

# **Increasing Power of Horizontal Axis Wind Turbine by Addition of a Shroud and Diffuser: Preliminary Results**

**Brett Floyd, Tavon Johnson**

Department of Mechanical Engineering  
West Virginia University Institute of Technology  
Montgomery, WV 25136  
Undergraduate Mechanical Engineering Students  
Email: bafloyd@mix.wvu.edu, tajohnson@mix.wvu.edu

**Farshid Zabihian, Yogendra Panta**

Department of Mechanical Engineering  
West Virginia University Institute of Technology  
Montgomery, WV 25136  
Email: farshid.zabihian@mail.wvu.edu, ympanta@mix.wvu.edu

## **ABSTRACT**

This project has been initiated to evaluate a potential method to increase the power output of a Horizontal Axis Wind Turbine (HAWT) by addition of a shroud and diffuser. The shroud, located at the inlet of the turbine, will be used to increase the inlet velocity to the turbine blades. The diffuser, located at the rear of the turbine, will be used to decrease the pressure through the nozzle. Through the usage of the COMSOL software, the team was able to confirm the fluid flow calculations through the shroud and diffuser. It was found that by adding a shroud, the wind velocity was increased by 55.4% at the location where the face of the turbine blades will be positioned. Behind the turbine blades where the flanged diffuser is positioned, it was found that the pressure was reduced to a vacuum pressure of -3.41 Pa leading to less back pressure effects on the turbine. To further validate the results found in the Computational Fluid Dynamics simulations, experimental analysis will be performed in coming weeks. The purpose of the experiment will be to verify the overall effect, increase in the power output, which the shroud and diffuser has on the bare small scale turbine that will be used. The testing model that will be used provides power output readings in digital form. This will allow measurements to be recorded accurately and compared to the bare turbine and with the components attached.

## **INTRODUCTION**

Alternative energy can deliver power to customers with minimal impact on the environment. Wind energy in particular has been the fastest growing source of electricity generation in the world since the 1990s, according to the U.S. Department of Energy [1]. Wind energy is inexhaustible and considered to be more clean or “green” than traditional energy methods such as oil or coal.

As the world moves closer to adopting renewable energy, innovative methods are required to ensure that we are receiving the maximum power possible from these renewable sources. Wind is playing a critical role in the country’s future, as a sustainable energy source. Currently wind turbines are used for a smaller demand of power output. If we are able to increase the power output, wind turbines will become more appealing for their sustainability.

## **THEORY**

Wind energy is becoming a popular renewable energy recourse around the world today. This source of energy is found to be one of the most environmentally friendly recourse available here on earth.

*Proceedings of the 2016 ASEE North Central Section Conference  
Copyright © 2016, American Society for Engineering Education*

Horizontal axis wind turbines have been used to generate power from the kinetic energy available in wind [2]. The total kinetic energy in the wind is determined by a given fluid mass moving at velocity “V” which is shown in Equation 1:

$$KE (J) = \frac{1}{2} mV^2 \quad (1)$$

Where:

M (kg) –Mass of Air

V (m/s) –Velocity of Air

For calculating power in the wind, the kinetic energy equation can be rearranged knowing Equation 2 derivation of mass[3]:

$$m = \rho v = \rho AL \quad (2)$$

Where:

$\rho$ (kg/m<sup>3</sup>) –Air Density

A (m<sup>2</sup>) – Area occupied by air

L (m) –Length

V (m<sup>3</sup>) –Volume

Taking the derivative of mass with respect to time yields Eq. 3:

$$\frac{dm}{dt} = \rho A \frac{L}{dt} = \rho AV \quad (3)$$

Notice that length over time is equal to velocity. Next, it is defined that the rate of energy over time equals power. This gives Eq. 4:

$$P = \frac{KE}{dt} = \frac{1}{2} \frac{dm}{dt} V^2 \quad (4)$$

Substituting in the mass time derivative gives equation 5, which indicates the power that is coming in to the turbine:

$$P_{in} (Watt) = \frac{1}{2} \rho AV^3 \quad (5)$$

Where:

$\rho$  (kg/m<sup>3</sup>) –Air Density

A (m<sup>2</sup>) – Swept Area of Turbine

V (m/s) –Wind Speed

Equation 6 determines the actual power that can be outputted from the winds energy by using the coefficient of power:

$$P_{out} (Watt) = P_{in} C_p \quad (6)$$

Where:

C<sub>p</sub>—Rotor Power Coefficient

The rotor power coefficient shown in the wind power equation above is known to have a maximum value equal to the Betz limit. According to the Betz limit there is a limit to the maximum convertible power that can be obtained from a wind turbine [4]. The maximum achievable efficiency of a bare wind turbine, i.e. a turbine without an added shroud and diffuser, is given by the Betz number  $\beta = 16/27 = 59.2\%$  that was discovered by Albert Betz in 1919. An efficiency of 59.2% was found only with the assumptions that there is no loss due to friction or turbulence and also from the use of an infinite number of turbine blades. After considering the various engineering requirements of a wind turbine, it is found that on average, HAWT's operate with a real world power coefficient between 3-45% [2]. This coefficient is then reduced even further after considering mechanical losses from bearings and gearboxes.

After studying the wind power equation, it can easily be seen that wind velocity has a large effect on power output from a horizontal axis wind turbine. One of the other parameters that has an effect on output power is the wind turbine's swept area. However, for the scope of this project, the swept area will be kept constant.

The effects of wind velocity on the output power of a HAWT has been a large focus point so far in this project. It can be seen from the wind power equation that if wind velocity could be doubled, the power would become eight times greater than it was originally. This is where the idea of adding a shroud to a HAWT came to mind. The shroud can be described in general as a nozzle similar to the one found at the end of a common household water hose. It has a conical shape where the inlet diameter is greater than the outlet diameter and is shown in Figure 1.



**Figure 1.**Conical Shroud

The effect that a shroud has on incoming air is that once the air enters the shroud at a velocity of  $V_{in}$ , it will then exit at a greater velocity of  $V_{out}$ . This can be proven through Eq.7, the Continuity equation:

$$(\rho * A * V)_{out} = (\rho * A * V)_{in} \quad (7)$$

Where:

- $\rho$  ( $\text{kg/m}^3$ ) –Air Density
- $V$  ( $\text{m/s}^2$ ) –Velocity of the fluid at the inlet and outlet
- $A$  ( $\text{m}^2$ ) –Circular area at the inlet and outlet

And further solving for the output velocity:

$$V_{out} = \frac{A_{in}}{A_{out}} V_{in} \quad (8)$$

Wind velocity also plays a large part in the rotational speed of a HAWT. The relationship between the wind speed and the rate of rotation of the blade, which is attached to the rotor shaft, is characterized by the Tip Speed Ratio (TSR). Eq. 9 further solves for the turbine's tip speed ratio:

$$\lambda = \frac{\text{Speed of Rotor tip}}{\text{Wind Speed}} = \frac{v_r}{V} = \frac{\omega * R}{V} \quad (9)$$

Where:

- V (m/s) –Wind Speed
- $\omega$ (rev/s) –Angular Velocity
- $v_r$ (m/s) –Velocity of Rotor tip
- R (m) –Rotor Radius

Once the TSR is calculated, it can then be used in the standard torque gain equation which is:

$$K = \frac{1}{2} \rho A R^3 \frac{C_p}{\lambda^3} \quad (10)$$

Where:

- $\rho$  (kg/m<sup>3</sup>) –Air Density
- A (m<sup>2</sup>) – Swept Area of Turbine
- R (m) –Turbine Rotor Radius
- $C_p$  –Rotor Power Coefficient
- $\lambda$  –Tip Speed Ratio (TSR)

From equation 9 we can see that the TSR will become smaller as wind velocity increases; therefore, using Eq. 10 the Torque gain (K) will then increase leading to a larger torque produced by the wind on the turbine blade. This relationship is shown in Eq. 11, the torque equation:

$$T = K \omega^2 \quad (11)$$

Where:

- K –Gain
- $\omega$  (rev/s) –Angular Velocity

Using the rotational speed value found by multiplying the velocity of the blade tip by the radius of the turbine, the horsepower generated from the turbine can be found in Watts using Eq. 12:

$$HP = \frac{T \times \omega}{5252} \times 745.7 \quad (12)$$

Where:

- T –Torque
- $\omega$  (rev/s) –Angular Velocity

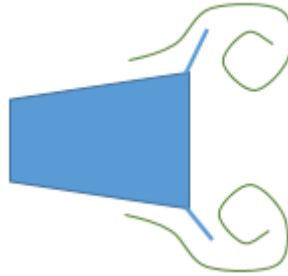
Along with the addition of the shroud, the group has also designed a diffuser that will be added at the rear of the turbine. As seen in Figure 2, the diffuser has a similar conical shape; however, it is positioned so that the smaller diameter is directly behind the turbine blades.



**Figure 2.** Conical Diffuser

The continuity equation applies to a conical diffuser, it is shown that the outlet velocity of the diffuser will decrease as the exit area increases. Also, the pressure found behind the turbine blades will be reduced leading to less of a power loss. It has been found that reducing back pressure behind the exit of a DAWT (Diffuser Augmented Wind Turbine) can have a profitable effect on the performance [5].

The addition of a flange to the outer perimeter of the diffuser which is shown in Figure 3, has been studied throughout this project and has given good results.



**Figure 3.** Wake region caused by the flanged diffuser.

As previous research shows, flanges have been proven to increase power output of a diffuser augmented wind turbine. Ohya et al. [6] found that by adding a flange onto the outside of a diffuser, wind speed through the inside of the diffuser was increased by 1.6-2.4 times that of the incoming wind speed. The reason for this is that the flange generates a low pressure region in the exit area of the diffuser by vortex formation and draws the outside wind into the diffuser. From the successful results found by Ohya et al. the group has tested this design through a CFD analysis in COMSOL with positive results and is shown in the Diffuser section of this report.

## EQUIPMENT

In order to better confirm the results found from numerical calculations and software modeling, an experiment must be designed and performed. The purpose of the experiment will be to verify the effects that the designed shroud and diffuser will have on a small scale HAWT shown in Figure 4.

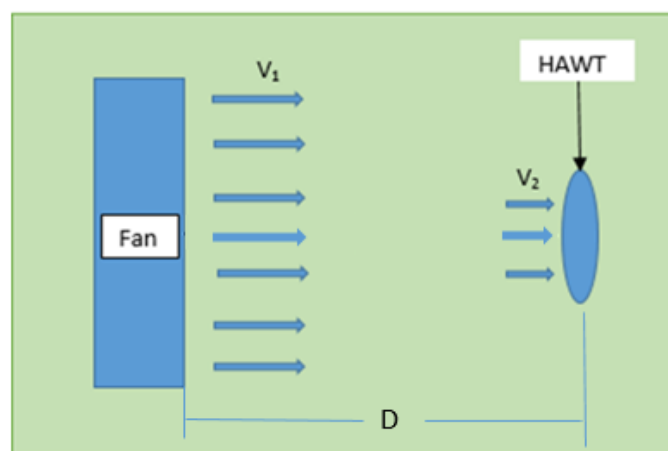


**Figure 4.** Small Scale HAWT

The small scale model provides power output readings in a digital form which can be recorded and used to compare the bare turbine to the turbine with the attached shroud and diffuser.

For this experiment to take place, there are a few pieces of equipment that will be needed. The first would be a pitot tube, which is a device that is used to measure wind velocity. The next piece of equipment that will be use is a Ventamatic High Velocity floor fan which will provide a maximum wind speed of 3.66 m/s and also a constant wind speed directed toward the small scale HAWT. Also required for this experiment is a measuring tape and ruler for measuring distances.

Using the previously mentioned devices along with the small scale HAWT kit, the team will follow a set of four steps in order to record the needed data for establishing worthy data that can be used to compare the effects of the added shroud and diffuser. For the procedure, please refer to Figure 5 shown below.



