International Journal of Engineering Innovation & Research
Volume 10, Issue 4, ISSN: 2277 – 5668

Effect of Shock Wave on the Blunt Body at Hypersonic Velocity

Suraj Sambhajirao Patil * and Sahil Chandel
Lovely Professional University, Punjab, India.
*Corresponding author email id: surajsambhajiraopatil@gmail.com

Date of publication (dd/mm/yyyy): 30/08/2021

***Abstract* – This paper is focused on the Computational analysis and comparison of various Nose cone shapes when the re-entry vehicle exceeds Mach 5 velocity. Since the nose cone undergoes extreme temperature and pressure during re-entry into earth's atmosphere which also produce shock wave at the nose cone, thus, it is wise to consider all the aerodynamic parameters in order to sustain the atmospheric drag. The blunt body structure of nose cone plays an influential role when designing a re-entry vehicle. Hence, several nose cone designs are studied for the sole purpose of discovering the most efficient design among them. Comparative analysis is carried out to see the disparity created due to varying radius of curvature. Flow analysis is an important measure to create the real time simulations in accordance with the stagnation conditions and bluntness ratio. Thus, it gives an effective result, how the various nose cone designs would perform in the desired environment. Flow analysis is carried out using ANSYS software. While the "Finite Element Method (FEM) is applied for the Structural Analysis of Nose cone design. Also, it is important to balance the Fineness ratio, as increase in Fineness ratio will lead to increase in skin drag. Apart from this, the subsequent effects of heat transfer, shockwaves, and chemical reactions are also considered. Based on the results, the most efficient nose cone design among the five chosen designs, is proposed.**

***Keywords* – Blunt Body, Hypersonic, Nose Radius, Shock Wave, Flow Analysis.**

I. Introduction

Designing the Nose cone is the most crucial phase while designing a spacecraft [1]. Shockwave formation, Aerodynamic drag, velocity, material selection are some of the influential factors. Hypersonic Velocity is attained when spacecraft exceeds five times the velocity of sound. At this velocity, strong shockwaves are formed closer the to the surface of body [3]. Thus, study of a blunt body at hypersonic velocity also involves the heat transfer, thermal stress and pressure force occurring in this region. At the hypersonic velocity, a significant role is played by gas dynamics and Nose cone design. Due to the ionization of gas molecules, radiative heat transfer increases. In order to find the efficient nose design for the hypersonic region, experimental study has been done by comparing five nose cone designs. Each design is different from the other due to different radius of curvature of nose cone. The flight conditions are set as per sea-level reference. While the Bluntness ratios is set as per the design parameters [1].

In this research paper, design parameters of nose cone are studied at hypersonic velocity. The factors like Shock Layer, Entropy Layer, Aerodynamic heating, Low Density are also considered. As Mach number keeps increasing, it leads to shock wave formation and thus, there is rise in the stagnation temperature and other stagnation flow condition such as pressure and density. Also, due to difference in flow properties, upstream and downstream of shockwave is separately studied. To visualize the aerodynamic effects on nose cone, ANSYS software is used to study the flow analysis and also to validate the results. In hypersonic flow region, high heat is produced due to the aerodynamic drag and the entropy layer. It also influences the Spacecraft velocity. But the shockwaves increases the velocity and pushes the shock front closer to the surface of body. Hence, the body design has a great impact on the contour of shockwave.

In the structural aspects, Blunt body design is effective in hypersonic region. The presence of large surface area allows better heat transfer and thus temperature of body is significantly less. Another advantage of using blunt body design is that the shockwave is formed far from the surface of body. The blunt shape maintains the structural integrity of spacecraft. Thus, the blunt body design allows the spacecraft to sustain extreme atmospheric conditions during re-entry. For the fabrication of structure, the material selection is to be of "Space grade". And the structural analysis is carried out using the Finite Element Method.

II. Geometrical Modelling

1. *Conic Nose Cone:*

This shape is generally preferred because it is easy to design and low cost of manufacture. However, the misconception behind using this shape is its Drag Characteristics. It is generally perceived that Conic design would encounter less drag. But in reality, this belief proves wrong [1].

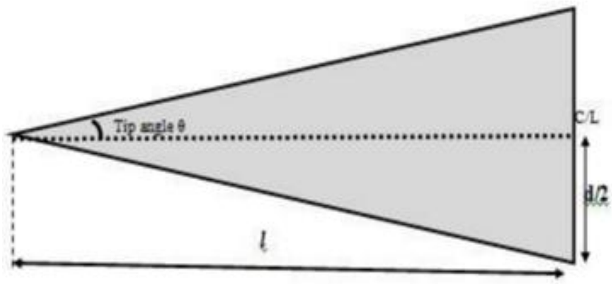

Fig. 1. Diagram of conic nose cone.

2. *Spherically Blunted Conic Nose Cone:*

The process to create a Spherically blunted nose cone shape involves, placing a spherical segment on a blunt cone shape. The shape, thus obtained, has more surface area than the Conical nose cone. Hence, the drag is reduced.

