Design and Analysis of Secondary Spike on Blunt Head

N. Sreekanth*, J. Akhil and S. R. Nagaraja

Department of Mechanical Engineering, Amrita School of Engineering, Bengaluru - 560035, Karnataka, India; sreekanth1990@gmail.com, sr_nagaraja@blr.amrita.edu, j_akhil@blr.amrita.edu

Abstract

The present paper investigates the heat flux and drag reduction techniques on a hypersonic re-entry vehicle. Many researchers have reported a reduction in these parameters by the introduction of an aero spike at the stagnation point of the blunt head of a re-entry vehicle. The effect of introducing a secondary surface (called the secondary spike), along with the aero spike, on the blunt head is discussed in this work. The study has been carried out by varying the location of the secondary spike with respect to the axis of the vehicle for a set of free stream conditions. Also, the effect of variation in free stream conditions on a re-entry vehicle with a primary and secondary spike has been investigated. It has been observed that some configurations work well for some free stream conditions but fail in comparison to a single spiked body for some other conditions.

1. Introduction

The design of re-entry vehicles has been a major area of study in the field of hypersonic flow in the recent past. Two of the major design constraints for a re-entry vehicle are aerodynamic heating and drag force. While a reduction in the drag will be experienced by streamlined bodies, this will generate problems like attached shock and a very thin shock layer, resulting in an increased heat flux. Thus, these constraints are conflicting in nature and conventional measures for addressing one of these will result in an adverse situation with respect to the other. Because of this, strategies to reduce both drag and aerodynamic heating have been explored widely by many researchers. A large number of drag and heat flux reduction techniques proposed by researchers include the concept of spikes. Spikes are slender bodies attached at the stagnation point of a re-entry vehicle, which displaces the bow shock away from the blunt head. It has been reported that the attachment of the spike alters the flow field in such a way that the pressure and temperature ahead of the blunt body is reduced, thereby reducing the drag and heat flux values on it. The heat flux values at the stagnation point of the spike will still be high, but there is a huge advantage in using thermal protection on the spike tip, rather than on the entire blunt head. In most of these studies, the reattachment of the shock wave on the blunt body has also been reported. This is critical because reattachment increases the drag and heat flux values on the blunt head to a great extent. From available literature, it has been seen that longer spikes delay the reattachment on the blunt surface. But larger values of L/D (ratio of spike length to the diameter of the blunt head) increase the stability concerns related to the design. As a means to prevent reattachment of the shock wave on the blunt body with an aero spike, the introduction of a secondary surface (called secondary spike) on the blunt body is proposed and analyzed in this work. As the reproduction of the actual conditions on a ground based laboratory is both expensive and technically challenging, computational methods are used, which can give results that are reasonable approximations of the actual flow phenomena. In many studies, numerical results so obtained have been compared with experimental and flow visualization data, yielding satisfactory agreement. Comparison of heat flux and drag coefficients for the vehicle with a secondary
spike have been done with the ones with only the primary spike, and the ones without a spike (blunt bodies).

2. Literature Review

A study of hypersonic flow over a spiked blunt cone was done by Gopalan et al.\(^1\). The flow field around a large apex angled spiked blunt cone has been analyzed at a hypersonic Mach number through experiments in a free piston-driven shock tunnel, and the results were compared with numerical results obtained from an in-house unsteady Navier-Stokes solver for laminar 2-Dimensional axisymmetric flows. Results show about 40–55% reduction in drag for the blunt cone using flat and hemispherical aero discs at lower angles of attack (up to 5). The work done by Tanno et al.\(^2\), shed light on a new force measurement technique for shock tunnels. Here the model was allowed to fall freely in the test section and was caught safely at the bottom. This method eliminates the requirement of mounting devices, thereby minimizing the effect of extraneous variables on the measurement. An analysis of Hypersonic flow over a multi-step after body was performed by Menezes et al.\(^3\), where the effect of a multi-step base on the total drag of a missile shaped body was studied in a shock tunnel at a hypersonic Mach number of 5.75. Experimental results indicated a reduction of 8% in total drag over the body with a multi-step base in comparison with the base-line (model with a flat base) configuration. The drag reduction in spherical spiked blunt body was studied by Mansour et al.\(^4\). Hypersonic flow with Mach number 6 over a spiked body was numerically simulated. The computed results show that the drag coefficient of the spiked blunt body is reduced with respect to no spiked body. Davis H. Crawford\(^5\) conducted studies on spiked blunt body under different Reynolds number. The paper emphasis on the reattachment point and the variation of pressure distribution and corresponding heat flux values at different Reynolds number. Studies conducted by Staylor et al.\(^6\) and Maull et al.\(^7\), among others, emphasize the strong dependence of the flow field on the ratio of length of the spike to the diameter of the blunt head (L/D). A value of 1.25 for the L/D was selected considering the data published in these works. Rajesh Yadav et al.\(^8\) studied spiked re-entry vehicles at a Mach number of 10.1 at a static pressure and temperature of 16066 Pa and 216.65 K respectively. Small reductions of 9% and 5% in the in reattachment heat flux and comparatively larger reduction of 44% in the drag force was observed. Ajinkya Kumar et al.\(^9\), studied on the temperature dependence on the aerodynamic drag. It was found from the experimental result that, as the flow temperature was increased a corresponding increase drag. Jiang, et al.\(^10\) experimentally investigated the reduction of drag and heat transfer by the combination of spike and jet on the tip of the spike at 40 angle of attack. It was observed that the spike altered the shock in front of blunt body and the jet reduced the aerodynamic heating on the spike tip.