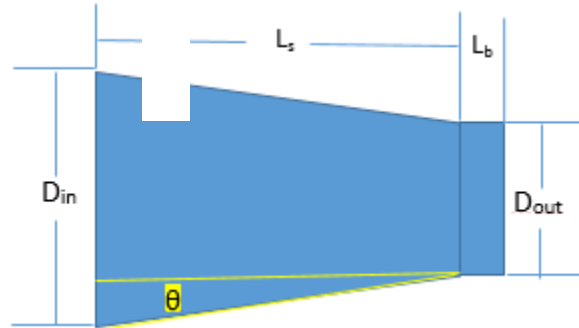
**Figure 5.** Experiment Layout

The first step in the experiment would be to measure the velocity  $V_2$  using the cup anemometer located at a distance “D” from the high velocity fan. Next, the bare small scale HAWT will be placed at the measured distance and the power output will be recorded. The third step would be to keep the turbine in the same location however for this trial, the shroud and diffuser will be added and the output power will be recorded. The final step consists of repeating steps one through three at various distances from the fan which will give data that will show the turbine performance at various wind velocities.

### **SHROUD DESIGN**

Based on theory outlined in the previous section, it can be identified in the power equation that one of the best ways to increase power effectively is to increase velocity. When determining the best way to increase the input velocity to the turbine blade, the fundamental principle of the continuity equation was considered strongly. Principally the continuity equation represents the engineering fundamental that what goes in a system must come out. Using this principle we understood that because the component will be in the air, the working fluid is the same so the density is approximated to be the same. According to the continuity equation the product of the velocity and area at the inlet must be equivalent to that of the outlet. To achieve this it was determined that to increase the fluid velocity, the area at the inlet must be larger than the outlet to increase the flow rate along a streamline.

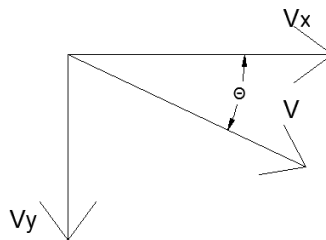
After understanding the principle of the continuity equation, measurements of the small-scale turbine planned for testing was done. Utilizing the length of the turbine blade ( $\approx 0.16\text{m}$ ), an input diameter parameter was set to be  $0.46\text{m}$ . A model of the shroud was drafted as a visual representation of the parameters:



**Figure 6.** Shroud Model with parameters

Utilizing the continuity equation, a study of the effects the shroud has on the wind velocity was conducted. The first phase of the study was computational utilizing an excel spreadsheet. In the instance of calculating velocity, the inlet velocity cannot be controlled as it is dependent on the atmospheric conditions. The main factor that will contribute to the change in velocity is the area ratio between the inlet and outlet. To determine the outlet velocity, reference Equation 8. Equation 8 shows that as the inlet area increases, the outlet velocity will also increase. Given the outlet area is as minimal as possible due to the swept area produced by the HAWT and the inlet velocity is not adjustable, the parameter that is adjustable is the inlet area. In order to keep the shroud outlet area as small as possible and maintain the highest incoming velocity at the blades tip, a clearance of  $\approx 0.001\text{m}$  was created—resulting in an outlet diameter of approximately  $0.37\text{m}$ .

Additionally, to ensure that the shroud is complimenting the wind with velocity in the x-direction as much as possible, a final parameter of the angle between velocity and normal vectors was considered. Utilizing basic trigonometric knowledge, decreasing the angle between the vectors  $V_x$  and  $V$  the value of the x component will be increased.



**Figure 7.** Wind velocity vectors

The parameter that was considered to be adjusted to an optimal value is the length of the shroud. When evaluating the wind velocity ( $V$ ), the angle  $\theta$  must be reduced to less than  $40$  degrees to maximize the velocity in the direction of the turbine blades ( $V_x$ ). Using the equation below, it was found that the highest velocity in the x direction was  $\approx 3.03 \text{ m/s}$  at an angle of  $35.75^\circ$ . The angle of attack was found

using Eq. 13:

$$\theta = \tan^{-1} \left( \frac{\frac{D_{in} - D_{out}}{2}}{L_s} \right) \quad (13)$$

