External aerodynamic analysis for Fuselage for various angles of attacks and speeds

Vijaykumar G Tile, Thousif S N, Sujay Ranganath, Yashawanth K, Nagalingegowda M
Assistant professor, Department of Mechanical Engineering,
MCE, Hassan

ABSTRACT

Wave drag is a significant 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 fuselage body. There are several techniques to the flow field ahead of a supersonic and hypersonic speed vehicles and the use of spikes are appears to be simplest and at an efficient means of reducing drag on the vehicle by bringing to control the flow separation at fuselage front face of wing body these devices called spikes, shocks will be shifting from fuselage front to spikes front. The present work involves study of subsonic transonic and supersonic speed flow over the fuselage model. Spikes have been considered for the purpose of drag reduction studies. The numerical results have been validated with literature data.

Keywords: Energy, temperature, computational fluid dynamics, aerofoil, wing, heat transfer, turbulence, aerodynamics.

1. Introduction:

A Numerical study of the air flow over aircraft fuselage will be using the ANSYS CFX, ICEM CFD tools to derive its aerodynamic characteristics. The flow model includes 3D unsteady Reynolds Averaged Navier Stokes (RANS) equations, along with the SST turbulence model. The results will be compared with literature data leading to improvements in the CFD modeling / confirmation of the adequacy of the modeling. The report focuses on the problems involved in the generation of surface model, mesh generation, implementation of realistic physical boundary conditions in the prediction of aerodynamics derivatives over the airfoil configuration at various angles of attack and rolling angle of attacks (**0**°, **10**°&**20**°AOA).

Investigation of airflow over aircraft fuselage is very essential to understand the flow complexity. In order to reduce the drag and lift forces to achieve better performance with less power consumption. Drag and lift force’s are very critical parameters to optimize the performance. To understand flow complexity and how to improve these two parameters with the help of vortex generator kind of devices. The analyses has to be carried out at $M\infty$=1.0, 2.0 flow regimes. All computations will be carried out using the RANS modeling which is available in commercially CFD package.

Drag is the resistance to the vehicle motion which originates from different sources. Higher the drag force, higher the thrust required to move the vehicle. Wave drag is caused by the formation of shock waves and vertices formations around the vehicle. Hence, reducing the fuselage drag has become an important issue now-a-days due to optimizations of energy efficiency.

Flow separation occurs in various flow conditions and at various locations around an aircraft, especially flow separation over the fuselage is of serious concern as it directly affects the aircraft performance. Spikes are the devices that can able to minimize the drag of the fuselage with minimum modification at fuselage front locations. These devices are made up of “simple” strip which is attached convex, concave and flat shape spikes. CFD method is more popular in now being used extensively in the aerodynamic design and performance analysis using these kind of devices on aircraft fuselage which be computed with reasonable accuracy. The studies have been carried out on three different spikes geometries i.e. concave, convex and flat.

2. Geometrical Modeling

The geometrical and fluid domain details of all the test cases that have been considered for studies are discussed in this section. 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 1-3 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.

3-D geometry of fuselage body has been built by using ICEM CFD industrial standard code and which is shown in below Fig Geometrical information of fuselage body without spike and with spike configurations have been shown in below Figs.

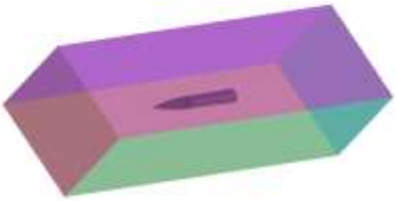

Fig.1Fuselage inside the wind tunnel

Fig.2 3-D view of Fuselage without spike

Fig.3 Fuselage with convex spike configuration

25mm height convex spike is been attached to the fuselage to separate the flow streams which can hits the fuselage fronts, this flow phenomenon will be reducing overall drag of the fuselage through simulations. This will be a good research work in terms understanding about fluid dynamics with and without spikes. If reduce the drag force with spike, it's been helping the community of aero vehicles to minimizes the drag force and indirectly increase the vehicle efficiency.

2.1 Computational Meshed Models:

Tetrahedron meshing (unstructured) method was followed for fuselage body and inside the wind tunnel to get more connectivity to transfer the mass and heat without and discontinuity. The mesh quality around fuselage is body fitted cells. Prism meshes are generated about 5 layers around the fuselage to capture the sub viscous effect of fluid flow around it. This mesh technique called unstructured-hybrid mesh method. So tetrahedron elements are considered for CFD simulation the mesh quality was well within required criteria and mesh propagation was pretty well. Below are the figs about tetrahedron meshes.

