Analysis of various bluff body shapes for hypersonic flight regimes

Kshitij Gandhi¹, Shruti Dipak Jadav², Simran T³ and Ankit Kumar Mishra⁴

¹ Department of Aeronautical Engineering, School of Aeronautics, Rajasthan, India
² Department of Aerospace Engineering, Sandip University, Maharashtra, India
³ Department of Aerospace Engineering, Sastra University, Tamil Nadu, India
⁴ Research Lead, Centre for Advanced Research and Development, LIPS Research Foundation, Kozhikode, Kerala, India

* kshitijgandhi10@gmail.com

Abstract. This paper deals with the analysis of fluid flow around bluff bodies using computational fluid dynamics. Two-dimensional rectangular body, semi-circular tip, triangular tip and elliptical nose cone shapes have been considered in the paper. The shapes are analysed at various Mach numbers, ranging from Mach 1.5 to Mach 10. The aerodynamic characteristics like Drag coefficient and drag force are investigated. Also, the formation of shock wave at higher flow regimes and its effects on the velocity and pressure are observed and analysed.

1. Introduction
Bluff bodies have a great impact on engineering applications (aeronautical, Aerospace, civil, mechanical, naval & Oceanic) & have their origin associated with the development of interaction between two shear layers and boundary layers including laminar, transition and turbulent flows. Surface like a circular cylinder which is commonly studied because of its collection of flow phenomena and surface roughness around a single rough circular cylinder is concerned with the complex flow over a bluff body. In all engineering applications, surface roughness influence & free stream turbulence are important & common disturbances. The distinct flow regimes observed during the flow past a bluff body are difference in the flow pattern, separation angle & drag coefficient, categorized base on the Flow characteristics. We can define bluff body as a body that, due to its shape causes the flow to separate over its surface. The flow separation occurs due to formation of high-pressure regions at edges or surfaces, unsteady vortices are also formed. These phenomena lead to formation of high drag and unsteady forces opposite to the flow direction.

The total drag on the body is mostly due to two reasons, the skin friction and the pressure. The pressure drag is minimum at subsonic speeds but as we move up to higher speeds the pressure drag increases and becomes a major contributor to the total drag. Pressure drag depends on the frontal area of contact of the body with the fluid. More the frontal area, more is the drag on that body. So it is, recommended to use pointed shapes at higher speeds but other criterion like the fineness ratio need to be also considered.

The k-omega (k−ω) turbulence model is mostly used for capturing the effects of turbulent flow conditions. It used a two-equation model. It not only solves the basic conservation equations, it also considers two transport equations. The two variables (k) and (ω) determine the energy and rate of dissipation respectively. The k-omega SST model is best among all the RANS models for prediction of...
flow separation. It also has better performance in adverse pressure gradients. It is a highly used method in the industry, due to its faster performance and high accuracy.

2. Geometry and Mesh preparation
The geometries are constructed two-dimensionally using the in-built Ansys software – Space Claim. The various shapes were decided mostly on the basis of their aerodynamic designs. These particular shapes are used so that we can observe the difference in the aerodynamic performance due to the changes in the shape. The length of all the shapes is kept constant – 56mm and maximum thickness of all the shapes is 12mm.

![Figure 1. Circular Tip Geometry.](image1.png)
![Figure 2. Rectangular Geometry.](image2.png)

![Figure 3. Triangular Tip Geometry.](image3.png)
![Figure 4. Elliptical nosecone Geometry.](image4.png)

| Body shapes | Length(mm) | Width(mm) |
|-------------|------------|-----------|
| Rectangle   | 56         | 12        |
| Triangular tip | 56     | 12        |
| Circular tip | 56      | 12        |
| Elliptical  | 56         | 12        |

The meshing is done on the default Ansys meshing software. The face meshing feature is used and multiple edge sizing have been applied to make the mesh finer.
3. Boundary conditions and CFD setup

The velocity inlet and pressure outlet condition are used for the analysis. The bluff body is kept as no-slip wall. The velocity is changed at the inlet for analysis of the shapes at various Mach numbers. For the setup, a pressure-based solver is used. The energy equation is turned on and SST K-omega turbulence model is used. The air is kept as ideal gas, to capture the compressibility effects on density due to high Reynolds number. In the Boundary conditions, the operating pressure is kept at sea level for all the cases. The scheme is kept as coupled and Green gauss node-based method is used. The second order upwind discretization is used.

4. Governing Equations

Governing Equations for SST k-omega turbulence model-

Turbulence Kinetic Energy

\[
\frac{\partial k}{\partial t} + U_j \frac{\partial k}{\partial x_j} = P_k - \beta^* k \omega + \frac{\partial}{\partial x_j} \left[ (v + \sigma_k v_T) \frac{\partial k}{\partial x_j} \right] \]

Specific Dissipation Rate

\[
\frac{\partial k}{\partial t} + U_j \frac{\partial k}{\partial x_j} = \alpha S^2 - \beta \omega^2 + \frac{\partial}{\partial x_j} \left[ (v + \sigma_k v_T) \frac{\partial k}{\partial x_j} \right] + 2(1 - F_2) \sigma_\omega \omega \frac{1}{\omega} \frac{\partial k}{\partial x_i} \frac{\partial \omega}{\partial x_i} \]

---

Table 2. Mesh details.

| Body shapes  | Number of elements | Number of nodes |
|--------------|--------------------|-----------------|
| Rectangle    | 30391              | 30808           |
| Triangular tip | 31010              | 31202           |
| Circular tip | 30876              | 30902           |
| Elliptical   | 30031              | 30193           |
Kinematic Eddy Viscosity

\[ v_T = \frac{a_1 k}{\max(a_1 \omega, SF_2)} \]  

Equation for Drag Calculation –

\[ D = \frac{1}{2} \rho v^2 C_d S \]  

5. Results
The simulations were run till the solution converged for all the cases. The contours of velocity and pressure were observed.

