Imperial Journal of Interdisciplinary Research (IJIR)
Vol-2, Issue-5, 2016
ISSN: 2454-1362, http://www.onlinejournal.in

CFD Analysis of Velocity Profiles for NACA 4412 Aerofoil

Jeyo Paul[1], Ajo Alexander[2] & Manikandan S[3]

[1,2]Final year B.Tech student, Department of Mechanical Engineering, MBCCET, India
[3]Assistant Professor, Department of Mechanical Engineering, MBCCET, India

Abstract*: In this paper we have obtained the velocity profile, drag and lift forces using CFD which can also be determined through experiments using wind tunnel testing. Here we have gone through an analytical method, this method is solely based on the principle of finite element method and computational fluid dynamics. The drawing and meshing of NACA 4412 is done in GAMBIT2.4.6 and CFD analysis is done in ANSYS FLUENT6.3.26.*

Keywords: CFD, Drag, Lift, NACA, GAMBIT, ANSYS FLUENT, Aerofoil

1. Introduction

Technology is all about the innovative ideas that we bring about in the simplest manner. Innovation is in a way, the most practical and economical path which is chosen to deal with the circumstances. In the present case, the detailed explanations and studies regarding the lift, drag and the variation in the angle of attack that occurs in an aerofoil is simulated using the ANSYS FLUENT software. It is practically impossible to test all these on a real aerofoil and more than that it is expensive and not at all economical. So in order to deal with these scenarios we used the GAMBIT software to develop initially the two dimensional structure of an aerofoil and mesh it accordingly to obtain the rigorously accurate calculations.

The ANSYS FLUENT software is computational fluid dynamic software that investigates thoroughly on an aerodynamic structure like the aerofoil, rather than any experimental method. Using this software had already altered our perspective regarding the aerofoil. The study of the aerofoil using the ANSYS FLUENT software made it clear about the drag force, lift force, coefficient of drag and coefficient of lift. Various angle of attacks such as 1^o, 6^o and 15^o were simulated and analyzed. All these had attack. From the iterations that we have undertaken, we understood a general trend about how the various parameters change in accordance with varying angle of attack.

2. Background

An aerofoil is an aerodynamic structure which primarily allows the flow of a fluid along its surface and analyzing how this fluid flow takes place is mainly specified in the studies that are conducted. An aerofoil have both low pressure region and high pressure region and flow takes place from the high pressure region to the low pressure region. And in order to understand how this flow takes place and how the parameters like velocity and pressure vary are discussed here. Studies regarding the meshing and grid independence study were also conducted. By focusing on the grid independence study, we analysed that the accuracy obtained by the original mesh does not vary when the mesh density and resolution was increased. All these were analysed in the GAMBIT software and ANSYS FLUENT was basically used to find out the convergence solution for the developed structure of the aerofoil.

3. Aerofoil Nomenclature

An airfoil or aerofoil is the shape of a wing, blade (of a propeller, rotor, or turbine), or sail. An aerofoil is a body of such a shape that when it is placed in an airstream, it produces an aerodynamic force.

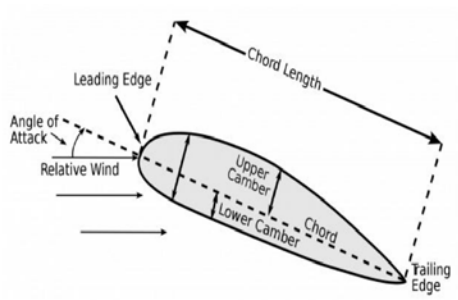

Fig.1 Aerofoil Nomenclature[3]

Chord and Chord length**:-**Chord is an imaginary reference line in the aerofoil section above and sometimes below which, the curvature of the upper and surface of the aerofoil is described. The chord can be found out by joining the most forward point on the leading edge to the rear most point on the trailing edge. The "chord length" is the length of the projection of the aerofoil on the chord.