When calculating values for the shroud, the inlet velocity was considered constant at a value of 2 m/s. This parameter was considered constant as our primary objective is to understand the effects the shroud dimensions have on the incoming wind as it exits the nozzle. Reviewing the results from, it can be observed that as the input diameter of the shroud is increased there will be an increase of wind velocity at the outlet. The wind velocity was determined to be maximum at a shroud angle of 35.75° calculating to be ≈3.11 m/s at the outlet. This output velocity delivers a theoretical 55.4% increase over the velocity provided at the inlet. Utilizing the power equation, we can observe that a 55% increase in wind velocity can lead to a significant increase in power output. The Reynolds number was calculated for each shroud and in each case, the wind flow through the shroud was found to be turbulent (Re>4,000).

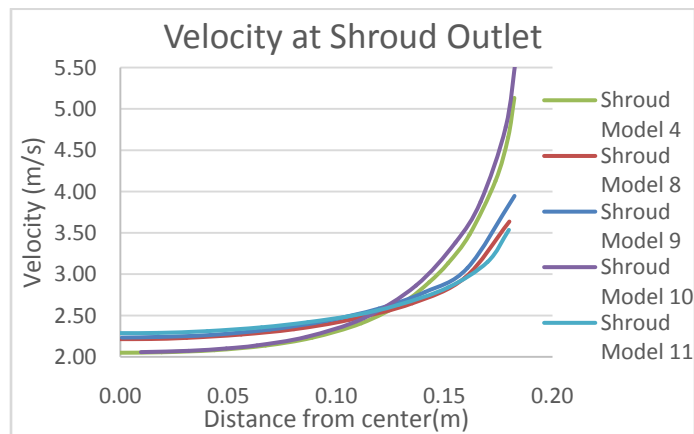
Sample Calculation:

$$Re = \frac{v * D}{\nu}$$

$$Re = \frac{10.21 \text{ (ft/s)} * 14.6 \text{ in} \left( \frac{1 \text{ ft}}{12 \text{ in}} \right)}{1.635 \times 10^{-4} \text{ (ft}^2\text{/s)}}$$

$$Re = 75,980.19$$

Several shroud models were studied through COMSOL to determine the effects of dimensional parameters of the shroud inlet diameter ( $D_{in}$ ), and also the length of the shroud ( $L_s$ ). While there were multiple shroud models that were studied, five models will be compared in this report. With an objective of increasing the wind velocity through the shroud, Figure 8 shows the velocity values of the wind as it passes through the length of the shroud.



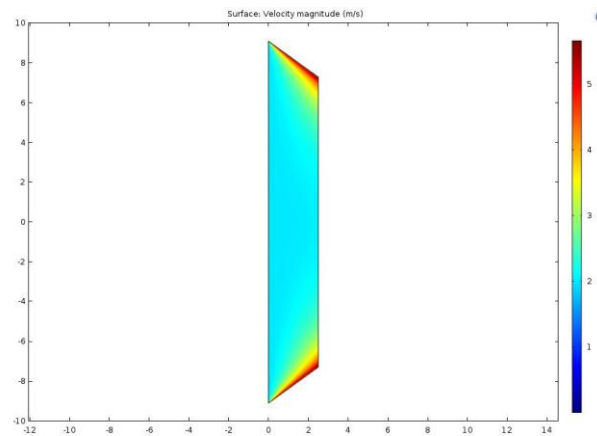
**Figure 8.** Plot of velocity values along the shroud outlet diameter.



The x-axis values shown in Figure 8 correspond to a point distance along the radius at the outlet of the shroud, with zero being the center of the shroud. These values can be assumed to be equal for the other half of the shroud since the configuration is symmetric about the horizontal axis.

In Figure 8, five shroud models are being compared. The five shrouds share the same output diameter of 0.37m and shroud length of 0.064m. It can be seen in Figure 8 that Model 10 had the largest output velocity at the furthest distance from the center, which corresponds to the location of the tip of the turbine blades.

