International Journal of Mechanical Engineering and Technology (IJMET)
Volume 8, Issue 10, October 2017, pp. 878–885, Article ID: IJMET_08_10_095
Available online at http://www.iaeme.com/IJMET/issues.asp?JType=IJMET&VType=8&IType=10
ISSN Print: 0976-6340 and ISSN Online: 0976-6359

DRAG REDUCTION FOR HYPERSONIC RE-ENTRY VEHICLES

G. Gopala Krishnan, Akhil and Nagaraja S R
Department of Mechanical Engineering, Amrita University, Bengaluru, India

ABSTRACT

The design of re-entry vehicles has been going on for decades and various researchers around the world have proposed different design strategies for reducing the aerodynamic drag and heating. The fact that these parameters are conflicting makes the design process all the more complicated. Using sharp fore-bodies would reduce the drag, but will provide only a small area to disperse the heat flux downstream of the shock wave. If a blunt fore-body is used, the increased area helps in efficient heat dissipation, but the drag force experienced increases drastically. Of the many strategies studied to counter these two effects, the use of spikes at the nose of blunt bodies, and injection of a counter flowing jet at the nose are common. In this paper, a numerical study is done on re-entry vehicles with spikes from the point-of-view of drag reduction. ANSYS Fluent is used to perform the analysis on blunt bodies with single-spike and twin-spike configurations, and the effect of counter flowing jet injection is discussed. The results indicate drag reduction ranging from 44% to 61%.

Keywords: Drag reduction, hypersonic re-entry vehicles

Cite this Article: G. Gopala Krishnan, Akhil and Nagaraja S R, Drag reduction for Hypersonic Re-entry Vehicles, International Journal of Mechanical Engineering and Technology 8(10), 2017, pp. 878–885.
http://www.iaeme.com/IJMET/issues.asp?JType=IJMET&VType=8&IType=10

1. INTRODUCTION

The various regimes of flow based on the Mach number, the hypersonic regime is arguably the most complicated one in terms of analysis. The effects like the formation of a thin shock layer and entropy layer, viscous interactions and very high temperatures become predominant as the Mach number increases. These are some of the reasons for the increased complexity. Because of these phenomena, hypersonic flow has to be considered distinct from supersonic flow, and hence, the design strategies to be adopted for hypersonic vehicles very vastly from those for supersonic vehicles. Designing vehicles capable of performing re-entry to the atmosphere is one of the major focus areas in hypersonic flow research. Mach numbers as high as 36 could be encountered during the re-entry of a space craft into the atmosphere. Such high Mach numbers result in two major issues that have to be considered during the design phase: Aerodynamic drag and Aerodynamic heating. The conflicting nature of these issues calls for non-conventional approaches for reducing both drag and heating. In terms of heat flux, blunt bodies are preferred, but the presence of the strong bow shock in front of the body results in a large wave drag. Strategies like employing a spike at the nose of a blunt body, injecting a counter flowing jet, energy deposition and firing a projectile from the fore-body are being studied experimentally and numerically. What is common in these techniques is that the flow field ahead of the blunt body is varied in such a way that the heat flux and pressure encountered at the blunt surface is reduced.

