California State University, Chico

Collegiate Wind Competition Team 2020

CHICO STATE
Alternative Energy Club

CHICO STATE
ENGINEERING

Technical Design Team:

Kate Gordon: Mechanical Engineering Student, Design Lead: gkate7997@gmail.com

Keenan Allen: Mechanical Engineering Student Lead: allenkeenanj@gmail.com

Alize Hall: Mechanical Engineering Student Lead: alizehall97@gmail.com

Project Development Team:

John Fielding: Construction Management Student, Co-design Lead: jfielding@mail.csuchico.edu

Nicholas Breuer: Mechanical Engineering Student, Co-design Lead: nbreuer1@mail.csuchio.edu

Dr. David Alexander: Faculty Advisor for Alternative Energy Club: dgalexander@csuchico.edu

May 20th, 2020

Department of Mechanical and Mechatronic Engineering and Sustainable Manufacturing

California State University, Chico CA 95929-0789

Table of Contents

Executive Summary

The Collegiate Wind Competition (CWC) is a national college level competition hosted by the U.S. Department of Energy and California State University Chico has been attending for the past 4 years. A senior level design engineer group from Chico State's Senior Project class was tasked in creating a new turbine for the 2020 competition. The CWC club on campus was then to take over the turbine and attend the CWC in hopes to win the Spring 2020 event. The senior project group was tasked to create a wind turbine to win the competition, as well as leave behind adequate and useful information for future Chico State teams.

The turbine design was centered on maximizing points explained in this document. This was done by research, prototyping, manufacturing, and testing. Research involved equating team members with what was already tested in large scale practice as well as previous winning designs from competition. Additionally, the group evaluated all aspects of the assembly including blade optimization, how to stop the turbine in an emergency situation, spec a generator to maximize power within our parameters, design of the mounting system, bearing choice for mounting, design of the nacelle, and lastly, a 3D printed duct to maximize air speed.

Calculations were completed for the bearing, turbine blades, duct/diffuser design in comparison to turbine blades without the duct, wind speed and factors of wind speed on the entire system, the duct frame, shafts, brakes, and forces on the entire system as a whole. The team completed a final thorough analysis of the completed design.

Wind Turbines have been designed and built for the past 4 years at CSU Chico, however, none have performed exemplary in the CWC. With an understanding of the Rules and Requirements provided by the CWC and many iterative tests performed in the wind tunnel, winning is very possible for the 2020 competition. As a team, it was agreed upon that the turbine needed to be a horizontal axis, downstream design, able to generate approximately 30+ Watts of power through the generator. By improving design through iterations and maximizing efficiency in the wind tunnel a superb product has been provided by CSU Chico that has a shot at winning the competition.

Design Objective

The turbine design satisfies all rules and requirements presented by the U.S. Department of Energy 2020 Collegiate Wind Competition. More specifically, each task of the competition and component of the turbine was thoroughly justified conceptually, analytically, by ANSYS simulation and SolidWorks FEA, and by physical experimentation. For example, examination of three of the primary components can be considered to better conceptualize the process that formulated the final design choices.

1. Blades: The blades were first conceptualized to have a greater width to length ratio than conventional full-scale wind turbines in aim to lower the cut-in wind speed for the "Cut-In Wind Speed Task." It was determined that although efficiency at high wind speeds may be decreased by doing so, at lower wind speeds it would increase which would benefit the overall performance score. This design choice was also justified utilizing excel and the competition point factors in which it was found that power generation at low speeds (2.5-5m/s) could substantially improve our score. As for simulations, Q-Blade was applied but was eventually dropped due to inaccuracy of results. Physical tests in the Chico State wind tunnel were then conducted which confirmed the desired results from blades that were reverse-engineered and 3D printed from the Energy Systems lab.
2. Duct/Diffuser: The Duct/Diffuser component was conceptually determined to capture more energy in the wind than free stream horizontal turbines by utilizing the corners of the square size constraint. Additionally, the component was conceptualized to reduce turbulence along the blade tips and to increase wind speed to the blades. Analytically, using the power in the wind and Bernoulli's equation (Equation 2.1, Figure 2.1), it was determined that increasing the area of the duct and decreasing the diameter of the turbine blades resulted in much greater power output. However, these analytical solutions didn't account for numerous factors and it was desired to keep the reverse-engineered blades of 30cm. After researching duct/diffuser cross sections and successful geometries, a solid model was input and iterated upon in ANSYS until the final design was concluded. To test the validity of the simulations, a small-scaled physical model experiment was performed in the Chico State wind tunnel, which confirmed our findings.
3. Turbine Tower: The Turbine Tower was designed with the intent of withstanding (minimal deflection 0.0005in) the forces of drag created by the large duct/diffuser mounted to its top surface (12.3 lbs). Additionally, the component's connection to the mounting plate of the duct/diffuser was required to rotate freely to satisfy the yaw requirement of the competition. To encompass these requirements, it was conceptualized that internal bearings were necessary and that the component be hollow to allow hydraulic lines and wiring to be passed through. Analytically, bearing calculations were applied to determine the minimum required forces the components had to withstand but after numerous design changes and consideration, the construction of the Turbine Tower was extremely overbuilt. Utilizing a face-mount crossed-roller bearing and thick steel tubing, deflection is nearly negligible. The reason for so greatly over engineering this component was not only to achieve little deflection, but was also to be sturdy enough to last for future collegiate wind competition teams to utilize. In depth, numeric results were concluded using SolidWorks FEA.

