



e-ISSN: 2319-8753 | p-ISSN: 2320-6710

IJRSET

International Journal of Innovative Research in
SCIENCE | ENGINEERING | TECHNOLOGY

INTERNATIONAL JOURNAL OF INNOVATIVE RESEARCH

IN SCIENCE | ENGINEERING | TECHNOLOGY

Volume 10, Issue 3, March 2021

ISSN INTERNATIONAL
STANDARD
SERIAL
NUMBER
INDIA

Impact Factor: 7.512

9940 572 462

6381 907 438

ijirset@gmail.com

www.ijirset.com



Computational Analysis of Vortex Generator on NACA 63210 Airfoil.

Shyam Rathod¹, Vatsal Acharya², Srujan Soma³, Divyesh Soni⁴, Rajeevkumar Baghel⁵

U.G. Student, Department of Aeronautical Engineering, Parul University, Vadodara, Gujarat, India¹

U.G. Student, Department of Aeronautical Engineering, Parul University, Vadodara, Gujarat, India²

U.G. Student, Department of Aeronautical Engineering, Parul University, Vadodara, Gujarat, India³

U.G. Student, Department of Aeronautical Engineering, Parul University, Vadodara, Gujarat, India⁴

Assistant Professor, Department of Aeronautical Engineering, Parul University, Vadodara, Gujarat, India⁵

ABSTRACT: This project was aimed at computational analysis of vortex generator on wing of NACA 63210 aerofoil. A vortex generator is designed using software package CATIA V5R20 and wing is designed in solid works 2020. Wing is investigated using simulation software Ansys (fluent) 17.1. In this project study of flow behaviour around vortex generator and its optimum location (x/c) from leading edge will be analysed to get maximum aerodynamic efficiency. Analysis result of wing with vortex generator and without vortex generator will get compared and will find at which optimum location it is delaying flow separation the most.

KEYWORDS: Vortex generator, Flow separation, CFD, optimum location, Turbulent flow.

I. INTRODUCTION

❖ What is flow separation?

At high angle of attack, it causes the boundary layer over airfoil to detach due to increase in adverse pressure gradient. Adverse pressure gradient means flow is forced to move from low pressure region to high pressure region. Thus, detachment of boundary layer causes wake region. Because of wake region flow over airfoil moves in reverse direction called reverse flow. This phenomenon is called stalling of airfoil/wing. The flow separation phenomenon causes increase in pressure drag which is not economical for any airplane. Thus, in order to make flow stay attached to airfoil surface a passive device called vortex generators is used.

❖ Vortex generator:

By experimentally it is seen that a turbulent boundary layer can resist more adverse pressure gradient than laminar boundary layer. Because of which turbulent boundary layer stays attached to airfoil even high adverse pressure gradient is present. So, if a turbulent boundary layer is used over airfoil instead laminar boundary layer the flow separation on an airfoil can be delayed. This is advantage for any airplane. So, if somehow a turbulent flow is produced over airfoil then the flow separation can be delayed. Thus, vortex generators are used over airfoil to produce turbulent boundary layer.

A vortex generator creates a vortex which is having high Reynolds number (more turbulent flow). Thus, with the help of vortex generator a turbulent boundary layer gets produced. Moreover vortex generators often use over cars, wind turbine blades etc.

❖ Problem statement

Flow separation is very undesirable situation for any aircraft. So vortex generators are placed on wing which can effectively delays the flow separation.

Existing researches on vortex generator are mostly done on dimensions of vortex generator but this project showcase, at which optimum location from leading edge in chord wise direction a vortex generator delays separation the most.

II. CFD METHODOLOGY

CFD methodology has been performed in three parts.

Pre-processing:

[i] CAD model:

We have designed rectangular wing of NACA 63210 airfoil in Solidworks 2020 and other most of parts in CATIA V5R20.



