Flow field analysis and testing of curved shroud wind turbine with different flange angle

Beshir Heyru¹* and Wondwossen Bogale²

Abstract: This research focuses on computational fluid dynamics modeling and experimental analysis of a shrouded wind turbine. A shroud turbine comprises a wind turbine with a flanged shroud encircling it for better wind energy utilization at low wind speed. The effects of diffuser shape and flange angle have been studied by performing flow field analysis in Ansys Fluent for better utilization and condensing of the shroud size. The CFD analysis shows that a cycloid diffuser shape gives 7% more velocity than a straight shape. In addition, the optimum flange angle for a cycloid diffuser is +10°, while compared with the normal flange angle, the optimal flange angle has increased velocity and power by 4.83% and 15%, respectively. This increment is due to the variation in the magnitude of vortex generation behind the flange. Besides, the performance of the proposed 3D, shrouded, and unshrouded wind turbines was numerically analyzed, and a prototype model for the proposed wind turbines was manufactured and tested. The simulation and experimental results agree and lay within the acceptable range. Compared with an unshrouded wind turbine with a similar swept area and wind speed, the shrouded wind turbine increased velocity and power by 1.58 and 5, respectively. This increase in speed and power makes the shrouded turbine suitable for application in low wind speed areas. Moreover, it can be helpful in sparsely populated off-grid areas to generate electricity by being mounted on rooftops.

Subjects: Mechanical Engineering; Fluid Mechanics; Renewable Energy

Keywords: Shroud; CFD; augmentation; axisymmetric; flanged shroud

ABOUT THE AUTHORS

Beshir Heyru, an Associate Researcher at the Ethiopia Space Science Institute of Technology, contributes to developing and controlling various satellite systems and aerospace infrastructures for space research and applications. He also has more than 2+ years of teaching experience. Wondwossen Bogale is an Associate professor at the School of Mechanical and Industrial Engineering at Addis Ababa University. He has more than 13+ years of teaching, research, and consultation experience in Wind Energy, Biogas, Biochar production, waste combustion, gasification, preparation of charcoal using agricultural waste, wind energy assessment, and various innovative research activities in energy sectors.
1. Introduction

