



INTERNATIONAL JOURNAL OF CREATIVE RESEARCH THOUGHTS (IJCRT)

An International Open Access, Peer-reviewed, Refereed Journal

Effect Of Four Mach Number And Angle Of Attack On Convex Blunt Nose

R VITTAL¹, CHANDRAPPA S², Ananda C B³

¹ Senior Scale Lecturer, Department of Mechanical Engineering, Government polytechnic Mulabagal, Karnataka, India.

² Lecturer, Department of Mechanical Engineering, Government polytechnic K.G.F, Karnataka, India.

³ Senior Scale Lecturer, Department of Mechanical Engineering, Government polytechnic Channasandra, Karnataka, India.

ABSTRACT

When designing vehicles capable of great speeds, wave drag must be carefully considered. Modifying the flow field ahead of the body may decrease drag. A retractable nose spike seems to be the simplest and most efficient way to lower drag on the vehicle by causing flow separation and altering the shock structure surrounding the body. There are various approaches to modify the flow field ahead of a supersonic or hypersonic body. The blunt nose of an aeroplane is the primary focus of the present investigation into flow at high speeds. In order to lessen the impact of drag, researchers looked at three different kinds of spikes. These computational models indicate that the optimal aero will reduce lift and drag. To back up the numerical findings, we utilized data from the literature. If you want better aerodynamics, less drag, or to accomplish any other design goal, you should change the blunt nose and spikes based on the results of the simulation. The data presented suggests that thermal protection may be achieved by blunting the front surface of an aircraft. Even with this reduction, the nose is still subject to a great deal of heat activity and needs far more thermal protection than the remainder of the vehicle. Modifying the frontal flow field is necessary to lessen wave drag, which is often generated by a blunt nose shape. A retractable nose spike is one method to implement this modification. One way to improve aerodynamic efficiency and decrease wave drag is to deploy a retractable spike, which may change the flow field.

Keywords: Mach number, Spike angle, CFD, Drag Force, Lift Force.

1. INTRODUCTION

As a subfield of fluid mechanics, fluid dynamics examines how various fluids, such as gases and liquids, move through a system. Aerodynamics and hydrodynamics are two of its subfields that investigate gases in motion. Among the many applications of fluid dynamics are the following: the prediction of weather patterns, the calculation of petroleum flow rates in pipelines, the modeling of fission bomb explosions, and the evaluation of aircraft forces and moments. Its principles are also used in traffic engineering, which treats traffic as an unbroken flow. These practical domains are based on the theoretical underpinnings of fluid dynamics, which include semi-empirical and empirical rules obtained from flow measurements and applied to real-world issues. Calculating the temperature, density, pressure, and velocity of the fluid as functions of both space and time is a common approach to solving fluid dynamics problems.

One that reduces drag An aero-spike's purpose is to lessen the impact of blunt body pressure drag while traveling at supersonic speeds. The aerospike generates an independent shock ahead of the body. By establishing a recirculating flow zone between the shock and the forebody, drag is reduced and the profile is made more aerodynamic. The Trident missile was the first to adopt this idea, and it is believed to have added 550 kilometers to its range. The Trident aero-spike consists of a flat circular plate attached to an extended boom. The moment the missile leaves the submarine and breaches the water's surface, it is launched. Because of the aero-spike, the nose could be rounded down, which decreased drag and made more space inside for propellant and cargo. This mattered greatly since the new Trident IC-4 missile, which superseded the Poseidon C-3, could go further thanks to its extra propulsion stage. For the post-boost vehicle to fit within the existing underwater launch tunnels, the third stage engine had to be placed in the middle and the reentry vehicles organized around it. An essential consideration when a body is in motion is the drag force. Drag rises while a car accelerates. Several things contribute to the vehicle's resistance to movement. You may sort drags by their origin. One example is the concept of form drag, which includes profile drag, friction drag, pressure drag, wave drag, and induced drag. Because it is pertinent to the current case study, only wave drag has been considered. The term for the pulling force that develops as a shockwave is wave drag.

