Analysis Of Fluid Dynamics For Concave Blunt Nose At Various Mach Speeds With A Zero Degree Spike Angle Of Attacks

Mamatha A [1], Ananda C B [2], Shivakumara N V [3]

[1,2,3]*Department of Mechanical Engineering, Government polytechnic Channasandra, Karnataka, India.*

ABSTRACT

Analyzing fluid dynamics around a blunt nose with various spikes and angles of attack involves understanding how the shape and orientation of the nose affect airflow and aerodynamic forces such as lift and drag. Begin by defining the geometry of the blunt nose and the various spikes that will be attached to it. This includes specifying dimensions, shapes, and positions of the spikes relative to the blunt nose. Use a CFD software package such as ANSYS Fluent, COMSOL Multiphysics, or OpenFOAM to simulate the flow around the blunt nose with different spike configurations. Compare the performance of different spike configurations and angles of attack based on metrics such as lift, drag, and aerodynamic efficiency. Identify which configurations produce the desired aerodynamic characteristics for the given application. Based on the simulation results, optimize the design of the blunt nose and spikes to improve aerodynamic performance, minimize drag, or achieve other design objectives.

The provided information suggests that blunting the front surface of an aerospace vehicle serves as a form of thermal protection. However, despite this blunting, the nose still experiences intense thermal action, necessitating even greater thermal protection than the peripheral parts of the vehicle. To address the issue of wave drag, which typically occurs due to the blunt nose shape, modifications to the flow field in front of the vehicle are necessary. One technique to achieve this modification is by employing a retractable nose spike. By deploying a retractable spike, the flow field can be altered, potentially reducing wave drag and improving aerodynamic performance.

Keywords: Zero degree Spikes, Concave Blunt Nose, computational fluid dynamics, Angle of Attacks, turbulence, aerodynamics.

1. Introduction

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%.

2. 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.

3. 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.

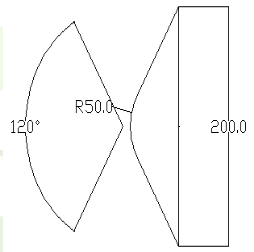


Fig. 1 : Blunt body without spike configuration

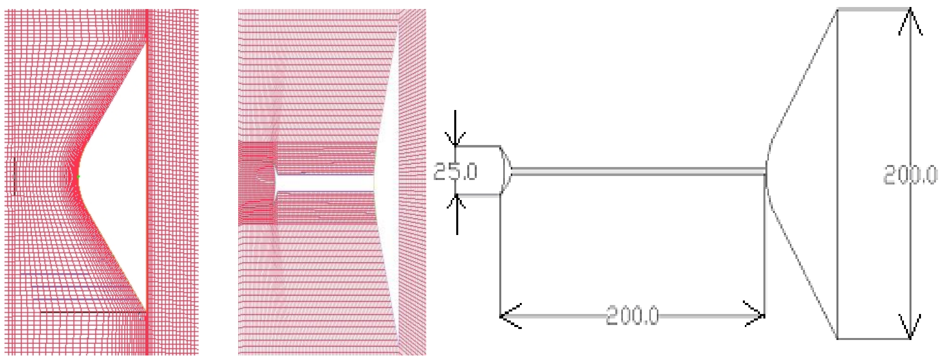


Fig. 2 : Meshed model for Blunt nose, Concave and concave spike configuration

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

5. CFD Results and Discussions

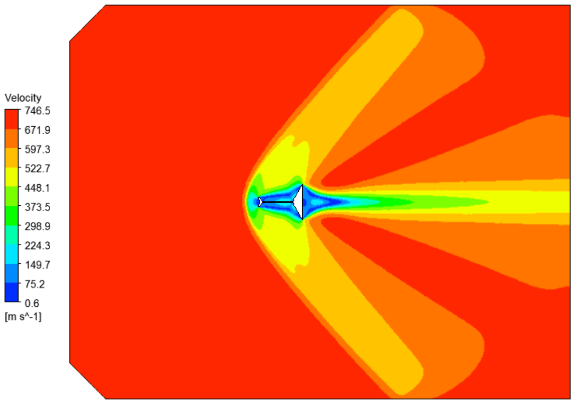
Below figures 3 shows the different variables contours of 0deg AOA and 2.0 M speed for concave spike attached to the blunt body, in the pressure contours it can be seen that the body fitted shock has been developed at just in front of concave spike and flow separated successfully at blunt wall location, 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 concave 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.

