Prototype of a windmill: Theoretical part of blade design

Frank Nagl[1], Thira Jearsiripongkul[2], Wiroj Limtrakarn[3]

Institut für Strömungslehre, Fakultät für Maschinenbau, Universität Karlsruhe
Baden-Württemberg, 76131 Karlsruhe[1]
Department of Mechanical Engineering, Faculty of Engineering, Thammasat University
Klong Luang, Pathumthani 12120, Thailand[2,3]
Tel: 0-2564-3001 ext 3194[1,2], 3144[3]
E-mail: franknagl@gmx.de[1], jthira@engr.tu.ac.th[2], limwiroj@engr.tu.ac.th[3]

Abstract

In the further need of renewable energy sources the generation of electrical power from the wind energy becomes more interesting. In this spirit this paper deals about the optimization of the blade design for a small scale low cost wind turbine. Nevertheless wind energy is gratis the wind turbine performance has to be optimized with the purpose of constructing and manufacturing a technically and financially effective wind turbine facility. The main task can be divided in two main sub tasks. The first one is to increase the extractable fraction of energy the wind is containing and the second on is to decrease the production and maintenance costs.

The main intense of this research was to increase the power output and so it was regarded to keep the wind turbine as simple as possible to keep the costs at a low level.

Keywords: wind turbine, blade design, BEM

Theoretical Basics

The intention of building a wind turbine is to extract the highest possible fraction of the energy the wind contains. So the starting point is the energy of the wind:

$$P = \frac{1}{2} m v^2$$
$$P = \frac{1}{2} \rho A v^3, \qquad (1)$$

where P is the power, m is the mass of the air, v is the air velocity, ρ is the air density and A is the area of the airflow.

The amount of extracted energy mainly depends on the velocity differences of the air flow up and downstream of the rotor. Considering eq. (1) and the continuity equation the following equation shows the coherence between the velocities:

$$P = P_1 - P_2$$
$$P = \frac{1}{2} \rho A_1 v_1^3 - \frac{1}{2} \rho A_2 v_2^3, \qquad (2)$$

where P_1 is power before the rotor A_1 is the flow area before the rotor and v_1 is the velocity before (after) the rotor. The values with index "2" indicate after the rotor.

The degree of extractable energy and so the quality of the wind turbine can be described with the efficiency coefficient $C_{p,mech.}$ This can be calculated like this:

$$c_{p,mech} = \frac{P_{mech}}{P_{wind}} \qquad (3)$$

$C_{p,mech}$ is the mechanical efficiency coefficient, P_{mech} is the extracted mechanical power and P_{wind} is the power the wind contains.

Like equation (2) shows (considering the continuity equation) the wind speed is only reduced by the wind turbine. And so the fraction of extractable energy is limited. This has been first proved by Alfred Betz in the year 1920 and so this limit is called the Betz limit:

$$c_{p,\max} = \frac{16}{27}$$
$$c_{p,\max} = 0,593. \qquad (4)$$

State of the art wind turbines reach a $c_{p,max}$ of about 0.5. [1]

Important parameters

As shown before the most important factor of the wind energy is the velocity of the wind. So the configuration of the wind turbine has to match to the mainly common wind characteristics in the area the facility should be established.

An important parameter describing the aerodynamics of a wind turbine is the tip-speed ratio λ:

$$\lambda = \frac{\omega_{turb} R}{v_{wind}}$$
$$\lambda = \frac{v_{tip}}{v_{wind}}, \qquad (5)$$

where ω_{turb} is the rotational speed of the turbine, v_{wind} is the wind velocity and v_{tip} is the rotor velocity at the tip. The highest values for Cp are typically obtained for λ in the range around 8 to 9 for a three bladed horizontal axis wind turbine [2].

That means by considering an average wind speed of 6m/s the speed at the tip should be about 50m/s. This means that the angle between relative air speed and the rotor plane must form a quite sharp angle. Therefore the angle of incidence can be calculated as:

$$\alpha = \arctan\left(\frac{1}{\lambda}\right)$$
$$\alpha = \arctan\left(\frac{v_{wind}}{\omega_{turb}}\right) \quad (6)$$

The angle α is defined for the tip so the local angle will vary along the length of the blade.

To construct this small scale type of wind turbine as simple as possible the blades will be fixed. In bigger and more complicated facilities the blades can be adjusted. But to adjust the rotor blades there are servo motors and a quite complicated control system necessary.