The aerodynamic factor of wave drag on a body in a hypersonic flow is crucial. To mitigate the heating issue, which becomes more apparent during aircraft ascent, a rounded body with a wide nose radius is necessary. Consequently, the car encounters increased wave resistance. Enhancing power from the propeller system while minimizing fuel consumption and requirements would preserve the vehicle's structural integrity and cargo capacity by lowering wave drag, an essential component of hypersonic flight. More than half of an airplane's mass is fuel, and a one percent reduction in drag boosts the range or cargo capacity by around ten percent.

2. LITERATURE REVIEW

Overview of the literature on computational aerodynamics of complicated flows over blunt bodies and aero spikes for drag reduction in subsonic, supersonic, and hypersonic flows. At supersonic speeds, blunt bodies experience forebody pressure drag, which may be mitigated with the use of a drag-reducing aero spike. The aero spike generates its own shock wave that travels in front of the body. A reticulating flow zone enhances body profile and lowers drag between the shock and the forebody.

In order to do CFD simulations, methods for generating both organized and unstructured meshes are required. Hybrid grids, which might be called chimera, composite, or patched grids, are sometimes made by combining these approaches. In this essay, we will go over the pros and cons of each of these alternatives. For flow problems with finite volume formulations including variables like temperature, pressure, velocity, etc., the solution is provided by the nodes inside each cell. A CFD solution's correctness is impacted by the grid's cell count [2, 20]. An important consideration for any moving object is drag. As a vehicle gains speed, the amount of drag it experiences increases. This refers to the many forms of resistance that the vehicle faces as it travels. Many different types of drag exist, each with its own origin story [3]. When an abrupt object is placed in front of a supersonic stream, it causes a shock wave to be generated. Shock waves are called oblique shock waves because they start at an angle to the surface of the blunt body. Computational fluid dynamics (CFD) became more popular due to fast computers and powerful numerical approaches [5, 6, 7]. Computational fluid dynamics (CFD) provides an affordable alternative to real-world flow simulation for fluid dynamics theory and experimentation. The practically applicable, analytically tractable method has been validated by its positive outcomes [9]. For this reason, it is possible to evaluate a CFD tool for handling shock capturing issues by comparing the numerical findings with the analytical data that is already accessible. This tool uses grid independence studies. To alleviate the burden of time-consuming CAD assistance during preliminary design investigations, a rapid geometry engine (RAGE) was created. Using a component-based methodology, the geometry tool constructs intricate airplane shapes. The primary components may be seen and understood using the basic production techniques described in [18]. To showcase the flexibility of the geometry tool, a variety of aerodynamic analysis methods with varying degrees of realism are used to a chosen geometry model.

3. COMPUTATIONAL MODEL

It is necessary to acquire sufficient geometric knowledge about a test case in order to simulate it on a digital computer. A well defined portion of the whole system, the fluid domain has boundaries that were set with care to avoid interfering with the problem's physics. In every one of these case studies, the computational domain is the fluid around the geometry, which pertains to external flow problems. Upstream zones often see shock capturing enhancements of 8-10 times the body width or base diameter, whereas distant fields may have enhancements of 3–5 times. It should be mentioned that while all the present case studies deal with outer flow problems, they only cover the fluid domain and not the solid body. There is an emphasis on symmetric models in the provided case studies. In both the spike-free and

spike-accompanied cases, the blunt cone body is represented for flow at a zero-degree angle of attack as well as two non-zero angles of attack, namely 5 degrees and 15 degrees.

The 2-dimensional shape of a blunt body was constructed using the ICEM CFD industrial standard code, as seen in Figure 1. In Figure 2 we can see the geometric data for blunt bodies with and without spike configurations. Computational fluid dynamics (CFD) simulation relies on both structured and unstructured mesh creation methods. Sometimes these techniques are combined to create hybrid grids, which are also called chimera, composite, or patched grids. The essay [1] goes over the pros and cons of each approach.

