ISSN 2229-5518

Computational Flow Analysis Of Hypersonic Reentry Blunt Body Using Fluent And Gambit

N. S. Harshavardhan .R [1]; Sanjana .K [2]; Sai Sharan .K [3]; Srinivas .G [4]

Abstract— In the modern years space vehicle like rockets, reentry vehicles irrespective their unique designs needed control surfaces at hypersonic speeds. This paper explores the problem of determining the aerodynamics of cone and its effects on drag, model geometry, reentry heating, and type of flow, inlet temperature, aerodynamic heating and thermal protection systems. Taking these parameters as the motivation to find on the different body conditions at varying the Mach number the shape of this paper started after conducting enough literature survey by finding the research gap. There is no much work carried out from Numerical analysis side comparing with the Experimental work. So, to carry out the Numerical analysis initial stage from the public domain literature cone shape obtained its geometry, for accurate measurements it will then be plotted on graphs to obtain millimeter scale coordinate points. Then the geometry is created in the GAMBIT and suitable mesh element will be chosen to analyze the body external flow analysis. After successful conducting the grid independence suitable boundary conditions will be applied to run the simulation till it get converges the solution. Once the solution is converging analysis will be taking place to best suitable values expecting from Aero-mechanical features to develop the shape of the body by validating with experimental work.

Index Terms—blunt body, Fluent and Gambit, analysis, grid, scale, Hypersonic, reentry

1 INTRODUCTION

A

Tmospheric entry is the crusade of an object into through the gases of a planet's atmosphere beginning outer space. There are two main kinds of atmospheric entry - uncontrolled entry, such as in the entry of celestial objects, space debris or bolides and controlled entry, such as the entry (or reentry) of technology capable of being navigated or subsequent a fixed course [2, 4, 6].

Atmospheric drag and aerodynamic heating can reason atmospheric breakup capable of completely crumbling smaller objects. These forces may reason objects with lower compressive strength to blast.

An substitute, low velocity, method of controlled atmospheric entry is buoyancy which is appropriate for planetary entry where thick atmospheres, strong gravity or both factors complicates high-velocity hyperbolic entry, such as the atmospheres of Venus, Titan and the gas giants [5,8,10].

The primary designs for the blunt body profile assume laminar equilibrium boundary layer flow over the entire nose radius of the shell [11, 19]. This postulation is of course, an oversimplification and turbulent flow is exposed in portions of the shock layer, which will raise the surface heating and recession rates of the blunt body [12, 13].

2 COMPUTATIONAL FLUID DYNAMICS

Computational fluid dynamics, usually shortened as CFD, is the analysis of systems linking fluid flow, heat transfer and related phenomena such as chemical reaction by means of computer-based simulation. The method is very powerful and spans a wide range of industrial and non-industrial solicitation area. Some of the illustrations are: Aerodynamics of aircraft and vehicles: lift and drag, Hydrodynamics of ships, Power plant: combustion in IC engine and gas turbines, Chemical

- *[1]N. S. Harshavardhan .R is currently pursuing bachelor's degree program in aeronautical engineering in MIT- Manipal University, Manipal, India-576104. E-mail: suryaharsha92@gmail.com*
- *[2]Sanjana Kolla is currently pursuing bachelor's degree program in aeronautical engineering in MIT- Manipal University, Manipal, India-576104. E-mail: glitzysanjana@gmail.com*
- *[3]Sai Sharan K is currently pursuing bachelor's degree program in aeronautical engineering in MIT- Manipal University, Manipal, India-576104. E-mail: sharan019mpl@gmail.com*
- *[4]Srinivas G is currently Assistant Professor, Dept of Aero & Auto Engg, MIT- Manipal University, Manipal, India-576104, E-mail: srinivasle06@gmail.com*

ISSN 2229-5518
process engineering: mixing and separation, polymer molding, External and internal environment of buildings: wind loading and space heating/ventilation

Since past aerospace industry has combined CFD techniques into the design, R&D and building of aircraft and jet engines. More recently the methods have been functional to the design of internal combustion engines, combustion chambers of gas turbines and furnaces. Likewise, motor vehicle manufacturers now routinely predict drag forces, under-bonnet air flows and the in-care situation with CFD. Increasingly FCD is attractive a vital component in the design of industrial products and procedures. A CFD code works in following steps like Pre-Processor, Solver, Post-Processor

The governing equations of fluid flow and heat transfer are: Continuity equation: The corporeal principle states the mass of the fluid is conserved. It can neither be created nor be destroyed. The final result of applying the physical principle of the conservation of mass to a finite control volume fixed in space. This equation is called the continuity equation [14, 15].

$$\frac{\partial \rho}{\partial t} + \frac{\partial(\rho u)}{\partial x} + \frac{\partial(\rho v)}{\partial y} + \frac{\partial(\rho w)}{\partial z} = 0 \qquad (1)$$

Momentum equation: The physical principle states that the rate of alteration of momentum equals the sum of the forces on the fluid particle (Newton's second law).