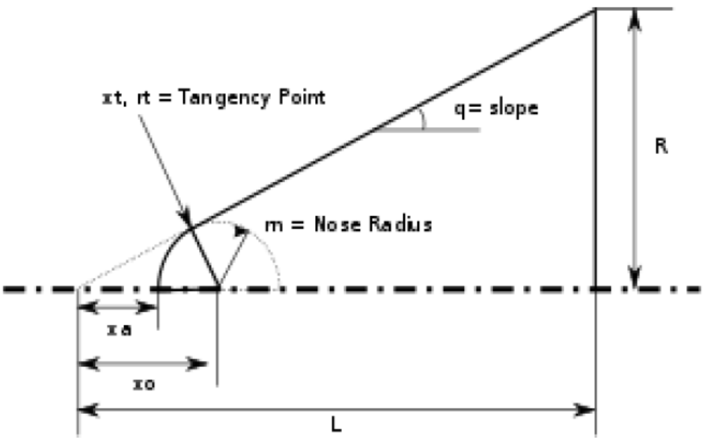

Fig. 2. Diagram of Spherically Blunted Conic Nose Cone.

3. *Bi-Conic Nose Cone:*

The shape of Bi-Conic nose is created by placing a cone on a frustrum of another cone. The diameter of the smaller frustrum is equal to diameter of the base of upper cone.

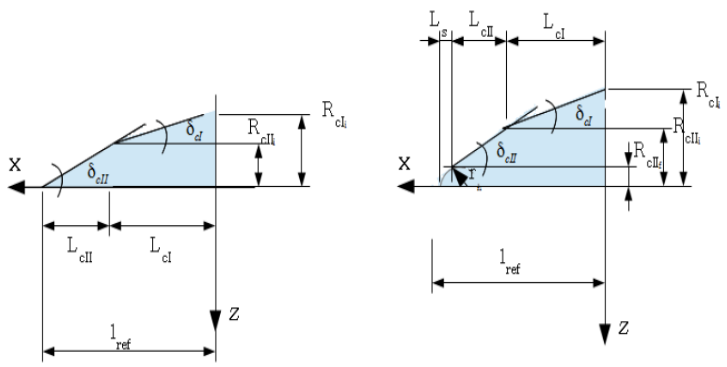

Fig. 3. Diagram of Bi-conic nose cone.

4. *Tangent Ogive Nose Cone:*

The nose profile is created by placing a spherical segment section on a rocket in a way that base of rocket is on the radius of circle (ogive radius) and tangent to the curve of nose cone.

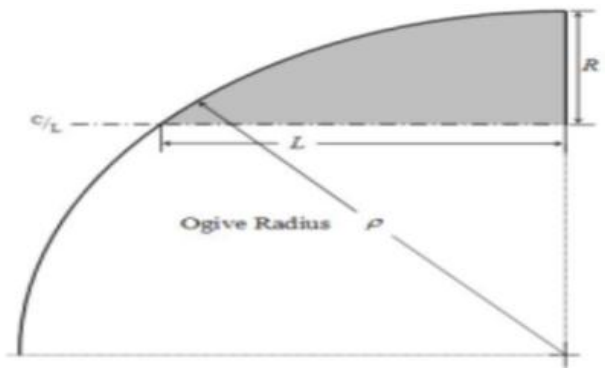

Fig. 4. Diagram of tangent ogive nose cone.

5. *Spherically Blunted Tangent Ogive Nose Cone:*

The profile of nose cone with spherically blunted tangent ogive shape is created by placing a spherical section on the front part of nose cone having tangent ogive shape.

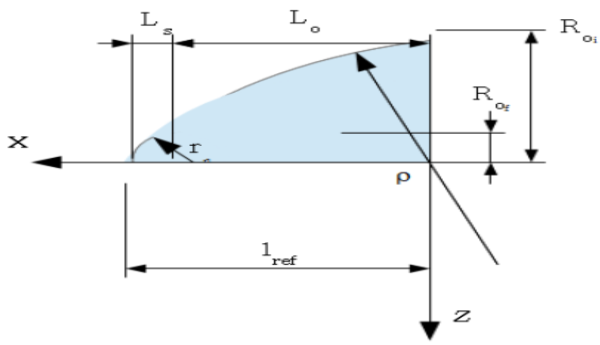

Fig. 5. Diagram of spherically blunted tangent ogive nose cone.

III. Meshing Procedure

Step 1 : Load the design in the design modular to initiate meshing.

Step 2: Select XY plane and divide domain by using face split.

Step 3: Select XY plane, enter on default mesh to generate mesh.

Step 4: Use Sizing function.

- Set advance Sizing to proximity and curvature.
- Relevance Centre-Fine.
- Set Max Face size to 12mm.
- Set max size to 15 mm.

Step 5: Added Face meshing and selected all split faces, respectively.

Step 6: Applied face meshing.

Step 7: Selected edge sizing function.

- Number of divisions is taken as 100.
- Bias factor is taken as 7.
- Procedure is repeated for every face split to gain the uniform meshing.

IV. Boundary Conditions

The mesh domain was given name by using name selection method. Far field is referring to the surrounding environment in the simulation. And the wall name selection is given to the surface of the design. The fluid medium is taken as air. After selecting pressure far field, Mach number and the gauge pressure is taken 5 Mach and 1 atm, respectively. The simulation is done on the bases of the gauge pressure value. The operating pressure value is set to zero.

Under the section of the reference values, computation is done from the far field to simulate the result. All other values were converted to the SI unit by the solver.

