

INVESTIGATION OF DRAG REDUCTION ON AIRCRAFT WINGS THROUGH PASSIVE FLOW CONTROL IN A CFD STUDY

M. Santhosh Kumar*, D. Madesh**, K. Gurunath*** & V. Akilan****

Department of Aeronautical Engineering, Dhanalakshmi Srinivasan Engineering College, Perambalur, Tamil Nadu

Cite This Article: M. Santhosh Kumar, D. Madesh, K. Gurunath & V. Akilan, "Investigation of Drag Reduction on Aircraft Wings Through Passive Flow Control in a CFD Study", Indo American Journal of Multidisciplinary Research and Review, Volume 4, Issue 2, Page Number 47-54, 2020.

Copy Right: © IAJMRR Publication, 2020 (All Rights Reserved). This is an Open Access Article distributed under the Creative Commons Attribution License, which permits unrestricted use, distribution, and reproduction in any medium, provided the original work is properly cited.

Abstract:

The drag force is an important parameter which is to be reduced and optimized to improve the performance of an aircraft wing. Investigation of airflow field over aerofoil wing is very essential to understand the flow complexity and its behavior. The vortex generator models are used to study the flow separation control. The vortex generators which are highly efficient aerodynamic devices that are used widely in both external and internal aerodynamics as means of flow control will be used to understand the detailed flow physics of controlled flow separation. This project is focused on CFD simulation and investigation of subsonic flow separation over a NACA0012 aerofoil with angle of attack 6° and 8° flow separation control with vortex generators. The uncontrolled baseline case and controlled case with passive vortex generator. The drag and lift characteristics of each shape of vortex generators was studied. The analysis was carried out with free stream velocities of 6 m/s and 8 m/s for both designs of vortex generators. The complete three dimensional flow field results and physics were studied for the flow over wing equipped with two different vortex generator shapes. The best vortex generator was chosen based on C_l and C_d values. All simulations was carried out using the RANS modeling using the CFD solver FLUENT

Key Words: Aircraft Wing, Drag Reduction, Vortex, Computational Fluid Dynamics (CFD).

Introduction:

Many flow cases in fluid dynamics face undesirable flow separation due to rising static pressure on wall boundaries. This occurs e.g. due to geometry as in highly curved turbine inlet duct or e.g. on flow control surfaces such as wing trailing edge flaps within a certain angle of attack range. Here, flow control devices are often used in order to enhance the flow and delay or even totally eliminate flow separation. Flow control can e.g. be achieved by using passive or active vortex generators (VG) that enable momentum mixing in such flows. This work focuses on passive VGs, represented by VG vanes that are mounted upright on the surface in wall-bounded flows. They typically have an angle of incidence to the mean flow and, by that, generate vortex structures that in turn allow for the desired momentum mixing in order to prevent flow separation.

The demand to design more cost effective high-lift configurations for future aircrafts that in turn comes along with better low-speed behavior during take-off and landing, increased safety, and less environmental impact is becoming more and more important for the aircraft industry and its manufacturers. At the same time, reduction of the complexity of existing high-lift systems is not only a trend but a need in order to e.g. advance flight safety, reduce overall weight, lower fuel emissions, increase the operating distance, just to mention a few. In total, it can be stated that the high-lift system has a meaningful impact on the total performance of the aircraft, economically as well as ecologically. But not only aircraft industry faces such challenges, all other industries that are dealing with fluid dynamical processes are constantly in the need to improve their products in order to either be and remain competitive on the market and/or to fulfill legal issues like certification processes due to changing laws and being present on different markets throughout the world. Flow control is a very effective way for improving existing fluid dynamical systems and can be a powerful tool in the conceptual design process step from the very beginning of the development cycle. The term "flow control" in fluid dynamics is generally used when a flow of some liquid or gas is changed by so-called flow control devices (such as vortex generators, VGs) in a beneficial way that the overall efficiency of the system is increased. In borderline cases, it might even occur that a fluid dynamical system would not work properly without such flow control devices applied. Passive VGs mix the fluid near surfaces and force higher momentum fluid from the free stream towards the wall and vice versa. This increases the near-wall velocity as well as the momentum and consequently the stability of the flow in terms of separation delay and prevention. During the development phase when an increasing amount of computational simulations is used now a days, VGs generally create the disadvantage of rather being computationally costly and time-consuming when included in a detailed analysis. Computational grids often fully resolve such flow control devices, leading to an enormous amount of additional nodes in their vicinity. In turn, this causes high computational costs. Therefore, flow control devices are often neglected in computations. But later during an experimental evaluation these devices are then often included in e.g. wind or water tunnel investigations where the impact can be studied more easily. This is of course a disadvantage for the product design

process that would generally benefit from a more detailed performance investigation even from the beginning of the development cycle.

