A NUMERICAL INVESTIGATION TO DETERMINE AND COMPARE DRAG CHARACTERISTICS OF A BLUNT BODY WITH VARIOUS CONFIGURATIONS OF AEROSPIKES AT MACH 6

Yadu Krishnan S, Hariprasad Thimmegowda,
Assistant Professor, Department of Aerospace Engineering, Alliance University, Bangalore.
Suraj R, Shameel Shaik, Leonard J P,
Students, Bachelor of Technology Department of Aerospace Engineering, Alliance University, Bangalore and
Vishnu Goodwill,
CFD Application Engineer, Fluid codes FZ LLC, Dubai.

Abstract

This paper deals with the numerical investigation of different aerospike models and their aerodynamic characteristics in hypersonic flow regime. The study shows that the spiked blunt body produces less drag than that of blunt body (without spike). The effect of few other configurations of spiked blunt body with different L/D ratio and various angles of attack were numerically studied at Mach 6. It has been found that the drag coefficient reduces for certain configuration significantly. The shock standoff distance is also calculated and validated with experimental and theoretical results.

Keywords: Blunt body, Aerospikes, L/D ratio, Drag reduction, hypersonic flow

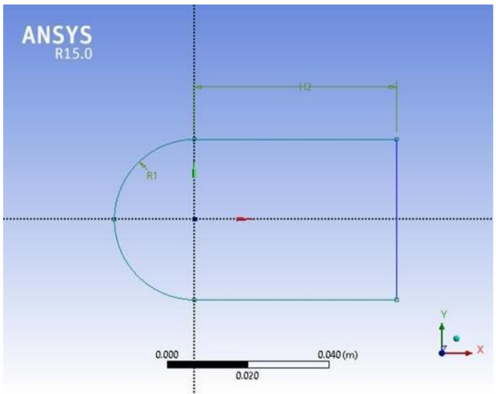
Introduction:

Most of the hypersonic vehicle has a blunt shaped nose to reduce the thermal load on the structure. Such a hypersonic flow creates a strong bow shock wave upstream of a blunt body. This enables the flow to be highly compressible in the downstream region of the bow shock wave. This, in turn, exerts high pressure on the surface of the body. The consequence is increased aerodynamic drag. Several techniques are available in literature to decrease such an adverse effect on the body of the space vehicle. One of the techniques available to reduce the aerodynamic drag is fixing the spike on the frontal area of the blunt body. The capability of the spike is that it generates oblique shock wave in the flow, and it produces a region of recirculating separated flow that protects the blunt nosed body from the incoming flow. This helps to reduces the drag and increases the lift coefficient. But the efficiency with which the drag is reduced depends upon many factors such as geometry of the spike, spike length and flow conditions.

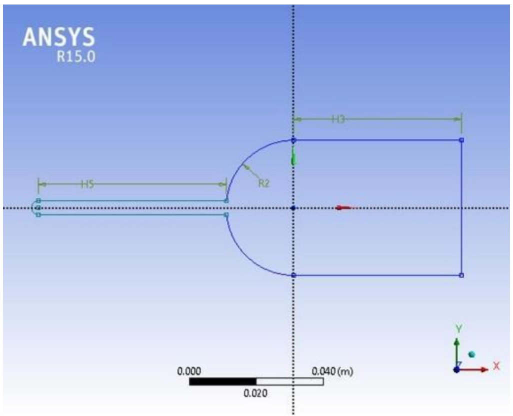
Many authors have investigated the effect of spike on blunt body for drag reduction at different Mach numbers and angles of attack. Studies have been conducted on spike length influence on aerodynamic drag at various angles of attack [1][2][4]. The experiments concluded with the results that the drag coefficient is reduced for the spikes which has higher L/D ratios (where, L is the spike length and D is the cylinder diameter). It is also noted from these results that the drag coefficient increases as the angles of attack increases. However, there is not much influence of Mach number on the length of the spike. Studies also have been conducted on spike geometry influence on effect of aerodynamic drag [3][4][5]. The results conclude that the blunt body with spike yields lesser drag as well as heat reduction when compared to blunt body having no spike. The experimental results show that the spike shape has an important influence on the aerodynamic characteristic, but it diminishes with the increase in angle of attack. The main aim of this paper is to numerically investigate the influence of spike length and geometry on effects of aerodynamic drag at various angles of attack and at Mach 6. The study will be carried out in ANSYS-Fluent for different angles of attack ranging from 0° to 10° with an increment of 2 degrees angle of attack.

