Outcome of Varied Mach Number and AOA on Convex Blunt Nose with Spike Angle Zero Degree

Mahesh [1], Srikanth [2], Channaveerayya [3]

[1, 3] *Lecturer, Department of Mechanical Engineering, Government Polytechnic Kalagi, Karnataka, India.*

[2] *Lecturer, Department of Mechanical Engineering, Government Polytechnic Bidar, Karnataka, India.*

Abstract: **Wave drag is an important parameter to be considered in designing high speed vehicles. It is well known that drag can be alleviated by modifying the flow field in front of the body. There are several techniques to the flow field ahead of a supersonic/hypersonic body and the use of retractable nose spike appears to be simplest and at an efficient means of reducing drag on the vehicle by bringing about flow separation and altering the shock structure around the body. The present work involves study of high-speed flow around the blunt nose of an aero vehicle. Three different types of spikes have been considered for the purpose of drag reduction studies. These computational studies suggest that suitable aero to reduce the drag and lift forces. The numerical results have been validated with literature data. Modify the blunt nose and spikes to increase aerodynamics, lower drag, or accomplish other design goals in light of the simulation results. The information presented states that thermal protection can be achieved by blunting the front surface of an aircraft. The nose still undergoes high levels of heat activity despite this blunting, thus it needs a lot more thermal protection than the remainder of the car. A modification to the flow field in front of the vehicle is necessary to alleviate the problem of wave drag, which is often brought on by the blunt nose shape. Using a retractable nose spike is one method to implement this modification. The flow field may be changed by deploying a retractable spike, which may lessen wave drag and enhance aerodynamic efficiency.**

Keywords: Convex Blunt Nose, Spike angle, CFD, Lift force, Mach number, drag force.

1. INTRODUCTION

Within the larger subject of fluid mechanics, fluid dynamics is the study of fluid flow, or the movement of liquids and gases. It includes a number of subfields, including hydrodynamics and aerodynamics, which examine gases in motion. Numerous tasks include the use of fluid dynamics, such as estimating airplane forces and moments, figuring out how much petroleum flows through pipelines at a mass flow rate, predicting weather patterns, and even simulating the detonation of fission weapons. In traffic engineering, where traffic is seen as a continuous stream, some of its concepts are also used. These practical disciplines are grounded on the systematic framework of fluid dynamics, which also includes empirical and semi-empirical rules obtained from flow measurement and applied to real-world issues. Calculating the fluid's many parameters as functions of place and time, including temperature, density, pressure, and velocity, is a common method of solving a fluid dynamics issue.

A drag-reducing the purpose of an aero-spike is to lessen blunt body pressure drag when traveling at supersonic speeds. The aerospike creates a detached shock in front of the body. Between the shock and the forebody, a zone of recirculating flow develops, which lowers drag and produces a more streamlined profile. This idea, which was initially used with the Trident missile, is believed to have extended range by 550 kilometers. An extensible boom with a flat circular plate attached is the Trident aero-spike. As soon as the missile leaves the submarine and breaches the water's surface, it is deployed. A blunter nose form was made feasible by the aero-spike, increasing interior space for freight and propulsion while reducing drag. This was important because the Trident IC-4 missile—which took the role of the Poseidon C-3 missile—had a third propulsion stage that gave it the extra range it needed. To fit inside the existing underwater launch tubes, the third stage motor had to be placed in the middle of the post-boost vehicle, with the reentry vehicles organized around it. Drag is a significant consideration when a body is in motion. An car feels more drag as it moves faster. It is the resistance to motion of the vehicle caused by several variables. Drag can be categorized according its source.

Among them are form drag, profile drag, induced drag, friction drag, pressure drag, and wave drag. Only wave drag has been examined here since it is relevant to the case study at hand. Wave drag is the drag force that arises during a shockwave's development.

