International Journal of Trend in Scientific Research and Development (IJTSRD)
Volume 5 Issue 4, May-June 2021 Available Online: www.ijtsrd.com e-ISSN: 2456 – 6470

# Computational Investigation on the Effect of Fences on Aerodynamic Characteristics of an Aircraft Wing

**Mohamed Hamdan. A**

Assistant Professor, Aeronautical Engineering, Dhanalakshmi Srinivasan Engineering College (Autonomous), Perambalur, Tamil Nadu, India

**ABSTRACT**

A computational study, to improve the stall characteristics of wing at high angles of attack, with and without fence configuration is done here. Wing fences, also known as boundary layer fences and potential fences are fixed aerodynamic devices attached to aircraft in the exactly middle of the span and two fences are fixed in the exactly middle of the wing span and two fences are fixed at 25 percentage of wing span from their corresponding wing tip. The main aim of this research is to improve the lift and stalling angle. A rectangular wing with different angles of attack is used. Modeling was done in CATIA V5 R20 and meshing and analyzing was taken in ANSYS workbench and CFX. Then the graph is drawn for $C_L$ and $C_D$ for various angles of attack and various models.

***KEYWORDS:*** *Stall, Lift, Fence and Configuration*

***How to cite this paper:*** Mohamed Hamdan. A "Computational Investigation on the Effect of Fences on Aerodynamic Characteristics of an Aircraft Wing" Published in International Journal of Trend in Scientific Research and Development (ijtsrd), ISSN: 2456-6470, Volume-5 | Issue-4, June 2021, pp.1605-1610, URL: www.ijtsrd.com/papers/ijtsrd43687.pdf

**IJTSRD43687**



## 1. INTRODUCTION

Aerodynamic performance enhancement plays an important role in improving flight performance. Even a slight improvement. Aerodynamic performance results in a huge amount of fuel savings. Improving aerodynamic performance of a vehicle mostly focuses on reducing the drag as much as possible. The study focuses on minimizing induced drag by applying wing fence.

A wing model with 45-degree sweep angle location with different height, at 10-degree angle of attack. From CFD analysis, it is observed that fence with height 2.5% of the root chord is effective on the upper surface, where fence with height 7.5% of root chord is effective on the lower surface. Particularly, for the selected model, fence with height 2.5% of root chord is very effective at 0.7 times to the span length location from the root on upper surface and fence with a height 7.5% of root chord very effective at the wing tip on the lower surface. If these two are employed together, a very good performance increment can be expected. Usage of wing fences also creates structural problems and it cannot be recommended specifically at a particular location for all the cases. Depending upon the geometry of a wing and flow condition, its fence location, size,

The wing is considered as the most important component of an aircraft, since a fixed-wing aircraft is not able to fly without it. The primary function of the wing is to generate sufficient lift force (L). While a wing designer is looking to maximize the lift, the other two drag force and pitching moment must be minimized. In fact, wing is assumed as a lifting surface that produces lift due to the pressure difference between lower and upper surfaces.

Stall fences are used in swept wings to prevent the boundary layer drifting outboard toward the wing tips. Boundary layers on swept wings tend to drift because of the span wise pressure gradient of a swept wing. Swept wing often have a leading edges fence of some sort, usually at about 35 percent of the span from fuselage centerline. The cross-flow creates a side lift on the fence that produces a strong trailing vortex. This vortex is carried over the top surface of the wing, mixing fresh air into the boundary layer and sweeping the boundary layer off the wing and into the outside flow. The result is a reduction in the amount of boundary layer air flowing outboard at the rear of the wing. This improves the outer panel maximum lift coefficient.

