

AOA and A Shifting Mach number Behavior: Their Effects on a Fifteen-Degree Spike Angle Flat Blunt Nose

Swamy S R ¹, Mohan Kumar G ², Smitha T K ³

Department of Mechanical Engineering, Government Polytechnic Hosadurga, Karnataka, India¹

Department of Mechanical Engineering, Government Polytechnic Mirle, Karnataka, India²

Department of Automobile Engineering, S J (Government) Polytechnic Bangalore, Karnataka, India³

ABSTRACT: Wave drag has to be taken into serious account while creating fast autos. Changing the flow field in front of the body might lessen drag. The most straightforward and successful method to lower the drag of the car seems to be a flat nose spike, which modifies the shock structure surrounding the body and creates flow separation. A body moving at supersonic or hypersonic speed may experience various changes in its flow field. This work is centered on the high-velocity flow around the aircraft's pointed nose. Studying the possibility of reducing drag included three distinct types of spikes. These computer models demonstrate how the proper aero will lower drag and lift forces. Publicized data has been used to verify the numerical results. For best aerodynamics, less drag, or any other design objective, fine-tune the blunt nose and spikes using the simulation data. The provided materials imply that the blunted front surface of an aircraft may provide thermal protection. Since the nose still produces a lot of heat activity even with this blunting, it requires much more thermal protection than the remainder of the vehicle. A blunt nose shape often causes wave drag, which has to be reduced by changing the frontal flow field. Use of a retractable nose spike is one way to make this adjustment.

KEYWORDS: Effect, Flat blunt body, Spike angle, Drag & lift coefficient, Fifteen-degree, Shifting Mach number.

I. INTRODUCTION

Under the broader discipline of fluid mechanics, fluid dynamics is primarily concerned with the study of fluid movement, often referred to as the motion of gases and liquids. Among the many sub-fields that study gas motion are hydrodynamics and aerodynamics. Prediction of weather patterns, pipeline petroleum flow rates calculation, modeling of fission bomb explosions, and aircraft forces and moments evaluation are just a few of the many uses of fluid dynamics. The many applications of fluid dynamics go beyond these. A number of the concepts used in traffic engineering are thought to be relevant when traffic is seen as an ongoing flow. A set of empirical and semi-empirical concepts, derived from flow measurements, are applied to practical situations in fluid dynamics. Many practical fields have their theoretical basis in this set of laws. Computed as functions of both space and time, the temperature, density, pressure, and velocity of the fluid are often used to address fluid dynamics problems. A common method for tackling fluid dynamics issues is this one.

Reducing glide is one such Wearing an aero-spike may help someone traveling faster than sound since it lessens the force of blunt body pressure. When the aerospike is positioned in front of the body, an isolated shock results. By means of establishing a recirculating flow zone between the shock and the forebody, the profile is improved aerodynamically, which lowers road resistance. Apparently, this idea, which was first applied to the Trident missile, increased the weapon's range by 550 kilometers. The extended boom of the Trident aero-spike is joined to a flat circular plate. The missile launches exactly as it emerges from the submarine and comes into touch with the water's surface. The aero-

International Journal of Innovative Research in Science, Engineering and Technology

(A High Impact Factor, Monthly, Peer Reviewed Journal) | Impact Factor: 7.215|

Visit: www.ijirset.com

Vol. 7, Issue 2, February 2018

spike's blunter nose shape decreased drag, allowing more space within for fuel and cargo. The Poseidon C-3 had been superseded by the new Trident IC-4 missile, which could go further because of its extra propulsion stage. This situation has great importance. The third stage engine had to be in the middle of the channel, and the reentry vehicles had to be grouped around it, in order to guarantee that the post-boost vehicle could fit within the pre-existing underwater launch tunnels. One has to consider drag while a body is moving. Drag increases with vehicle speed. It is noteworthy that many elements affect the resistance of the car to motion. One may arrange them according to where the drags came from. Within this category include induced drag, form drag, profile drag, friction drag, pressure drag, wave drag, and other forms of drag. Since wave drag is pertinent to the ongoing case study, just that has been considered. The force of dragging that results when a shockwave peaks is referred to as wave drag.