Figure:1- Wing design in Solidworks 2020

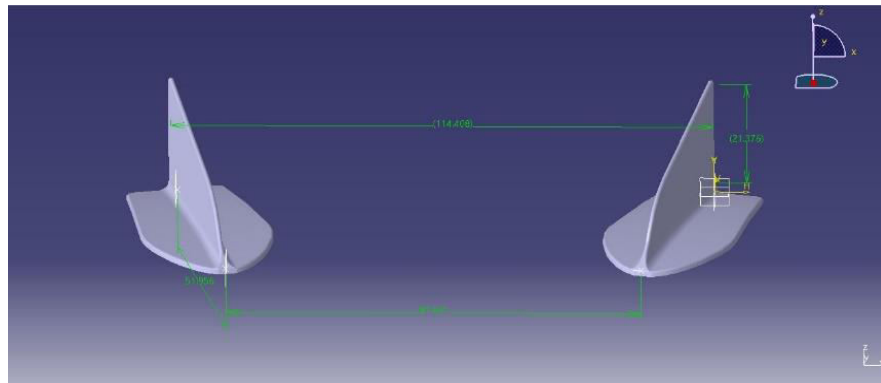


Figure:2-Vortex generator design in CATIA V5R20

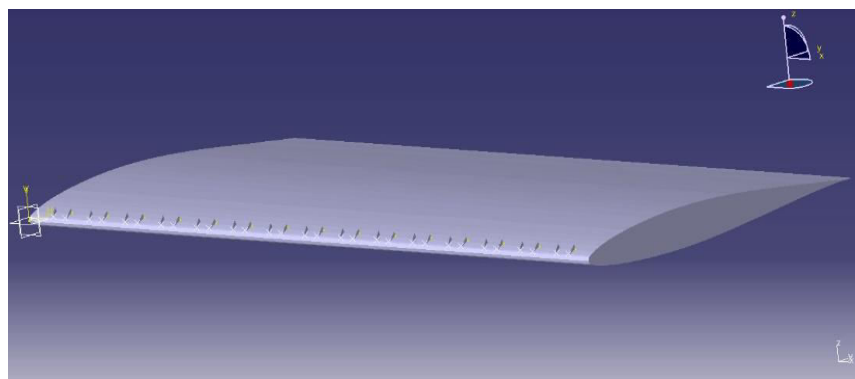


Figure-3: Vortex generator mounted on wing designed in CATIA V5R20

[ii] Fluid domain:

It is a space portion where the solution of CFD simulation is calculated.

In this project shape of domain is Rectangular. To analyse the downstream flow, downstream area of domain is 10x of wing chord. Width of the domain is 2x of wing span to analyse the wing tip vortices.

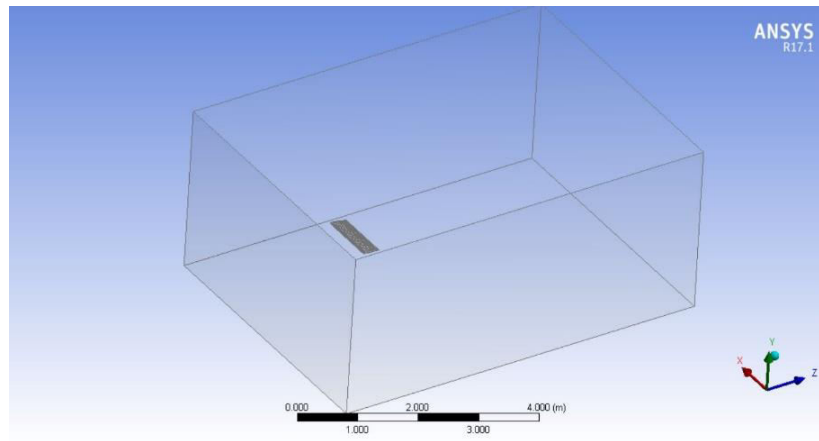


Figure:4- CFD Simulation

[iii] Meshing:

Mesh is the process of breaking up a physical domain into smaller domain/element/control volume, in order discretises partial differential equations into algebraic form we need finite number of points which are called as nodes. The process which carries out nodes on domain is called Meshing process.

❖ Type of mesh:

- a) Structured mesh:
A structured mesh is characterized by uniform patterned connectivity of nodes.
2D element: Quadrilaterals
3D element: Hexahedra
- b) Unstructured mesh:
An Unstructured mesh is characterized by non uniform patterned connectivity of nodes.
2D element: Triangular
3D element: Tetrahedral
- c) Hybrid mesh:
A hybrid mesh is a mesh that contains structured and unstructured elements.

❖ Advantages of Unstructured mesh:

- Unstructured mesh is best to use for discretising complex geometries
- For complex geometries it gives more accurate result than structured mesh
- Less effort have required to generate mesh than structured mesh
- Less space is required to save the unstructured mesh than structured mesh
- Good for smoothing and capturing sharp edges.



❖ Meshing over wing:

Looking over these advantages for this particular project Unstructured tetrahedral mesh has been used. The meshing is done in Ansys fluent itself. Below image shows meshing over wing at zero-degree angle of attack.