Different techniques to include VGs in computations are used: fully re-solved, partly resolved/modeled, and fully modeled approaches exist. Fully resolved VGs are completely embedded in the three-dimensional (3D) computational mesh, leading to very fine mesh sizes in their vicinity. Partly resolved/modeled approaches model the VGs by e.g. volume forces that are in turn added to the governing equations. Still, the generated structures need to be resolved and a higher mesh density is needed downstream of the VGs. Fully modeled approaches, like the one described in this thesis, also model the generated structures or the influence of the modeled VGs. This approach further reduces the mesh size compared to the partly resolved/modeled approach. For a long time, passive VGs have been used in order to control flows. Pearcey (1961) suggested certain design criteria for successful boundary layer flow control. His studies contained many different unsuccessful and successful designs as e.g. co-rotating and counter-rotating VGs, multiple row systems, and vane-type VGs of different geometries, see also figure 1.3. In particular, he found out that it was practical to set the VG height to wing chord ratio to $h_{VG}/c = 0.01$ when using counter-rotating set-ups on wings. In addition, other parameters like the distance between VG pairs, the length of the VG chord, the VG angle of incidence, and the distance between two VG blades were investigated.

This project work focuses on study of aerodynamic characteristics of a chosen wing with and without vortex generator. CFD simulation tool ANSYS fluent has been extensively used to predict lift and drag of the wing with and without vortex generators and for different angle of attack and mach number. The flow physics of two different vane type vortex generators has been studied and its effect on aerodynamic forces has been explained.

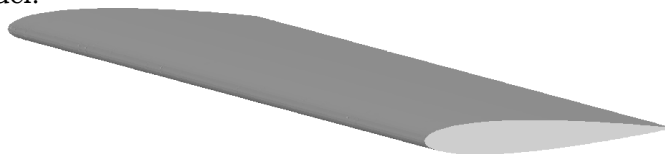
Methodology:

Methodology for CFD simulation for predicting the performance of vortex generator on NACA 0012 wing for drag reduction involves

- Modeling the NACA0012 wing using the CAD software CATIA.
- Pre-processing the modeled wing – CAD cleanup, domain creation and meshing.
- Applying boundary condition, solver setting and performing the CFD simulation for the modeled wing using FLUENT CFD solver.
- Monitoring the lift coefficient, drag coefficient for the base case of the wing for different angle of attack 6 and 8 degrees with two free stream velocities $V = 6 \text{ m/s}$ and $v = 8 \text{ m/s}$
- Post-processing the results.
- Modeling the wing with vortex generators locating on 20 % chord from leading edge on the wing's upper surface.
- Again CFD simulations are carried out to calculate the drag and lift acting on the wing.
- Post processing the results of the wing with vortex generators.
- Comparing the aerodynamic coefficients lift C_l and Drag C_d and comparing by visualizing the properties of air over the wing.
- Reporting the inference in the results

Model Construction:

Three-dimensional model of NACA0012 wing was designed using the CAD design software CATIA V5 and exported as STEP files. The STP or STEP file format is a third party file format most probably supported in all commercially available CFD tool. The IGS files were then imported into ANSA, the mesh generator. In ANSA, the imported geometry is checked for topology. Then the flow domain is created from the imported wing CAD model.

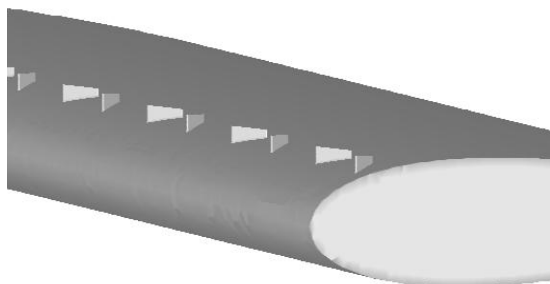


CFD Simulation of Modelled Wing:

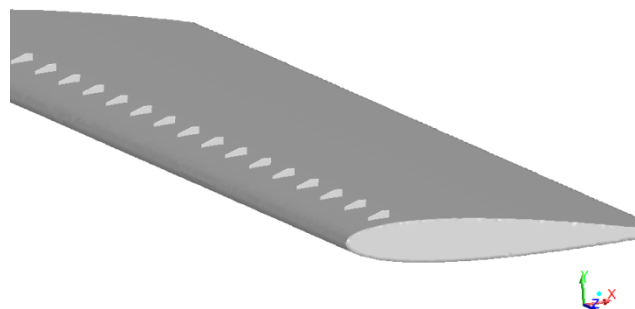
The CFD simulation to predict the effect of vortex generators external flow aerodynamic lift and drag prediction of the modeled NACA0012 wing has been carried out for two different AOA and two types of vortex generators. The main operating parameters chosen for simulation is the free stream velocity (v) of 6 m/s and 8 m/s are used for this simulation cases. At first the lift coefficient and drag coefficient of the wing was obtained from CFD simulation for the base case of the wing (without vortex generator). Then the simulations are carried out for the wing with vortex generator as shown in the CFD test simulation matrix table. CFD simulation test matrix for wing without and with vortex generators

