Quest Journals
Journal of Research in Mechanical Engineering
Volume 4 ~ Issue 1 (2018) pp: 28-34
ISSN(Online):2321-8185
www.questjournals.org

Research Paper

Investigation of Fluid Dynamics for Concave Spike Blunt Nose at Various Mach Speeds with a Five Degree AOA

Srikanth [1], Channaveerayya [2], Mahesh [3]
[1] *Department of Mechanical Engineering, Government polytechnic Bidar, Karnataka, India.*
[2,3] *Department of Mechanical Engineering, Government polytechnic Kalagi, Karnataka, India.*

ABSTRACT
Analyzing fluid dynamics around a blunt nose with different spikes and angles of attack requires knowing how the nose's shape and orientation impact airflow and aerodynamic forces like lift and drag. Begin by determining the shape of the blunt nose and the spikes that will be affixed to it. This involves describing the spikes' diameters, forms, and placements relative to the blunt nose. To model the flow around the blunt nose with various spike configurations, use a CFD software program like ANSYS Fluent, COMSOL Multiphysics, or OpenFOAM. Compare the performance of various spike designs and angles of attack using measures like lift, drag, and aerodynamic efficiency. Determine which combinations result in the necessary aerodynamic properties for the specific application. Based on the simulation findings, tweak the blunt nose and spikes to increase aerodynamic performance, reduce drag, or achieve other design goals. According to the information presented, blunting an aircraft vehicle's front surface provides thermal protection. Despite this blunting, the nose still receives severe thermal activity, demanding considerably more thermal protection than the rest of the vehicle. To overcome the issue of wave drag, which is often caused by the blunt nose form, adjustments to the flow field in front of the vehicle are required. One way to do this alteration is to use a retractable nose spike. By deploying a retractable spike, the flow field may be changed, potentially lowering wave drag and enhancing aerodynamic performance.

Keywords: *Concave Blunt Nose, computational fluid dynamics, Angle of Attack, turbulence, aerodynamics, drag force.*

I. Introduction

Fluid dynamics is the sub-discipline of fluid mechanics dealing with fluid flow: fluids (liquids and gases) in motion. It has several sub-disciplines itself, including aerodynamics (the study of gases in motion) and hydrodynamics (the study of liquids in motion). Fluid dynamics has a wide range of applications, including calculating forces and moments on aircraft, determining the mass flow rate of petroleum through pipelines, predicting weather patterns, and reportedly modeling fission weapon detonation. Some of its principles are even used in traffic engineering, where traffic is treated as a continuous fluid.

Fluid dynamics offers a systematic structure that underlies these practical disciplines and that embraces empirical and semi-empirical laws, derived from flow measurement, used to solve practical problems. The solution of a fluid dynamics problem typically involves calculation of various properties of the fluid, such as velocity, pressure, density, and temperature, as functions of space and time.

A drag-reducing Aero-spike is a device designed to lessen the fore-body pressure drag on blunt bodies at supersonic speeds. The aerospike generates a disconnected shock ahead of the body. Between the shock and the forebody, a zone of recirculating flow forms, acting as a more streamlined forebody profile and lowering drag. This idea was initially used on the Trident missile, and it is believed to have extended range by 550 kilometers. The Trident aero-spike is made out of a flat circular plate attached on an extendable boom that is deployed immediately after the missile bursts through the water's surface following its launch from the submarine. The aero-spike allowed for a more blunter nose shape, which boosted interior space for cargo and propulsion while reducing drag. This was necessary because the Trident IC-4 was equipped with a third propulsion stage to accomplish the requisite improvement in range over the Poseidon C-3 missile it replaced. To fit inside the existing undersea launch tubes, the third stage motor had to be installed in the middle of the post-boost vehicle, with the reentry vehicles grouped around it. Drag is a critical characteristic to consider while a body is in motion. The faster the vehicle speed, the greater the drag put on it. It is the resistance caused by many factors for vehicle motion. Drag can be classified by its source. Some of these are wave drag, friction drag, pressure drag, induced drag, form drag, and profile drag. Only wave drag has been examined here, as it is relevant to the current case study. Wave drag is the drag force that occurs during the creation of a shockwave.