X-component momentum

$$\rho\frac{Du}{Dt} = \frac{\partial(-p+\tau_{xx})}{\partial x} + \frac{\partial \tau_{yx}}{\partial y} + \frac{\partial \tau_{zx}}{\partial z} + S_{Mx} \qquad (2)$$

Y-component momentum

$$\rho\frac{Dv}{Dt} = \frac{\partial \tau_{xy}}{\partial x} + \frac{\partial(-p+\tau_{yy})}{\partial y} + \frac{\partial \tau_{zy}}{\partial z} + S_{My} \qquad (3)$$

Z-component momentum

$$\rho\frac{Dw}{Dt} = \frac{\partial \tau_{xz}}{\partial x} + \frac{\partial \tau_{yz}}{\partial y} + \frac{\partial(-p+\tau_{zz})}{\partial z} + S_{Mz} \qquad (4)$$

Energy equation: The physical principle states that the amount of change of energy is equal to the sum of the rate of heat addition to and the rate of work done on the fluid particle (first law of thermodynamics).

$$\frac{\partial}{\partial t}\left[\rho\left(e+\frac{V^2}{2}\right)\right] + \nabla \bullet \left[\rho\left(e+\frac{V^2}{2}\right)V\right] = \rho\dot{q} - \nabla \bullet (\rho V) + \rho(f \bullet V) + \dot{Q}'_{viscous} + \dot{W}'_{viscous}$$

International Journal of Scientific & Engineering Research, Volume 5, Issue 5, May-2014
ISSN 2229-5518

