CFD ANALYSIS OF WIND TURBINE AIRFOIL

[1]Sanket S. Unde, [2]Kailasnath B. Sutar*

[1]Researcher, [2]Associate Professor

[1,2]Department of Mechanical Engineering

[1,2]Bharati Vidyapeeth (Deemed to be University), College of Engineering, Pune, India

Abstract: This paper describes the simulation study of the INDTH 4412 wind turbine airfoil at angle of attack 0° to 15°, keeping velocity constant. The aerodynamics of the airfoil which is used for wind turbine blade has a striking influence on the overall aerodynamic efficiency of wind turbine. Geometric modelling of the airfoil and its CFD simulation is carried out on ANSYS Fluent. For a computational domain, separate faces of enclosure were projected and edge meshing was selected in a manner to create fine mesh near the airfoil.

Keywords - Airfoil, Lift Coefficient, Drag Coefficient, Angle of Attack

I. INTRODUCTION

Airfoil is defined as the cross section of a body held stationary in a real fluid, which is flowing at a uniform velocity '*V*' in order to create a convenient aerodynamic force. The cross sections of wind turbine blades, compressor blades, aircraft wings, windmill blades, hydrofoils and propeller blades are examples of airfoil. The most critical part in wind turbine design is shape of an airfoil. Wind turbines convert the kinetic energy in the wind into mechanical energy. The shape of an airfoil can be defined by several parameters shown in figure 1.

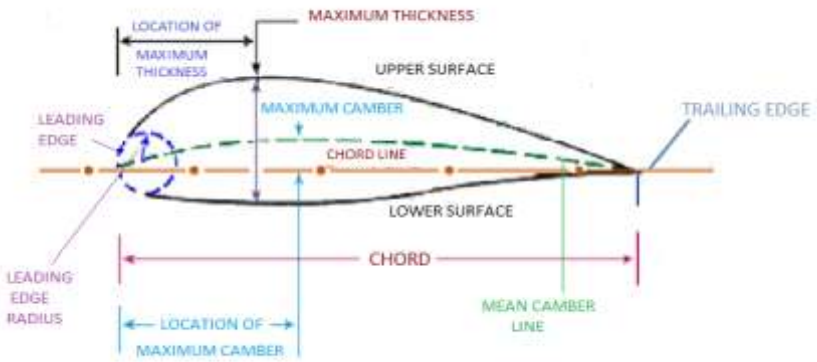


Figure 1: Airfoil terminology

All parameters are defined as follows:

- Chord line: Straight line drawn from leading edge to trailing edge.
- Chord: Length of chord line.
- Mean camber line: Curve line drawn from leading to trailing edge. It is equidistance from lower surface to upper surface of an airfoil.
- Maximum camber: Maximum distance between mean camber line and chord line.
- Leading edge: The front edge of an airfoil.
- Trailing edge: The rearmost edge of an airfoil.
- Angle of attack: Angle between chord line and air flow.

Wind energy is form of solar energy [1]. Dash [2] analysed the NACA0012 wind turbine airfoil at various angle of attack, at constant Reynolds number and found CFD analysis as an efficient alternative to experimental method. Yao *et al.* [3] compared the aerodynamic performance of airfoil to experimental data. Patil et al. [4] analysed the effect of low Reynolds number on drag and lift for the wind turbine blade and found that as Reynolds number increases, lift and drag coefficient also increases. Eleni et al. [5] studied the various turbulent models and found that, the turbulent model used in commercial CFD does not give accurate result at high angle of attack. Ameku et al. [6] designed 3 kW wind turbine generator with thin airfoil blade and found that average generator output power is 1105 W and power coefficient is 0.14.

Referring to the existing literature, the present study aims to analyses flow field for an air foil INDTH 4412, developed for small wind turbine blades [7]. The 2D CFD simulations were performed at various angles of attack considering constant velocity of 12 m/s. Steady state governing equations of continuity and momentum where solved using standard k- ε (2 equation) turbulence model.