Geometric Modelling and Grid Generation:

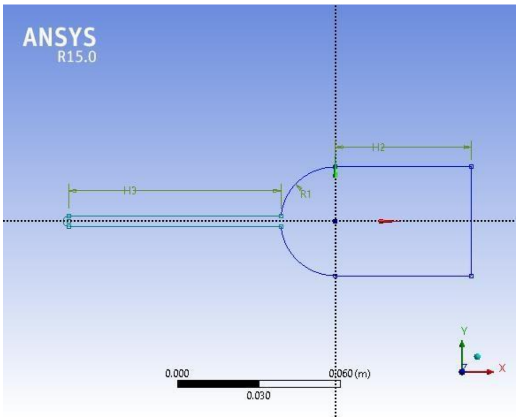
For any simulation, creating the geometry of the model is the first step. In this paper, five different geometries are been considered and created in ANSYS workbench. As a second step, the mesh has to be generated. The mesh generated is unstructured mesh and generated using ICEM CFD. Boundary conditions should be specified based on the flow conditions. To obtain accurate results, grid independence test was done. After completion of pre-processing, the file has to be exported to solver. To solve this problem, Spalart-Allmaras turbulent modelling was used to calculate the aerodynamic characteristics.



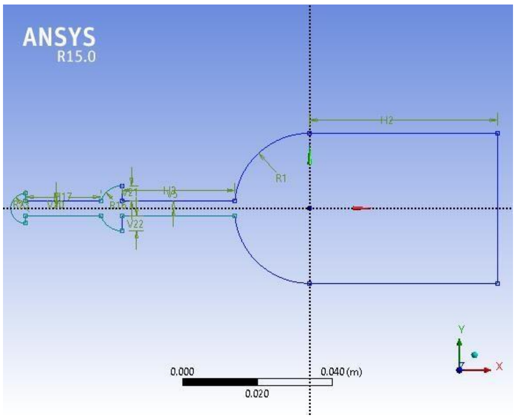
(a)



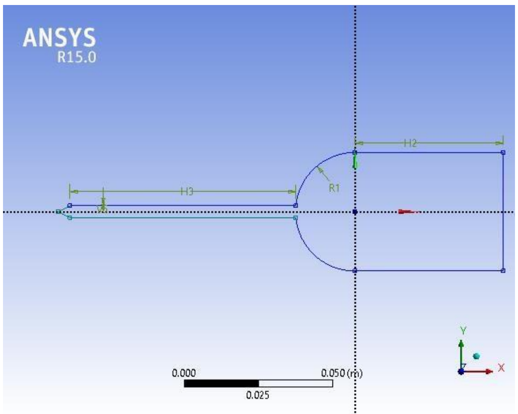
(b)



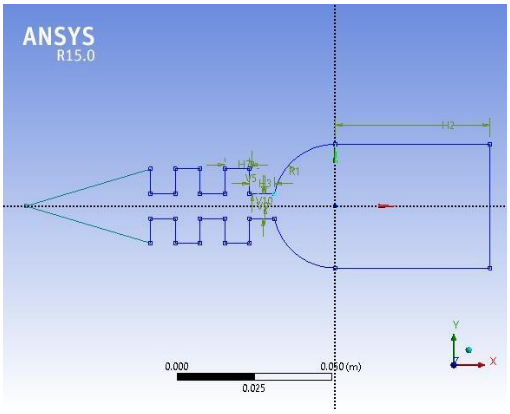
(c)



(d)



(e)



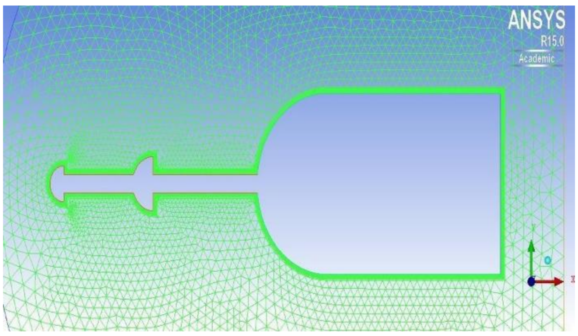
(f)

FIGURE 2: GEOMETRY OF BLUNT BODY WITH DIFFERENT SPIKE CONFIGURATION.

