Paper ID #45650

Relating Kinetic Energy Changes to Power Generation in a Mechanical Engineering Wind Turbine Lab

Dr. Chuck H. Margraves, University of Tennessee at Chattanooga

Dr. Chuck Margraves is a UC Foundation Associate Professor and Graduate Coordinator of Mechanical Engineering at the University of Tennessee at Chattanooga. His current research focus is on STEM Education, particularly in the area of energy sustainability, at the collegiate and high school levels.

Prof. KIDAMBI SREENIVAS, University of Tennessee at Chattanooga
Trevor S. Elliott, University of Tennessee at Chattanooga
Lance Isaac Rose, University of Tennessee at Chattanooga

Relating Kinetic Energy Changes to Power Generation in a Mechanical Engineering Wind Turbine Lab

Abstract

All mechanical engineering students at the University of Tennessee Chattanooga are required to take a senior-level experimentation lab that covers topics in multiple areas of Mechanics as well as Thermal Fluid Sciences. Two faculty members teach the course, consisting of a one-hour course and a three-hour lab. One of the main goals of this course is to reinforce much of the undergraduate material students have covered in the program's first three years. Recently a Wind Turbine experiment was added to the lab that demonstrated the effectiveness of blade orientation to power generation for a range of wind speeds. The work provided in this study seeks to expand the effectiveness of this lab by demonstrating how the change in kinetic energy across the blades, measured using a pitot tube, is converted into rotational power, and finally into electrical power using a generator. One of the benefits of this experiment is that it allows the faculty to introduce both thermal fluid science concepts as well as mechanics in a single experiment, unlike many of the other labs, which tend to focus on only one area. Results showed that the change in kinetic energy across the blades resulted in approximately 77 watts of power decrease, due to an increase in cross-sectional area, losses, and turbine extraction, while the average rotational power was measured to be 21 watts. The electrical power produced was less than 1 watt showing the low conversion efficiency for this particular generator. The second portion of this study aimed to determine the power generation numerically using the commercially available Computational Fluid Dynamics (CFD) Software, Fluent. While good agreement was found on the calculated rotational power from the model, changes in kinetic energy were substantially different than those measured. Future work on the computational model is currently being considered.

Introduction

At the University of Tennessee Chattanooga, all mechanical engineering students participate in a senior-level experimentation lab that covers a broad range of topics in mechanics and thermal-fluid sciences. One of these labs focuses on alternative power generation using a wind turbine provided by Turbine Technologies Windlab™. Students observe the changes in power output by the turbine's generator at various loads, blade angles, and wind speeds. What is not explicitly addressed in the lab is the actual change in kinetic energy across the blades. In prior research conducted on this turbine, one study examined how blade design and orientation affect power extraction [1]. The University of Queensland also developed modules for use in an academic setting for this lab [2]. Neither, however, looks specifically at the origin of this energy or how it is transformed into what eventually becomes electrical power. Thus, the first objective of this research is to demonstrate the actual change in kinetic energy across the turbine blades, which results in electrical power production. A second objective is to quantify losses in the system by comparing the measured changes in kinetic energy to the power delivered to the blades in the form of torque and rotational speed. A final objective is to demonstrate the benefit of creating an accurate Computational Fluid Dynamics (CFD) model to show students how computational models can be used to improve the efficiency of future designs.

Experimental Setup

The Turbine Technologies WindLab™ consists of three main parts: a motorized fan to drive airflow, a cylindrical constant area wind tunnel equipped with a flow straightener at the entrance to the tunnel, and a three-blade wind turbine-generator combination stationed at the end of the tunnel. The turbine apparatus is shown in Figure 1 below.

Figure 1: Depicts the entire wind turbine apparatus

As stated in the introduction, the first objective of this work is to measure the kinetic energy change across the turbine blades. This is completed by applying Equation (1) to the measured velocities:

$$\Delta \dot{KE} = \frac{1}{2}\left(\dot{m}_1 {V_1}^2 - \dot{m}_2 {V_2}^2\right) \quad (1)$$

where $\Delta \dot{KE}$ is the change in the rate of kinetic energy,
$\dot{m}_1$ and $\dot{m}_2$ are mass flow rates before and after the turbine (should be equal),
and V_1 and V_2 are the average velocity before and after the turbine.