A crucial and essential aerodynamic issue is wave drag on a body in hypersonic flow. A blunt body with a big nose radius is needed to lessen the heating issue, which is most visible during the flight's ascent. The car has increased wave drag as a result. Reducing this wave drag, which is essential in hypersonic flow, would increase the propulsive system's thrust while lowering fuel consumption and propulsive system requirements, protecting the vehicle's structural integrity and cargo capacity. More than half of an aircraft's weight is made up of fuel, and a 1% decrease in drag adds around 10% to the vehicle's range or cargo capacity.

2. LITERATURE REVIEW

Literature review on the use of aero spikes to reduce drag in subsonic, supersonic, and hypersonic flow as well as aerodynamic study utilizing computational methods for complicated flow over the blunt body. A device intended to lessen blunt bodies' forebody pressure drag at supersonic speeds is called a drag-reducing aero spike. A detached shock is produced ahead of the body by the aero spike. A zone of reticulating flow forms between the shock and the forebody, acting as a more streamlined body profile and lowering drag.

Mesh generation techniques, both organized and unstructured, are essential for CFD simulation. Hybrid grids, often referred to as chimera, composite, or patched grids, are occasionally produced by combining these approaches. This article [1] discusses the advantages and disadvantages of each of these strategies. The solution to a flow issue (temperature, pressure, velocity, etc.) in a finite volume formulation is achieved at nodes within each cell. The accuracy of a CFD solution depends on the number of cells in the grid [2, 20]. Drag is an important consideration for a body in motion. With increasing vehicle speed comes an increase in drag on the vehicle. It is the resistance that the vehicle experiences when moving due to a variety of factors. Drag may be divided into many categories based on its origins [3]. When a supersonic flow passes in front of a blunt body, a shock wave is created. This kind of shock wave is referred to as an oblique shock wave as it originates at an angle to the surface of the blunt body. Computational fluid dynamics (CFD) [5, 6, 7] started to become more and more popular as fast computers and efficient numerical methods continued to advance. CFD provides an inexpensive alternative to actual flow simulation for fluid dynamics theory and experimentation. The analytically accessible answer has been verified to be practical to apply, since it produces satisfactory results [9]. As a consequence, using grid independence studies, one may evaluate a CFD tool for handling shock capturing problems by comparing the numerical findings with the existing analytical data. For early design study, a rapid geometry engine (RAGE) has been created that eliminates the need for labor-intensive CAD assistance. The geometry tool uses a component-based methodology to construct intricate aircraft designs. There is a presentation and discussion of basic techniques for generating the main components [18]. To highlight the geometry tool's adaptability, a selected geometry model is studied using a variety of fidelity-ranging aerodynamic analysis techniques.

3. COMPUTATIONAL MODEL

It is necessary to acquire sufficient geometric knowledge about a test case in order to mimic it on a digital computer. The fluid domain is a roughly defined area of the entire system, with carefully determined outside bounds to avoid interfering with the problem's physics. In all of the current case studies, the computational domain is the fluid around the geometry (external flow concerns). In the upstream zone, it is typical to enhance the shock capturing up to 8–10 times the body width or base diameter, and in the distant field, up to 3-5 times.

It should be highlighted that just the fluid domain—and not the solid body—is being represented in the current case studies because they all entail outside flow problems. The case studies that are being presented use symmetric models. For both cases (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 created using the ICEM CFD industrial standard code, as seen in Figure 1. Figure 2 displays geometric data for blunt bodies with and without spike configurations. In CFD simulation, structured and unstructured mesh generating techniques are crucial meshing methods. These approaches are occasionally combined to create hybrid grids, also known as chimera, composite, or patched grids. The benefits and drawbacks of each of these approaches have been covered in this article [1].