The growing interest in small-scale wind turbine applications in low wind speed regions has become an essential issue for rural electrification in developing countries (Agha et al., 2018). According to the African Development Bank report, Ethiopian installed wind power is 324 MW, with a potential of 10,000 MW, implying the country harnesses only 3.24% of the potential capacity. Moreover, compared to the overall energy demand, the scale of wind power usage is still very insufficient (Leghari & Shaikh, 2017). The large-scale conventional horizontal axis wind turbines are efficient, and modern megawatt wind turbines have a power coefficient of up to 40—45% (Ozgener & Ozgener, 2007). However, a large-scale wind turbine needs high wind speed to function. Compared with large-scale wind turbines, small-scale wind turbines can work at low wind speed, generate minimal noise, and have no known safety risks. Despite many advantages, small wind turbine models have been developed and installed worldwide (Hiranara et al., 2005; Leung et al., 2010). Ohyá et al. (Göltenbott et al., 2017). Showed that the velocity augmentation strongly depends on the duct shape and design parameters in the experimental analysis of the different ducts. Hansen et al. (Hansen et al., 2000). Showed that the power coefficient of an airfoil-shaped diffuser around a wind turbine is directly proportional to the mass of wind flow at the diffuser inlet, without considering the diffuser efficiency. Abe and Ohyá (Toshimitsu et al., 2005). Shows that the thrust coefficient of a flanged diffuser augmented wind turbine is smaller than a bare wind turbine. Jafari and Kosasih (Jafari & Kosasih, 2014a). Showed that in the numerical analysis of the frustum shrouding diffuser, the velocity augmentation strongly depends on the geometry of the diffuser. Heikal, Elyazeed, and Nawar et al. (Heikal et al., 2018). Studied the effect of flange angle and the diffuser flange depth for the wind lens turbine system; this turbine system has a square cross-section shape at the location of turbine installation. The numerical result shows that the +10° flange angle gives 28% power extraction from other flange angles. Francesco, Daniele, and Damiano (Avallone et al., 2020). Numerically investigate the effect of blade length and tip clearance ratio for diffuser augmented wind turbine. The result shows that a longer blade reduces the rotor thrust coefficient and the tip clearance ratio significantly affects the far-field noise. Ali et al. (Ali & Kim, 2021). Performed analytical and computational methods for airborne wind turbine installation at high heights. K.Foreman (Foreman, 1981). Refers to a process of evaluating the technical and economic viability of a diffuser-assisted wind turbine. The results show that small rated output of wind energy could lead to an effective and competitive cost of electricity generation. Luptz and Shumer (Lubitz & Shomer, 2014). Examine the practicability of the shrouded turbine compared to the unshrouded horizontal axis wind turbine and the necessity of weighing the benefits of a diffuser against the additional costs. R. Bontempo et al. (Bontempo et al., 2021; Bontempo & Manna, 2020a, 2020b). Conducted the performance of ducted wind turbines based on the actuator-disk model and a free-wake ring-vortex actuator-disk model and also established essential theoretical models for design and performance analysis for wind-concentrator systems. M.Rivarolo (Rivarolo et al., 2020). Experimentally investigates that the ducted horizontal axis wind turbine increased the power by 2.5 compared to the bare turbine. V. Dighe et al. (Dighe et al., 2019). Determine ducted wind turbines as an energy harvesting device employed in urban environments where non-uniform intakes can degrade aerodynamic and acoustic performance. Bichitkar et al. (Siva Sankara Raju et al., 2020). Using the Ansys CFD tool, investigate the flange angle variation, found a + 15 deg to be the best flange angle for accelerating higher wind speed at the diffuser intake. Vaz, Okulov, and Wood (Vaz & Wood, 2016). Bring forth alternative finite blade functions that maintain the finite limit on the axial velocity to maximize the power output of a diffuser-augmented wind turbine. O. Igra et al. (Liu & Yoshida, 2015). Showed that the airfoil-shaped diffuser increases the power augmentation by 2.4 factors than a bare wind turbine. Kosasih and Hudin (Kosasih & Saleh Hudin, 2016). Showed that the power coefficient of the shrouded wind turbine was still more significant than that of the unshrouded wind turbine; even when there was a lot of turbulence, diffuser augmentation was still possible. Lubitz and Shomer (Lubitz & Shomer, 2014). Investigate the practicality of the shrouded turbine relative to the unshrouded horizontal axis wind turbine and analyze the advantage of incorporating the diffuser with the additional cost. Sørensen and Hansen (Hansen et al., 2000). Showed that the increasing wind flow through
the rotor of a diffuser is directly proportional to the force acting inside the diffuser. Gilbert and Foreman (Tukaram). Introduce 1D and semi-empirical theory applied to the shrouded turbine and concentrate on concentrating and accelerating wind energy in a diffuser with diffuser inclination and controlling flow separation by using holes along the diffuser to understand the flow pattern inside a diffuser's wall. Bet and Grassmann (Bet & Grassmann, 2001) investigated the inclusion of a wing ring structure near the diffuser outflow. Based on the study, the shrouded turbine enhances the power by a factor of 2 compared to an unshrouded turbine using an airfoil type diffuser. Fletcher et al. (Nikolić et al., 2015). Based on extended blade element momentum theory analyzes the effect of wake spin, blade Reynolds number, diffuser efficiency and area ratio for shrouded wind turbines. However, diffuser axial push and far-wake velocity were not accounted. Anzai et al (Amer et al., 2012; Harrington, 2006). Showed that diffuser velocity speed-up ratio, torque and output power of the unit shows the same behavior which gained for experimental outcomes existing from the literatures. J. Vaz and D. Wood (Vaz & Wood, 2018). Proposed a novel performance study of a shrouded wind turbine that takes the influence of diffuser efficiency and thrust force. Ranjbar and B. Rafiei (Ranjbar et al., 2021). Shows that incorporating an optimal duct around a vertical axis wind turbine can increase power output as does a horizontal axis wind turbine, but it is more efficient in the location of consistent wind speeds. The introduction of shrouded turbines, on the other hand, is critical for maximizing the use of wind energy at low wind speeds, however, by adding a shroud around the turbine and concentrating wind energy through a diffuser, which increasing the wind stresses on the turbine. Therefore, based on the literature review, investigating the design parameters of the shroud is very significant to condense the size of the shroud further. Furthermore, the effect of cycloid or curved sectional shape with varied flange angles on the performance of shrouded wind turbines has never been explored. Therefore, in this research work, the effect of the diffuser shape and its flange angle will be studied in detail to increase the use of wind energy at low wind speed areas.

