Analysis of NACA 4412 Airfoil using Computational Fluid Dynamics in ANSYS Software

Kaushal Shrivastava[1] Prof. Sandeep Jainb[2] Dr. Lokesh Bajpaic[3]
[1]M. Tech. Student [2]Associate Professor [3]Head of Dept.
[1,2,3]Department of Mechanical Engineering
[1,2,3]Samrat Ashok Technological Institute, Vidisha 464001, (Madhya Pradesh) India

Abstract— This paper focuses primarily on some specific parameters (like coefficient of drag, coefficient of lift, pressure and velocity variation) that mainly effects aerodynamic efficiency of aerofoil. The airfoil of a plane has crucial influence on its aerodynamic efficiency, hence the selection of a suitable airfoil section for the plane is of utmost importance. In this paper, National Advisory Committee for Aeronautics (NACA) 4412 airfoil profile is considered for analysis of its various parameters. The geometry of the airfoil is created using ANSYS16.2 and CFD analysis is carried out using CFX at 4^0 and 8^0 angle of attack (AOA). The coefficient of lift and drag values are calculated for 1×10^5 Reynolds number.

Key words: Airfoil, Coefficient of Drag, Coefficient of Lift, CFD

I. Introduction

An airfoil means a two dimensional cross-section shape of a wing whose purpose is to generate lift when exposed to a moving fluid. The cross sections of wings, windmill blades , propeller blades, compressor and turbine blades in a jet engine, and hydrofoils are example of airfoils. The geometry of an airfoil is shown in Figure 1. [1]

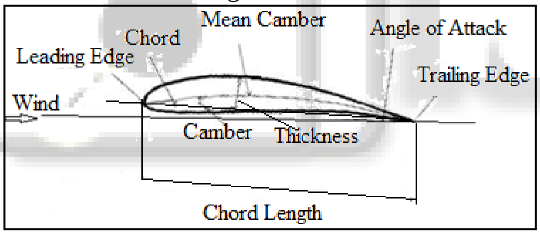

Fig. 1: Diagram of airfoil's geometry

II. Lift and Drag

Lift on a body is characterized as the power on the body in a direction normal to the stream direction. Lift can be generated, if the fluid flows in a circulatory stream around the body, for example, lift exists around a spinning cylinder[2]. The speed above the body is increased thus the static pressure is decreased while speed underneath is slowed down, giving an increase in static pressure, by Bernoulli's theorem. In this way, there is a normal force generated in the upward direction which is called the lift force.

The drag on a body in an incoming flow is defined as the force on the body in a parallel flow direction. For a plane to operate efficiently, the lift force should be high and drag force should be low. Lift and drag force are shown in figure 2 [3]. For small angle of attack, lift force is high and drag force is low. If the angle of attack (α) increases beyond a certain value, both lift force and drag force increases, hence proper angle of attack (AOA) should be known to have an optimum lift to drag ratio.

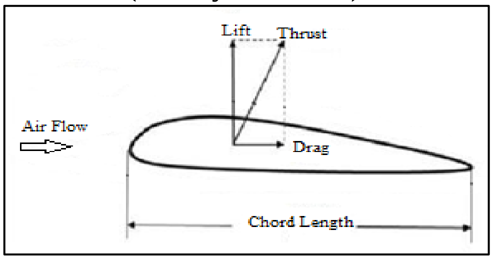

Fig. 2: Forces on a stationary airfoil in airflow

III. Airfoil Description

The word 'airfoil' is an Americanization of the British term 'aerofoil' which itself is derived from the two Greek words Aeros ("of the air") and Phyllon ("leaf"), or "air leaf"[4].

The NACA four digit wing sections define the profile[5] as :

1) First digit from left describes the maximum camber as percentage of the chord length.
2) Second digit from left describes the distance of maximum camber from the airfoil leading edge in tens of percents of the chord.
3) Last two digits describes the maximum thickness of the airfoil as percent of the chord.

The airplane like Luscombe 8E, z326 uses NACA 4412 aerofoil. This profile has also been used in some sports airplane and aircraft, the example involves[6] aeronca series aircraft like aeronca 65-tac defender, AAI AA-2 MAMBA aircraft, aeronca 11ac chief etc.

IV. Modeling and Simulation

The data for airfoil is taken from NACA database where various airfoil data for wind turbine, aeroplane are stored. Then, data of NACA 4412 profile, which is used in z326 airplane, is imported to Ansys Design Modeler and a curve is generated. Thereafter, a surface is generated to complete the modelling of the airfoil. Airfoil used in this study is shown here at 4^0 angle of attack (AOA) in figure 3 while the points are shown at 0^0 AOA around the foil.