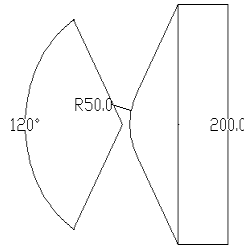


Fig. 1 : Blunt body without spike configuration

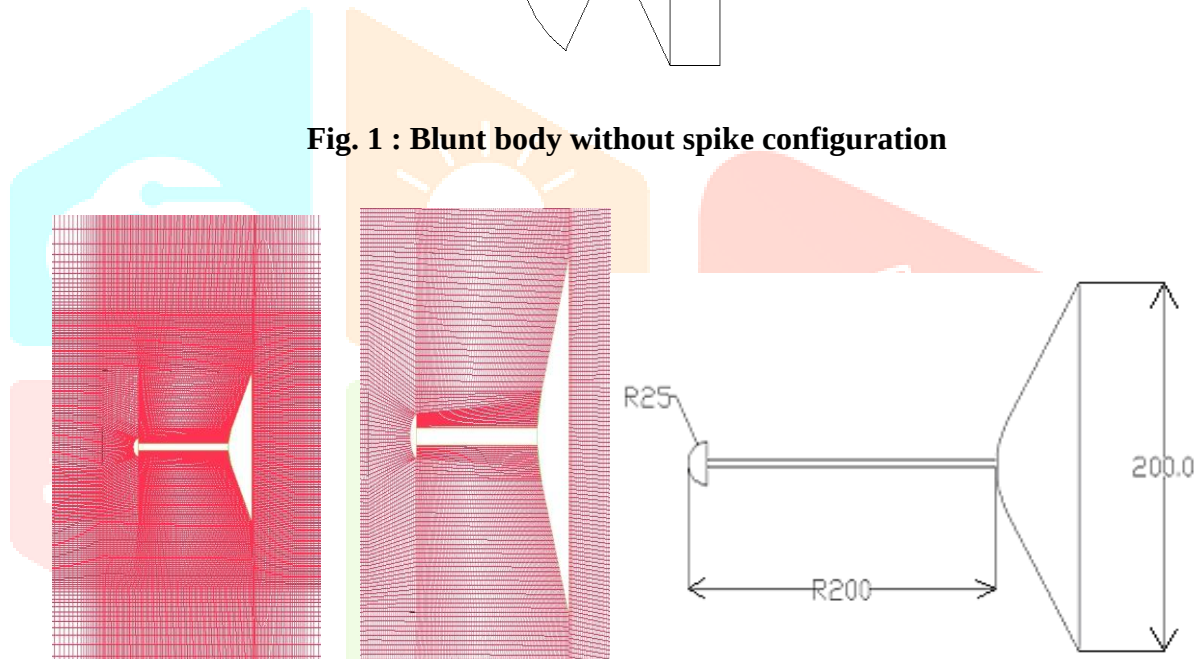


Fig. 2 : Meshed model for Blunt nose, Convex and convex spike configuration

4. PROBLEM STATEMENT

The flow geometry constraint and the flow field beginning condition must be established in order to get precise solutions, and the fluid must satisfy the governing equations. Integrating the governing equations requires exact measurements of temperature, pressure, and velocity at the region's border. When something is ephemeral, we need to know the value of the dependant variable at time $t=0$ and use first-order temporal derivatives. This is called the initial condition. Boundary conditions are extra limitations imposed on the actual borders of the fluid region. But there's no need for beginning conditions in this case since steady state is assumed. At 5 kilometres (16404 feet) above sea level, the typical metrics used to characterize the air conditions are temperature, pressure, and velocity. It takes a detour toward an unconscious route. Outputs from the inner domain are projected onto all variables in the computational domain. As a result of its viscosity on the solid surface of the blunt body, the fluid is believed to cling to the wall. The no-slip criteria states that the solid must not be moving at the same

speed as the fluid around it. Since the fluid is assumed to be non-slippery, the surfaces of the blunt cone model are subjected to the wall boundary condition.