Wave drag on a body in hypersonic flow is a vital and fundamental aerodynamic problem. To reduce the heating problem, which is most noticeable during the ascent portion of the flight, a blunt body with a large nose radius is required. This causes more wave drag on the vehicle. This wave drag, which is critical in hypersonic flow, must be reduced in order to maximize the thrust of the propulsive system while keeping fuel consumption and propulsive system needs low, hence enhancing the vehicle's payloads and structural integrity. Fuel accounts for over half of the aircraft's basic weight, and a 1% reduction in drag increases the vehicle's payload capacity or range by around 10%.

II. Literature Review

Structured and unstructured mesh generation methods are important meshing techniques in CFD simulation and sometimes these are used combinations of structured and unstructured grids called as hybrid grids (referred to as chimera, composite or patched grids). Each of these methods has their own advantages and disadvantages which have been discussed here [1]. The solution to a flow problem (velocity, pressure, temperature, etc.) in finite volume formulation is obtained at nodes inside each cell. The accuracy of a CFD solution depends on the number of cells in the grid [2, 20]. Drag is an important parameter to be considered for a body in motion. Higher the vehicle speed, higher the drag exerted on the vehicle. It is the resistance cause due to different sources for the vehicle motion. Drag can be classified based on the source of it [3]. When the supersonic flow passes over a blunt body, a shock wave will form in front of the body. This kind of shock wave is called an oblique shock wave because it forms at some angle to the surface of blunt body. The Steady advancement of high speed computers and also due to the development of efficient numerical algorithms, the area of Computational Fluid Dynamics (CFD) [5, 6, 7] started gaining importance. CFD complements experimental and theoretical fluid dynamics by providing an alternative cost effective means of simulating real flows. It has been confirmed that the analytically available solution can be used in practical as it gives the acceptable results [9]. Therefore with the grid independence studies, the numerical results can be compared with the available analytical results, in turn one can validate a CFD tool for solving shock capturing problems. A rapid geometry engine (RAGE) has been developed to allow for preliminary design analysis without labor-intensive CAD support. The geometry tool builds complex aircraft configurations using a component-based approach. Basic algorithms for creating the primary components are presented and discussed [18]. A select geometry model is analyzed with several aerodynamic analysis methods ranging in fidelity to further demonstrate the versatility of the geometry tool.

III. Computational Model

It is vital to have sufficient geometric information of every test case in order to simulate it on a digital computer. The fluid domain is a nearly cut section of the entire system whose outside borders are determined in such a way that the problem's physics remain unaffected. In all of the current case studies, the computational domain is the fluid around the geometry (external flow issues). For shock capturing concerns, it is common practice to go up to 8-10 times the body's width or base diameter in the distant field and 3-5 times in the upstream zone.

It should be highlighted that just the fluid domain is being modeled, not the solid body, as all of the current case studies involve exterior flow concerns. The models used in the current case studies are symmetrical. For both situations (without and with spike), the blunt cone body is modeled for flow at zero angle of attack and two non-zero angles of attack (5 degrees and 15 degrees).

The 2-D shape of a blunt body was generated using the ICEM CFD industrial standard code, as illustrated in Figure 1. Figure 2 depicts geometric information for blunt bodies without and with spike arrangements.

