Computational Study On Blade Of Micro-Capacity Wind Turbine

Roshan S. Kodare, Dr. Kailasnath B. Sutar

Abstract— Nowadays electricity is one of the most important basic needs for the human being. We have to shift from conventional to non-conventional energy resources, since conventional energy resources will deplete day by day. Among the renewable energy sources, the cheapest is the wind energy and available in large intensity. Electricity which is produced by wind turbine is having without damaging natural balance and can be available in affordable cost. Field testing of HAWT wind turbine is carried out in order to calculate the output power of wind turbine and it is compared with the theoretical power. This paper describes the Geometric modelling of Wind Turbine Blade using INDTH 4412 airfoil. Optimum angle of attack for INDTH 4412 airfoil was found to be 4°, and the Computational study for the Blade is carried out at 4° angle of attack. C-mesh domain is used to discretize the computational domain. Computational Fluid Dynamic (CFD) analysis is performed on the blade and study is carried to determine lift and drag coefficient, results of which help to improve performance of Wind turbine.

Index Terms— Renewable energy; Wind Energy; HAWT; CFD; Blade Geometry; Lift coefficient; Drag coefficient;

1 Introduction

1.1 Energy Resources

A key factor that influence the sustainable development of any nation is Energy. It is a vital building block in human development.[1] Energy resources may be classified as primary resources, suitable for end use without conversion to another form and secondary resources, where substantial conversion of energy from a primary source to usable form of energy is required. Based on the time required to regenerate an energy resource they are classified as Renewable resources and Non-renewable resources. Nonrenewable energy sources include natural gas, petroleum made from crude oil and natural gas liquids, coal and uranium. Most of the energy in use is supplied by these non-renewable energy sources, but they take a very long time to form and hence their supplies are limited. Renewable energy sources include biomass, solar energy, wind energy, geothermal energy and hydropower. Unlike non-renewable resources they are replenished naturally in a short period of time. Day after day, the sun will shine, the wind will keep blowing, and rivers won't stop flowing. With the rapid growth of the global human population, the demand for energy also increases. Only means to reduce the dependencies on fossil fuels thus reducing greenhouse gas emission, is to adopt renewable energy technology.[2]

1.2 Wind Energy Systems

The role of a wind energy system is to generate electricity for it captures mechanical energy from the airflow and converts it to electrical energy. For the former purpose it consists of a wind turbine rotor, and the latter an electrical machine usually a generator..[4]

• *Roshan S. Kodare, Department of Mechanical, Bharati Vidyapeeth (Deemed to be University) College of Engineering, Pune, India. Email: roshankodare@gmail.com*

- Classification of Wind Turbines

Wind turbines are classified majorly into two types, Horizontal axis wind turbines (HAWT) and Vertical axis wind turbines (VAWT). This classification is based on the orientation of their rotating axis. Classification of wind turbines based on rotor diameter are Large scale and Small scale wind turbine. Wind turbines having a rotor diameter ranging from 50m to 100m are classified as large scale wind turbines. They have power capacity of 1 MW and 3 MW. Small scale wind turbines have a rotor diameter ranging from 3m to 10m. It produces power between 1.4–20 kW.[5] Micro capacity wind turbine range from 50 W to 2 kW. Micro wind turbine is installed in urban area where wind area is low.

Other Classification are:[4]

- Lift type: They are recognized as low speed turbines. Horizontal axis wind turbines and Darrieus wind turbine are lift-based turbines.
- Drag type: They are recognized as high speed turbines. Savonius wind turbine are drag-based turbines.[7]
- Vertical axis wind turbines (VAWT)

Vertical axis wind turbines have their rotor axis in vertical direction. The VAWT's are advantageous at locations where the wind directions frequently changes.[6]
There are two major classifications of VAWT:

1. Darrieus Wind Turbine: These wind turbines are lift-based wind turbines. They consists of a number of straight or curved blades mounted on a vertical framework.
2. Savonius Wind Turbine: These are drag-based wind turbines. They consists of 2 - 3 scoops having 'S' shaped cross-section when viewed from top. This turbine experiences lesser drag.