The disadvantage of fixed blades is of course the fact that only in a small range of wind speed the facility works with the maximum efficiency coefficient.

A specific characteristic of a fixed blade wind turbine is the passive stall control. That means with increasing wind speed the angle of incidence also increases and from a certain value the blade will stall. Stall occurs when the critical angle of attack (generally 15°) is exceeded, causing an immense loss of lift and a large increase in drag due to disruption of airflow.

Process of simulation

For Computational Fluid Dynamics (CFD), there are many commercial solvers available. Each solver has special advantages and disadvantages which have to be compared to choose the most suitable solver. In this case mainly two commercial software products were used. For first simple performance estimations CFdesign has been used and in the further progress to calculate the extractable mechanical power ANSYS Workbench with the Add-on Ansys CFX has been used.

Fluid mechanical problems can be described by a set of Navier-Stokes equations which describes the process of momentum and mass transfer. To solve these partial differential equations for example ANSYS CFX is based on the finite volume technique [3].

To calculate the aerodynamic forces acting on the rotor blades the blade element momentum theory (BEM) has shown to be very effective. The BEM has shown to give a good accuracy on one hand and to be quite time efficient on the other hand. In this method the turbine blades are divided in a special number of independent elements along the length of the blade. At each section a 2D force balance is applied involving the lift and drag forces with the torque and thrust produced by the section. At the same time a balance of axial and angular momentum is applied. This produces a set of non-linear equations which can be solved numerically for each section. In the following the necessary equations for the BEM method will be introduced:

The BEM theory considers the forces in flow direction and the tangential force due to torque in the shaft.

The lift force F_{Lift} per unit is perpendicular to the relative velocity of the wind:

$$F_{Lift} = \frac{\sigma c}{2} v_{rel}^2 c_L \quad (7)$$

where c is the chord of the aerodynamic profile and c_L is the lift coefficient.

The drag force F_{Drag} per unit length is parallel to v_{rel} and is given by:

$$F_{Drag} = \frac{\sigma c}{2} v_{rel}^2 c_D \quad (8)$$

where c_D is the drag coefficient.

To calculate these forces information about the lift and drag coefficients are required. These parameters have to be obtained by 2D wind-tunnel experiments or are given in the airfoil data table as a function of α, which is the angle of incidence:

$$\alpha = \psi - \beta \quad (9)$$

Regarding Figure 1, there can be further seen that:

$$\tan\psi = \frac{(1-\alpha)U_\infty}{(1+\alpha')\omega r} \quad (10)$$

If α exceeds about 15°, the blade will stall. That means boundary layer on the upper surface becomes turbulent which will result an immense increase of the drag force and a decrease of the lift force. So the lift and drag coefficients have to be projected to the normal and tangential directions:

$$c_N = c_L \cos\psi + c_D \sin\psi \quad (11)$$
$$c_T = c_L \sin\psi + c_D \cos\psi \quad (12)$$

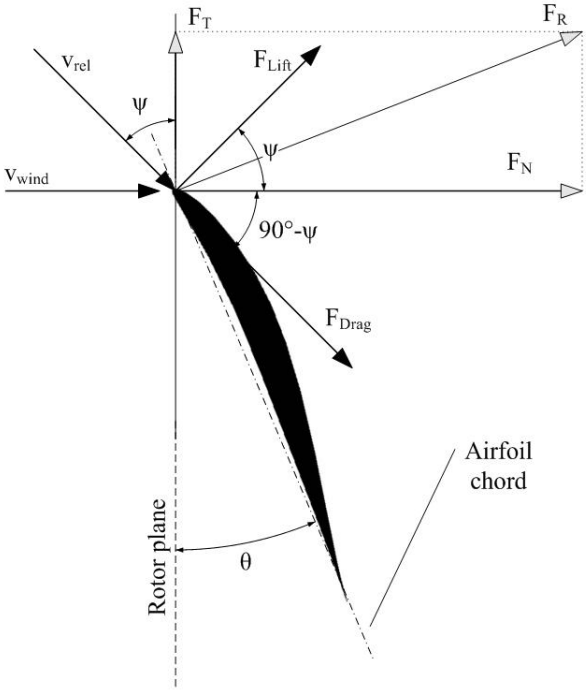

Figure 1. Acting forces

where θ is the pitch angle.