3. Models and Boundary Conditions

The model analyzed here is a hemispherical nose with a cylindrical after-body. The diameter of the body, D, is 40 mm. The length of the after body is also 40 mm. An aero spike with an L/D ratio of 1.25 is introduced at the nose of the hemisphere. The end of the spike is a hemispherical section. A circumferential projection, which is termed as the secondary spike, is introduced on the hemispherical body, as shown in Figure 1. The secondary spike extends to a length that is 0.6 times the diameter of the hemisphere. The analysis is done by varying the location of the secondary spike with respect to the axis of the body. Under the same free stream conditions, the flow field over the model with the secondary spike located at distances of 5 mm, 10 mm and 15 mm from the axis are analyzed. The results of these analyses are compared with the model having a single spike, and one without a spike.

The fluid is assumed to be ideal gas which follows Sutherland equation on the thermal conductivity, with dynamic viscosity of \(1.794 \times 10^5\) Ns/m². Computational efforts can be drastically reduced by the assumption of axisymmetric flows when a 2D mesh can be used for computation. An axis boundary condition has been implemented on the longitudinal axis of the Hemisphere-Cylinder in this problem across which the radial momentum flux and the gradients are zero. The body of the hemisphere-cylinder is modeled as a fully catalytic wall at a fixed temperature of 300 K with a no slip velocity condition for viscous flows (\(u=0, v=0\)). The boundaries of the fluid domain are treated with the pressure far-field, where the free-stream pressure of 16066 Pa, and temperature of 216.65 K, and a Mach number of 10.1 was used. The effect of location of the secondary spike was modeled under these conditions. In the second part of the study, conducted on the 10mm secondary spike, Mach number values of 2, 5 and 7 were used to study the flow field, keep-
4. Results and Discussion

ANSYS 14.5 is used to analyze the flow over the mentioned models. Flow over a body with a single spike was simulated first to validate the solver. The results so obtained were in agreement with experimental results available in literature.

In the first part of the study, the free stream conditions are fixed and the location of the secondary spike is varied by changing the distance from the axis. Simulations are done for distances of 5 mm, 10 mm and 15 mm from the axis. Figure 2 shows the heat flux plot \( q/q_0 \) vs \( x/R \) and the temperature contour. It can be seen from the plot that there is not much improvement in the heat flux reduction in comparison to the case with single spike. The reattachment of the shock wave on the blunt head can be seen in the temperature contour.

The pressure plot \( p/p_0 \) vs \( x/R \) and the pressure contour shown in figure 3 also show no significant improvement in the drag reduction. The drag coefficient \( CD \) value was seen to reduce from 0.17 for a single spike, to 0.16 due to the addition of the secondary spike.

Figures 4 and 5 shows the heat flux and pressure plots, and the temperature and pressure contours for the secondary spike located at 10 mm from the axis. In comparison with the single spike, the secondary spike at 10 mm shows an improvement both in terms of drag reduction and heat flux reduction. From the temperature and
pressure contours, it can be observed that there is only a weak reattachment, thus reducing the peak heat flux and the peak pressure significantly. A drag reduction of almost 30% and a heat flux reduction of 52.92% are observed because of this.

It can be observed from Figure 6, that the heat flux reduction is much more enhanced when the secondary spike is located at 15 mm from the axis. The temperature contour shows that the reattachment is very weak, resulting in a drastic reduction of heat flux in comparison with the single spike.

But in terms of drag, as can be observed from the plot and contour in Figure 7, this configuration presents unfavorable results. The pressure of the fluid between the two spikes is on the higher side, thus resulting in an increase in the drag coefficient. While the heat flux value reduced by 88.23% in comparison to a single spiked model, the
Figure 5.  $p/p_0$ plot and pressure contour of 10 mm secondary spike.