The Darrieus wind turbine may be difficult to build but provide larger efficiency.[8]

- Horizontal axis wind turbines (HAWT)

Horizontal axis wind turbines are those whose rotor axis is in horizontal direction. These turbines will work efficiently when facing the wind. HAWT have the ability to self-start and Yaw in the direction of the wind.[5] There are more options and technologies available for selecting HAWT's and are comparatively economical.[6] Performance of the wind turbine can be improved by selecting appropriate number of blades and also their radius.[10][11] HAWT still can achieve larger efficiency compared to VAWT and are easier to manufacture.[9]

1.3 Computational Fluid Dynamics

Computational fluid dynamics is the use of applied mathematics, physics and computational software to visualize how fluid flow as well as how the fluid flow affect objects as it flow past. It is based on Naiver-Stokes equation which defines single phase fluid flow.

- *General Concept of Airfoil*

Airfoil is defined as cross section of a body held stationary in a fluid in order to generate useful aerodynamic force. Shape of the airfoil forms a vital step in wind turbine design. An aerodynamic force is produced when airfoil shaped body moves through a fluid. When the body placed in a flow field, lift is produced in a direction perpendicular to direction of motion. While another force component parallel, called drag is produced. Geometry of the airfoil consists of following terms:[13]

- a leading edge and a trailing edge, the point one at the front and other at the back respectively that has the maximum curvature.
- the cord line connects the leading and the trailing edge.
- the mean camber line is the locus of the points midway between the lower surface and the upper surface
- the pitching moment is independent of angle of attack and lift coefficient at the aerodynamic centre, which is a chord-wise length.
- the pitching moment is zero at the centre of pressure.

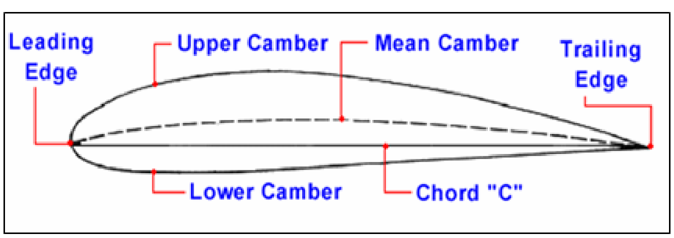

Fig.1 General Airfoil

- *Angle of attack*

Angle of attack is the angle between the oncoming flow and the body's reference line. In aerodynamics, angle of attack specifies the angle between the chord line of the wing of a fixed-wing aircraft and the vector representing the relative motion between the aircraft and the atmosphere.[12]

- *Reynolds number*

It the dimensionless number that defines the characteristic of fluid. It is the ratio of Inertia to Viscous force.

Let

ρ - Density of air

μ – Viscosity of fluid V – Air stream velocity A – Projected area

C – Chord length l - Airfoil span

L- Length

$$\mathrm{Re} = \frac{\rho u L}{\mu} = \frac{uL}{\nu}$$

2 Q-BLADE SIMULATION

INDTH 4412 airfoil was used to model the blades. This airfoil was designed to provide low drag coefficient and high lift coefficient.

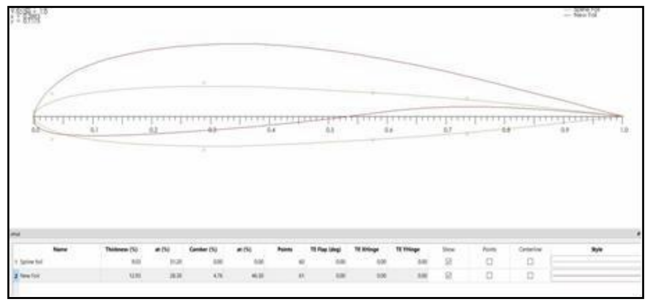

Fig.2 Design airfoil INDTH 4412

The rotor blade model was designed using Q-Blade software, the length of blade is 0.8m, maximum chord length is 0.117m and tip position at end is 0.031m. Blade is sectioned into 10 parts each section 0.08m in length. The blade should have maximum chord length upto 25% of the total length, in this case its upto 0.2m.