II. LIFT AND DRAG

Airflow over an airfoil portion creates two types of aerodynamic forces viz. drag force and lift force [8]. Lift on body is defined as the component of total force (F_R) in the direction perpendicular to the direction of motion. Thus the lift force exerted by the fluid in the direction perpendicular to the direction of motion. Lift force occurs only when axis of the body is inclined to the direction of flow. Drag on the body is defined as the component of total force (F_R) in the direction of motion. Thus the drag force exerted by the fluid in the direction of motion [9]. The resultant of both lift and drag forces act at distance *C/4* from the leading edge (figure 2).

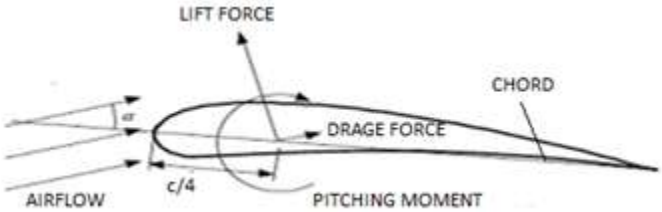


Figure 2: Lift and drag forces on a stationary airfoil

Lift coefficient:

$$C_l = \frac{\frac{\text{Lift force}}{\text{Unit length}}}{\frac{\text{Dynamic force}}{\text{Unit length}}} = \frac{\frac{L}{l}}{\frac{1}{2}\rho V^2 C} \quad (1)$$

Drag coefficient:

$$C_d = \frac{\frac{\text{Drag force}}{\text{Unit length}}}{\frac{\text{Dynamic force}}{\text{Unit length}}} = \frac{\frac{D}{l}}{\frac{1}{2}\rho V^2 C} \quad (2)$$

Momentum coefficient:

$$C_m = \frac{\text{Pitching moment}}{\text{Dynamic moment}} = \frac{M}{\frac{1}{2}\rho V^2 AC} \quad (3)$$

Where;

ρ - Density of air
μ – Viscosity of fluid
V – Air stream velocity
A – Projected area
C – Chord length
l - Airfoil span

III. Simulation On An Airfoil

CFD analysis of the airfoil was done in following steps:

3.1 Geometry Creation

The geometry was performed in ANSYS DesignModeler. It was created with 3D curve coordinates and C mesh domain was drawn for discretization of the computational domain. Figure 3 shows computational domain of the airfoil.

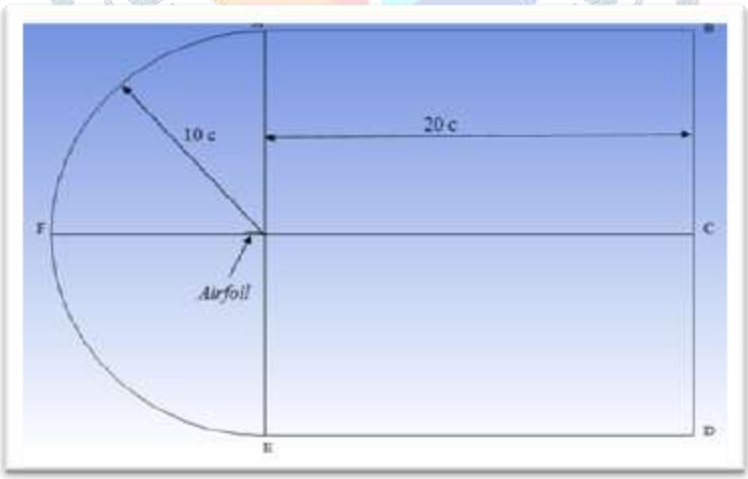


Figure 3 Computational domain of airfoil INDTH 4412

3.2 Mesh Generations

Meshing was performed in ANSYS AUTODYN PrepPost. Mesh are made using quadrilateral dominant method. The entire model is meshed into Quad and Tri element. The model is meshed with 14892 nodes and 14600 elements shown in figure 4. Inlet and Outlet boundaries and the Airfoil in ANSYS is shown in figure 5.