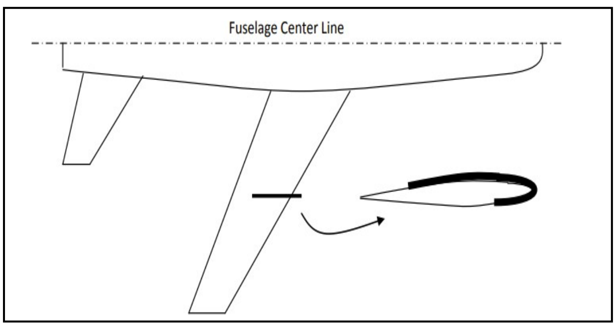



**Figure 1.1 Wing with fences**

## 2. STALL

The conventional stall is generally defined as a sudden loss in lift at an AOA just above that of maximum lift coefficient. However, for aircraft without a true maximum lift coefficient, it is better to consider the following definition of stall speed. Stall speed is the minimum steady speed attainable or usable in flight. However, it has become increasingly common to define this based on other characteristics such as a high sink rate, an undesirable attitude, loss of control about any axis, or deterioration in handling qualities. Stall is normally associated with flow separation that has occurred over large portions of a lifting surface. The results of stall are a decrease in lift, increase in pressure drag, and a change in pitching moment. The type of boundary layer has a significant impact on stall. Because flow separation begins at the boundary layer, higher velocity gradients associated with turbulent boundary layers better resist separation. This ultimately allows the flow to remain attached to the surface longer, thus delaying stall.

## 3. SWEPT WING WITH FENCES

Fence also works as a vortex Fences generator. In principle, vortex generators are used to delay separation. They are normally small and shaped like an airfoil or thin plane which protrudes from the surface. They are positioned at an angle to provide vortex generation. The key is that the vortex captures energy from the free stream and transfers it to the boundary layer, and this helps to delay separation.

The location, length, height, and shape of the fence are significant variables that must be adjusted dependent upon specific aircraft attributes. Some sources suggested blanket guidelines. Among the design guidelines were suggestions that extending the fence beyond one-third of the local chord does not significantly increase its effectiveness. Another was that they are more effective when they wrap around the leading edge. The most common span-wise location for wing fences is between 40 percent and 60 percent of the wing span.

The most outboard span wise location of a fence found through research was 76 percent semi span. Additionally, fences need to be much taller than the boundary layer to be effective. The main objective of this computational study is to reduce the induced secondary flow (Span-wise flow) of a swept backward wing by applying wing fences which results in a better lift generation, reduced induced drag and improved stall characteristic.

Installing a wing fence changes the lift distribution on a swept back wing as depicted in Figure 1.2. On the inside of the fence, the local lift per unit span is higher. On the outside of the fence, lift per unit span is lower. This shift in load is usually beneficial to stall behavior. Generally, the load is reduced on the wing tip and the boundary layer is maintained in such a way that separation is inhibited.

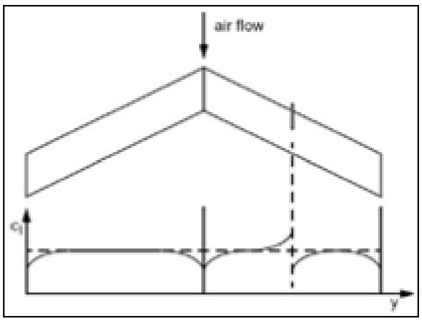


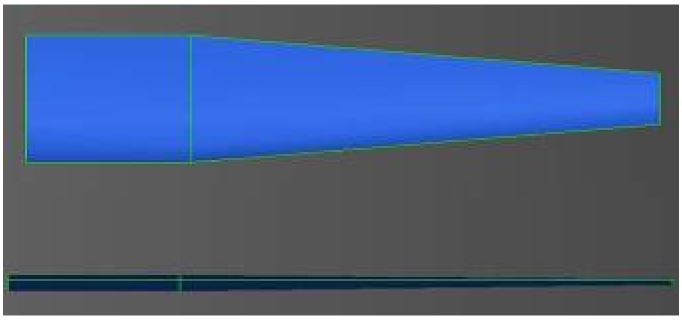

**Figure 3.1. Swept Wing with Fence - Lift Distribution**

## 4. CFD METHODOLOGY

The methodology includes the CAD Modeling, Meshing, Boundary conditions & Solver set up, Solution progress and post processing. Three different configurations of fences are considered for the present numerical simulation.

### 4.1. CAD Modelling

The CAD model of the wing configuration is shown in the figure 3.1. The top and front view of the overall wing configuration is shown figure 4.1. The overall length of the wing considered for the analysis is 10.9 m and the chord length is taken as 3 m. NACA 0012 is considered for the present investigation. The taper ratio is considered as 0.27.