3 COMPUTATIONAL METHODOLOGY

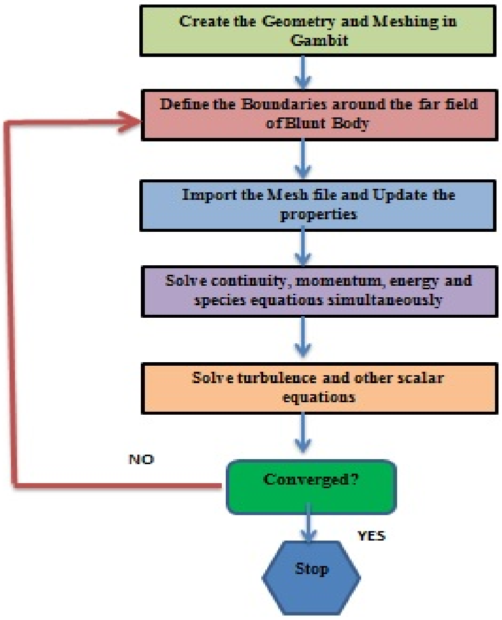


Figure 1: Flow process in Gambit and Fluent Software

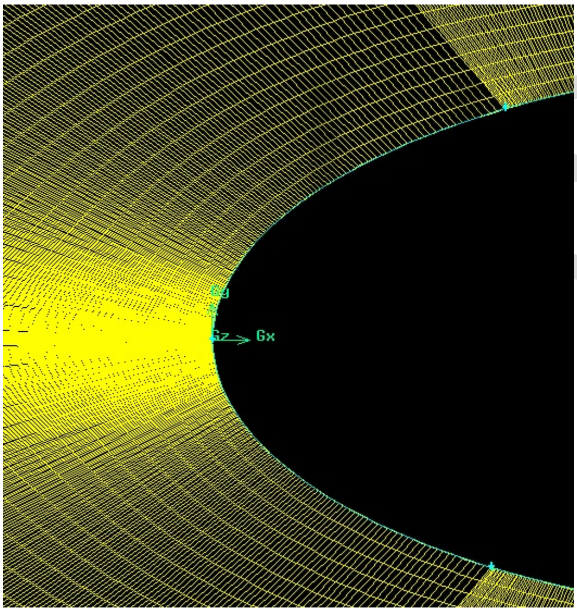


Figure 2: Meshing near the cone region

The hypersonic flow over the re-entry analysis needs to be modeled both mathematically and physically so that simulations of the flow and resultant response are studied using gambit and fluent software. The geometry of the body is created using the GAMBIT software which is axis-symmetric over the X-axis. Here the suitable mesh has been computed using triangular mesh over the surface of the blunt cone. The mesh should be very fine near the nose part and the far field also created around the body to created artificial environment as like experimental setup can see in the Fig 2-4. The dimensions of the far field are as like the experimental setup given in the literature. The inlet of the far field is taken as velocity inlet and outlet of the far field considered as pressure outlet. Further imported the meshes file into FLUENT for analysis there the suitable boundary conditions and operating conditions are given to get all the performance parameter during the reentering into the earth atmosphere.

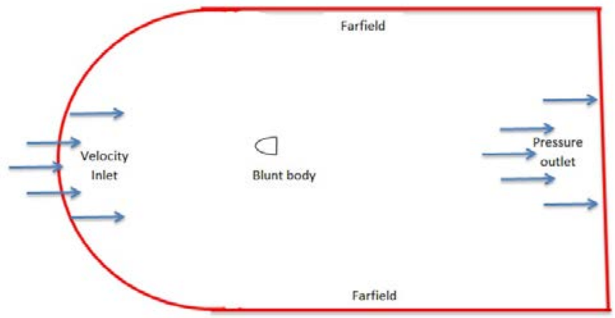


Figure 3: Gambit Flow setup over the Blunt body

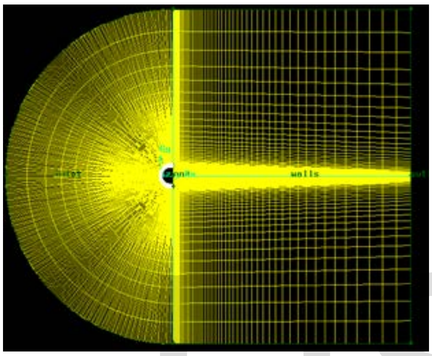


Figure 4: Gambit Meshing over the Blunt Body

4 RESULTS AND DISCUSSIONS

After successful completion of convergence solution, in the flow field around the blunt cone body, gas molecules which influence on the body experience a change in momentum, and by the random molecular impact of the molecule this change transmitted to the neighboring molecules. In this fashion, info about the presence of blunt body diffused to the surrounding flow via molecular impacts. Uncertainty the upstream flow is supersonic the disturbance wave pile up and merge, form a standing wave in front of the body. That is the reason ahead of the blunt body a bow shock is caused because deflection angle is very great at the body nose. This bow shock wave is clearly seen in the figure and ahead of this flow properties vicissitudes significantly. Utmost frequently used possessions in aerodynamic analysis of any object are given as, Pressure, Density, Temperature and Flow velocity.

From the analysis flow field around the blunt cone body, outstanding to hypersonic flow a shock wave generate ahead of the body. This shock wave is named bow shock .This shock is isolated from the body due to the high deflection angle .Pressure deviations extremely across the shock wave, at the stagnation point pressure is at peak value since at the stagnation pressure shock wave is normal to the body. Area of the curve shows the sonic region here velocity of the flow is subsonic.

From the Blunt cone analysis pressure is maximum at the stagnation point, and decreasing along the body. Pressure increase at stagnation point due to the bow shock. Among the bow shock and the body the pressure outline is reddish, it demonstrations that at that region pressure is extremely high, flow compressed due to the normal shock. Because at the apex of the body shock is sturdiest and normal to the flow. The disorder created by the body on the flow changes density drastically around the body. Reddish area just fast of the body shows high density region because at the stagnation point shock wave is strongest and normal to the body.

We can also observe that the temperature takes an important role in high speed aerodynamics because at high Mach number the kinetic energy of flow reappear in the form of internal energy of the fluid, this phenomenon is called viscous intemperance. Once the fluid temperature rise, a high temperature gradient set up between body and fluid and this basis

ISSN 2229-5518
high heat transfer rate. The above all analysis can see from Fig 3 to 10. Even the values are well matching with the experimental data [16] that can be seen in Fig 12.