Static Performance Analysis

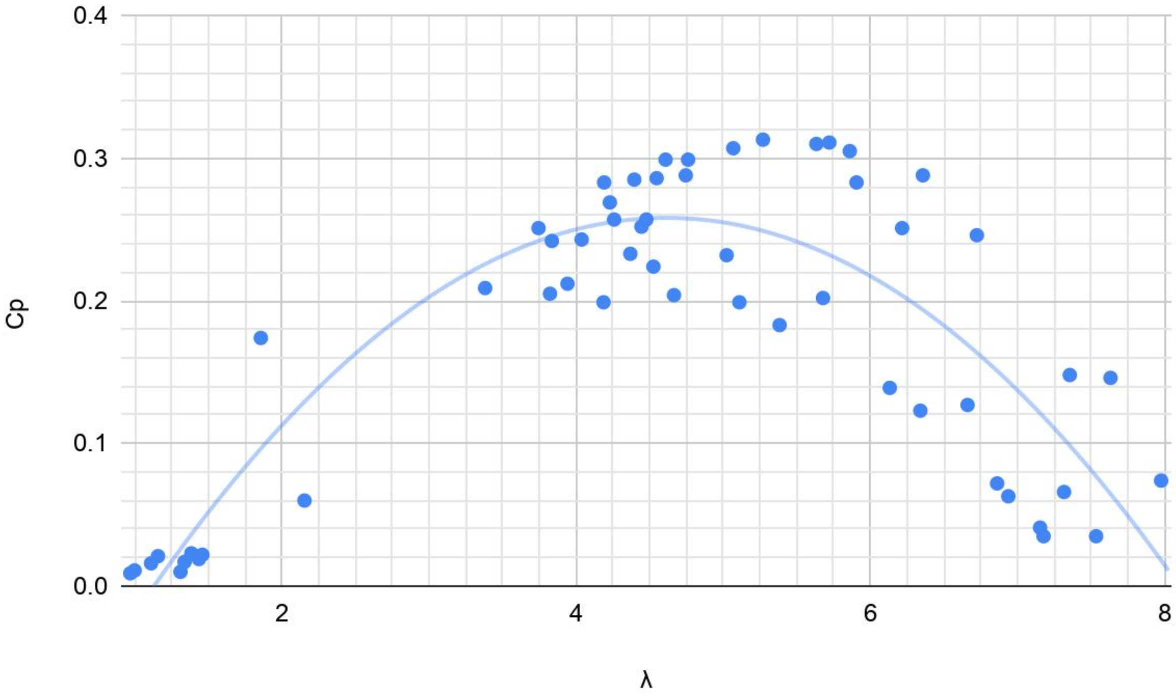


Figure 1: C_p *vs. Lambda*

Expected Mechanical Loads and Associated Safety Factors

Once the turbine was completed in Solidworks, we performed a full analysis to look for forces on the weakest parts of the system, which we calculated to be the duct. Originally a value of 10 pounds was calculated through Ansys, but the actual force was measured in the wind tunnel at Chico State to be approximately 2 pounds. The support system of the duct was revised after concerns of the system not visibly showing strength. This concern led to a steel plate being added around the 3D printed duct for extra support and rigidity.

Blade strength was another concern for the project. After some testing with PET-G plastic 3D printed blades, and the blades failing below runaway conditions, ideas were brainstormed for new modifications to the current design. Just before the school campus was closed, carbon fiber 3D printed blades were in the works to be printed and tested. A modified design with a thicker, stronger base towards the nose cone (where the previous set of blades broke) was constructed and was also to be tested against the carbon fiber printed blades (with the old blade design). This comparison would have reflected a good depiction between the two designs to let us know if the change in 3D printed materials is worth the effort or if blade design is a better idea. If both of these designs failed at runaway conditions, the stronger design of the two was to be modified with a metal rib running through the length of the blade for added rigidity.

All designs were to be tested in the Chico State wind tunnel as well as tested without the 3D printed duct due to possible damage of the duct if blades are to fracture. Once a final design was to be completed the blades would have been tested in the wind tunnel with the duct.

Turbine Yaw System

The turbine yaw system uses a damped bearing and downstream blades to make changes for yaw. Having blades oriented downstream creates a sail effect where drag forces equalize in the central position, thus whenever there is rotation it adjusts to the same downstream position in the wind. Tests were conducted on the yaw system to test the behaviour.

Electrical Analysis

Generator Model: The 24V version of the Faulhaber 3564-B was selected for several reasons. The first reason was the max operating RPM close to 10,000 which was needed to avoid the issue of burning out a motor as well as to take full advantage of the high speed of the blades. The second major reason was the small form factor which would allow for minimal space required for the nacelle and improve the potential for capturing air. Finally, the 24V model was selected because it operated closest to the desired RPM as well as leaving ample voltage headroom for powering any 5v electronics.

Nominal Voltage	24	Volts
Rated Power	126	Watts
Back-emf	2.08	mV/RPM
Current constant	0.05	A/mNm
Max Speed	29000	RPM
Dimensions: (Not including Shafts)	35 OD: 64 Length	mm

Table 1: Generator Specs

Power electronics

Since the chosen generator is a 3-phase brushless motor it has to be rectified before it reaches the PCC. The solution was designed and modeled using software and has yet to be tested and optimized. The solution is a full-wave rectifier using diodes, inductors, and capacitors

Electrical Load

The electrical load is a simple resistor which allows for easy analysis during the design stage. Additionally, on the turbine side of the electronics is a small load in the form of a servo and linear regulator which will account for a portion of the load.

