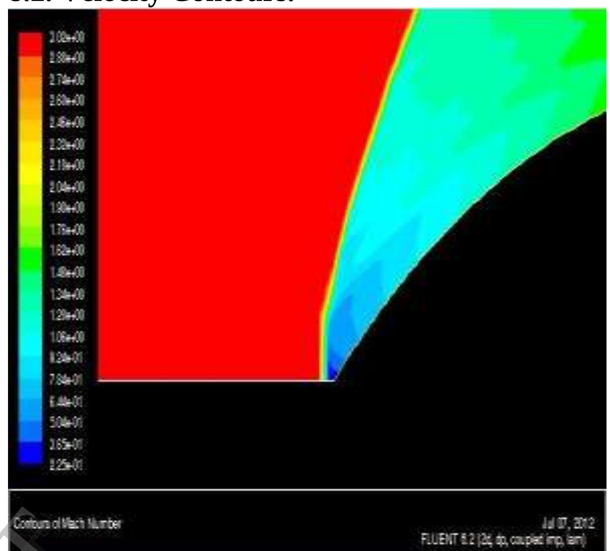

Figure: 8.3.Velocity contour over a Spherical nose.

From the velocity contour over the spherical nose, it is clear that a high Mach region occurs before the nose section. But, near to the nose there occurs a region of low Mach flow.

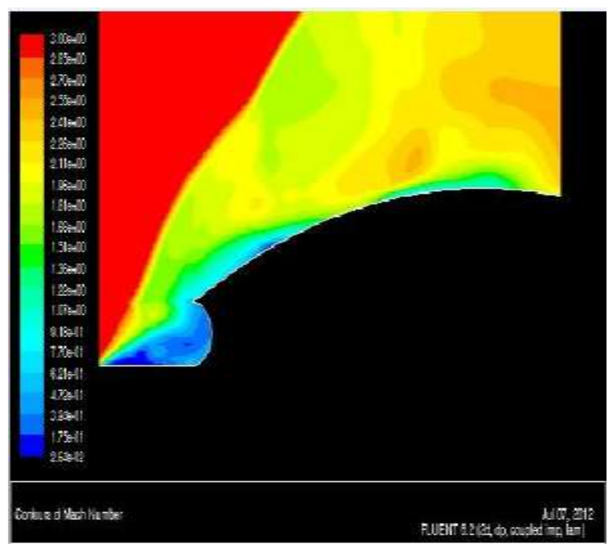

Figure: 8.4.velocity contour over a nose cavity.

Thus, the Mach flow near to the nose cavity of the spherical model was very low subsonic. A bow shock region was detected. But, if the model would have been a slope wedge an oblique shock region would have been detected.

8.3. Temperature contours.

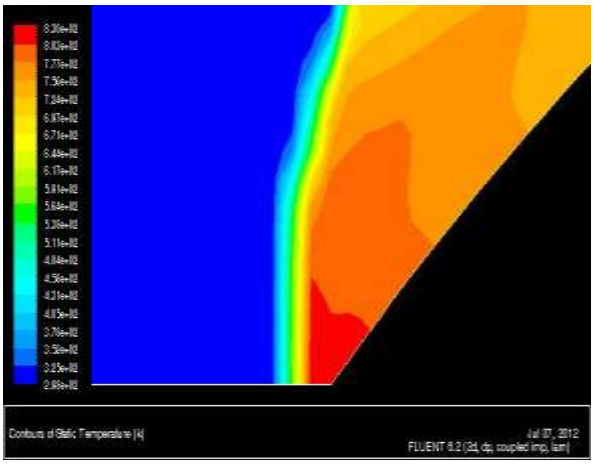

Figure: 8.5.Temperature contour over the spherical nose.

