Aerodynamic Design and Analysis of Horizontal Axis Wind Turbine (HAWT) Blades Using CFD

Mr. M. Eswaran[1], Mr. A. Dyson Bruno[2], Dr. P. Emmanuel Nicholas[3]
[1]Lecturer, [2,3]Assistant Professor
[1,2,3] PSNA College of Engineering and Technology,
Dindigul

***Abstract* –In the recent years wind power is broadly considered and utilizecd as one of the most promising renewable energy sources. In the recent research study, aerodynamic analysis of horizontal axis turbine is accomplished by using CFD. Blades which are mostly feasible for commercial grade wind turbines embody a straight span-wise profile along with airfoil shaped cross sections. Wind tunnel test is implemented in order to test aerodynamic efficiency wind turbine blade. In this paper, the researchers' choice is NACA 4421 airfoil for analysis. CFD Analysis of HAWT Blade is executed at various blade angles with the aid of ANSYS CFX and also by correlating that result with experimental results. HAWT efficiency remarkably rely upon the blade profile and its orientation. The researchers are able to identify the optimum angle at which HAWT provides constant output.**

Keywords: HAWT, CFD, NACA4421

INTRODUCTION

Wind turbine technology is one of the effective means to implement this renewable resource in order to produce environmentally friendly electrical energy. As it is an intricate system it depends upon the iunification of multiple engineering disciplines, which comprises of structures, aerodynamics, controls and electrical engineering. The main objective of the wind turbines is to capture maximum energy from the wind energy. Best design parameters has to be selected for each and every constituent of the wind turbine. It leads to increase in efficiency and life cycle. The two stages of a wind turbine blade design process are aerodynamic design and structural design.

I.1. Objective

Maximum aerodynamic efficiency at specific wind speed. To optimize blade geometry to give the maximum power for a given wind speed The objectives of the research are to establish two Dimensional and three Dimensional CFD models of wind turbine blade and rotor, so as

- To analyse the aerodynamic performance of different aero foils.
- To predict wind turbine power output at different wind speeds.
-

II. *Airfoil Nomenclature*

Chord length – It is calculated as the length from the LE to the TE of a wing cross section which is similar to the vertical axis of symmetry.

Mean camber line – It is a line which is halfway between the upper and lower surfaces.

Leading edge (LE)- It is the front most point on the mean camber line.

Trailing edge (TE) – It is the most rearward point on mean camber line.

Camber – It is the maximum distance between the mean camber line and the chord line which is measured perpendicular to the chord line - 0 camber or un cambered means the aerofoil is symmetric above and below the chord line.

Thickness – It refers the distance between upper surface and lower surface which is measured perpendicular to the mean camber line.

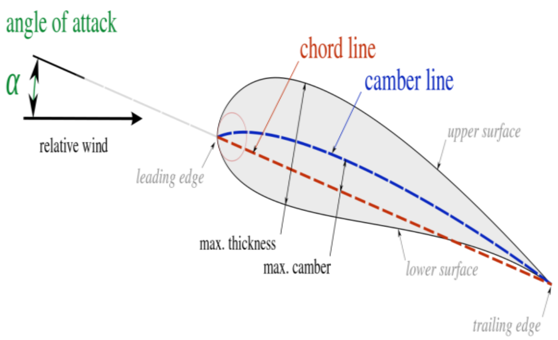


Fig.1. Airfoil Nomenclature

III. NACA 4 DIGIT AIRFOIL SPECIFICATION

This NACA airfoil series is controlled by 4 digits e.g. NACA 4421, which designate the camber, location of the greatest camber and thickness. If an airfoil number is

NACA MPXX

e.g.

NACA 4421

then M will be regarded as the maximum camber which is divided by 100. In the example, M=4 hence the camber is 0.04 or 4% of the chord. Likewise, P is the position of the maximum camber that is divided by 10. In the model, P=4 consequently the maximum camber is at 0.4 or 40% of the chord. Mean while XX is noted as the thickness divided by 100. In the illustration, XX=21 thus the thickness is 0.21 or 21% of the chord.