Figure 6.  $q/q_0$ plot and temperature contour of 15 mm secondary spike model.

Figure 7.  $p/p_0$ plot and pressure contour of 15 mm secondary spike.
drag coefficient suffered an increase by almost 60%, from 0.1722 to 0.27.

The second part of the work aims at studying the effect of variation of free-stream conditions, specifically the Mach number, on the flow-field over a body with the secondary spike. With similar treatment for boundaries as mentioned above, the simulations were run for Mach numbers of 2, 5 and 7. For this, the model with the secondary spike located 10 mm away from the axis was selected, as it gave favorable results with respect to both our design constraints. Similar graphs as above were plotted by making the pressure and heat flux non-dimensional, by expressing them as ratios with the corresponding maximum values ($p_{\text{max, ns}}$ and $q_{\text{max, ns}}$) obtained for a blunt body without spike under the same conditions.

Figures 8 and 9 show that both heat flux and pressure are higher compared to the body with a single spike for a flow Mach number of 2. The shock reattachment, which was very weak for a Mach number of 10.1, is a lot more significant in this case, as can be observed from the pressure and temperature contours. The value of CD is 0.363 which is slightly higher than that for the single-spiked body, which was 0.3527.

The plots for heat flux and pressure variation for Mach 5 and 7 can be seen in Figure 10. For a Mach number of 5, we see an increase in the peak heat flux and a reduction in the peak pressure in comparison with a body with a single spike. Peak heat flux is almost twice that of the single spiked body, where as CD is reduced to 0.186 in comparison with a single spiked model having a CD of 0.216. There is reduction in peak pressure, but a substan-

![Figure 8](image1.png)  
**Figure 8.** $q/q_{\text{max, ns}}$ plot and temperature contour 10mm secondary spike at M=2.

![Figure 9](image2.png)  
**Figure 9.** $p/p_{\text{max, ns}}$ plot and pressure contour 10mm secondary spike at M=2.
The employment of secondary spike along with other methods of heat flux and drag reduction techniques, such as opposed jet injection and energy deposition, can also be considered and studied.

5. Conclusion and Future Work

The effects of secondary spike on drag and heat flux reduction is studied and the results obtained show that secondary spike can be employed as means to reduce the drag and heat flux values. However, it is also observed that the location of the secondary spike and the free stream conditions are critical in determining the effectiveness of the secondary spike. Employing the secondary spike could result in detrimental effects, as observed in some simulations. A more extensive study investigating the effect of parameters like secondary spike length should also be done in order to get a clear picture of the impact of secondary spike on the flow field. The addition of secondary spike could result in structural stability issues, which should also be studied before it can be implemented. Also, employment of secondary spike along with other methods of heat flux and drag reduction techniques, such as opposed jet injection and energy deposition, can also be considered and studied.

6. References

1. Gopalan J, Menezes V, Reddy KPJ, Hashimoto T, Sun M, Saito T, Takayama K. Flow fields of a large-angle, spiked blunt cone at hypersonic Mach numbers. Trans Jpn Soc Aeronaut Space Sci. 2005; 48(160):110–6.
2. Hideyuki T, Tomoyuki K, Kazuo S, Katsuhiro I, Masahiro T, Kazuhisa F, Lawrence S, Hannemann K. Free-flight force measurement technique in shock tunnel. AIAA.
3. Menezes V, Kumar S, Maruta K, Reddy KPJ, Takayama K. Hypersonic flow over a multi-step afterbody, Shock Waves. 2005; 14(5/6):421-4.
4. Mansour K. The drag reduction in spherical spiked blunt body. Acta Astronautica. 2014; 99:92-8.
5. Crawford DH. Investigation of the flow over a spiked-nose hemisphere-cylinder at a Mach number of 6.8. Technical note, D-118. National Aeronautics and Space Administration.
6. Staylor WF. Flow field investigation for large-angles cone with short spikes at a mach no of 9.6. Nasa Technical Note NASA TND-5754. 1970
7. Maull DJ. Hypersonic flow over axially symmetric spiked bodies. J Fluid Mech 1960; 8(P-4):584-92.
8. Yadav R, Velidi G, Guven U. Aerothermodynamics of generic re-entry vehicle with a series of aerospikes at nose. Acta Astronautica. 2014; 96:1–10.

9. Krishnakumar A, Sharma P, Tambi Y, Ashok B. Establishing temperature dependency of the aerodynamic drag using CFD and experimental analysis. IJST. 2016; 9(31).

10. Jiang Z. Experimental demonstration of a new concept of drag reduction and thermal protection for hypersonic vehicles. Acta Mech Sin. 2009; 25:417-9.