From the above contour plot, a region of high temperature is created during a supersonic flow of Mach 3.This, may sometimes causes the missiles to disintegrate or its payload characteristics may change in its chemical composition.
The contour shows that the temperature may go even upto 830 K which causes serious disasters. Thus, to reduce this factor a region of small parabolic curvature was made in the spherical model (design 2)to reduce the temperature effects. The ideal gas equation suits well to these models.

PV= nRT………………………… **(8.1)**Thus, pressure α temperature and inversely proportional to the volume of the fluid flow.
Where, R is a universal gas constant and is having a value of 8.314 KJ/kg mole.K

A programming code was done based on the ideal gas equation and its relation with the shock formation. The simple code was written in C ++.The code was shown at the chapter 11.

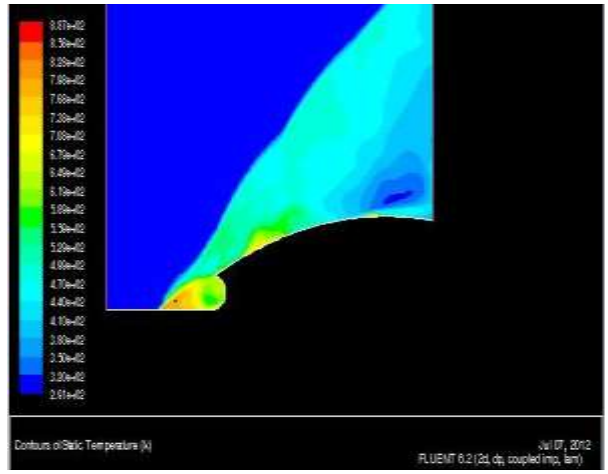

Figure: 8.6.Temperature contour over the nose cavity.

The full body contour plot shown above proves that that a region of less temperature exists before the cavity. A temperature region of less than 530 K for Mach 3 speed was found near the cavity region.

9. XY plots.

XY plots of the contour plots are shown here. These plots were for only the aerodynamic gradients over area of interest (the nose region) and not for the farfield region.

9.1. XY Plots of Nose Cavity model.

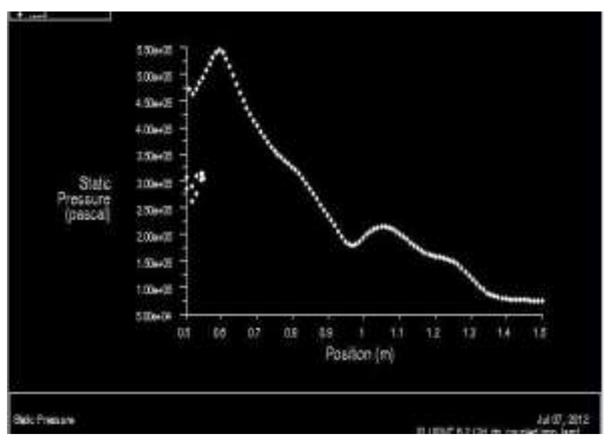

Figure: 9.1.XY plot of pressure contour.

From the XY pressure plot it is shown that;
1. A low pressure region of about 3*10^6 Pa exists in the nose cavity region.
2. A region of high pressure starts just after the nose cavity and the holds a value of about 4.75 *10^6 Pa and it reduces upto 0.05*10^6 Pa at the aft body of the nose section.

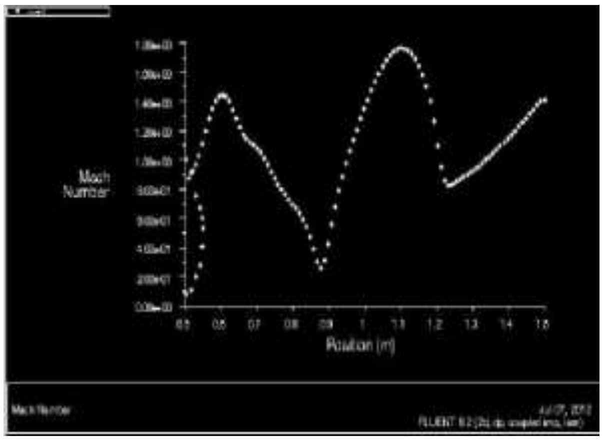

Figure: 9.2.XY plot of the velocity contour (Mach number)

A region of subsonic Mach in the cavity region and at the aft body section of the nose the fluctuating velocity gradients occurs. From the graph it is clear that a low subsonic region occurs due to recirculation in the nose cavity region. This reduces pressure at the stagnation point.

to reduce the temperature effects. The ideal gas equation suits well to these models.