Figure 7. Rectangular body Velocity contour.  
Figure 8. Circular tip Velocity contour.  
Figure 9. Triangular Tip Velocity contour.  
Figure 10. Elliptical nosecone velocity contour.  
Figure 11. Triangular tip pressure contour.  
Figure 12. Rectangular body pressure contour.  
Figure 13. Circular Tip pressure contour.  
Figure 14. Elliptical nosecone pressure contour.
Figure 7, 8, 9 and 10 show the velocity contours of each shape at Mach 2. It can be seen that the velocity drops by a huge margin across the shock wave that is formed ahead of the bluff bodies. The maximum drop in velocity is observed in the rectangular body and least can be seen in the elliptical body. Figure 11, 12, 13 and 14 show the pressure contours of the shapes at Mach 2. We can observe pressure concentration at the tip of the shapes and also the formation of the shockwave due to which we observe the velocity drop. The concentration of pressure is dependent on the contact area of the shape with the fluid flow and this leads to formation of drag on body. It is also observed that as we move up to higher Mach numbers the shock cone angle decreases and much more higher concentration of pressure on the body which then causes the increase in temperature. The drag coefficient and drag forces being experienced by each shape at different Mach numbers is listed in the Table 3 and 4. It is observed that the elliptical nosecone shape experiences the least drag among the other selected shapes and as we move up to higher Mach numbers we see that the drag coefficient drops down for the elliptical shape.

| Mach Number | $C_d$ of Rectangular body | $C_d$ of Triangular tip | $C_d$ of Circular tip | $C_d$ of Elliptical nose shape |
|-------------|---------------------------|-------------------------|-----------------------|-------------------------------|
| 1.5         | 0.164                     | 0.122                   | 0.117                 | 0.098                         |
| 2           | 0.186                     | 0.150                   | 0.137                 | 0.081                         |
| 3           | 0.185                     | 0.134                   | 0.135                 | 0.063                         |
| 5           | 0.180                     | 0.139                   | 0.128                 | 0.046                         |
| 10          | 0.190                     | 0.124                   | 0.115                 | 0.040                         |
Table 4. Drag Force at various Mach numbers.

| Mach Number | Drag of Rectangular body (N) | Drag of Triangular tip (N) | Drag of Circular tip (N) | Drag of Elliptical nose shape (N) |
|-------------|------------------------------|---------------------------|-------------------------|----------------------------------|
| 1.5         | 2825.56                      | 2199.12                   | 2129.92                 | 1718.83                          |
| 2           | 5684.78                      | 5019.59                   | 4628.57                 | 2519.47                          |
| 3           | 12746.82                     | 9619.14                   | 9886.84                 | 4211.20                          |
| 5           | 34462.85                     | 27817.04                  | 26023.42                | 9084.50                          |
| 10          | 144458.53                    | 98920.07                  | 93313.456               | 36810.8                          |

Figure 17. Drag Coefficient at various Mach numbers.

Figure 18. Drag force at various Mach Numbers.
6. Conclusion

In the result section, four different shapes, mainly two-dimensional rectangular shape, circular tip, triangular tip and elliptical nosecone, were analysed at various Mach numbers. The analysis was done in supersonic and hypersonic flow regimes. The results were obtained using the software - ANSYS Fluent. The drag coefficients, drag force and various other aerodynamic factors affecting a bluff body were studied and analysed. It is concluded that the elliptical shape is aerodynamically better shape, for hypersonic flow regimes, among the other selected shapes.

7. References

[1] Gera. B et al., CFD analysis of 2D unsteady flow around a square cylinder, International Journal Applied Engineering Research, Din Digul, Volume 1, No. 3 2010
[2] Sibha Veerendra Singh et al., CFD Analysis of Different Bluff Bodies, International Journal of Novel Research in Electrical and Mechanical Engineering, Volume-2, Issue-3, Sept-Dec 2015
[3] Thakur, V Yadav, T Rajiv B , 2017 Drag Optimization of Bluff Bodies Using CFD for Aerodynamic Applications, International Journal of Computational Engineering Research (IJCER), ISSN 2250-3005 7(4)
[4] Ying, X Y Xu, F Y, Zhang, Numerical Simulation and Visualization of flow Around Rectangular Bluff Bodies, The Seventh International Colloquium on Bluff Body Aerodynamics and Application, September 2-6
[5] Suresh, V, Senthilkumar, C, Nadaraja Pillai, S, Arunvinthan, Fore body Effect on Bluff Body Drag Reduction at Low Reynolds Number, International Journal of Applied Engineering Research, ISSN 0973-4562 10(33)
[6] Scott M., Murman, Lift and Drag Behaviour of unconstrained Bluff bodies, NASA/TM-2010-216406 [7] Pankaj Kumar, Santosh Kumar Singh, Flow past a bluff body subjected to lower subcritical Reynolds number, Journal of Ocean Engineering and Science, 13:6
[8] Alex Mendonca Bimbato, et al., Study of Surface Roughness on a bluff body- the formation of Asymmetric separation bubbles, Energies 13, 6094
[9] Thirumalai R and Sadhasivam C., Experimental Measurement and Computational Investigation on the Effect of Inlet Winglet Turbulators on the Heat Exchange Behaviour of Radiator of Tubes, Journal of Applied Fluid Mechanics, Vol. 11, Special Issue, pp. 135-139, 2018.