2. DESIGNING OF HAWT

1. Determine the rotor diameter from power equation. Power generated due to wind speed is given by following equation.

Power in the Wind P = Cp η½ρ πR^2 V^3

Cp --- Coefficient of performance (0.4 for a modern three bladed wind turbine)

Ƞ --- Expected Mechanical (or) Electrical efficiency (0.9 would be a suitable value)

ρ --- Effect of air density

R --- Tip Radius

V --- Wind velocity

2. Choose Tip Speed Ratio (λ): For a)Electrical power Generation Pick 4<λ<10

3. Choose Number of Blades (B=3)

4. Select an Aero foil.

5. Attain and study thoroughly the lift and drag coefficient Curves (CL & Cd) by using CFD Software.

6. Determination of chord length:

C=8 πR cos β/ 3Bλr

Here tan β=λr(1+a')/(1-a) λr=Ωr/v

B - Relative flow angle

λr - Local tip speed ratio

a - Axial induction factor

a' – Angular induction factor

Ω – Blade rotational speed

7. Divide the blade into N elements and moreover 10 to 20 elements are typically used.

8. Relative flow angle

β= 900 –(2/3)tan-1 (1/λr)

a= {1 + (4cos2 β/σ' CL sin β)}-1

a'=1-3a/4a-1 here σ' –local solidity

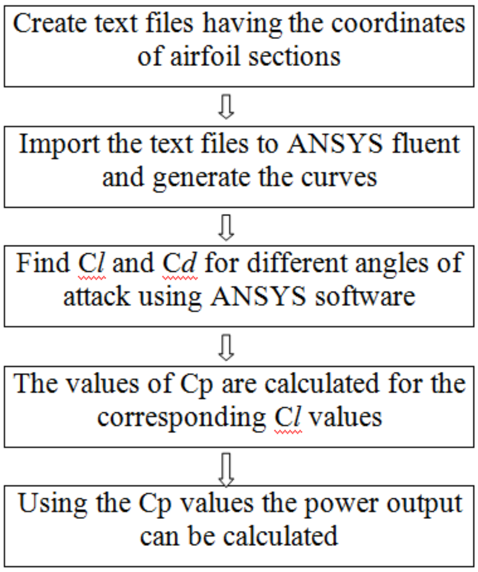
σ'=Bc/2πr

9. Calculate the rotor performance and then modify the design procedure.

$$Cp=\frac{8}{\lambda^2}\int_{\lambda_h}^{\lambda} Q\,\lambda^3 \; a'(1-a)\{1-\frac{Cd}{CL}\tan\beta\}d\lambda r$$

3. METHOD OF ANALYSIS

The aerofoil NACA 4421 is chosen for blade modeling as shown in fig.3. NACA 4421 profiles are obtained from Ansys fluent. The blade is modeled for the specification given in Table1.



NACA 4421 Profile	
Root chord length	1635mm
Tip chord length	620 mm
Length of blade	10640mm
Hub diameter	318.5mm
Hub length	1446 mm
Hub to blade (neck)	1460 mm