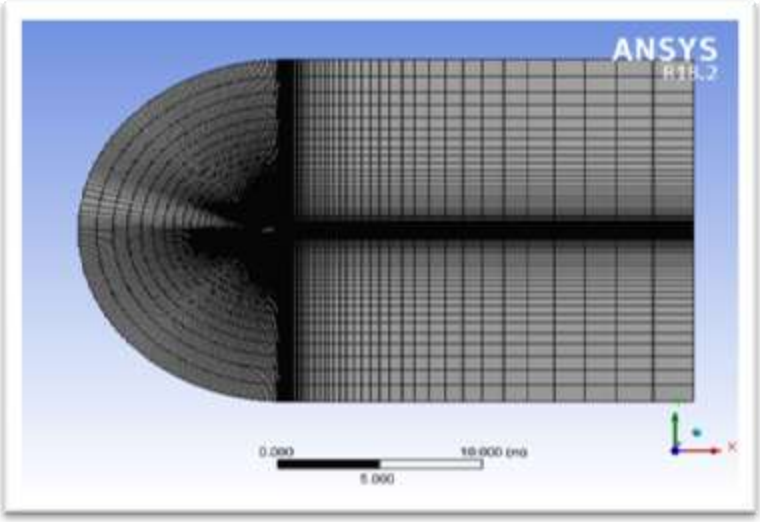


Figure 4 Meshing domain of airfoil INDTH 4412

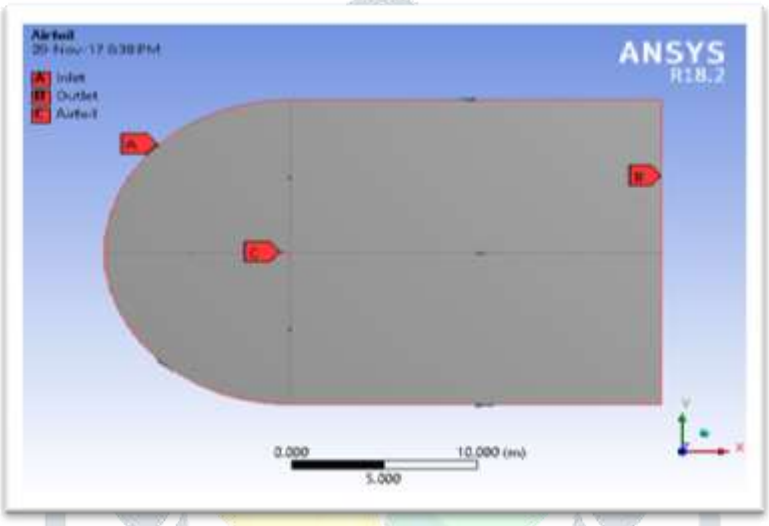


Figure 5 Inlet, outlet and airfoil in ANSYS

3.3 Input And Boundary Conditions

Input and boundary conditions used during CFD simulation of the air foil are shown in table 1.

Table 1 Fluent setup parameters

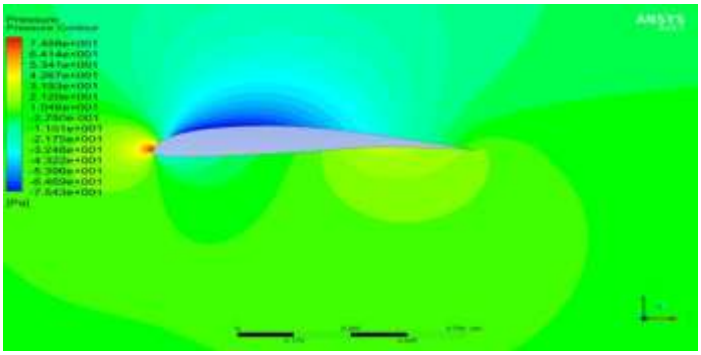
Air foil	INDTH 4412
Solver type	Density based
Time	Steady
Velocity formulation	Absolute
Model	Viscous-Standard k-ε (2 equation), Standard Wall Function
Air Density	1.22 kg/m^3
Kinetic Viscosity	1.7894 × 10^{-5} kg/m-s
Temperature	288 K
Wind Speed	12 m/s
Method	Gradient (Least Square Cell Based) Flow (Second Order Upwind) Turbulent Kinetic Energy (Second Order Upwind) Turbulent Dissipation Rate (Second Order Upwind)
Boundary Condition	Velocity Inlet Pressure Outlet Stationary wall with no sleep shear condition