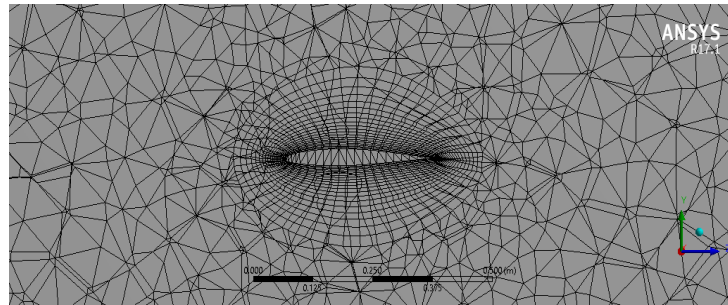


Figure:5-Meshing

Meshing element sizing:

Mesh type	Tetrahedral
Nodes	258011
Elements	1332805
Minimum element size	4.4899e-003m
Maximum face size	7.0e-0.002m
Maximum test size	9.5e-0.002m

Face sizing:

To refine mesh more around the wing face sizing using mesh control. Below table shows the sizing of elements of face sizings.

Element size	3.5e-0.002m
Behaviour	Soft
Local minimum size	4.4899e-0.003m

Inflation layer:

To capture the boundary layer around wing an inflation layer is pasted around wing. Below table shows the sizing of elements in inflation.

Boundary	Wing
Transition ratio	0.5
Maximum layers	20

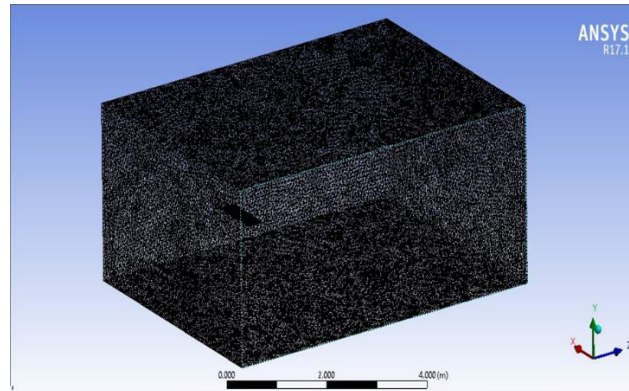


Figure:6- Boundary layer

Solver:

Second step of the CFD is solver. Often referred to as setup or physics setup in Ansys fluent. For this project pressure-based solver is used with steady time.

❖ Models:

K-epsilon ($k-\epsilon$) turbulence model is the most common model used in computational fluid dynamics (CFD) to simulate mean flow characteristics for turbulent flow conditions. It is two equation model of partial differential equation. First one is turbulent kinetic energy (k), and second is the rate of dissipation of turbulent kinetic energy (ϵ). For this project K-epsilon ($k-\epsilon$) turbulence model is used.

Boundary conditions:

Model	K-E Turbulence Model
Velocity At Inlet Of Domain	5 M/S In Z Direction
Pressure At Inlet Of Domain	Atmospheric
Boundary Wall	Stationary Wall
Density	1.225 (Constant)
Pressure At Outlet Of Domain	0 Gauge Pascal
Viscosity	1.7894e-05 (Constant)
Reference Temperature	298.15k

❖ Iterations and solution converge:

The standard solution initialization method is being used prior to starting calculation activity. The solution is initialized from the inlet of the domain.

To analyse wing without vortex generator total 150 iterations has been applied. And iterations are completely converged.

Post-processing:

The third step of the CFD analysis is post processing. After the iterations gets converged, we have to check what result we have obtained. Ansys post processing is a powerful tool provide additional reporting options in a graphical representation and Post processing tool also has graph plotting, animation capabilities which used to solve complex engineering problems.

Graphical representation of post processing includes streamlines, vectors, contours, path lines, volume rendering, animations etc.

For this project pressure contour, velocity contour, streamline, vector is plotted and their images are plotted below.

❖ Pressure contour:

A completely stalled wing pressure contour has been plotted which is at angle of attack 14-degree A plane intersecting mid span of the wing is made and on that plane pressure contour is obtained. A level for the representation is set to 110.the minimum pressure is -43.2641 Pa obtained at upper surface and maximum pressure is 14.4356 Pa obtained at lower surface near leading edge.

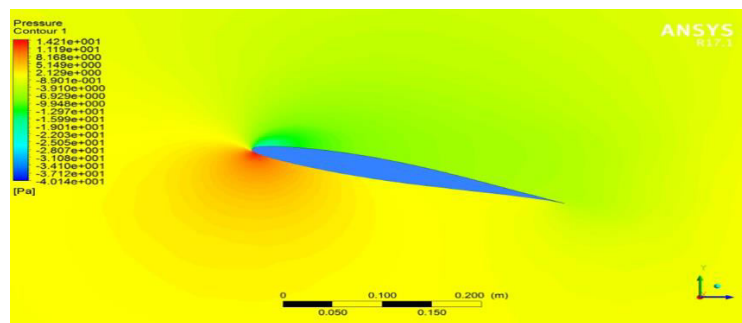


Figure:7- Pressure contour

❖ Velocity contour:

A completely stalled wing velocity contour has been plotted which is at angle of attack 14 degree. Minimum velocity is 0 m/s which represents stagnation point and maximum velocity is 7.6677 m/s.