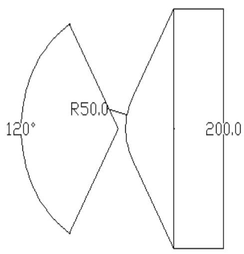


Fig. 1 : Blunt body without spike configuration

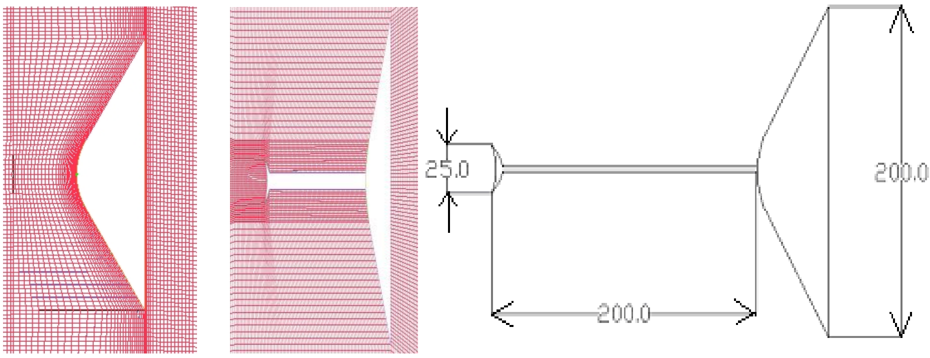


Fig. 2 : Meshed model for Blunt nose, Concave and concave spike configuration

IV. Problem Statement

The governing equations must be fulfilled inside the fluid, and particular solutions can only be derived by specifying the flow geometry constraint as well as the flow field's starting condition. As a result, velocity, pressure, and temperature must be appropriately determined at the region's border in order for the governing equations to be integrated. In transitory situations, the time derivative is of first order, and the dependent variable's value at time t=0 must be specified. This is known as the initial condition. Boundary conditions refer to additional conditions imposed on the fluid region's physical boundaries. However, in the present case, steady state is assumed, hence no beginning conditions are used. Inlet, typical parameters used here are velocity, pressure, and temperature, which represent the ambient air conditions at a height of 5 kilometers (16404 feet) above sea level. It opens to an ambient outlet. All variables from the interior domain are projected to the computational domain's output. Wall, on the solid surface of the blunt body, the fluid is believed to adhere to the wall due to viscosity. This is known as the no-slip condition, and it demands that the solid and neighboring fluid have no velocity relative to one another. As a result, the wall boundary condition is applied to the blunt cone model surfaces, and the fluid is considered to be non-slippery.

The Continuity Equation:
$$\frac{\partial \bar{U}_j}{\partial x_j} = 0$$

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

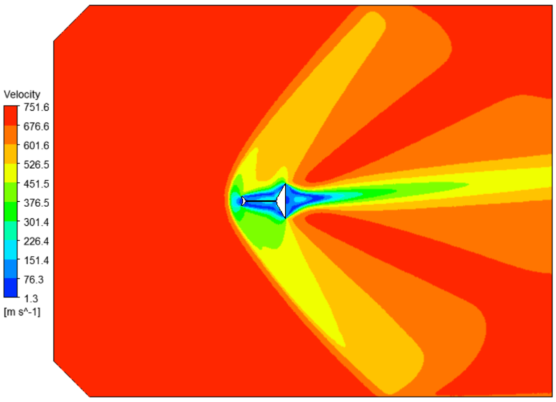
The Energy Equation:
$$\frac{\partial}{\partial t}(\rho \bar{h}) + \frac{\partial}{\partial x_j}(\rho \bar{U}_j \bar{h}) = -\frac{\partial}{\partial x_j}(Q_j + \rho \overline{u_i'' h'})$$