Maull [1] investigated the flow over axisymmetric spiked bodies at a Mach number of 6.8 and showed that the pressure and heat flux distribution depended primarily on the *l/d* (length of spike to blunt body diameter) ratio. For some ranges of *l/d* ratio the flow was found to be unsteady resulting in oscillations which pose a serious structural stability issue. The practical use of a spike must therefore be limited to bodies which do not exhibit these unsteady changes in flow pattern. Analysis of hemispherical bodies with spike showed steady flows. Mansour.et.al. [2] numerically analyzed drag reduction in spherical spiked blunt body at the Mach number of 6. Reynolds Average Naiver–Stoke equations were solved numerically and standard k–ε model was used for turbulence modeling. The computed results showed that at *l/d*=1.5 the drag has been reduced by 40% of the drag which is produced on no spike model. Numerical study of 2–D axisymmetric re-entry model was done by Rajesh Yadav, et al. [3]. The model had multiple spikes and these are arranged in series. These are mounted at the stagnation point of the blunt body. The model was analyzed for flow of chemically reacting air at Mach 10.1 for various *l/d* ratios and the results were compared with that of re-entry model without spike. An overall *l/*d ratio of 1.5 for the model with spikes in series, with length ratio (Fig. 1) l_1/l_2 = 1.5 showed best results for drag reduction. Kalimuthu et.al. [4], have experimentally studied the effect of the spike length, shape, spike nose configuration and angle of attack on the reduction of the drag using hypersonic wind tunnel at Mach 6. Among the various spikes experimented, the inclusion of hemispherical disk at the tip of the spike showed higher reduction in drag at various angles of attack. Free piston-driven shock tunnel experiments were carried out by Jagadeesh et.al [5] to study the flow field around a large angled spiked blunt cone. Shock oscillations near the cone were observed, and they were more pronounced for a conical spiked model compared to a disc spiked one. Numerical results in agreement with experimental results were obtained using an in-house 2D axisymmetric unsteady Navier-Stokes solver. An investigation on drag and heat flux reduction on highly blunted cones with and without spikes at various angles of attack in a hypersonic shock tunnel was carried out by Menezes et.al [6]. Force and heat flux measurements along with electric discharge visualization results were compared with numerical results. The measurements indicated around 55% reduction in drag for the blunt cone with flat-disk spike at zero angle of attack for a free-stream Mach number of 5.75. The unsteady nature of flow in the separated region and the increase in drag at higher angles of attack were explained as a consequence of the flow re-attachment point. Finley [7], Jiang [8] and Huang [9], among others, studied the effect of opposing jets and spikes on heat flux and drag reduction. Significant reduction in heat flux was seen in the numerical simulations on an aero disked nose cone by Gerdroodbary et.al [10] where forward injection of jets from the stagnation point of the sphere was modeled at hypersonic flows. Sreekanth et.al [11] have used secondary spike on a blunt head to reduce the drag coefficient.

These results helped in selecting the geometrical parameters for the analysis of drag reduction on re-entry vehicles. An *l/d* ratio of 1.5 is selected for hemispherical bodies with spikes having both conical and hemispherical tips. The effect of a counter-flowing jet from the tip of the spike on the aerodynamic drag is also investigated for some cases. A twin spike configuration with conical spikes is also modeled and analyzed in terms of drag reduction. The details of the numerical methodology and the results obtained for these analyzes is explained in the coming sections.

2. NUMERICAL METHODOLOGY

ANSYS Fluent is used to perform the analysis on axisymmetric models shown in Fig. 1. A pseudo transient method is used and the solution was advanced to steady state; the conservation of mass, momentum and energy equations being solved in each time-step. Convergence is assumed when the scaled residual of a conserved variable reaches a limiting value of 10^{-5}. As one can observe from Fig.1, five different models were analyzed, among which four are single-spike models and one is a twin-spike model. The models have a base diameter of 40 mm, with a spike length (for single-spike) of 60 mm and spike diameter of 5 mm. For the hemispherical disc at the end of the spikes, the diameter is 3.5 mm, which is same as the height of the conical disc. For counter flow of air jet, an opening of 2 mm diameter provided at the nose of the spike. For the twin-spike model, a geometry similar to that used in R Yadav et. al [3] is used, except for the fact that the spikes were conical.