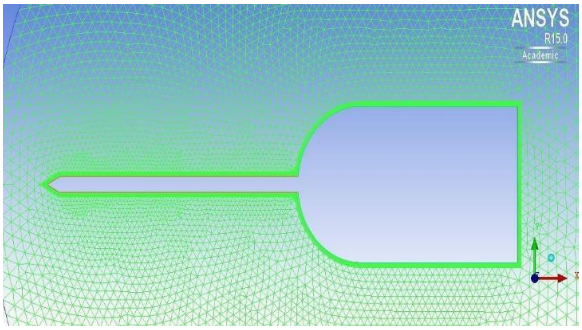
The blunt body, as shown in figure 2(a), is considered as the standard model for the comparative analysis with other models which has a same base diameter of 0.04 m and the overall length of the cylinder is 0.05 m with a hemispherical cap at the end having a radius of 0.02 m. The second and third model chosen for the analysis has a forward-facing spike of length 1.5 and 2 times, respectively, the diameter of the standard model. The spike has hemispherical end and the geometry is as shown figure 2(b) and 2(c). The fourth model has two different spikes arranged in series with different radii of hemispherical ends as shown in figure 2(d) . The overall length of the spike is 0.06 m while the radii for the two hemispherical ends are given as 0.006 m and 0.004 m respectively. The fifth model, shown in figure 2(e), consists of forward spacing sharp pointed conical spike having an overall length of 0.08m while the dimensions for the blunt body remains the same. The sixth model, shown in figure 2(f), consists of multiple rows of identical disks each having length of 0.008m with a conical end having a length of 0.04m

Meshing:

To get a better behaviour of the fluid flow characteristics prism layer is introduced near the walls. The mesh was further refined with help of repairing tools such as Repair mesh and Check mesh. For example, the mesh generation along with prism layers can be visualized for blunt body with series of spikes and pointed spike as shown in figure 3(a) and 3(b).



(a)



(b)

FIGURE 3: COMPUTATIONAL GRID FOR BLUNT BODY **(a)** WITH SERIES OF SPIKES **(b)** POINTED SPIKE

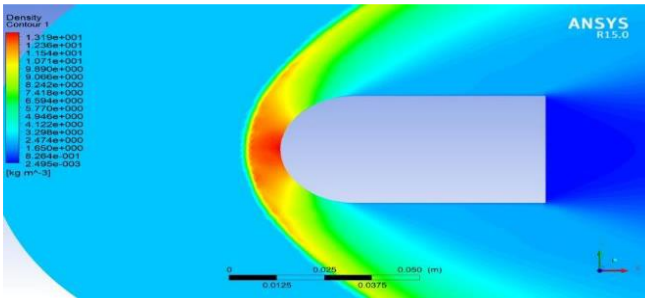
Boundary Conditions:

The boundary conditions are inlet face as pressure far field while the outlet face assumed to be pressure outlet. The spike as well as the blunt body are given no-slip wall boundary conditions. The body of the hemisphere cylinder is to be a fully catalytic wall at a fixed temperature of 300 K with a no slip velocity condition for viscous flows. All other boundaries of the flow domain are assumed to be pressure far field except the outlet boundary that is assumed to be pressure outlet with Mach number and static pressure and temperature specified. The inlet boundary has a free stream pressure of 101325 Pa and temperature of 300 K. The reference area and reference length are 0.001256637 m^2 and 0.04 m respectively.