One crucial component of aerodynamics that needs to be considered is the effect that wave drag has on a body in hypersonic flow. Ascending the flight makes the heating issue more apparent; a big nose radius and a blunt body may be able to lessen the issue. Wave drag will so rise for the car. Reduced wave drag, a crucial element of hypersonic flow, would improve propeller system power and lower fuel consumption and system requirements. This would allow the automobile to keep its load capacity and structural integrity. A automobile may enhance its range or load capacity by around 10% for every one percent reduction in drag. Conversely, gasoline makes up over half of an airplane's weight.

II. LITERATURE REVIEW

The present state of the art in computer analysis of complicated fluid dynamics over aero spikes and blunt bodies is summarized in this part, with an emphasis on lowering drag in subsonic, supersonic, and hypersonic flows. A kind of drag known as forebody pressure drag is experienced by blunt objects flying at supersonic speeds. Use of an aero spike designed to minimize drag may help to lessen this drag. In front of the body, the aero spike produces a very obvious shockwave. Less drag and better body shape are the outcomes of a region of linked flow that forms between the shock and the front of the body.

Computational fluid dynamics (CFD) modeling depends critically on techniques for creating both structured and unstructured models. Combining different techniques may produce hybrid grids, often referred to as chimera, composite, or patched grids. This paper [1] looks at the benefits and drawbacks of each of these strategies. The values at the nodes within each cell in a finite volume formulation control the solution to a flow issue. Temperature, pressure, velocity, and other pertinent statistics are among these values. The number of cells in the grid, which ought to be between 2 and 20, affects the correctness of a CFD solution. A major consideration in understanding an object's motion is drag. The amount of drag a car experience grows with its speed. For a variety of causes, the driving car runs into several challenges. Drag may be divided into many categories according to its origins [3]. A abrupt object in front of a supersonic flow causes a shock wave to generate. Because an oblique shock wave begins at an angle with respect to the blunt mass's surface, it is so called.

As computer speed has improved and complex numerical techniques have been developed, computational fluid dynamics (CFD) has become more and more common [5, 6, 7]. Studying fluid dynamics theory and carrying out experiments is made more affordable using computational fluid dynamics (CFD) than with actual flow simulation. The good outcomes of the simply understood and studied solution have shown its practical value [9]. As such, it is possible to evaluate the efficiency of a CFD tool in handling shock capturing issues by contrasting the numerical findings with the available analytical data. This program uses assessments of grid independence. Initially exploring designs may now be done using a rapid geometry engine (RAGE) rather of time-consuming CAD assistance. Component-based construction of intricate airplane designs is the method used by the geometry tool. The essential elements are shown and explained with the use of fundamental techniques [18]. We use many fidelity aerodynamic study methods on a chosen geometry model to show the flexibility of the geometry tool.

International Journal of Innovative Research in Science, Engineering and Technology

(A High Impact Factor, Monthly, Peer Reviewed Journal) | Impact Factor: 7.215|

Visit: www.ijirset.com

Vol. 7, Issue 2, February 2018

III. COMPUTATIONAL MODEL

One has to be sufficiently familiar with the geometric characteristics of a test case in order to replicate it on a digital computer. The fluid domain in the total system is a well-defined zone with properly selected limits surrounding it to prevent interfering with the physics of the issue. All case studies that come after, particularly those involving external flow issues, use the fluid around the geometry as their computational domain. Usually, the shock capturing is raised in the upstream zone to a magnitude of 8–10 times the body width or base diameter and in the remote zone to 3-5 times. Even if every case study now in publication has problems with external flow, it is important to remember that these studies only examine the fluid domain and not the solid body. Samples of symmetric models are provided in the case studies. In three distinct angles—zero, five, and fifteen degrees—we model the blunt cone body for flow. This applies to both situations—the one with the spike and the one without.