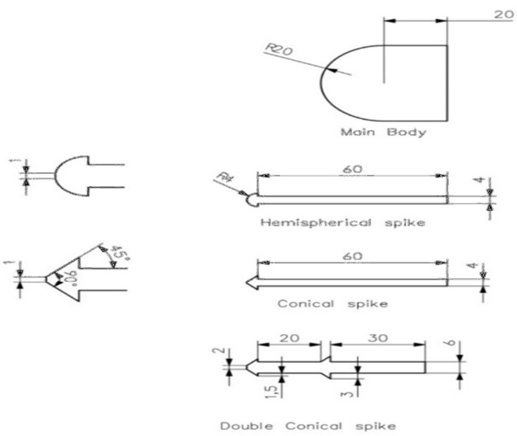

Figure1 Models Analyzed

A free-stream Reynolds number of 1.72×10^6 per unit length is assumed and the flow is considered to be turbulent. The Reynolds Averaged Navier-Stokes (RANS) equation is solved using the Spalart-Allmaraz one equation model for turbulence. The ideal gas assumption is used for air and the dynamic viscosity and thermal conductivity is considered to vary in accordance with the Sutherland's equations. The fluid domain is considered to be pressure far-field with a free-stream pressure of 16066 Pa, temperature of 216.65 K and a Mach number of 10.1. Walls of the re-entry models are considered to be at constant temperature of 300 K and catalytic wall with no slip is used in the analysis.

3. RESULTS AND DISCUSSION

The same authors [11] have previously performed a heat transfer analysis on the same cases at the same set of boundary conditions. This was done after completing the validation of the solver and ensuring grid independence. Thus, grid generation is done as per our previous work. To study the effect of various spike configurations, a simulation was run for a re-entry model with the same base diameter, without any spike or counter-flowing jet. The Drag coefficient C_D for this case was found to be 0.887. For each case, the pressure distribution and the corresponding pressure contour around the body is shown.

Case I: Single-Spike

As mentioned earlier, single-spike configurations with both hemispherical and conical tips are analyzed. Fig. 2 shows the pressure contour and axial pressure distribution for a single-spike with conical tip, in comparison with the model without spike.

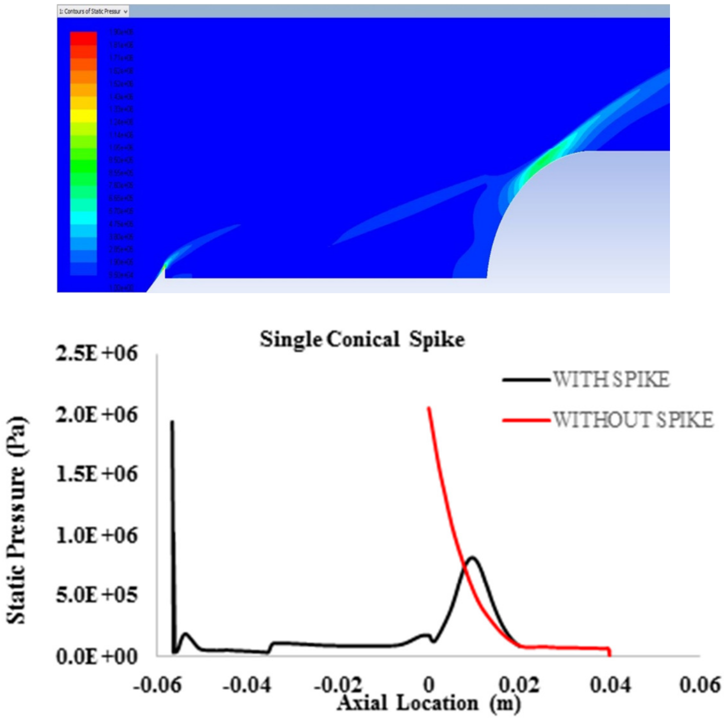

Figure 2 Pressure Distribution over a Single Conical Spike

The maximum pressure on the blunt surface is about 7.6×10^5 Pa. Thus, the spike changes the location of the shock wave in such a way that the peak pressure is experienced at a small area at the tip of the spike, while the larger area of the blunt surface experiences lower pressure. Fig. 2 also shows the reattachment of the shock on the blunt surface resulting in a peak in the pressure at that location. The drag coefficient for this test case is 0.356, which is a drastic reduction in comparison with the case without the spike. Fig. 3 shows similar contour and plot for the single-spike model with hemispherical spike tip. A slight reduction in the peak pressure and the C_D value is observed because of the shock being detached compared to the conical tip. The reattachment point is also slightly shifted because of the nature of the shock wave. The C_D value for this case is 0.345.