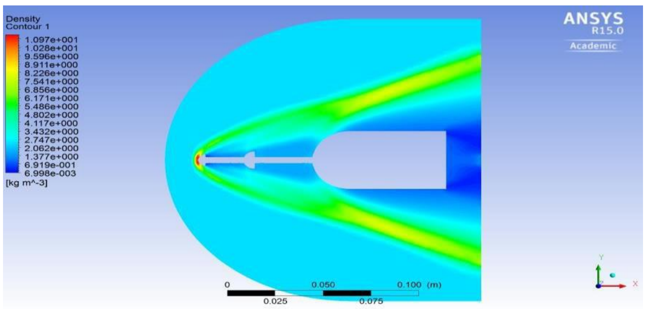
Results and Discussions:

The analysis of the influence of spike length and geometry on effects of aerodynamic drag at various angles of attack and at Mach 6 were carried out. As an illustration two results are being discussed here. (Please note remaining results and validation will be given in full length paper).

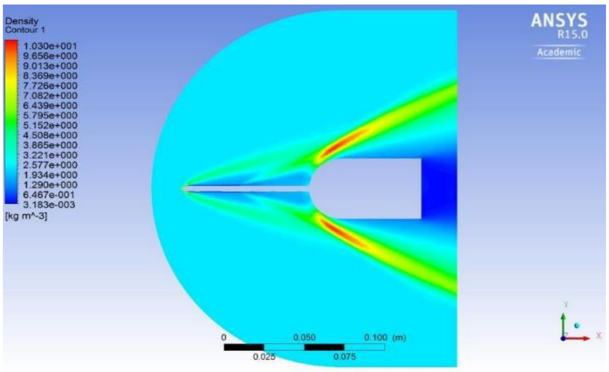
Density Contours:



(a)



(b)



(c)

FIGURE 4a: DENSITY CONTOUR FOR BLUNT BODY WITH SERIES OF SPIKES.
FIGURE 4b: DENSITY CONTOUR FOR BLUNT BODY.
FIGURE 4c: DENSITY CONTOUR FOR BLUNT BODY WITH POINTED SPIKE.

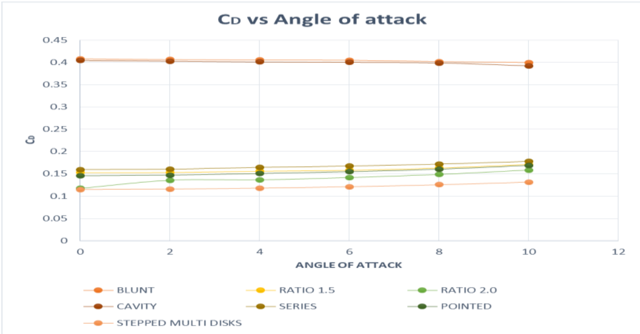


FIGURE 5: C_D VS AOA GRAPH FOR VARIOUS SPIKE MODELS.

Conclusion:

From the analysis results, it is evident that the blunt body mounted with stepped multiple rows of disks conical end is more efficient in drag reduction when compared to the other models. The flow visualization for the different models which involves the shock formation, the boundary layer separation as well as the recirculation regions were also captured and studied with the help of different profile for pressure, density, velocity and temperature. (please note only density contours are given in figure 4). The effect of spike length was also studied, in conclusion the length has a strong influence on the flow characteristics. The longer spike increased the recirculation region in turn reduce the aerodynamic drag effectively.

References:

[1] R. Kalimuthu, R.C. Mehta, E. Rathakrishnan, “Drag reduction for spike attached to blunt- nosed body at Mach 6”, Journal of Spacecrafts & Rockets 47 (1) (2010) 219–222.

[2] M. Yamauchi, K. Fujii, F. Higashino, “Numerical investigation of supersonic flows around a spiked blunt body”, Journal of Spacecrafts & Rockets. 32 (1) (1995) 32–42.

[3] Wan T, Liu C M, “Drag Reduction Optimization for Blunt Body with Aerospikes”, Journal of Aeronautics and Aerospace Engineering 10.4172/2168-9792.1000202

[4] Rajesh Yadav Gurunadh, Velidi Ugur Guve, “Aerothermodynamics of generic re-entry vehicle with a series of aerospikes at nose”, 10.1016/j.actaastro.2013.11.015.

[5] S.S. Milicev, M.D. Pavlovic, “Influence of spike shape at supersonic flow past blunt-nosed bodies: experimental study”, AIAA 40 (5) (2001) 1018–1020.