Table 1. Blade specification

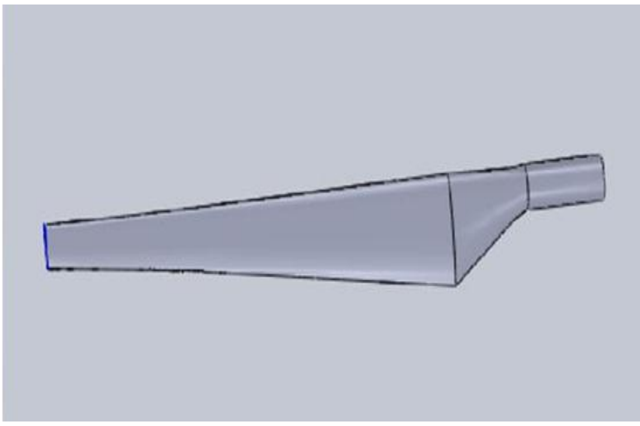


Fig 3 Wind Turbine Blade

4. CFD ANALYSIS PROCEDURE

1. Cavity model of horizontal axis wind turbine blade is created.

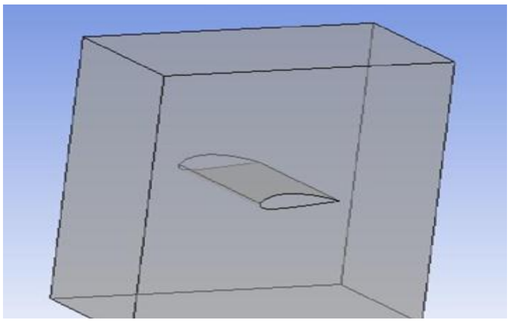


Fig 4. Cavity model of NACA 4421 airfoil

2. Save the above the cavity model in IGES file Format and import this IGES file Into ANSYS CFX.
3. Geometry in ANSYS CFX is generated.
4. Meshing geometry.
5. ANSYS CFX for Pre- processing.
6. Air domain is created.

Domain Type	Fluid Domain
Fluid	Air Ideal Gas
Domain Motion	Stationary
Heat Transfer Model	Total energy
Turbulence Model	K-Model

Table 2 Dimensions of Domain

7. Define inlet
8. Define outlet
9. Define solver control criteria.

5. RESULT AND DISCUSSION

Inlet velocity for the experiments and simulations is 16 m/sec and turbulence viscosity ratio is 10. In ANSYS CFX, turbulent flow solution was completely used. A simple solver had been employed and it was set Zero for operating the pressure. For the "linear" region, calculation must be done earnestly. [7, 8].The airfoil profile and boundary conditions are all created. In cavity domain one inlet, outlets other are symmetry boundary.