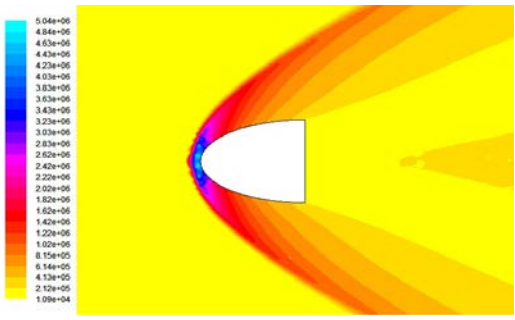


Figure 5: Pressure distributions over the Blunt body

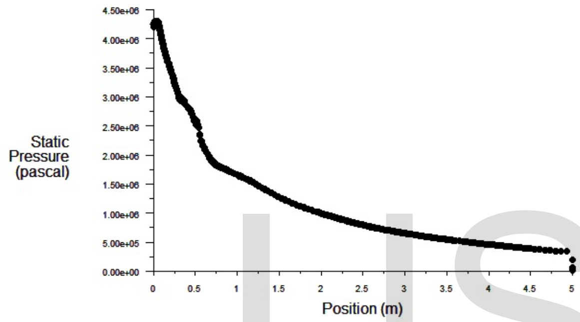


Figure 6: Static Pressure contour over the Blunt body

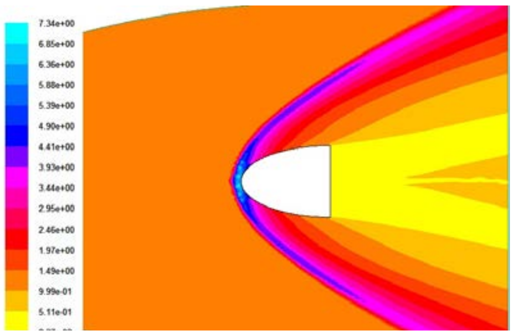
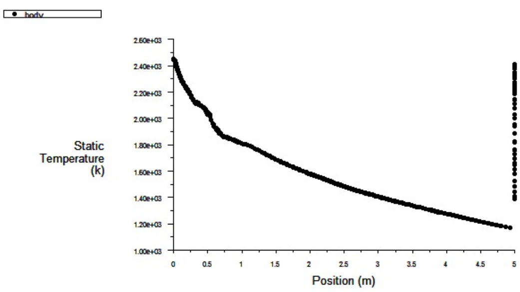


Figure 7: Density distributions over the Blunt body



ISSN 2229-5518

Figure 8: Density contour over the Blunt body

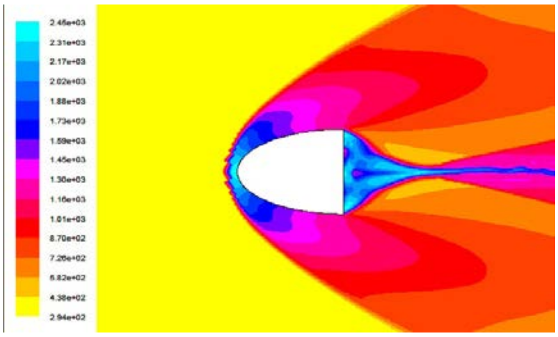


Figure 9: Temperature distributions over the Blunt body

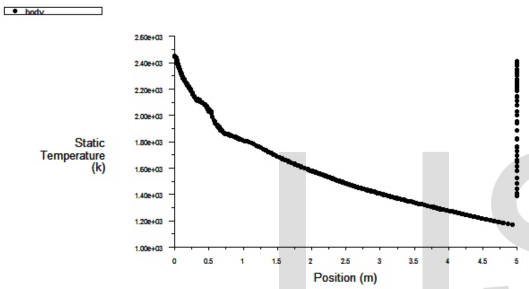


Figure 10: Temperature distributions over the Blunt body

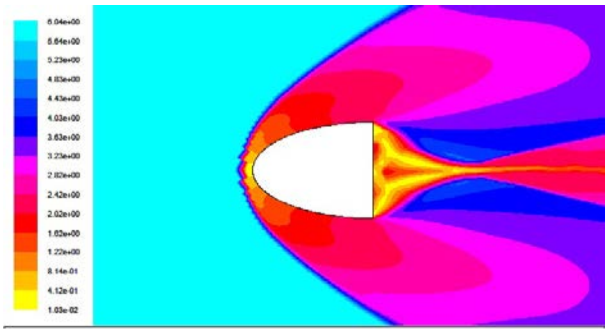


Figure 11: Mach number distributions over the Blunt body

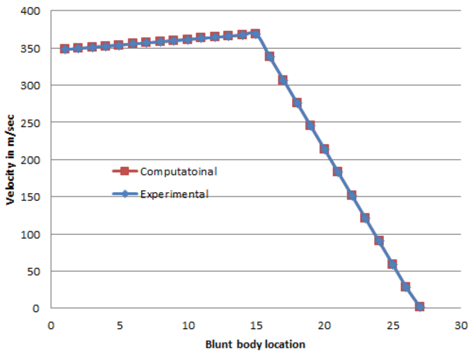


Figure 12: Computational Vs Experimental validation on Blunt body [16]

