International Journal of Engineering Research & Technology (IJERT)
ISSN: 2278-0181
Vol. 6 Issue 09, September - 2017

Study and Analyse Airfoil Section using CFD

Rajat Veer*, Kiran Shinde*,
Vipul Gaikwad*, Pritam Sonawane*,
*Student,
Department of Mechanical Engineering,
RMD Sinhgad School of Engineering,
Pune.

Yogesh Sonawane#
#Assistant Professor,
Department of Mechanical Engineering,
RMD Sinhgad School of Engineering,
Pune.

Abstract— **In this report coefficient of drag and coefficient of lift is obtained with the help of CFD, it is also obtained by an experiment which is conducted on wind tunnel. Though both of the methods give approximate result for the same test section, an experimental process has greater cost and quite laborious than that of CFD. Analysis of airfoil over two-dimensional subsonic flow at a various angle of attacks and operating at Reynolds's number is obtained. The result shown by CFD has closely agreed with experiment result, thus CFD is a mature tool to predict the performance of test section at any angle of attack.**

Keywords— Flow separation, angle of attack, CFD, coefficient of lift and coefficient of drag, pressure coefficient etc.

INTRODUCTION

It is fact of common experience that a body in motion through a fluid experiences a resultant force mainly a resistance to a motion. A class of body exists, for which the component of resultant force normal to the direction of motion is many times greater than the component resisting the motion [1]. An airfoil is a streamlined body found in airplanes, propellers, turbines and many other applications. When an airfoil body passing through any fluid it produces an aerodynamic force which is due to pressure distribution over the body surface and shear stress distribution over the body surface. This aerodynamic force can be resolved into two components known as lift and drag. The force which acts in perpendicular to the direction of motion is called as lift force, and the force which is parallel to the direction of motion is called as drag force. Lift is generated by aerofoil primarily depends upon surface area and angle of attack. The drag force mainly depends upon the body surface and fluid which is flows over it. This lift and drag force are obtained with the help of wind tunnel. A wind tunnel is a machine which used in aerodynamic research to study the effects of air moving on solid objects. A wind tunnel consists of a tubular passage with the object under test mounted in the middle. Air is then moving past the object by a powerful fan system or other means. The test object often called a wind tunnel model, is instrumented with suitable sensors to measure aerodynamic forces, pressure distribution, or other aerodynamics-related characteristics. The advance in computational fluid dynamics modeling on a high speed digital computer has reduced the demand for wind tunnel testing. However, CFD results are still not completely reliable and wind tunnels are used to verify CFD predictions.

GEOMETRY

In aerodynamics, Airfoil design is a major facet. Different flight regimes show different results. There are basic dissimilarities between symmetric and asymmetric aerofoil like at zero degree angle of attack, Asymmetric airfoils can generate lift, while a frequent inverted flights suits symmetric airfoil as in the case of an aerobatic airplane. Thus without boundary layer separation, we can use various angles. Subsonic airfoils have a round leading edge around which it is naturally insensitive to angle of attack [2].

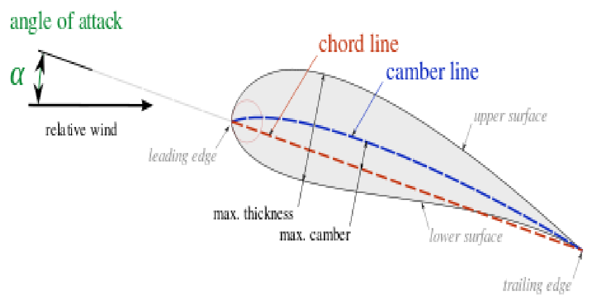

Fig 1. Airfoil Geometry

The point at front of the airfoil that has maximum curvature is known as a Leading edge. The trailing edge is defined as the point of minimum curvature at the rear of the airfoil. The straight line joining the leading edge and trailing edge of airfoil section is chord line. Chamber line is the locus of point's midway between the upper and lower surface of an airfoil. The Angle of attack is measured by taking a difference between free stream velocity and chord line. It is the ratio of a span of an aerofoil to the chord length of an aerofoil is called as the Aspect ratio.

COMPUTATIONAL FLUID DYNAMICS (CFD)

Computational Fluid Dynamics is a branch of fluid mechanics that analyze problems involving fluid flow by using numerical analysis and data structure. The interaction of liquid and gases with a surface defined by boundary condition are solved with the help of computers, which performs its calculations. CFD is commonly accepted as referring to the board topic encompassing the numerical solution, by computational methods, of the governing equations which describe fluid flow, the set of Navier-stokes equation, continuity and any additional conservation equation like energy or species concentration. CFD is considered as a bridge between the pure experiment and pure theory. Computational fluid dynamics predict the performance of a system before

International Journal of Engineering Research & Technology (IJERT)
ISSN: 2278-0181
Vol. 6 Issue 09, September - 2017

installing it in real life. CFD predict which design changes are most crucial to enhance the performance. Moreover, there are several unique advantages of CFD over experimental-based fluid system design.