Wing Model	Inlet Free Stream Velocity Condition (V)	Angle of attack, α (degree)	
NACA0012 Wing (Without Vortex Generators)	6 m/s	6	8
	8 m/s	6	8
NACA0012 Wing (With Vortex Generators)	6 m/s	6	8
	8 m/s	6	8

Then the wing has been modelled with two different configuration of vortex generator (VG) VG model-1 and VG model-2. The VG model-1 is a vane type vortex generator which has two vanes in which two vanes are positioned in 40 degree angle with a gap between each vanes. The VG model-1 has a trapezoidal shape with the maximum protrusion height of 0.8mm which is similar to the submerged type VG's used now a days.



Wing with VG Model-1



Wing with VG Model-2

The VG model-2 is a strip type with single strip of 20 No's on the wing. Then CFD simulations have been carried out for both the type of VG's for selected angle of attack and free stream velocities. The designed VG model-1 and VG model-2 is shown in the are shown in the figure.

Meshing:

Surface Mesh:

After repairing up the geometry, the surface mesh is generated in ANSA tool itself. All the surfaces are discretized using tri surface element. As the geometry has some skewed surfaces, tri surface elements are used to capture the geometry. The figure 2 shows the surface mesh of the wing with wind tunnel domain.

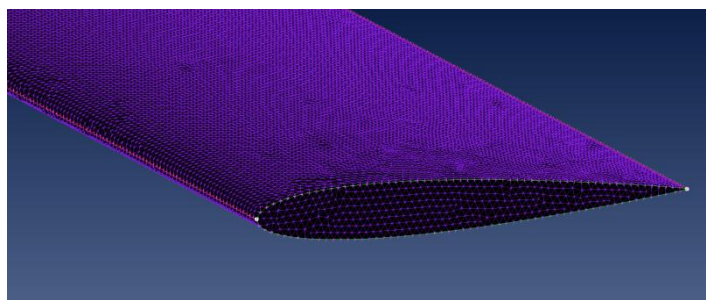


Figure 2: Surface mesh of the modeled wing (Base case)

Volume Meshing:

In general CFD domain may contain both static region and dynamic regions. The static regions are the zones where there wont be any moving part, thus meshing is much simple with tetrahedral elements. The dynamic regions in the CFD domain are the zones that has motion which might have sliding motion or linear motion. The volume mesh for the static zone of the created external domain wintunnel of the wing was generated using ANSYS-TGRID which is a robust volume mesh generator. The volume of the static zone of the extracted domain (Wing along with wind tunnel domain) is discretized using tetrahedral elements for interior of the tunnel and prism elements for the formation boundary layer. The prism elements are wedge shaped elements in which the higher order differential form of the navier-stokes system of equations are solved at each and every cell centroid of the elements for high accuracy results.

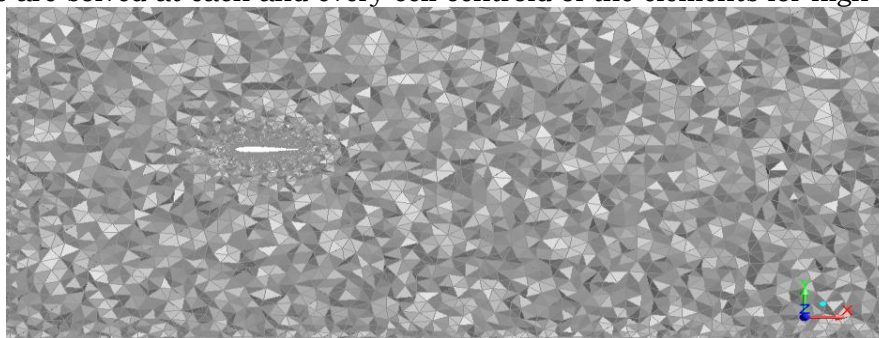


Figure 3: Volume mesh of wind tunnel region of the domain

The figure 3 shows the volume mesh of the static region i.e., surface of the wing and wind tunnel domain.

Since Fluent solver uses finite volume discretization scheme, the flow governing equations, turbulence equations and other calculations are carried out in the each and every node at cell centroid of the volume elements. Finer the mesh, accurate are the results.