PV= nRT………………………… **(8.1)**

Thus, pressure α temperature and inversely proportional to the volume of the fluid flow.
Where, R is a universal gas constant and is having a value of 8.314 KJ/kg mole.K

A programming code was done based on the ideal gas equation and its relation with the shock formation. The simple code was written in C ++.The code was shown at the chapter 11.

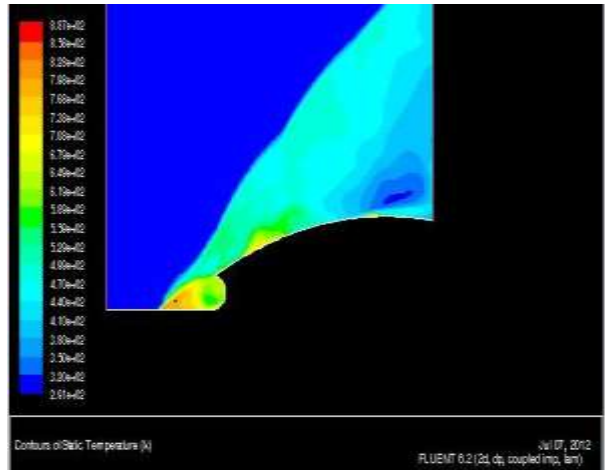

Figure: 8.6.Temperature contour over the nose cavity.

The full body contour plot shown above proves that that a region of less temperature exists before the cavity. A temperature region of less than 530 k for Mach 3 speed was found near the cavity region.

9. XY plots.

XY plots of the contour plots are shown here. These plots were for only the aerodynamic gradients over area of interest (the nose region) and not for the farfield region.

9.1. XY Plots of Nose Cavity model.

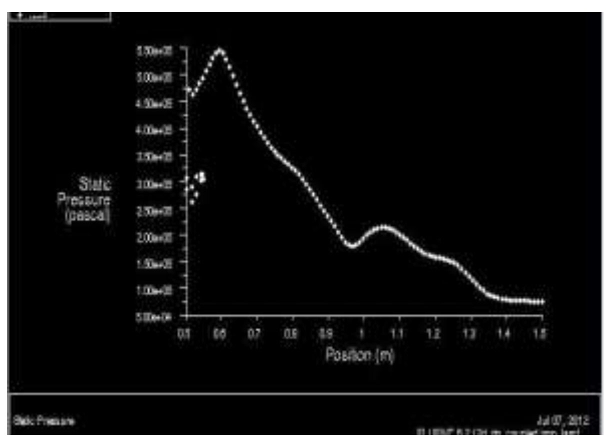

Figure: 9.1.XY plot of pressure contour.

International Journal of Engineering Research & Technology (IJERT)
ISSN: 2278-0181
Vol. 2 Issue 4, April - 2013

From the XY pressure plot it is shown that;
1. A low pressure region of about 3*10^6 Pa exists in the nose cavity region.
2. A region of high pressure starts just after the nose cavity and the holds a value of about 4.75 *10^6 Pa and it reduces upto 0.05*10^6 Pa at the aft body of the nose section.