- Local pressure:- 1 atm.
- Temperature :- 300K .
- Mach No:- 5.
- Solver set to solve 2nd order discretisation method.

This process is repeated for every design setup during simulation to avoid error and to gain proper comparative result.

V. Simulation Technique

In this we have use Density Based Solver in Workbench as it is more preferred than Pressure Based Solver for the compressible hypersonic flow over a body. All the simulations are done in 2D, over steady state & by allowing energy equation & following K-epsilion standard model. In this we have kept operating pressure zero since the vehicle speed is more than Mach 0.1. While performing simulation fluid is assumed to be Ideal gas & viscosity follows Sutherland Law for more accurate results in terms of compressibility and viscous effect.

International Journal of Engineering Innovation & Research
Volume 10, Issue 4, ISSN: 2277 – 5668

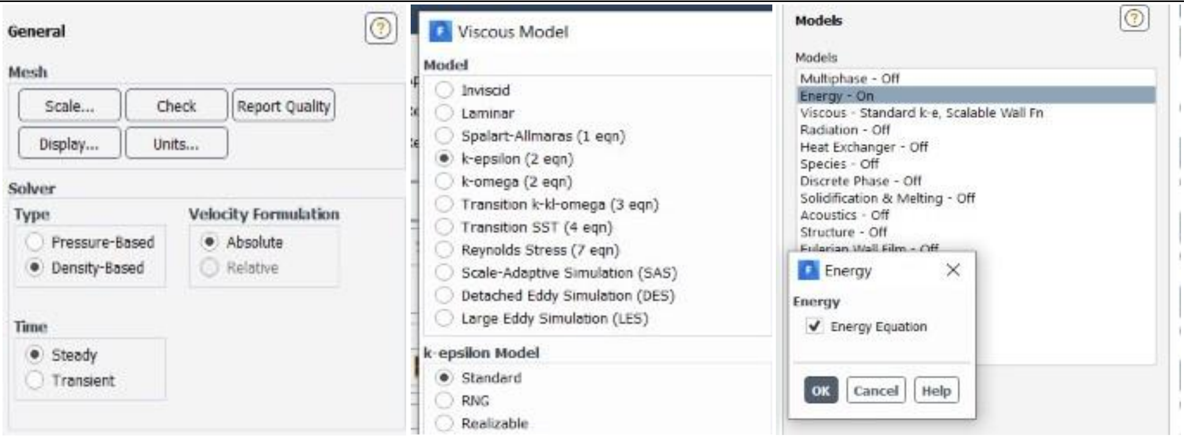

Fig. 6. Ansys models.

VI. Simulation Results

1. *Conic Nose Cone:*

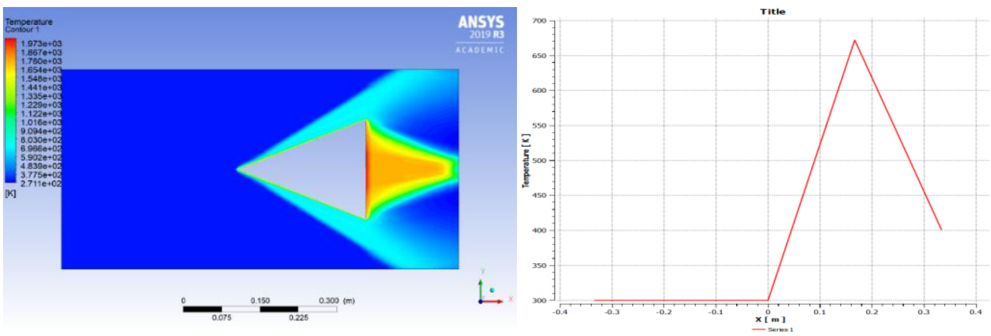

Fig. 7. Contour and graph for temperature.

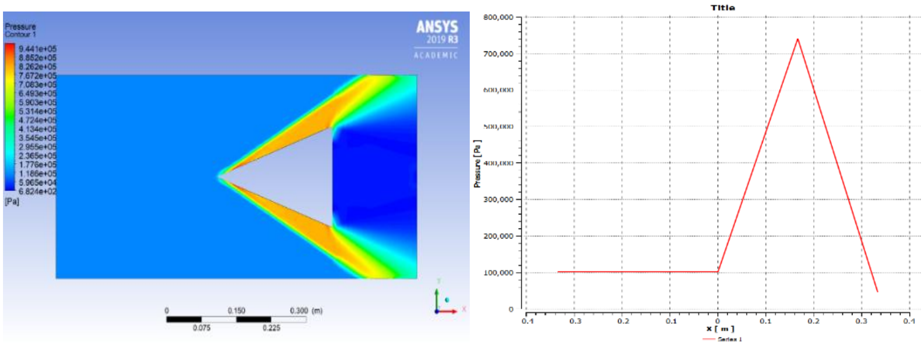

Fig. 8. Contour and graph for pressure.

