Horizontal Axis Wind Turbine Blade Design

Ryoichi Samuel Amano[1] and Ryan Malloy[2]

[1] *Professor of Mechanical Engineering, University of Wisconsin-MilwaukeeWI,* amano@uwm.edu
[2] *Graduate Student, University of Wisconsin-Milwaukee, WI,* rjmalloy@uwm.edu

ABSTRACT

Recently there has been an increase in the demand for the utilization of clean renewable energy sources. This is a direct result of a rise in oil prices and an increased awareness of human induced climate change. Wind energy has been shown to be one of the most promising sources of renewable energy. With current technology, the low cost of wind energy is competitive with more conventional sources of energy such as coal. This paper explores the possibility increasing the number of profitable sites by optimizing wind turbine blade design for low wind speed areas.

Wind turbine blade profiles are often constructed using the Blade Element Momentum theory (BEM). This theory will produce the angle of twist and chord length for a given airfoil cross section and rotation speed at a finite number of positions along the span of the blade. From these two dimensional sections a three dimensional shape can be extruded. The BEM theory accomplishes this by treating a given cross section as an independent airfoil which processes wind with a speed and direction that is a vector sum of the on coming wind and the wind generated by rotation. Since the direction and magnitude of the wind generated by rotation changes as a function of span wise position, so too must the airfoil cross section. The BEM theory is not entirely accurate if the data for the airfoil cross sections that are used have not been corrected for rotational motion. It is for this reason that CFD analysis is necessary for new blade designs.

It was this method that was used to construct a straight edge blade prototype whose optimal on coming wind and rotation speeds were 7m/s and 20rpm. The blade has a length of 20m and uses a constant airfoil cross section NACA 4412. To test this new design the performance both blade will be measured using CFD at a wind speed of 10m/s. In addition to this straight blade a swept edge blade will also be tested. This swept edge blade will have the same characteristics as the straight edge except for the trajectory of the edge. Each cross section will have the same dimensions and be at the same distance from the hub as its corresponding section in the straight edge blade.

A diffuser shaped domain was chosen with a 120 degree slice taken lengthwise along the axis. Each side of the domain was given periodic boundary conditions. The front and top planes were given as velocity inlets. The rear plane was give as a pressure outlet. The domain extended 5 diameters upstream of the blade and 10 diameters downstream of the blade. The domain had a radial height of 5 diameters at the front and 8 diameters at the back. Several different mesh schemes were used in an effort to both resolve the boundary layer surrounding the blade and hub and obtain a computationally feasible domain. A rectangle around 4m a side and a length roughly 20m was constructed around the blade.

A start size of .05m was used at the surface of the blade along with a growth rate and maximum size of 1.3 and 0.5. The wedge containing this rectangle and the rest of the blade/hub was meshed with a constant density mesh of 0.5m. The rest of the domain was given a growth rate of 1.08 and maximum size of 10 extending from this wedge.

For the swept blade the rectangle was modified slightly to accommodate the geometry. This approach is based off of the work done by Mandas and Carcangiu[i]. The final mesh contained 1.7 million elements.

The inlets were given an undisturbed velocity with a set turbulence level. The periodic boundaries were set to rotational. Finally the fluid being chosen as a moving reference frame was given a rotational speed of 2.09rad/s. The Turbulence closure model used was the the κ-ω SST model. This model was used with success in a similar application by Ferrer and Munduante [ii]. After these analyses, it was demonstrated that lower pressures are achieved on the back side of the swept tip blade as opposed to the straight edge blade. This results in approximately an 8% increase in torque. It can be assumed that this increase in

torque is due to the different edge geometries since every other aspect of the blades are the exact same. This result can also not be attributed to the increase in surface area associated with the swept blade since this difference is negligible.

It was further shown that the pressure profiles are very much the same between leaned and straight blades. This is to be expected since the same flat surface area is exposed to the on coming wind which is normal to the plane in which the geometry difference lies.

Table 1: Surface Area Comparison

Edge Geometry	Surface Area(m^2)
Straight	89.3
Swept	90.77

Table2: Mechanical Torque Comparison

Edge Geometry	Torque on Shaft(kN-m)
Straight	14.8
Swept	16.1

Table 1 shows the surface area for both straight and swept blades. Both areas are quite similar to each other. From Table 2 it should be noted that both geometries generate considerably less torque than conventional large scale turbines such as NTK500/41 using LM 19.1 blades which generates a torque of approximately 47.1 kN-m at the same speed. The difference in torque may be attributed to the fact that BEM predictions aren't entirely accurate as stated before. This theory is helpful in determining an initial design with non calibrated airfoil data or new designs using calibrated airfoil data. Since these blades did not use calibrated airfoil data it is expected that they under perform. Future designs may incorporate several different airfoil cross sections at different points along the blade which is currently done in most commercially available turbine blades.

Despite this under performance it can still be concluded that a swept edge has the possibility of increasing toque at certain wind speeds since that was the only variable changed between these two cases. The reason for this is the lower pressures generated towards the tip on the down stream side of the swept blade as opposed to the straight edge blade. The swept geometry appears to have no effect on the pressure magnitude or distribution on the up stream side of the blade.

Future work will include running more trials at different wind speeds and with different degrees of sweep. It is of interest to better understand how the swept tip design affects torque generation at the stall point. It is generally accepted that a swept tip will delay the stall point but to what degree and its relation to the degree of sweep is unknown. Also changes in the cross section of the blade itself will be made as mentioned earlier as is typically done with conventional blades. Finally a loading analysis will be done to determine the effect a swept edge has on static loading. One issue with blades designed for low wind speeds is that they experience high stresses high wind speeds found in the occasional storm for example. It has been hypothesized that a swept tip will help relieve the stress found at the hub/blade interface with a span twist. It is this affect that will be investigated.

REFERENCES

---

[i] Mandas, N. and Carcangiu, C., 2006, "Numerical Prediction of Horizontal Axis Wind Turbine Flow," EWEC Conference No. 466.

[ii] Ferrer, E. and Munduante, X, 2007, "Wind Turbine Blade Tip Comparisons Using CFD", *Journal of Physics Conference series 7, 012005*

---

[3] Hansen M.O.L., 2000, *Aerodynamics of Wind Turbines*; James & James, London.