Aerodynamic Analysis Of Horizontal Axis Wind Turbine Blade

[1] Mr. Monir Chandrala, [2] Prof. Abhishek Choubey, [3] Prof. Bharat Gupta

[1]M.Tech. [Machine Design] Student, Department of Mechanical Engineering, Oriental Institute of Science & Technology, Bhopal (MP)

[2] Asst.Professor, Department Of Mechanical Engineering, O.I.S.T, Bhopal (MP)

[3] Asst.Professor and Head, Department Of Mechanical Engineering, O.I.S.T, Bhopal

ABSTRACT:

Wind energy has been shown to be one of the most viable sources of renewable energy. With current technology, the low cost of wind energy is competitive with more conventional sources of energy such as coal. Most blades available for commercial grade wind turbines incorporate a straight span-wise profile and airfoil shaped cross sections. For the testing of aerodynamic efficiency wind turbine blade, wind tunnel test is carried out. In this paper we select NACA 4420 airfoil for analysis. CFD Analysis of HAWT Blade is carried out at various blade angles with help of ANSYS CFX and comparing that result with experimental results. HAWT efficiency is highly depends on the blade profile and its orientation. We can find optimum angle at which HAWT gives constant output.

Keywords— NACA Airfoil, CFD analysis, Wind Tunnel Test

I INTRODUCTION

Wind turbines interact with the wind, capturing part of its kinetic energy and converting it into usable energy. This energy conversion is the result of several phenomena. The wind is characterized by its speed and direction, which are affected by several factors. Atmospheric turbulence causes important fluctuating aerodynamic forces on wind turbines. Turbulence is an important source of aerodynamic forces on wind turbine rotors. Turbulence is an irregular motion of fluid that appears when fluids flow past soil surfaces or when streams of fluid flow past or over each other.

We select NACA 4420 airfoil. The blade is made up of single airfoil. The chord length is varying throughout the blade. Analysis is done by changing the blade angle with different wind speed. We can see that which blade angle give optimum power output with constant wind speed.

II LIFT AND DRAG

Lift on a body is defined as the force on the body in a direction normal to the flow direction. Lift will only be present if the fluid incorporates a circulatory flow about the body such as that which exists about a spinning cylinder. The velocity above the body is increased and so the static pressure is reduced. The velocity beneath is slowed down, giving an increase in static pressure. So, there is a normal force upwards called the lift force.

The drag on a body in an oncoming flow is defined as the force on the body in a direction parallel flow direction. For a windmill to operate efficiently the lift force should be high and drag force should be low. For small angles of attack, lift force is high and drag force is low. If the angles of attack (α) increases beyond a certain value, the lift force decreases and the drag forces increases. So, the angle of attack plays a vital role.
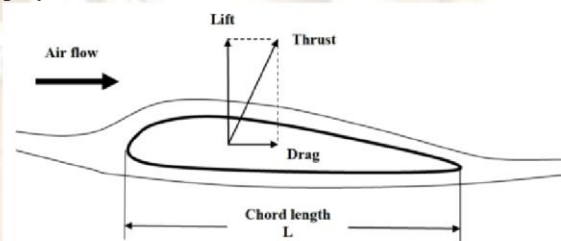

Fig 1. Forces on a stationary rotor blade in airflow

III NACA AIRFOIL

An airfoil means a two dimensional cross-section shape of a wing whose purpose is to either generate lift or minimize drag when exposed to a moving fluid. The word 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".

3.1 Airfoil Nomenclature

Chord length – length from the LE to the TE of a wing cross section that is parallel to the vertical axis of symmetry

Mean camber line – line halfway between the upper and lower surfaces

Leading edge (LE) is the front most point on the mean camber line,

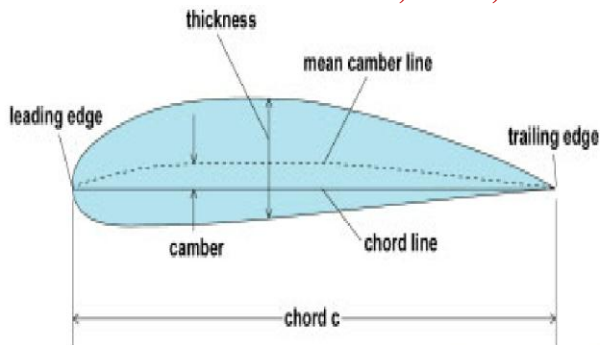**Trailing edge (TE)** is the most rearward point on mean camber line