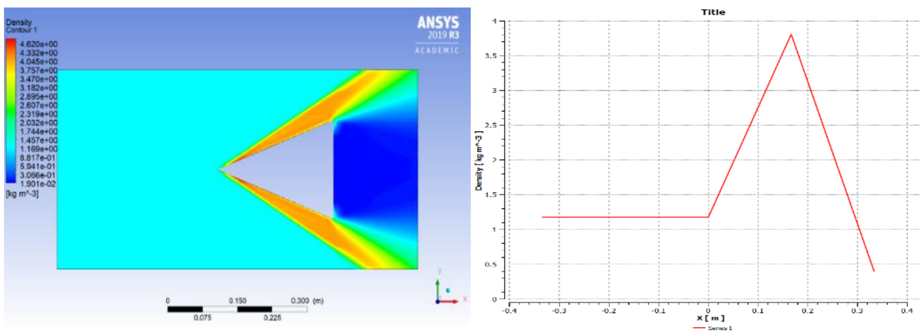

Fig. 9. Contour and graph for density.

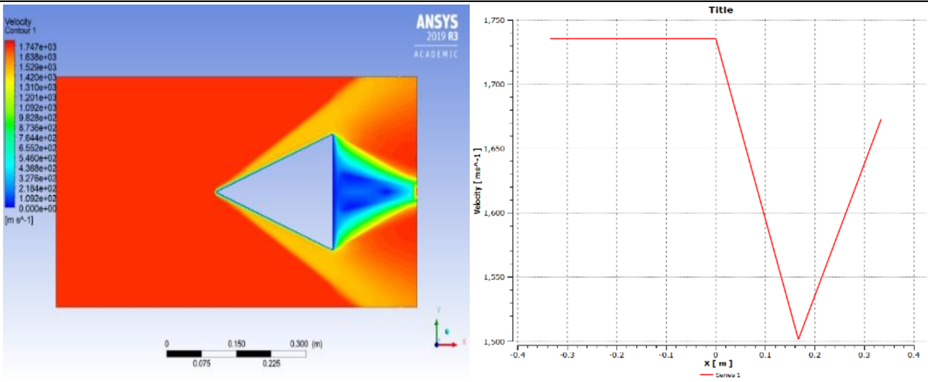

Fig. 10. Contour and graph for velocity.

VII. Spherically Blunted Conic Nose Cone

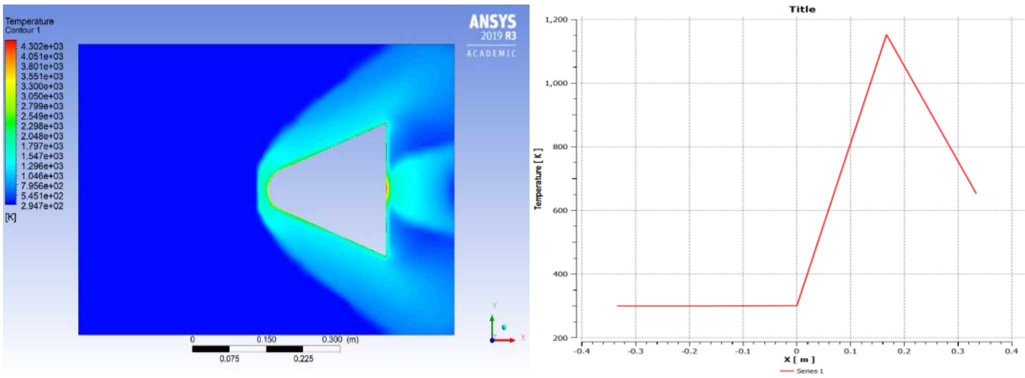

Fig. 11. Contour and graph for temperature.

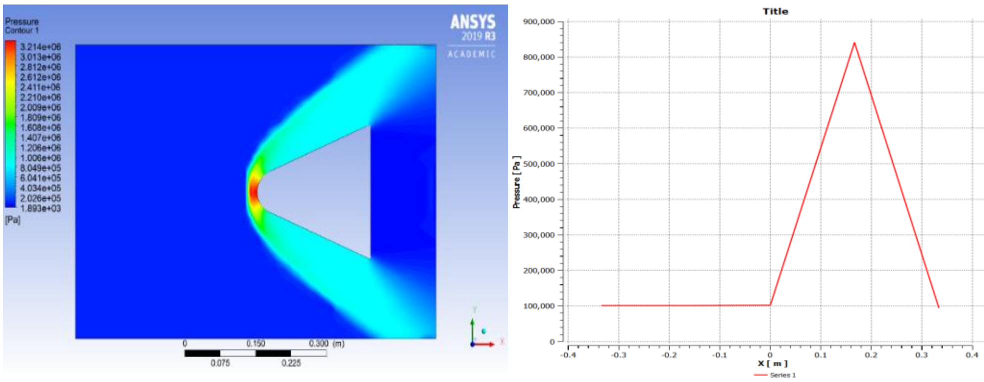

Fig. 12. Contour and graph for pressure.

