

A STUDY ON AIRFOIL CHARACTERISTICS OF A ROTOR BLADE FOR WIND MILL

Dhatchanamurthy.P¹, Karthikeyan .L.M², Karthikeyan.R³

¹*Department of Aeronautical Engineering, Kathir College of Engineering (India)*

²*Department of Aeronautical Engineering, Techno Global University (India)*

³*Department of Aeronautical Engineering, Er.Perumal Manimekalai College of Engineering, (India)*

ABSTRACT

All over the world there is a substantial effort in design, development and installation of wind turbines for renewable energy production. Aerodynamic design processes involved in the design of wind rotor for power generation from wind. The design of airfoil shape of rotor blade is a most complex process for wind mill design. This article aims to study on characteristics of an airfoil of a low power wind turbine blade for rural application in domestic situation. As per the Betz's theory, forces over a turbine segment has obtained and the airfoil characteristics of a blade and it has analyzed using CFD software.

Keywords : *Airfoil, CFD, Rotor Blade, Wind Mill*

I. INTRODUCTION

The power from the wind is a natural source of renewable energy. In the present scenario, there is a perceived doom arising from the complete utilization of non-renewable fossil fuels, the possibility of unlimited power hidden in the natural wind provides a hope for the future. There are many efforts in design, development and erection of wind turbine for generation of electrical power and connecting it to the local electric grid system. Design of large power wind turbines for specific location assured of steady winds of high velocities, but the design of wind mills from the range of 3 to 10 kilowatt (kW) for domestic and farms involves difficult processes. The effort in this study is for the design of wind rotor of about 5 kW capacity of functioning in a wind of about 8 m/s. The emphasis here is in trying to understand the main design issues and to design a rotor for the small-power turbine following the classical method due to Betz.

II. GLOBAL THEORY OF WIND TURBINE DUE TO BETZ

The estimation of the wind power production capacity can complete using kinetics wind energy equation. Ali Musyafa [1] explains a method to estimate the production capacity of wind turbines. The Rotor is used to extract the power from wind. In practice the wind mill rotor will have certain number of blades placed equally around a hub. Typically a high speed wind rotor will have three blades. Ali Musyafa and Ronny Dwi Noriyati [2] has indicated the implementation methods for pitch angle in wind turbine position for maximum power production. The maximum

power obtained from wind can calculate from Betz's law. Magdi Ragheb and Adam M. Ragheb [3] explain the global theory of wind turbine due to Betz. According to Betz's simplified analysis, there is a limit to the maximum value of power from a rotor and from oncoming wind. Fig 2 shows the wind turbine blade designed as per Betz's criterion.

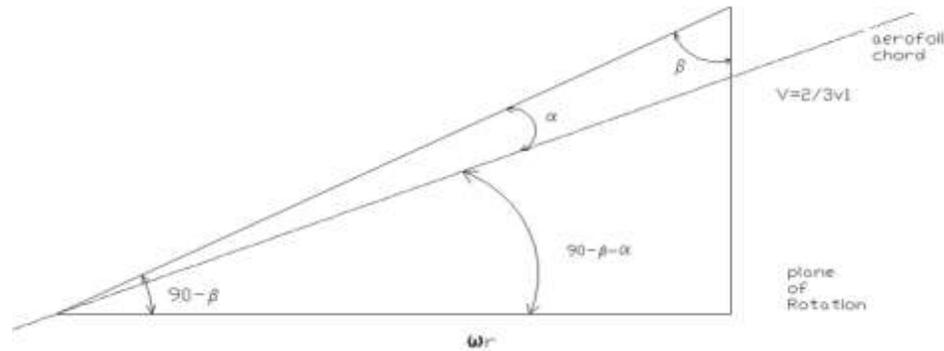


Fig 1: Velocity triangle at radius 'R' in relation to local airfoil

2.1 Design of a wind rotor: The design targets

In this study, a fast wind rotor for generating power of 5 KW has selected. For this high speed rotor we have three blades. For such a wind turbine, based on Betz hypothesis that:

- (1) Wind rotor is ideal, in the sense it has no hub and has an infinite number of blades which offer no resistance or drag to the passage of air through it.
- (2) The speed of air in front of it, at it and far behind is uniform across the cross sections.
- (3) The velocity everywhere is purely axial, that is the component of swirl induced by the blades is ignored.
- (4) The individual blades, made of aerofoil sections, have no tip losses and no tip vertices are shed.