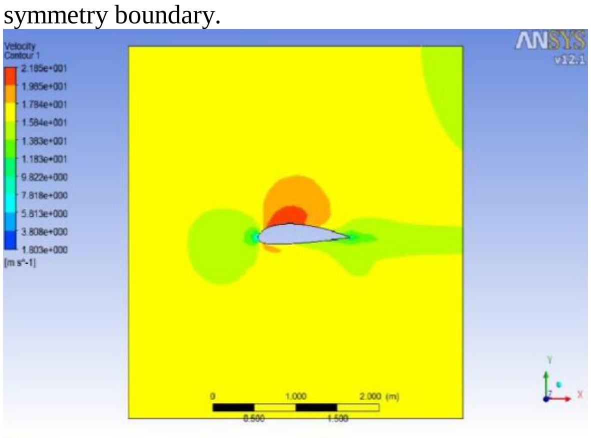


Fig 5. Velocity plot -0^0 blade angle

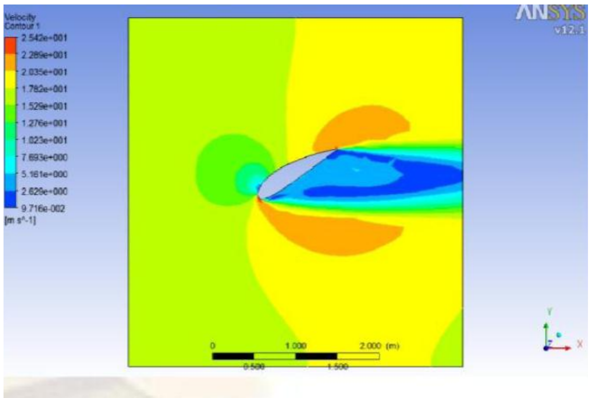


Fig 6 . Velocity plot -22.5^0 blade angle

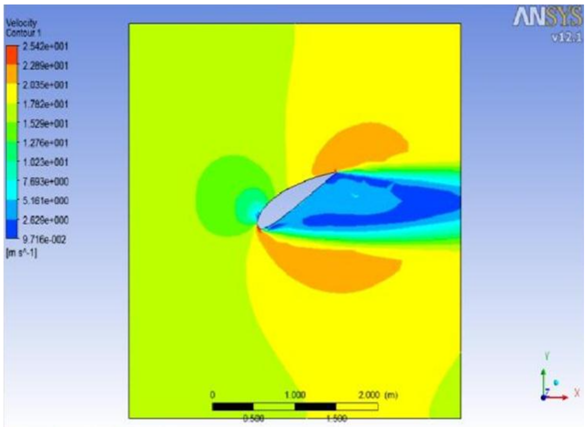


Fig 7. Velocity plot -30^0 blade angle

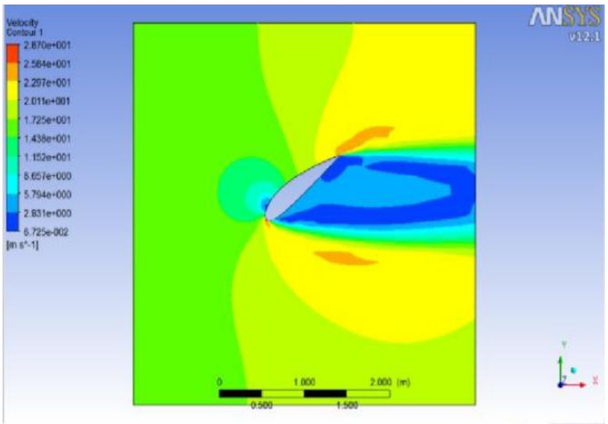


Fig 8. Velocity plot -37.5^0 blade angle

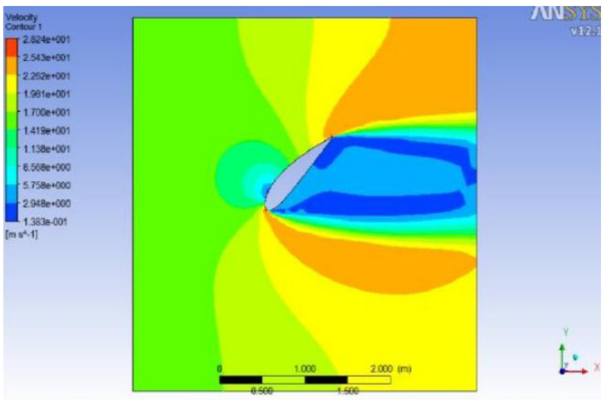


Fig .9. Velocity plot -45^0 blade angle

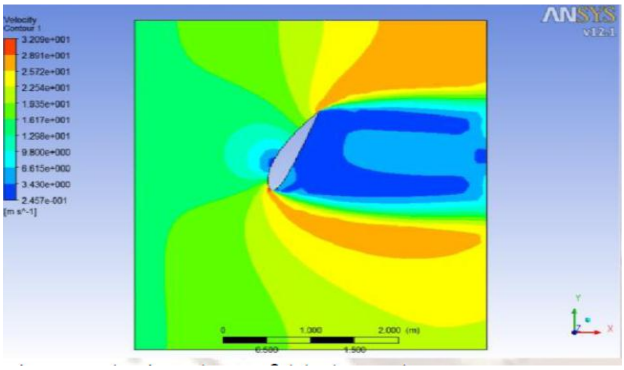


Fig 10 . Velocity plot -60^0 blade angle

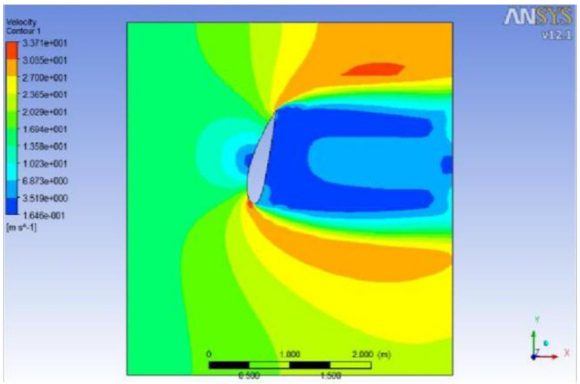


Fig 11 . Velocity plot -75^0 blade angle

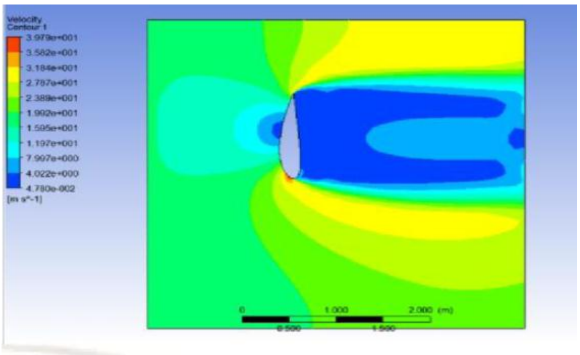


Fig 12. Velocity plot -90^0 blade angle

S.No	Blade angle	Velocity(m/s)	Density(kg/m)	Power(W)
1	0	15.5	1.225	1457710.87
2	22.5	16	1.225	1603379.2
3	37.5	16.05	1.225	1618457.90
4	45	16.10	1.225	1633630.84
5	60	16.17	1.225	1655031.85
6	75	16.94	1.225	1902902.40
7	90	17.8	1.225	2207680.92

Table 3.Effect of power in various angle of blade

Fig 13.Effect of power in various angle of blade

6. CONCLUSION

The researchers, in this paper, identify a horizontal axis wind turbine blade with NACA 4421 which is designed and analyzed for different blade angle and wind speed. The CFD analysis is executed by using ANSYS CFX software. In the given figure, the velocity distribution at various blade angles is clearly shown. The upper surface on the airfoil experiences a higher velocity when compared to the lower surface which is easily observed through this.

1.8 References

- NACA airfoil series.pdf
- H. Abbott, A.E. von Doenhoff, L. Stivers, NACA Report No. 824 – Summary of Airfoil Data, National Advisory Committee for Aeronautics.
- Thumthae C, Chitsomboon T. Numerical simulation of flow over twisted-blade, horizontal axis wind turbine. The 20th conference of mechanical engineering network of Thailand
