CFD Analysis of NACA 2415 and 23012 Airfoil

Radhakrishnan P M[1], Dheepthi M[2]

[1]*Master of Technology, Department of Aeronautical Engineering*
[2]*Master of Technology, Department of Aeronautical Engineering*
Hindustan Institute of Technology and Science, Chennai

Abstract

Two dimensional models for the airfoils were created, drawn and meshed in ANSYS Mechanical using geometry data gathered by the National Advisory Committee for Aeronautics. ANSYS FLUENT is used for computing flow over the two dimensional NACA 2415 and 23012 airfoil at an angle of attack of 4°. The computations are carried out at high Reynolds number at standard roughness conditions. One of the popular meshes for simulating an airfoil in a stream is a C-Mesh, and that is what we will be using. At the inlet of the system, we will define the velocity as entering at a 4° angle of attack, and at a total magnitude of 1. We will also define the gauge pressure at the inlet to be 0. As for the outlet, the only thing we can assume is that the gauge pressure is 0. As for the airfoil itself, we will treat it like a wall. From the analysis, C_P (Coefficient of Pressure) distribution, C_L (Coefficient of Lift) and C_D (Coefficient of Drag) is estimated and compared with the experimental results.

Keywords: Airfoils, CFD, Fluent, Lift, Drag

1. Introduction

An airfoil is the shape of a wing, blade (of a propeller, rotor, or turbine), or sail (as seen in cross-section). An airfoil shaped body moved through a fluid produces an aerodynamic force. The component of this force perpendicular to the direction of motion is called lift. The component parallel to the direction of motion is called drag. The lift on an airfoil is primarily the result of its angle of attack and shape. When oriented at a suitable angle, the airfoil deflects the oncoming air (for fixed-wing aircraft, a downward force), resulting in a force on the airfoil in the direction opposite to the deflection. This force is known as aerodynamic force and can be resolved into two components: lift and drag. Most foil shapes require a positive angle of attack to generate lift, but cambered airfoils can generate lift at zero angle of attack.

2. NACA Series

The NACA airfoils are airfoil shapes for aircraft wings developed by the National Advisory Committee for Aeronautics (NACA). The shape of the NACA airfoils is described using a series of digits following the word "NACA". There are different types of NACA series namely NACA four-digit series, NACA five-digit series and NACA six-digit series.

2.1 NACA 2412

The NACA 2412 airfoil has a maximum camber of 2% of chord located at 40% of chord from the leading edge with a maximum thickness of 12% of the chord. Four-digit series airfoils by default have maximum thickness at 30% of the chord from the leading edge.

2.2 NACA 23012

The NACA 23012 airfoil has a maximum camber of 1.8% of chord located at 12.7% of chord from the leading edge with a maximum thickness of 12% of the chord located at 29.8% of chord.

3. Pre – Processor

3.1 Geometry & Grid Generation

The 2-Dimensional airfoil geometries were created and meshing was done in Ansys. The airfoil is assumed to have a chord length of 1 meter with the part of the airfoil at (0, 0, 0) and the trailing edge is at (1, 0, 0). One of the popular meshes for simulating an airfoil in a stream is a C-Mesh which is shown in Figure 2.
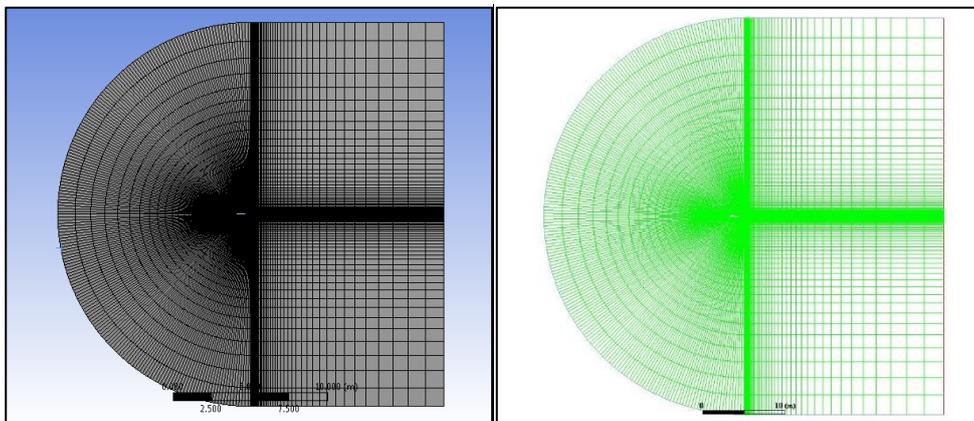

Figure 2: Grid Generation

3.2 Initial & Boundary Conditions

The computational domain extends far upstream of the airfoil where the boundary condition are defined as velocity inlet, outlet was defined as pressure outlet and airfoil was defined as wall. The problem considers flow around an airfoil at 4° angle of attack. Here we consider only two dimensional flow therefore only x direction velocity and y direction velocity which is shown in Table 1.