**4.1 CAD Model of the wing considered for present CFD analysis**

Figure 4.2 shows the profile of NACA series considered for the computational analysis. Three different lengths of the fences are considered with 25%, 50% and 75 % of the chord length. There are three fences positioned over the surface of the wing as shown in figure 3.3, 3.4 and 3.5.

**Figure 4.2 Aerofoil NACA 0012 – Considered for Wing Generation**

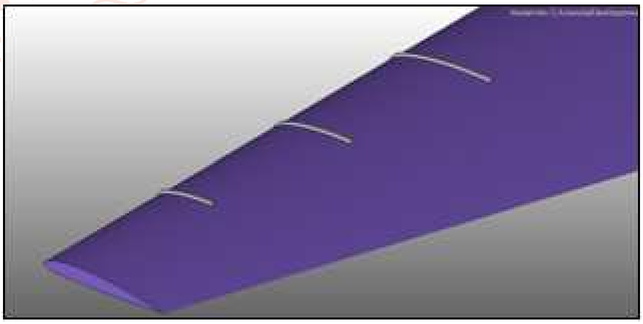


**Figure 4.3 Wing with Fence Configuration 1: 25%length**

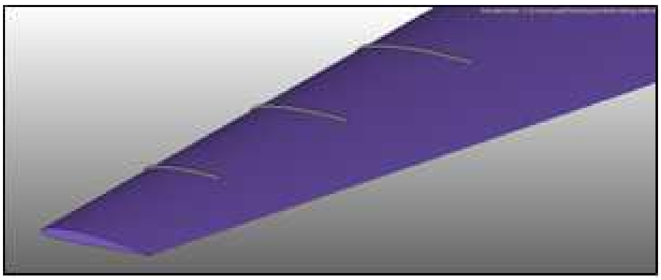


**Figure 4.4 Wing with Fence Configuration2: 50% length**

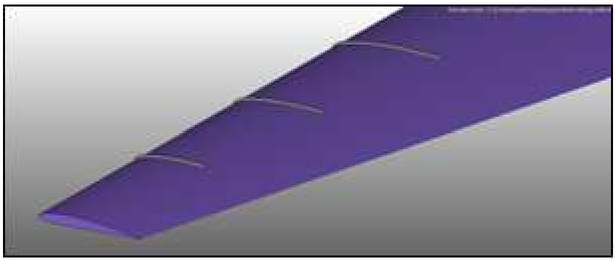


**Figure 4.5 Wing with Fence Configuration 3 :75% length**

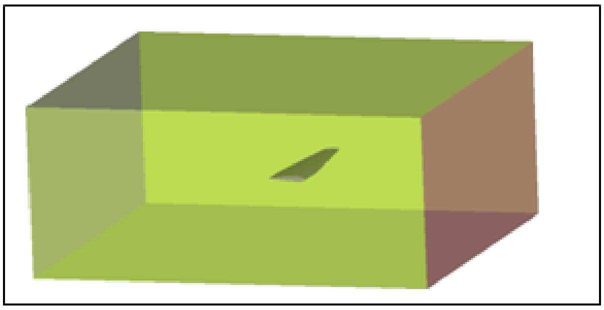


**Figure 4.6 Flow Domain around the wing Configuration**

## 4.2. MESHING

The flow domain is generated around the wing configuration in such a way that the domain extensions are as per the standard flow requirements. The rear end of the domain is extended for a distance of 6 times of the chord length whereas in the upstream is taken as 4 times of the chord length. CFD meshing – flow domain discretization. The meshing is done using HYPERMESH software. Figure 3.7 shows the mesh refinement near the leading edge of the wing. The mesh size near this region is kept as 3mm and the growth rate is maintained as 1.2. The maximum element size of the mesh is kept as 25mm.