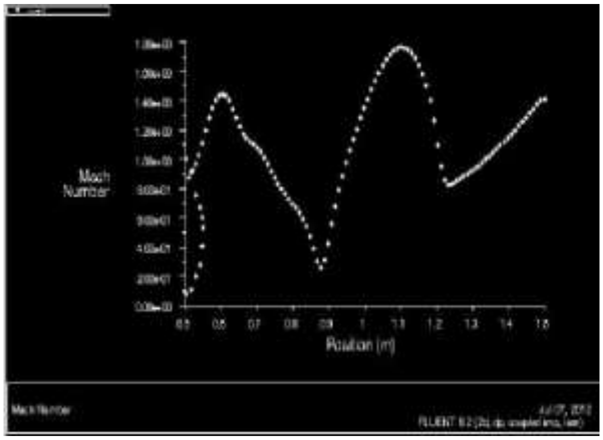

Figure: 9.2.XY plot of the velocity contour (Mach number)

A region of subsonic Mach in the cavity region and at the aft body section of the nose the fluctuating velocity gradients occurs. From the graph it is clear that a low subsonic region occurs due to recirculation in the nose cavity region. This reduces pressure at the stagnation point.

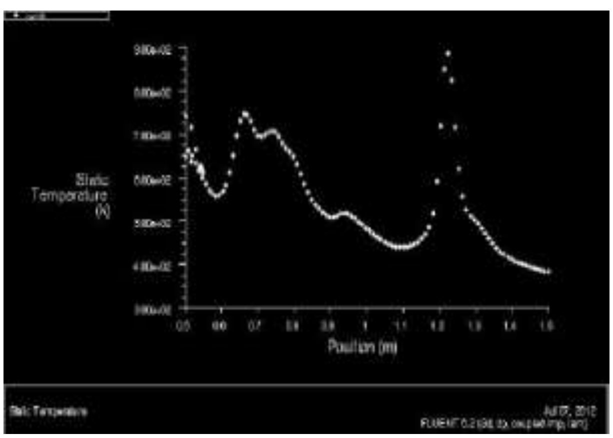

Figure: 9.3.XY plot of the temperature contour.

The XY plot suits well to the temperature contour plot of the model. A temperature region of 530 k at the nose cavity at a position of 0.5 units is shown.

9.2. XY Plots of the Spherical model.

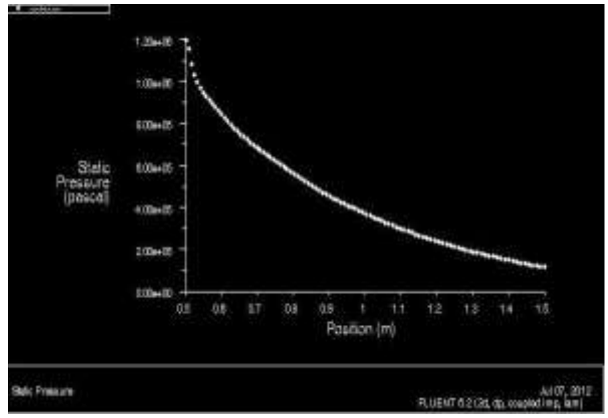

Figure: 9.4.XY plot of the pressure contour.

At the position of 0.5 units a high pressure region exists, this region exists near to the forebody of the nose section.Thus,as the position of the nose section extends the pressure gradient on it reduces drastically. But, why the pressure gradient varies (why it is reducing).The reason is due to separation of the flow. Initially a high compressible force of high velocity flow of Mach 3 strikes at the sharp point of the nose section. So, high pressure the 0.5 units position. This is because no recirculation region appears before the nose. As no recirculation velocity region occurs, high surface pressure region appears. As a result of high surface pressure region high aerodynamic drag occurs.