Since we are only interested in the forces normal and tangential to the rotor plane the lift and drag forces can be projected in this directions:

$$F_N = F_{Lift}\cos\psi + F_{Drag}\sin\psi \tag{13}$$

$$F_T = F_{Lift}\sin\psi + F_{Drag}\cos\psi, \tag{14}$$

where F_N/F_T is the force normal/tangential to the rotor plane. So F_N and F_T represent the resulting thrust and torque.

Further, the solidify σ is defined as the fraction of annular area in the control volume, which is covered by the blades:

$$\sigma(r) = \frac{c(r)N}{2\pi r}, \tag{15}$$

where N is the number of blades.

The normal force and the torque on the control volume with the thickness dr, is since F_N and F_T are forces per length:

$$dT = NF_N dr,$$

$$dT = \frac{1}{2}\rho N \frac{U_\infty^2(1-\alpha)^2}{\sin^2\psi} cC_N dr \tag{16}$$

and

$$dQ = rNF_N dr,$$

$$dQ = \frac{1}{2}\rho N \frac{U_\infty^2(1-\alpha)\omega r(1+\alpha')}{\sin\psi\cos\psi} cC_T r dr \,. \tag{17}$$

The two induction factors α and α′ are finally defined as:

$$\alpha = \frac{1}{\frac{4\sin^2\psi}{\sigma C_N}+1} \tag{18}$$

$$\alpha' = \frac{1}{\frac{4\sin\psi\cos\psi}{\sigma C_T}-1}\,. \tag{19}$$

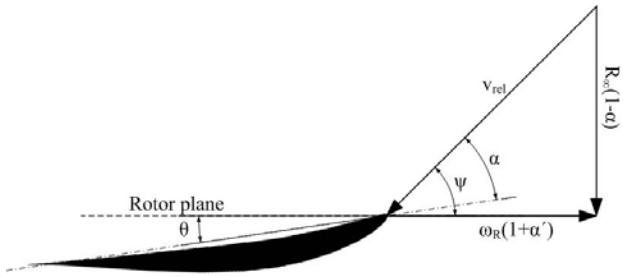

Figure 2. Inductions factors α and α′

Since the different control volumes are assumed to be independent, each 2D blade element may be treated separately and therefore the results for one radius can be computed before another one is solved. For each control volume the BEM algorithm can be divided in eight steps:

1. Initialize α and α′; typically α = α′=0
2. Compute the flow angle (ψ) using eq. (10)
3. Compute the local angle of attack using eq. (9)
4. Read $C_L(\alpha)$ and $C_D(\alpha)$ from airfoil data table
5. Compute C_N and C_T from eq. (11) and eq. (12)
6. Calculate α and α′ from eq.(18) and eq.(19)
7. If α and α′ have changed more than a certain tolerance: go back to step 2. else continue
8. Compute the local forces on each element of the blades

This is a principle description of the BEM method [4].

Basic Model definition

The model description and its accuracy have to be as simple as possible to save computing time but precise enough to get useful results.

In this case a simplified model has been used. The undisturbed airflow of the wind has been estimated laminar with constant speed. The model of the blades, the hub and the tower has been generated according to airfoil data sheet. To simulate the losses and the resistance of the generator and the bearing a constant decelerating torque has been applied to the rotor hub.

Choosing a blade geometry

Finding the optimal blade geometry requires an algorithm which can basically be divided into the following steps:

1. Chose a blade geometry which could be suitable

for the given parameters.

2. Simulation of the aerodynamic quality of the chosen blade geometry
3. Modify blade geometry
4. repeat step 2 und 3 until an expected Cp value is reached
5. Build a prototype of the blade to prove the estimated results from the simulation
6. If the blade performance is proven to be good enough the whole device can be built

The air flow around the profile causes drag and lift. The reason for that are different velocities around the airfoil. Air with lower velocity has a higher pressure than air with higher velocity. The slower moving air under the profile causes the lift force of the rotor. The drag force results from the air pulling against the rotor and holding it back. The result of those two forces is the thrust. Consequently the airfoil performance is determined by the ratio of drag and lift.

The first two chosen profiles are a NACA 6 series profile: NACA 63-908 with a mean line parameter of 0.3 and a self defined one: called "Profile 2".