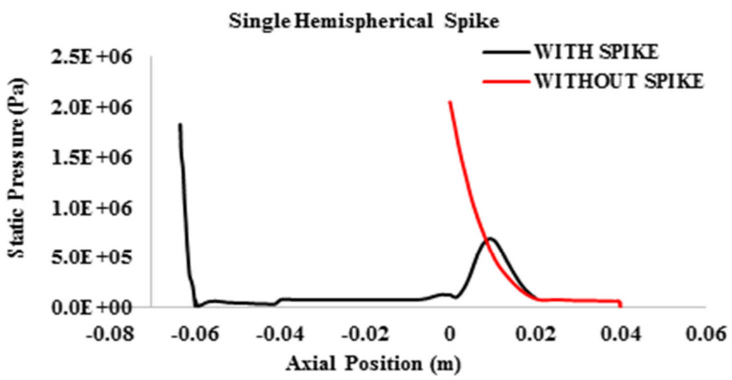

Figure 3 Pressure Distribution of Single Hemispherical Spike

Case II Single spike with Jet

Reduction in heat flux due to a Counter-flowing jet at the nose of the spike has been reported by many authors. In this section, the effect of the jet on aerodynamic drag is discussed. Single-spike models with both conical and hemispherical tips, with a sonic counter-flowing jet of air from the 2 mm opening at the nose are studied. For both these cases, results similar to that obtained in the previous section are seen. The flow field is altered in such a way that the pressure experienced at the blunt surface is reduced, thereby resulting in drag reduction. The pressure contours and axial pressure distribution for conical and hemispherical tipped spikes with jet injection can be seen in figures 4 and 5 respectively. The shock is pushed further away from the leading edge of the spike. In comparison with the corresponding cases without jet injection, it can be seen that the peak pressure and Drag coefficient reduction is slightly less for single-spike re-entry models with jet injection. The C_D values for these cases are 0.464 and 0.493 respectively for the conical and hemispherical spikes. Again, for these cases, the geometry at the tip of the spike is not making much difference to the results.