The mass flow rate may be approximated by Equation (2) below:

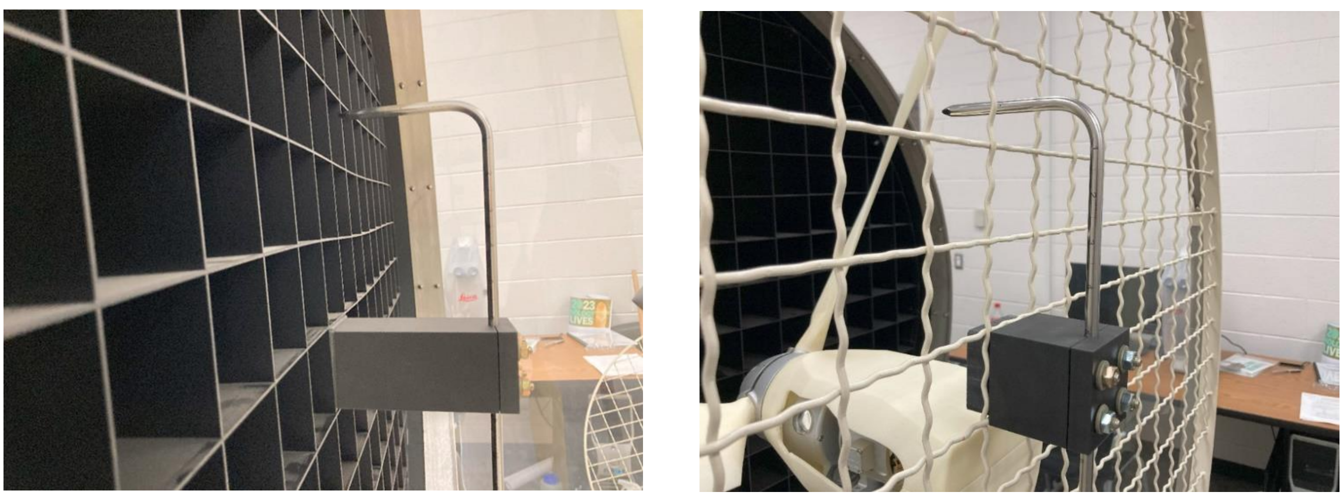
$$\dot{m} = \rho V_{avg} A_c \quad (2)$$

where ρ is the density of air,
V_{avg} is the average velocity across A_c,
and A_c is the cross-sectional area.

To solve equations (1) and (2), the average air velocity before and after the turbine must be known, so a device capable of measuring relatively low wind speeds was purchased. Knowing that the WindLab™ can safely run at wind speeds of around 10 m/s, a manometer (PCE-PDA

01L), coupled with a pitot tube, was chosen. This device had an upper-velocity limit of approximately 18m/s and an accuracy of one percent. This device's range would ensure that the wind speed readings would fall on the scale in the mid to upper range yielding a less significant error. The dimensions of the pitot tube were measured so that mounts, allowing for the systematic measurement of wind speed, could be created. Since the upstream flow straighteners were of a different cross-section compared to the downstream protective grid, two different-sized mounts (designed to hold the pitot tube) that could be inserted into the appropriate openings were 3D printed.

Before taking wind speed measurements, the angles of the three turbine blades were set at forty-five degrees according to the scale markings at the base of each blade. The generator excitation was set at approximately 7 volts, all three loads were set at 50 ohms, and the windspeed was adjusted to 10m/s according to the onboard control panel. Next, individual wind speed measurements were obtained at the center of each opening of the downstream protective grid and the upstream flow-straightener grid. These measurements were then used to calculate the average wind speed before and after the turbine blades. This resulted in two hundred and fifty-six points of measurement on the downstream side, and one hundred and eighty points on the upstream side of the turbine. The tip of the pitot tube protruded approximately 5cm upstream into the squares of the protective grid and resided just downstream of each square of the flow-straightener grid. While obtaining the upstream measurements, the turbine blades were held stationary to gain access to the flow-straightener grid. Figure 2 shows the point of measurement for the upstream and downstream locations.



(a) (b)

Figure 2: Pitot tube at the (a) upstream and (b) downstream locations.

The second goal of this work was to measure the rotational power of the blades based on the torque and rotational speed. The torque is calculated using Equation (3).

$$\tau = Fr \tag{3}$$

where τ is torque,
F is the force on the blades,
and r is the radial distance from the center of rotation.

After obtaining the torque, the rotational power can be calculated using Equation (4).

$$P_{rot.} = \tau\omega \tag{4}$$

where $P_{rot.}$ is the rotational power
and ω is the angular velocity.