NACA 6 series profiles have been successfully used in small scale wind turbine facilities. One example of effective usage as rotor blades for a 1.5 kW facility in turkey is the NACA 63-622. The NACA 63-nnn series blade can be preferred to other blades for performance improvement because of the fact that these blades have shown excellent properties for wind turbine applications and their high average power coefficients. [5]

The shape of the "Profile 2"-airfoil is described by a list of points and their x- and y-coordinates. This profile has been chosen because of its similar properties to the NACA profile. The main differences in shape are the radius of the body in the front part and the camber of the underside and of the rear upside of the airfoil. The ratios of lift to drag of both profiles at different angles of attack are nearly the same. The average ratio of lift to drag at different attack angles is about 130. So the two chosen profiles show at the first sight nearly the same performance attributes. Therefore the aerodynamic simulations will show if these two airfoils have appreciable different aerodynamic characteristics on closer examination.

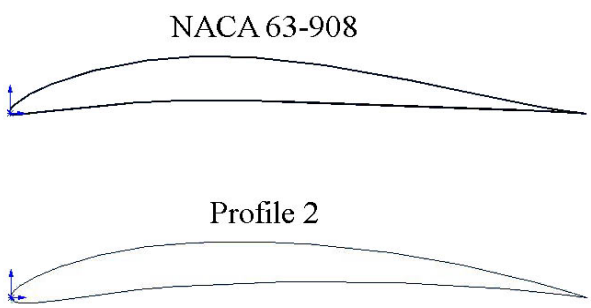

Figure 3. Shape of the chosen Airfoils

Additional Increase of the power output

Like other former research projects have shown, there exist ways to increase the power output even above the Betz Limit. With a diffuser around the rotor it is possible to change the characteristics of the airflow in a positive way. On the one hand the diffuser increases the wind velocity at the entrance of the tube and reduces turbulence [6]

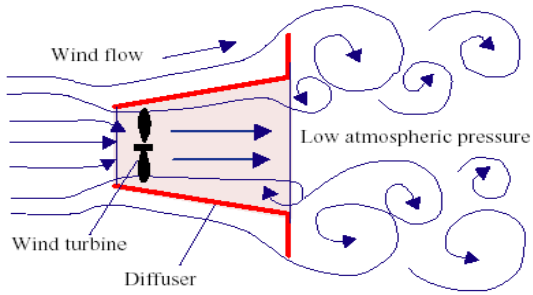

Figure 4. Schematic principle of a diffuser

A more specific description of the active principle, shape and potential of a diffuser will follow in the presentation.

First simple simulations

At the moment are just some results about the aerodynamic quality of the chosen blade design available. Therefore first simulations with CFdesign have been done. The chosen model is a simplified 3D model of the NACA 63-908 and a similar shaped airfoil ("profile 2") both without tightening towards the tip and without built in twist. Further there was just the rotor (hub with blades) without the tower regarded. A decelerating momentum has been applied to the hub. The drag and lift coefficients are given. The undisturbed air flow is modeled as laminar with constant velocity. Further there are some more special simulation parameters which should be regarded to get correct results [7]:

- Meshing: For an adequate meshing at least two rows of masked nodes of the solid path throughout the fluid path are required.
- Fluid: The fluid should be considered as compressible, because there will be regions in the fluid which are due to the motion isolated from other regions. An incompressible would not allow the pressure waves to travel throughout the medium and this may cause instability. Additionally objects that are supposed to move due to the flow-induced forces may not move at all.

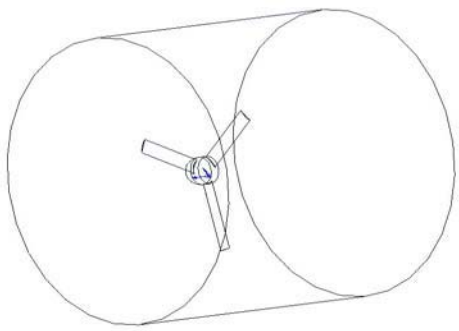

Figure 5. Model of the rotor and the surrounding airflow

First results

Due to the performed simulations the results make a comparison between the two chosen profiles possible. The results from the first simulation just content the quality of the aerodynamic performance and very coarse estimations about the resulting rotational speed and torque of the rotor.