Meshing Details:

The grid tests has been carried out for the modelled wing along with wind tunnel with increased mesh count in order to optimize the accuracy of the solution of the differential form of navier stokes equation. The mesh count after the grid study for the wing along the wind tunnel domain are shown in table

Table 1: Meshing details of the wing with wind tunnel domain

Surface Mesh Details	
Element Type	Triangular [TRI]
Maximum Quality	0.6
Surface mesh count	250760
Volume Mesh Details	
Element Type	Tetrahedral TET-for interior of wind tunnel
Maximum Quality	0.87
Volume mesh count	1750570

The volume mesh contains tetrahedron cells with the cell count of 175057

Solver Set Up Methodology:

ANSYS-FLUENT is used as the solver for this case.

- External flow over the modeled NACA0012 wing is assumed to be 3-D, turbulent and in-compressible in nature.
- Steady simulation has been chosen to predict the drag for the base case of the wing(without vortex generators)
- Pressure based solver from the solver options has been chosen.
- K-Omega SST (2 Equation) RANS turbulence model has been chosen to model turbulence of the free stream air flow over the wing.
- Segregated solver is used for pressure-velocity coupling.
- SIMPLE (Semi Implicit Pressure Linked Equations) algorithm is used to solve the problem.

The Boundary conditions for this case are

- Boundary condition at Inlet - Velocity inlet boundary condition is set with the value of 6 m/s and 6 m/s for the required cases.
- Boundary condition of wing - Wall boundary condition with no slip condition.
- Boundary condition of tunnel wall - symmetry
- Pressure outlet boundary condition is to be pressure outlet with 0 Pascal(static pressure).

Results and Discussion:

CFD simulation of flow over the modeled wing with and without vortex generators on the wing in this work has been done as three stages.

- CFD analysis of the base case of the modeled wing has been carried with the operating condition as free stream velocity 6 m/s and 8 m/s. Drag, lift coefficients and has been obtained during the simulation for the respective angle of attacks 6 degree and 8 degree.

- Then, CFD analysis has been carried out for the wing with vortex generator case. This simulation has been carried out to predict the change of drag and lift values because of vortex generators VG model-1 and VG model-2 placed on 20% chord of the wing.

- The lift and drag coefficients obtained from CFD simulation are tabulated in the table 2 and 3

Table 2: Lift Coefficient for base model of the wing

S.No	Case	Angle of attack (α) degree	Predicted Cl value for each free stream velocity (v)	
			6 m/s	8 m/s
1	Base case of the wing (without vortex generator)	6	0.151	0.17
		8	0.16	0.1828

Table 3: Drag Coefficient for base model of the wing

S.No	Case	Angle of attack (α) degree	Predicted Cd value for each free stream velocity (v)	
			6 m/s	8 m/s
1	Base case of the wing (without vortex generator)	6	0.021	0.0241
		8	0.023	0.026

CFD results for base case of the wing (without vortex generator)

Case for wing at angle of attack = 6 Degree

For velocity v = 6 m/s

CFD Predicted Cl for this case is 0.151

CFD Predicted Cd for this case is 0.021

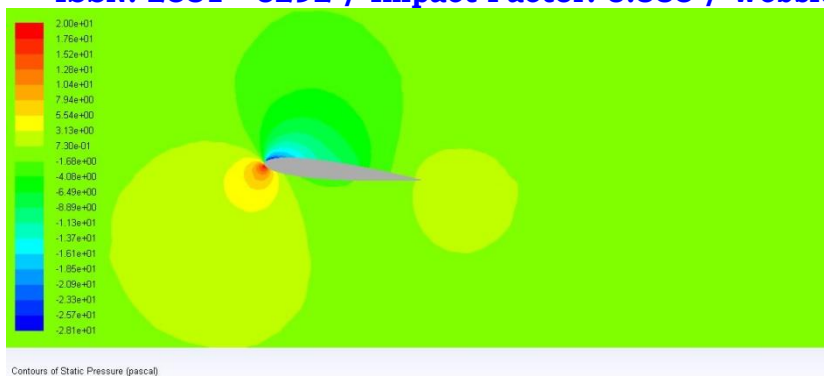


Figure 4: Static pressure contour of flow over wing at AOA = 6 and $V = 6$ m/s

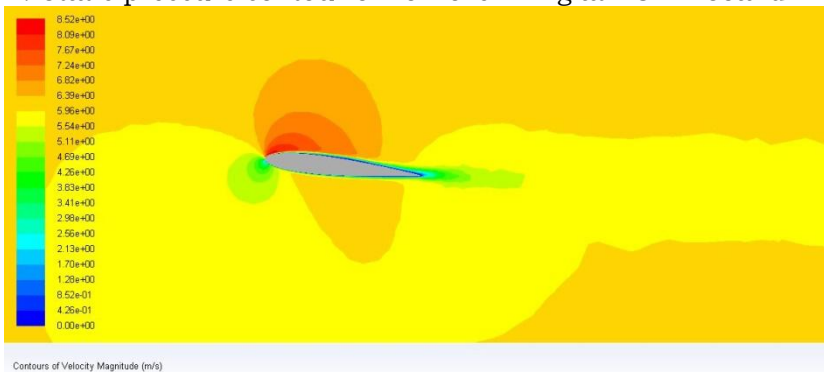


Figure 5: Velocity contour of flow over wing at AOA = 6 and $V = 6$ m/s

For velocity $v = 8$ m/s

CFD Predicted C_l for this case is 0.16.