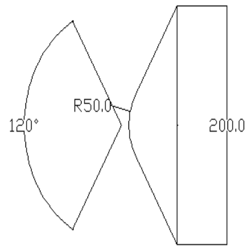


Fig. 1 : Blunt body without spike configuration

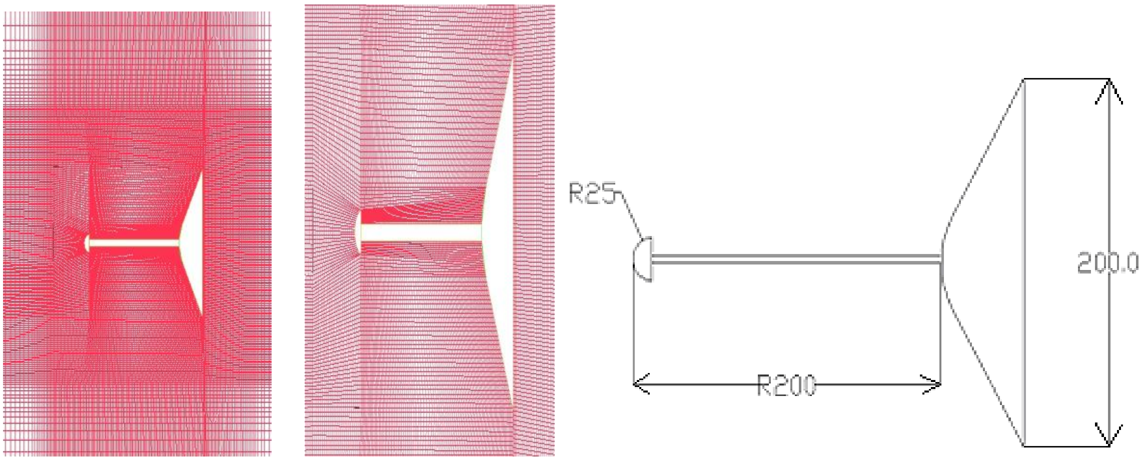


Fig. 2 : Meshed model for Blunt nose, Convex and convex spike configuration

4. PROBLEM STATEMENT

The fluid must satisfy the governing equations, and the flow geometry constraint and the beginning condition of the flow field must be defined in order to derive particular solutions. Consequently, temperature, pressure, and velocity must be accurately measured at the region's border in order to integrate the governing equations. When something is transitory, the time derivative is of first order and the value of the dependant variable at time t=0 must be stated. This is what we call the starting condition. Boundary conditions are additional limitations imposed on the fluid region's physical bounds. However, no beginning conditions are needed in this case because steady state is assumed. The typical characteristics used to characterize the air conditions at an elevation of 5 kilometers (16404 feet) above sea level are temperature, pressure, and velocity. It goes towards a subliminal channel. All variables from the inner domain are projected to the output of the computational domain. Wall: The fluid's viscosity on the solid surface of the blunt body is assumed to be the reason it sticks to the wall. The no-slip condition states that there must be no velocity between the surrounding fluid and the solid. 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:
$$\frac{\partial}{\partial t}(\rho \bar{U}_i) + \frac{\partial}{\partial x_j}(\rho \bar{U}_i \bar{U}_j) = -\frac{\partial \bar{P}}{\partial x_i} - \frac{\partial}{\partial x_j}(\bar{\tau}_{ij} + \rho \overline{u_i'' u_j''})$$

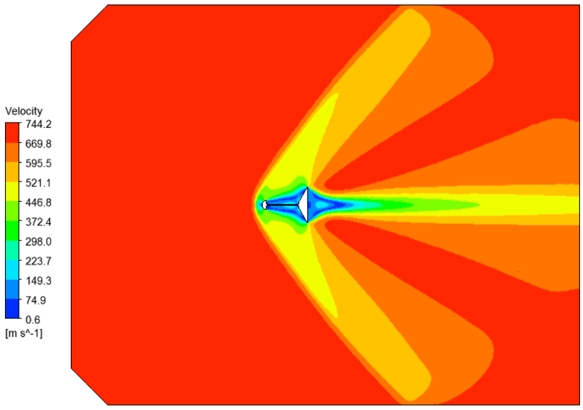
The Energy Equation:
$$\frac{\partial}{\partial t}(\rho \bar{h}) + \frac{\partial}{\partial x_j}(\rho \bar{U}_j \bar{h}) = -\frac{\partial}{\partial x_j}(Q_j + \rho \overline{u_i'' h'})$$

