Influence of Nose Radius of Blunt Cones on Drag in Supersonic and Hypersonic Flows

A. Hemateja¹, B. Ravi Teja¹, A. Dileep Kumar¹, Rakesh S.G¹{
Department of Mechanical Engineering
Amrita School of Engineering, Bengaluru,
Amrita Vishwa Vidyapeetham, Amrita University, India

¹e-mail: sg_rakesh@blr.amrita.edu

Abstract. The objects moving at high speeds encounter forces which tend to decelerate the objects. This resistance in the medium is termed as drag which is one of the major concerns while designing high speed aircrafts. Another key factor which influences the design is the heat transfer. The main challenge faced by aerospace industries is to design the shape of the flying object that travels at high speeds with optimum values of heat generation and drag. This study deals with computational analysis of sharp and blunt cones with varying cone angles and nose radii. The effect of nose radius on the drag is studied at supersonic and hypersonic flows and at various angles of attack. It is observed that as the nose radius is increased, the heat transfer reduces & the drag increases and vice-versa. Looking at the results, the optimum value of nose radius can be chosen depending on the need of the problem.

Keywords: Aerodynamic Drag, Blunt Cones, Nose Radius, Flow Over Blunt Bodies.

1. Introduction
In order to attenuate aerodynamic drag and heat during the re-entry of spacecraft, flight orientation can be altered by varying the angle of attack. When a spacecraft enters into earth’s atmosphere some involuntary forces may cause the spacecraft to topple and excess amount of heat generated may also cause the spacecraft to disintegrate. Hence a detailed study of the aerodynamic properties is crucial. The flying objects have to be designed accordingly so that they encounter less heat and drag while moving at high speeds through the medium. A lot of research is being carried out in this area and the researchers have come up with an idea of making the nose of the body blunt. Navneeth Soori et al. [1] have reported that even though blunt bodies result in reduction of drag, the heat flux generated on the surface of the body is immense [1]. Studies have been carried out by Luther Neal Jr. et al. [2] to analyse aerodynamic characteristics of a specific cone configuration at an angle of attack ranging up to 180° with various degrees of nose bluntness. The models are spherically blunted with bluntness ratios h/R of 0, 0.32, and 0.65. A blowdown type of tunnel has been considered which has a test duration ranging from 60 to 90 seconds, depending on the stagnation pressure and configuration.

Experiments have been conducted by Reginald M. Machell et al. [3] on flow over blunt-nosed cones at Mach 5.8, over a range of Reynolds numbers to determine the significant parameters governing the pressure distribution. The vertex angle and the nose radius have been varied in the study. The study involves observation of shock shapes and measurement of static pressure on
spherically blunt cones. They have reported that for the models having more bluntness the shock wave shape was dominated by the effects of the blunt nose, whereas, for the more pointed models the shock shape was dominated by the conical portion of the model and the separation distance of the shock wave from the nose of the models at zero yaw varied linearly with the radius of the spherical nose of the model. Wilson F.N. Santos [4] have studied the aerodynamic performance of round leading edges using a series of numerical simulations by parametrically changing the nose radius and angle of attack using Direct Simulation Monte Carlo method. These calculations examine drag coefficient, lift coefficient and lift to drag ratio. The analysis shows that round leading edges provide smaller stagnation point heating and larger total drag coefficient than flat-faced leading edges. They have also reported that lift coefficient decreases by increasing the nose radius of the leading edge whereas heat flux is inversely proportional to the nose radius.

W.E. Moeckel [5] has conducted experiments to study the flow separation ahead of a blunt axi symmetric body at various Mach numbers. It is observed that minimum drag is obtained with rod tip projections equal to about three times the nose radius. Aerodynamic characteristics of breathing blunt nose configuration at hypersonic speeds has been studied by Yasumasa Watanabe et al. [6]. This technique is the most promising technique for reducing the drag. It is found from the results that the breathing results in 5% reduction in drag but the lift to drag ratio remains unchanged. Results obtained from investigation of drag of blunt bodies in free flight at Mach numbers from 0.6 to 2.3 by Harvey A.Wallskog and Roger G. Hart [7] shows that small degree of spherical bluntness on parabolic nose decreases in total drag for Mach no. range from 1 to 1.4 whereas larger degrees of spherical bluntness up to 0.5 times maximum nose radius did not produce an increase in drag up to Mach 1.1. G. Gopala Krishnan et al. [8] and N. Sreekanth et al. [9] have carried out studies on aerodynamic drag reduction by introducing sharp fore-body design like spikes on blunt cones and have reported reduction in the drag considerably.