Fig 2. Airfoil nomenclature

Camber – maximum distance between the mean camber line and the chord line, measured perpendicular to the chord line

- 0 camber or uncambered means the airfoil is symmetric above and below the chord line **Thickness** – distance between upper surface and lower surface measured perpendicular to the mean camber line

3.2 NACA Four digit series (4420)

1. First number is camber in percentage of chord(**4**)
2. Second number is location of maximum camber in tenths of chord measured from LE(**40**)
3. Last two digits give maximum thickness in percentage of chord(**20**)[1]

IV   DESIGNING OF HAWT

Power generated due to wind speed is given by following equation.

$$P_w = \frac{1}{2} \times \rho \times A \times u^3 \times \frac{16}{27}$$

Where,
P_w= Power of wind (watt)
ρ = Air density (Kg/m^3) (1.225 Kg/m^3)
A= Area of segment of the wind being considered u = Undistributed wind speed (m/s)
The wind velocity is taken as 8 m/s

We can find the value of C_L and C_D from Design Foil Workshop for different angle of attack.
For AOA 10 degree,
C_L=1.279, C_D=0.0144

Lift and Drag forces find out by following equation

$$F_L = \frac{1}{2}\rho \times u^2 \times C \times lb \times C_L$$

$$F_D = \frac{1}{2}\rho \times u^2 \times C \times lb \times C_D$$

Where,
C= chord length-1.1 m
lb= length of blade element-2 m
C_L = lift coefficient
C_D = drag coefficient

V   METHOD OF ANALYSIS

The aerofoil NACA 4420 is chosen for blade modeling as shown in fig.3. NACA 4420 profiles are obtained from Qblade software [2] .The modeling is done with Solid Works. The blade is modeled for the specification given in Table1.
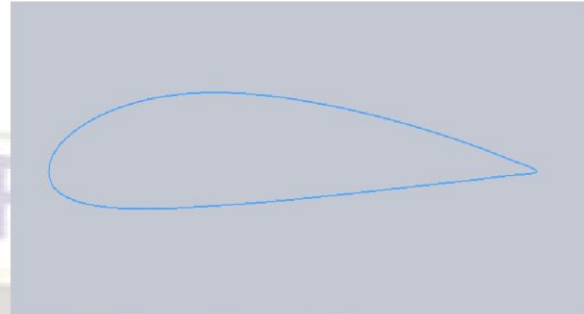

Fig3 .NACA 4420 airfoil profile

NACA 4420 Profile	
Root chord length	1651 mm
Tip chord length	650 mm
Length of blade	10700 mm
Hub diameter	337.5 mm
Hub length	1465 mm
Hub to blade (neck)	1475 mm

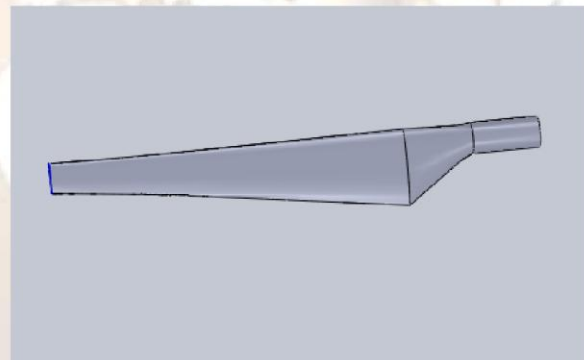Table 1. Blade specification

Fig4. Wind turbine blade

5.1Numerical method

The numerical method utilized for the simulation had a density based solver with implicit formulation, 2-D domain geometry, absolute velocity formulation, and superficial velocity for porous formulation. For this test, a simple solver and an external compressible flow model for the turbulence was utilized. There are different equations used for flow, turbulence, species, and energy [3]. A simple method was used for the pressure-velocity coupling. For the discretization, a standard pressure was used, and density, momentum and turbulent kinetic energy were set to second order upwind. [4, 5].

