CFD Simulations of Flow over Airfoils:
An Analysis of Wind Turbine Blade Aerodynamics

By: John Hamilla, Mechanical Engineering
Advisor: Maria-Isabel Carnasciali, Ph.D.

Abstract

Wind turbines are rapidly becoming a hot topic when faced with today's sustainable energy challenges. As such, attention needs to be turned to analyzing the flow over such structures. Computational Fluid Dynamics (CFD) software was used to study the flow fields surrounding fixed airfoils. The simulations provided data on how air flows past various blade shapes. With more realistic simulations, progress can be made on evaluating unique designs.

Introduction

Wind turbines are becoming an increasingly popular solution to generate clean, sustainable, power. This emerging technology was the underlying focus of this study.

More specifically, this study aimed at gaining an understanding of the flow fields around a wind turbine structure. Computational Fluid Dynamics (CFD) software was used to simulate air flow past two distinct shapes. Given the flow fields, one could visualize the effects particular shapes have on the air stream. The air flow gives rise to lift and drag forces on the airfoils. Lift and drag play an important role on the performance of a wind turbine. A high amount of drag would inhibit a rotor's motion, ultimately wasting energy that is captured from the wind.

Background

Wind turbines transform the kinetic energy of the wind into mechanical energy. This mechanical energy is then turned into electricity. Wind turbines are tools to capture energy for later use.

Wind turbines come in various shapes and sizes. The focus of this project is primarily on horizontal axis wind turbines. These are the kind typically thought of and usually consist of propeller-like blades. These turbines are lift devices, as opposed to drag devices. Drag devices rely on the wind to push parts. This is relatively inefficient. Lift devices instead use airfoils. The rotors of this type of wind turbine are shaped to create lift as wind flows past. A component of the generated lift creates a torque on the blade which causes them to rotate around an axis. With this setup, the rotational speed of the blade can surpass that of the wind (*Wind Energy*).
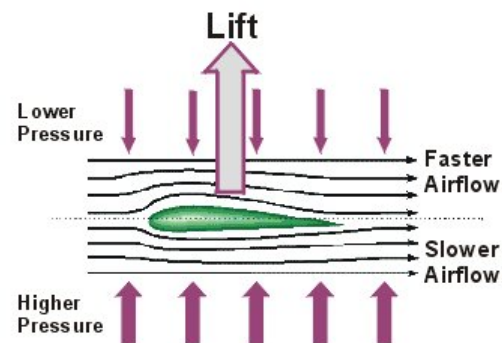

Figure 1 – lift generation from fluid flow over an airfoil

When air flows over an object such as an airfoil, the air stream is forced to move around it as depicted in Figure 1. Typically the object is shaped in such a way that an air stream has to travel further over one side than the other. This difference forces one air stream to travel faster than the other. The Bernoulli Effect tells us that as fluid's flow velocity increases, the static pressure it exerts decreases (Cengel). The

difference in flow speeds creates a pressure differential. The pressure difference creates a kind of circulation around the airfoil which speeds up the flow above and slows the flow below. This results in the force commonly known as lift. Lift is perpendicular to the wind direction (Figure 1).

In practice, an airfoil is not generally aligned with the air flow. The angle at which an airfoil makes with the oncoming wind is known as the angle of attack. Generated lift is heavily dependent on the angle of attack. The lift coefficient is linearly proportional to the angle of attack, and has an approximate slope of 2π/radian (*Aerodynamics*). There is a limit at which the angle of attack will no longer increase lift. At this point, the airfoil stalls. At high angles of attack the drag component caused by pressure, which is normally miniscule, becomes very large. The boundary layer of the air also begins to separate.

The drag force is created by the air as it flows over the airfoil. Because of the no-slip condition, a fluid exerts a tangential shear force on the surface of the airfoil. This force is in the direction of motion of the fluid. Normal pressure forces exerted by the air also have components which contribute to drag. Drag acts to oppose the motion of the airfoil. And in doing so, drag reduces the efficiency of a wind turbine.

Coefficients are used as a way to quantify the lift and drag forces on an object (Cengel). These coefficients are determined by the frontal area, wind speed, density of the fluid, and the force experienced by the object due to the fluid flow.

$$C_D = \frac{2F_D}{\rho v^2 A}, \qquad C_L = \frac{2F_L}{\rho v^2 A}$$

In the case of a wind turbine, the airfoils are mounted around an axis. As they rotate, the blades experience a combination of air flows. First is the obvious air flow from wind. Second is a flow due to the rotation of the blade. This flow from the rotation is, in an ideal case, perpendicular to the wind. The resultant vector sum of the flows is the wind experienced by the blade (*Wind Energy*). The vectors can be seen in Figure 2. From this wind, lift is generated. A component of this lift applies a torque to rotate the blade. The base of the wind turbine has to be strong enough to withstand the moments caused by

components of the remaining lift and drag, which act to push the structure back.