• CFD provides more detail and comprehensive information.

• Ability to study system under zero hazardous conditions at and beyond their normal performance limits.

• Power consumption of CFD is low as compared to a wind tunnel.

CFD Analysis of temperature, velocity, and chemical concentration distribution can help an engineer to understand the problem correctly and provide ideas for getting the best resolution.

ANGLE OF ATTACK

When you stretch your arm out through window of car which moves at high speed, you can feel that your arm pull back while colliding with oncoming air and when you hold your hand outside of window in parallel to the direction of road and just inclined it in certain angle, you feel like your hand push upward it is because of oncoming air strikes at your hand. Angle Of Attack is the angle between the reference line of a body and relative wind or oncoming air [3]. On an airfoil such as one on a wind turbine, it is the angle between the chord line and relative wind vector. The relation between an angle of attack, the coefficient of lift and coefficient of drag is as follows:

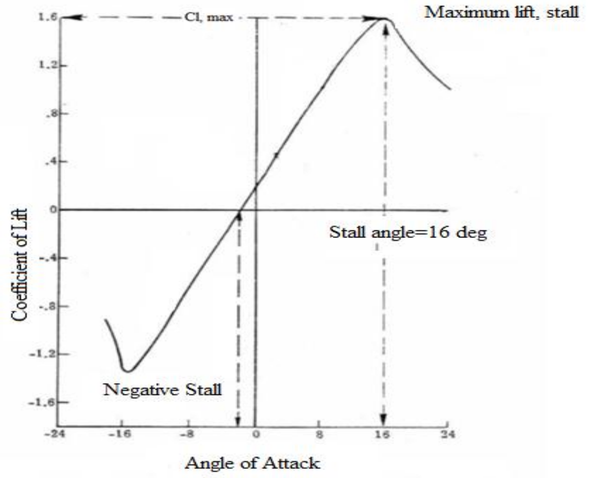

Fig 2. Variation of Angle of attack vs Coefficient of Lift

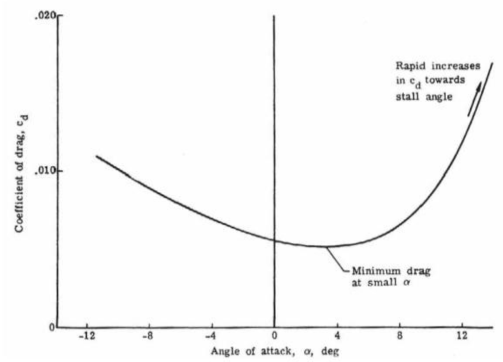

Fig 3. Variation in Angle of attack vs Coefficient of Drag

A typical graph of coefficient of lift against the angle of attack of the airfoil section is studied. One of the first things noticed is the fact that at an angle of attack of 0°, there is a positive coefficient of lift, and, hence, positive lift. One must move to a negative angle of attack to obtain zero lift coefficients. It will be remembered that this angle is called the angle of zero lift. A symmetric airfoil was shown to have an angle of zero lift equal to 0° as might be expected. From the diagram we can see that as an angle of attack increased coefficient of lift associated with it is also increases up to a certain maximum point known as a stall angle. Above this angle, however, the lift coefficient reaches a peak and then declines. The angle at which the lift coefficient (or lift) reaches a maximum is called the stall angle. Beyond the stall angle, one may state that the airfoil is stalled and a remarkable change in the flow pattern has occurred.

Originally the value of drag coefficient is zero at zero degree angle of attack. But then as we increase an angle of attack drag coefficient will also increase before and after stall condition occurs. Minimum drag coefficient occurs at a small positive angle of attack corresponding to a positive lift coefficient and builds only gradually at the lower angles. Near to the stall angle, C_D increase rapidly because the greater amount of turbulent and separated flow occurred.

FLOW SAPERATION

All solid objects travelling through fluid experience viscous forces occur in the layer of fluid close to the solid surface which acquires a boundary layer of fluid around them. Boundary layers can be in the form of laminar or turbulent. By calculating the Reynolds number of the local flow conditions we can make a decision that the flow is laminar or turbulent. When the boundary layer travels far enough against an adverse pressure gradient and at the same time the speed of the boundary layer relative to the object falls almost to zero then Flow separation occurs. In aerodynamics, flow separation can often result in increased drag. For this reason, much effort and research have gone into the design of aerodynamic and hydrodynamic surfaces which delay flow separation and keep the local flow attached for as long as possible.