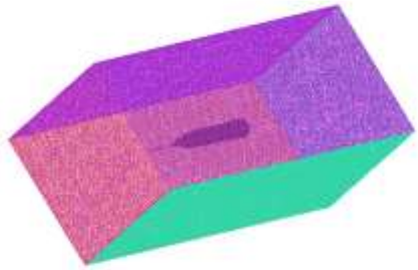

Fig.4 Fuselage meshed model inside the wind tunnel

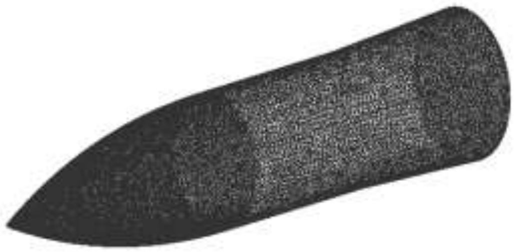

Fig.5 Mesh propagation on Fuselage surface

3. 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. 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 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 fuselage surfaces and the fluid at these surfaces is assumed to have no-slip condition. The numerical code uses an industry standard CFD code solving 3D RANS equations with a second order upwind scheme. The solution sought is for steady state and a diminishing residuals falling by 1×0^{-6} is set as the criteria for convergence. For a clarity the main equations considered for the solution is provided below.

1. The Continuity Equation:

$$\frac{\partial \bar{U}_j}{\partial x_j} = 0$$

2. The Momentum Equation:

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

3. The Energy Equation:

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

4. CFD Results

Below figure shows the different variables contours for 0° AOA and for without spikes and with spikes. 2 Mach speed flow over fuselage is considered to analyze the flow behavior. The pressure and velocity contours it can be seen that the body fitted shock has been developed strongly and it is attached to the starting point of fuselage. Shock waves are symmetry in the case of zero angles of attacks. Fig 6(a) shows the velocity contours plots for 0° AOA’s at 2 Mach speed. Similarly pressure and temperature and density changes at different locations with contours plots shows in Fig 6(b), 6(c) and 6(d) which represents the without spikes and flow over fuselage structure. Fig 7(a) to Fig 7(d) represents the fluid variables for with spikes design. Below figure shows the different variables contours for 0° AOA. Shock waves are captured accurately and the fluid behavior is changing due to compressible effect in front of fuselage.