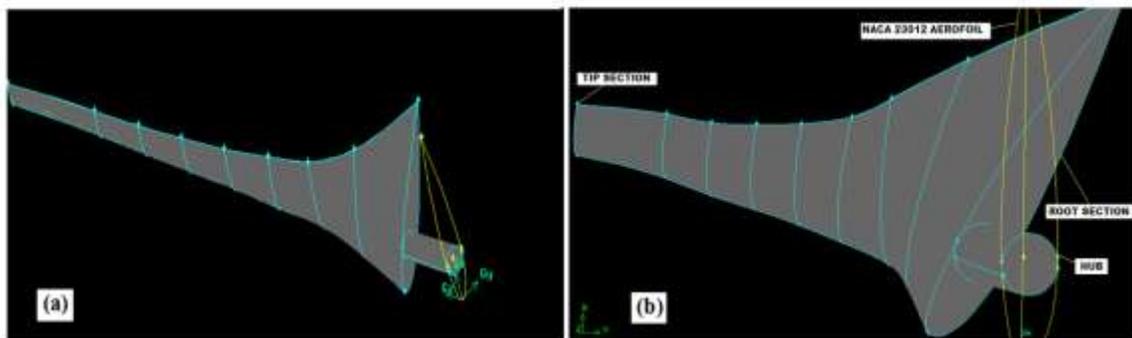


Fig 2: Wind turbine blade designed as per Betz's criterion; (a) Airfoil is NACA 23012, Design power = 5 kW, Tip speed ratio = 8, Wind speed= 7m/s, Blade diameter = 8.54 m; (b) 3 blade propeller with NACA 23012 airfoil, Tip speed ratio = 8, Wind speed = 7 m/s, Design power = 5 kW

Under these assumptions, we can obtain the distribution of local chord of the blade aerofoil as a function of the local radius r , the number of blades n , and the rotor tip-speed-ratio λ_D

$$t(r) = 2\pi R \frac{1}{n} \frac{C_{Lmax}}{C_L} \frac{1}{\lambda_D^2 \left(\frac{r}{R}\right)^2 + 4/9}$$

Where C_L is the lift coefficient corresponding to the max L/D for the aerofoil used.

2.2 Choice of The Aerofoil

There are many kinds of airfoils used for the design of wind-turbine rotor blades. Few examples are Gottingen 623, NACA 4412, 4415, 4418, 23012, 23015 and 23018. Sandip. A. Kale and Ravindra N. Varma [4] has studied aerodynamics design of a horizontal axis micro wind turbine blade using NACA 4412 profile.



Fig 3: NACA 23012 Airfoil

In this study we choose NACA 23012 airfoil for the rotor blade. In this case,

$$\lambda_D = \pi / 108 \cdot R / C_L / \sqrt{(r/R)^2 + 1/154}$$

$$\lambda_D = 0.0291 \cdot R / C_L / \sqrt{(r/R)^2 + 0.00649}$$

For the NACA 23012 aero foil our CFD analysis shows a maximum L/D of about 53 corresponding to an incidence of 8 degrees for which the lift coefficient is 0.96 using a value of $C_L = 0.96$, then the above equation becomes :

$$t(r) = 0.0303 / \sqrt{(r/R)^2 + 0.00649}$$

Table 1: Distribution of $t(r)$, $\beta(r)$ and I as a function r/R

r/R	$t(r)$ meter	$\beta(r)$ degree	I degree
0.0	1.54	0	82
0.1	0.9677	50.19	31.81
0.2	0.5763	67.38	14.62
0.3	0.4000	74.56	7.44
0.4	0.3046	78.23	3.77
0.5	0.2456	80.54	1.46
0.6	0.2057	82.09	-0.09
0.7	0.1762	83.21	-1.21
0.8	0.1545	84.05	-2.05
0.9	0.1375	84.71	-2.71
1.0	0.1239	85.24	-3.24

2.3 Diameter ‘D’ of the rotor

This being the high speed rotor we use the thumb-rule $P=0.20 \times D^2 \times V^3$ to determine the diameter D for a given wind speed V.