V. CFD Results and Discussions

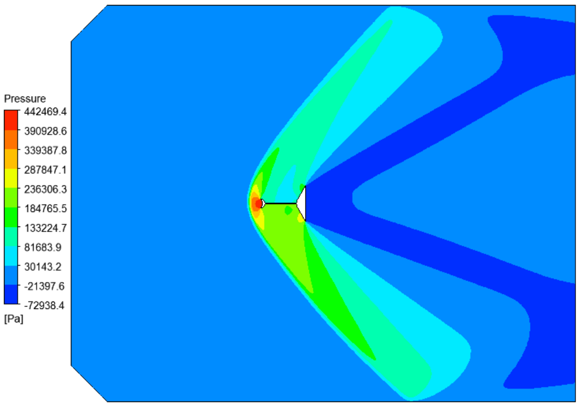
Below figures 3 and 4 shows the different variables contours of 5deg AOA (Angle Of Attach) and 2.0 Mach and 4.0 Mach speed for concave spike attached to the blunt body. The pressure contours illustrate the distribution of pressure around the concave spike and the blunt body. It is mentioned that a body-fitted shock has developed just in front of the concave spike. This indicates that the shock wave is closely following the shape of the spike. Additionally, noted that flow separation occurs successfully at the blunt wall location. This separation could lead to a reduction in the force exerted on the blunt wall, potentially reducing aerodynamic drag. It is observed that the peak pressure contour is captured upstream of the spike. This suggests that the highest pressure occurs in this region, likely due to compression effects caused by the spike's shape and the incoming flow conditions. This increased pressure could have implications for aerodynamic performance and structural loading.

5.1 Concave spikes 5 deg AOA and 2.0 Mach

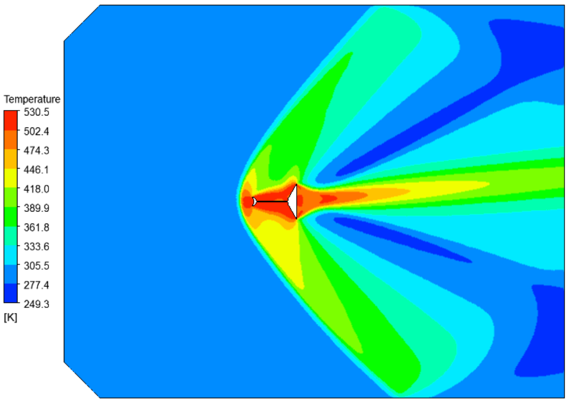
Velocity, Pressure, Density, Temperature Contours & Velocity Vectors, Streamlines plots