Durability

The use of a buck-boost converter and linear low-dropout regulator is used for regulating the voltage into the load at or below 5 volts.

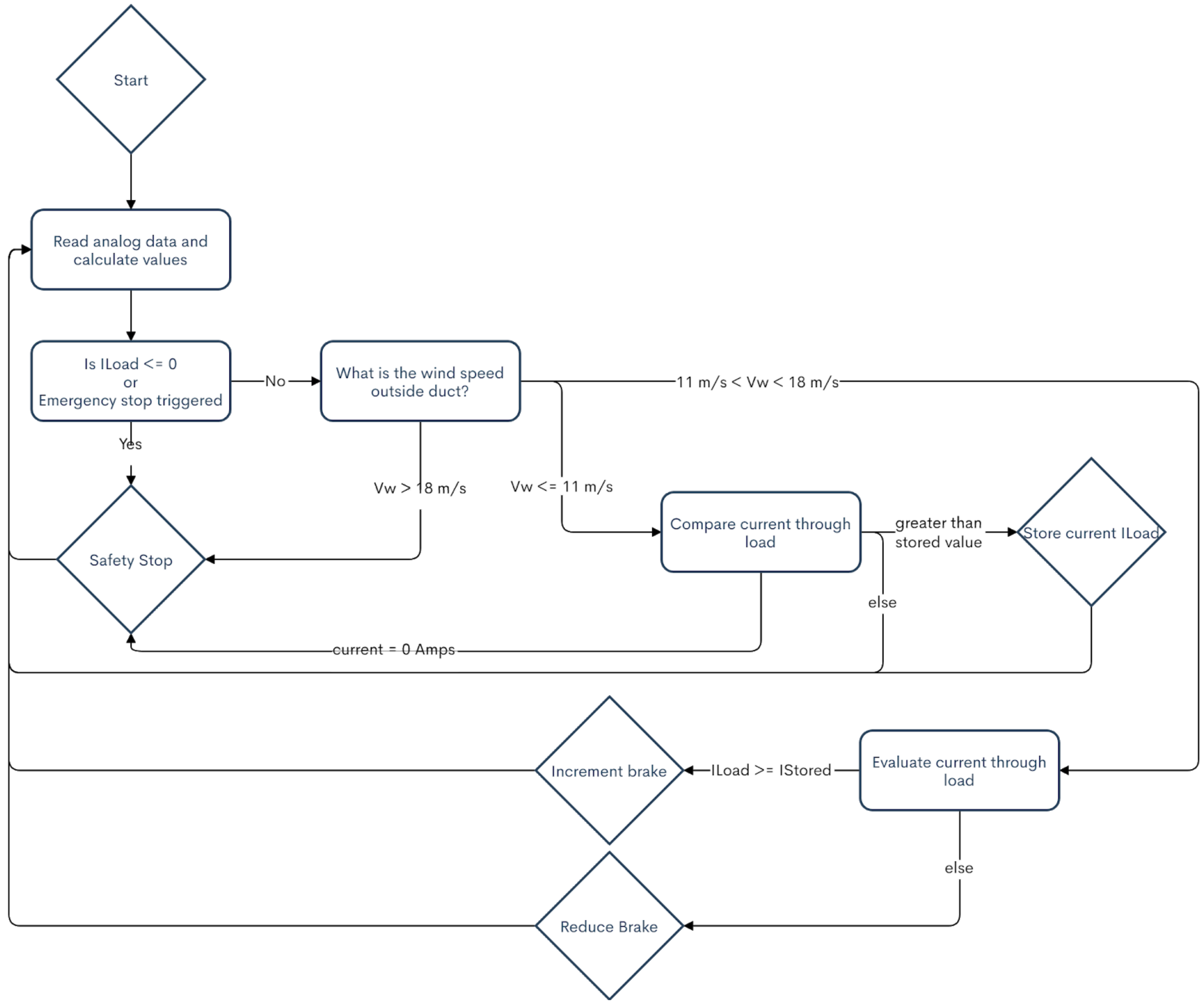


Figure 2: Control Diagram

Specific operational states include;

- Collecting data: reading analog current and wind speed sensors, as well as e-stop button. Converts analog values to real-world equivalents
- Safety stop: activates full braking force regardless anything that's going on
- Evaluating data: compare real-world values of wind speed and current to competition restrictions
- Control of RPM and power: increment or decrement PWM values for braking servo corresponding to current values

Analysis and Simulation

To validate the duct/diffuser design choices, flow simulations were conducted on SolidWorks which were then improved upon with more accurate simulations composed by ANSYS CFD. Once reasonable data was collected from ANSYS, a 5:16 scale duct/diffuser was 3D printed and tested in the Chico State wind tunnel to compare simulated versus physical testing.

Utilizing SolidWorks flow simulations, it was concluded that airflow velocity is doubled near the center of the throat of the duct/diffuser. As seen in Figure 3, this increase in wind speed is depicted in the orange to red coloring but there does not appear to be clear turbulence which is questionable because the duct poses as such a large obstacle. Taking lack of turbulence into account, it was determined that a more accurate simulation was necessary; ANSYS CFD.