For our case $P=5000 \text{ W}, V=7\text{m/s}$

$$\text{i.e } 5000=0.20 \times D^2 \times 7^3 \quad \text{or } D=8.54\text{m}$$

For thin wind rotor t (r) is given by (for $R=4.27\text{m}$)

$$=0.1294/\sqrt{(r/R)^2+0.00649} \quad (\text{m})$$

From this we can compute t (r) as shown in the table 1.

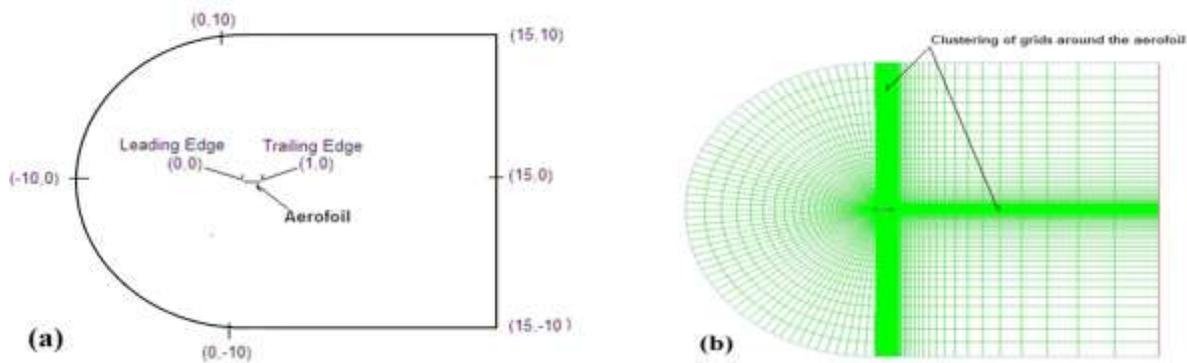


Fig 4: (a) Aerofoil is placed between (0,0) and (1,0); (b) Quadrilateral grids around the airfoil with suitable clustering

2.4 Setting angle for the profiles

At any section r we have the velocity triangle shown in fig 1, under optimum or Betz's condition.

$$\begin{aligned} \tan \beta &= \omega r/v = 3/2 \omega r/v_1 = 3/2 \omega r/v_1 \cdot (r/R) \\ &= 3/2 \cdot \lambda_D \cdot (r/R) \end{aligned}$$

Where by definition $\lambda_D = \omega R/v_1$, v_1 being free stream velocity. For the present case $\lambda_D=8$ and we have

$$\tan \beta = 12(r/R)$$

We can now compute $\beta(r)$ as function of r/R as shown in the table 1.

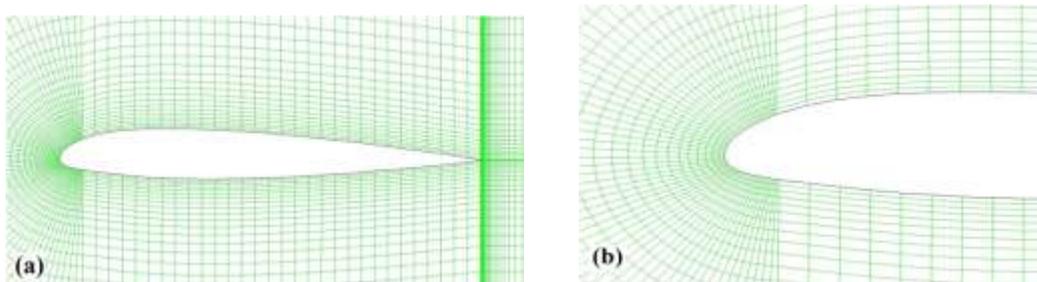


Fig 5: (A) Close View Of The Grid Around Naca 23012 Aerofoil; (B) Close View Of The Grid At The Leading Edge