5. CFD RESULTS AND DISCUSSIONS

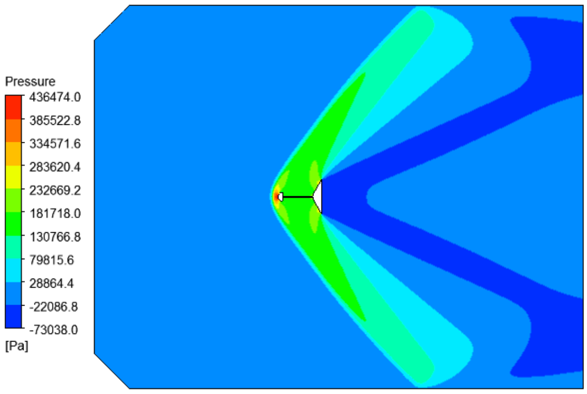
Figures 3 and 4 below show the several variable shapes for the convex spike attached to the blunt body, with a zero degree angle of attachment (AOA) and speeds of 2.0 and 4.0 mach. The pressure contours show the distribution of pressure around the convex spike and blunt body. It is alleged that a body-fitted shock generated just in front of the convex spike. This implies that the shock wave is following the shape of the spike rather precisely. The successful occurrence of flow separation at the blunt wall position has also been observed. Aerodynamic drag may be reduced if this separation results in a reduction in the force applied to the blunt wall. It is observed that the contour of peak pressure is captured upstream of the spike. This implies that the region with the highest pressure is probably the result of compression effects brought on by the form of the spike and the incoming flow conditions. The structural loads and aerodynamic performance may be affected by this increased pressure.

5.1 Convex spikes 0 deg AOA and 2.0 Mach

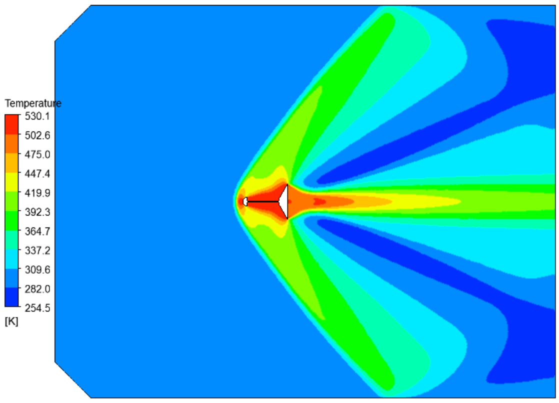
Velocity, Pressure, Density, Temperature Contours & Velocity Vectors, Streamlines plots