5.2Flow analysis

The computational flow analysis is also performed for NACA 4420 profile. The three

sections are considered for flow analysis at the blade from root to tip show in Table.2.

Section	Distance from hub (m)	Chord length (m)
1	2.94	1.651
2	6.82	1.1
3	10.7	0.65

The NACA 4420 airfoil profile is created in solid work software and the CFD analysis is performed in ANSYS CFX. A smart fine mesh is created for the flow area.

5.3 CFD analysis procedure
1. Create cavity model of horizontal axis wind turbine blade.
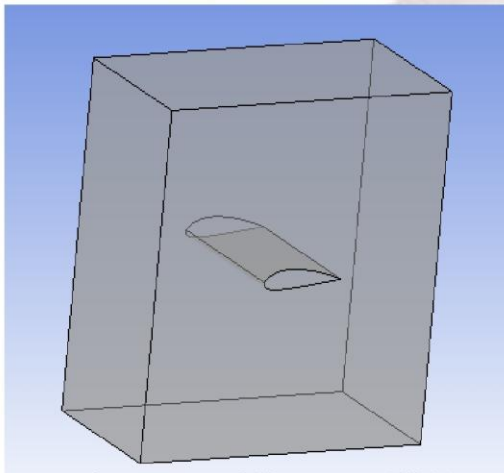
Fig 5. Cavity model of NACA 4420 airfoil

2. Save as above cavity model in IGES file format and import this IGES file into ANSYS CFX.
3. Generate geometry in ANSYS CFX.
4. Meshing the geometry.
5. ANSYS CFX for pre-processing
6. Create air domain

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

7. Define inlet
8. Define outlet
9. Define solver control criteria[6]

5.4 Geometry and boundary conditions
Inlet velocity for the experiments and simulations is 16 m/sec and turbulence viscosity ratio is 10. A fully turbulent flow solution was used in ANSYS CFX. A simple solver was utilized and the operating pressure was set to zero. Calculation was done for the "linear" region. [7, 8].The airfoil profile and boundary conditions are all created. In cavity
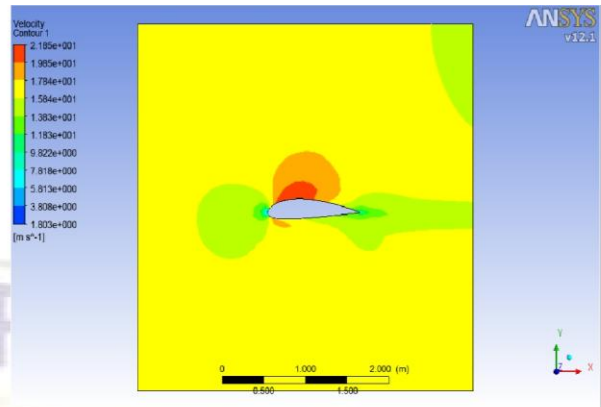
domain one inlet, outlets other are symmetry boundary.
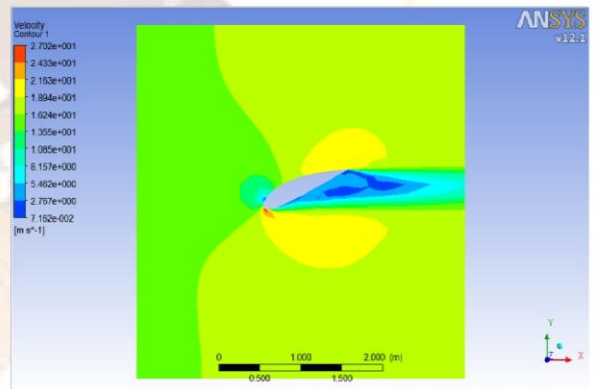