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

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

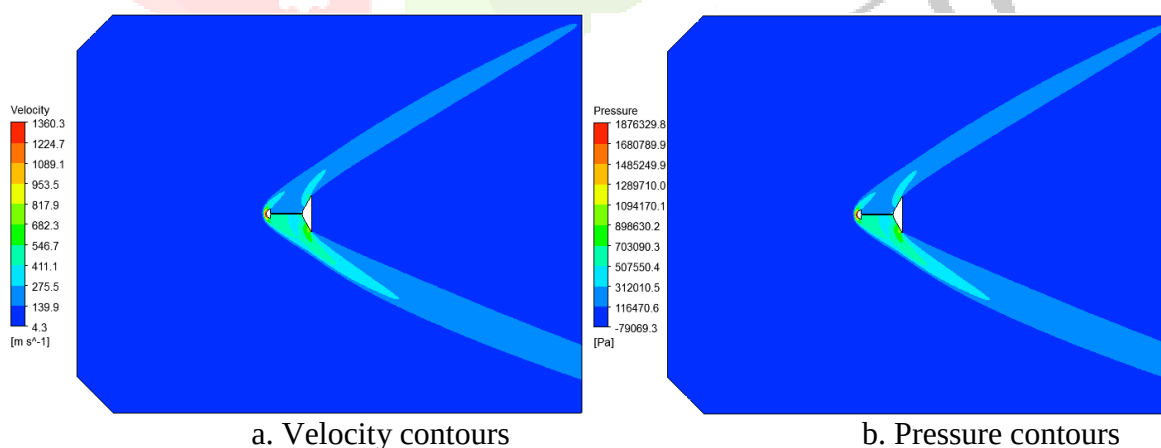
The Energy Equation:
$$\frac{\partial}{\partial t}(\rho \bar{h}) + \frac{\partial}{\partial x_j}(\rho \bar{U}_j \bar{h}) = -\frac{\partial}{\partial x_j}(Q_j + \rho \overline{u_j'' h'})$$

5. CFD RESULTS AND DISCUSSIONS

At zero degrees of angle of attachment (AOA) at speeds of 4.0 mach, the convex spike coupled to the blunt body in Figures 3 and 4 below displays a diversity of morphologies. The pressure contours show the distribution of pressure around the convex spike and blunt body. The time immediately before the convex spike is considered to have been a body-fitted shock. The shape of the spike seems to be rather similar to the shock wave, according to this. Additionally, it has been shown that the blunt wall position is optimal for flow separation. Reduced aerodynamic drag may result from this separation if it lessens the force acting on the blunt wall. It is observed that the contour of peak pressure is recorded upstream of the spike. The place with the highest pressure was most likely created by the compression effects induced by the geometry of the spike and the incoming flow conditions. The aerodynamic performance and structural stresses might be affected by this increased pressure.