Camber:-Camber is the name given to describe the curvature of the upper and the lower surfaces of an aerofoil section. Camber is define as the ratio of the vertical intercepts and the chord length, the vertical intercept being measured as the length between the surface concerned and the chord. The length of the intercepts between chord and the upper surface expressed as the percentage of chord length is called "upper camber" and that between the chord and lower surface expressed as the percentage of chord length is called the "lower camber". The "mean camber" is the vertical intercepts between the geometrical centerline of the aerofoil section at the center of the chord expressed as a percentage of chord length and the "mean camber line" is a line drawn half way between the upper surface and the lower surface; that is the geometrical centerline.

Thickness ratio:-Thickness ratio is the ratio of the maximum thickness to the chord length.

Relative wind and angle of attack:-The angle of attack of an aerofoil in a relative wind is the angle made by the direction of the relative wind on the chord of the aerofoil and is denoted by "α".

Lift:-The component of the aerodynamic force that is perpendicular to the relative wind.

Drag:-The component of the aerodynamic force that is parallel to the relative wind.

Leading edge:-The leading edge is the point at the front of the aerofoil that has maximum curvature.

4. Mathematical Formulation

The force which is exerted on a body by a flowing fluid in the direction of flow is called drag. The component of this force normal to the body is called lift force. The lift of an aerofoil can be explained with the help of Bernoulli's equation. According to Bernoulli's equation, for an incompressible steady state flow, pressure increases if the flow velocity decreases and vice versa.

When the air passes over the aerofoil, velocity increases as the air continues to flow from its leading edge to the upper surface of the aerofoil. The pressure is decreased in that area. But on the other hand, velocity decreases as the air passes through the bottom of the aerofoil and the pressure is increased. This positive pressure acting upward is the main reason for producing lift.

When a fluid separates from a body, it forms a separated region between the body and the fluid stream. This low pressure region behind the body where recirculation and backflows occur is called the separated region. The larger the separated region, the larger the drag force. Wake can be defined as a region of flow trailing the body where the effects of the body on velocity are felt. Wake consists of vortices which are responsible for creating drag by creating negative pressure in that region. Wake can occur in an aerodynamic body with a relatively large angle of attack (larger than 15 degree for most aerofoils). This is known as stalling point. Negative pressure and drag force become dominant from stalling point [5].

The lift and drag forces associated with the aerofoil can be explained in terms of coefficient of lift and coefficient of drag.

$$F_D = \frac{1}{2} \rho A V^2 C_D$$

$$F_L = \frac{1}{2} \rho A V^2 C_L$$

Where, C_L and C_D are the coefficient of lift and coefficient of drag corresponding to the lift force (F_L) and drag force (F_D) respectively. $\frac{1}{2} \rho A V^2$ denotes the dynamic pressure. The lift and drag force can be obtained by determining the values of dynamic pressure and coefficient of lift and drag respectively.

5. Analytical Method

In this method NACA 4412 aerofoil were generated by importing the co-ordinates to GAMBIT in DAT format. The generated 2D geometry of the aerofoil is shown in fig.2.

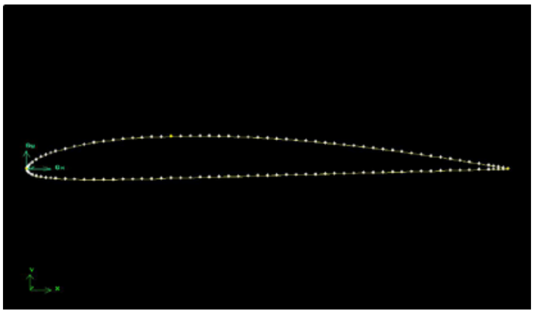

Fig.2: 2D geometry in GAMBIT

The meshing of aerofoil profile was also done in GAMBIT. This meshing process and principles are based on the theory of finite element analysis method. The final meshed geometry surrounding the aerofoil is as shown in the fig.3