2. Methodology

2.1. Design parameters of the shroud

Numerical analysis is used to determine the optimum shape of the shroud for a small horizontal axis wind turbine, considering the various design factors shown in Figure 1, and Table 1 shows the dimensions selected for this analysis.

The flow field within the domain is governed by a fluid flow equation combined with a nonlinear PDE. The exact solution of the governing equation is challenging due to the non-linearity of the partial differential governing equation of the flow field. However, it can be solved numerically using numerical software. Therefore, the numerical analysis and the governing equations are discussed in the following sub-sections.

2.2. Computational analyses

2.2.1. Governing equations

The computational domain governing equation is viscous flow, which describes three fundamental conservation principles. Equations 1, 2 and 3 are the Navier Stokes equations of mass, momentum, and energy. However, the fluid utilized is incompressible, and the temperature does not change significantly. Therefore, equations 4 and 5 are modified to account for incompressible fluids, ignoring the energy equation.

\begin{equation}
\frac{\partial \rho}{\partial t} + \rho \nabla V = 0
\end{equation}
\[
\rho \frac{\partial \mathbf{V}}{\partial t} = \rho \mathbf{g} + \nabla \cdot \tau - \nabla P = 0
\]

(2)

\[
\rho \frac{\partial \mathbf{V}}{\partial t} = \rho \mathbf{g} + \nabla \left(k \nabla^2 T \right) + \theta
\]

(3)

\[ \nabla \cdot \mathbf{V} = 0 \]

(4)

\[
\rho \frac{\partial \mathbf{V}}{\partial t} = \rho \mathbf{g} + \mu \left(\nabla^2 \mathbf{V}\right) - \nabla P
\]

(5)

2.2.2. Computational condition and boundary condition
Assume the flow is 2D axisymmetric, uniform, and incompressible, where similar methods have in (Abe & Ohya, 2004; Hu & Wang, 2015; Shives & Crawford, 2010). Figures 2 and 3 depicted the current computational conditions and the grid system surrounding a flanged shroud.
### Table 1. The dimension of each model

| Shape of shroud         | Throat diameter $D_T [m]$ | Rotor diameter $D_r [m]$ | Arearatio unitless | Length ratio unitless | Brim height $h_f [m]$ | Angel of the brim with a horizontal line | Inlet shroud length $L_{shroud[m]}$ |
|-------------------------|--------------------------|--------------------------|--------------------|-----------------------|-----------------------|------------------------------------------|----------------------------------|
| Curved                  | 0.52                     | 0.5                      | 2.35               | 1.03                  | 0.5 $D_T$             | 90°                                      | 0.8                              |
| Straight                | 0.52                     | 0.5                      | 2.35               | 1.03                  | 0.5 $D_T$             | 90°                                      | 0.8                              |
| Curved with Flange angle| 0.52                     | 0.5                      | 2.35               | 1.03                  | 0.5 $D_T$             | -35° to +30°                             | 0.8                              |
2.2.3. CFD fluent set-up

The flow field expresses by the continuity and incompressible Reynolds-averaged Navier–Stokes equations, and the turbulent model is a transition SST Turbulence model based on the recommended flow Reynolds number. In the analysis of turbulent flow, the $y^+$ values are in the range of 0.00187 to 0.3797, which is the acceptable range for the transition SST Turbulence model ($y^+ < 1$; Shukla et al., 2015). The first-order numerical scheme computes the convection and diffusion terms (Owis et al., 2015). The computational domain boundary conditions are as follows: steady flow, no-slip, zero-pressure, axisymmetric, and slip boundary conditions set at the inlet, wall, outlet, top, and bottom edge, respectively; similar methods have in (Harrington, 2006). In addition, a uniform flow with 5% free-stream turbulence was specified (Hu & Wang, 2015). Iterative CFD investigations use the Ansys Fluent CFD package, a helpful tool for simulating fluid flow in complex geometries.