CFD Predicted C_d for this case is 0.023.

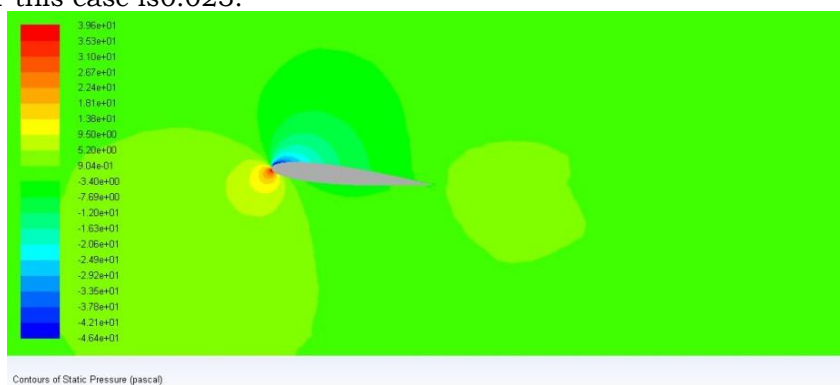


Figure 6: Static pressure contour of flow over wing at AOA = 6 and $V = 8$ m/s

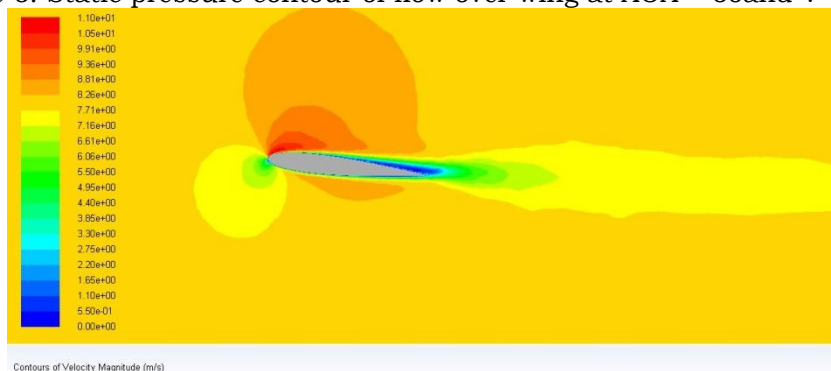


Figure 7: Velocity contour of flow over wing at AOA = 6 and $V = 8$ m/s

The figure 4 to 7 shows the pressure contour & velocity contour base case of the wing with angle of attack 6 degree for the velocities 6 m/s and 8 m/s. We can clearly see the flow separation on the wing as increases angle of attack increases due to inertial dominance and adverse pressure gradient on the wing.

The C_l/C_d ratio of 7.1904 was seen for the case with velocity 6 m/s and C_l/C_d ratio of 6.956 was seen for the case with velocity 8 m/s. C_l/C_d ratio is an important parameter in the aerodynamic analysis that for the given inlet velocity conditions it should higher value. It can be clearly seen for the C_l/C_d values of both 6 m/s and 8 m/s, there is a decrease in the value. This is due to flow separation on the wing because of higher adverse pressure gradient for the increased velocity.