OPERATING PARAMETERS

Analysis on the airfoil profile is carried out to find the values of C_D and C_L at different values of angle of attack .Here we are going to analyse the Airfoil of Chord Length 160 mm using CFD. For that we take some initial assumptions or boundary conditions for our problem which are as follows.

a) Span of airfoil = 0.3 m

b) Density of air = 1.208 kg/m^3

c) Length of airfoil = 0.16 m

d) Velocity of wind = 30.48 m/s

e) Fluid = Air as ideal

f) Operation Pressure = 13 bar

International Journal of Engineering Research & Technology (IJERT)
ISSN: 2278-0181
Vol. 6 Issue 09, September - 2017

MESH GENERATION

Today there is numerous analysis software which is getting used for geometry-integrated mesh generation and post-processing purpose. ANSYS ICEM CFD has desired effect that it keeps close relationship with geometry during mesh generation and post-processing. From geometry to mesh generation in an analysis, it provides refined geometry acquisition, mesh generation, post-processing and mesh optimization tools. The mesh generation properties in ICEM CFD offers from, a capability to parametrically create grids from geometry in multi-block structured, unstructured hexahedral, tetrahedral, hybrid grids consisting of hexahedral, tetrahedral, pyramidal and prismatic cells; and Cartesian grid formats combined with boundary conditions.

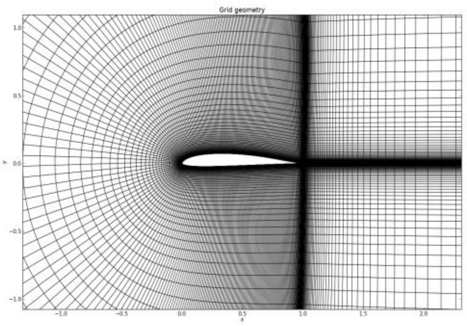

Fig 4 . Mesh generation

I. RESULT

We have plotted the pressure contours as well as velocity contours for chord length of 0.16 m and changing the values of angle of attack. Following are the cases at the stall angle and few degrees before and after it.

a) *Pressure Contours*

Next Figure shows pressure distribution on the upper and lower surface of airfoil for 10º angle of attack. There is positive pressure on the upper surface and negative pressure on the lower surface. Value of lift coefficient is 0.17769.

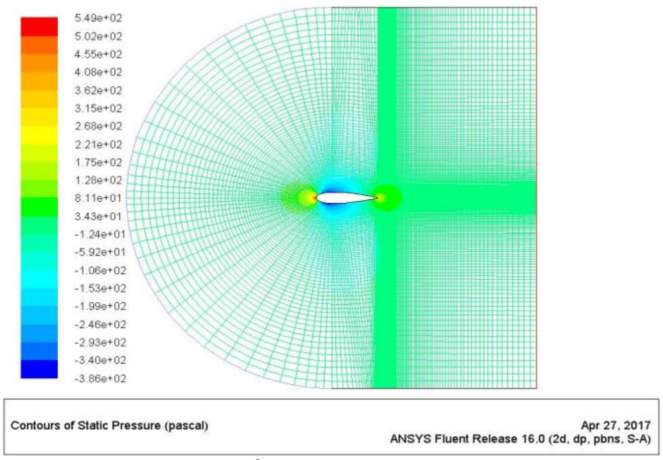

Fig 5. Contour of static pressure at 10 Degree AOA

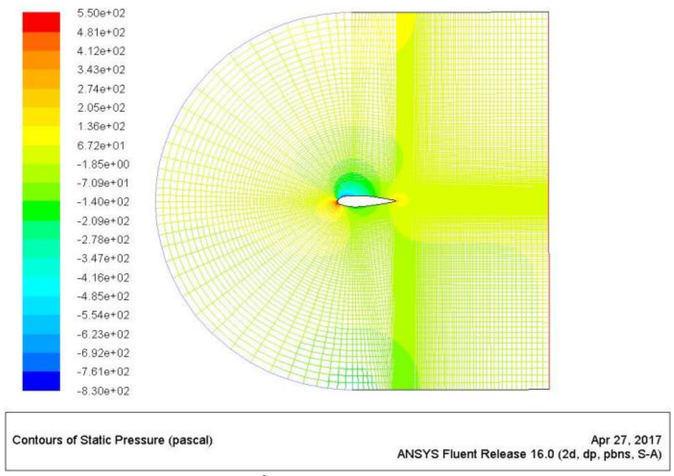

Fig 6. Contour of static pressure at 14 Degree AOA

Above figure shows pressure distribution on the upper and lower surface of airfoil for 14º angle of attack. The value of lift coefficient is maximum at this angle of attack. Hence it is called as angle of attack. Value of lift coefficient is 0.2095. Maximum lift is due to maximum pressure difference on the upper and lower surface of the airfoil.