**Figure 4.7 Mesh refinements near the leading edge of the wing**

**Figure 4.8 Mesh refinements near the trailing edge of the wing**

**Figure 4.9 Mesh flow from leading edge to trailing edge of the wing**

Figure 4.10 and 4.11 shows the cut sectional view of the volume mesh around the wing and the fence configuration

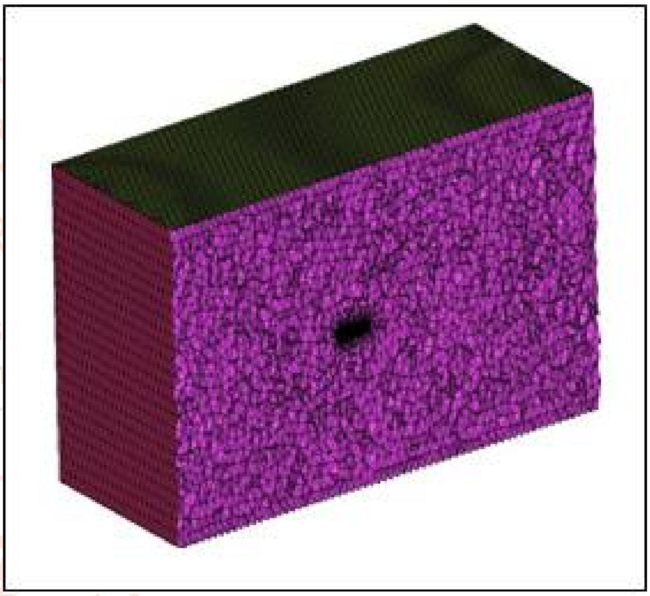


**Figure 4.10 Volume mesh cut sectional view of the flow domain**

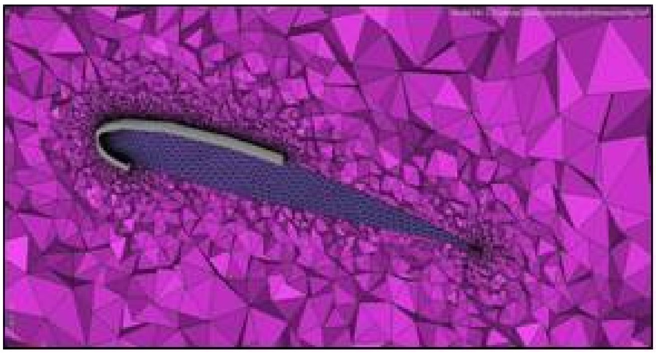


**Figure 4.11 Volume mesh cut sectional view around the Fence**

## 4.3. BOUNDARY CONDITIONS

The boundary conditions are applied to the meshed model. The domain inlet is given with velocity inlet boundary condition whereas the domain outlet is given with pressure outlet boundary condition. Standard wall functions are given to the wall boundary condition of the wing and fences. The consolidated boundary conditions are given in the table 3.1.

**Table 4.1 Boundary Conditions**

| Boundary | Type | Value |
|---|---|---|
| Domain Inlet | Velocity Inlet | 100 m/s |
| Domain Outlet | Pressure Outlet | 0 |
| Wing Surfaces | Wall | Standard Wall Conditions |
| Fences | Wall | |

**Table 4.2 Solver Settings**

| Model | Type | Value |
|---|---|---|
| Flow | Low speed incompressible | 100 m/s |
| Solver type | Pressure based | 0 |
| Turbulence | K-Epsilon | Standard wall conditions |
| Solution Control | Number of iterations | 500 |

## 5. RESULTS AND DISCUSSION

### 5.1. CONVERGENCE SOLUTIONS:

The convergence criteria for all equations (continuity, momentum and turbulence) are set to be 0.001 in Ansys Fluent. A pressure based Navier Stokes solver solves these flow equations with simple algorithm. Figure 5.1 shows the convergence history of flow and turbulence equations which are solved for the wing with fences.

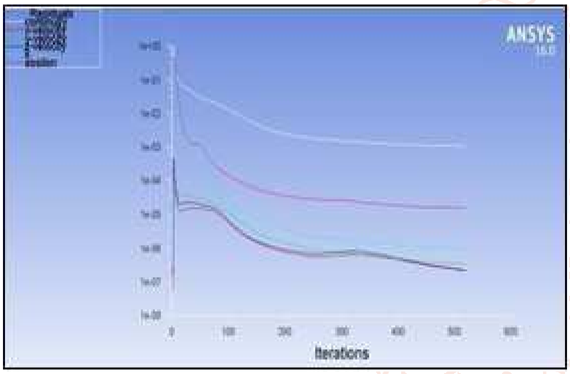



**Figure 5.1 Convergence History of the Flow and Turbulence equations**

### 5.2. COMPUTATIONAL RESULTS

The converged solutions are post-processed to plot various contours of pressure, velocity and turbulence intensity. The results obtained for both the cases with fence and without fences are compared and analyzed for better aerodynamic performance.

