Fluid Dynamics Analysis for Blunt Nose with Various Spikes and Angles of Attacks

KASHINATH S. V. H[1], RAVICHANDRA S. C[2], G. N. YOGAVARDHAN SWAMY[3]
[1, 2] Department of Mechanical Engineering, Government polytechnic, Channapatna, Ramanagar, Karnataka, India.
[3] Department of Mechanical Engineering, Government polytechnic, Nagamangala, Mandya, Karnataka, India

Abstract- ***The wave drag exerted on a body in supersonic/hypersonic flow is very critical and important problem of aerodynamic analysis. In order to minimize the drag force is essential to use a blunt body with a large nose radius which in turn induces drag for the vehicle motion. The present work involves study of supersonic/hypersonic flow around the blunt nose of an aero vehicle. Three different types of spike have been considered for the purpose of drag reduction studies. Blunting of the front surface is considered in a certain sense as a way of thermal protection of an aero vehicle. But still, this blunted nose experiences the most intensive thermal action, therefore it requires thermal protection to even a greater extent than the peripheral part of the aero vehicle. The problem of wave drag can be alleviated by modifying the flow field in front of the body. One of the techniques to modify the flow fields is using retractable nose spike. In order to complement the numerically simulated results, they have been compared with the available experimental results.***

Indexed Terms- Energy, Temperature, Computational Fluid Dynamics, Blunt Nose, Heat Transfer, Turbulence, Aerodynamics.

I. INTRODUCTION

A Drag-Reducing Aero-spike is a device used to reduce the fore-body pressure drag of blunt bodies at supersonic speeds. The aero-spike creates a detached shock ahead of the body. Between the shock and the fore-body a zone of recirculating flow occurs which act like a more streamlined for body profile, reducing the drag. This concept was first introduced on the Trident missile and is estimated to have increased the range by 550 km. The Trident aero-spike consists of a flat circular plate mounted on an extensible boom which is deployed shortly after the missile breaks through the surface of the water after launch from the submarine. The use of the aero-spike allowed a much blunter nose shape, providing increased internal volume for payload and propulsion without increasing the drag. This was required because the Trident IC-4 was fitted with a third propulsion stage to achieve the desired increase in range over the Poseidon C-3 missile it replaced. To fit within the existing submarine launch tubes the third stage motor had to be mounted in the center of the post-boost vehicle with the reentry vehicles arranged around the motor. 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. Some of them are wave drag, friction drag, pressure drag, induced drag, form drag, profile drag etc. Only wave drag has been discussed here which is relevant to the present case study (for details of the other types of drags, refer [3]). Wave drag is the drag force which comes into picture with the formation of shock wave.

The wave drag exerted on a body in hypersonic flow is a very critical and important problem of aerodynamics. In order to minimize the heating problem, which dominates during the ascent phase of the flight, it is essential to use a blunt body with a large nose radius. This increases wave drag on the vehicle. This wave drag which is crucial in the hypersonic flow has to be minimized in order to make the best use of the thrust of the propulsive system and keep the fuel consumption as well as the propulsive system requirements low, improving the payloads and structural integrity of the vehicle. Fuel is almost half the aircraft's base weight and a 1% reduction in drag enhances the vehicle pay load capacity or the range by around 10%.

II. COMPUTATIONAL MODEL

It is necessary to have enough details of geometry of any test case for the purpose of modeling it on digital computer. Fluid domain is a virtually cut portion of the whole system whose outer boundaries are decided in such a way that the physics of the problem do not get affected. In all the present case studies, computational domain is the fluid surrounding the geometry (external flow problems). It is usual practice to go up to 8-10 times the width or base diameter of the body in the far field regions and 3-5 times in the upstream region for shock capturing problems.

It should be noted that, only the fluid domain is being modeled and not the solid body, since all the present case studies are being external flow problems. The models considered in the present case studies are symmetric. For both the cases (without spike and with spike) the blunt cone body is modeled for flow at zero angle of attack and two non zero AOA (5 deg & 15 deg).

2-D geometry of blunt body has been built by using ICEM CFD industrial standard code and which is shown in Fig 1. Geometrical information of blunt body without spike and with spike configurations has been shown in Fig 2.