Aerodynamic Performance

For the simulation of the aerodynamic performance of the two different blades, a stripe of each airfoil has been modeled. This stripe was surrounded with a box-shaped airflow. For each stripe the airflow for different pitch angles has been simulated. As values for the pitch angle the values -6°, -3°, 0°, 3° and 6° have been chosen. The wind has been modeled as a laminar airflow with a constant speed of 5m/s.

Regarding the resulting pressure and velocity distributions around the airfoils the results from the angle of incidence witch -6° seem to most useful. The velocity and pressure distribution above and under the airfoil show high varying values due to the lift and drag forces will probably the highest.

In a second set of simulation, with the -6° configuration and an increased inflow speed (10m/s) the turbulence intensity distribution of the two profiles has been regarded. These distributions show a higher turbulent intensity especially in the rear part of the "profile 2".

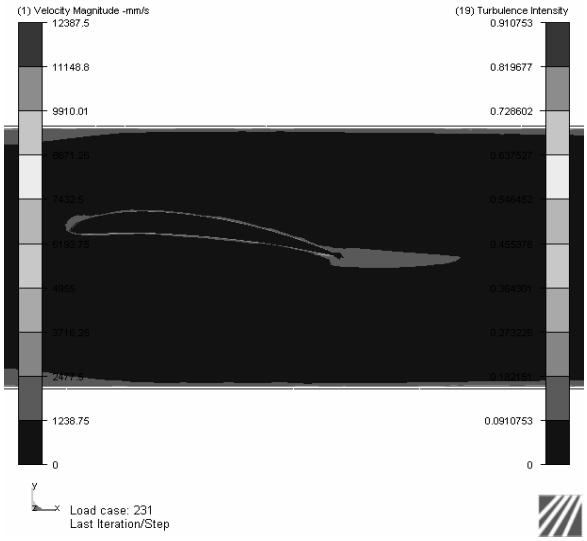

Figure 6. Turbulence distribution around the "profile 2" airfoil

Around the NACA 63-908 profile with the same configuration is nearly no turbulence intensity at all. Comparing the two different airfoils in the -6° configuration the NACA 63-908 airfoil shows advantages in the aerodynamic performance.

Coarse torque estimation

The results are only qualitative and show that the NACA 63-908 airfoil seems to have a higher resulting torque than the "Airfoil 2". The resulting torque of the NACA profile seems to be about 25% higher. This first estimation of the resulting torque is probably not reliable and has to be proven in further simulations and in wind tunnel experiments.

First Conclusions and outlook

The first simulations show that the NACA 63-908 profile seems to have some performance advantages compared to "Profile 2". So in the following these first results have to be verified. If the advantages of the NACA profile can be confirmed, this profile will be taken for further optimization iteration steps to find an optimal airfoil design. At the time of sending in this paper there were only results of a first set of simple simulations available. During the next time a first estimation of the power output of the introduced airfoil and its further modifications will be available. In addition the simulation model will be extended by the diffusor which is supposed to deliver a further increase of the power output.

These results shall be presented in addition to the first described results at the 20th Conference of Mechanical Engineering Network of Thailand.

Acknowledgments

This research was accomplished at the Thammasat University, Thailand with support from the University Karlsruhe, Germany.

References

[1] Ackermann, T., 2005. Wind Power in Power Systems. John Wiley & Sons, Ltd England.
[2] Jungbauer, A., 1998. Diplomarbeit: Windenergienutzung in einem regenerativen Energiesystem. Institut für Hochspannungstechnik, Eletrotechnik-Wirtschaft und Energieinnovation, Technische Universität Graz.
[3] ANSYS Inc., 2005. ANSYS CFX-Solver, Release 10.0: Theory. ANSYS Inc.
[4] Björk A. AERFORCE Subroutine Package for unsteady Blade-Element/Momentum Calculations. Technical Report TN2000-07, 2000. FFA Bromma Sweden.
[5] Ozgener, O., 2005, A small wind turbine system (SWTS) application and its performance analysis. Energy conversion and Management 47 (2006) pp. 1326-1337.
[6] Matsushima, T, 2004. Characteristics of a highly effective propeller type small wind turbine with a diffuser. Renewable Energy 31 (2006) pp. 1343-1354.
[7] Blue Ridge Numerics Inc., 2005. cfdesign user´s guide Version 8.0. Blue Ridge Inc.