Figure 5.2 shows the static pressure variations over the wing without fence. It can be observed from the figure that the static pressure at the leading edge of the wing raised to maximum pressure which is further distributed over the wing.

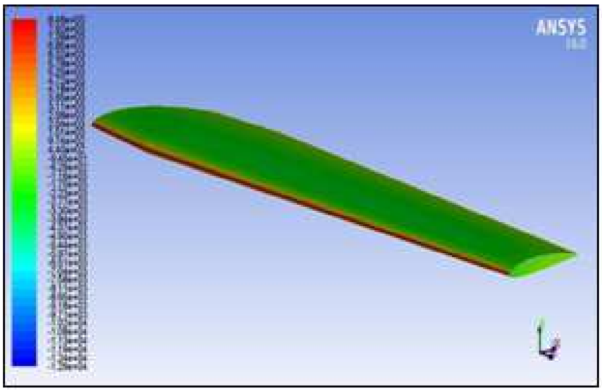



**Figure 5.2 static pressure variations over the wing without fence**

Figure 5.3 shows the static pressure variation over the wing with fences. There are three fences located along the span of the wing. The three types of fences are used with varying fence length with 25%, 50% and 75% of the chord length.

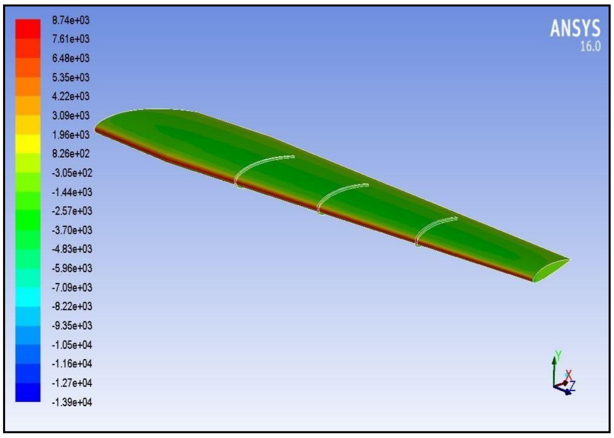



**Figure 5.3 static pressure variations over the wing with fences**

Figure 5.4 shows the turbulence intensity variations over the wing without fence. It can be observed from the figure that the turbulence intensity at the leading edge of the wing raised to maximum intensity level which is further distributed over wing

Figure 5.5 shows the turbulence intensity variation over the wing with fences. It can be observed from the figure that the effect of fence is to distribute the flow evenly over the wing which leads to reduce the flow separation.

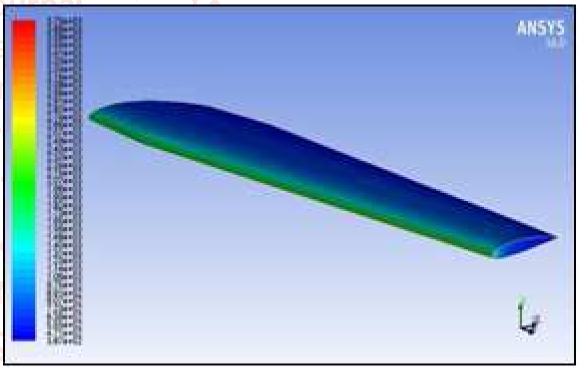



**Figure 5.4 Turbulence intensity variations over the wing without fence**

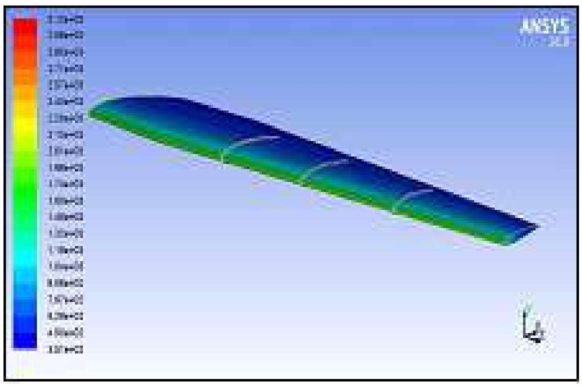



**Figure 5.5 Turbulence intensity variations over the wing with fence**

Figure 5.6 shows the velocity variation across the wing cross section without fence. It can be noticed from the figure that the stagnation in the region of leading edge and the flow separation in the region of trailing edge has been captured and predicted properly. Figure 5.7 shows the velocity variation across the wing configuration with fence.