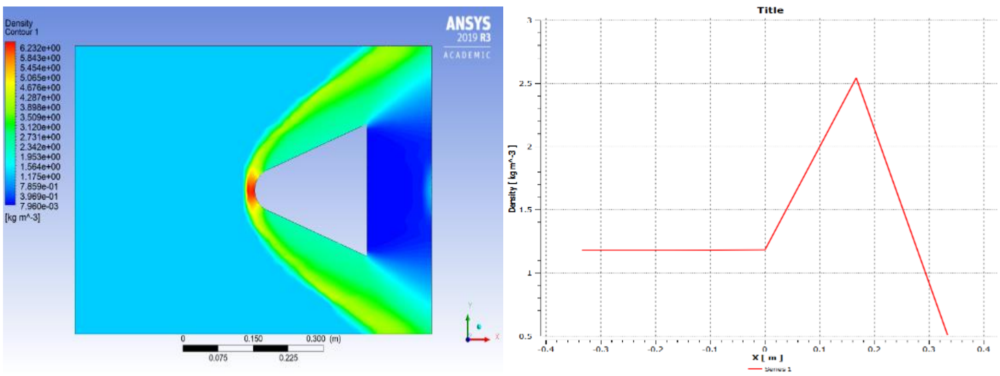

Fig. 13. Contour and graph for density.

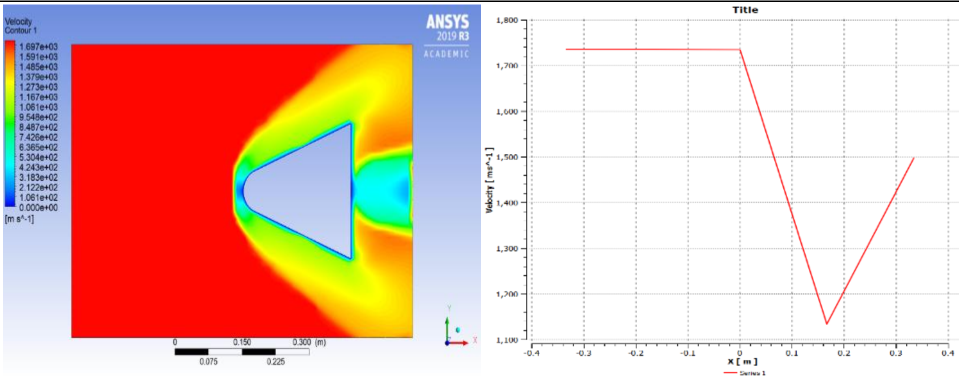

Fig. 14. Contour and graph for velocity.

1. *BI-Conic Nose Cone:*

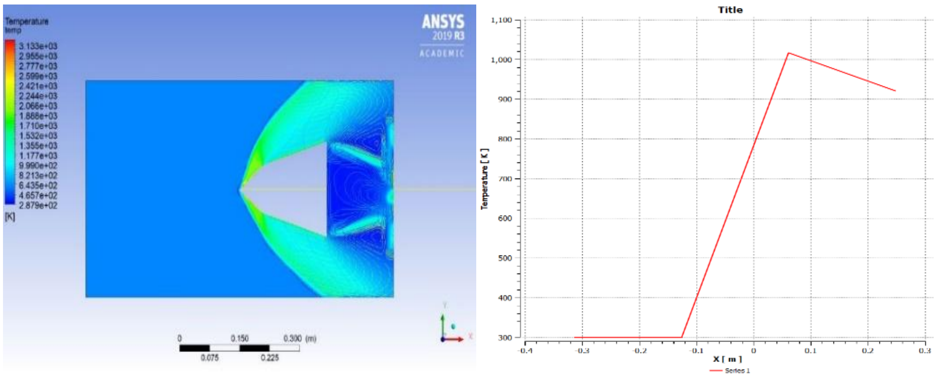

Fig. 15. Contour and graph for temperature.

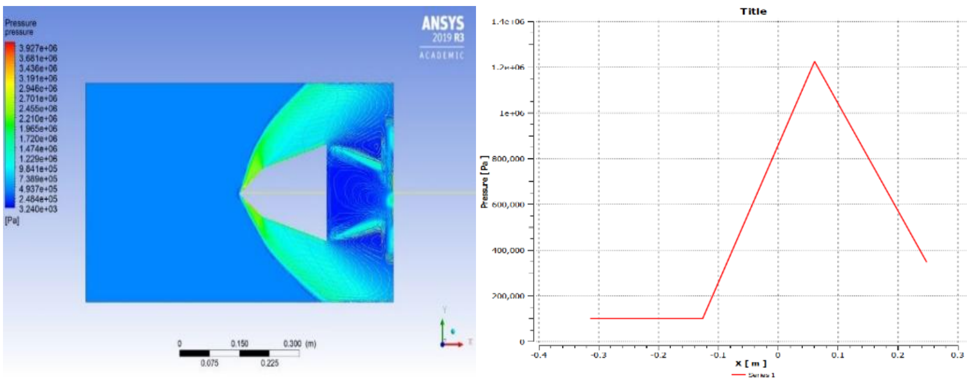

Fig. 16. Contour and graph for pressure.