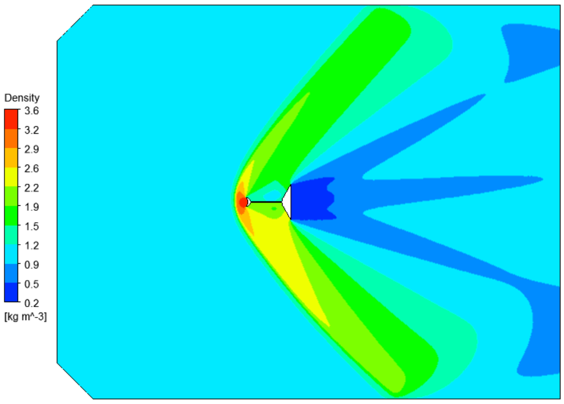
a. Velocity contours



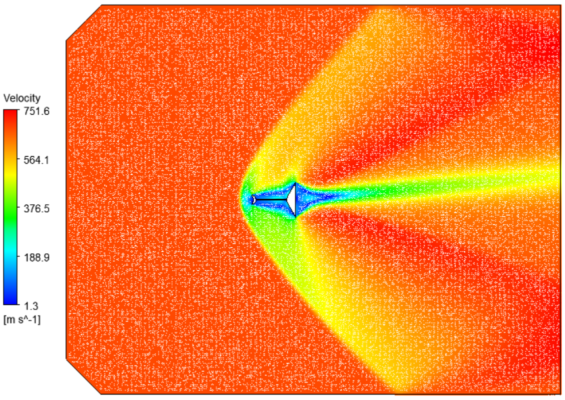
b. Pressure contours



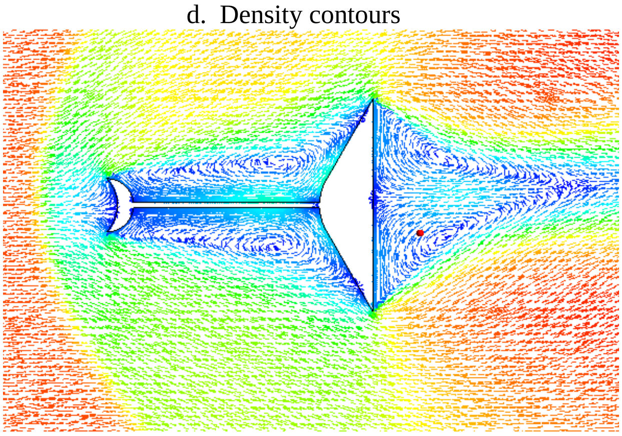
c. Temperature contours



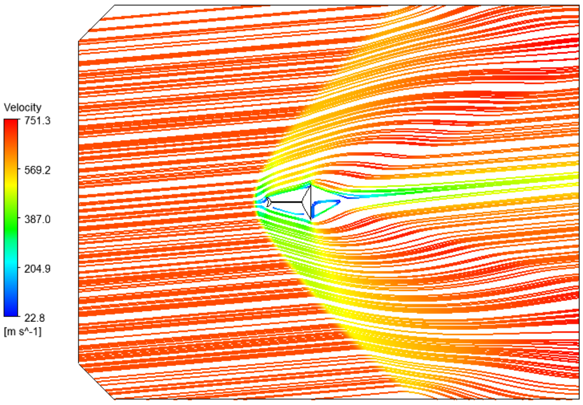
d. Density contours



e. Velocity Vectors



f. Zoomed view



g. Velocity Streamlines

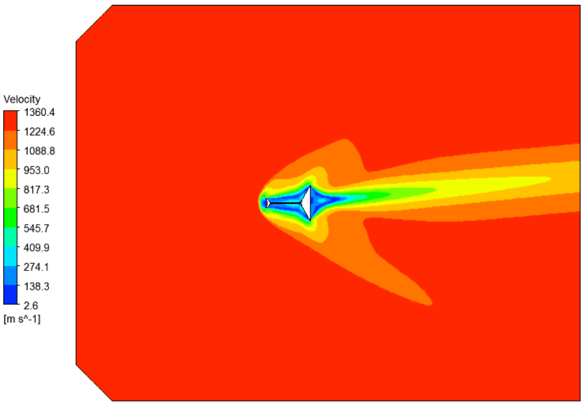
Fig. 3 : Concave spike blunt nose fluid behavior for 2 Mach speed and 5 deg AOA

Above figure shows the different variables contours at 5 deg AOA for 2.0Mach speed, in the pressure and velocity contours it can be seen that the flow streams are separated due to concave spite at upstream of blunt face. Shock waves are asymmetry. Since flow passes 5 deg inclined direction with high speed, lower part of blunt seeks higher shock waves than upper part. Similarly Temperature, Density contours are captured and there is no disturbance for free stream flow.

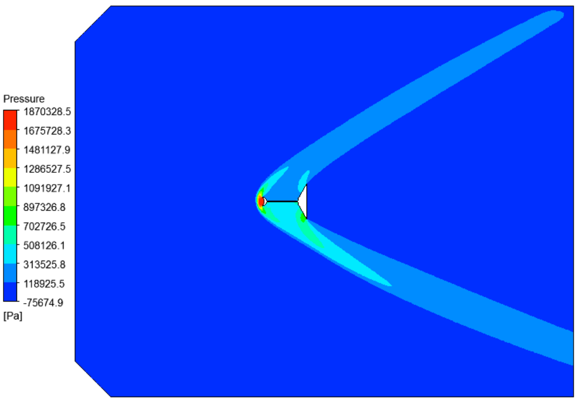
Vector plot for concave spikes of blunt nose body is shown in figure. It can be seen that the speed and direction of moving fluid is at 5 deg inclined movement and fluid striking the spike and flow separates successfully. So that peak fluid force appears on lower part of blunt surface. Air recirculation at downstream is tilted. And similar flow patters can be seen in streamline plot.