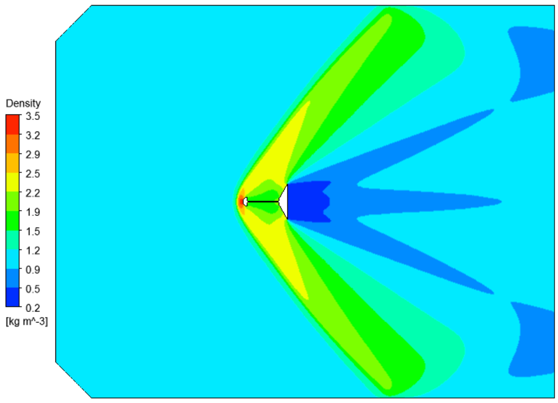
a. Velocity contours



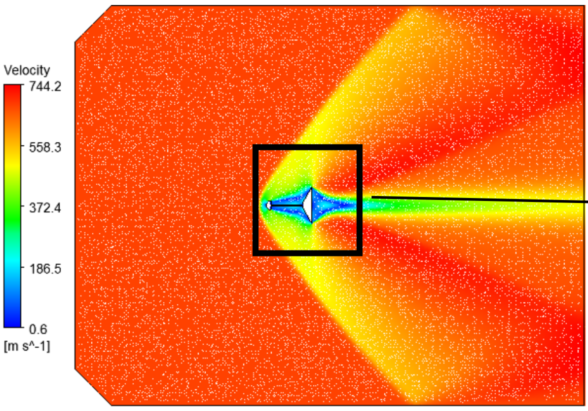
b. Pressure contours



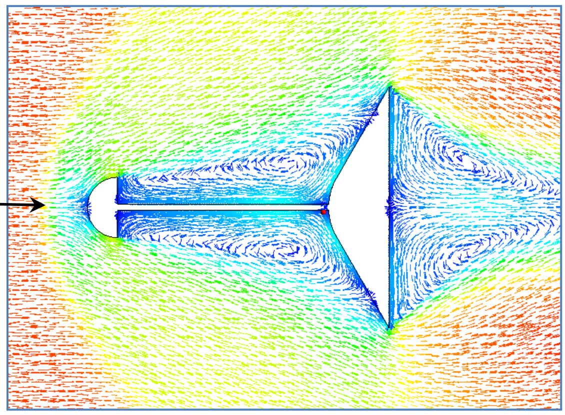
c. Temperature contours

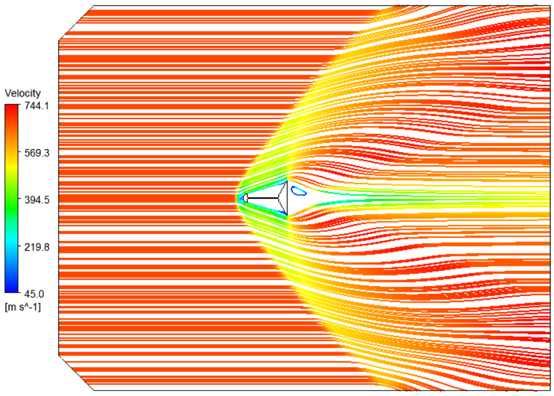


d. Density contours



e. Velocity Vectors


f. Clipped view



g. Velocity Streamline

Fig. 3 : Convex spike blunt nose fluid behavior for 2 Mach speed and 0 deg AOA

Above figure shows the different variables contours of 0 deg AOA and 2.0 M speed for convex spike attached to the blunt body, in the pressure contours it can be seen that the body fitted shock has been developed at just in front of convex spike and flow separated successfully at blunt wall location, this will reduce the force on blunt wall. Pressure variation can be seen that the peak pressure of contour is captured at the spike upstream.

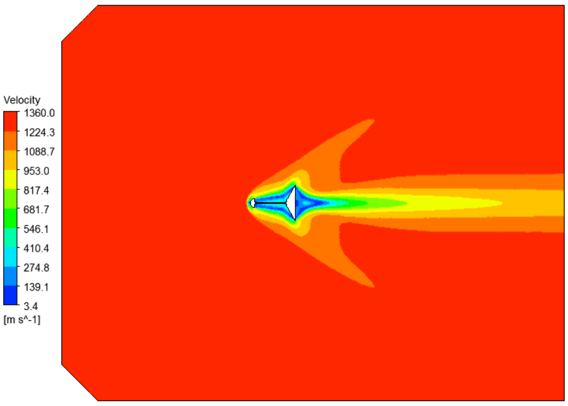
Velocity contour patterns plots have been captured for convex spikes blunt nose body, it can be seen that there is shockwave (low velocity zone) in the upstream of the spike and also it can see that there is a recirculation flow pattern captured in the downstream of the blunt body, and the shocks are symmetric in nature.

Temperature Contour plots for convex spikes blunt nose body as shown in fig. It can be seen that peak temperature at convex spike. Temperature is also high in downstream of the spike and low in blunt body. Density contour plots for blunt nose with convex spike as shown in figure, it can be seen that peak density appears at spikes surface due to thermal properties changes.