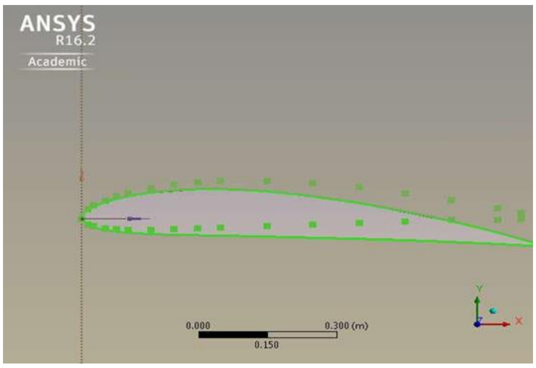

Fig. 3: Geometry of Airfoil created in ANSYS 16.2

For the flow analysis of the AIRFOIL, mesh is created in the ANSYS16.2 CFX software. Figure 4 shows the meshing drawing of the airfoil having 'sweep meshing' with 'body of influence' around the close vicinity of airfoil. The 'edge sizing' having number of divisions of 500 and 12 'inflation layers' with growth rate of 1.2, to make results more close to its realistic condition, is also added.

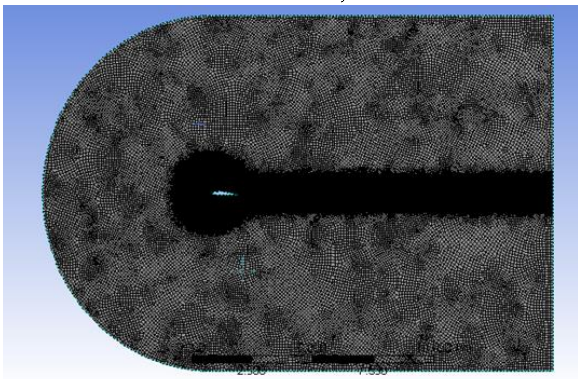

Fig. 4: Meshed diagram

The shape of the boundary for meshing is chosen as C shaped boundary at the leading edge and the rectangular shaped at the trailing edge. The number of nodes and elements are 226984 and 113207 respectively, which are of 'quad' and 'tri'shape.

'Pressure based steady state' solver is used which has wide range of its applicability from low speed incompressible flow to high speed compressible flow regimes. It can solve the momentum equation and pressure correction equation in segregated and coupled manner. The pressure velocity coupling is chosen as 'coupled' in this study, which is more robust and efficient single phase implementation technique for steady state flows with better performance as compared to segregated schemes. The momentum under spatial discretization is chosen as 'second order upwind' which is more complex and takes more time to get converges to the solution, but more accurate as compared to first order upwind. The turbulence model is set at 'Shear Stress Transport (SST), k-omega' which is three-equations eddy viscosity model and has good behaviour in adverse pressure gradient and separating flow regimes.

CFX solver is set at some specific parameters which are as follows in table 1:

Solver	Pressure based steady state
Turbulence model	Shear Stress Transport (SST)
Pressure velocity coupling	Coupled
Momentum	Second order upwind
Chord-length	1 m
Inlet velocity	50 m/s
Turbulent viscosity ratio	10
Viscosity (kg/m-s)	1.7894
Density (kg/m^3)	1.225

Table 1: CFX solver description

V. RESULTS AND DISCUSSIONS

The force vectors are obtained on the airfoil after simulation at both AOA's which shows the increased value of vectors at higher angle of attack but there is also increment in drag force. The lift force obtained is 12.9116 N at 4^0 AOA with 0.282938 N as drag force and 45.634 as lift to drag ratio, while at 8^0 AOA the value of lift and drag ratio are 18.7676 N and 0.463497 N respectively with 40.49 as lift to drag ratio.