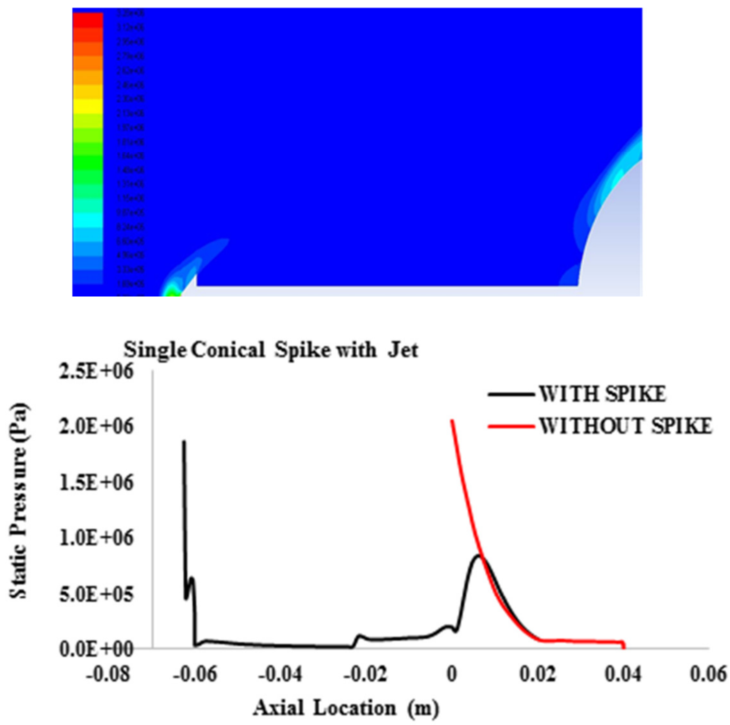

Figure 4 Pressure Distribution of Single Conical Spike with Jet

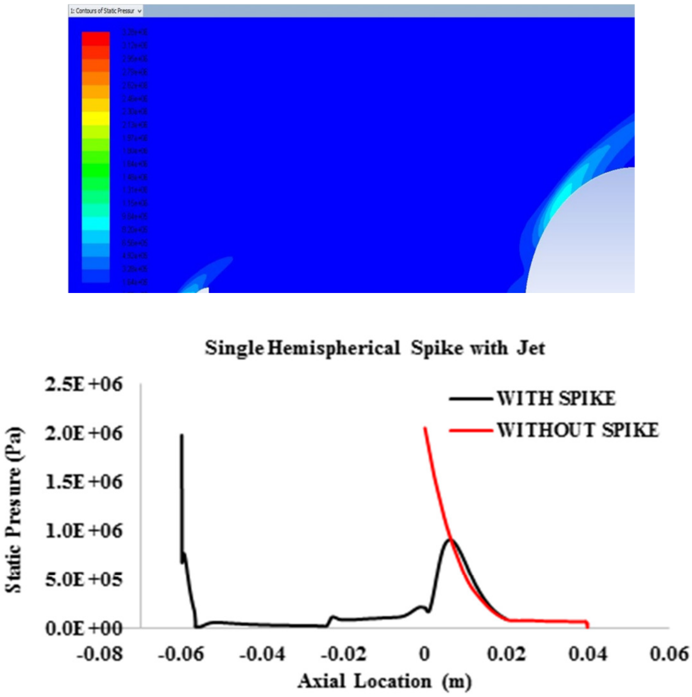

Figure 5 Pressure Distribution of Single Hemispherical Spike with Jet

Case III: Twin Conical Spike

In our previous study [12], the re-entry configuration with two conical spikes in series gave the best result in terms of reduction of heat flux. Here, we see the effect of twin spikes on pressure distribution and drag. The second spike pushes the reattachment point further downstream, which can be seen in Fig. 6. This results in a significant drag reduction compared to blunt re-entry model, but in comparison with some of the cases discussed earlier, the reduction is not very significant. A C_D value of 0.473 is obtained which is comparable with that of single-spike models with jets, and slightly higher than those without jets.

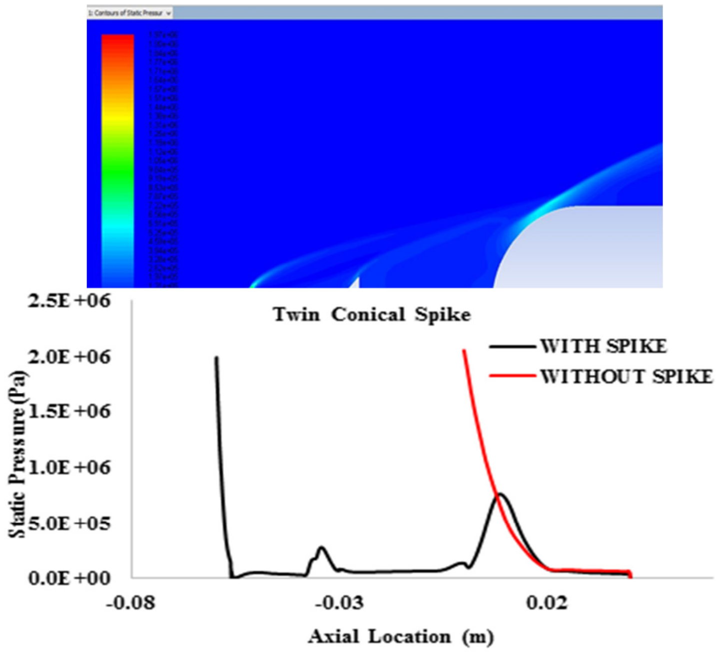

Figure 6 Pressure Distribution of Twin -Spike

4. CONCLUSION

Numerical investigation of different re-entry configurations is performed in terms of reduction in Aerodynamic Drag. These results, along with the heat flux reduction results of our previous work [12] are summarized in Table.1. The percent reduction in the values of peak heat flux and Drag coefficient are tabulated for comparison.