Table 1: Initial Conditions

Inlet	X velocity	0.9975 m/s
	Y velocity	0.0697 m/s
	Z velocity	0
	Magnitude	1
	Gauge Pressure	0
Outlet	Gauge Pressure	0

3.2 Turbulence Model

The standard k-ε model falls within this class of turbulence model and has been used in present case. The second order upwind scheme and Pressure-Velocity scheme has been used to solve the flow variables. FLUENT solver is set at some specific parameters which are tabulated in Table 2.

Table 2: Fluent Solver Description

Solver gradient	Least square cell based
Momentum	Second order upwind
Turbulent kinetic energy	Second order upwind
Turbulent dissipation	Second order upwind

4. Results and Discussions

At an angle of attack of 4°, flow is simulated over the airfoil to calculate the contours of static pressure, velocity vector and turbulence kinetic energy. Also to estimate the coefficient of lift, coefficient of drag and coefficient of pressure distribution.

4.1 Contours of Static Pressure

Flow has a stagnation point just under the leading edge and hence producing lift as there is a low pressure region on the upper surface of the foil as shown in Figure 3 and Figure 4. We can also observe that Bernoulli's principle is holding true; the velocity is high (denoted by the red contours) at the low pressure region and vice-versa. There is a region of high pressure at the leading edge (stagnation point) and region of low pressure on the upper surface of airfoil.
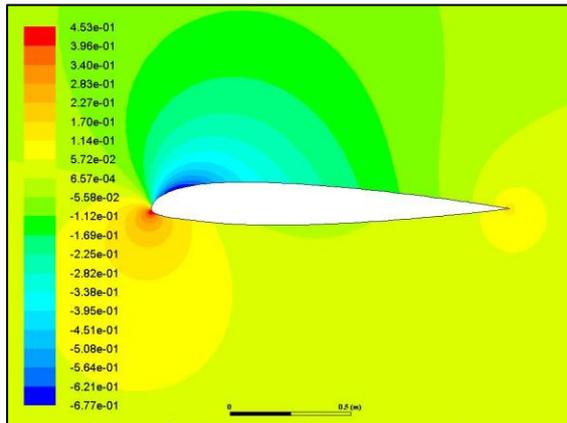
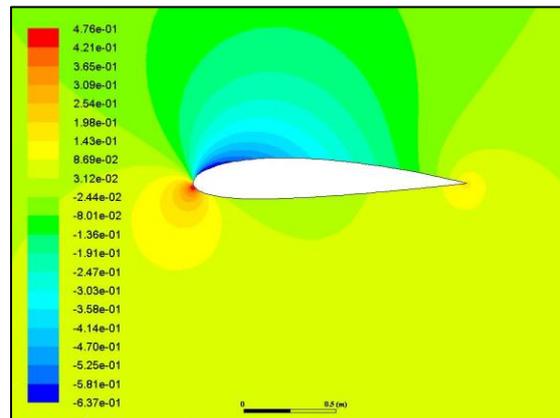**Figure 3:** Contour of Static Pressure - NACA 23012

Figure 4: Contour of Static Pressure - NACA 2415

4.2 Contours of Velocity Vector

At 4˚ angle of attack the stagnation point is slightly shift towards the trailing edge via bottom surface hence it will create low velocity region at lower side of the airfoil and higher velocity acceleration region at the upper side of the airfoil (as seen in Figure 5 and Figure 6) and according to principle of Bernoulli's upper surface will gain low pressure and lower surface will gain higher pressure. Hence value of coefficient of lift will increase and coefficient of drag will also increase but the increasing in drag is low compare to increasing in lift force. Here high velocity is seen near the leading edge at upper side of the airfoil and low velocity is seen near the leading edge at lower side of the airfoil.
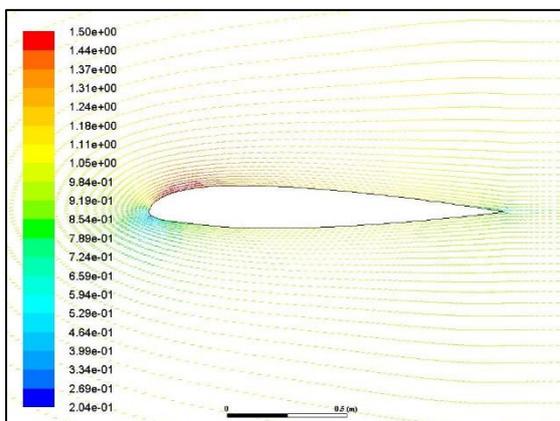