The nacelle is the enclosure behind the rotor hub. Inside the nacelle are some major components for
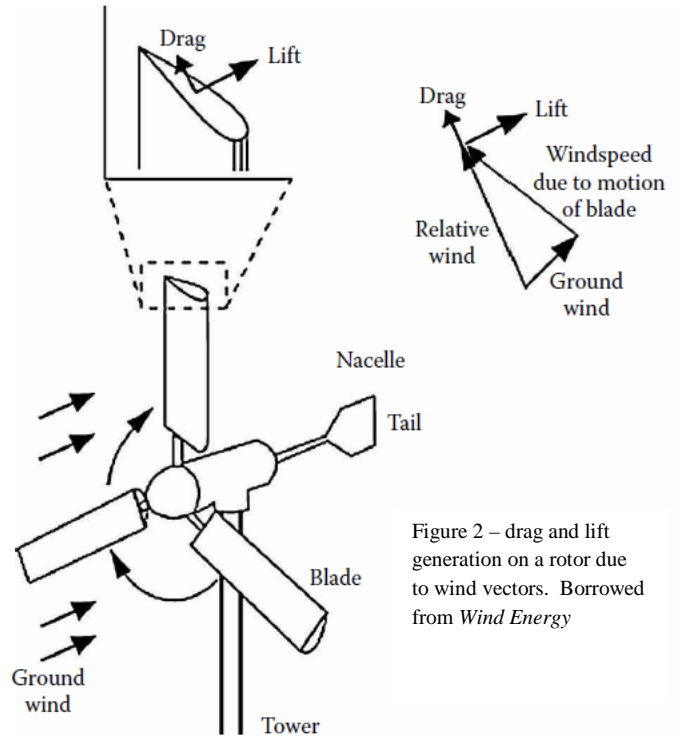

Figure 2 – drag and lift generation on a rotor due to wind vectors. Borrowed from *Wind Energy*

turning the collected energy into electricity. A gear box transforms the rotation from the rotors to a speed more suitable for the generator (*Wind Energy*). A tail is attached to smaller wind turbines, like in Figure 2, in order to ensure that the wind turbine faces the direction of the oncoming wind. Larger wind turbines accomplish this with a motor in the base of nacelle.

Large companies are spearheading the effort to create sustainable wind energy. General Electric continues to roll out a line of new wind turbines. These large structures are typically able to produce 1.5MW of electrical power, with some designs capturing 2.5MW or even 4.5MW. Their most available design, the 1.5MW 1.5-77 wind turbine has a 37m blade and is built to be either 65m or 80m from the ground (*Wind Turbines*). At this altitude, the turbine is expected to handle an average wind speed of 10m/s. Larger wind turbines are capable of harnessing the faster winds from higher altitudes.

Materials and Methods

CFD software was chosen to run the simulations. Before the simulations could be run, working CAD models needed to be drawn up. Solidworks® was used to create 3-D models of various rotor blades. The models were then imported into ANSYS® Workbench. Around each of these geometries a mesh was generated. The mesh can be thought of as 3-dimensional wire-frame where numerical solutions are obtained for the flow fields around the geometries.

The shapes used for the meshes were based on what was at hand. The three-bladed rotor and half-propeller blade are based off of simple toy pieces. It was not practical at the time to attempt to draft a wind turbine blade, especially since the dimensions were unknown and proved to be difficult to reproduce.

Shown in Figure 3 are the meshes used. The geometries were placed in differently oriented cylindrical boundaries. The mesh is intended to be used to calculate the properties of the fluid flow. Therefore it is necessary that the boundaries are far enough as to not to cause interference with the areas of interest.

Three-bladed Rotor (left)
- 1658019 nodes
- 6630360 elements
- 0.1932 minimum orthogonal quality

Single Blade (right)
- 134101 nodes
- 757635 elements
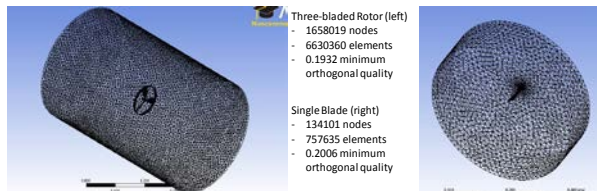- 0.2006 minimum orthogonal quality

Figure 3 – (left) wireframe mesh of the 3-bladed rotor, (right) wireframe mesh of the single propeller blade