5.2 Concave spikes 5 deg AOA and 4.0 Mach

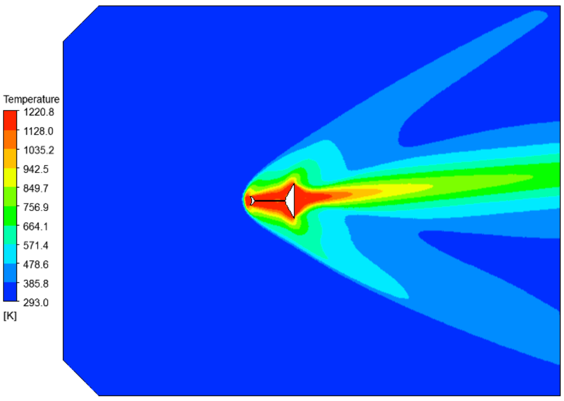
Velocity, Pressure, Density, Temperature Contours & Velocity Vectors, Streamlines plots



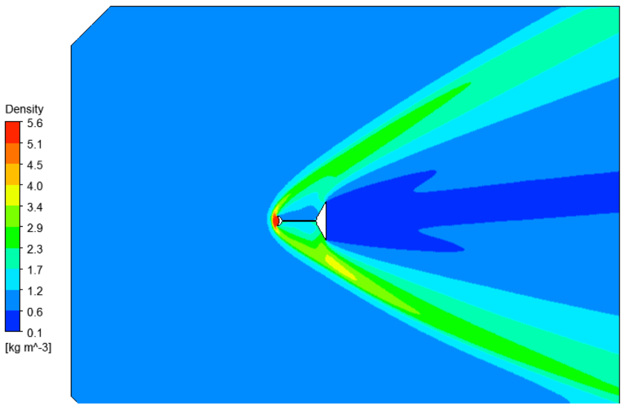
a. Velocity contours



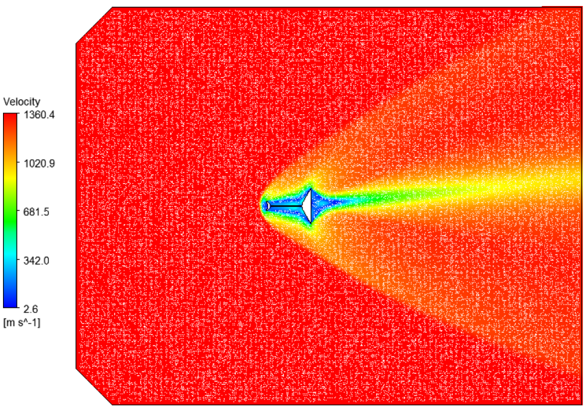
b. Pressure contours



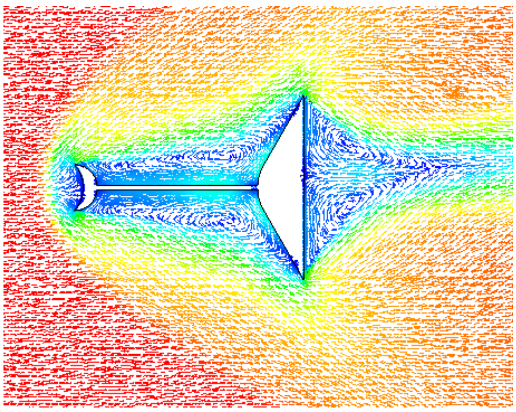
c. Temperature contours

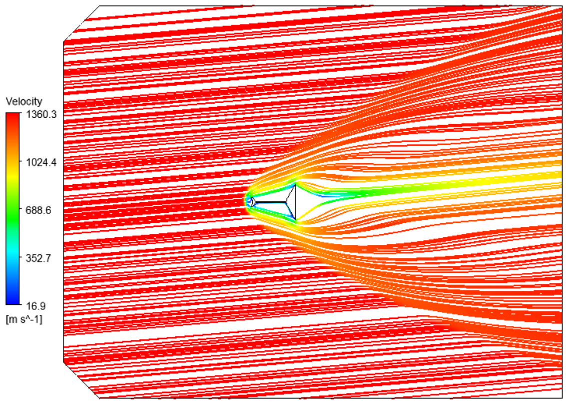


d. Density contours



e. Velocity Vectors


f. Zoomed view



g. Velocity Streamlines

Fig. 4 : Concave spike blunt nose fluid behavior for 4 Mach speed and 5 deg AOA

Above figure shows the different variables contours at 5 deg AOA for 4.0Mach speed, in the pressure and velocity contours it can be seen that the flow streams are separated due to concave spite at upstream of blunt face. Shock waves are asymmetry. Since flow passes 5 deg inclined direction with high speed, lower part of blunt seeks higher shock waves than upper part. Similarly Temperature, Density contours are captured and there is no disturbance for free stream flow.

Vector plot for convex spikes of blunt nose body is shown in figure. It can be seen that the speed and direction of moving fluid is at 5 deg inclined movement and fluid striking the spike and flow separates successfully. So that peak fluid force appears on lower part of blunt surface. Air recirculation at downstream is tilted. And similar flow patters can be seen in streamline plot.

VI. Conclusions

When attached to a blunt cone, this spike successfully shifts the flow re-attachment point away from the model. By doing so, it alters the shock structure, which helps mitigate wave drag and results in a reduction in the drag coefficient. Various spike configurations were compared, and it was found that the spike with a concave and flat disc shape showed the most promise in terms of reducing wave drag. This conclusion was drawn based on the drag coefficient and the percentage of drag reduction observed with different spike configurations. The shape of the spike plays a crucial role in modifying the flow field around the blunt body. In particular, the concave and flat disc-shaped spike configuration was found to be