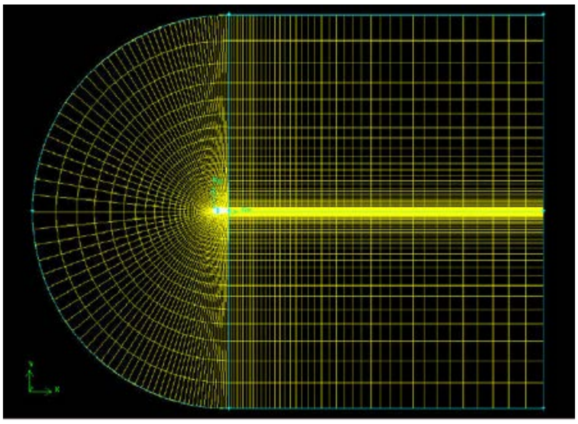

Fig.3: Final meshed region surrounding the aerofoil geometry

Once the meshing is completed, the final step in GAMBIT is to define the boundary types. The aerofoil is defined as a wall across which the flow occurs. The completed mesh of the region surrounding the aerofoil is exported as two dimensional mesh file for the purpose of simulation in ANSYS FLUENT. The specification of the mesh is given in table 1.

Table 1: Specification of aerofoil mesh

Number of Elements	14400
Number of Nodes	14705
Number of zones	5

The software used for simulation and analysis is the ANSYS FLUENT 6.3.26. The FLUENT is launched as two dimensional double precision abbreviated as 2ddp. Once the program is launched the exported aerofoil mesh file is imported from the working directory. The input or reference values for the iteration process can be well explained from table 2 which also describes the various angles of attack at which simulations were carried out.

Table 2: Input values set for the aerofoil

No.	Input	Value
1	Velocity of flow	0.015Mach or 51m/s
2	Operating pressure	101325 Pa
3	Model	Inviscid
4	Density of fluid	1.225 kg/m^3
5	Kinematic viscosity	1.7894e-05 kg s/m^2
6	Angle of attack	$1^0, 6^0, 15^0$
7	Fluid	Air as ideal

6. Results and Discussions

The sole purpose of this simulation based paper was to compare the different values of velocity obtained by changing the angles of attack of NACA 4412 aerofoil and thus finding out which one is the most efficient between them. It is obvious that there were significant changes in velocity when the angle of attack of the aerofoil was changed.

The velocity on the upper surface of the aerofoil has a higher value than the lower surface. Thus the pressure at the upper surface is low compared with the lower surface of aerofoil. According to Bernoulli's theorem, the object will move from higher pressure region to lower pressure region. Thus the lift occurs in aerofoil.

The C_L & C_d verses number of iterations, velocity vectors, velocity contours at angle of attack 10, 60 & 150 are shown below.

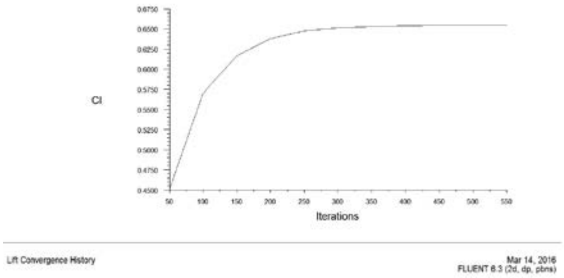

Fig.4 Plot between the coefficient of lift and number of iteration at an angle of attck 1^0

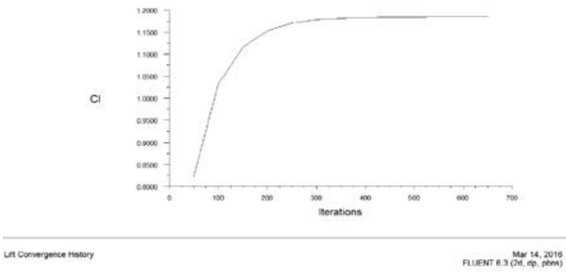

Fig.5 Plot between the coefficient of lift and number of iterationsat an angle of attack 6^0