5.1 Convex spikes 5 deg AOA and 4.0 Mach

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



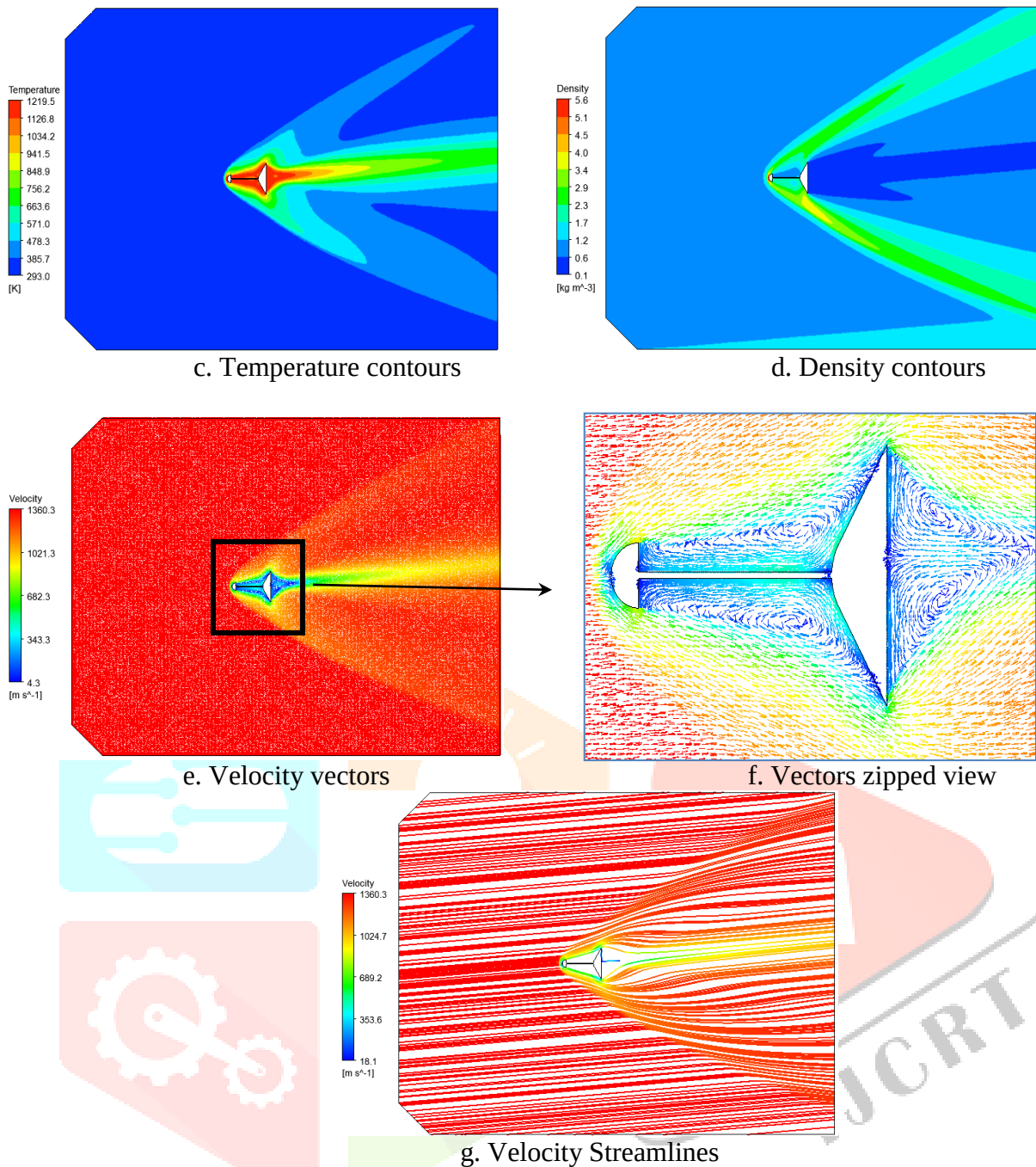


Fig. 4 : Convex spike blunt nose fluid behavior for 4 Mach speed and 5 deg AOA

At 5 degrees angle of attack and 4.0 meters per second, the convex spike connected to the blunt body exhibits a variety of varied forms, as seen in the image above. At the spot immediately ahead of the convex spike, the body-fitted shock has formed, and the flow has effectively divided at the blunt wall's position, lowering the stress on the wall. The pressure variation clearly shows that the upstream spike is where the contour's peak pressure is contained.

Convex spikes with blunt nose bodies have had velocity contour pattern plots created. The patterns indicate a low-velocity zone upstream of the spike and a recirculation flow pattern downstream of the blunt body. All of the shocks are equally powerful.