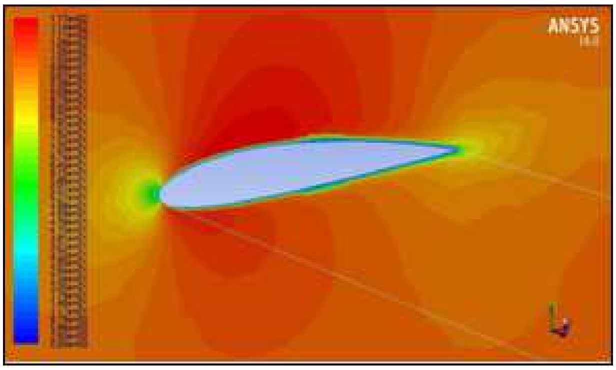



**Figure 5.6 Velocity variations across the wing cross section without fence**

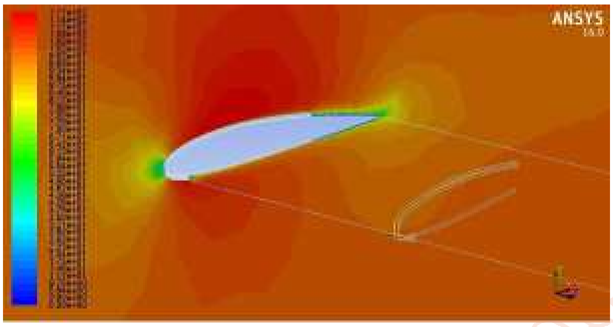



**Figure 5.7 Velocity variations across the wing cross section with fence**

Figure 5.8 shows the static pressure variation across the wing cross section without fence. It can be noticed from the figure that the stagnation in the region of leading edge leads to rise in static pressure and low-pressure region above the wing has been captured properly by CFD simulation. Figure 5.9 shows the static pressure across the crosses section of the wing with fences.

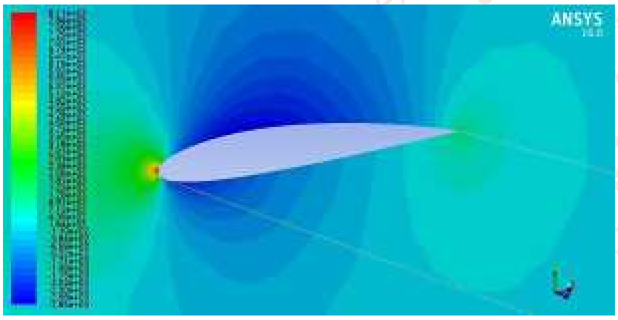



**Figure 5.8 Static Pressure Variations across the wing cross section without fence**

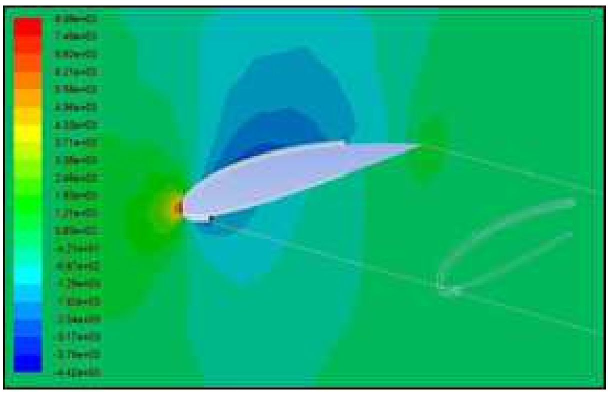


**Figure5.9 Static Pressure variations across the wing cross section with fence**

Figure 5.10 shows the intensity of turbulence across the wing without fence. It is observed from the figure that the turbulence intensity is maximum at the leading edge and further reduced along the flow direction