Fig 6.Velocity plot-0° blade angle

Fig 7.Velocity plot-22.5° blade angle
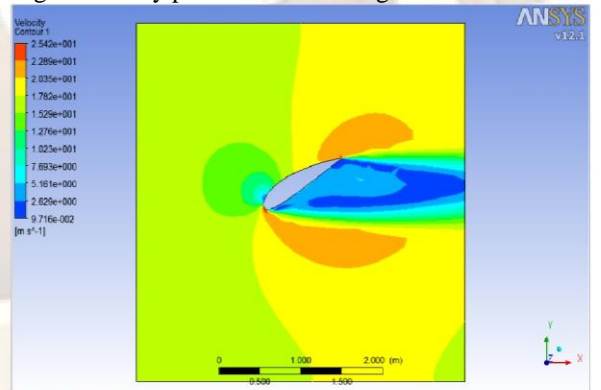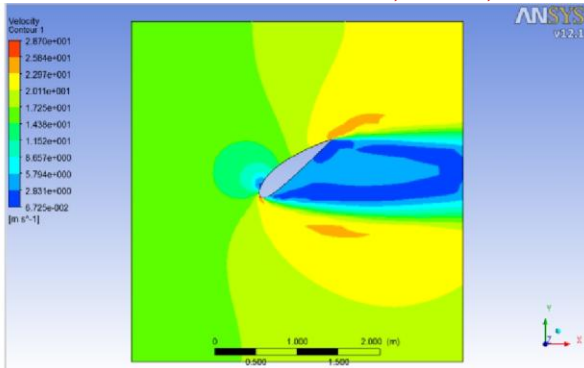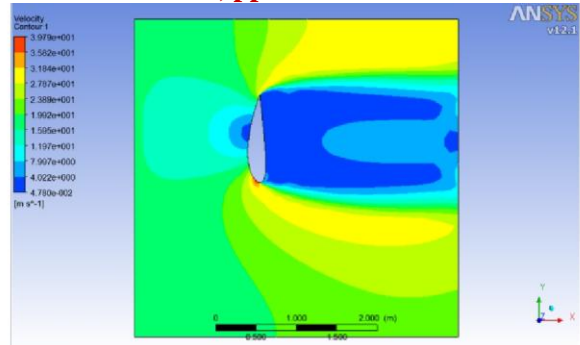
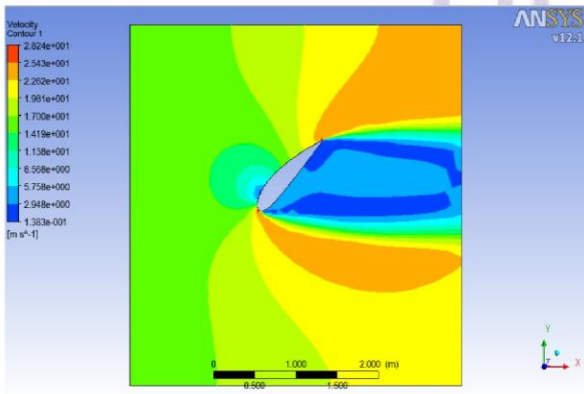

Fig 8.Velocity plot-30° blade angle
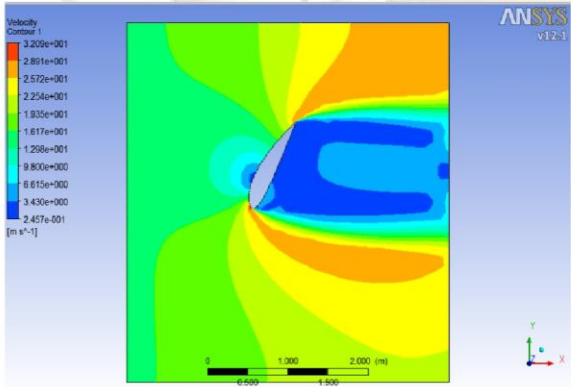
Fig 9.Velocity plot-37.5⁰ blade angle

Fig13.Velocity plot-90⁰ blade angle

Fig10.Velocity plot-45⁰ blade angle
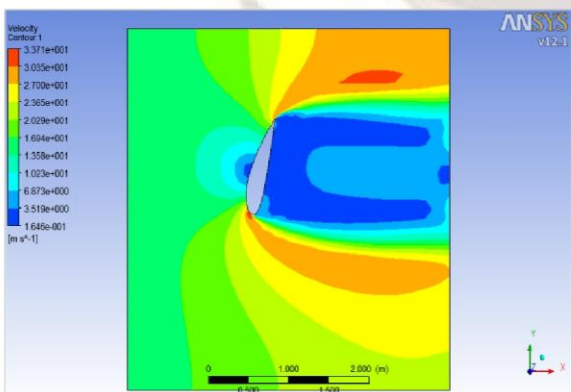
Sr. No	Blade Angle ⁰	Velocity (m/s)	Density (kg/m³)	Power (W)
1	0	15.5	1.225	1457710.87
2	22.5	16	1.225	1603379.2
3	37.5	16.05	1.225	1618457.90
4	45	16.10	1.225	1633630.84
5	60	16.17	1.225	1655031.85
6	75	16.94	1.225	1902902.40
7	90	17.8	1.225	2207680.92