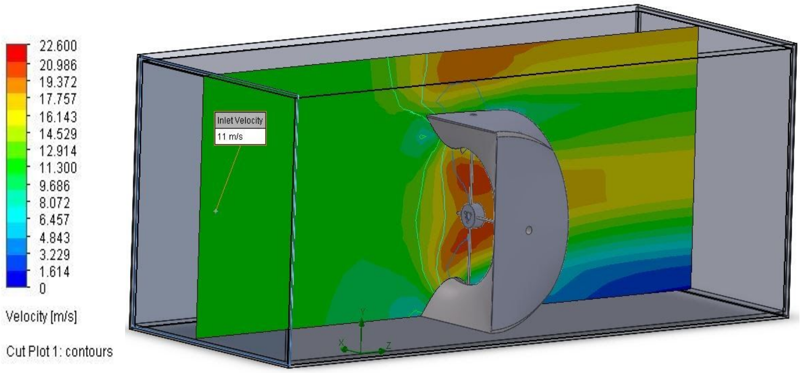


Figure 3: SolidWorks Flow Simulation of Duct/Diffuser at 11m/s

Applying ANSYS CFD analyses to the duct/diffuser at an airflow speed of 11m/s, clear visualization of turbulence is depicted as seen in the dark blue coloring in Figure 4, which was expected/desired. After repetitive iteration of the duct/diffuser design, it was concluded that wind speed through the throat of the component increases by an average factor of approximately 1.5 with the maximum increase being as much as 2.0 along the outermost edges of the throat. Findings from ANSYS are accurate considering factors such as the yaw effect, surface finish, and moving blades were not taken into account. One important discovery when simulating varying wind speeds is, there is always a concentration of higher wind speeds along the outermost edge of the throat which can be beneficial but also harmful to the turbine's performance. Higher wind speeds at the outermost edge of the throat and at the tips of the turbine blades can lead to a gain in power production and decrease in cut-in wind speeds, but can also cause abnormal and non uniform loading on the blades which complicates the design and iterative process of the blades as the airflow they experience cannot be treated as uniform. These concentrations of high wind speeds are depicted in red in Figure 3.

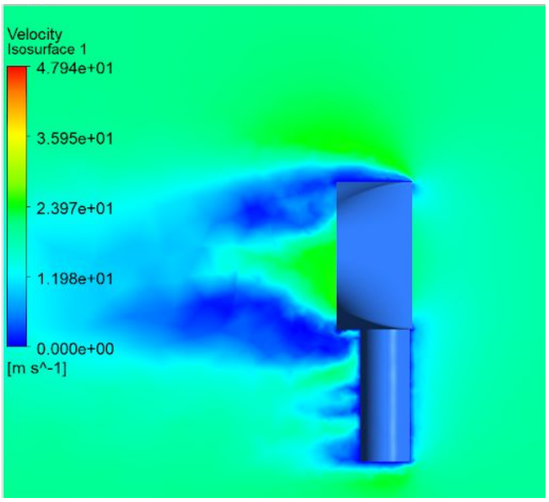


Figure 4: ANSYS CFD on Duct/Diffuser at 11m/s

To be confident in the utilization of ANSYS CFD as a primary simulator for design iteration, further experiments were necessary to conclude ANSYS's accuracy. The experiment conducted to prove ANSYS's accuracy was to manufacture a 5:16 scale prototype of the duct/diffuser component and to test it in the Chico State wind tunnel to compare drag coefficients. If drag coefficients have significant correlation, then it could be concluded that the force of the wind in reality and in the simulation acts similarly and it can be utilized for design iteration analyses.

The results of the physical test of the 5:16 scale duct/diffuser correlated with the ANSYS simulation of the same 5:16 scale model. The physical test yielded a drag coefficient of 1.28 where the simulated model yielded a drag coefficient of 1.45. Although these coefficients are not identical, many differences in set up were not taken into account such as the dimension imperfections in 3D printing, the drag of the tower of the physical test, surface finishes, and perfectly uniform airflow and airflow speeds. The physical 5:16 scale model is shown in Figure 5 and the surface roughness can be seen due to the layering of the 3D printed PLA. The ANSYS results of the 5:16 scale model are depicted in Figure 6 and the turbulence causing the drag forces are visualized in blue.

Figure 5: 5:16 Scale Model of Duct/Diffuser in Chico State Wind Tunnel (Cd=1.28)

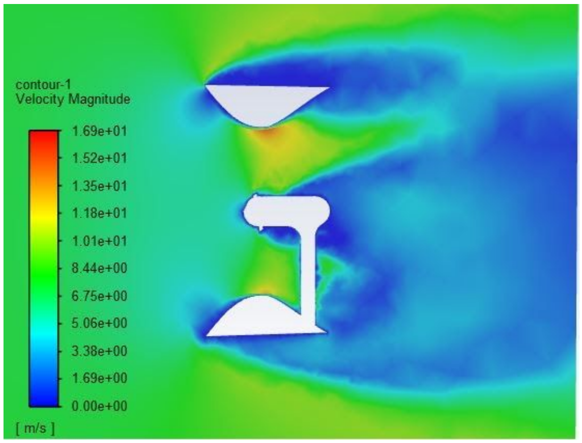


Figure 6: 5:16 Scale Model of Duct/Diffuser in ANSYS at 8m/s (Cd=1.45)

Once confident with the results from ANSYS simulations, additional tests were completed to obtain airflow speed through the throat of the duct/diffuser at varying wind tunnel velocities of 2.5, 8, 11, and 20 m/s. The following figures visually compare and contrast the final results of the full-scale duct/diffuser in the competition wind tunnel. Figure 7 shows a 3D view of where the contour cross sections are being taken, Figures 8 through 9 illustrate wind speeds of 2.5, 11, and 20 m/s with the same legend/color scheme, and Figures 10 through 11 depict magnified views of 2.5, 8, 11, and 20 m/s wind speeds for more precise readings.