To obtain the torque in equation (3), it became necessary to devise a method for estimating the net or total force on the blades. Thus, at the same air speed setting of 10 m/s (as reported by the turbine) at which the manometer measurements were taken, the net torsional force exerted on the three blades was estimated by hanging a spring gauge vertically from the enclosure wall and attaching it to one of the blades. The spring gauge was made plumb using a level, and the blade was considered horizontal when its perpendicularly oriented hub collar was also made plumb using the level. In this position, the radial distance (r) was measured from the hub shaft's center to the point where the spring gauge was attached to the blade. The net torsional force was measured at two different radial distances on the same blade, and an average value was obtained. Figure (3) shows the measurement setup.

(a) (b)

Figure 3: The two images show how (a) the spring gauge, and (b) the point at which the radial distance was measured

Simulation Setup

Initially, for the Computational Fluid Dynamics (CFD) work, Turbine Technologies provided an STL file of a single turbine blade. Because the STL format is computationally demanding, due to the large number of independent triangles, efforts were undertaken to recreate a simpler geometry in SolidWorks. The result was a greatly simplified blade created in a STEP file format, which was nearly dimensionally identical to the original. A model of the blade hub was also received from Turbine Technologies and recreated. With the two simplified parts, a full three-blade turbine assembly was modeled, with the same forty-five-degree blade angle used during the experimental work. The next step was to create a dimensionally accurate replica of the fluid domain. Because the flow straighteners effectively reduce the cross-sectional area of the upstream velocity field, the domain was shaped like a perfectly symmetrical diffuser with a smaller upstream cross-section and a larger downstream cross-section. Figures (4) and (5) below depict the turbine's geometry and flow domain respectively.

Figure 4: Computational model of the three-turbine blade geometry used in the CFD simulation.

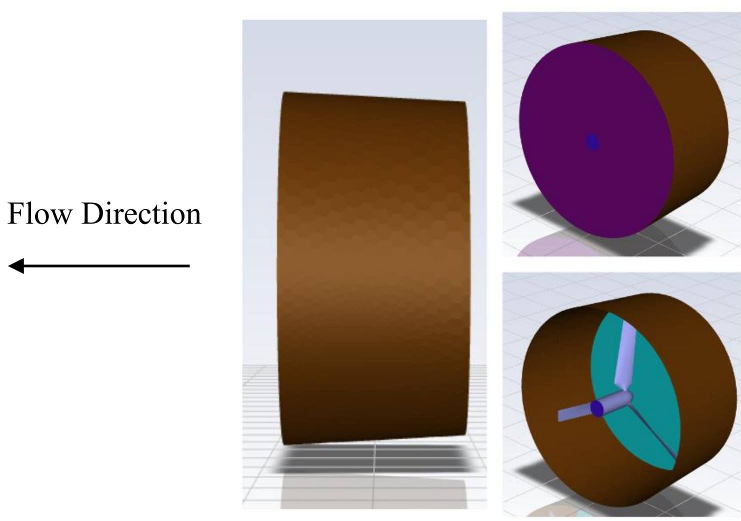


Figure 5: The images above depict the tapered shape of the fluid domain and the orientation of the turbine model within the domain.

The last geometric consideration was to ensure that the contact surface between the blade wall and the fluid was well-defined by manually sharing topology between the two bodies. The next step in this simulation was to create a mesh, using Fluent's Meshing software [3]. The blade wall was treated as a body of influence with a relatively small cell size on and around its walls growing to a larger cell size along the boundaries of the domain. By sharing the topology during the geometry stage, a conformal mesh was ensured. The next consideration was to set the cell-zone conditions for the simulation, and after some real-world testing, it was decided to give the fluid domain and the blade wall an angular velocity of one hundred ninety-four revolutions per minute (based on experimental results) relative to the enclosure. This, along with the assumption that the enclosure wall is perfectly symmetrical, allows the simulation to be considered a steady state as the orientation of the enclosure wall relative to the blade and domain appears unchanging at any instant in time. The last step needed to complete the simulation was to define the remaining boundary conditions which included the following: a uniform inlet velocity of 8.00 m/s (rounded from the experimental result of 8.06 m/s), a pressure outlet at atmospheric pressure (zero-gauge pressure), and a no-slip condition at the enclosure wall. The selected inlet velocity was based on the pitot tube anemometer's average reading rather than the turbine's reported wind speed of 10 m/s.

Results

After collecting the necessary data and employing the equations above, the calculated results of this experiment may be examined. Table 1 provides experimental values for the cross-sectional area, average velocity, and mass flow rate for the upstream and downstream locations. A velocity drop across the turbine of approximately 1.23m/s resulted in a rate of change in kinetic energy of approximately 77 watts due to an increase in cross-sectional area, losses, and turbine extraction.