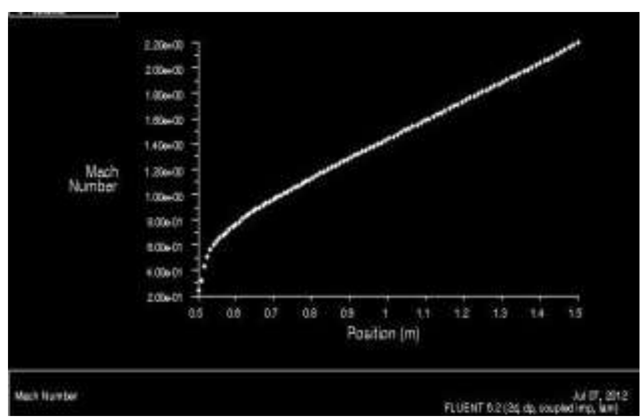

Figure: 9.5.XY plot of the velocity contour.

Near to the nose section a velocity gradient value of 4.00*e-01(1.5 Mach) exists. It reaches upto 2.25 Mach (6.12*e-01) in the forebody section of the nose.

Thus, these results were used to calculate the Mach number after the shock too. The famous ideal gas equations can be used to calculate the aftershock Mach values. This derivation results are discussed in section 9.

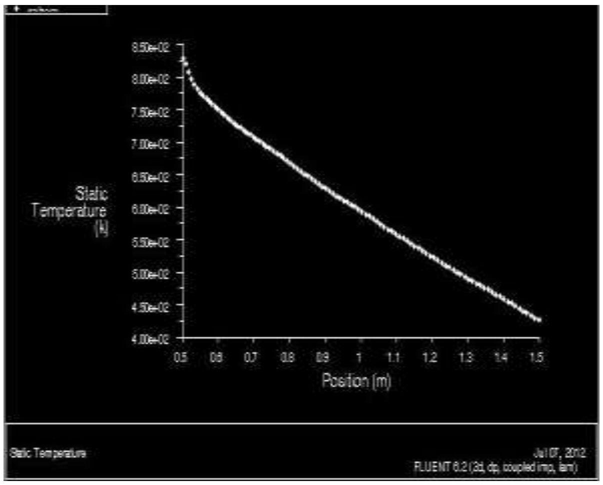

Figure: 9.6.XY plot of the temperature contour.

A region of high temperature occurs near to nose and as the position of the body changes temperature gradient effects reduces. As the stagnation pressure increases near to the nose section region, wall heat flux also increases. This is because of high aerodynamic drag formation that occurs with this model.

10. Ideal gas equations and calculations.

The shock angle is given by β is given by the term

$$\beta = \sin^{-1}(1/M) \quad \textbf{(10.1)}$$

The velocity before and after shock formation is given by velocity ratio [5]

$$\frac{(V1)}{(V2)} = \frac{(\Upsilon+1)*M^2\sin^2\beta}{(\Upsilon-1)*M^2\sin^2\beta+2} \quad \textbf{(10.2)}$$

Where;
V1 = velocity before the shock.
V2 = velocity after the shock.

Mach-Shock relation:

$$M_2^2 = \frac{2 + (\Upsilon-1)*M_1^2}{2\Upsilon*M_1^2 - (\Upsilon-1)} \quad \textbf{(10.3)}$$

Where;
M_1= Mach number before shock.
M_2= Mach number after shock.
ϒ= CP/CV=1.4(for atmospheric conditions)

10.1. Obtained Results

The obtained results suits well to both models. These calculations are done to prove the isentropic relation of the shock formation in the models.

Shock angle equation;
$\beta = \sin^{-1}(1/M)$
$\beta = \sin^{-1}(1/3) = 19^{o}\ 47'$

Velocity ratio;

$$\frac{(V1)}{(V2)} = \frac{(\Upsilon+1)*M^2\sin^2\beta}{(\Upsilon-1)*M^2\sin^2\beta+2}$$