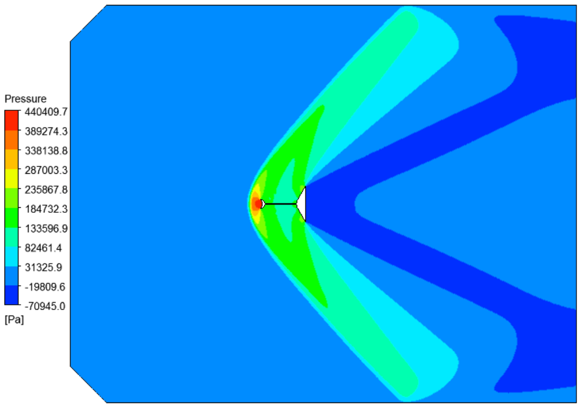
Temperature Contour plots for convex spikes blunt nose body as shown in fig. It can be seen that peak temperature at concave spike. Temperature is also high in downstream of the spike and low in blunt body. Density contour plots for blunt nose with concave spike as shown in figure, it can be seen that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

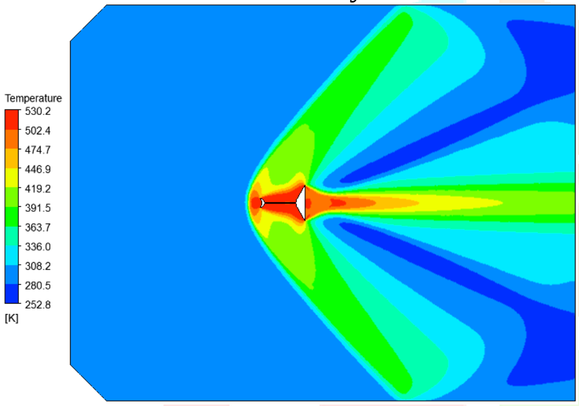
Velocity, Pressure, Density, Temperature Contours & Velocity Vectors, Streamlines plots