Table 1: Change in the Rate of Kinetic Energy (Experimental)

Density of Air (kg/m^3)		1.20		
Location	Cross-Sectional Area (m^2)	Average Velocity (m/sec.)	Mass Flow Rate (kg/sec.)	Change in Rate of Kinetic Energy (watts)
Upstream	0.766	8.06	7.4	76.6
Downstream	0.858	6.83	7.1	
		Relative Error (%) =	4.2	

The calculated mass flow rate before and after the blades showed a reasonable correlation with an approximately 4% difference. This difference could be related to the fact that a constant density was assumed across the blades as it was not possible to get local pressure calculations to account for the change in density as the flow slowed down. It should also be noted that the blades had to be held in a fixed position when making the upstream velocity measurements. Also, these measurements were made approximately 0.27 diameters from the tip of the leading edge of the turbine blade. Ideally, the velocity measurements would be made just before and just after the blades to minimize any losses created by other factors. However, this was not possible under our current setup.

The next experimental results obtained were derived from the spring gauge measurements. Table 2 includes the measured forces with their associated nondimensionalized radial distances, as well as the calculated torque, angular velocity, and rotational power for each of the two measurement locations. The radial distance is normalized by the maximum radius of 0.5m. The average rotational power was calculated to be approximately 21 watts.

Table 2: Rotational Power (Experimental)

Force (N)	Normalized Radial Distance	Torque (N*m)	Angular Velocity (Rad./sec)	Rotational Power (watts)
2.9	0.68	0.99	20.3	20.0
5.2	0.41	1.07	20.3	21.8
			Average Rotational Power =	20.9

As discussed previously it was not possible to measure the kinetic energy just before and just after the blades as desired. Therefore it is difficult to directly compare these results with the changes in kinetic energy. Because of the large support structure needed to hold the turbine in place, there are potentially large losses associated with air movement around it. There is also a slight increase in the flow area downstream of the turbine. Finally, the electrical power measured from the generator was less than 1 watt, which can be attributed to the low efficiency of the generator that is being used.

In addition to the experimental results, the CFD simulation also provided values for the change in the rate of kinetic energy and the rotational power. Figure 6 shows the velocity field near the turbine blades.

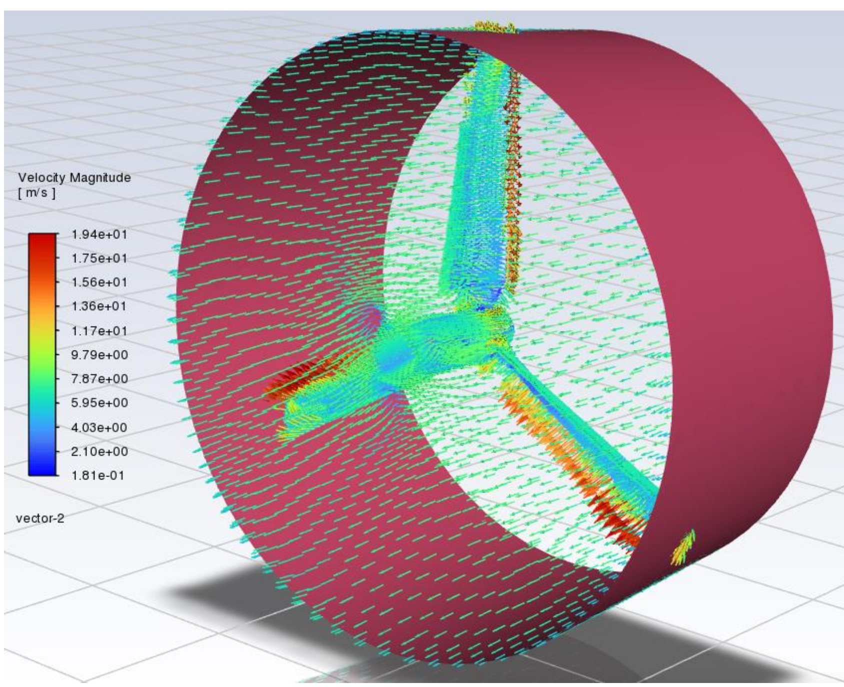


Figure 6 – Velocity field near the turbine blades.

Table 3 presents the cross-sectional area, average velocity, and mass flow rate for the same upstream and downstream measurement locations from the experimental setup. The result was a change in the rate of kinetic energy of approximately 45 watts.