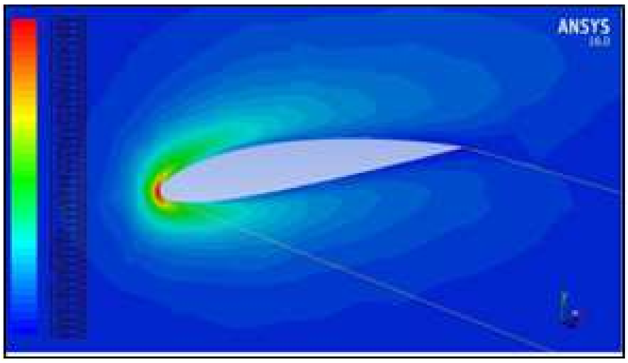



**Figure 5.10 Intensity of turbulence across the wing cross section with fence**

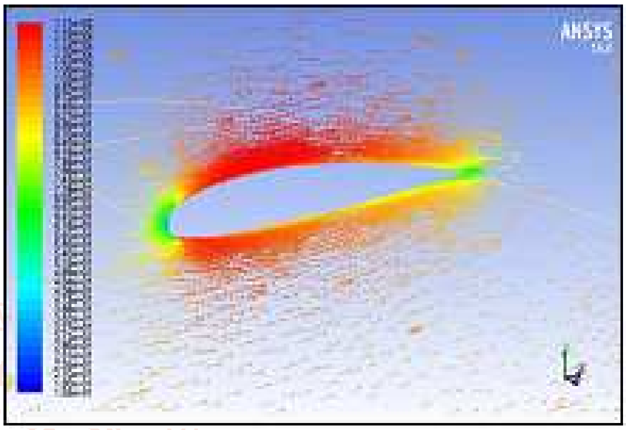



**Figure5.11 Intensity of turbulence across the wing cross section with fence**

Figure 5.12 shows the velocity vector distributions across the wing and along the flow direction. The velocity vector plot along the flow with fences is shown in Figure 5.13. It can be noticed from the figure that the flow vector orientations have been captured correctly by CFD predictions.