2. Geometrical modelling
A 2D wedge inside a rectangular enclosure with 400mm x 200mm cross section is considered. The base diameter of the wedge is 50mm and the length of the wedge varies with cone angle and nose radius. The geometry has been divided into a number of faces using face split tool. This helps in giving a finer mesh in the area which is of interest in our study.

Figure 1. Models on which simulations have been carried out
(θ: Cone Angle, AoA: Angle of Attack, NR: Nose Radius)
3. Meshing
A structured mesh is used in order to achieve accurate results. The meshing is done symmetrical about x-axis. The number of elements on each side can be changed using edge sizing technique. The entire geometry has been divided into 50000 to 80000 elements where the minimum size of an element is 0.2mm and maximum size of the element is 22mm. The mesh near the nose is made finer and the area ahead of the shock is left coarse in order to reduce the calculation time. Mapped face meshing is used to ensure uniform meshing throughout the model as shown in Figure 2.

![Figure 2. Sharp and blunt bodies after structured meshing](image)

4. Boundary conditions
For the inlet and walls, pressure far field conditions are defined, whereas for the outlet pressure outlet conditions are set (Figure 3). For the wedge body, stationary wall conditions are preferred. The inputs for pressure far field like static pressure, Mach number, temperature, flow direction, turbulence parameters are obtained from free stream conditions set at an altitude of 25000m above the sea level as shown in Table 1 below.

| Altitude (meters) | Temperature (K) | Pressure (Pa) | Density (Kg/m$^3$) | Viscosity (N-s/m$^2$) |
|------------------|----------------|--------------|-------------------|---------------------|
| 25000            | 221.65         | 2.51102E+3   | 3.94658E-2        | 1.46044E-5          |

![Figure 3. Boundary named selections](image)
5. Simulation technique
Density based approach is used as it the most preferred for compressible high speed flows. Simulations are carried out in 2D, steady state enabling energy equation. K-epsilon standard model is considered. Operating pressure is zero since Mach number is greater than 0.1. The fluid is assumed to be ideal gas and Sutherland viscosity law is used for better accuracy in terms of compressibility and viscous effects (Figure 4).

![Figure 4. Simulation Technique](image)

6. Validation
The values of pressure and temperature obtained after the analysis of the models with zero angle of attack and zero nose radius in the simulation technique are verified with the theoretical calculations. Calculation of one model is shown below.

At inlet the conditions that are considered are: \( P_1 = 2511 \text{ Pa}, \ T_1 = 221.645 \text{ K}, \ M_1 = 2, \ \theta = 15^\circ \)
From oblique shock tables for \( \theta = 15^\circ \) and for \( M = 2, \ \beta = 46.35^\circ \)
From the equation \( Mx_1 = M_1 \times \sin \beta, \ Mx_1 = 1.447 \)
From normal shock tables for \( Mx_1 = 1.48, \ Mx_2 = 0.708, \ P_2/P_1 = 2.389, \ T_2/T_1 = 1.307 \)
From the equation we get \( M_2 = Mx_2/\sin (\beta-\theta) = 1.3608, \ P_2 = 5998.77 \text{ Pa}, \ T_2 = 289.696 \text{ K} \)

The difference between the actual values and the theoretical values for pressure and temperature are 90Pa and 4K.

7. Results
The simulations have been carried out on a number of models and the lowest drag was seen for the cone with 30° cone angle, nose radius of 0mm and Mach number of 8 with zero angle of attack \((Cd=0.252)\) which is 80.49% less compared to the \( Cd \) value of the blunt body with nose radius of 20mm, at Mach 2 and 10° angle of attack \((Cd =1.2685)\). However, it is observed that the heat flux generated in the former is much higher compared to the latter.
8. Density Contours and Graphs

Density contours obtained for few of the cases are shown in Figure 5 to Figure 9.