As we can see from the velocity triangle β is the direction of the resultant velocity at the Plane of the rotor at a radius r . For its best performance the aero foil should take this "free stream" at an optimum incidence α , where L/D is Maximum. We have chosen NACA 23012 as the aero foil and this aero foil has a maximum L/D of about 70 around an incidence of 8° , where the lift coefficient is 0.96. We choose the keep the incidence same for all the sections although one may choose different incidence. With thin the following distribution of setting angles Z results along the radius of the rotor blades. The table 1 shows the distribution of chord $t(r)$, local flow angle with respect to the flow direction $\beta(r)$, setting angles (I) as a function r/R .

III. COMPUTATIONAL FLUID DYNAMICS

Thanks to the continuous developments in more and more powerful computers, with enormous speed and storage capabilities, the field of Computational Fluid Dynamics (CFD) has now reached a stage of maturity where it is claimed that it possible to do, routinely, computations to obtain simulation of complex flows over complex geometries and for real-life flow situations -- very reliably and in a short time. Over the years the field of grid generation has grown from simple rectangular body piercing grids to body conforming grids. There are also zonal methods for grid generation for dealing with special set of geometry in given shape, overlapping grids and unstructured grids. The field of grid generation has indeed grown into a fully mature field. We use in this study FLUENT which is the flow code and the GAMBIT which is the grid generation code.

We have studied in this report the geometric parameters such as extent of domain for computations, the role of relaxation parameters in the process of numerical convergence of numerical solution. Furat Abdal Rassul Abbas and Mohammed Abdulla Abdulsada, [5] have studied the simulation of wind-turbine speed control using MATLAB. For validation we have chosen baseline NACA 23012 aerofoil as a candidate since our study in this project concerns the same aerofoil with split flap on it. This validation study contains comparison of basic aerofoil lift, drag vs. angle of attack as a basis of validation. We have chosen standard experimental results available in literature. Based on this study we fix up range of parameters for our investigation of flow over the split flap.

3.1 Characteristics of NACA 23012 as validation Exercise

Fig 3 shows NACA 23012 aerofoil and Table 1 contains standard coordinates. These coordinates are imported to GAMBIT and quadrilateral grids are created over sub-domains into which the main domain of fluid flow has divided. The numerical flow domain taken for this study has shown in Fig.4 (a). The aerofoil is placed between $(0,0)$ and $(1,0)$. The chord of the aerofoil is thus unity. Fig.4(b) shows the quadrilateral grids around the airfoil with suitable clustering. Close view of the grid around NACA 23012 aerofoil as shown in fig 5(a). Fig 5(b) shows closely spaced grid around leading edge close to the body in order to pick up rapid changes taking place in that region. In this study, spalart allmaras one equation code has choose for model.

We have retained default options for parameters in the model and we have used second order upwind finite differencing. In fluent for solving simultaneous equations, various iterative methods are available and that results from discretisation of equation of conservation of motion from finite volume grid chosen. It's been our experience

that converging results are obtained with relaxation factor continuity x and y and turbulence viscosity has set as 0.3. We had chosen inherent choice as an available option in fluent. Fig 6 shows the lift and drag characteristics of basic NACA 23012 aerofoil compared with Experimental values.