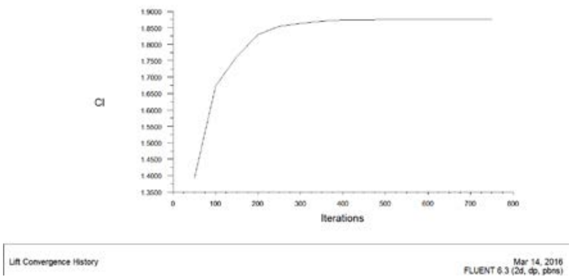

Fig.6 Plot between the coefficient of lift and number of iterations at an angle of attack 15^0

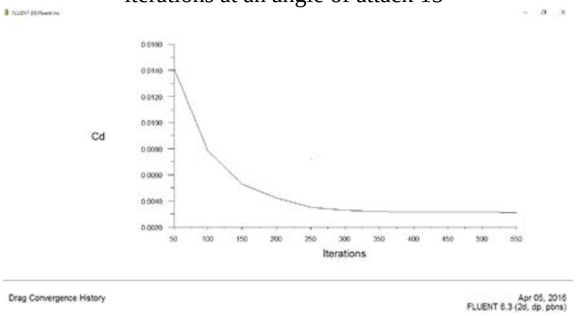

Fig.7 Plot between the coefficient of drag and number of iteration at an angle of attck 1^0

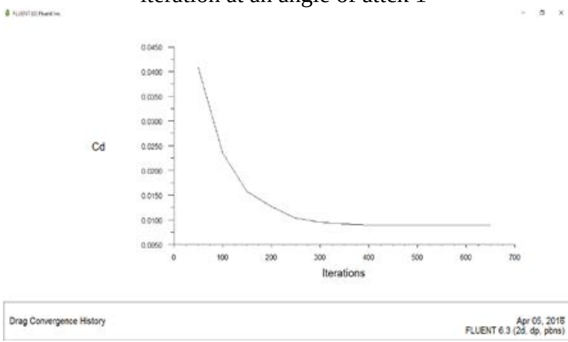

Fig.8 Plot between the coefficient of drag and number of iterationsat an angle of attack 6^0

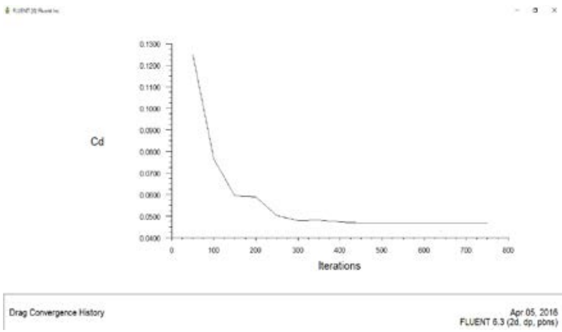

Fig.9 Plot between the coefficient of drag and number of iterations at an angle of attack 15^0