Orthogonal quality was used to judge the quality of the meshes. Orthogonal quality is the measure of quality of an element. It ranges from 0 to 1 where 1 is that of a perfect cell. Generally a mesh with a minimum orthogonal quality of 0.1 is considered acceptable. An orthogonal quality of 0.2 is considered to be relatively good. Both meshes were adjusted to meet this standard. Solid cut-away views of these meshes are also shown in Figure 4.
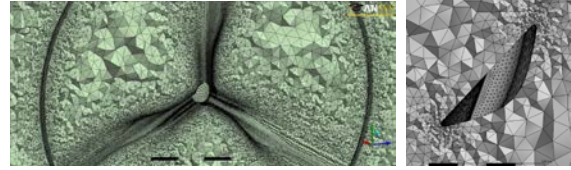

Figure 4 – close up section views of each mesh

With the meshes complete, it was possible to start setting up the simulations. The meshes were brought into Fluent, a program within the ANSYS Workbench. Fluent uses the meshes to model the fluid space. It solves the Navier-Stokes equations numerically at each node of the mesh. An iterative method is used by fluent to converge on a solution of the case.

In order to run the simulation, specific parameters needed to be set. First some general settings had to be established. For example, gravity was neglected, time was treated as a steady case, velocity was taken to be in an absolute reference frame, and the solver used would be pressure based. Next the turbulence model had to be selected. The k-epsilon method was used for most of the simulations run. When compared to other methods, such as the Spalart-Allmaras, it proved to have better convergence, and therefore more accurate results. The k-epsilon model is one of the most widely used models. More specific methods also had to be specified. Pressure, momentum, dissipation, and energy were all modeled using second order functions. These higher order functions are generally more accurate than first order approximations, but are also more time consuming.

Boundary conditions had to be set for the different areas of each mesh. The faces that would make up the objects (i.e. the rotor blades) were set to be treated as walls with zero velocity. The fluid could not pass through these boundaries. For the mesh of the three-bladed rotor, one of the flat circular faces of the enclosure was set to be a velocity inlet for the fluid. The remaining faces of the enclosure were set to be pressure outlets. Fluid is forced in at a given speed and leaves the mesh at the opposite end, or through the sides. The other mesh was handled slightly differently. The single propeller blade was oriented in a cylinder such that the air travels parallel to the flat boundaries. In this case, the curved face of the mesh was set to be treated as a velocity inlet. This allowed

for the specification of magnitude and direction of the fluid flow. The fluid flow was allowed to leave the system through the flat sides, which were also treated as pressure outlets. Ideally the meshes are large enough so that any flow patterns caused by the boundaries are not influential to the flow around the objects.

Reference values needed to be set within Fluent. Some values were temperature, enthalpy, pressure, velocity, viscosity, etc. Temperature was changed to 0 degrees Celsius. Pressure was set to 0Pa gauge or atmospheric in order to meet standard temperature and pressure conditions. Enthalpy was defined as 0kJ/kg, since the model as setup does not consider heat transfer. Viscosity is determined by the fluid, which is air. Velocity was altered between simulations. Simulations were primarily run with velocities of 3m/s, 5m/s, and 10m/s. These velocities are comparable to low altitude wind speeds in the West Haven area.
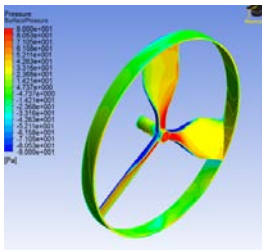
Results and Discussion

Figure 5 – surface pressure on the rotor

Once the meshes were generated, simulations were run in ANSYS/Fluent. These simulations yielded solutions to the three dimensional flow fields around the objects. It was decided that special attention would be given to the pressure and velocities of flow fields. Another part of ANSYS® Workbench, CFD Post, was used to
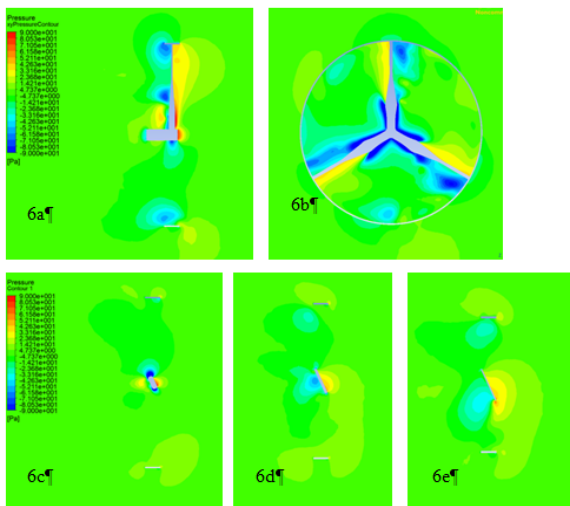
Figure 6 – Pressure contours in the plane of a (a) side section, (b) front section, and section of a blade taken at (c) 0.015m, (d) 0.025m, and (e) 0.035m from the axis of rotation. Flow is from right to left in 6a, 6c, 6d, and 6e. Flow is into the plane in 6b.