IV. Results And Discussions

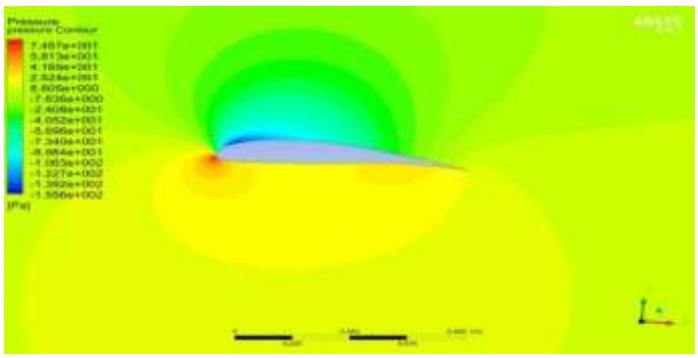
4.1 Distribution of pressure on the airfoil

The contours of pressure distribution at different angles of attack obtained from CFD simulation are shown in figure 6 (a-d). The simulation was carried out using density based solver. It was found that there is a region of high pressure at the leading edge (stagnation point) and region of low pressure on the upper portion of the airfoil. According to Bernoulli's principle, whenever there is high velocity, we have low pressure and vice versa. With increase in angle of attack, maximum pressure is observed at lower side of the airfoil. As the pressure

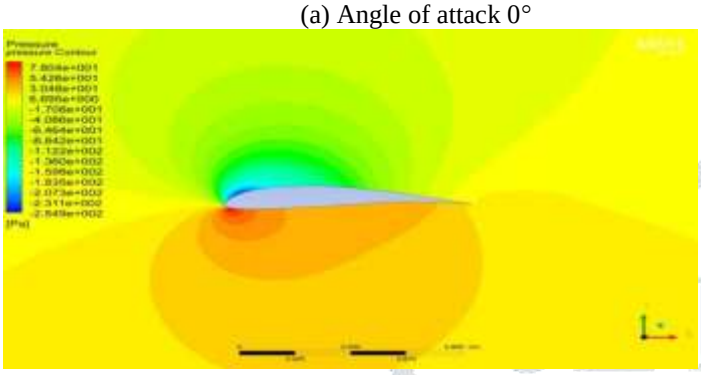
on the lower side is higher than that of the incoming flow stream, the airfoil is forced upward normal to the incoming flow stream on account of lift force.