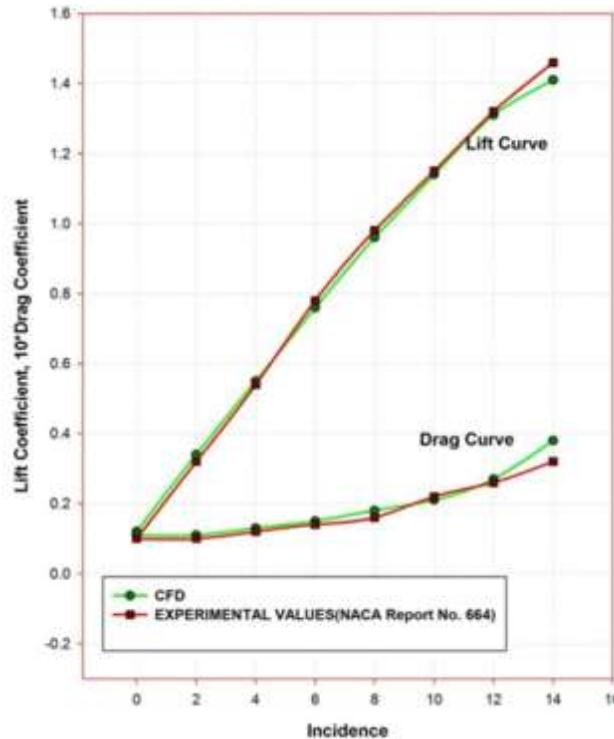


Fig 6: Lift and Drag characteristics of basic NACA 23012 aerofoil compared with experimental values

3.2 Flow Simulation Versus Prediction Capabilities of CFD

Although considerable progress has been made in the field of computational fluid dynamics by the way of excellent algorithms, of flow models for particularly turbulent flows as well as the equations of conservation laws of fluid dynamics, very many software's available in the literatures. In general fluid mechanics particularly that there is large number of extraneous parameters peculiar to algorithm or to flow model, turbulent etc. one of the essential steps in CFD namely the grid generation also would involve several parameters peculiar to geometry, some of these would include the need for special grids around specific flow regions where flow gradients are high as well as peculiarity of grids for capturing boundary layer. We would like to distinguish the simulation capabilities from predictive capabilities of the given software. We mean by simulation capability the possibility of obtaining any experimental results considered to be accurate but known as the time of computation.

In simulation, it may so turn out that on reasoning the code it's just possible to mimic the experimental results. Often this may take some time. In itself this capability also is of practical importance because after having fixed up the

parameters space and after assessing that these will give the best results. One way use the code along with the parameter space for generating data for geometries quite close to the geometry use in simulation exercise to generate useful practical results. In contrast to this if software were to have predictive capability, one should be able to obtain good comparison with standard result with some default options for parameters in the code. While more and more software achieving predictive capability there still some room for improving other codes in direction of CFD.

VI. CONCLUSION

The effort in this study is for the design of a low power wind turbine in the range of 3 to 10 kW. This contains the CFD analysis of the aerofoil chosen for the blade and the procedure for the design of a wind turbine rotor. This is an attempt to design a wind rotor of about 5 kilowatt capacity capable of functioning in a wind of about 8m/s. This study consists of Computerized Fluid Dynamics (CFD) analysis of the aerofoil chosen for the blade and the procedure for the design of a wind turbine rotor. The procedure outlines the main issues involved in the determination of the diameter and of the radial chord distribution for a three bladed horizontal axis wind turbine.

REFERENCES

- [1] Ali Musyafa, Comparative Analysis of Small-Scale Wind Turbine Design for the Low Rate Wind Speed, Asian Journal of Natural and Applied Sciences, 1(3), 2012.
- [2] Ali Musyafa and Ronny Dwi Noriyati, Implementataion of Pitch Angle Wind Turbine Position for Maximum Power Production, Academic Research International, 3(1), 2012.
- [3] Magdi Ragheb and Adam M. Ragheb, Wind Turbines Theory - The Betz Equation and Optimal Rotor Tip Speed Ratio, Fundamental and Advanced Topics in Wind Power, Intech Publication, 2011, 19-38.
- [4] Sandip. A. Kale and Ravindra N. Varma, Aerodynamic Design of a Horizontal Axis Micro Wind Turbine Blade Using NACA 4412 Profile, International Journal of Renewable Energy Research, 4(1), 2014.
- [5] Furat Abdal Rassul Abbas and Mohammed Abdulla Abdulsada, Simulation of Wind-Turbine Speed Control by MATLAB, International Journal of Computer and Electrical Engineering, 2(5), 2010, 1793-8163.