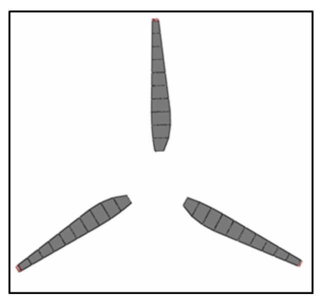

Fig.3 Geometry of Blade generated in Q-Blade

Different parameters of INDTH 4412 airfoil is analyzed using Q-Blade software. The important parameters include lift coefficient and drag coefficient and their results are plot on graph. Also the thrust coefficient and power coefficient are plotted. For simulation Reynolds number and wind speed was kept constant at 822000 and 12m/s respectively. The air density 1.225 kg/m3 at ambient temperature is considered. The kinematic viscosity 1.789 10-5 kg/m-s is taken varying the angle of attack between 0° to 15°.

The Fig. 4 depicts the behaviour of Cl/Cd ratio, as it increases upto angle of attack 4° and then decreases. Thus the optimum angle of attack that provides low drag and high lift coefficient is 4°.

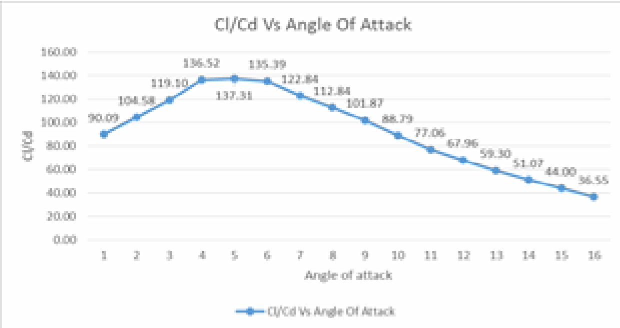

Fig. 4 Graph of Cl/Cd Vs angle of attack

The hub of the rotor has a radius of 0.075m, on which the three blades are connected. These capture the energy from the wind flowing through it. A respective CAD model of the blade was made using SolidWorks 2016.

3 CFD ANALYSIS

ANSYS 16.0 (Fluent) is used to analyze the performance of the blade.

3.1 Geometry Creation

The SolidWorks model was imported into ANSYS Fluent, and C-mesh domain was generated around the blade geometry. . Now the blade geometry was removed using Boolean operation out from the C-mesh Domain, and a Computational fluid domain was obtained.

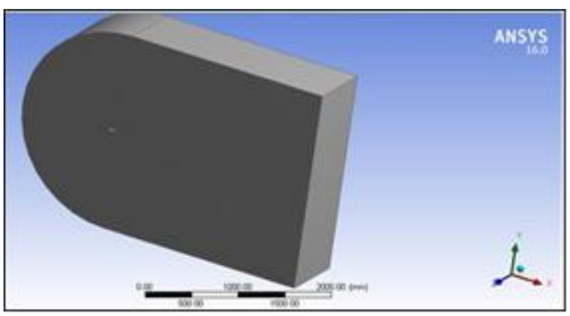

Fig. 5 Geometry of Blade

3.2 Mesh Generation

Automatic Mesh method was used to mesh the Computational Domain, which discretizes the domain into Quad/tri elements. Named Selection was provided to the Inlet, Outlet and Airfoil. Mesh Statistics showed that the node count is 829040 and no. of elements equal to 800880.

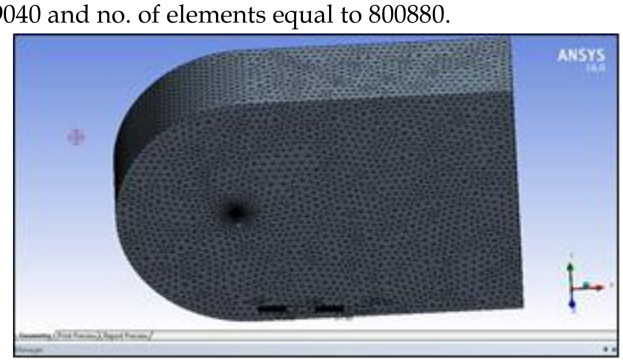

Fig.6 Meshed Computational Domain

The mesh settings were set to obtain refined mesh, which included setting relevance centre to 60. Relevance depicts the quality or state of mesh by closely connecting nodes of elements. The mesh can also so controlled by setting the advanced sizing function, for this case it was set to proximity and curvature.