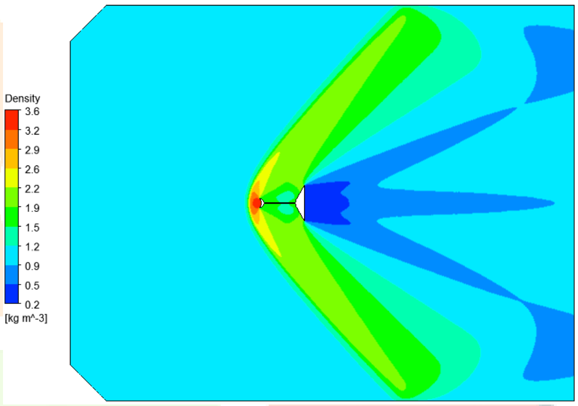
a. Velocity contours



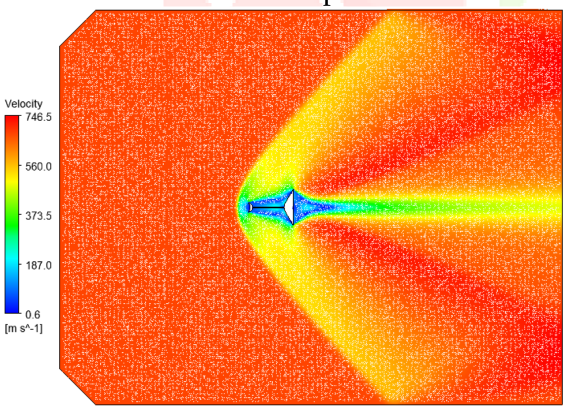
b. Pressure contours



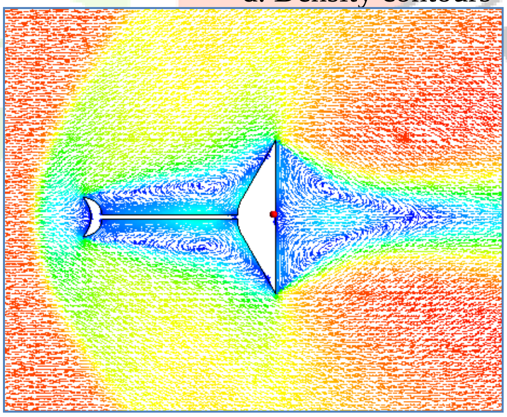
c. Temperature contours



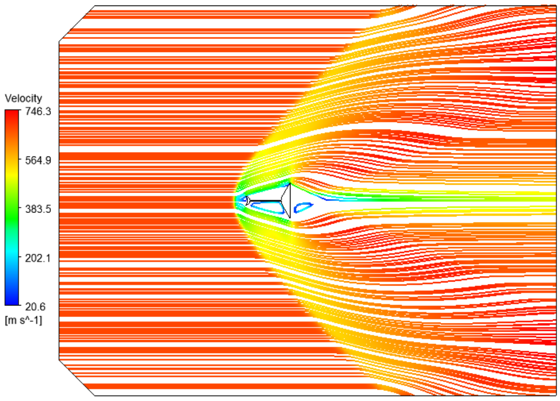
d. Density contours



e. Velocity Vectors



f. Zoomed view



g. Velocity Streamlines

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

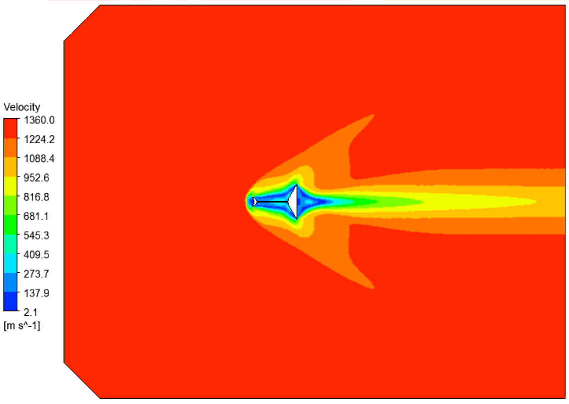
Below figures 4 shows the different variables contours of 0deg AOA and 4.0 M speed for concave spike attached to the blunt body, in the pressure contours it can be seen that the body fitted shock has been developed at just in front of concave spike and flow separated successfully at blunt wall location, 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 concave 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.

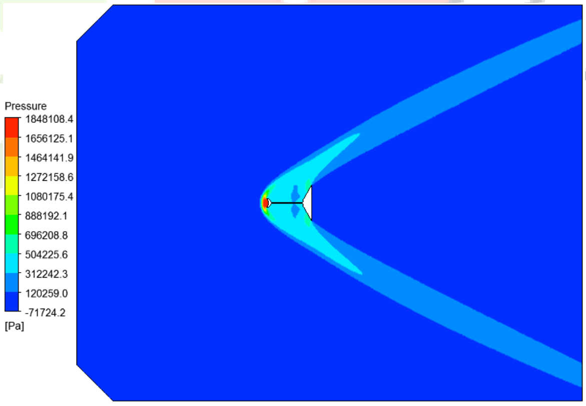
Temperature Contour plots for convex spikes blunt nose body as shown in fig. It can be seen that peak temperature at concave spike. Temperature is also high in downstream of the spike and low in blunt body. Density contour plots for blunt nose with concave spike as shown in figure, it can be seen that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

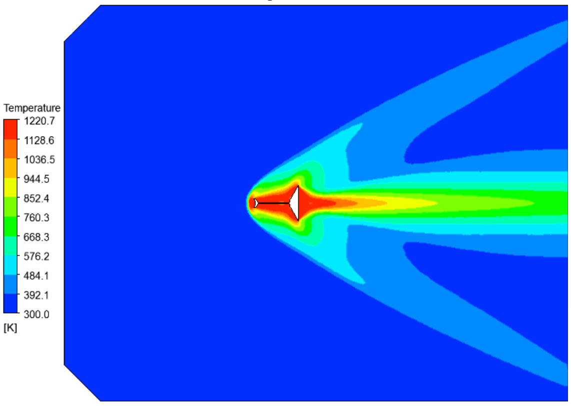
Velocity, Pressure, Density, Temperature Contours & Velocity Vectors, Streamlines plots