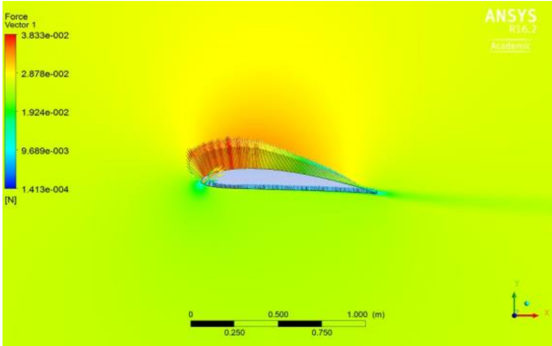

Fig. 5: Force Vectors at 4^0AOA

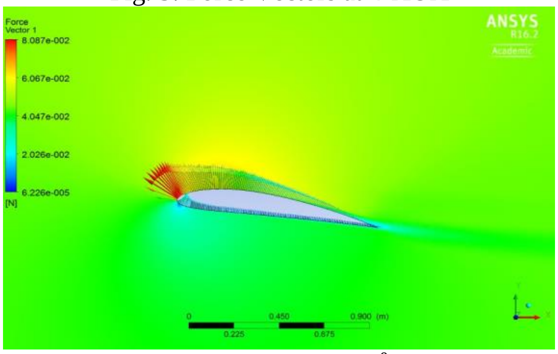

Fig. 6: Force Vectors at 8^0 AOA

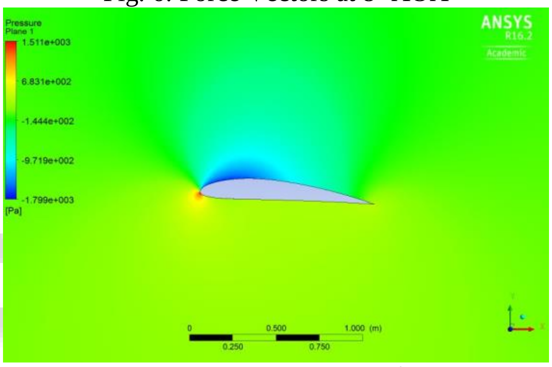

Fig. 7: Pressure variation at 4^0 AOA

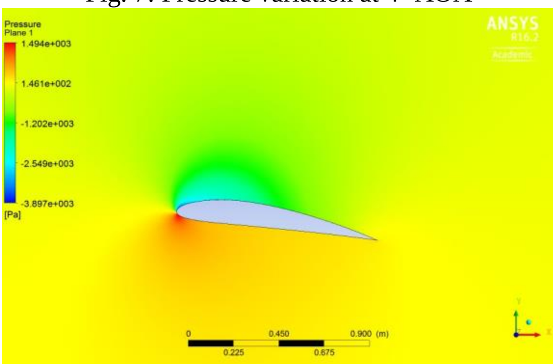

Fig. 8: Pressure variation at 8^0 AOA

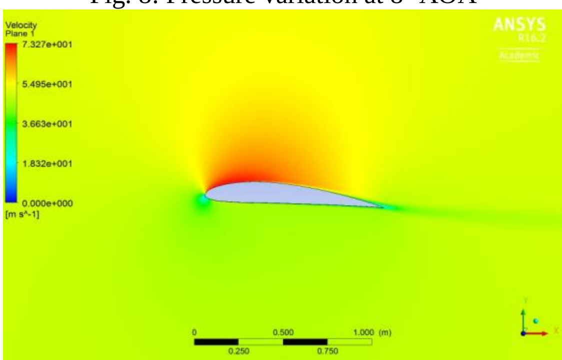

Fig. 9: Velocity at 4^0 AOA

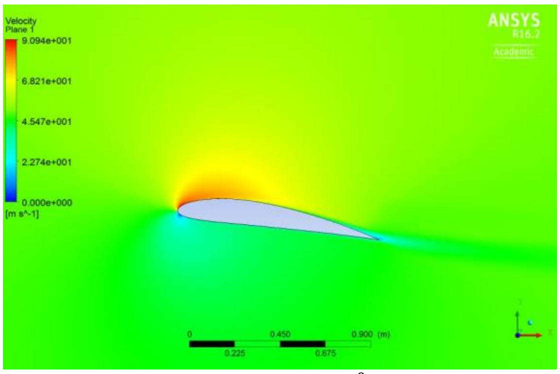

Fig. 10: Velocity at 8^0 AOA

VI. CONCLUSION

From the pressure diagram, it is concluded that there is a region of high pressure at the leading edge (stagnation point) and region of low pressure on the upper surface of airfoil. From Bernoulli's equation, it is clear that whenever there is high velocity, there is low pressure and vice versa[7]. The pressure on the lower surface of the airfoil is greater than that of the incoming flow stream and as a result, it effectively pushed the airfoil upward, normal to the incoming flow stream. The pressure variation diagram clearly shows the increase in pressure in the lower region of the airfoil at 80 angle of attack(AOA) as compare to that of 40 AOA which leads to increase in the lift coefficient.

With the result of lift force, drag force and lift to drag force at different AOA's it can be concluded that on increasing angle of attack though lift force increases but of drag force also increases as lift to drag rato is decreased.

REFERENCES

[1] S.Kandwal, S. Singh. "Computational Fluid Dynamics Study of Fluid Flow and Aerodynamic Forces on an Airfoil". IJERT, pp 1-7. 2012.
[2] Monir Chandrala, Abhishek Choubey, Bharat Gupta. "Aerodynamic Analysis of Horizontal Axis Wind Turbine Blade". IJERA, pp 1244-1248.2012.
[3] Nathan logsdon. "A procedure for numerically analyzing Airfoils and Wing sections". Thesis 1-53. 2006.
[4] Sunada S."Comparison of wing characterstics at an ultra low Reynolds number". Journal of aircraft, pp 331-338. 2002.
[5] Ngoc Anh Vu, Jae Woo Lee, Jung Il Shu. "Aerodynamic design optimization of helicopter rotor blades including airfoil shape for hover performance". Chinese journal of Aeronautics, pp 1-8. 2013.
[6] Kaushik Devmurari, "A Review on Design and Aerodynamic Behavior of Airfoil". IJSRD, pp 2127-2129. 2015
[7] Kulfan BM. "Universal parametric geometry representation method". Journal of Aircraft, pp 142-158.2008

