ISSN: 2278-0181
NCERAME - 2015 Conference Proceedings

CFD Analysis of Airfoil wing with Vortex Generators for Different Angles of Attack (NACA 64215)

[1]Srinidhi. R, [2]Sunil. K, [3]Sunil Kumar. J. N, [4]Sharath R Nambiyar,
[5]Prof. Vijaykumar. G. Tile.
Department of Mechanical Engineering,
Malnad College of Engineering,
Hassan.

***Abstract:*- Investigation of airflow over airfoil wing is very essential to understand the flow complexity. In order to reduce the drag and improve lift forces to achieve better performance with less power consumption for NACA 64215, which is newly developed aerofoil wing, drag and lift forces are very critical parameters to optimize the performance. Aim is to understand the flow complexity and how to improve these two parameters with the help of vortex generator kind of devices. Vortex Generators are highly efficient aerodynamic devices that are used widely in both external and internal aerodynamics as means of flow control. The project work (report) deals with the CFD analysis of VGs on a wing. Complete three dimensional results to be for the flow over wing equipped with Vortex Generators with various shapes and locations over the wing. The drag and lift characteristics of each shape and locations of VG's have to be studied. The analysis has to be carried out at subsonic (M=0.5), transonic (M=1.0) and supersonic (M=2.0) flow regimes. The best shape and location of the VG's for each of the flow regimes will be determined. All computations will be carried out using the RANS modeling which is available in commercially CFD package.**

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

1. INTRODUCTION:

Flow separation occurs in various flow conditions and at various locations around an aircraft, especially flow separation over the wing is of serious concern as it directly affects the aircraft performance. The oldest and the simplest means of flow control for separated areas is using mechanical devices called vortex generators (VGs). These devices are made up of "simple" sheet metal plates which are placed at upstream of the separated area with a predetermined cross angle with the flow. The VGs create vortices, transferring momentum from the outer region of the boundary layer into the inner flow region thereby increasing the kinetic energy of the boundary layer as well as its ability to resist to adverse pressure gradient which results in sustain the flow separations. CFD is now being used extensively in the aerodynamic design and performance analysis using these VG's on aircraft wing which can be computed with reasonable accuracy. The studies have been carried out on two different VG geometries i.e., trapezoidal shaped configurations.

2. COMPUTATIONAL MODEL

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 5-8 times the width or base diameter of the body in the far field regions and 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 VG's and with VG's) the wing body is modeled for flow at zero angles of attack.
2-D geometry of wing body has been built by using ICEM CFD industrial standard code and which is shown in Fig1. Geometrical information of wing body without VGs and with VGs configurations has been shown in Fig2.

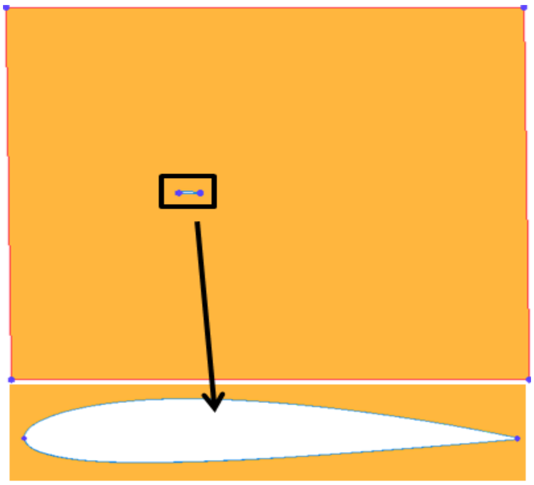


Fig.1 Wing body and wind tunnel modeling

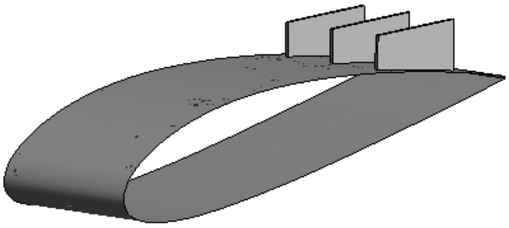


Fig.1(a) VGs are placed on wing trailing portion

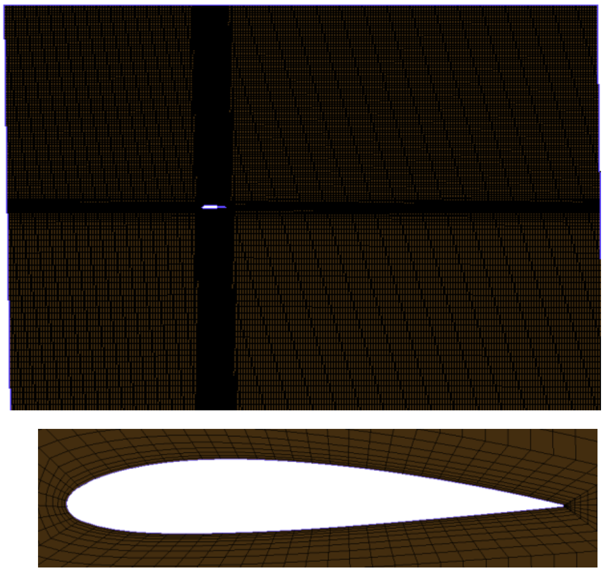


Fig. 2 Meshed model for wing and wind tunnel of structured mesh

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.

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 above the sea level. Opening, it is 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.