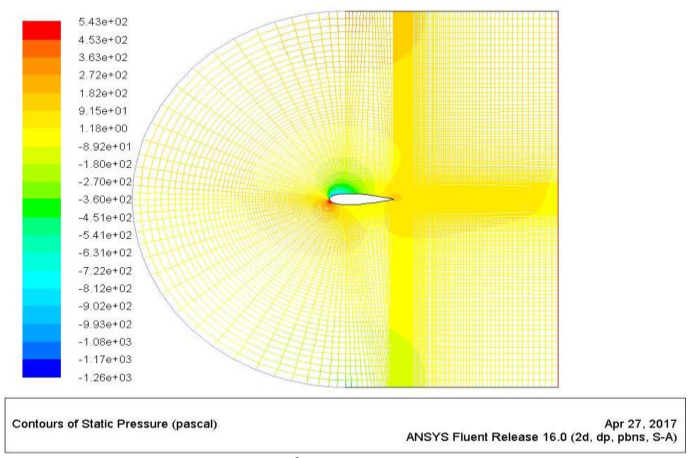

Fig 7. Contours of pressure at 16 Degree AOA

Above figure shows pressure distribution on the upper and lower surface of airfoil for 16º angle of attack. There is positive pressure on the upper surface and negative pressure on the lower surface. Value of lift coefficient is 0.08087.It is observed that lift generated decreases after the stall angle.

b) Velocity Contours

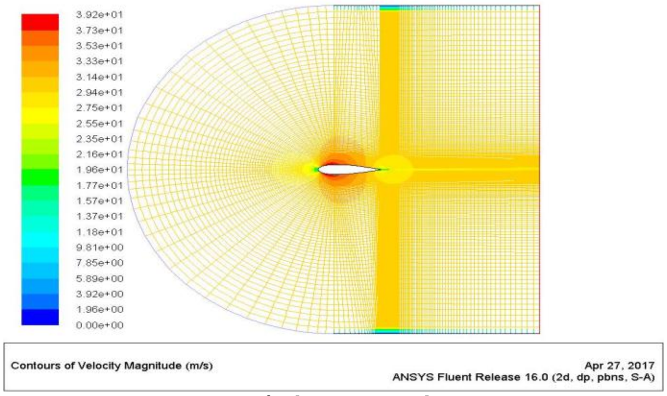

Fig 8. Contour of velocity magnitude at 10º AOA

Above figure shows the magnitude of velocity contour over an airfoils for 10º angle of attack. It is seen that flow separation takes place close to trailing edge. Hence lift generated is more compared to 16º angle of attack.

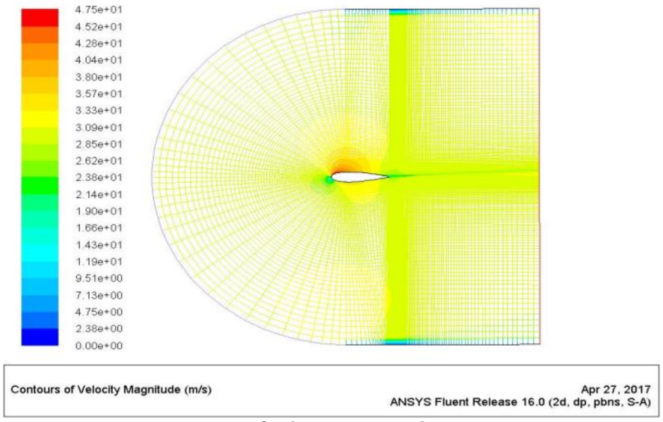

Fig 9.Contour of velocity magnitude at 14º AOA

Above figure shows the magnitude of velocity contour over an airfoil for 14º angle of attack. It is seen that flow is streamlined op to the trailing surface. Flow separation is very less. Hence, maximum lift is generated.