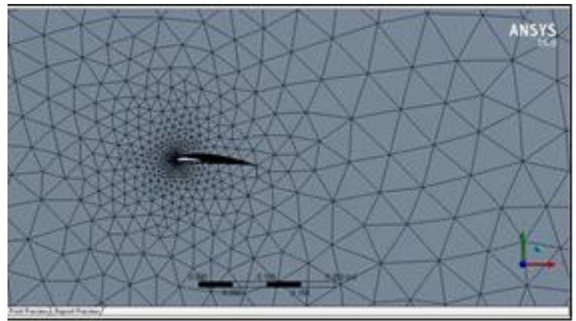

Fig.7(a) Fine mesh near Airfoil

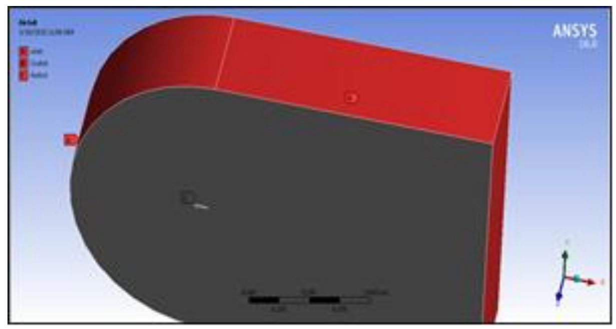

Fig.7(b) Named Selections

Create named selections is used to label the faces to be referred for performing simulations. The named selections such as Inlet, Outlet and Airfoil is given to the respective faces. The Ansys Solver recognizes these named selections, and sets the cell zone conditions for the same.

3.3 Fluent Setup

Setup parameters and input parameters to perform the Simulation are shown in table.

TABLE 1 SETUP PARAMETERS

Profile of Airfoil	INDTH 4412
Solver type	Pressure based
Time	Steady
Velocity formulation	Absolute
Model	Viscous – Standard k-ε (2 equation), Standard Wall function
Air Density	1.22 kg/m³
Kinetic Viscosity	1.7894×10^{-5} kg/m-s
Temperature	288K
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

The simulation is to predict the properties of boundary layer, for which Standard k-ε (2 equation) model is used. It is suitable for simulations that need to determine lift and drag forces. All ε-based models may also be used with enhanced wall functions, but to reduce computational time standard wall function is used.[15]

4 Results

The below graph in Fig. 8(a) & 8(b) are plotted for Co-efficient of drag and Co-efficient of lift against number of iterations. The graph for Cd convergence shows decrease in the value of drag and that of Cl convergence shows increase in the value of lift. So the graphs depict that the blade which was designed to produce less drag and high lift will perform well.

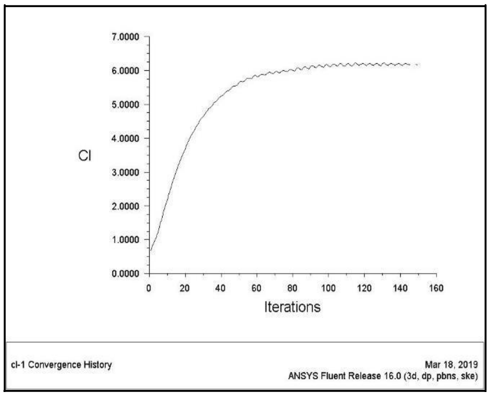

Fig.8(a) Plot for Coefficient of lift, Cl

The Pressure distribution over the computational domain was obtained for the blade geometry. The Fig. 9(a) displays the pressure distribution for 4° angle of attack and wind speed of 12 m/s. Higher pressure can be clearly observed near the leading edge of the blade. There is low pressure region on the upper portion near the blade, while high pressure beneath the blade. There will be reversal in the flow at the trailing edge, due low pressure on the upper portion near the leading edge. The difference in pressure results into development of lift force on the blade and reduced drag.