visualize the solutions. The simulations were run with varying wind speeds. Wind speeds of 3m/s and 5m/s seemed to provide the most consistent results. The rotor with three blades was run first. Because of
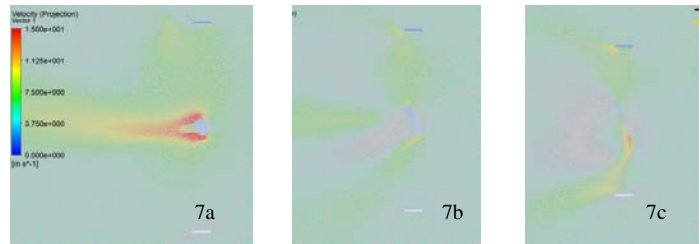
Figure 7 – Velocity vector plots at (a) 0.015m, (b) 0.025m, (c) 0.035m from the axis of rotation. Flow is from right to left.

the complexity of the structure, it proved to be the most difficult and time consuming to calculate. Focus will be primarily on the simulation that was run with a wind speed of 5m/s. The simulation was run for 500 iterations and reached a

Figure 8 – surface pressure on the propeller blade

residual continuity of 7.38E-04. The residual continuity is a measure of the convergence of the calculation. A residual continuity of 1E-06 is regarded as ideal, however smaller than 1E-03 is generally acceptable. This calculation returned information on the object's drag coefficient, which was determined to be 0.0112.
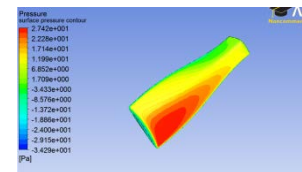
Pressure contours in different cross-sections are shown in Figure 6. It can be seen that the pressure increases as the air collides into the rotors. At the same time, pressure directly behind each blade drops significantly. The pressure also varies along the radius (Figure 6c – 6e). The air pressure is dramatically lower closer to the axis of rotation. Because these simulations treated the object as stationary in space, it makes sense that air pressure would be lower at this part. This indicates that the air is moving faster around the thinner part of the blades. This is verified in the velocity vector plots –shown in Figure 7. The air is set to normally travel at 5m/s. Around the root of the blade, this speed accelerates up to 15m/s.

The second series of simulations focused on flow over a single propeller blade. This geometry is a more developed airfoil than a comparable blade taken from the previous geometry. This airfoil was also tested with varying wind speeds. Results from the 5m/s simulation will be discussed. This particular simulation ran for 2500 iterations and had a residual continuity of 3.93E-03. The convergence is of an acceptable order of magnitude. The simulations for this geometry were calculated much more quickly due to simpler geometry. Therefore more iterations were able to be calculated in a given time. This airfoil in this orientation had a drag coefficient of 0.0373.
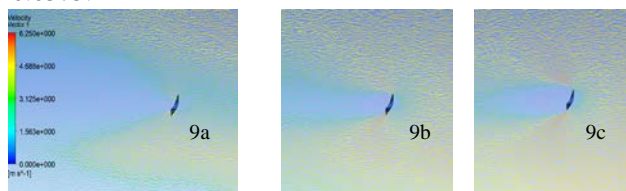

Figure 9 – velocity vector plots at (a) 0.02m, (b) 0.04m, and (c) 0.06m from the axis of rotation.

In this unique orientation, it can be seen (Figure 9) that the air travels faster when it is forced to travel more distance. The velocity of the air is much higher in the area where the flow separates. Closer to the root of the blade, the flow is more conservative, and does not separate as dramatically. Behind the entire length of the blade, there is a pocket of stagnant air. This is caused by the separation of the flow fields.

Next Steps

With multiple simulations done, attention can be directed into further improving the simulations. One proposed solution would involve creating dynamic meshes, or meshes that would deform and rotate, better modeling how rotor blades actually behave.

With more accurate models, a conservation of energy method may allow for the determination of power captured by a rotor set up. From there, tests can be done to see how altering details of a rotor may change its efficiency.

Once some realistic designs are created, it may be possible to make prototype airfoils with the 3D printer. This would allow for wind tunnel testing to validate the models.

References

Aerodynamics of Wind Turbines. Earthscan, 2008. Retrieved from www.knovel.com.

Cengel, Yunus, John Cimbala, and Robert Turner. *Fundamentals of Thermal-Fluid Sciences*. New York, NY: McGraw-Hill, 2012. 4th Edition.

Wind Energy: Renewable Energy and the Environment. Taylor and Francis Group, LLC, 2009.

Wind Turbines. General Electric Company. Web. Aug. 2, 2012. <http://www.ge-energy.com/>

Biography

John Hamilla is currently in his junior year as a Mechanical Engineering major and mathematics minor at the University of New Haven. After graduating, he plans to continue his studies in graduate school while also working as an engineer. Research has taught John the value of determination and reminded him of his passion for learning. In his free time, John enjoys playing tennis and golf with friends.