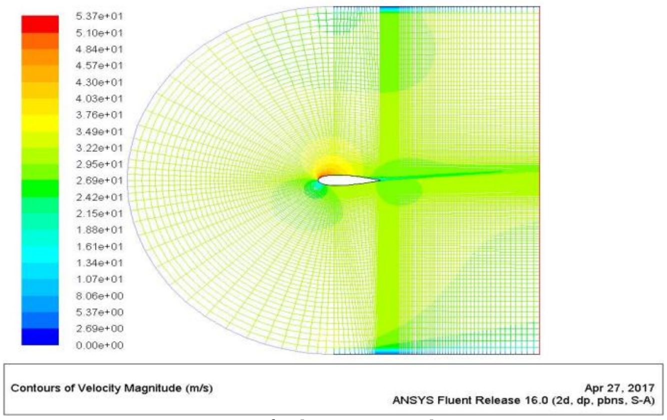

Fig 10. Contour of velocity magnitude at 16º AOA

Above figure shows the magnitude of velocity contour over an airfoil for 16º angle of attack. It is seen that flow separation takes place away from the trailing edge. Hence lift generated is less compared to 16º angle of attack.

And by conducting experiment on wind tunnel we obtained values of coefficient of lift and drag and compare it with values obtained from CFD to show that how precise CFD is. We obtain values of coefficient of Lift and Drag by using CFD and the values are as follows:

Angle Of Attack	C_L	C_D
-2	-0.65889	0.01594
0	-0.47815	0.017426
2	-018890	0.01557
5	0.17141	0.03884
8	0.49431	0.013542
10	0.69852	0.013031
13	0.80013	0.012667
14	0.90890	0.12790
15	0.92026	0.12532
16	0.87554	0.04377

Here we obtain values of coefficient of Lift and Drag experimentally and are as follows:

Angle of attack (α)	Area of Airfoil(A m²)	Lift force (F_L)	Drag force (F_D)	C_L	C_D	C_L/C_D
-2	0.048	-2.53	1	-0. 0939	0.01642	-5.72
0	0.048	0	1	0	0.01642	0
5	0.048	4	1	0.2532	0.01642	5.21
8	0.048	5	1	0.4	0.01642	8.07
10	0.048	6	1	0.5353	0.01642	10.65
12	0.048	7	1	0.6784	0.01642	12.01
14	0.048	8	1	0.8122	0.01642	12.27
15	0.048	6	1	0.8924	0.01642	5.89
16	0.048	6	1	0.8491	0.01642	4.84

Finally after comparing both of the above charts we can say that CFD is mature tool to predict the performance of airfoil section. CFD showed precise results which also obtain from experimentally.

Angle Of Attack	C_L by software	C_L by experimental	% error
-2	-0.65889	-0. 0939	0.8574
0	-0.47815	0	1
2	-018890	0.175	1.94
5	0.17141	0.2532	0.27
8	0.49431	0.4	0.19
10	0.69852	0.5353	0.233
13	0.80013	0.6784	0.152
14	0.90890	0.8122	0.106
15	0.92026	0.8924	0.0303
16	0.87554	0.8491	0.03

CONCLUSION:

The objective of the work was to understand the flow pattern over an airfoil and study the stalling effect using CFD. From the present study, it is seen that CFD contributes to the significant understanding of flow pattern over an airfoil. The following are the important conclusions drawn from the studies carried out in the present work.

International Journal of Engineering Research & Technology (IJERT)
ISSN: 2278-0181
Vol. 6 Issue 09, September - 2017

1) For airfoil with chord length 160 mm, the coefficient of lift increases from -0.47815 for 0° to 1.2026 for 15° angles of attack and again decreases to 0.8754 for 16°.

2) The values of CD and CL obtained from CFD are close to those obtained from wind tunnel test. The small variation is due to the leakage losses in the wind tunnel.

3) The maximum value of lift coefficient is obtained at the stall angle. The value of maximum lift goes on increasing with increasing the cord length.

4) After the stall angle, the flow separation occurs away from the trailing edge which reduces the lift generated.

REFERANCES

[1] Karna S. Patel, Saumil B. Patel, Utsav B. Patel, Prof. Ankit P. Ahuja, “CFD Analysis of an Aerofoil”, International Journal of Engineering Research, March 2014, ISSN:2319-6890, Volume No.3, Issue No.3, PP 154-158.

[2] Ashish Kadve, Dr. Prashant Sharma, and Abhishek Patel. “Review on CFD Analysis on aerodynamic dsigh optimization of wind turbine rotor blade”, International Journal of Innovation and Emerging Research in Engineering, February 2016, Volume 3, Issue 5, PP 178-183.

[3] P. B. Makwana, J. J. Makadiya, “CFD Analysis of Airfoil at High Angles of Attack”, International Journal of Engineering Research and Technology, April 2014, ISSN 2278-0181 Volume 3, Issue 4, PP 430-437.

[4] Anderson R F, the Aerodynamic Characteristics of Six Commonly Used Airfoils over a Large Range of Angle of Attack, National Advisory Committee for Aeronautics, 1931.