Case for wing at angle of attack = 8 Degree

For velocity $v = 6$ m/s

CFD Predicted C_l for this case is 0.17

CFD Predicted Cd for this case is 0.0241

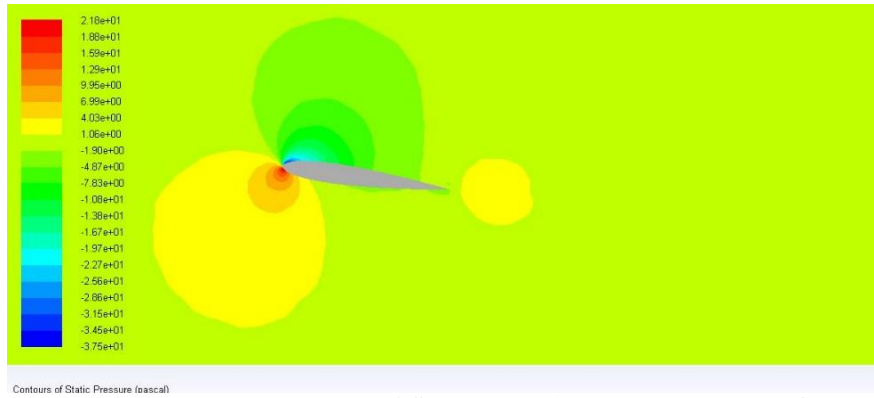


Figure 8: Static pressure contour of flow over wing at AOA = 8 and V = 6 m/s

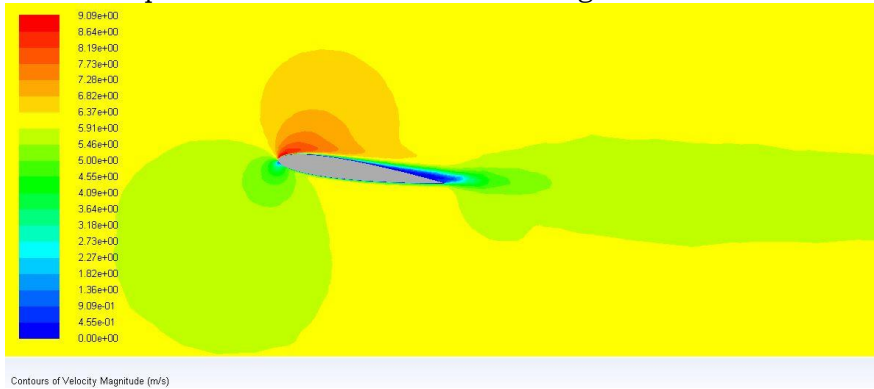


Figure 9: Velocity contour of flow over wing at AOA = 8 and V = 6 m/s

The laminar separation bubbles of flow from suction side of the wing can be seen where there is decrease in velocity and increase in pressure (adverse pressure) in between 40 to 50 percent of the chord from leading edge.

The surface pressure distributions will be greatly affected with high pressure values instead of low suction pressure thus causing low pressure wake region with eddies. Thus further reattachment was not seen and it has have been visualized.

For velocity $v = 8$ m/s

CFD Predicted Cl for this case is 0.1828

CFD Predicted Cd for this case is 0.026

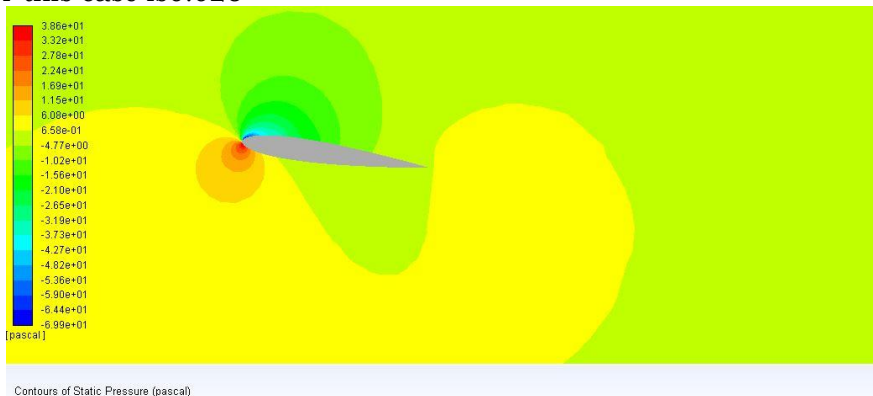


Figure 10: Static pressure contour of flow over wing at AOA = 8 and V = 8 m/s

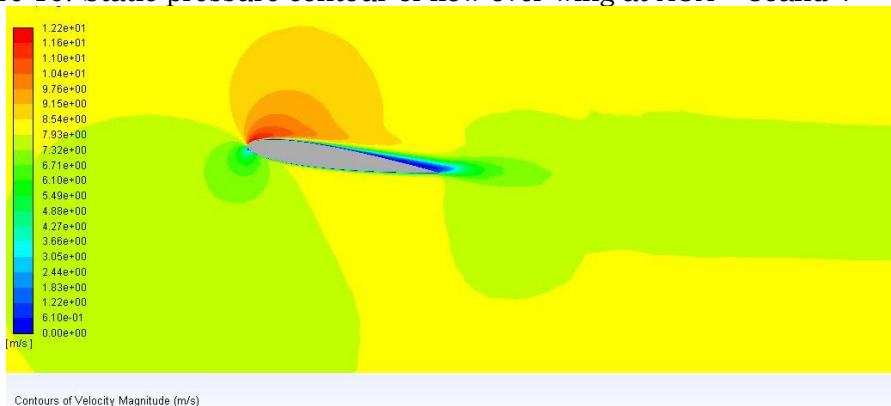


Figure 11: Velocity contour of flow over wing at AOA = 8 and V = 8 m/s

The figure 8 to 11 shows the pressure contour & velocity contour for the base case of the wing with angle of attack 8 degree for the velocities 6 m/s and 8 m/s. The contours clearly shows the flow cases with both the angle of attacks has earlier separation.

