© 2016 IJSRSET | Volume 2 | Issue 3 | Print ISSN : 2395-1990 | Online ISSN : 2394-4099
Themed Section: Engineering and Technology

CFD Simulation of an Airfoil at different Angle of Attack

Mragank Pratap Singh, Dr. Mahendra Singh Khidiya, Sahil Soni

Department of Mechanical Engineering, College of Technology And Engineering, MPUAT, Udaipur, Rajasthan, India

ABSTRACT

This research describes simulation of computational fluid dynamics (CFD) problems that occur in physical practices. The primary focus is on the simulation of the airflow around the airfoil. The fluid flow simulations are obtained with the FLUENT software package of ANSYS. The model was prepared with the software SOLIDWORKS. The pre-processing includes the creation and modelling in SOLIDWORKS and modification of the surface mesh in ANSYS. Further, a way of analysing the results and some of the outputs of the simulations and analysis has been presented. The CFD simulations were performed on the computational model of an airfoil. The computations were performed for different angle of attacks. It means that the laminar and turbulent flow and several combinations of the angle of attack have been considered.

The research aims to perform a CFD analysis on an aircraft model using FLUENT solver. While performing the simulations, meshing techniques, pre-processing and post processing sections and evaluation of a simulation are being learnt. Coefficient of lift and drag were also recorded. These values were also compared by running different simulations with change of input parameter i.e. angle of attack.

Keywords: Angle of attack; coefficient of drag; coefficient of lift; stall angle of attack

I. INTRODUCTION

An aircraft is a machine that is able to fly by gaining support from the air, or, in general, the atmosphere of a planet. It counters the force of gravity by using either static lift or by using the dynamic lift of an airfoil, or in a few cases the downward thrust from jet engines.

Different in-flight parameters like payload-capacity, endurance, maneuverability, fuel-consumption, noise-emission[1] etc. depend on the aerodynamic characteristics like lift, drag, vortex etc. The aerodynamic forces of foremost importance are Lift and Drag. The induced drag, which is the drag caused by the lift takes up to almost 33% of the total drag during cruise and it is even more significant in low speed which is up to 80-90% of the total drag[1]. Due to design of the Airfoil, the pressure difference is generated between the top and bottom of the airfoil. This is due to the fact that air flows with greater velocity on the upper surface of the as compared to power surface. Thus, creating low pressure on the top surface by Bernoulli's theorem. However on a finite wing, there is a leakage of air molecules at the wing tip which causes downwash, thus generating vortices at the trailing edge of the wing. Wing tip sails are attached to the wings in such a way they use local airflows about the wing tips induced by the generation of lift on the wing to produce thrust. Drag reduction is a big challenge posed in this area, but it is achievable. Specific design of airfoils are made into use to reduce drag. When simulating the flow over airfoils, transition from laminar to turbulent flow plays an important role in determining the flow features and in quantifying the airfoil performance such as lift and drag[3]. Stall Angle is the angle between the chord line of an airfoil and the undisturbed relative airflow at which stalling occurs where stalling refers to the condition when there is a sudden reduction in the lift generated by the wing[2]. Further research in this field is required.

- **Chord line**: - It is a straight line joining the leading edge to the trailing edge. It bisects the airfoil into two parts for a symmetric airfoil but may not do so for an asymmetric airfoil. It defines another important parameter Angle of attack.

- **Angle of attack**: - It is the angle which the chord line makes with the direction of motion of plane. It is an important parameter which affects the coefficient of lift and drag.
- **Chamber line**: - It is a line joining leading edge and trailing edge and dividing the airfoil into two symmetrical parts. It may or may not be a straight line.
- **Lift coefficient**: - It is a dimensionless coefficient that relates the lifting force on the body to its velocity, surface area and the density of the fluid in which it is lifting.
- **Drag coefficient**:- It is a dimensionless coefficient that relates the dragging force on the body to its velocity, surface area and the density of the fluid in which it is moving.
- **Stall angle of attack**:- It is the angle of attack at which the lift coefficient is maximum and after which the lift coefficient starts to decrease.