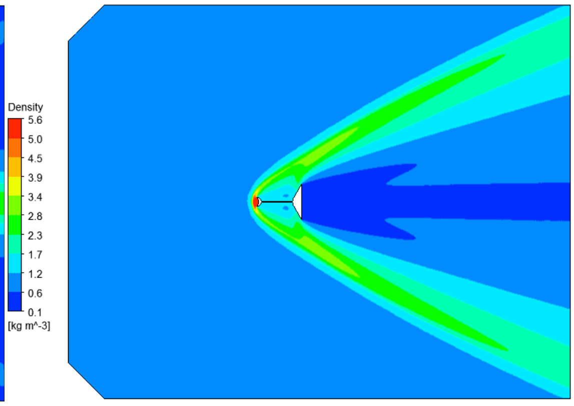
a. Velocity contours



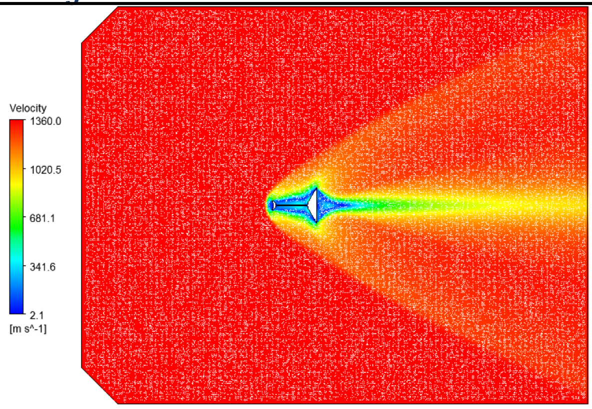
b. Pressure contours



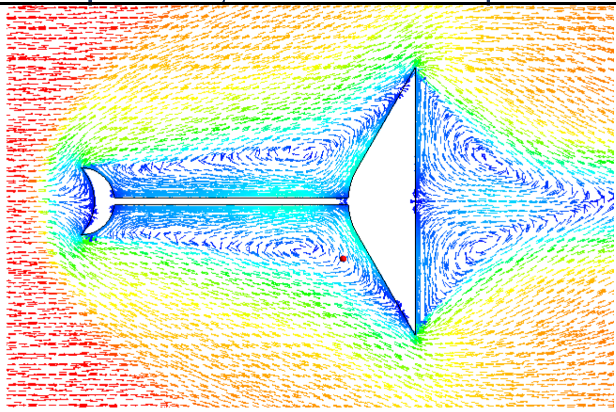
c. Temperature contours



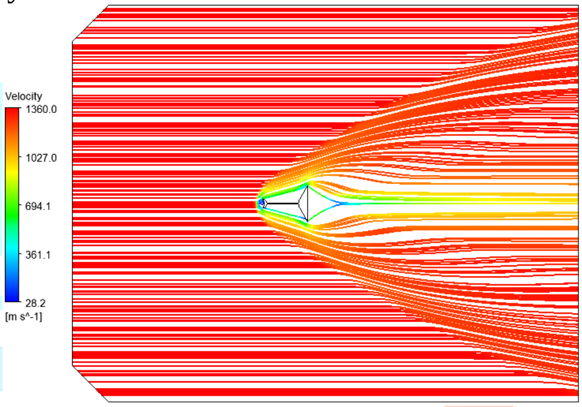
d. Density contours



e. Velocity Vectors



f. Zoomed view



g. Velocity Streamlines

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

6. Conclusions

The provided information highlights the effectiveness of using a spike attached to a blunt body to alter the shock structure and reduce wave drag. Specifically, the flat aero disc-tipped spike, when attached to the blunt cone, successfully pushes the flow re-attachment point away from the model. This alteration in the shock structure helps mitigate wave drag, resulting in a reduction in the drag coefficient. Comparative analysis of different spikes indicates that the spike with a concave and flat disc shape may be considered the most effective in terms of reducing wave drag. This conclusion is drawn based on the drag coefficient and the percentage of drag reduction observed with different spike configurations. The utilization of a concave and flat disc-shaped spike likely contributes to modifying the flow field around the blunt body, leading to a more favorable shock structure and reduced wave drag. By pushing the re-attachment point away from the model, this spike configuration helps minimize the adverse effects of shock waves on aerodynamic performance. Overall, the findings suggest that the selection of the spike shape plays a crucial role in wave drag reduction. The concave and flat disc-shaped spike emerges as a promising choice for enhancing the aerodynamic efficiency of the blunt body, as it effectively alters the shock structure and contributes to reducing drag coefficients.

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