4.1 Flow patterns over fuselage for w/o spikes

Fig a. Velocity contours

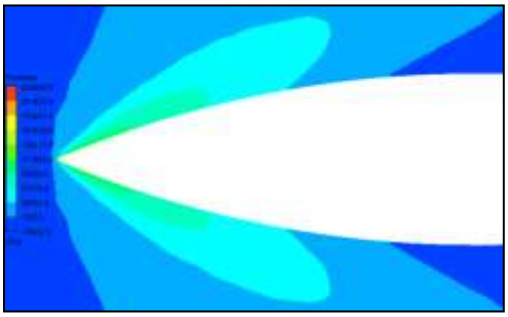

Fig b. Pressure Contours

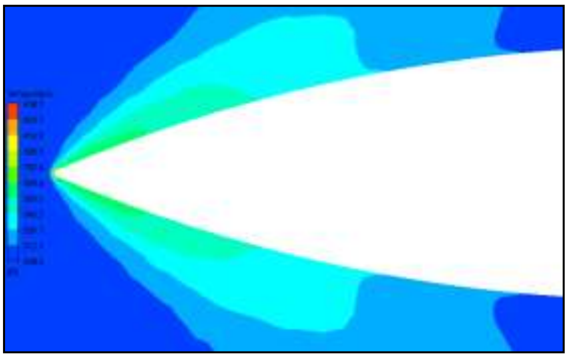

Fig c. Temperature contours

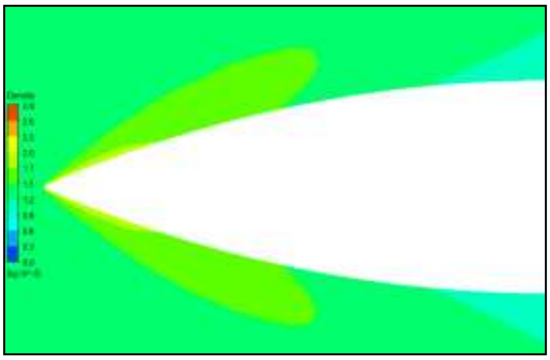

Fig d. Density contours

Fig.6(a to d) Flow variables over fuselage for without spikes

4.2 Flow patterns over fuselage for with spikes

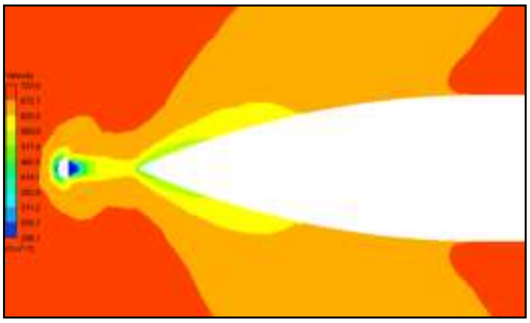

Fig a. Velocity contours

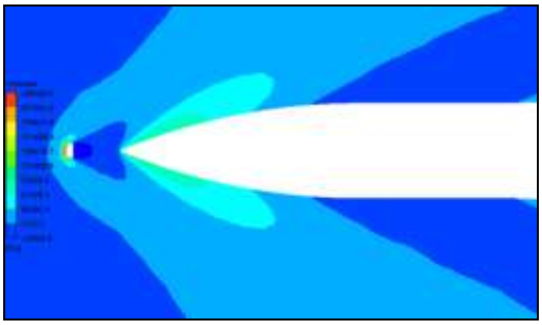

Fig b. Pressure contours

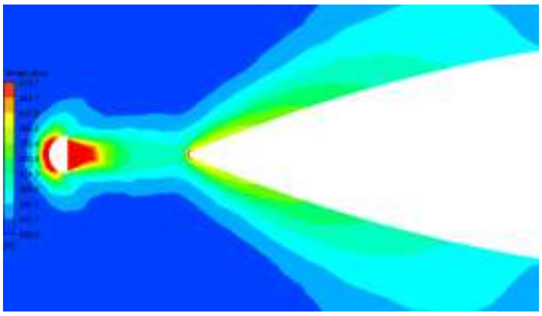

Fig c. Temperature contours

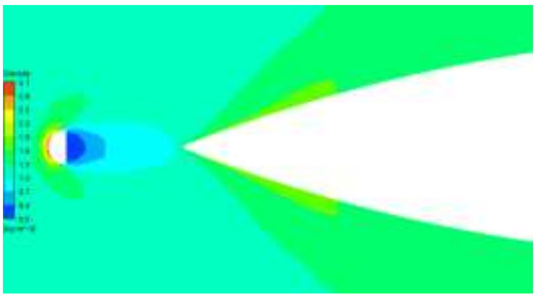

Fig d. Density contours

Fig.7 Flow variables over fuselage for with spikes

Above figure shows the different variables contours, in the pressure contours it can be seen that the body fitted shock has been developed strongly and it is attached frontal area of the fuselage. Pressure variation can be seen that the peak pressure of contour is captured at the starting point of the fuselage body.

Velocity contour patterns plots have been captured for fuselage body, it can be seen that there is shockwave (low velocity zone) in the upstream of the fuselage leading point and also it can see that there is a low velocity pattern captured in the downstream of the fuselage, and the shocks are symmetrical in nature for both upper and lower portion of the fuselage.

Temperature Contour plots for fuselage body as shown in fig. It can be seen that peak temperature at leading edge and is goes on decrease at along the length of fuselage. Temperature is also high in downstream of the fuselage. Density contour plots for fuselage body shown in figure, it can be seen that peak density appears at leading point of the fuselage due to compressibility effect of working fluid (thermal) properties changes in significantly.

Velocity contours around wing body is been plotted, these velocity contours are symmetry in nature upper and lower section of fuselage. There is a low velocity at leading portion. These kind of velocity distribution is expected and same is well captured in simulations.

Velocity and pressure contours line are added in above figure, this line graph is also represent that low velocity at starting point of the fuselage and correspondingly high pressure at same location of fuselage. This clearly represents that the compressibility effect of working fluid. This effect is symmetrical each other for zero degree AOA.

5. CONCLUSIONS

Comparison of Drag coefficients for with and without spikes

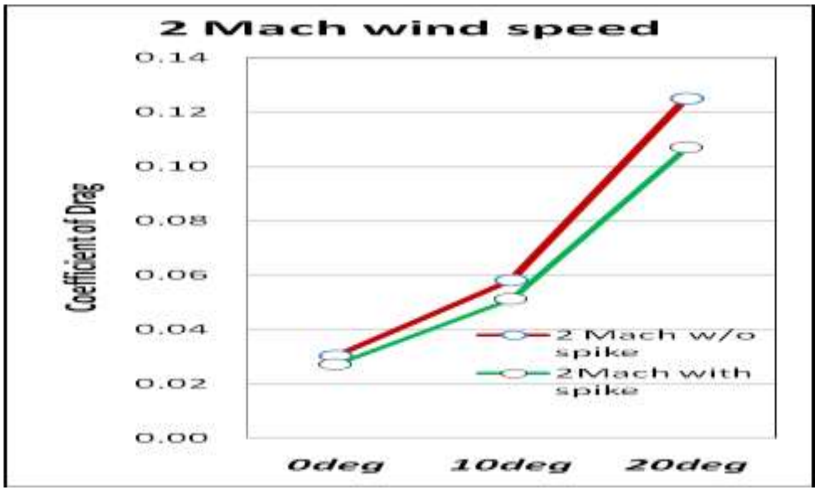

Fig.8 Drag comparison with and without spikes

Spikes are efficient devices to reduce the drag force and its coefficients, in this study one can able see the difference of flow patterns with and without spikes. Drag coefficients are also reduced by almost **12.13%** for all different AOA, which is very good sign where energy consumption can optimized through these kind of devices. Recent days the aero vehicles numbers are increasing for domestic or military purpose as well. Saving energy is the key issue to increase the performance to the cost ratio optimistically. In the report tried with one of the size and shape of spike, there is very much scope in optimizing the size with various simple shapes to further optimization of drag force coefficients. CFD results are well captured the flow field and its behavior over fuselage, these results are well correlating with literature data which were published in various aerodynamics forum worldwide.

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.