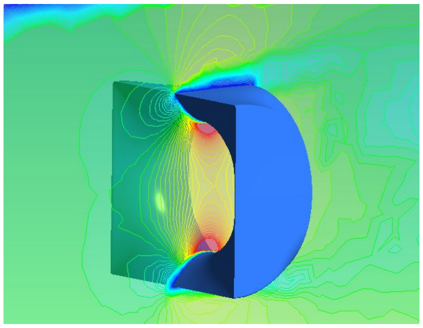


Figure 7: 3D View of Contour Cross Section Location

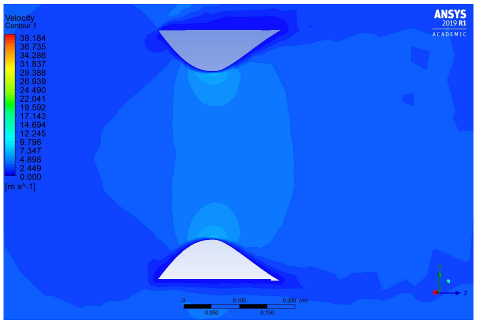


Figure 8: 2.5m/s Wind Speed 0-40m/s Legend

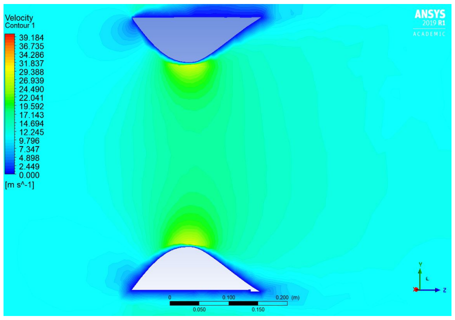


Figure 9: 11m/s Wind Speed with 0-40m/s

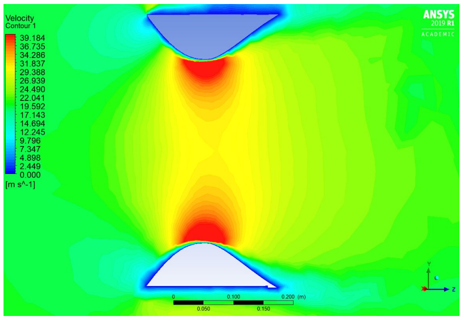
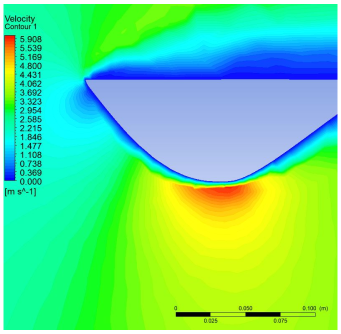
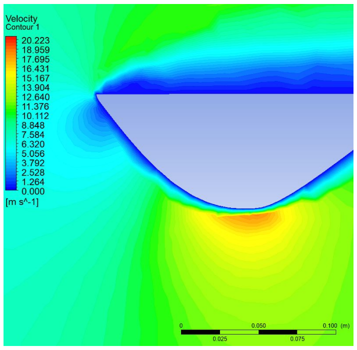


Figure 10: 20m/s Wind Speed with 0-40m/s





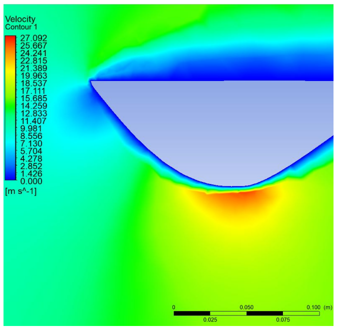


Figure 11: 2.5m/s Wind Speed

Figure 12: 8m/s Wind Speed

Figure 13: 11m/s Wind Speed

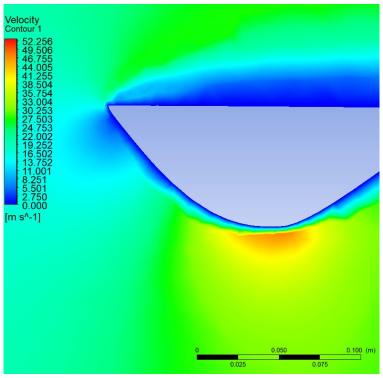
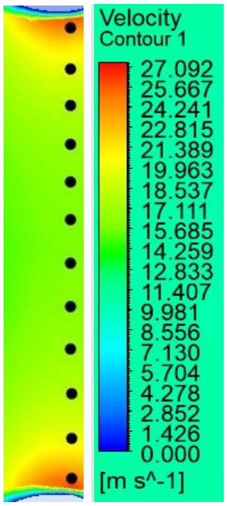
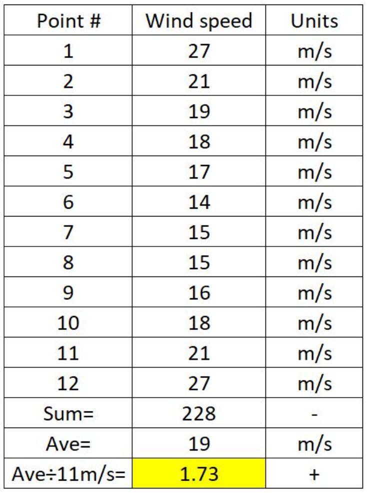


Figure 14: 20m/s Wind Speed

As predicted, wind speed increased through the throat. Taking twelve measurements through the rectangular cross sectional portion of the duct/diffuser that the blades were to be mounted in as depicted in Figure 14, wind speed increased an average factor of 1.73 (Table 2).