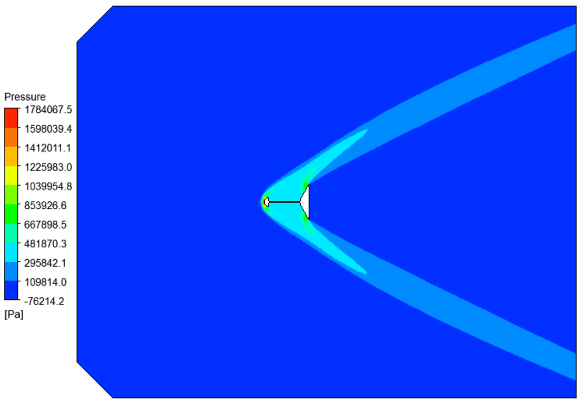
Vector plot for Blunt nose body is shown in figure. It can be seen that the speed and direction of moving fluid do not change until the moving fluid striking the convex spikes. Due to spike in location flow gets separate and flow hitting on blunt face is less. Streamline plots are the lines of constant value of the stream function. It can seen figure that there is a considerable flow recirculation is being captured in the downstream of the blunt surface.

5.2 Convex spikes 0 deg AOA and 4.0 Mach

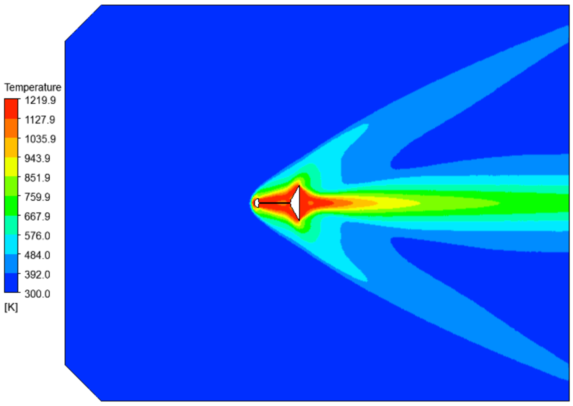
Velocity, Pressure, Density, Temperature Contours & Velocity Vectors, Streamlines plots