4. CFD RESULTS

Below figure shows the different variables contours for 0 deg AOA and for without VGs and with VGs designs. 2 Mach speed flow over wing 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 leading edge of wing. Shock waves are symmetry in the case of non zero angles of attacks. Fig 3 shows the velocity contours plots for 0 deg AOA's at 2 Mach speed. Similarly pressure and temperature and density changes at different locations with contours plots shows in Fig 3 to Fig 6 which represents the without VGs and flow over wing structure. Fig 7 to Fig 10 represents the fluid variables for with VGs design. Below figure shows the different variables contours for 0 deg AOA. Shock waves are captured accurately and the fluid behavior is changing due to compressible effect in leading edge of wing.

Flow patterns over wing

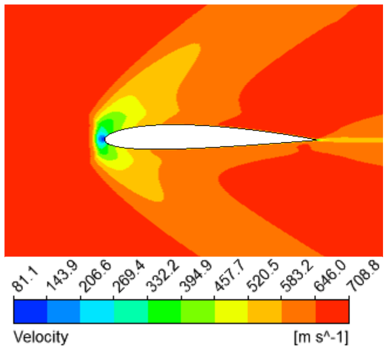


Fig.3 Velocity Contours over wing

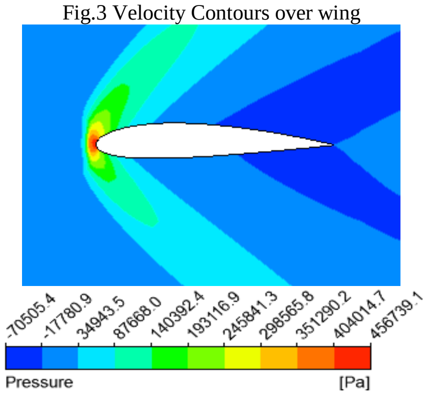


Fig.4 Pressure Contours

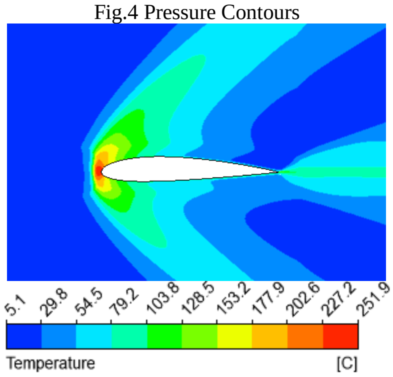


Fig.5 Temperature Contours

ISSN: 2278-0181
NCERAME - 2015 Conference Proceedings

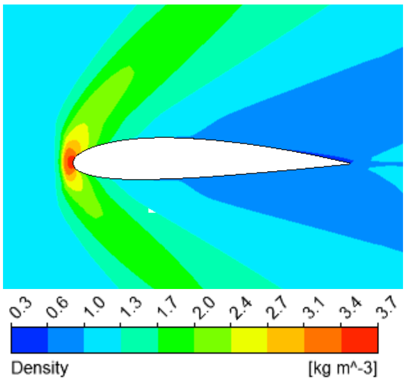


Fig.6 Density Contours

Flow patterns over wing with VGs

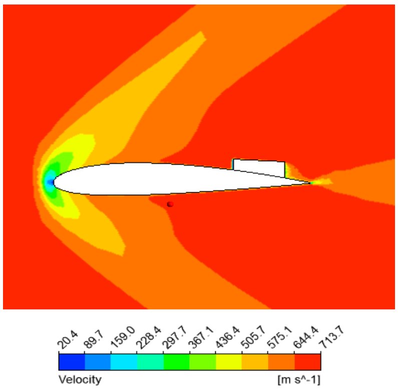


Fig.7 Velocity Contours over wing attached VGs

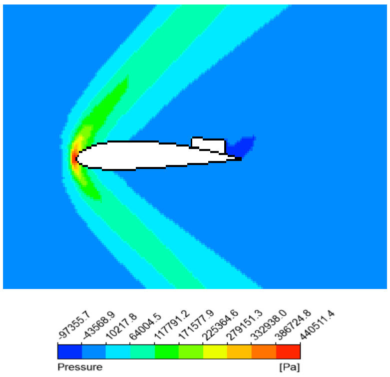


Fig.8 Pressure Contours

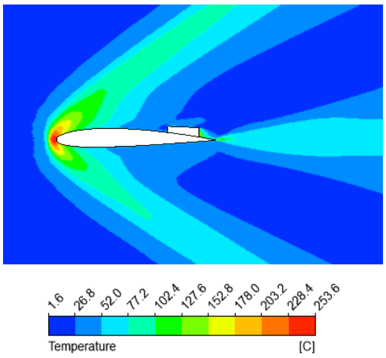


Fig.9 Temperature Contours

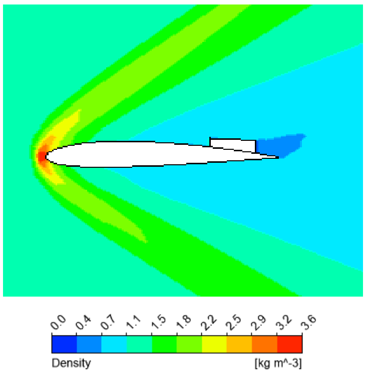
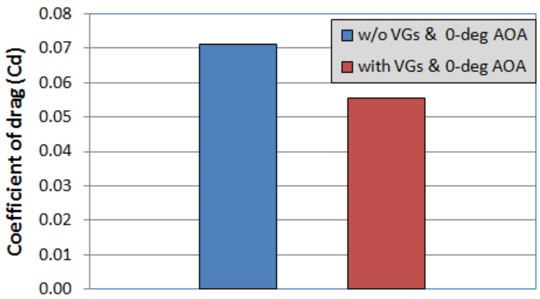


Fig.10 Density Contours



Coefficient of drag is more in case of without VGs are attached to the wing; Cd is reduced by 27% with VGs design. This could be very prominent devices to reduce the overall drag force which is result in significant reducing the fuel consumption for same loading conditions.

5. 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.