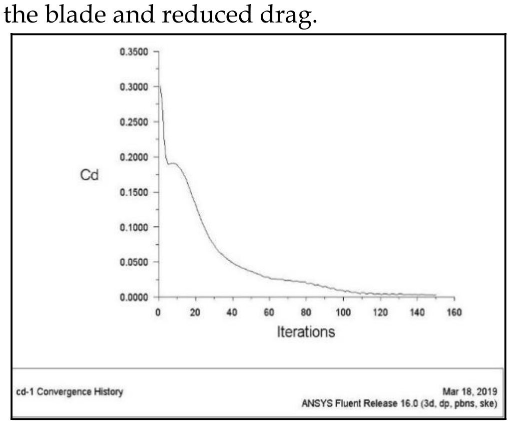

Fig.8(b) Plot for Coefficient of Drag, Cd

The Contour plot in Fig. 10(a) for Static pressure shows the pressure distribution over the surface of the blade. The tip experiences high pressure of around 9.62 Pa while there is negative pressure on the top portion of the blade that causes reversal in the flow.

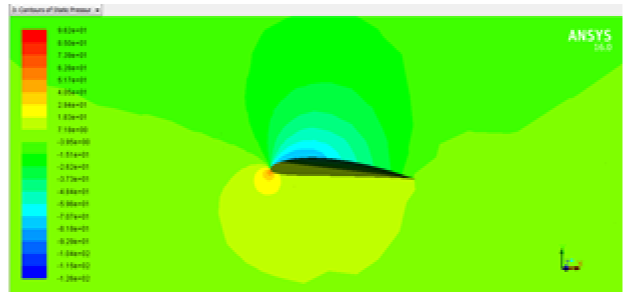

Fig.9(a) Pressure Distribution at Start of blade

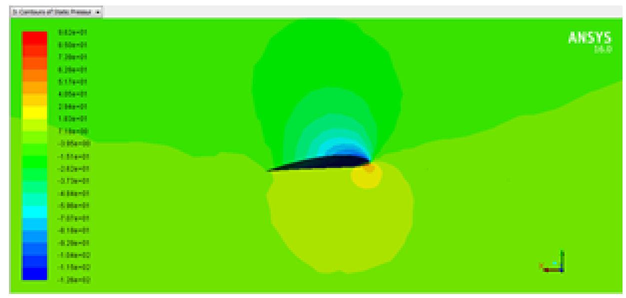

Fig.9(b) Pressure Distribution at tip of blade

The Vector plot for velocity in Fig. 10(b) illustrates the velocity magnitude at different locations or points on the blade.

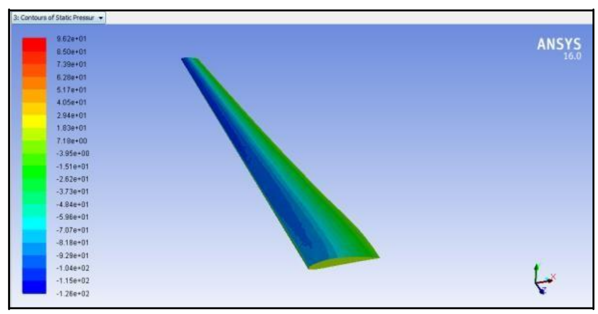

Fig.10(a) Pressure Contour

At the leading edge there forms a stagnation point thus the velocity is zero at that point. The top portion has maximum velocity of 18.2 m/s which goes on decreasing towards the trailing edge.