$=(1.4+1)*(3\text{^}2)*\sin^2(19.47)/(1.4-1)*(9)*\sin^2(19.47) +2$

$= 2.399710 / 2.399951$

$= 0.9998$

Mach-shock relation;

$$M_2^2 = \frac{2 + (\Upsilon-1)*M_1^2}{2\Upsilon*[M_1^2 - (\Upsilon-1)]}$$

$M_2^2 = 2 + (1.4-1)*(3\text{^}2) / 2*1.4[3\text{^}2 – (1.4-1)]$

$M_2^2 = 5.6 / 24.08 = 0.225581.$

$M_2 = 0.48.$

International Journal of Engineering Research & Technology (IJERT)
ISSN: 2278-0181
Vol. 2 Issue 4, April - 2013

11. C++ Program Code.

The above calculations can also be found by using a simple C++ code as follows;
The program code was written to provide the results of the following relations;
1. Velocity ratio.
2. Mach-shock relation.
3. Shock angle.

11.1. Program shock calculations.

```
#include<iostream.h>
#include<conio.h>
#include<math.h>
void main ( )
{
clrscr ( );
float v1, v2, v3, ang,beta,gamma=1.4, M1,M2,MM2;
cout<<"enter the M1 value before shock "<<endl;
cin>>M1;

//getting the value of beta= sin-1(1/M1)

ang=asin (1/M1); // to get sin-1
beta=ang*(180/3.14); //to convert sin-1 to degrees

cout<<"obtained beta value is "<<beta<<endl;

//To get the value of velocity ratio

float ssin =sin (ang);
float ssin2=pow (ssin, 2);
v1= (gamma+1)*(m1*m1)*ssin2;
v2= ((gamma-1)*(m1*m1)*ssin2)+2;
v3=v1/v2; // velocity ratio
cout<<endl;
cout<<"VELOCITY RATIO IS "<<v3<<endl;

// To get the value of M2

MM2=(2+(gamma-1)*(m1*m1))/
(2*gamma*(m1*m1-(gamma-1)));
float M2=sqrt(MM2);
cout<<"value of Mach number 2 aftershock is
"<<M2;

getch ( );
}
```


11. Acknowledgement.

We would like to express our deep acknowledgements to Dr.Ugur Guven, HOD, Dept. of CFD, UPES for helping students in preparing themselves to publish many papers and to do many research works in applying the thoughts of CFD in many areas.
We would also like to express our thankfulness to Mr.KarthikSundarraj,Mr.Sourabh Bhat, Mr.Gurunadh Velidi of Dept. of Aerospace, UPES for providing technical assistances in the field of CFD. Our sincere thanks to the group KAMTS for helping us throughout the academic period.

12. Conclusion.

Mach number of the supersonic regime ranges from greater than 1 to 3 (i.e. $1<M<3$).Thus, if Mach number is greater than unity then Reynolds number is also greater than unity. In the supersonic region the fluid is considered to be ideal. Obtained results help to get the velocity ratio by comparing pre and post shock velocities. Similarly, Mach number values before and after shock was also obtained by using ideal gas relations.

Supersonic region is associated with the shock waves, mostly moving downstream. In analyzing supersonic nose cone models it was found that the high temperature, pressure regions was found to appear in front of nose. To reduce this temperature effect a nose cavity model was studied and implemented in supersonic cruise type nose cone section of the missiles. The results obtained were analysed and found that the temperature in the payload section of the missiles was reduced greatly.

Existing spherical nose cone model generated a high pressure and a temperature value of nearly 830 K.But, the cavity model generated a comparable less temperature of nearly 530 K.This resulted due to the occurrence of the recirculation region of flow before the cavity region. This, inturn reduces the surface pressure effects and inturn reducing wave and aerodynamic drag to a great extent.

References.

[1]Flow Simulation over Re-Entry Bodies at Supersonic & Hypersonic Speeds, by Shiva Prasad et.al

[2]Effect of nose cavity on the heat fluxes to Re-entry vehicle in titan's atmosphere, by Karthik Sundarraj et.al

[3]Flow field analysis over aero-disc attached to blunt nosed body at Mach 6, by R.C.Mehta et.al.

[4] Fluent 6.2.16. User guide.

[5]Liepman & Roshko "Elements of Gas Dynamics".