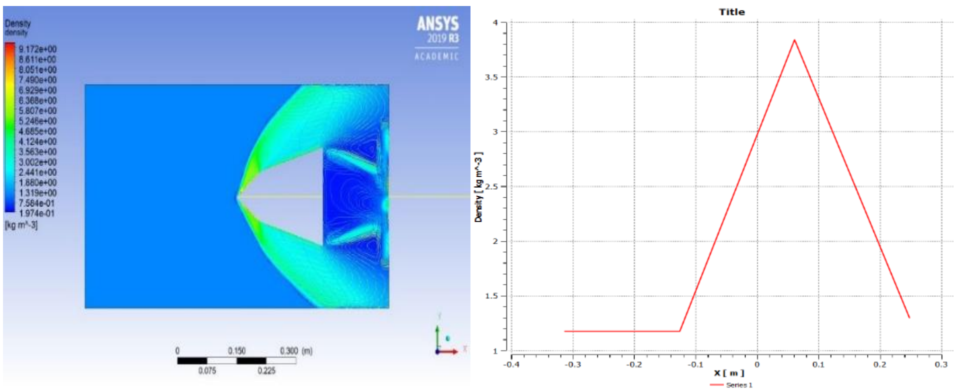

Fig. 17. Contour and graph for density.

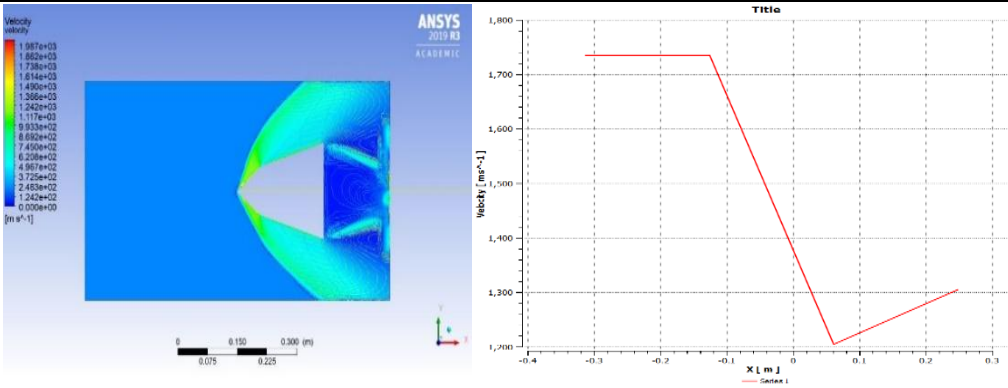

Fig. 18. Contour and graph for velocity.

2. *Tangent Ogive Nose Cone:*

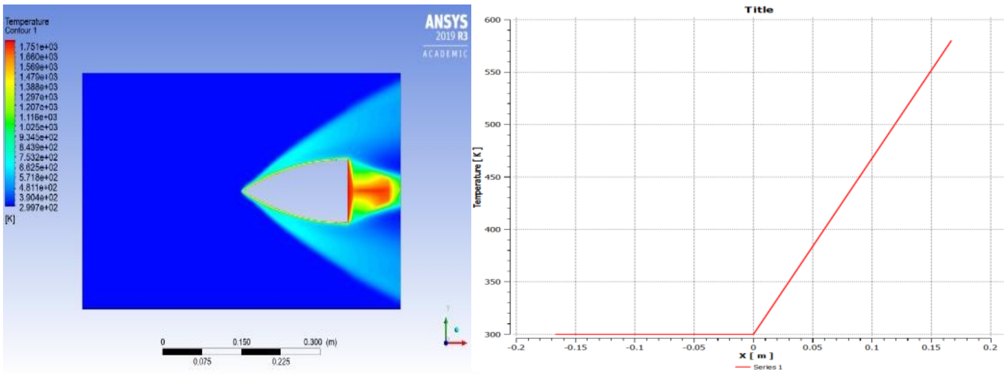

Fig. 19. Contour and graph for temperature.

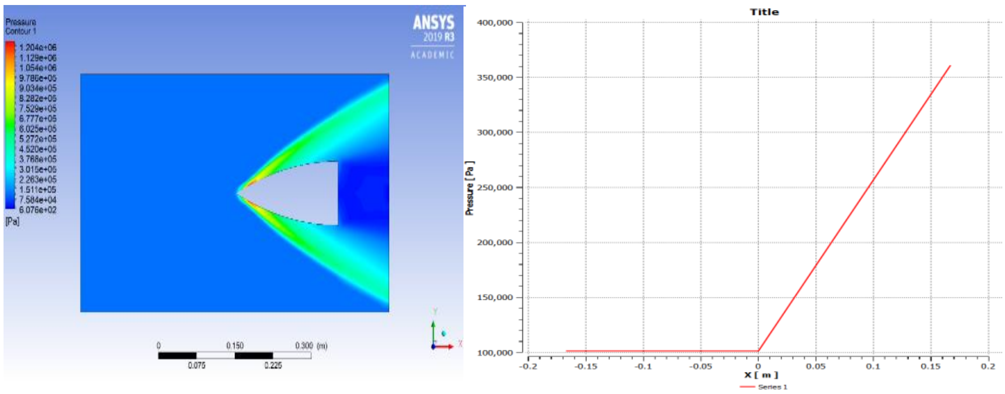

Fig. 20. Contour and graph for pressure.