Chervonenko[4] showed the effect of attack angle on the non-stationary aerodynamic characteristics and flutter resistance of a grid of bent vibrating compressor blades. Bacha et al[5] presented drag prediction in transitional flow over two-dimensional airfoils. Eleni et al[6] evaluated the turbulence models for the simulation of the flow over NACA 0012 airfoil. Ramdenee et al[7] investigated on modeling of aerodynamic flutter on a NACA (National Advisory Committee for Aeronautics) 4412 airfoil with application to wind turbine blades. Johansen[8] also evaluated laminar/turbulent transition in airfoil flow.

II. METHODS AND MATERIAL

MODELLING- The geometry is generated in SOLIDWORKS. Here the coordinates are taken from UIUC official website.

Figure 1. NACA- 6409 Airfoil Source- CAD LAB, CTAE

Mesh Generation- the IGS file of the airfoil is then imported to ANSYS 15.0. Then the mesh is generated with coarse mesh.

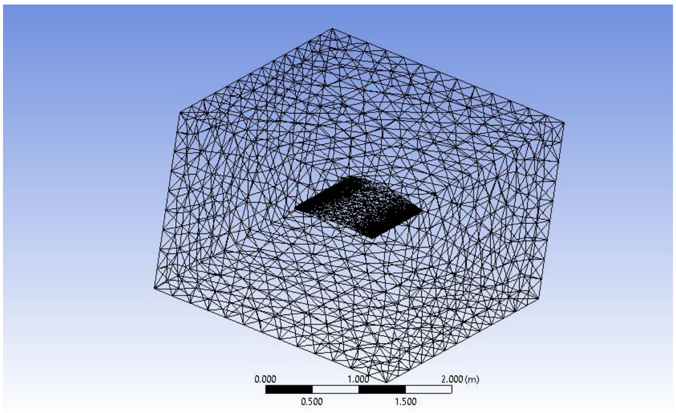


Figure 2. Mesh generation Source- CAD LAB, CTAE

ANALYSIS- CFD Analysis and study of results are carried out in 3 steps: Pre-processing, Solving and Post-processing by using FLUENT solver in ANSYS work bench.

Velocity Inlet: The inlet boundary conditions involve velocity components for varying angle of attack, turbulence intensity and turbulent viscosity ratio.

Pressure Outlet: Ambient atmospheric condition is imposed at outlet.

Operating conditions:

Velocity- 200 m/s
Nodes- 19669
Elements- 110255
Pressure based.
Model k-epsilon: standard.
Solution control gauss-seidel method.

III. RESULTS AND DISCUSSION

Analysis set up of all cases is carried out in ANSYS FLUENT Solver. Result analysis is done using ANSYS Post processor. Velocity and pressure plots are plotted for all the cases of study.