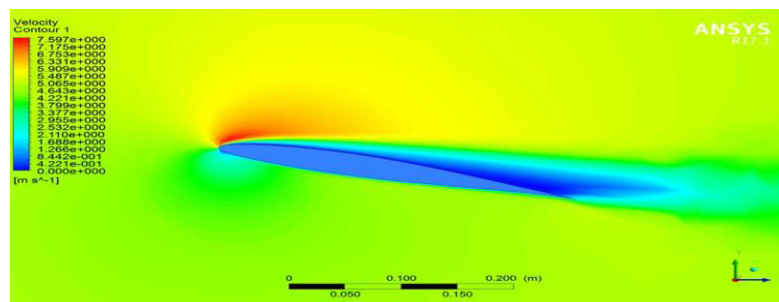


Figure:8- Velocity contour

❖ Streamlines:

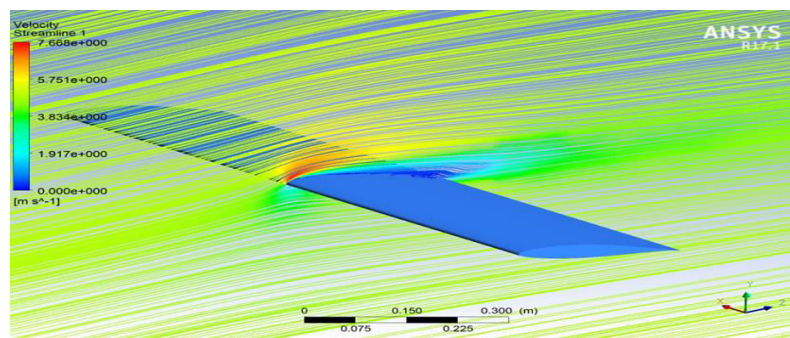


Figure:9- Streamline

❖ Lift coefficient vs angle of attack:

A Matlab is having very good for graph plotting capabilities. So a Matlab code is made to plot a graph between lift coefficient and all angle of attack.

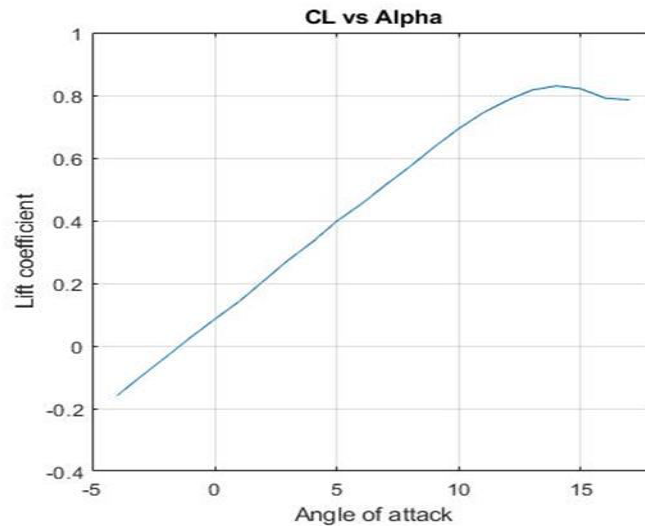


Figure:10- C_L versus alpha

From this plotted graph it is seen that wing stalls at angle of attack 14 degree. Furthermore, it is seen that C_L remains linear from zero lift α till stalling α after that C_L drops.

III. CONCLUSION

By the simulation of wing of NACA 63210 airfoil without vortex generator it is observed that from zero lift angle of attack up to stalling angle of attack, wing stalls near 14° angle of attack and boundary layer separation start from trailing edge at 12° angle of attack.

REFERENCES

- [1] Fundamental of aerodynamics: J D Anderson;
- [2] Aerodynamic drag reduction by vortex generator: Md. Rasedulislam&etal.
- [3] Introduction to computational fluid dynamics : H K Versteeg & W Malalasekera;
- [4] Optimized vortex generator in flow separation control around NACA 0015 profile: H. Tebbiche and etal.
- [5] Tec-science.com
- [6] Effects of vortex generators on a blunt trailing-edge airfoil for wind turbines
- [7] Experimental parameter study for passive vortex generators on a 30% thick airfoil



INNO  **SPACE**
SJIF Scientific Journal Impact Factor

**Impact Factor:
7.512**

ISSN INTERNATIONAL
STANDARD
SERIAL
NUMBER
INDIA



INTERNATIONAL JOURNAL OF INNOVATIVE RESEARCH

IN SCIENCE | ENGINEERING | TECHNOLOGY

 **9940 572 462**  **6381 907 438**  **ijirset@gmail.com**



www.ijirset.com

Scan to save the contact details