Figure 1 shows a two-dimensional depiction of a blunt body produced using the ICFM CFD industrial standard code. The geometric information for blunt bodies with and without spike configurations is shown in Figure 2. Computer fluid dynamics (CFD) modeling requires the use of both structured and unstructured mesh generation techniques. Hybrid grids, often called chimera grids, composite grids, or patched grids, may be produced by combining these techniques. The benefits and drawbacks of every one of these methods are covered in this paper [1]. The article is available here.

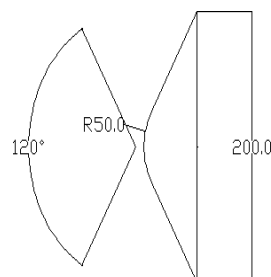


Fig. 1: Blunt body without spike configuration

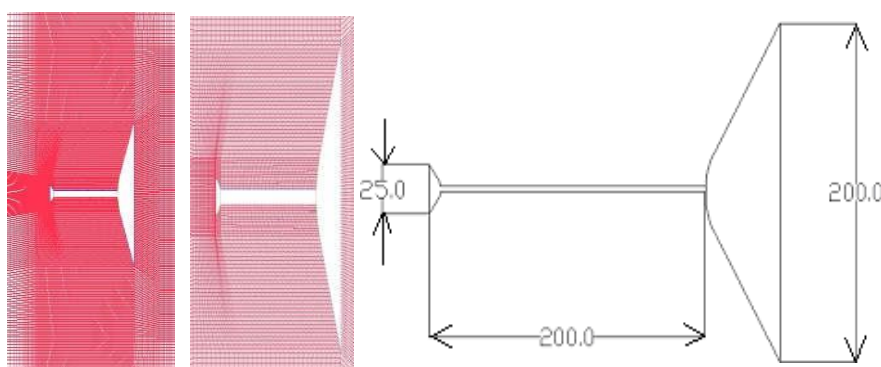


Fig. 2: Meshed model for Blunt nose, Flat and Flat spike configuration

International Journal of Innovative Research in Science, Engineering and Technology

(A High Impact Factor, Monthly, Peer Reviewed Journal) | Impact Factor: 7.215|

Visit: www.ijirset.com

Vol. 7, Issue 2, February 2018

IV. PROBLEM STATEMENT

Exact solutions require that the fluid obey the governing equations, the flow geometry constraint be specified, and the flow field beginning conditions be fulfilled. Precise measurements of the temperature, pressure, and velocity at the border of areas are required to effectively integrate the governing equations. First-order temporal derivatives must be applied to anything ephemeral, and the dependent variable's value at time $t=0$ must be known. The value of the dependent variable is therefore established. It is this that we call the beginning condition. A further constraint that has to be taken into account is the physical boundaries of the fluid region. Still, beginning conditions are not necessary in this case since steady state is assumed. Three most often used measurements to characterize the air at a height of 5 kilometers (16404 feet) immediately above sea level are temperature, pressure, and velocity. A subconscious route has as its intended course its intended purpose. Following projection onto every variable in the computational domain, the inner domain outputs are examined. Given its high viscosity on the solid surface of the blunt body, the fluid is believed to stick to the wall. This explains the notion that it sticks to the wall. The solid and the fluid around it have to have no relative velocities toward one another in order to be non-slip. As a result, the wall boundary condition applies to the surfaces of the blunt cone model and the fluid is considered non-slippery.