Table 1 Comparison of Heat Flux and Drag reduction for the models considered

Model	% reduction in peak heat flux	% reduction in CD
Single conical spike without jet	31	59
Single hemispherical spike without jet	27	61
Single conical spike with jet	33	47
Single hemispherical spike with jet	29	44
Twin-conical-spike	44	46

Out of the models studied, the single-spike model with hemispherical tip with an *l/d* ratio of 1.5 gives the best drag reduction of 61%, closely followed by the one with a conical tip, reducing the C_D by 59%. Similar models with jet injection, as well as the twin-spike model results in drag reduction compared to a blunt re-entry model, but they are not as significant as the single-spike models without jet. But choosing a re-entry vehicle configuration should be done considering the drag and heating effects simultaneously. In that regard, it can be seen from Table.1, that the twin-spike configuration giving 44% reduction in peak heat flux and 46% reduction in drag coefficient is optimum among the models studied.

However, these results are specific to the free-stream conditions mentioned. The effect of free-stream conditions on these geometries should be studied before generalizing the results. Chemical reactions and dissociation of air at the extreme conditions encountered in hypersonic applications could render the ideal gas assumption inaccurate. The study can be extended by considering these factors to get a better estimate of these design parameters.

REFERENCES

[1] Maull D J, 1960, Hypersonic flow over axially symmetric spiked bodies, Journal of Fluid Mechanics, vol. 8, p.584-592

[2] Mansour K, Khorsandi, 2014. The drag reduction in spherical spiked blunt body. Acta Astronautica, Volume 99, p. 92-98.

[3] Yadav, Rajesh; Velidi, Gurunadh; Guven, Ugur, 2014. Aerothermodynamics of generic re-entry vehicle with a series of aerospikes at nose. Acta Astronautica, Volume 96, p. 1-10.

[4] R. Kalimuthu, R.C. Mehta, E. Rathakrishnan, 2008. Experimental investigation on spiked body in hypersonic flow. The Aeronautical Journal (2008), pp. 593-598

[5] Jagadeesh Gopalan, Viren Menezes, K.P. Jagannatha Reddy, Tokitada Hashimoto, Mingyu Sun, Tsutomu Saito, Kazuyoshi Takayama, 2005. Flow Fields of a Large-Angle, Spiked Blunt Cone at Hypersonic Mach Numbers. Transactions of the Japan Society for Aeronautical and Space Sciences Vol. 48 (2005) No. 160 P 110-116

[6] Menezes V., Saravanan, S., Jagdeesh, G., Reddy, K. P.J, 2003. Experimental investigation of hypersonic flow over highly blunted cones with aerospikes, AIAA Journal, Vol. 41, No.10, 1955-1966.

[7] P J Finley, 1966. The flow of a jet from a body opposing a supersonic free stream. Journal of Fluid Mechanics, Volume 26, Issue 2, pp. 337-368

[8] Zonglin Jiang, Yunfeng Liu, Guilai Han, Wei Zhao, 2009. Experimental demonstration of a new concept of drag reduction and thermal protection for hypersonic vehicles. Acta Mechanica Sinica June 2009, Volume 25, Issue 3, pp 417–419

[9] Wei Huang, Jun Liu, Zhi-xun Xia. (2015) Drag reduction mechanism induced by a combinational opposing jet and spike concept in supersonic flows. Acta Astronautica AA5427, 15-27

[10] M. Barzegar Gerdroodbary, M. Imani, D.D. Ganji, 2014. Heat reduction using counter flowing jet for a nose cone with aero disk in hypersonic flow. Aerospace Science and Technology 39 (2014) 652–665

[11] Sreekanth N, Akhil J and Nagaraja S R 2016 Design and Analysis of Secondary spike on blunt head Indian Journal of Science and Technology, Vol 9(43)

[12] G Gopalakrishnan, Akhil J, Nagaraja S R, 2015. Heat transfer analysis of hypersonic re-entry vehicles with spikes. Proceedings of the 23rd National Heat and Mass Transfer Conference and 1st International ISHMT-ASTFE Heat and Mass Transfer Conference

[13] Eppakayala Naresh, Pinnamaneni Dileep kumar, B. Nagaraj goud and Anil kumar.N, Drag Reduction over a Circular Cylinder, International Journal of Civil Engineering and Technology, 8(8), 2017, pp. 1334–1345.