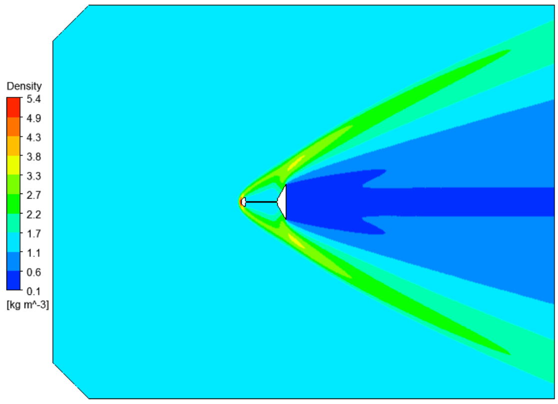
a. Velocity contours



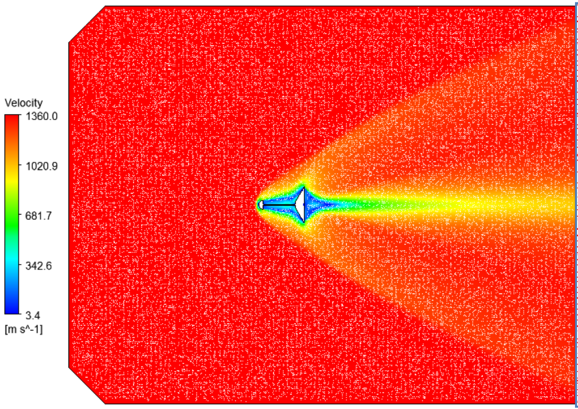
b. Pressure contours



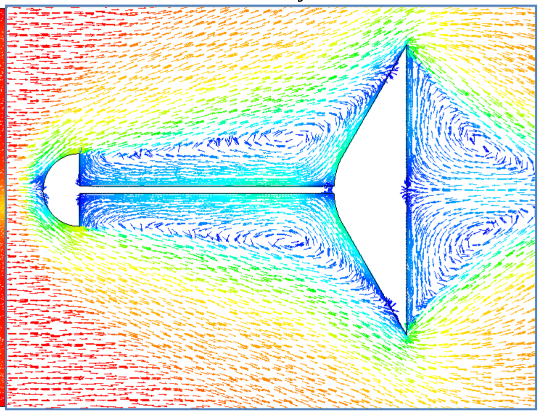
c. Temperature contours



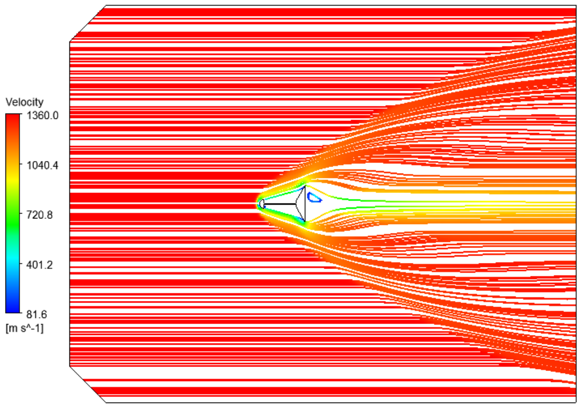
d. Density contours



e. Velocity Vectors



f. Clipped view



g. Velocity Streamlines

Fig. 4 : Convex spike blunt nose fluid behavior for 4 Mach speed and 0 deg AOA

Above figure shows the different variables contours of 0deg AOA and 4.0 M speed for convex spike attached to the blunt body, in the pressure contours it can be seen that the body fitted shock has been developed at just in front of convex spike and flow separated successfully at blunt wall location, this will reduce the force on blunt wall. Pressure variation can be seen that the peak pressure of contour is captured at the spike upstream.

Velocity contour patterns plots have been captured for convex spikes blunt nose body, it can be seen that there is shockwave (low velocity zone) in the upstream of the spike and also it can see that there is a recirculation flow pattern captured in the downstream of the blunt body, and the shocks are symmetric in nature.

Temperature Contour plots for convex spikes blunt nose body as shown in fig. It can be seen that peak temperature at convex spike. Temperature is also high in downstream of the spike and low in blunt body. Density contour plots for blunt nose with convex spike as shown in figure, it can be seen that peak density appears at spikes surface due to thermal properties changes.

Vector plot for convex spikes Blunt nose body is shown in figure. It can be seen that the speed and direction of moving fluid do not change until the moving fluid striking the convex spikes. Due to spike in location flow gets separate and flow hitting on blunt face is less. Streamline plots are the lines of constant value of the stream function. It can seen figure that there is a considerable flow recirculation is being captured in the downstream of the blunt surface.

6. CONCLUSIONS

When this spike is fastened to a blunt cone, it successfully shifts the flow re-attachment point away from the model. This is accomplished by altering the shock structure, which reduces wave drag and the drag coefficient. It was found that the spike with the convex, flat disc-shaped tip had the most potential for reducing wave drag after examining several spike shapes. This result was drawn using the drag coefficient and the percentage of drag decrease seen with different spike designs. The flow field surrounding the blunt body is altered depending on the geometry of the spike. By moving the re-attachment point away from the model, the convex and flat disc-shaped spike arrangement in particular was found to be helpful in reducing the detrimental effects of shock waves on aerodynamic performance. The research indicates that optimizing the design of the spikes may significantly reduce drag coefficients and increase the aerodynamic efficiency of the blunt body. The convex and flat disc-shaped spike exhibits considerable potential since it can change the shock structure and reduce wave drag. Taking everything into account, your work highlights the critical role that spike design plays in improving aerodynamic performance, particularly in reducing wave drag on blunt bodies. With the use of this knowledge, aerospace vehicle designs might be improved, leading to an overall increase in 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