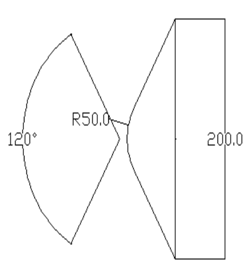

Fig. 1 : Blunt body without spike configuration

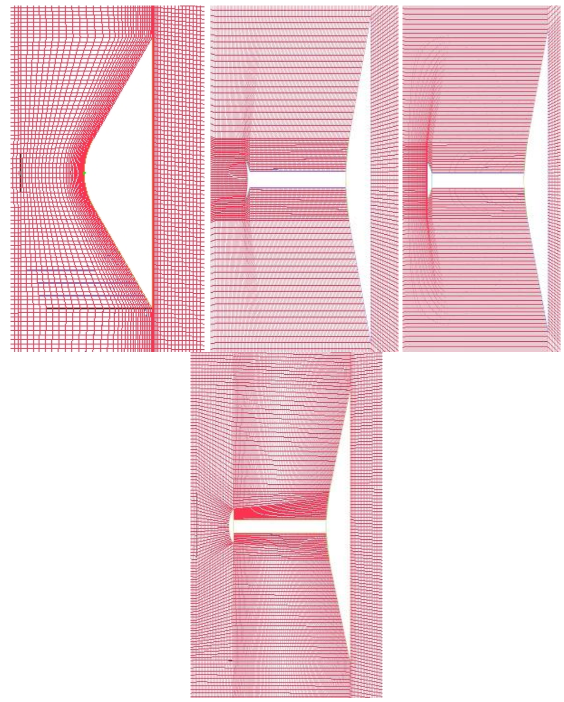

Fig. 2 : Meshed model for Blunt nose, Concave, Flat disc and Convex spikes

III. PROBLEM STATEMENT

The governing equations must be satisfied in the interior of the fluid and the specific solutions can be obtained only by prescribing the constraint of flow geometry as well as the initial state of the flow field. Hence on the boundary of the region velocity, pressure and temperature must be suitably defined to permit integration of the governing equations. In the transient problems, the time derivative is of first order and the value of dependent variable at time t=0 must be given. This is called as initial condition. Other conditions prescribed on the physical boundary of the fluid region are called Boundary conditions. However, in the present problem, steady state has been assumed and hence no initial conditions have been applied. Inlet, typical values used here are velocity, pressure and temperature, these are the ambient atmospheric conditions at an altitude of 5 kms (16404 ft) above the sea level. Opening, it open to ambient outlet. At the outlet of the computational domain, all variables are extrapolated from the interior domain. Wall, on the solid surface of the blunt body, the fluid is assumed to stick to the wall by the action of viscosity. This is called as no-slip condition and it requires that the solid and adjacent fluid do not have a velocity relative to each other. Hence the wall boundary condition is used at the blunt cone model surfaces and the fluid at these surfaces is assumed to have no-slip 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: $\frac{\partial}{\partial t}(\rho \bar{h}) + \frac{\partial}{\partial x_j}(\rho \bar{U}_j \bar{h}) = -\frac{\partial}{\partial x_j}(Q_j + \rho \overline{u_i'' h'})$

IV. CFD RESULTS AND DISCUSSIONS

Below figure shows the different variables contours for 0 deg AOA (angle of attacks) and non zero degrees of angles of attacks. Two different mach numbers are considered 2 mach and 4 mach. From the pressure and velocity contours, it can be seen that the body fitted with spike has been developed strongly shock waves and it is attached frontal area of the blunt nose. Shock waves are asymmetry in the case of non zero angles of attacks. Fig 3 shows the velocity contours plots for 0, 5 and 15 deg AOA's at 2 Mach speed with blunt nose. Similarly pressure and temperature and density contours shows in Fig 4 for 15 deg AOA.

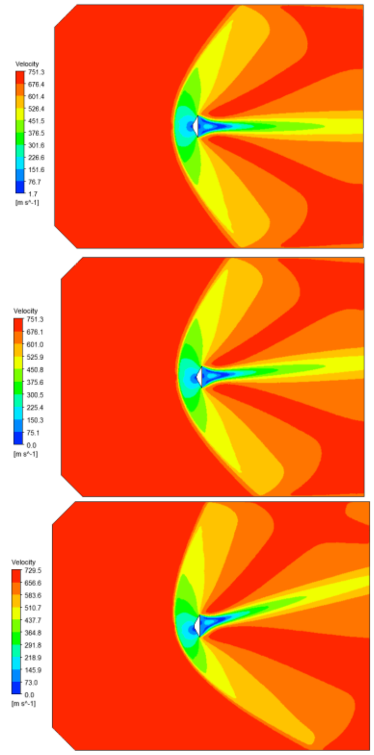

Fig. 3 : Blunt nose 2-Mach velocity contours for 0 deg, 5 deg and 15deg angles of attacks