The Continuity Equation:

$$\frac{\partial \bar{U}_j}{\partial x_j} = 0$$

The Momentum Equation:

$$\frac{\partial}{\partial t}(\rho \bar{U}_i) + \frac{\partial}{\partial x_j}(\rho \bar{U}_i \bar{U}_j) = -\frac{\partial \bar{P}}{\partial x_i} - \frac{\partial}{\partial x_j}(\bar{\tau}_{ij} + \rho \overline{u_i'' u_j''})$$

The Energy Equation:

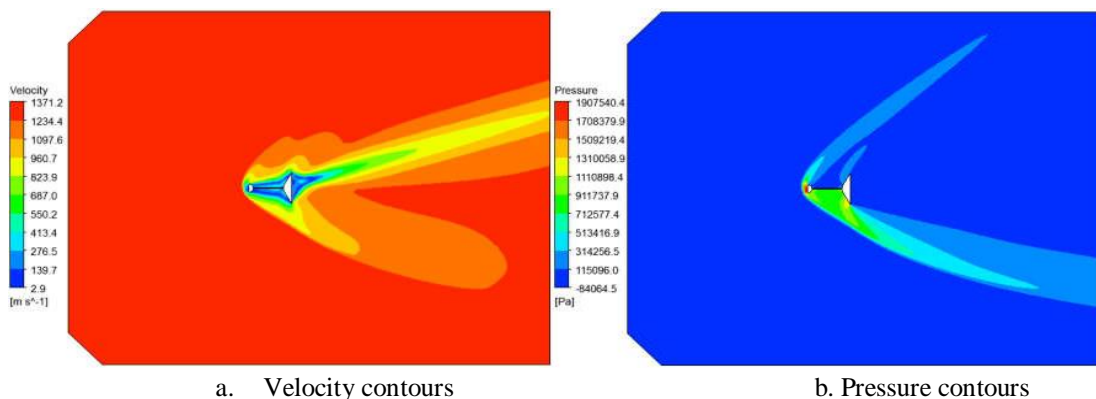
$$\frac{\partial}{\partial t}(\rho \bar{h}) + \frac{\partial}{\partial x_j}(\rho \bar{U}_j \bar{h}) = -\frac{\partial}{\partial x_j}(Q_j + \rho \overline{u_j'' h'})$$

V. CFD RESULTS AND DISCUSSIONS

Different approaches of attaching the flat spike to the blunt body are shown in Figures 3 and 4. These shapes show at 2.0 and 4.0 Mach and zero AOA. Distribution of pressure around the blunt body and flat spike is seen by the pressure contours. Stories go that body-fitted shock peaked before the Flat surge. It follows that the morphology of the shock wave and the spike is same. The flow separation is optimized by the blunt wall placement. If this separation decreases blunt surface force, aerodynamic drag may decrease. Curves showing peak pressure upstream of surges. Peak pressure was therefore probably influenced by compression effects from the spike shape and the entering flow conditions. Aerodynamics and structural strains might be impacted by higher pressure.

Flat spikes 15-degree AOA and 2.0 Mach

Velocity, Pressure, Density, Temperature Contours & Velocity Vectors, Streamlines plots

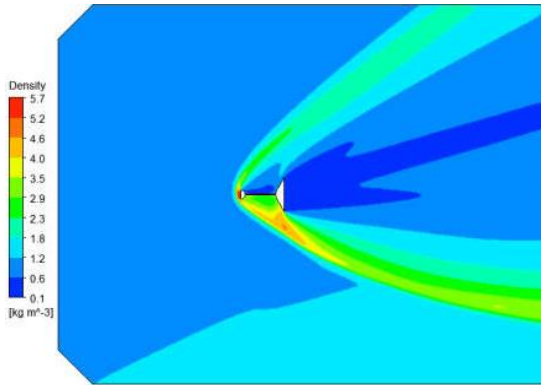


International Journal of Innovative Research in Science, Engineering and Technology