CASE 1: Angle of attack is **ZERO** degree and velocity 200 m/s

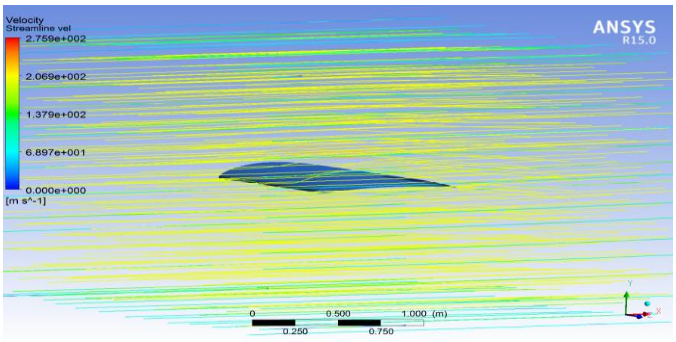


Figure 3. Velocity Streamline at 0 degree angle of attack Source- CAD LAB, CTAE

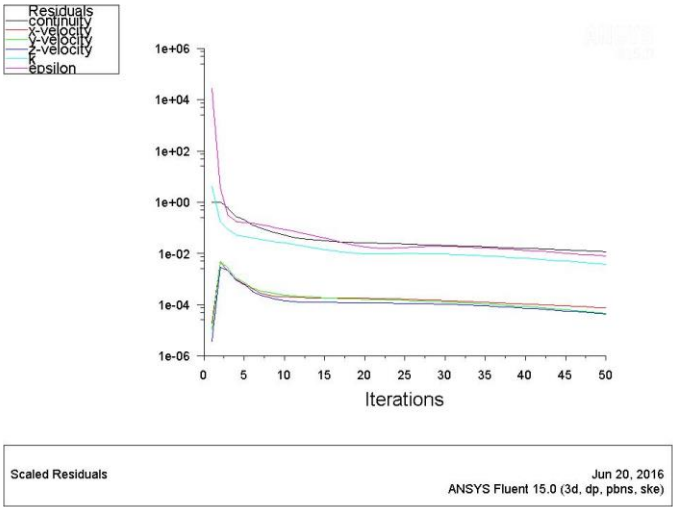


Figure 4. Velocity residuals at 0 degree angle of attack Source-CAD LAB, CTAE

Case-2: At 16 degree angle of attack (stall)

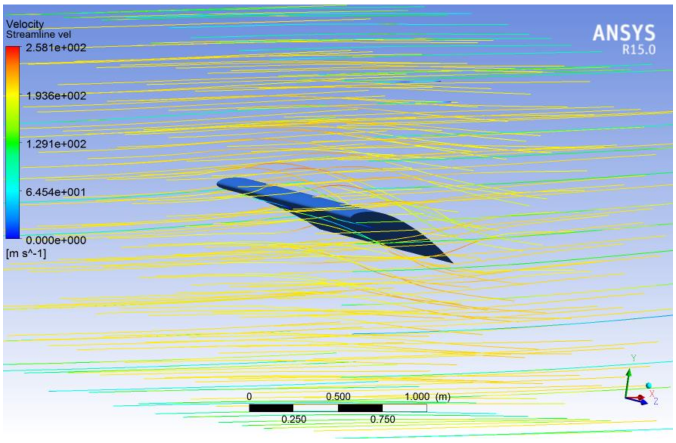


Figure 5. Velocity streamline at 16 degree angle of attack Source-CAD LAB, CTAE

TABLES AND GRAPHS

After performing simulation method at various angles of attack, the values of Coefficient of Lift and Drag were observed.

Table-1: Coefficient of Lift vs. angle of attack

Angle of Attack	Coefficient of lift (C_L)
0	0.223
2	0.295
4	0.357
6	0.414
8	0.469
10	0.517
12	0.587
14	0.673
16	0.702
18	0.275

Graph-1: Coefficient of Lift vs. Angle of attack

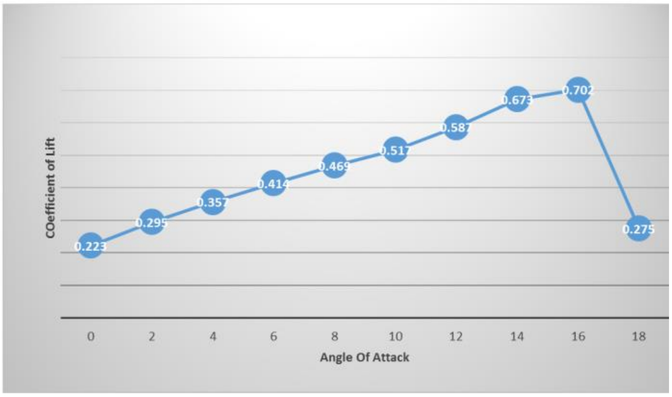
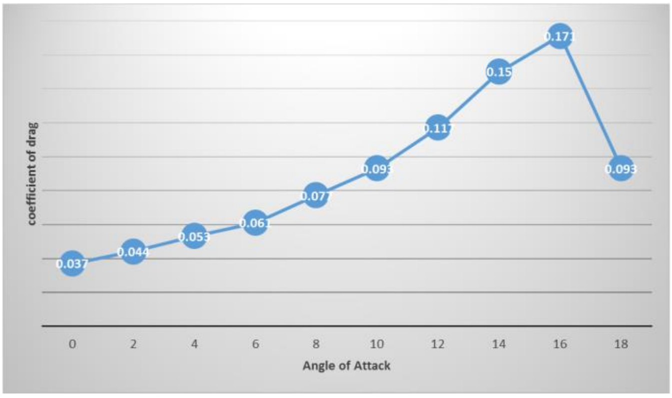