The C_l/C_d ratio of 7.0539 was seen for the case with velocity 6 m/s and C_l/C_d ratio of 7.030769 was seen for the case with velocity 8 m/s. It can be clearly seen from the above mentioned C_l/C_d values of both 6 m/s and 8 m/s, there is a decrease in the C_l/C_d .

Simulation Case with Vortex Generator on the Wing:

For VG model-1

For AOA = 6 Degree, velocity $v = 6$ m/s

CFD Predicted C_l for this case is 0.12918155

CFD Predicted C_d for this case is 0.019263811

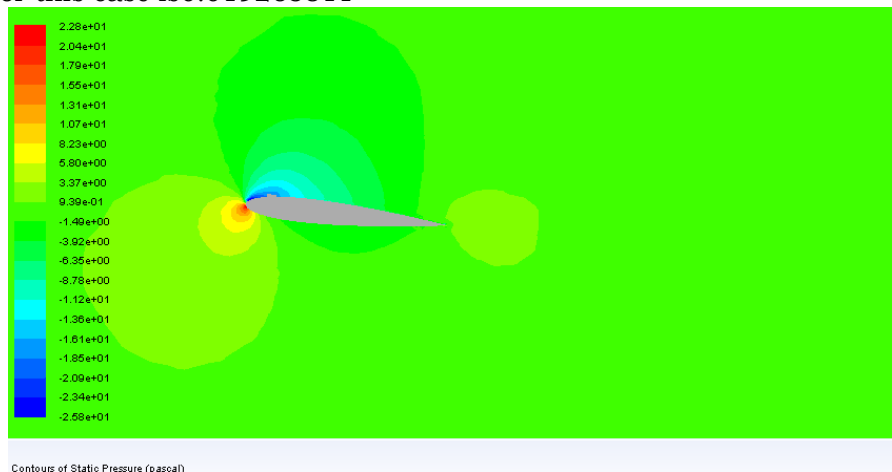


Figure 12: Static pressure contour of flow over wing with VG-1 at AOA = 6° and $V = 6$ m/s

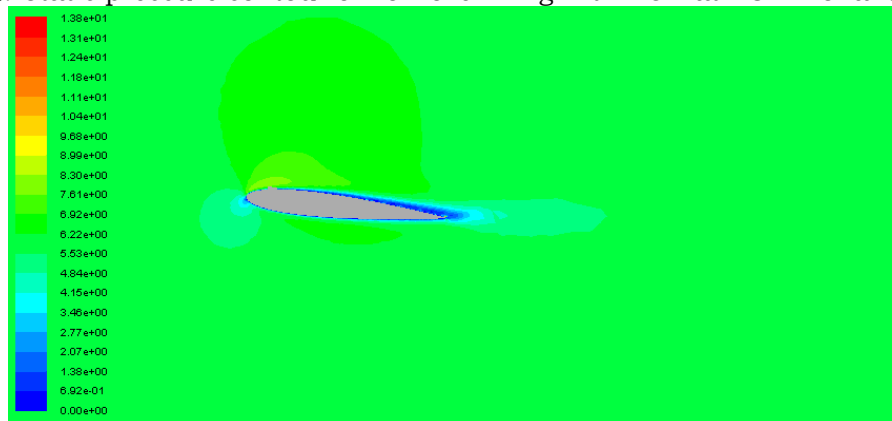


Figure 13: Velocity contour of flow over wing with VG-1 at AOA = 6° and $v = 6$ m/s

The C_l/C_d ratio of 7.9423 was achieved for this simulation case of wing with VG Model-1.. When comparing this with the C_l/C_d ratio of base case which has the C_l/C_d value of 7.1904, there is 9.46703 percent increase in lift to drag ratio.