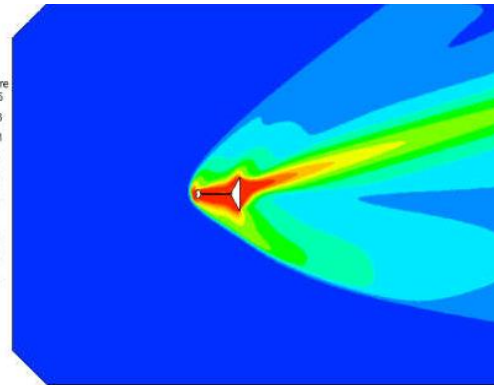
(A High Impact Factor, Monthly, Peer Reviewed Journal) | Impact Factor: 7.215|

Visit: www.ijirset.com

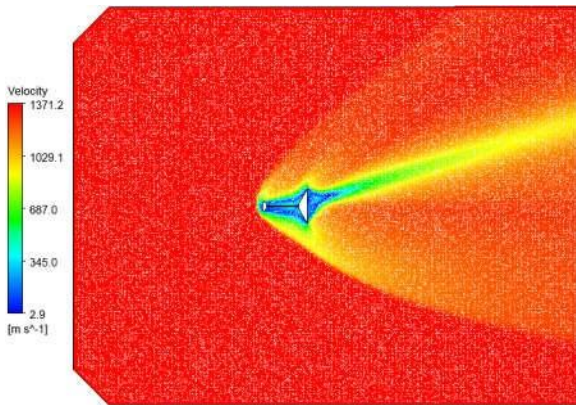
Vol. 7, Issue 2, February 2018



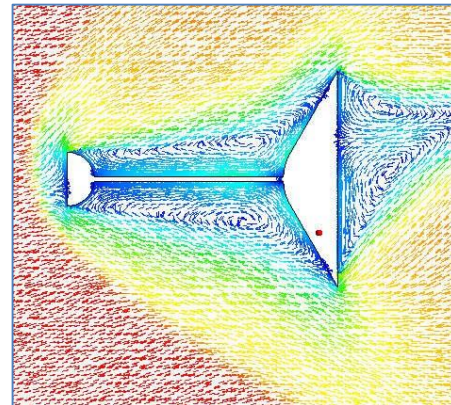
c. Density contours



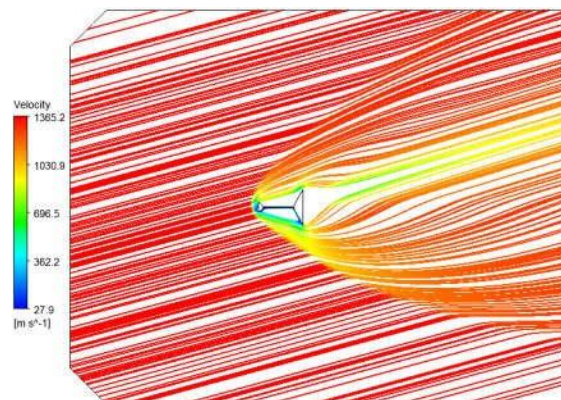
d. Temperature contours



e. Velocity Vectors



f. Zoomed view



g. Velocity Streamlines

Fig. 3: Flat spike blunt nose fluid behavior for 2 Mach speed and 15 deg AOA

International Journal of Innovative Research in Science, Engineering and Technology