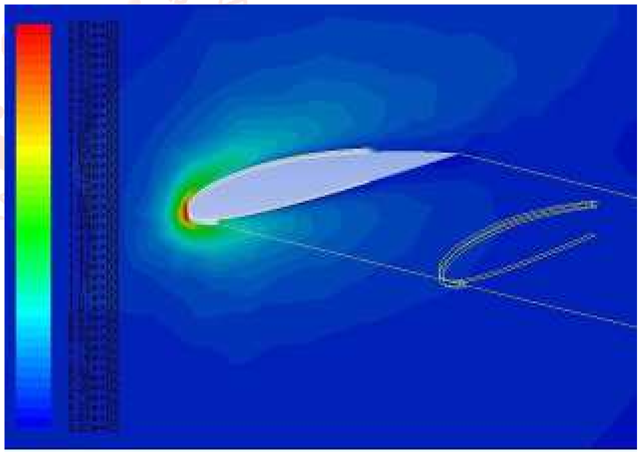


**Figure 5.12 Velocity Vectors around the wing without fence**


**Figure 5.13 Velocity Vectors around the wing with fence**

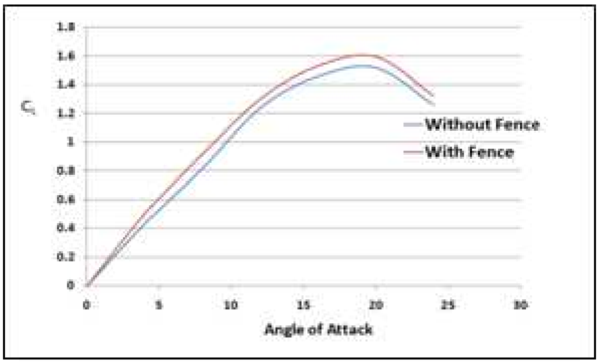



**Figure5.14 Comparison of wing with and without fences: Lift Co-efficient Vs Angle of Attack**

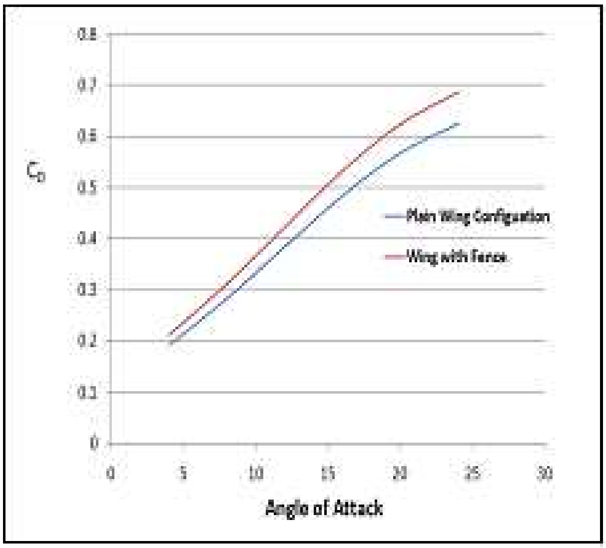



**Figure 5.15 Comparison of wing with and without fences: Drag Co-efficient Vs Angle of Attack**

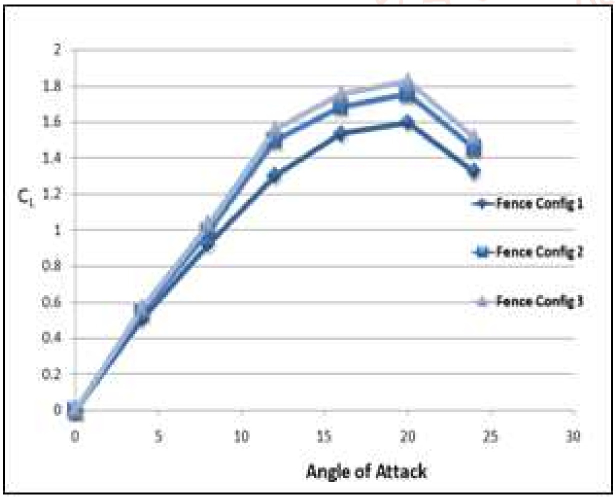



**Figure 5.16 Comparison of wing with various length of fences: Lift Co-efficient Vs Angle of Attack**

From the above graphs it can be noted that the lift coefficient is significantly affected by the insertion of fences. Also, the drag forces are very much reduced due to presence of fences. The stall angle also significantly extended when we use the fences over the wing.

Figure 4.14 shows the comparison of three different fence lengths over the wing. The lift forces are significantly increased due to the presence of fence. But there is no significant variation in lift between 50% and 75% length configuration.

## 6. CONCLUSION

The CFD simulations of the wing configuration with and without fences have been carried out successfully using ANSYS Fluent commercial code. The CFD methodology developed during this project can be successfully applied for the low-speed aerodynamics of wing configuration with and without fences. From the CFD results it can be noted that the insertion of fences significantly influences the aerodynamic characteristics of the wing performance. The variation of fence length also influences the lift characteristics of the wing. There is no significant variation of lift among the fences with 50% and 75 % of chord length. But the fences with 25% and 50% of the chord length show a significant variation of lift characteristics.

## REFERENCES

[1] Charles W. Harper and Ralph Lo Maki "A review of the stall characteristics of swept wings" Ames Research Center, Mofett Field CalzJ (2002).

[2] Cummings, Russell, James R. Forsythias, Scott A. Morton, And Kyle D. Squires "Computational challenges in high angle of attack flow prediction" Progress in Aerospace Sciences (2003).

[3] D. K. Mandal1*, S. Bandyopadhyay2 and S. Chakrabarti3 (2009). "A numerical study on the flow through a plane symmetric sudden expansion with a fence viewed as a diffuser", International Journal of Engineering, Science and Technology Vol. 3, No. 8, 2011.

[4] Grinstein, Fernando F. And Christerfureby, "Implicit large eddy simulation 42 of high-re flows with flux-limiting schemes". AIAA computational fluid dynamics conference (2003).

[5] Vavilis, Panagiotis S. And John A. Ekaterinaris, "Computational investigation of flow control over wings". AIAA Aerospace Sciences Meeting and Exhibit (2007).