5 CONCLUSIONS

From this paper conclude that the Design of flow variables at a point just behind the bow shock wave settle that at the apex the bow shock wave can be treated as a normal shock. From the hypothetical formulation we conclude that aerodynamic heating of the body initially depends on its kinetic energy and bluntness of the nose cone decrease the aerodynamic heating over the body by generating the strong bow shock. The CFD study of minimum-drag bodies at supersonic and moderate hypersonic speeds at the level of Mach no 3 to 7 approves that, the bodies with the lowest drag have to be geometrically blunt. At maximum Mach number a detached bow shock at the front of the body produces, which is highly impact the flow properties round the blunt cone body. Mach number unexpectedly decreases extremely behind the wave and flow compacted to a high level at the stagnation point. Even the temperature increase at stagnation point is very high; due to this high heat transfer rate is set up among the flow and blunt cone body. Certainly this paper will give better aeromechanical feature for further investigations of the space explorations.

REFERENCES

[1] Fedorov, A. V., and Khokhlov, A. P., 2001, Prehistory of Instability in a Hypersonic Boundary Layer, Theoretical Computational Fluid Dynamics, 14, 359-375.

[2] Mack, L. M., 1975, On The Application of Linear Stability Theory and The Problem of Supersonic Boundary Layer Transition, AIAA Journal, 13, 423-448.

[3] Fedorov, A. V., Receptivity of a High Speed Boundary Layer to Acoustic Disturbances, Journal of Fluid Mechanics, 491, 101-129.

[4] Egarov, I. V., Fedorov, A. V., and Soudakov, V. G., 2005, Direct Numerical Simulation of Supersonic Boundary Layer Receptivity to Acoustic Disturbances, AIAA Paper, 97.

[5] Balakumar, P., 2002, Stability of Hypersonic Boundary Layers Over a Compression Corner, AIAA Paper, 2848.

[6] Mair, W. A, 1952, Experiments on Separation of Boundary Layers on Probes in front of Blunt Nosed Bodies in a Supersonic Air Stream, Phil. Mag. Ser. 7, Vol 43, No 343, 695.

[7] Billig, F. S., 1967, Shock-Wave Shapes around Spherical and Cylindrical-Nosed Bodies, Journal of Spacecraft, 4, 6, 822– 823.

[8] Yankilevsky DZ. The optimal shape of an earth penetrating projectile. Int J Solids Struct 1983;19:25–31.

[9] Moss, J. N., Price, J. M., Dogra, V. K., and Hash, D. B., "Comparison of DSMC and Experimental Results for Hypersonic External Flows," AIAA Paper 95-2028, June 1995.

[10] Horvath, T. J., McGinley, C. B., and Hannemann, K., "Blunt Body Near Wake Flow Field at Mach 6," AIAA Paper 96-1935, June 1996.

[11] Hollis, B. R., and Perkins, J. N., "High- Enthalpy and Perfect-Gas Heating Measurements on a Blunt Cone," Accepted for publication in AIAA Journal of Spacecraft and Rockets, 1996.

[12] Kastell, D., Horvath, T. J., and Eitelberg, G., "Nonequilibrium Flow Expansion Around a Blunted Cone," 2nd European Symposium on Aerothermodynamics, ESTEC, Noordwijk, The Netherlands, Nov., 1994.

[13] Krasil'shchikov A.P. et al. Experimental Studies of Bodies of Revolution in Hypersonic Flows. Fizmatlit, Moscow, 2007

[14] Glotov G.F. (2003) Aerothermodynamics of Flying Vehicles in Photographs. TsAGI, Zhukovsky, 2003

[15] Coleman G.T. A Study of Hypersonic Boundary Layers over a Family of Axisymmetric Bodies at Zero Incidence. Preliminary Report and Data Tabulation, Imperial College of Science and Technology. England, I. C. Aero. Rept. 73- 06, 1973.

[16] Niranjan Sahoo , "Experiments on a Blunt Cone Model in a Hypersonic Shock Tunnel" 16th Australasian Fluid Mechanics Conference, 2-7 December 2007. Pg 503-506.