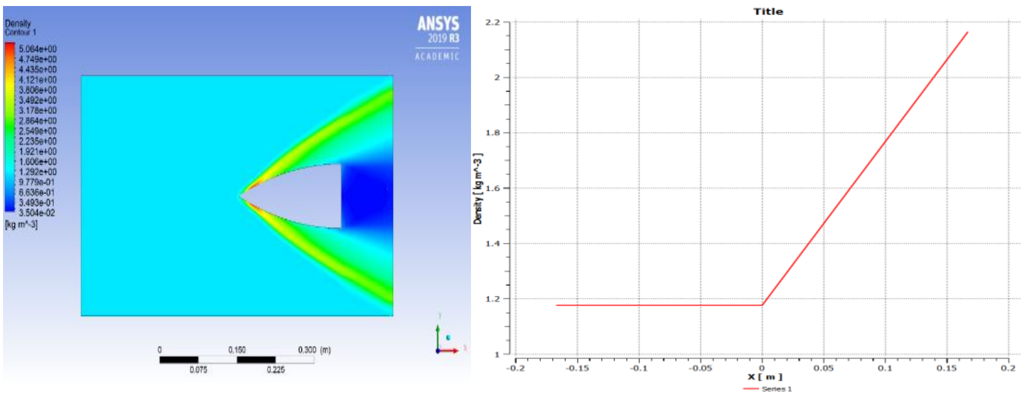

Fig. 21. Contour and graph for density.

International Journal of Engineering Innovation & Research
Volume 10, Issue 4, ISSN: 2277 – 5668

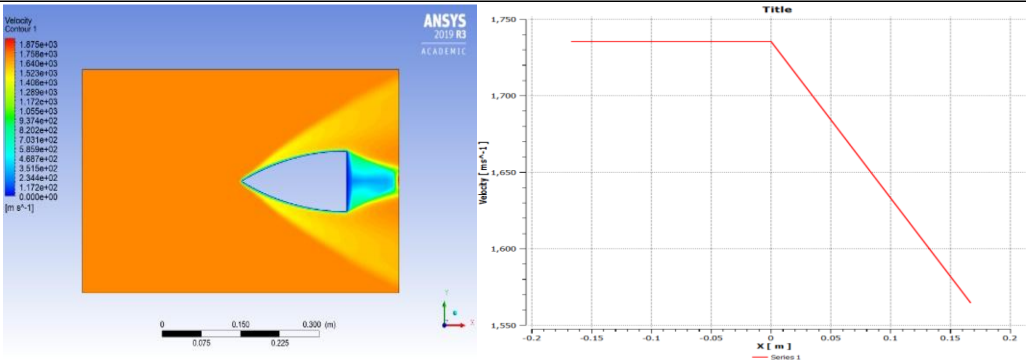

Fig. 22. Contour and graph for velocity.

3. *Spherically Blunted Tangent Ogive Nose Cone:*

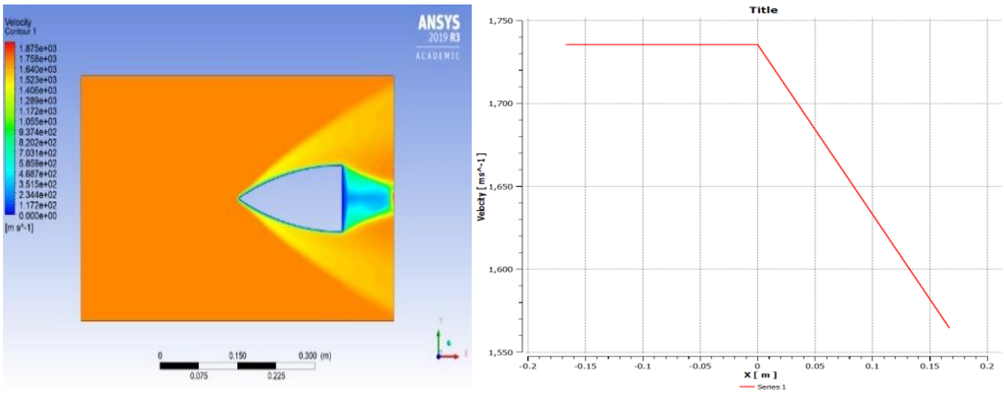

Fig. 23. Contour and graph for velocity.

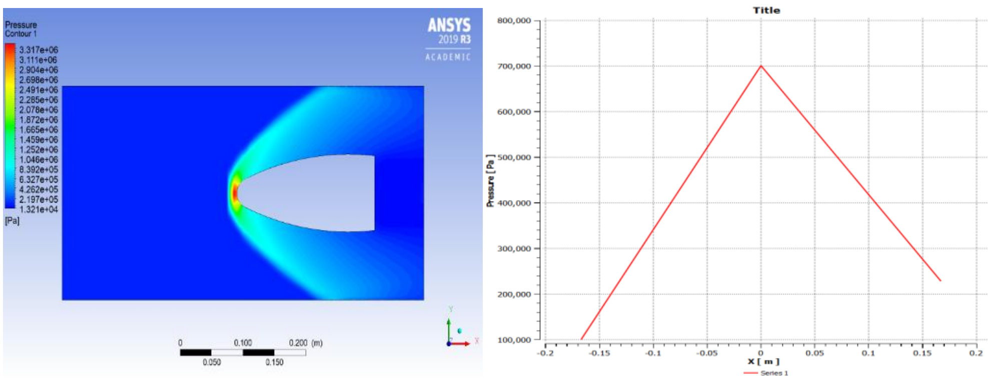

Fig. 24. Contour and graph for pressure.