2.2.4. Computational domain for the whole assembled shrouded wind turbine

As shown in Figure 4, the complete CAD drawing revolving and stationary fields are modeled independently and assembled in the Fluent.

The non-revolving area has a shroud and a cylinder to interface the two domains. In contrast, the rotating domain has a rotor turbine and a rotating cylinder. Figure 5 shows the resulting mesh for stationary and rotating domains loaded into CFD Fluent.

2.2.5. Physical set-up with Ansys pre-processing

The flow field expresses by the continuity and incompressible Reynolds-averaged Navier–Stokes equations. The Transition SST turbulence model analyzes turbulent flow (Sorribes-Palmer et al.,
In the analysis of turbulent flow, the $y^+$ values are in the range of 0.0127 to 0.4567, which is the acceptable range for the SST Turbulence model ($y^+ < 1$; Shukla et al., 2015). The first-order numerical scheme computes the convection and diffusion terms (André & Maia, 2014). As illustrated in Figure 6, the velocity inlet and pressure outlet boundaries are applied to the inbound and outbound faces of the outer cylinder, respectively. The curved surface of the outer cylinder has a symmetry boundary condition imposed on it. The rotor faces have been assigned a wall with no-slip boundary conditions. The calculated average rated wind velocity at the inlet is six m/s, the gauge pressure is zero in the pressure outlet, and the turbulence intensity is 5% (Shukla et al., 2015). The interior cylinder faces linking the stationary and rotating parts. The rotor blades and the hub are moving walls attached to the moving frame, and a no-slip boundary condition sets at the walls.

The rotating zone has been set to the inside cylinder and assigned the calculated angular velocity of 63.33 rad/sec for a 6:1 tip speed ratio, and similar methods have been in (Jafari & Kosasih, 2014b). An
unsteady flow defines by a time step where the properties of the flow vary with time. By definition, the total flow time for a wind turbine is the ratio of $2\pi$ to the rotational speed of the rotor (Shukla et al., 2015). The study was conducted with a 1/20 incremental time step, corresponding to 0.005 sec, to capture 20-time points per rotation. The stationary and spinning domains have different interface types. Fluid-Fluid and Defining expressions are excellent techniques to streamline a CFD scenario (Jafari & Kosash, 2014b). Table 2 illustrate the defined expression terms.

### 2.3. Experimental set-up

In the wind tunnel, the authors also tested the performance of a prototype shroud turbine for implementation purposes. Figure 7 shows the test rig.

| Name                        | Expressions | Description               |
|-----------------------------|-------------|---------------------------|
| Rotor radius ($r$)          | 0.9m        | Turbine radius            |
| Tip speed ratio ($\lambda$) | 6           | The ratio of tip velocity to free velocity |
| Free steam velocity ($V_0$) | 6m/s        | Stream velocity           |
| Omega ($\omega$)            | $\frac{\omega V_0}{r}$ [rad/sec] | Angular velocity |
| Torque at rotor             | Torque$_x$@rotor | The entire rotor is selected |
| Total time                  | $\frac{2\pi}{\omega}$ [sec]  | Total simulation time     |

Figure 7. Multimeter connected in parallel with load to measure DC voltage set-up (left) and its electrical circuit (right).
3. Result and discussion

3.1. Grid independence test and model verification

Six different meshes are used to find a constant solution even as the mesh size increases. Used a finite number of elements to split the domain by applying edge sizing, inflating procedures, and global and local meshing to achieve discretization. Each mesh has the same physical set-up and boundary conditions, and the orthogonal quality of the mesh was tested and found to be within an acceptable range [0.15–1]. The simulation result was obtained by constructing the SST turbulence model and assessing the velocity speed-up ratio to optimize the shrouded shape, flange angles, and the torque convergence used for assembled shrouded wind turbine. Figure 8 shows that around 75,329 elements are required for mesh independence because the velocity augmentation variation after the indicated mesh size is 0.028. Thus, the significant enhancement is not predictable from the further refinement of the mesh.