Point #	Wind speed	Units
1	27	m/s
2	21	m/s
3	19	m/s
4	18	m/s
5	17	m/s
6	14	m/s
7	15	m/s
8	15	m/s
9	16	m/s
10	18	m/s
11	21	m/s
12	27	m/s
Sum=	228	-
Ave=	19	m/s
Ave÷11m/s=	1.73	+

Figure 15 (left): Rectangular Blade Mounting Cross Section Area with Corresponding Legend
Table 2 (right): Wind Speed Factor of Increase Approximation

Testing

Durability Test

The turbine design would be tested at the maximum wind speed it will feel at competition which is 25 m/s. During this test the blades would be stopped and the turbine should not fracture in any way. The design never reached wind speeds over 22 m/s so this test was not able to be completed.

Yaw Test

In order to test the yaw system of the wind turbine it was placed within a wind tunnel where it would be in constant wind flow and the base could be turned to test the ability to self-correct. As expected the system was observed to work better in higher wind speeds, but still worked in wind speeds as low as about 9 m/s. However at low wind speeds, rather than re-orienting to center the turbine was observed to orient itself as high as 20 degrees off center. While this was not expected, the fact there was close to 3 inches of clearance toward any side of the wind tunnel suggested this behaviour was likely a result of the small wind tunnel rather than standard operational behavior.

Brake Test

The brakes were tested in three of the four requirements:

1. The brakes were able to be triggered by an e-stop and slow the blades down to less than 10 percent of the rotor RPM at 11 m/s wind speed.
2. When the turbine is disconnected from the load, the spring takes over successfully triggering the brakes and stopping the Rotors.
3. Brakes were able to be engaged and stop the Rotors by detecting high wind speeds.
4. The regulation of the rotor RPM feature did not make it into this build.

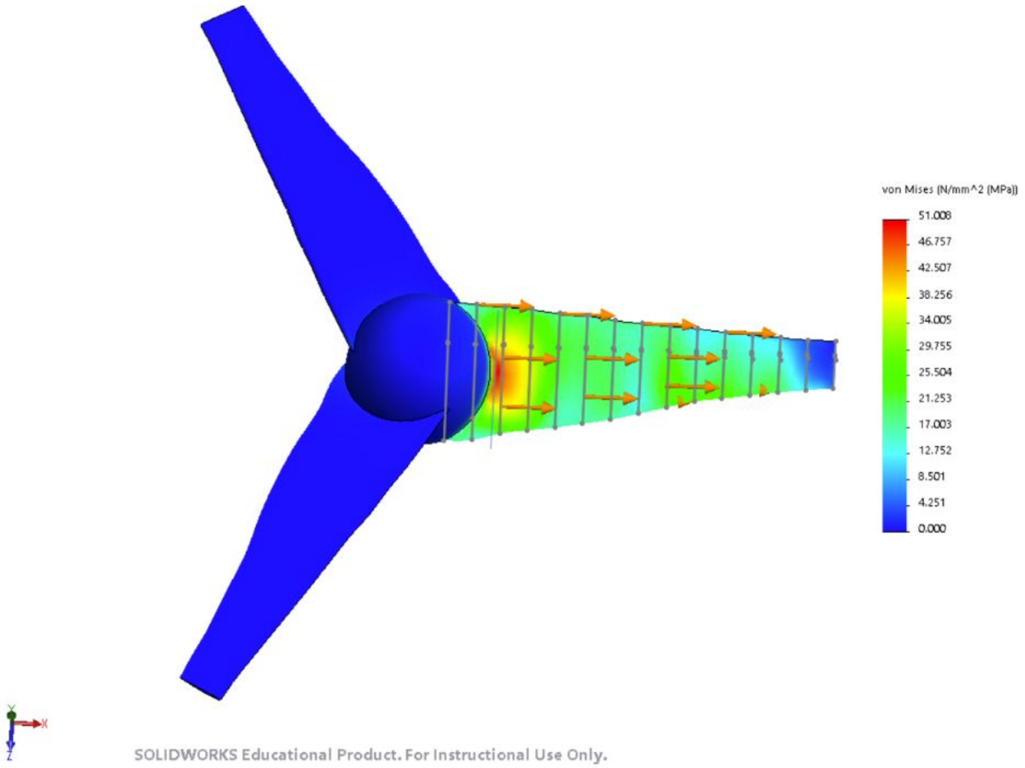


Figure 16: FEA of turbine blade Von Mises Stress

Rotor Strength Test 1

The rotors were tested to their breaking point by placing them in the CSU Chico wind tunnel on a free spinning mount with negligible friction and slowly ramping the wind speed until failure. The blades broke at an RPM of 9,000 with a wind speed of 22 m/s. The 3D printed PET-G plastic material was not enough to withstand this speed even with an epoxy coating. Runaway for this blade configuration was determined to be 19,700 RPM, so this does not pass the CWC requirements of safety by having blades that survive runaway conditions. A laser tachometer and oscilloscope were used to measure RPM during this test with an uncertainty of 50 RPM. However, a greater uncertainty is to be expected due to human recording of RPM at time of failure.