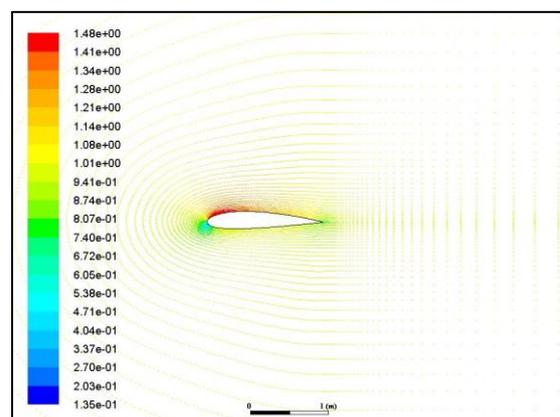**Figure 5:** Contour of Velocity Vector – NACA 23012

Figure 6: Contour of Velocity Vector – NACA 2415

Radhakrishnan P M and Dheepthi M

4.3 Contours of Turbulent Kinetic Energy

At 4° angle of attack, turbulence is carried out under second order upwind scheme and least square cell based scheme. Turbulence of the flow is defined by using Reynolds number, where Reynolds number is directly proportional to velocity. Here the turbulence is high over the upper surface of the airfoil because velocity is high near the upper surface of the airfoil and the turbulence is low near the lower surface of the airfoil because velocity is low near the lower surface of the airfoil which is shown in the Figure 7 and Figure 8.
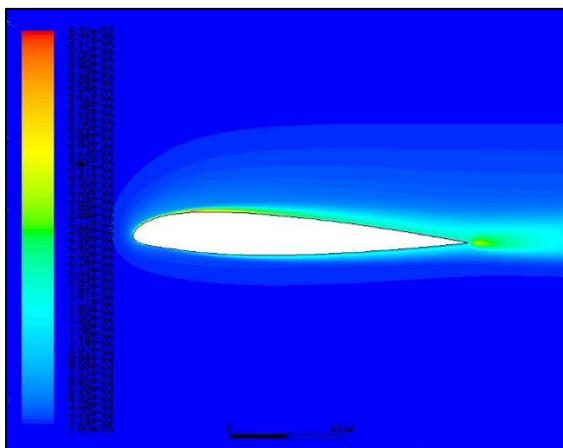

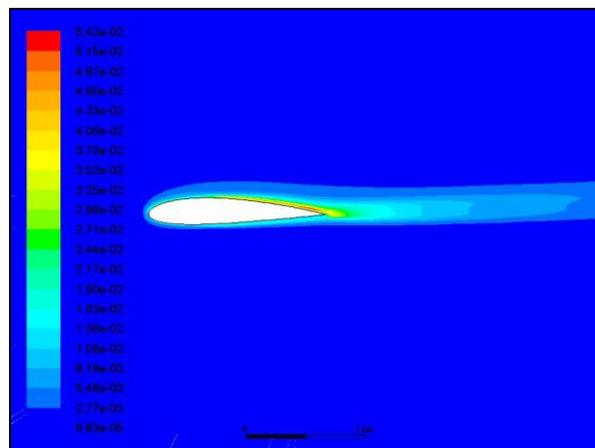**Figure 7:** Contour of Turbulent Kinetic Energy – NACA 23012

Figure 8: Contour Turbulent Kinetic Energy – NACA 2415

4.4 Coefficient of Pressure

The coefficient of pressure (C_P) is a dimensionless number which describes the relative pressures throughout the flow field. This pressure distribution is simply the pressure at all points around an airfoil. Typically, graphs of these distributions are drawn so that negative numbers are higher on the graph, as the C_P for the upper surface of the airfoil will usually be farther below zero and will hence be the top line on the graph.