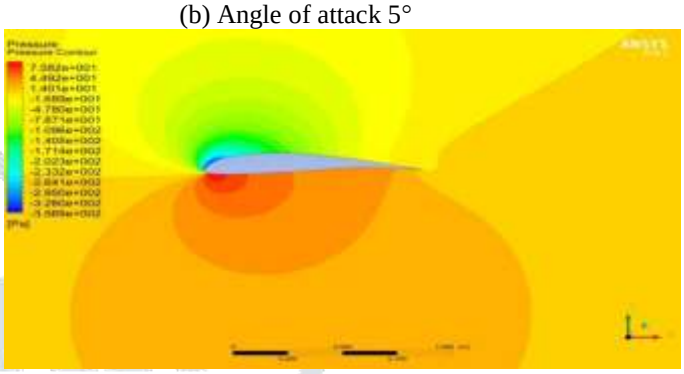
(a) Angle of attack 0°



(b) Angle of attack 5°

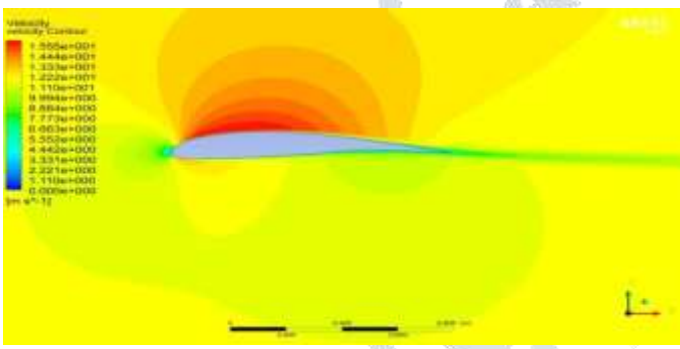


(c) Angle of attack 10°

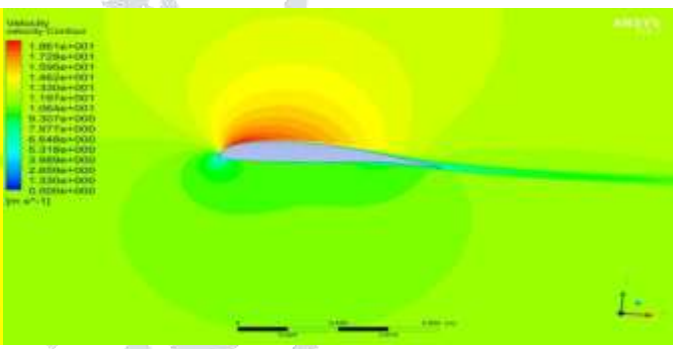


(d) Angle of attack 15°

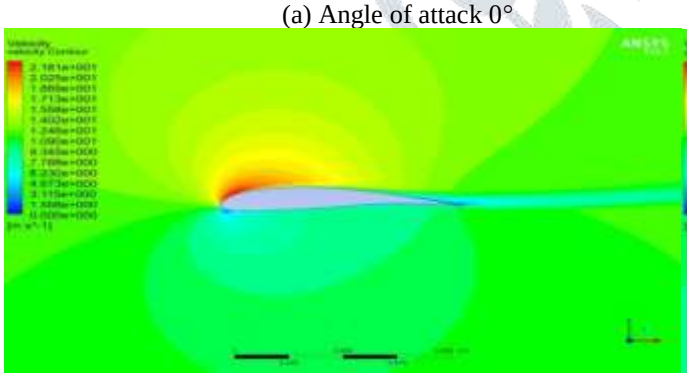
Figure 6: Pressure contours at different angles of attack



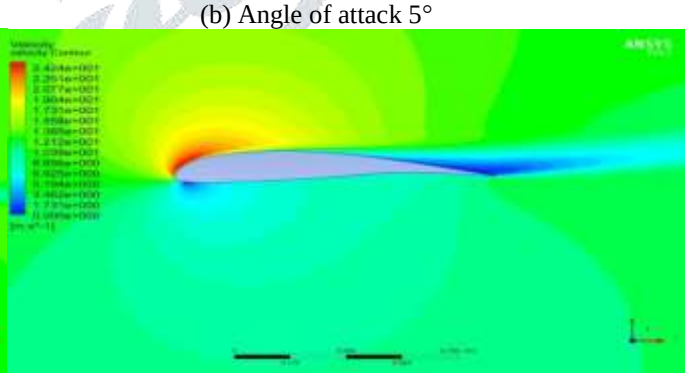
(a) Angle of attack 0°



(b) Angle of attack 5°



(c) Angle of attack 10°



(d) Angle of attack 15°

Figure 7: Velocity contours at different angles of attack

4.2 Distribution of velocity on the airfoil

From 7 (a-d), at the leading edge (stagnation point) the velocity of flow is almost zero. The flow accelerates on the upper portion of the airfoil and the velocity of flow decreases along the lower portion. Generation of back flow (vertex) at trailing edge is observed as angle of attack near to 10° and so on.

4.3 Curve of lift/drag coefficient

The coefficient of lift and drag is calculated at angle of attack 0° to 15° using the standard k- ε (2 equation) turbulence model. The figure 8 shows that coefficient of lift/drag ratio increases with increase in angle of attack up to 5° and above this angle, coefficients of lift/drag ratio decreases with increase in angle of attack.