In order to test the runaway speed and design blades to ensure failure does not occur at competition maximum wind speed a rotor strength test was performed. The rotor blades on the wind turbine are one of the most crucial parts of this design, due to this they must be very sturdy and be able to handle strong winds beyond what will be seen at the future competition due to the duct funneling wind. This test is to ensure they do not see any unforeseen stresses in a motor "runaway" situation. The RPM was measured until failure of the rotor blades. Our blades failed at approximately 16m/s and 9,000rpm (see photos below). The failure was due to the stress in tension on the material. We are currently printing these blades out of PET-G and covering them with epoxy to harden them further. We plan to print future iterations of blades out of a Polycarbonate (PC) based Carbon Fiber (CF) filament and then covering it in epoxy. This PC-CF material is superior in stiffness as well as overall durability compared to PET-G. There will be testing of these blades tested up to 19,732rpm (calculated runaway rpm speed). If this fails, we will modify the design to hold a metal “rib” on the base of the blade where the fracture is taking place and/or increase the cross-sectional area of the base of the blade where the fracture is occurring.

Figure 17: Remaining blades and hub (left) and nose cone (right) after rotor strength test in wind tunnel.

Where,
W˙=Power (watts)
T = Torque (n-m)
ω=Angular velocity (rad/s)
ρ=Density (of fluid e.g.wind) (kg/m^3)
A = Cross-sectional Area of blade
U = Velocity of wind (m/s) = (.55 multiplier is due to the duct funneling wind to an overall 55% increase)
25m/s + 25m/s*.55 = 38.75 m/s
λ = TSR (Tip Speed Ratio
R = Radius of blades

$$\lambda = \frac{\omega * R}{U} =$$

$$8 = \frac{\omega * .15m}{38.75\frac{m}{s}}$$

$$\omega = 2066.667\frac{rad}{s} * \frac{1\,rot.}{2\pi\,rad} * \frac{60sec}{1\,min} = \mathbf{19,735\,rpm}$$

Appendix

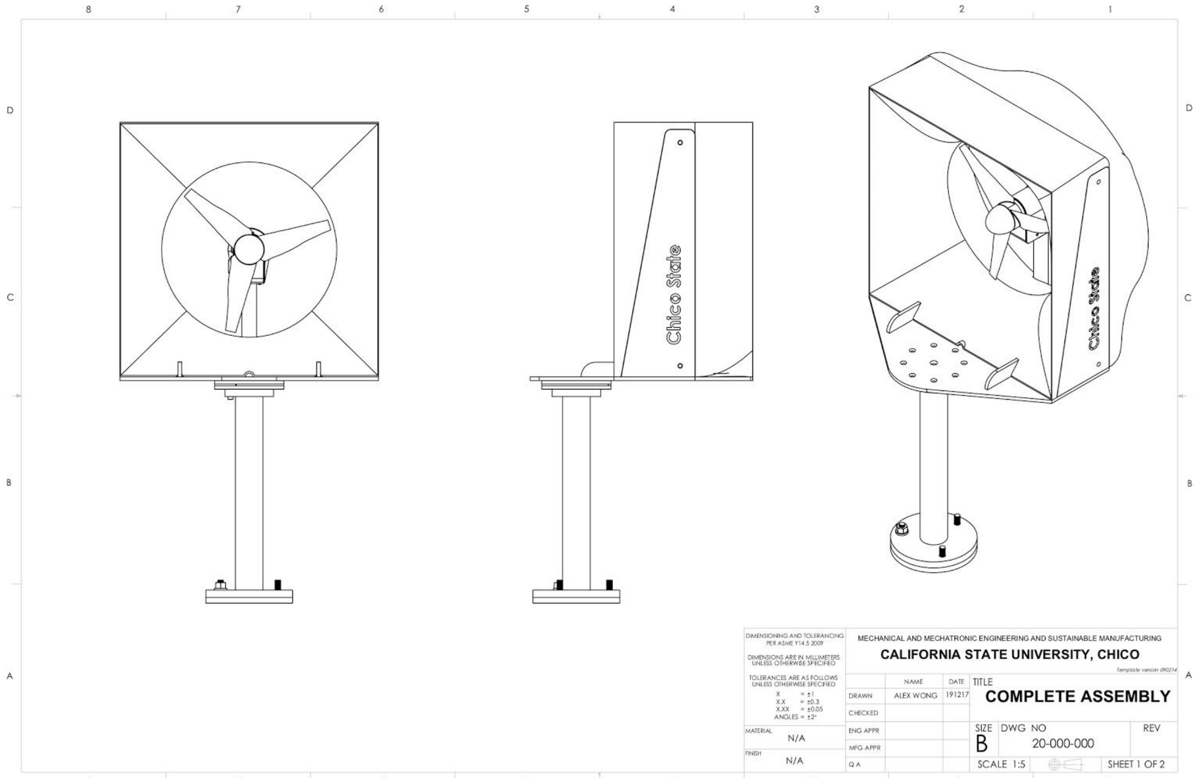
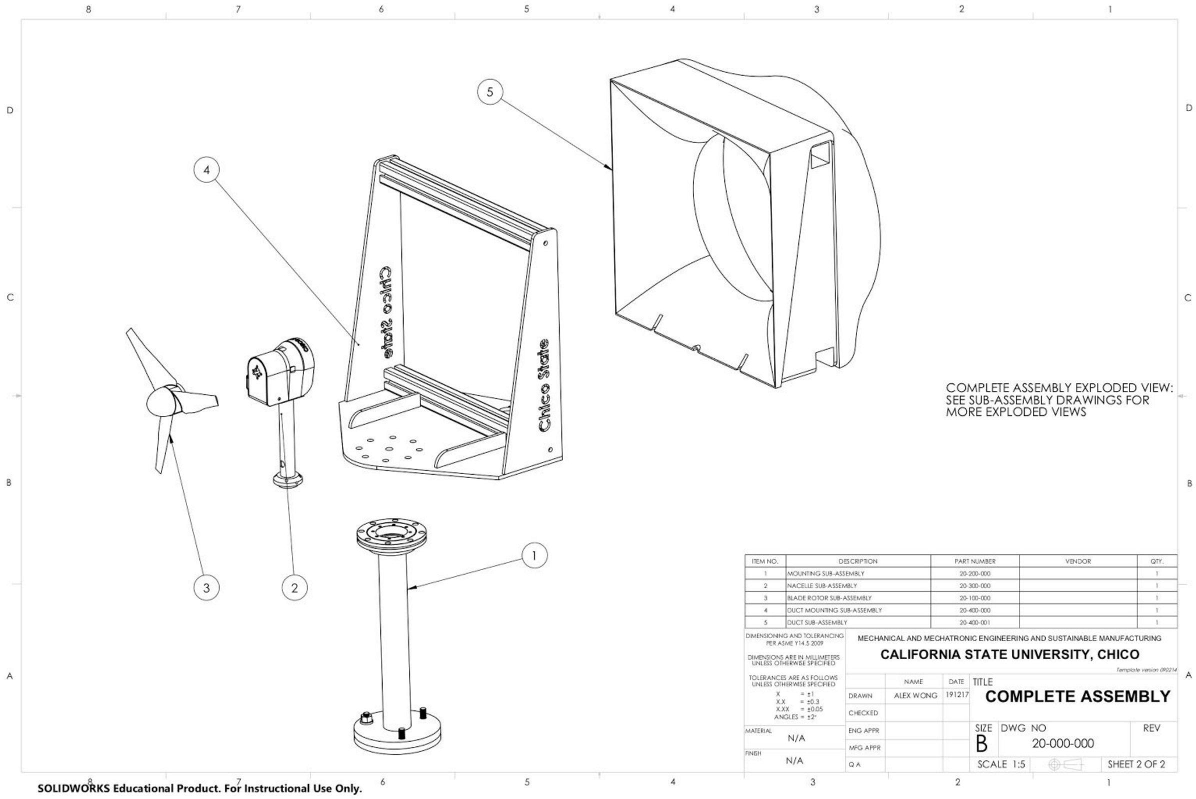