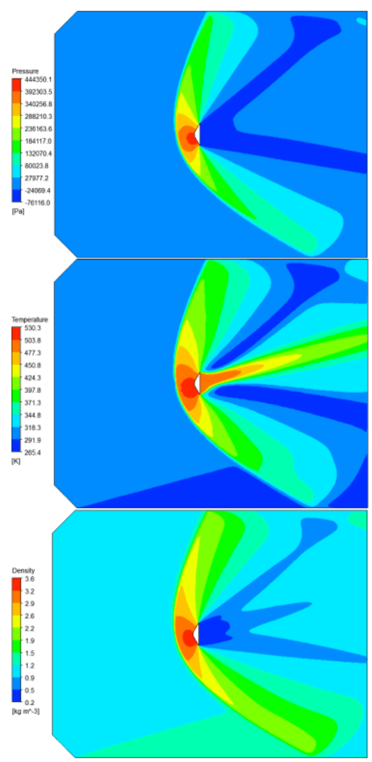

Fig. 4 : Blunt nose 2-Mach Pressure, Temperature, Density contours for 15deg angles of attacks

Below figure shows the different variables contours for 0 deg AOA and non zero degrees of angles of attacks. Fig 5 shows velocity contours for 0 deg 5 deg and 15 deg for 4 mach speed, Fig 6 shows pressure contours for similar AOA's. Shock waves are asymmetry in non zero AOA. Asymmetry shocks flow patterns will offers more drag force against symmetry.

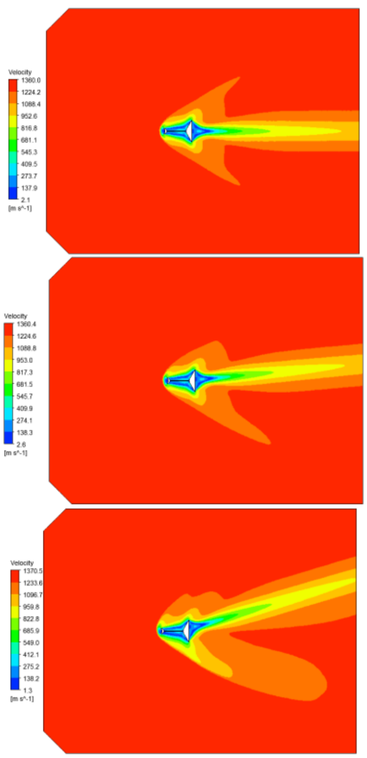

Fig. 5 : Concave spike with blunt nose 4 mach velocity contours for 0 deg, 5 deg and 15 deg angles of attacks

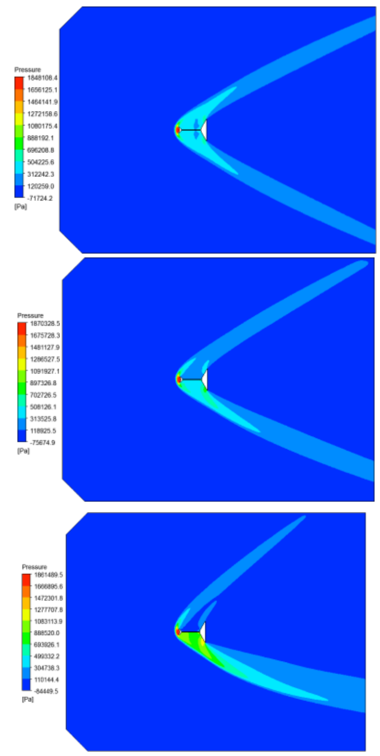

Fig. 6 : Concave spike with blunt nose 4Mach pressure contours for 0 deg, 5 deg and 15 deg angles of attacks

Similarly CFD simulations are carried out for convex and flat disc spikes, the flow patterns are observed similar to above cases. The shock structure was altered by the attachment of the spike to the blunt body. The flat aero disc tipped spike used on the blunt cone accomplishes the task of pushing the flow re-attachment point off the model fore-body as shown in the respective figure. No other among the considered spikes was able to avoid the re-attachment point. The drag coefficient and percentage of drag reduction for different spikes have been compared. From this analysis, spike with Concave & flat disc may be regarded as the best choice, from the point of view of wave drag reduction. Comparison of spikes in terms of drag reduction for 4 Mach and 0-deg AOA

Configuration details	Numerical values of drag coefficients
Blunt Nose Body	1.34

Convex spike	0.660
Concave spike	0.539
Flat spike	0.551

CONCLUSION

The shock structure was altered by the attachment of the spike to the blunt body. The flat aero disc tipped spike used on the blunt cone accomplishes the task of pushing the flow re-attachment point off the model for a body as shown in the respective figure. No other among the considered spikes was able to avoid the re-attachment point. The drag coefficient and percentage of drag reduction for different spikes have been compared. From this analysis, spike with Concave & flat disc may be regarded as the best choice, from the point of view of wave drag reduction.

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