Convex spikes and blunt nose bodies' temperature contour plots are shown in Figure. The highest point is seen at the convex spike. Furthermore, the blunt body has a low temperature whereas the area downstream from the spike has a high temperature. The picture shows density contour plots for a blunt

nose with a convex spike. The highest density occurs at the spike's surface due to variations in thermal properties.

The convex peak vector diagram A figure with a pointed muzzle is seen. Both the direction and the speed of the fluid are clearly unaltered until it reaches the convex spikes. The position spike causes the flow to split, resulting in fewer frequent impacts with the blunt face. A streamline graphic shows the continuous lines in a stream function. According to the figures, a considerable quantity of flow recirculation is being captured after the blunt surface.

6. CONCLUSION

After fastening this spike to a blunt cone, the flow re-attachment point is effectively moved away from the model. Reducing wave drag and the drag coefficient is achieved by modifying the shock structure. It was determined that the spike with the convex, flat disc-shaped tip had the best chance of reducing wave drag after several other designs were examined. A combination of the drag coefficient and the percentage of drag reduction obtained with different spike designs was used to generate this result. The form of the spike determines the flow field that encircles the blunt body. To mitigate the detrimental effects of shock waves on aerodynamic performance, it was discovered that relocating the re-attachment point away from the model—specifically, the disc-shaped spike arrangement with convex and flat surfaces—was advantageous. Reducing drag coefficients and enhancing blunt body aerodynamic efficiency may be achieved, according to the study, by altering the spike design. Because it can change the shock structure and reduce wave drag, the flat, convex disc-shaped spike has great potential. When all factors are considered, your study shows that spike design is highly important for improving aerodynamic performance, especially for blunt bodies in reducing wave drag. With this new information, engineers may be able to create more efficient designs for flying vehicles.

References

1. S. W. Yuan, "Foundations of fluid mechanics", PHI Publications, 1988.
2. K. Muralidhar & T. Sundararajan, Computational fluid flow and heat transfer, Narosa publishing house, 1984.
3. Pradip Niyogi, S. K. Chakrabartty, M. K. Laha, Introduction to Computational Fluid Dynamics, Pearson Education Series, 2005.
4. 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.
5. 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.
6. 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.

7. J.S.Shang, Plasma injection for hypersonic blunt body drags reduction, AIAA Journal, Vol.40 No-6, June 2002.
8. 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.
9. 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
10. David L. Rodriguez* and Peter Sturdza^{2†}, A Rapid Geometry Engine for Preliminary Aircraft Design, Desktop Aeronautics, Inc., Palo Alto, CA, 94301
11. 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
12. Timothy, Baker. Mesh generation: Art or science? MAE Department, Princeton University, Princeton, NJ 08540, USA
13. 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
14. Mark Filipiak, Mesh Generation, Version 1.0, Edinburgh Parellel Computing Centre, University of Edinburgh, November-1996.
15. H. K. Versteeg & W. Malasekera, An introduction to Computational Fluid Dynamics-The finite volume method, Pearson Prantice Hall, 1995.
16. John. D. Anderson, Jr, Fundamentals of Aerodynamics, McGraw Hill International Editions, 1985.
17. H W Liepmann & A Roshko, Elements of Gas Dynamics, John Wiley & Sons, Inc. – Galcit Aeronautical series, 1965.
18. John. D. Anderson, “Computational Fluid Dynamics – the basics with applications”, McGraw Hill Inc, 1985.
19. Joel. H. Ferziger and Milovan Peric, “Computational Methods for Fluid Dynamics”, 3rd revised edition, Springer Verlag publications, 2003.
20. C. A. J. Fletcher, Computational techniques for fluid dynamics – 1, fundamental and general techniques, 2nd edition, 1990.
21. J.F.Thompson, A composite grid generation code for general 3D regions — the Eagle code, AIAA J., Vol. 26 (3) pp.271-272 (1988).