![Density Contour](image1.png)

**Figure 5.** Density Contour
(θ = 30°, NR = 0, M = 2, AoA = 0°)

![Density Contour](image2.png)

**Figure 6.** Density Contour
(θ = 30°, NR = 0.1D, M = 2, AoA = 0°)

![Density Contour](image3.png)

**Figure 7.** Density Contour
(θ = 30°, NR = 0.2D, M = 2, AoA = 0°)

![Density Contour](image4.png)

**Figure 8.** Density Contour
(θ = 30°, NR = 0.3D, M = 2, AoA = 0°)

![Density Contour](image5.png)

**Figure 9.** Density Contour
(θ = 30°, NR = 0.4D, M = 2, AoA = 0°)

Plots of $C_d$ vs $M$ for various cases are shown in the following graphs (Figure 10 to Figure 14).

![Cd vs Mach No.](image6.png)

**Figure 10.** $C_d$ vs Mach No. plot for all cone angles (NR = 0, AoA = 0°)

![Cd vs Mach No.](image7.png)

**Figure 11.** $C_d$ vs Mach No. plot for all cone angles (NR = 0.1D, AoA = 2.5°)
Figure 12. Cd vs Mach No. plot for all cone angles (NR = 0.2D, AoA = 5°)

Figure 13. Cd vs Mach No. plot for all cone angles (NR = 0.3D, AoA = 7.5°)

Figure 14. Cd vs Mach No. plot for all cone angles (NR = 0.4D, AoA = 10°)

9. Conclusions
If reducing the drag alone is of prime importance, then the sharp cone configuration could be chosen even though they provide lesser heat flux reductions when compared to other configurations. In case of blunt bodies, even though heat flux generated is less, higher drag is experienced. For all values of nose radius, with increase in Mach number the Cd value decreases. With increase in angle of attack, the Cd value increases for all Mach numbers. This study can be extended in multiple directions by considering other configurations, different free stream conditions and other design and analysis parameters.

10. References
[1] Navneeth Soori 2015, Numerical study of heat flux developed on an aero-spike structure in hypersonic flow in Aerospace Conference, 2015 IEEE, pp. 1-7.

[2] Luther Neal, Jr., 1963, Aerodynamic characteristics at a Mach number of 6.77 of a 9 deg cone configuration, with and without spherical after bodies, at angles of attack up to 180 deg with various degrees of nose blunting.

[3] R. M. Machell and W. T. O'Bryant 1956, An Experimental Investigation of the Flow Over Blunt-Nosed Cones at a Mach Number of 5.8°.

[4] Wilson F.N. Santos Nov 2007 Nose radius influence on drag and lift of round leading edges in hypersonic flow p. 11.

[5] W.E. Moeckel Dec. 1951, Flow separation ahead of a blunt axially symmetric body at Mach numbers 1.76 to 2.10 Cleveland, Ohio.
[6] Yasumasa Watanabe, Kojiro Suzuki and Ethirajan Rathakrishnan Aerodynamic characteristics of breathing blunt nose configuration at hypersonic speeds, University of Tokyo, Kashiwa, Chiba, Japan.

[7] Harvey A. Wallskog and Roger G. Hart 1953, Investigation of the drag of blunt nosed bodies of revolution in free flight at mach numbers from 0.6 to 2.3, Langley Field, Va.

[8] G. Gopala Krishnan, Akhil J. and Nagaraja S.R. 2015, Heat Transfer Analysis on Hypersonic Re-entry Vehicles with Spikes, 1st International ISHMT-ASTFE Heat and Mass Transfer Conference IHMT2015, Thiruvannanthapuram, India, December 2015.

[9] N. Sreekanth, J. Akhil and S.R. Nagaraja 2016, Design and Analysis of Secondary Spike on Blunt Head, Indian Journal of Science and Technology, Vol. 9(45), December 2016.