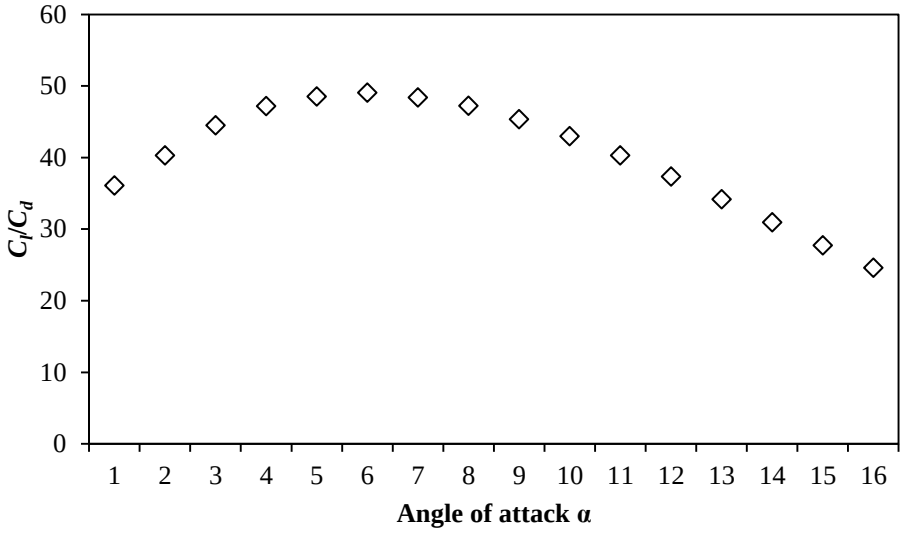


Figure 8: Graph of lift/drag coefficient versus angle of attack for an airfoil

V. Conclusion

Analysis of aerodynamic performance of INDTH 4412 airfoil was carried out using ANSYS-Fluent at angle of attack 0° to 15° with constant velocity of 12 m/s using the standard k- ε (2 equation) turbulence model. It was found that there was a region of low pressure on the upper portion and the high pressure on lower portion of the airfoil. The velocity on the upper portion of the airfoil was higher than the lower portion. It was observed that there was increase in lift to drag ratio till α = 5° and with further increase in angle of attack, there was decrease in lift to drag ratio.

References

[1] Wind energy, https://openei.org/wiki/Wind_energy Cited on 2nd October, 2017

[2] Dash, Ankan. "CFD Analysis ofWind Turbine Airfoil at Various Angles of Attack."

[3] Yao,Ji, Weibin Yuan, Jianbin Xie, Haipeng Zhou, Mingjun Peng and Yong Sun. "Numerical simulation of aerodynamic performance for two dimensional wind turbine airfoils." *Procedia Engineering* 31(2012): 80-86.

[4] Patil, Bhushan S., and Hitesh R. Thakare. "Computational Fluid Dynamics Analysis of Wind Turbine Blade at Various Angles of Attack and Different Reynolds Number." *Procedia Engineering* 127 (2015): 1363-1369.

[5] Eleni, Douvi C., Tsavalos I. Athanasios, and Margaris P. Dionissios. "Evaluation of the turbulence models for the simulation of the flow over a National Advisory Committee for Aeronautics (NACA) 0012 airfoil." *Journal of Mechanical Engineering Research* 4, no. 3 (2012): 100-111.

[6] Ameku, Kazumasa, Baku M. Nagai, and Jitendro Nath Roy. "Design of a 3 kW wind turbine generator with thin airfoil blades." *Experimental thermal and fluid science* 32, no. 8 (2008): 1723-1730.

[7] Dhurpate Priyanka. "Development of an airfoil for small wind turbine blades." M. Tech. Dissertation report. Bharati Vidyapeeth (Deemed to be University) College of Engineering, Pune, India. June 2016.

[8] Mechanics, Dr RK Bansal-Fluid. "Hydraulic Machines." (2009).

[9] Manwell, James F., Jon G. McGowan, and Anthony L. Rogers. *Wind energy explained: theory, design and application*. John Wiley & Sons, 2010.