- Kim B, Kim J, Kikuyama K, Rooij V, Lee Y. 3-D numerical predictions of horizontal axis wind turbine power characteristics of the scales delft university T40/500 model. The fifth JSME-KSME fluids engineering conference, Japan.
- Mandas N, Cambuli F, Carcangiu CE. Numerical prediction of horizontal axis wind turbine flow. European wind energyconference, Athens, Greece;
- Ansys cfx tutorial 12.pdf
- Laursen J, Enevoldsen P, Hjort S. 3D CFD rotor computations of a multimegawatt HAWT rotor. European wind energy conference, Milan, Italy.
- Thumthae C, Chitsomboon T. CFD Simulation of horizontal axis wind turbine in steady state condition. The 2nd Thailand national energy conference, Thailand.
- Alexandros Makridis and John Chick, 2009, CFD Modeling of the wake interactions of two wind turbines on a Gaussian Hill, EACEW 5 Florence, Italy. 19th-23rd July 2009
- Carlo Enrico Carcangiu, 2008, „wind turbine functioning and aerodynamics‟, CFD-RANS Study of Horizontal Axis Wind Turbines, Italy
- Chalothorn Thumthae and Tawit Chitsomboon. Optimal angle of attack for untwisted blade wind turbine. Renewable Energy, Volume 34, Issue 5, May 2009, Pages 1279-1284
- Chalothorn Thumthae and Tawit Chitsomboon. Optimal Pitch for Untwisted Blade Horizontal Axis Wind Turbine. In: SEE (Sustainable Energy and Environment) 2006, The 2nd Joint International Conference. Bangkok, Thailand. 21-23 November 2006.
- Chuichi Arakawa, Oliver Fleig, Makoto Iida and Masakazu Shimooka. Numerical Approach for Noise Reduction of Wind Turbine Tip with Earth Simulator, Journal of the Earth Simulator, Volume 2, March 2005, 11-33, pages 11-33
- C.K Cheung and W.H Melbourne. Wind Tunnel Blockage Effects on a Circular Cylinder in Turbulent Flows. In: 7th Australasian Hydraulics and Fluid Mechanics Conference. Brisbane, Australia. 18-22 August 1980.