Utilizing the computational knowledge from the equations drafted above, simulations were developed utilizing the COMSOL-CFD module. Several shroud models were drafted in the software. Model number 4 will be highlighted in this report as it was the choice with the most optimal output. Figure 9 shows the CFD simulation of the model.



**Figure 9.** Velocity magnitude of Shroud design

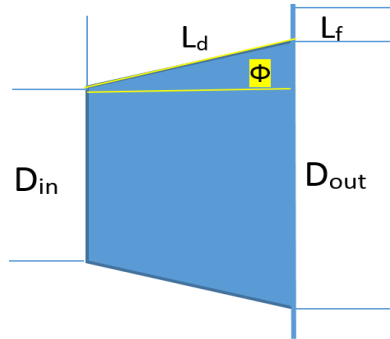
This shroud was created with a 0.064m shroud length and an input velocity of 2 m/s, which is hypothesized to be the velocity of the wind tunnel for experimental analysis next semester. Observing Figure 9 we can identify the velocity of the wind at the surface of the shroud, point closest to the tip of the turbine blade, is of highest value. It was determined that the value at the tip of the turbine blade would be approximately 3 m/s. This result yields a 3.6% difference between calculations and simulations. We can also confirm that, in areas of interest, the velocity through the shroud has increased by over 50%. This velocity increase is significant as we recall from the theory section the relationship between Gain and Power output.

## **DIFFUSER DESIGN**

The study of the diffuser was initiated to reduce the negative effect of back pressure that takes place behind the wind turbine blades. When decreasing the pressure through the diffuser, the velocity will be increased as the reduction in pressure acts as a vacuum. Keeping this objective in mind, the diffuser was first designed utilizing hand calculations and later confirmed through CFD analysis in the COMSOL software.

The geometry of the diffuser was created utilizing information gathered which related the length of the diffuser to its diameter. It is recommended that the length of diffuser,  $L_d$ , be short relative to the

Diameter, i.e.,  $L_d < 0.4D$  [7]. With this ratio an angle  $\phi$  can be determined. Figure 10 shows the relationship between  $\phi$  the length of the diffuser, and the inner and outer diameters.



**Figure 10.** Schematic of Diffuser

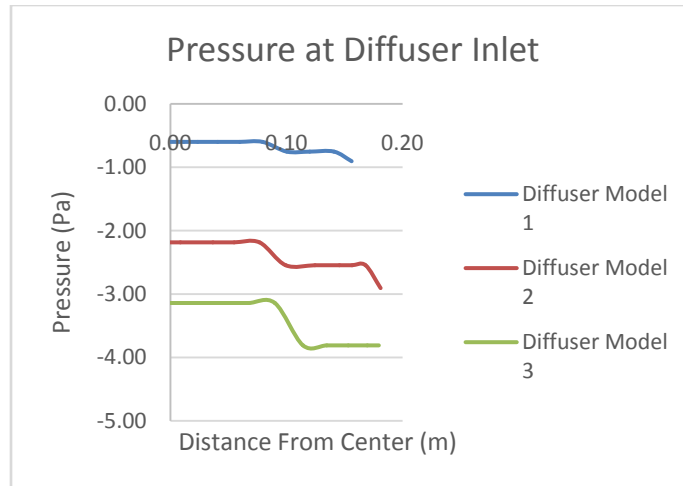
Examining the optimal angle of the diffuser ( $\Phi$ ), it is found that on average a  $4^\circ$  angle is most effective [7]. It was found from the CFD analysis done in COMSOL that an angle of  $2^\circ$  gave the best result when the flange was added. The dimensions of the diffuser that gave an angle of  $2^\circ$  was an inlet diameter of 0.37m, output diameter of 0.381m, and a total length of 0.1524m. Hand calculations can be shown below for the pressure at the inlet of the diffuser where the inlet velocity is 3.11 m/s:

Sample Calculations:

$$V_{out} = \frac{A_{in}}{A_{out}} V_{in} = \frac{(0.37(m))^2}{(0.381(m))^2} \times 3.11 \left(\frac{m}{s}\right) = 2.933 \frac{m}{s}$$

$$\begin{aligned} P_{in} &= \frac{1}{2} \rho (V_{out}^2 - V_{in}^2) \\ &= \frac{1}{2} \left( 1.199 \left(\frac{kg}{m^3}\right) \right) \left( \left(2.933 \frac{m}{s}\right)^2 - \left(3.11 \frac{m}{s}\right)^2 \right) \\ &= -0.641 Pa \end{aligned}$$

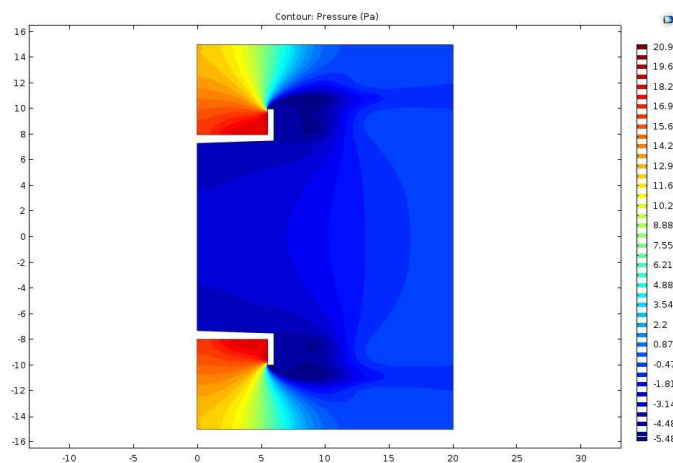
Several diffuser models were studied through COMSOL to determine the effects of various dimensional parameters of the diffuser such as the length ( $L_d$ ), outlet diameter ( $D_{out}$ ), and also the length of the added flange ( $L_f$ ). As there were multiple diffuser models that were studied, three models will be compared in this report. With an objective of decreasing the pressure behind the turbine blades, three diffusers were compared in Figure 11 to show the pressure values at the inlet of the diffuser i.e. the location behind the turbine blades.



**Figure 11.** Plot of pressure values at the inlet of the diffuser.

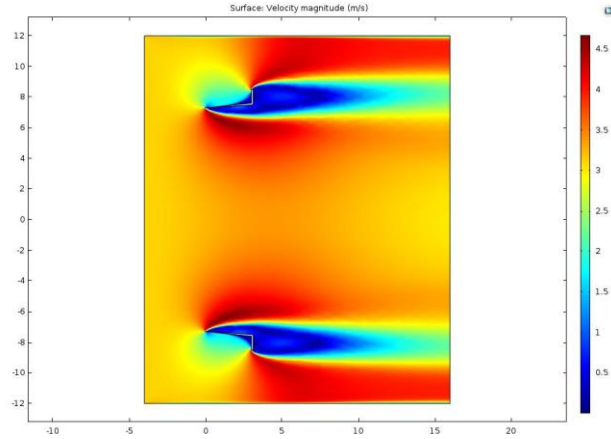
The length values shown in Figure 11 correspond to the location along the radius at the inlet of the diffuser, with zero being the center. These values can be assumed for the other half of the diffuser since the diffuser is symmetric about the horizontal axis.

In Figure 11, three diffuser models are being compared. The three diffusers share the same input diameter of 0.37m, output diameter of 0.381m, as well as the total length of 0.1524m. The difference is that Diffuser Model 1 was not equipped with a flange, Diffuser Model 2 had a 0.0254m flange, and Diffuser Model 3 had a 0.0508m flange. It can be seen that Model 3 which had the larger flange, shown in Figure 12, gave the most negative pressure at the inlet of the diffuser due to the wake region produced by the flange.



**Figure 12.** CFD Pressure analysis through Diffuser 3

Figure 12 shows how strong vortices produced by adding a flange to the outlet of the diffuser draws an increase in wind speed. By the vortices creating a vacuum effect, the wind will be drawn through the diffuser at a higher velocity; therefore, the pressure through the diffuser will be decreased. This diffuser model was created in COMSOL using dimensions used in hand calculations which are related to the small scale HAWT that will be used for experimental analysis. Figure 13 shows the velocity effect through the diffuser model equipped with a flange and was simulated in the COMSOL software.