(A High Impact Factor, Monthly, Peer Reviewed Journal) | Impact Factor: 7.215|

Visit: www.ijirset.com

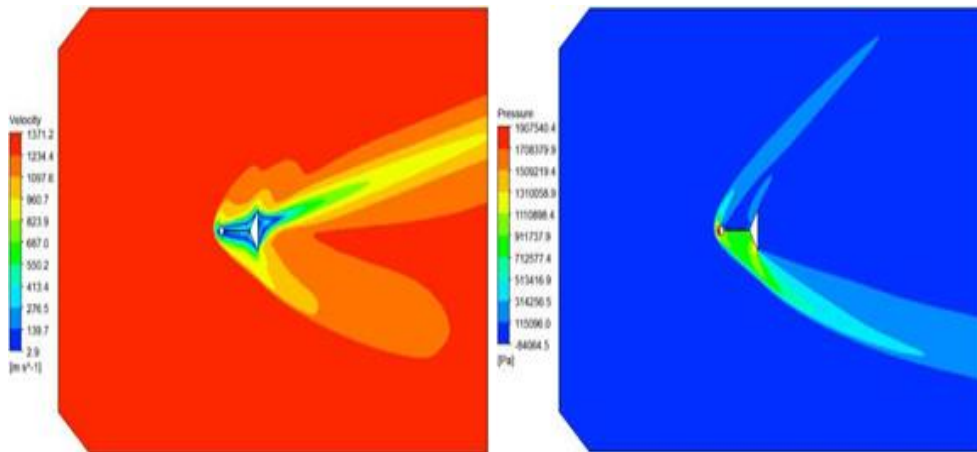
Vol. 7, Issue 2, February 2018

The pressure and velocity contours in the above picture demonstrate how the flow streams are divided by flat spike at upstream of the blunt face for 2.0M speed at 15 degrees AOA. Waves of shock are asymmetric. Lower section of blunt wants greater shock waves than upper part because flow moves in a 15-degree inclined direction at a fast speed. Similar contours of temperature and density are obtained, and unrestricted stream movement is not hampered.

In figure 3, shown is a vector plot for the flat spikes of the blunt nose body. The fluid is traveling at a 15-degree inclined angle, as indicated by the successful separation of flow after hitting the spike. Thus, the lowest portion of the blunt surface shows the highest fluid force. There is skewed air recirculation downstream.

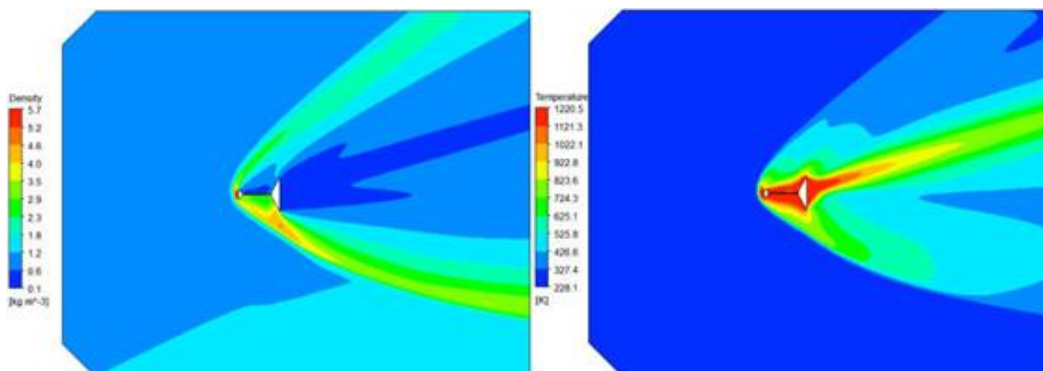
Flat spikes 15 deg AOA and 4.0 Mach

Velocity, Pressure, Density, Temperature Contours & Velocity Vectors, Streamlines plots