effective in pushing the re-attachment point away from the model, thereby minimizing the adverse effects of shock waves on aerodynamic performance. The findings suggest that selecting the appropriate spike shape can significantly contribute to reducing drag coefficients and enhancing the aerodynamic efficiency of the blunt body. The concave and flat disc-shaped spike emerges as a promising choice due to its effectiveness in altering the shock structure and reducing wave drag. Overall, your analysis highlights the importance of spike configuration in improving aerodynamic performance, particularly in reducing wave drag on blunt bodies. This information could be valuable for optimizing the design of aerospace vehicles and improving their overall efficiency.

References

[1]. Mark Filipiak, Mesh Generation, Version 1.0, Edinburgh Parellel 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.
[8]. J.F.Thompson, A composite grid generation code for general 3D regions — the Eagle code, AIAA J., Vol. 26 (3) pp.271-272 (1988).
[9]. S. W. Yuan, "Foundations of fluid mechanics", PHI Publications, 1988.
[10]. K. Muralidhar & T. Sundararajan, Computational fluid flow and heat transfer, Narosa publishing house, 1984.
[11]. Pradip Niyogi, S. K. Chakrabartty, M. K. Laha, Introduction to Computational Fluid Dynamics, Pearson Education Series, 2005.
[12]. 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.
[13]. 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.
[14]. 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.

[15]. J.S.Shang, Plasma injection for hypersonic blunt body drags reduction, AIAA Journal, Vol.40 No-6, June 2002.
[16]. 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.
[17]. Snežana S. Milićev1, 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
[18]. David L. Rodriguez* and Peter Sturdza2†, A Rapid Geometry Engine for Preliminary Aircraft Design, Desktop Aeronautics, Inc., Palo Alto, CA, 94301
[19]. 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
[20]. Timothy, Baker. Mesh generation: Art or science? MAE Department, Princeton University, Princeton, NJ 08540, USA
[21]. 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