Figure 18: Engineering mechanical drawing of all components final assembly.



ITEM NO.	DESCRIPTION	PART NUMBER	VENDOR	QTY.
1	MOUNTING SUB-ASSEMBLY	20-200-000		1
2	NACELLE SUB-ASSEMBLY	20-300-000		1
3	BLADE ROTOR SUB-ASSEMBLY	20-100-000		1
4	DUCT MOUNTING SUB-ASSEMBLY	20-400-000		1
5	DUCT SUB-ASSEMBLY	20-400-001		1

Figure 19: Engineering mechanical drawing of all components in exploded assembly.

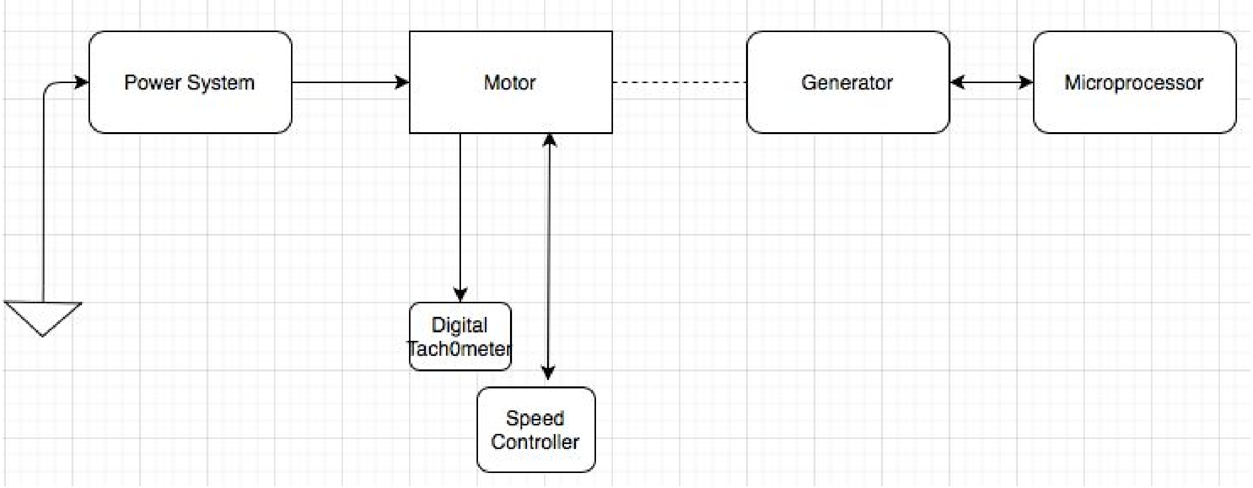


Figure 20: Electrical one-line diagram for turbine.