Table 3: Change in the Rate of Kinetic Energy (Simulation)				
Density of Air (kg/m^3)		1.20		
Location	Cross-Sectional Area (m^2)	Average Velocity (m/sec.)	Mass Flow Rate (kg/sec.)	Change in Rate of Kinetic Energy (watts)
Upstream	0.766	8.00	7.3	45.4
Downstream	0.858	7.18	7.3	

The torque and rotational power were also obtained from the simulation software. The reported torque was approximately 1 newton-meter, and the rotational power was approximately 20 watts.

Table 4: Rotational Power (Simulation)		
Torque (N*m)	Angular Velocity (Rad./sec)	Rotational Power (watts)
0.997	20.3	20.3

In summary, the downstream velocity, torque, and rotational power values were all very similar for both the simulation and the experiment. The deviation between the change in rate of energy values, however, was more significant. Table 5 compares these values based on their relative error, treating the experimental value as the true value and the simulation value as the estimated value.

Table 5: Summary					
Data Source	Upstream Velocity (m/s)	Downstream Velocity (m/s)	Torque (N*m)	Rotational Power (watts)	Change in Rate of Kinetic Energy (watts)
Experimental	8.06	6.83	1.030	20.9	76.6
Simulation	8.00	7.18	0.997	20.3	45.4
Relative Error (%)		5.2	3.2	3.1	40.8

While a reasonable correlation between the simulation and experiment is shown in Table 5, there is a variation that requires some explanation. One reason for the simulation's slightly higher downstream velocity may be the use of simplifying assumptions when creating the simulation model. The model assumes a perfectly symmetrical enclosure and neglects to include such factors as surface variation or the turbine's stationary tower. The wake from this tower, for example, could lead to additional air speed reduction which the simulation does not account for. The model also imposes an experimental angular velocity on the turbine blades rather than allowing them to freely spin. This would likely cause the air to not expend as much kinetic energy across the blades. The slight difference in torque may have to do with the fact that the experimental torque was measured with the blade stationary.

Class Implementation and Learning Objectives

There are two different courses where the work completed from this research may be implemented in the classroom. The first is in the lab discussed above. During the one-hour lab preparation portion of the class, the experiments conducted and the CFD results will be presented as part of the lab material for the week. This material will then be assessed as part of a pre-lab quiz taken before the start of the lab. Students will also be shown the instruments used to collect the data from the thermal fluids side (pitot tube) and a mechanics side (spring gage) during the lab. Unfortunately, due to the time to take all the data it will not be possible to repeat the experiments during the lab.

The second class where this material may prove useful, is a new senior level elective course that will focus on differing numerical techniques involving both solid and fluid mechanics. This class is currently under development and the work completed here is being considered for a module of the class that will focus on comparing experimental and numerical work. The main thrust of this work is to help students link numerical simulations to actual physical results. One of the main learning objectives, particularly of the elective course, will be to teach students how to assess the accuracy of a numerical simulation.

Conclusion

The primary objective of this work was to demonstrate how a drop in kinetic energy across turbine blades results in the generated power of a wind turbine. By using a pitot tube and simple mounting apparatuses, a significant drop in the average wind speed resulted in a maximum drop of 76.6 watts, due to an increase in cross-sectional area, losses, and turbine extraction. Such a clear demonstration could give students a deeper and more tangible understanding of energy transfer. The second objective was to demonstrate how the change in kinetic energy is initially transferred to rotational power as well as losses accrued during this process. The measured torque, and known rotational velocity, resulted in a power of 20.9 watts. This represented a loss of approximately 55.7 watts from that supplied by the change in velocity of the air. The third and final objective was to show students the potential benefit of tying a computational fluid dynamics simulation to an experimental setup. While the results were mixed when compared with the experimental results, the addition of the computational model can greatly enhance the student's understanding of this important engineering tool.

References

[1] A. Khalid and C. Roper, “Research Learning Experience while Designing Wind Turbine for Low Speed Wind Applications”, 2016 ASEE Annual Conference & Exposition, June 26-29, New Orleans LA, 2016.

[2] M. Evrat and R. Sharma, “Laboratory Modules For Wind Turbine Experiments Using the WindLab Facility At The University of Queensland”, School of Information Technology and Electrical Engineering, Power and Energy System, The University of Queensland, St. Lucia, QLD, 2015.

[3] Ansys® Fluent with Fluent Meshing, Release R2, ANSYS, Inc., 2024.