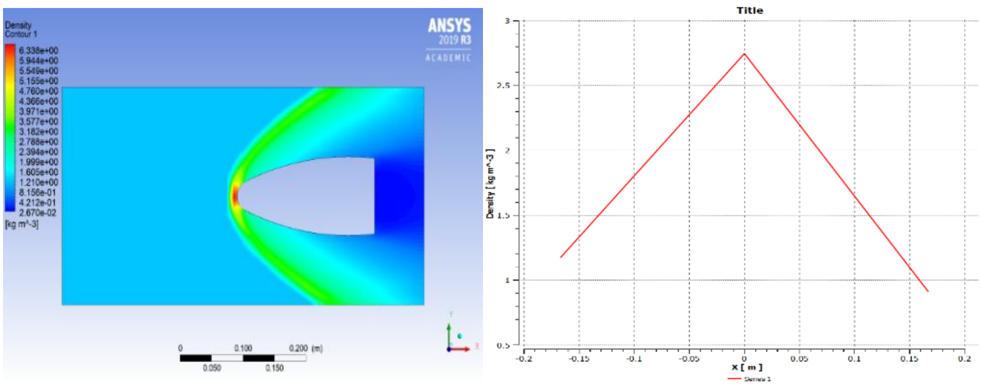

Fig. 25. Contour and graph for density.

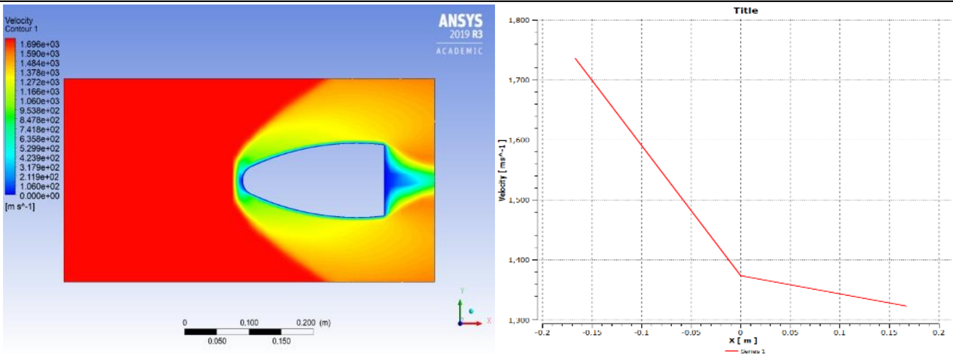

Fig. 26. Contour and graph for velocity.

VIII. Conclusions

After extensive study and comparing results, the "Spherically Blunted Tangent Ogive" shows promising result among all five designs. Based on the Flow analysis and Simulation, this blunt shape proves to be quite effective. A De-Attached Bow shock is produced by the design during simulation. As a result, velocity is reduced by greater margin. We can conclude that the design will likely encounter less pressure force during the flight at hypervelocity. This design is more effective at Heat transfer. This will ensure less damage to the surface and also the thermal stress is comparatively less. Thus, this Blunt shape would be able to sustain extreme atmospheric drag at hypersonic velocity.

Although this design has few drawback and constraints, it is used in spacecrafts by many space agencies. The efficiency of this design is greater in comparison with all five designs which were mentioned earlier in the paper.

References

[1] "Influence of nose radius of blunt cones on drag in supersonic and hypersonic flows", "A. Hemateja, B. Ravi Teja , A. Dileep Kumar , Rakesh S.G", "IOP Conference series: Materials Science and Engineering", "2017".

[2] "Experiments on a blunt cone model in a hypersonic shock tunnel", "Niranjan Sahoo", "16th Australasian Fluid Mechanics Conference", "2007".

[3] "Hypersonic flow past nose cones of different geometries: A Comparative Study", "Ashish Narayan, S Narayanan and Rakesh Kumar", "Simulation: Transactions of the Society for Modeling and Simulation International", "2017".

[4] "Aerothermodynamics design and optimisation of hypersonic research re-entry vehicle", "Manjunatha H, C.N. Nataraj", "International Journal of Science and Research (IJSR)", "2014".

[5] "Computational flow analysis of hypersonic re-entry blunt body using fluent and Gambit", "N.S. Harshavardhan. R; Sanjana K.; Sai Sharan K.; Srinivas G.", "International Journal of Scientific & Engineering Research, Volume 5, Issue 5", "May-2014".

[6] "Influence of nose-tip bluntness on conical boundary layer at Mach 10", "Guillaume Grossir, Fabio Pinna, Olivier Chazot", "AIAA Journal", "1999".

[7] "Rapid prediction of hypersonic blunt body flows for parametric design studies", "W. Schuyler Hinman, Craig T. Johansen", "Aerospace Science and Technology", "2016".

Author's Profile

First Author
Suraj Sambhajirao Patil, He is an Aerospace Engineering student from Lovely Professional University, Punjab, India. He has a strong knowledge about manufacturing & simulations & has done many projects. He is also the Founder of EUROAVIA-LPU which is an European Aerospace Student Originations.

Second Author
Sahil Chandel, He is an Aerospace Engineering Student from Lovely Professional University, Punjab, India. He has own many national level quiz competitions & is part to NCC. He is also member of Team Peregrine-A student Hybrid Rocket Team. email id: sahilchandel3108@gmail.com