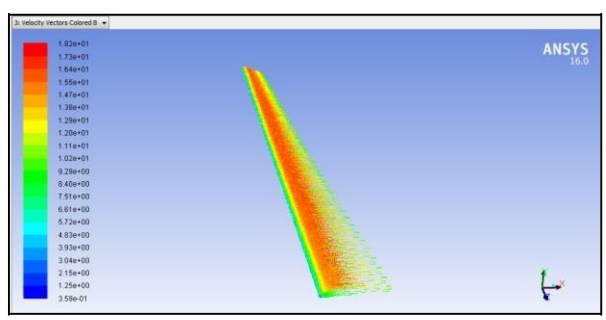

Fig.10(b) Velocity Vectors

5 Conclusion

The paper introduces the basics of small capacity wind turbine. It includes the literature review on the history of Small capacity wind turbines, classifications based on the applications and output needed. Due to increase in the demand for energy (electricity), there has been an increase in the consumption of non-renewable resources. Thus use of alternative resources has become essential, to meet the ever increasing energy usage. Even environmental guidelines need to be followed and strict environment norms are to be set, due to rise in CO^2 emissions. Computational Fluid Dynamics (CFD) was performed on the blade to determine the lift and drag forces generated and Pressure distribution contours. The Q-blade and Ansys Fluent simulations depict that an angle of attack 4° generates optimum Cl/Cd ratio. The results of simulations, show that the airfoil can be used to develop blades for a HAWT. The study also satisfied that using this blade geometry to build wind turbine is economically and technically feasible renewable energy resource.

REFERENCES

[1] Renewable Energy in India: Current Status and Future Prospects. Arun Kumar Singh Tomar , K.K. Gautam
[2] Chong, W. T., Muzammil, W. K., Wong, K. H., Wang, C. T., Gwani, M., Chu, Y. J., & Poh, S. C. (2017). Cross axis wind turbine: Pushing the limit of wind turbine technology with complementary design.
[3] Applied Energy, 207, 78-95.Energy Statistics 2018 (Twenty Fifth Issue) Central Statistics Office, Ministry Of Statistics And Programme Implementation, Government Of India, New Delhi
[4] Nima Madani "Design of a Permanent Magnet Synchronous Generator for a Vertical Axis Wind Turbine" XR-EE-EME 2011:013
[5] "A review on small scale wind turbines" Abhishiktha Tummala.
[6] Aierken Dilimulati, Ted Stathopoulos, Marius Paraschivoiu "Wind turbine designs for urban applications: A case study of shrouded diffuser casing for turbines" Journal of Wind Engineering & Industrial Aerodynamics 175 (2018) 179–192
[7] M. Ragheb "Components of Wind Machines" 2/28/2014
[8] Herzo Agenda, n.d., "Wind Energy 4. Plant Concepts", www.herzo-agenda21.de, accessed 08 August, 2015.
[9] J. Walker and N. Jenkins, 1997, "Wind Energy Technology", Wiley, first edition.
[10] A. Jha, 2010, "Wind Turbine Technology", CRC Press, first edition.
[11] T. Burton, N. Jenkins, D. Sharpe and E. Bossanyi, 2011, "Wind Energy Handbook", Wiley-Blackwell, second edition.
[12] P. J. Schubel and R. J. Crossley, 2012, "Wind turbine blade design", Energies, no. 5, pp. 3425–3449.
[13] Karthikeyan, N., Murugavel, K. K., Kumar, S. A., & Rajakumar, S. (2015). Review of aerodynamic developments on small horizontal axis wind turbine blade. Renewable and Sustainable Energy Reviews, 42, 801-822.
[14] Patel, H., & Damania, S. (2013). Performance Prediction of Horizontal Axis Wind Turbine Blade. International Journal of Innovative Research in Science, Engineering and Technology, 2(5).
[15] Mara, B. K., Mercado, B. C., Mercado, L. A., Pascual, J. M., & Lopez, N. S. (2014, November). Development and validation of a CFD model using ANSYS CFX for aerodynamics simulation of Magnus wind rotor blades. In 2014 International Conference on Humanoid, Nanotechnology, Information Technology, Communication and Control, Environment and Management (HNICEM) (pp. 1-6). IEEE.