Table 3.Effect of power in various angle of blade
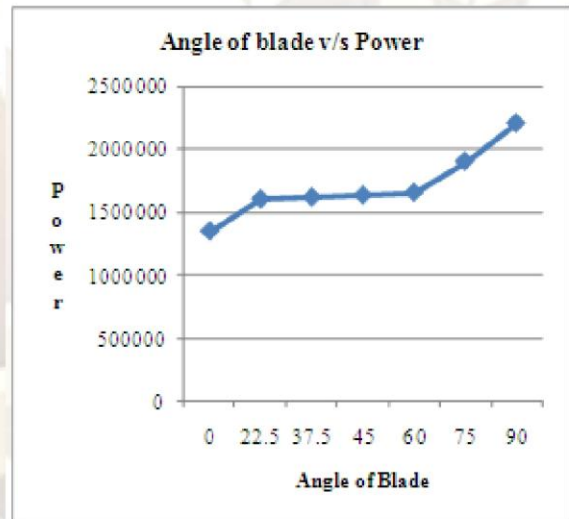

Fig11.Velocity plot-60⁰ blade angle

Figure 14.Effect of power in various angle of blade

VI   WIND TUNNEL TESTING

This airfoil made up of aluminium. I put the airfoil at different angle of attack with a constant velocity 16 m/s and find pressure distribution over an airfoil.

6.1 Operating procedure

Fig12.Velocity plot-75⁰ blade angle

The following sections describe the initial steps to be followed in preparing the instruments.

Step-1: Connect manometer limbs to the various static pressure taps of the airfoil and fill the manometer reservoir with water.

Step-2: Initially, the lift and drag readouts should be set to zero with the model attached to it.

Step-3: The incidence angle is changed by loosening the bolts and manually positioning the airfoil at the required incidence angle.

Fig15. Wind tunnel testing

The upper surface and lower surface of an airfoil are divided in to 6 point for pressure measurement.

VII   RESULTS AND DISCUSSION

In this paper a horizontal axis wind turbine blade with NACA 4420 is designed and analysed for different blade angle and wind speed.

The CFD analysis is carried out using ANSYS CFX software. The velocity distribution at various blade angles is shown in fig.6-13.These results are coinciding with the wind tunnel experimental values. Hence the results are validated with the experimental work.

It could be observed that the upper surface on the airfoil experiences a higher velocity compared to the lower surface. By increasing the velocity at higher mach numbers there would be a shock wave on the upper surface that could cause discontinuity.

From the figure 14 we can see that we get constant power between blade angles 22.5⁰ to 60⁰.

REFERENCES

[1]   NACA airfoil series.pdf

[2]   H. Abbott, A.E. von Doenhoff, L. Stivers, NACA Report No. 824 – Summary of Airfoil Data, National Advisory Committee for Aeronautics.

[3]   Thumthae C, Chitsomboon T. Numerical simulation of flow over twisted-blade, horizontal axis wind turbine. The 20th conference ofmechanical engineering network of Thailand.

[4]   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.

[5]   Mandas N, Cambuli F, Carcangiu CE. Numerical prediction of horizontal axis wind turbine flow. European wind energyconference, Athens, Greece;

[6]   Ansys cfx tutorial 12.pdf

[7]   Laursen J, Enevoldsen P, Hjort S. 3D CFD rotor computations of a multimegawatt HAWT rotor. European wind energy conference, Milan, Italy.

[8]   Thumthae C, Chitsomboon T. CFD Simulation of horizontal axis wind turbine in steady state condition. The 2nd Thailand national energy conference, Thailand.