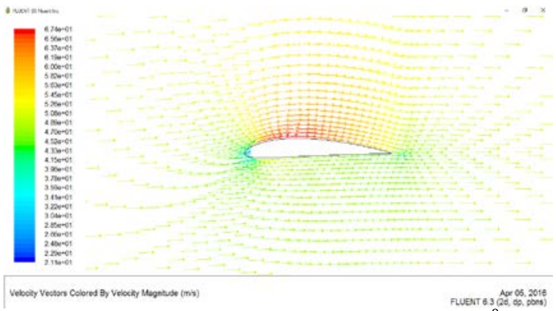

Fig.10 Velocity vector for an angle of attck 1^0

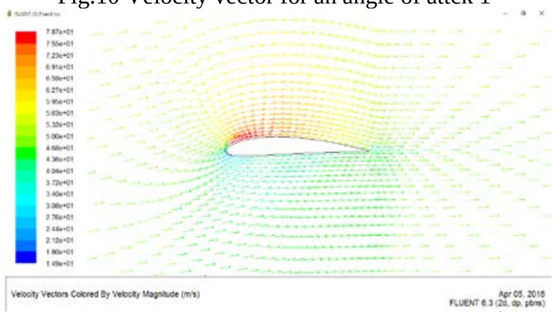

Fig.11 Velocity vector for an angle of attack 6^0

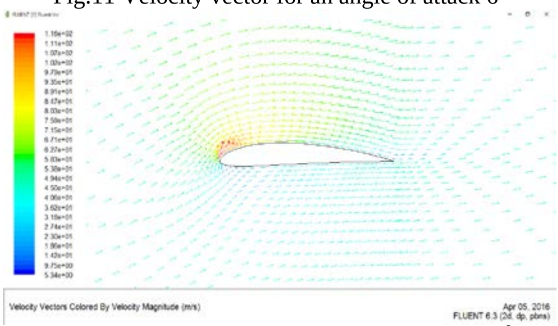

Fig.12 Velocity vector for an angle of attack 15^0

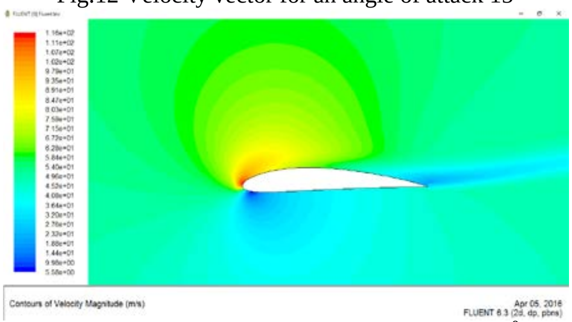

Fig.13 velocity contour for an angle of attack 1^0

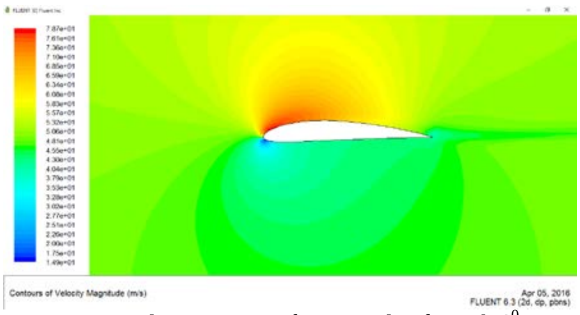

Fig.14 velocity contour for an angle of attack 6^0

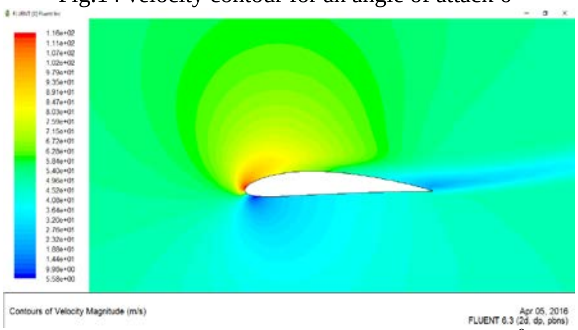

Fig.15 velocity contour for an angle of attack 15^0

7. Conclusion

From the study conducted over the velocity profile of NACA 4412 we could analyze that the variation in the velocity has been increasing with the increase in the angle of attack. Also as the angle of attack is increased we could also analyze that there is a rapid increase in the lift force developed.

8. References

[1] H. Glauert,"The Element of Aerofoil and Airscrew Theory".

[2] A.G Chervonenko, "Effect of attack Angle on the Non stationary 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. 78-81,1991.

[3] Karna .S. Patel, Saumel B. Patel, Utsav B. Patel, Prof. Ankit P. Ahuja, UVPCE, Ganpat University, CFD Analysis of Aerofoil, International journal of engineering research, volume 3,Issue No.3, page: 154-158.

[4] Michael .S .Selig, Bryan .D. McGranahan , Wind Tunnel Aerodynamic tests of six airfoils for use on small wind turbines, Department of aerospace engineering, UIUC, ASME Journal of solar energy engineering, volume 126, November 2004.

[5] MD. Safayet Hossain, Muhammad Ferdous Raiyan, Mohammed Nasir Uddin Akanda, Nahed Hassan Jony, A Comparative flow analysis of NACA 6409 and NACA 4412 aerofoil, International jornal of research in engineering and technology, volume 3, issue No: 10, October-2014.