If the airfoil is placed at some angle of attack in a moving fluid, pressure variation occurs over the airfoil. Since the airfoil is kept at 4° angle of attack in a moving fluid (air), there occurs pressure variation over the airfoil. Pressure distribution is calculated over the airfoils which is shown in Figure 9 and Figure 10.
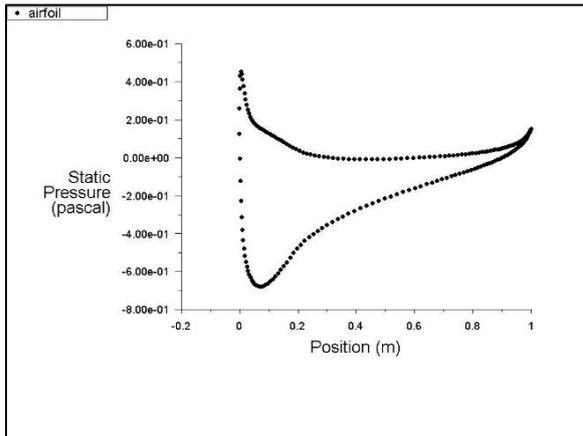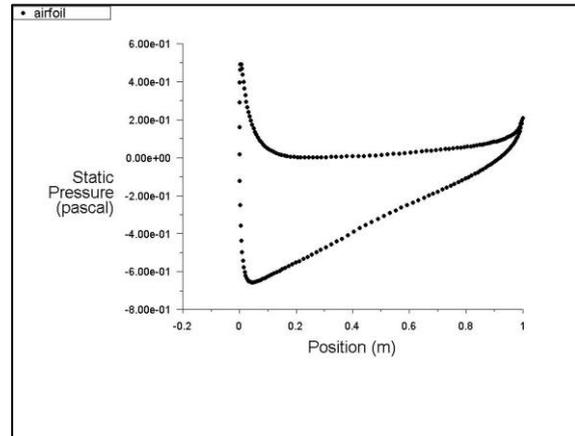
Figure 9: C_P Distribution – NACA 23012

Figure 10: C_P Distribution – NACA 2415

4.5 Coefficient of Lift

The coefficient of lift (C_L) is a dimensionless coefficient that relates the lift generated by an airfoil to the fluid density around the body. If the airfoil is placed at some angle of attack in a moving fluid, it will create some lift. Since the airfoil is placed at 4° angle of attack, it generates lift. The coefficient of lift is calculated which is given in Table 3.

4.5 Coefficient of Drag

The coefficient of drag (C_D) is a dimensionless coefficient that relates the drag generated by an airfoil to the fluid density around the body. If the airfoil is placed at some angle of attack in a moving fluid, it will create some drag. Since the airfoil is placed at 4° angle of attack, it generates drag. The coefficient of drag is calculated which is given in Table 3.

Table 3: Estimated C_L and C_D

NACA	C_L	C_D
23012	0.59081796	0.00368160
2415	0.35908761	0.00210803

Radhakrishnan P M and Dheepthi M

5. Conclusion

Analysis of NACA 23012 and NACA 2415 airfoil is carried out using ANSYS FLUENT. The simulation captures all the important features of the flow. A special care was taken to build good quality mesh with grid clustering to capture flow separation. The contours of static pressure, velocity vector, turbulence kinetic energy and the coefficient of lift, coefficient of drag and coefficient of pressure distribution is calculated. Results concluded that coefficient of pressure is maximum at leading edge and trailing edge while lower at thick surfaces. Since the angle of attack is 4°, the velocity of the fluid is high above the airfoil, which is mainly due to the angle of attack, therefore lift and drag is generated.

6. References

[1] Abbott IH, Von Doenhoff AE (1959). Theory of Wing Sections. Dover Publishing, New York.

[2] Bacha WA, Ghaly WS (2006). Drag Prediction in Transitional Flow over Two-Dimensional Airfoils, Proceedings of the 44th AIAA Aerospace Sciences Meeting and Exhibit, Reno, NV.

[3] Badran O (2008). Formulation of Two-Equation Turbulence Models for Turbulent Flow over a NACA 4412 Airfoil at Angle of Attack 15 Degree, 6th International Colloquium on Bluff Bodies Aerodynamics and Applications, Milano, 20-24 July.

[4] Fluent Inc. (2006), Fluent 6.3 User's Guide.

[5] Johansen J (1997). Prediction of Laminar/Turbulent Transition in Airfoil Flows. Risø National Laboratory, Roskilde, Denmark.

[6] Menter FR (1994). Two-Equation Eddy-Viscosity Turbulence Models for Engineering Applications. AIAA J., 32: 1598-1605.

[7] McCroskey WJ (1987). A Critical Assessment of Wind Tunnel Results for the NACA 0012 Airfoil. U.S. Army Aviation Research and Technology Activity, Nasa Technical Memorandum, 42: 285-330.