Table-2: Coefficient of Drag vs. Angle of attack

Angle of Attack	Coefficient of Drag (C_D)
0	0.037
2	0.044
4	0.053
6	0.061
8	0.077
10	0.093
12	0.117
14	0.150
16	0.171
18	0.093

Graph-2: Coefficient of Drag vs. Angle of attack



Above performances shows that as we increase angle of attack lift tend to increase, though this trend does not remain the same. After few degrees of increase in angle of attack lift start reducing drastically, this angle of attack called stall angle of attack. Attack flow ascends towards the middle of the aircraft wings. This induces more turbulence to the flow and increase the sound.

IV. CONCLUSION

Results shows the simulated flow over airfoil section and it observed that as angle of attack increases, lift also increases. The Stall angle of attack for airfoil is found out to be at 16 degrees. For an angle of attack 16 degrees at velocity of 200m/s it was observed that the lift force was 0.22303355 N and related coefficient of lift was 0.702 and coefficient of drag was 0.171.The lift and drag depend on the airfoil shape and it is depending upon the velocity distribution, but also on the wing planform and on the wing area.

V. REFERENCES

[1] Shivasharanayya Hiremath and Anandkumar.S.Malipatil, "CFD Simulations of Aircraft Body with Different Angle of Attack and Velocity" by International Journal of Innovative Research in Science, Engineering and Technology (An ISO 3297: 2007 Certified Organization) in Vol. 3, Issue 10, October 2014.

[2] Arvind Prabhakar and Ayush Ohri (2013), "CFD Analysis on MAV NACA 2412 Wing in High Lift Take-Off Configuration for Enhanced Lift Generation" by Aeronautics & Aerospace Engineering.

[3] Ravi.H.C, Madhukeshwara.N and S.Kumarappa,"NUMERICAL INVESTIGATION OF FLOW TRANSITION FOR NACA-4412 AIRFOIL USING COMPUTATIONAL FLUID DYNAMICS" by International Journal of Innovative Research in Science, Engineering and Technology (Vol. 2), Issue 7, July 2013.

[4] A.G Chervonenko, 1991, Effect of attack Angle on the Nonstationary Aerodynamic Characteristics and Flutter Resistance of a Grid of Bent Vibrating Compressor Blades, Ukrainian Academy of Sciences, Plenum Publishing Corporation, Ukraine, Volume 39, No. 10, pp. 78-81.

[5] Bacha WA, Ghaly WS, 2006, Drag Prediction in Transitional Flow over Two-Dimensional Airfoils, Aerospace Sciences Meeting, USA, AIAA 20062048.

[6] Douvi C. Eleni, Tsavalos I. Athanasios and Margaris P. Dionissios, 2012, Evaluation of the Turbulence Models for the Simulation of the Flow over an Aerofoil, Journal of Mechanical Engineering Research, Greece, Volume 4, No.3, pp. 100-111.

[7] Drishtysingh Ramdenee, H. Ibrahim, N.Barka, A.Ilinca, 2013, MODELING OF AERODYNAMIC FLUTTER ON A NACA 4412 AIRFOIL WIND BLADE, International Journal of Simulation and Process Modelling, Inderscience Publishers, Canada, Volume 8, No. 1 , pp. 79-87.

[8] Johansen J, 1997, Prediction of Laminar/Turbulent Transition in Airfoil Flows, Journal of Aircraft, Aerospace Research Central, Denmark, Volume 36, No. 4, pp. 731-734.