For AOA = 6 Degree, velocity $v = 8$ m/s

CFD Predicted C_l for this case is 0.12126

CFD Predicted C_d for this case is 0.015273234

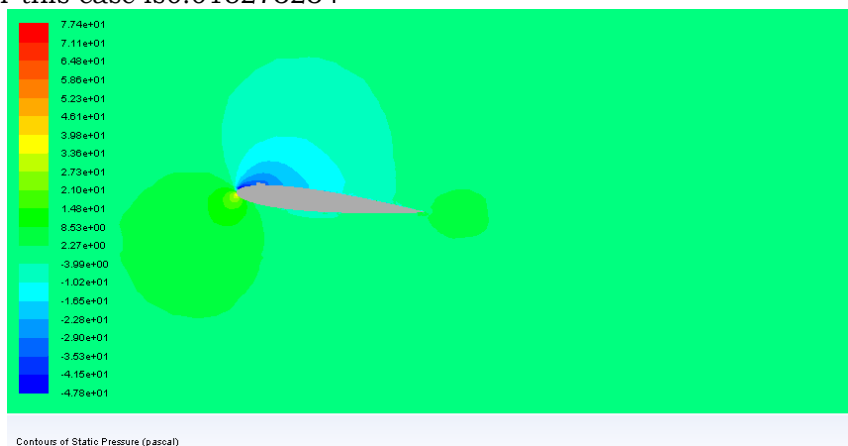


Figure 14: Static pressure contour of flow over wing with VG-1 at AOA = 6° and $V = 8$ m/s

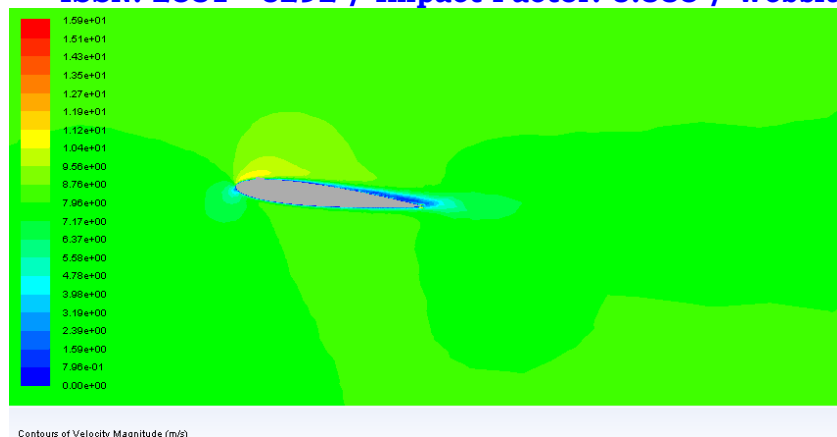


Figure 15: Velocity contour of flow over wing with VG-1 at AOA = 6° and V = 8 m/s

From the above results it can be clearly understood that at high speed of 8 m/s, the effect on wing with increment in lift was seen and whereas at low speeds decrease in performance because of lesser vortices

Conclusion:

The Passive flow control over a NACA0012 aerofoil using vortex generators is studied by performing numerical simulations. The purpose of this investigation deals with the control of boundary layer separation of NACA0012 wing positioned at angle of attack 6 degree and 8 degree. The study focuses on to visualize the vortex generators performance for counter-rotating vortex generator VG Model-1 and single strip type vortex generator VG Model-2 configuration. The simulation results are studied in the aspect of lift to drag ratio. The lift-to-drag ratio gives relevant measure about the impact of VG's on aerodynamic performance. From simulation results, it has been observed that, at low AOA and low speed there is improvement in CL/Cd ratio due to delayed flow separation for wing with VG Model-1. These results indicate that an increase in vortex intensity generates a flow in their surroundings similar to that produced by a perturbation, disturbing the boundary layer in such a way that increases the Co/Cd ratio. The results of VG Model-2 it is understood that at high speed of 8 m/s, the effect on wing with increment in lift was seen and whereas at low speeds there is a decrease in performance because of lesser vortex intensity over the wing surface.

References:

1. Deepanshu Srivastav. Flow Control over Airfoils using Different Shaped Dimples. 2012 International Conference on Fluid Dynamics and Thermodynamics Technologies (FDTT 2012) IPCSIT Vol. 33 (2012) IACSIT Press, Singapore.
2. Leticia Hime, César Celis Pérez, Luís Fernando Figueira da Silva. A review of the characteristics of submerged air intakes. 18th International Congress of Mechanical Engineering November 6-11, 2005, Ouro Preto, MG.
3. Mohan Jagadeesh Kumar M. Effect of Vortex generators on Aerodynamics of a Car: CFD Analysis. International Journal of Renovations in Engineering & Technology. Vol 2 Issue 1 April 2013
4. Raju Govindharajan, K.N. Parammasivam, S. Sathyanarayanan. Design of vortex generators for light transport vehicles (ltvs) using CFD. The Eighth Asia-Pacific Conference on Wind Engineering, December 10-14, 2013, Chennai, India.
5. Zare Shahneh. Computational analysis of tetrahedral vortex generator effect on the attenuation of shock induced separation. International Journal of Applied Science and Technology (IJAST). Vol. 2 No. 3; March 2012