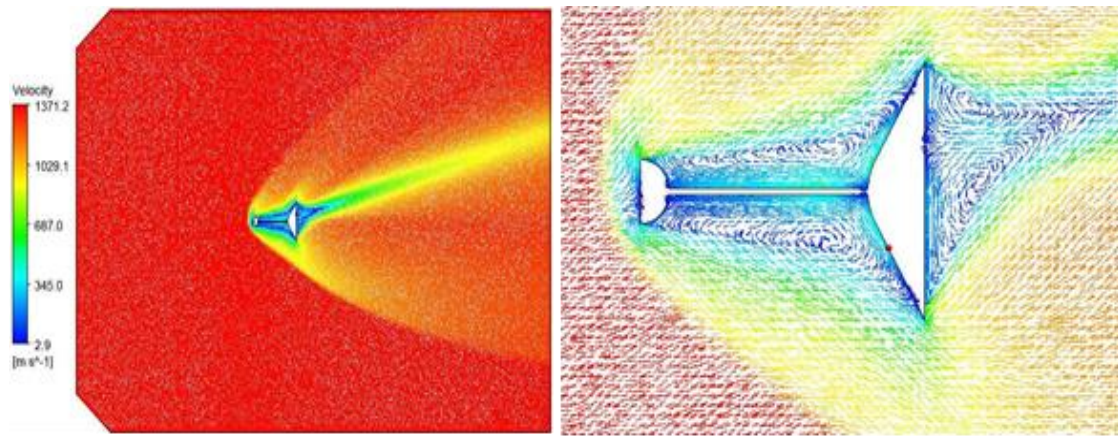
a. Velocity contours

b. Pressure contour



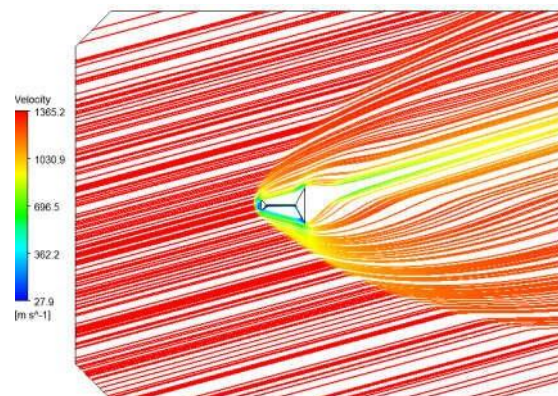
c. Density contours

d. Temperature contours



e. Velocity Vectors

f. Zoomed view



g. Velocity Streamlines

Fig. 4: Flat spike blunt nose fluid behavior for 4 Mach speed and 15 deg AOA

The contours of various variables at 15 degrees above the angle of attack (AOA) for a velocity of 4.0 M are illustrated in the figure above. Upon examining the pressure and velocity contours, one can observe that the flow streams are segregated as a result of flat spike located upstream of the blunt face. The characteristics of shock waves are asymmetry. Given the high velocity in a 15-degree inclined trajectory, the lower portion of the projectile induces greater shock waves compared to the upper portion. In the same way that temperature and density contours are captured, unconstrained stream flow is unaffected.

The figure 4, shows a vector map of the blunt nose body's flat spikes. The fluid is traveling at a speed and direction of 15 degrees of inclination, and it effectively separates upon impact with the spike. Thus, the lowest point of the blunt surface is where the maximum fluid force is felt. The downstream air recirculation is skewed.

International Journal of Innovative Research in Science, Engineering and Technology

(A High Impact Factor, Monthly, Peer Reviewed Journal) | Impact Factor: 7.215|

Visit: www.ijirset.com

Vol. 7, Issue 2, February 2018

VI. CONCLUSION

Attached to a blunt cone, the spike separates the flow re-attachment point from the model. Modifications of the structural shock lower the drag coefficient and wave drag. Wave friction was minimized by the flat disc-shaped spike the most out of all the designs that were tested. Drag coefficient and percentage of drag reduction from different spike designs were used to get this result. The flow field of the blunt body is affected by the shape of the spike. In the Flat and flat disc-shaped spike configuration in particular, the negative impacts of shock waves on aerodynamic performance were reduced by moving the re-attachment point away from the model. According to the study, the spike design might be changed to greatly lower drag and improve the blunt body's aerodynamics. It is very possible that the flat, disc-shaped spike will improve shock structure and lower wave friction. Your study shows that generally, spike design enhances aerodynamic efficiency; wave drag on angular structures is especially lowered. Gaining this understanding could improve the efficiency and architecture of aviation vehicles.