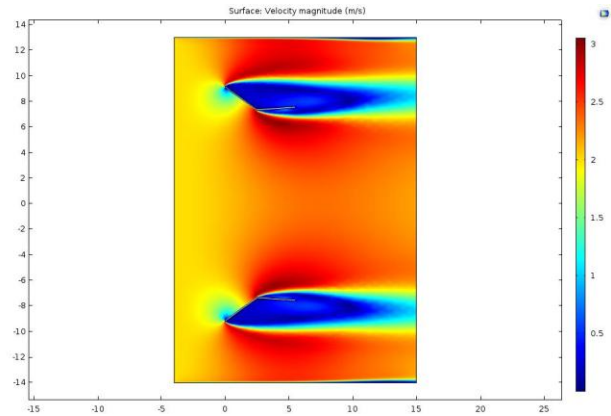


**Figure 13.** Velocity magnitude of Diffuser 3 Design

From Figure 12, we can observe the effect a flange, and inclination of the diffuser has on the pressure surrounding the component. Observing the diagram legend we can see that the wake region produces a negative pressure. This negative pressure produces a vacuum like effect, which explains the significant increase of velocity in Figure 13 through the diffuser.

### **OBSERVATIONS**

Based on the designs that were discussed in the Shroud and Diffusion discussion of the report, a combined model of the two components was constructed. Figure 14 shows the velocity magnitude of the flow passing through the shroud and diffuser components.



**Figure 14.** Velocity magnitude of Shroud and Diffuser

The results from Figure 14 shows a significant increase in velocity between the two components. In the vicinity of the intersection of the two components will be the turbine blades, the placement of the turbine blades will be strategic. The input velocity for the simulation was 2 m/s, the area where the tip of the turbine blades are located measures a velocity reaching between 2.8-3 m/s. This increase in velocity can be attributed to the shroud usage of the conservation of mass principle and the vortices created from the inclination and flange on the diffuser.

## CONCLUSION

By adding the shroud and diffuser, the team was able to increase the input wind velocity from 2 m/s to an average of 2.91 m/s at the exit of the shroud with an angle of 35.75°. Without considering losses, the output power was compared in Table 1.

**Table 1.** Comparison of output power.

	$V_{wind}$ (m/s)	Swept Area (m <sup>2</sup> )	$P_{out}$ (Watt)
Bare Turbine	2	0.107	0.51
Added Shroud and Diffuser	2.91	0.107	1.58

As shown in Table 2, the power has increased from 0.51W to 1.58W by adding the designed shroud and diffuser. In order to verify the large increase in power output, the team will carry out the experiment as future work.

## REFERENCES

- [1] Energy Department. "U.S. Wind Energy Production and Manufacturing Reaches Record Highs". Energy.gov. Web. August 6, 2013.
- [2] The Royal Academy of Engineering. "Wind Turbine Power Calculations" raeng.org.uk/publications. Pp. 1-5.
- [3] Professor Mel Tyree. Department of Renewable Resources, University of Alberta, Edmonton Canada. "Derivation of WindPower Equation: Playing with Newtonian Physics". PDF. 5/24/2008.
- [4] Mahmoud Huleihil and GedalyaMazor. "Wind Turbine Power: The Betz Limit and Beyond". Dx.doi.org/10.5772/52580. Pp. 1-28.
- [5]Dr. Gerard J.W. van Bussel. Journal of Physics: Conference Series 75 (2007) 012010. "The Science of Making Torque from Wind: Diffuser experiments and theory revisited". IOP Publishing. 2007. Pp. 1-13.
- [6] Yuji Ohya, Takashi Karasudani, and Xing Zhang. "A Shrouded Wind Turbine Generating High Output Power with Wind-Lens technology.
- [7] Yuji Ohya, Takashi Karasudani, Akira Sakurai, Ken-ichi Abe, Masahiro Inoue. "Development of a Shrouded Wind Turbine With a Flanged Diffuser". Journal of Wind Engineering and Industrial Aerodynamics. 1996-2008. Pp. 1-16.