REFERENCES

1. Mark Filipiak, Mesh Generation, Version 1.0, Edinburgh Parallel Computing Centre, University of Edinburgh, November-1996.
2. H. K. Versteeg & W. Malasekera, An introduction to Computational Fluid Dynamics-The finite volume method, Pearson Prantice Hall, 1995.
3. John. D. Anderson, Jr, Fundamentals of Aerodynamics, McGraw Hill International Editions, 1985.
4. H W Liepmann & A Roshko, Elements of Gas Dynamics, John Wiley & Sons, Inc. – Galcit Aeronautical series, 1965.
5. John. D. Anderson, “Computational Fluid Dynamics – the basics with applications”, McGraw Hill Inc, 1985.
6. Joel. H. Ferziger and Milovan Peric, “Computational Methods for Fluid Dynamics”, 3rd revised edition, Springer Verlag publications, 2003.
7. C. A. J. Fletcher, Computational techniques for fluid dynamics – 1, fundamental and general techniques, 2nd edition, 1990. J.F. Thompson, A composite grid generation code for general 3D regions — the Eagle code, AIAA J., Vol. 26 (3) pp.271-272 (1988).
8. S. W. Yuan, “Foundations of fluid mechanics”, PHI Publications, 1988.
9. K. Muralidhar & T. Sundararajan, Computational fluid flow and heat transfer, Narosa publishing house, 1984.
10. Pradip Niyogi, S. K. Chakrabarty, M. K. Laha, Introduction to Computational Fluid Dynamics, Pearson Education Series, 2005.
11. S. M. Deshpande & S. V. Raghuramarao, “Numerical methods for compressible flows based on kinetic theory of gases”, AR & DB Centre of Excellence for Aerospace CFD, IISc – Bangalore, July 2002.
12. Viren Menezes - PhD thesis, Investigation of aero-spike induced flow field modifications around large angle blunt cone flying at hypersonic mach number, Aerospace Engg Dept, IISc – Bangalore, Feb-2003.
13. K.Sateesh, P.S.Kulkarni, G. Jagadeesh, M. Sun, K. Takayama, Experimental and numerical studies on the use of concentrated energy deposition for aerodynamic drag reduction around re- entry bodies, AIAA, CFD Conference USA.
14. J.S.Shang, Plasma injection for hypersonic blunt body drags reduction, AIAA Journal, Vol.40 No- 6, June 2002.
15. K. Satheesh, G. Jagadeesh and P. S. Kulkarni, Hypersonic wave drag reduction in re-entry capsules using concentrated energy deposition, ISSW24, July 12 – 19th, 2004, Beijing, China.
16. Snežana S. Milićević¹, Miloš D. Pavlović¹, Slavica Ristić², Aleksandar Vitić², ON THE INFLUENCE OF SPIKE SHAPE AT SUPERSONIC FLOW PAST BLUNT BODIES, University of Belgrade, Faculty of Mechanical Engineering 27 marta 80, 11000 Belgrade, Yugoslavia
17. David L. Rodriguez* and Peter Sturdza², A Rapid Geometry Engine for Preliminary Aircraft Design, Desktop Aeronautics, Inc., Palo Alto, CA, 94301
18. A.N. Volkov a, Yu.M. Tsirkunov a, B. Oesterle b,* Numerical simulation of a supersonic gas– solid flow over a blunt body: The role of inter-particle collisions and Two-way coupling effects, International Journal of Multiphase Flow 31 (2005) 1244–1275
19. Timothy, Baker. Mesh generation: Art or science? MAE Department, Princeton University, Princeton, NJ 08540, USA
20. S. P. Kuo¹, “Shock Wave Modification by a Plasma Spike: Experiment and Theory”, Department of Electrical & Computer Engineering, Polytechnic University, 6 MetroTech Center, Brooklyn, NY 11201, USA. Received October 14, 2004; accepted November 9, 2004