Ohya et al. (Amer et al., 2012) used the experimental results to validate the model. The diffuser geometry for Ohya experimental set-up is: intake diameter \( D_{\text{in}} = 12\text{cm} \), outside diameter \( D_{\text{out}} = 24\text{cm} \), area ratio 4, angle of inclination 3\(^\circ\), and length ratio 7:7. The numerical analysis employs the same diffuser duct geometry as the author’s experiment. Figure 9 depicts the axial velocity distribution near the diffuser centerline in both circumstances. Therefore, the difference between the two curves ranges from 0.435 percent to 9.36 percent. Consequently, it shows that the current numerical approach by CFD fluent and Ohya experimental results are in good agreement.

Figure 10 and Table 3 clearly show the grid information for the computed torque for an optimal shroud wind turbine. As shown in Table 3, a finer mesh of around 9.5 million will solve with higher accuracy than others, but it will take a longer computation time. Nevertheless, about 7.15 million elements were required for mesh independence. A solution was considered grid-independent for mesh sizes greater than M4 since the torque output difference after M4 mesh size is so minimal that no substantial improvement is predicted from further mesh refinement.

![Figure 8. Stream wise velocity distribution for different mesh elements.](image-url)
3.2. Pressure and velocity distribution in the domain of the shroud for both models

As demonstrated in Figure 11, numerical modeling provides more detailed information on the flow inside and outside a flanged shroud. Such information is essential in analyzing the effect of design parameters on shroud performance.

Figures 12–13 clearly show the acceleration process of approaching flow. When air flows inside and outside a flanged shroud, the wind velocity increases at the diffuser entrance; this is due to the creation of low pressure at the diffuser wake, which acts as a vacuum, sucking in and speeding up the wind. In addition, the flange of the shroud creates a vortex at the back of the flange, which
causes low pressure in the wake of the diffuser; this causes intense pressure and draws additional wind into the diffuser.

3.2.1. Velocity augmentation for both case
Figure 13 shows that the curved sectional shape of the shroud causes a more significant zone of lower pressure inside the diffuser, which corresponds to faster velocity and improved performance. The curved sectional shape of the flanged cover increases the speed at the diffuser inlet from 5 to

| Table 3. Mesh independency test for the optimized shroud turbine |
|---------------------------------|-----|-----|-----|-----|-----|-----|
| Mesh type | M1  | M2  | M3  | M4  | M5  | M6  |
| Total number of elements | 4,147,262 | 5,436,250 | 6,432,435 | 7,132,232 | 8,530,546 | 9,430,562 |
| Torque calculated (Nm) | −8.93 | −9.61 | −9.81 | −9.98 | −10.01 | −10.04 |
| Percentage of Torque error (%) | 7.614 | 2.081 | 1.732 | 0.300 | 0.2997 | 0.256 |

Figure 11. Velocity distributions in straight sectional shape (left) and curved sectional shape (right) shroud.

Figure 12. Pressure distributions in straight sectional shape (left) and curved sectional shape (right) shroud.
8.34 m/s, as shown in Figure 13, but only to 7.79 m/s in the straight section. As a result, a curved shroud augments velocity by 7% more than a straight shroud. The primary reason for this is that the boundary-layer flow along the inside wall of the curved diffuser does not exhibit significant flow separation, as illustrated in Figure 14.

Figure 14 illustrates that the starting point of flow separation in a curved shroud is 40 cm from the inlet, where the wall shear stress dissipates but in straight, it starts to separate at 22 cm: this demonstrates that the flow separation points within a diffuser significantly impact airflow acceleration through the diffuser. As a result, increasing the velocity and reducing the size of a flanged shroud is critical in developing small and medium-scale horizontal axis wind turbines using flanged diffusers.

3.2.2. Velocity augmentation for different flange angle
The effect of flange angles for a diffuser curved sectional shape was investigated as a variable parameter on velocity increment at the diffuser entrance. The only difference between the samples is the flange angles ranging from—35° to 30°. All samples have the same sectional shape of the flanged shroud and identical dimensions of shroud length, entrance diameter, exit diameter, and flange length.

The numerical result in Figures 15 and 16 demonstrates that the +10° optimum flange angle has accelerated the wind speed at the diffuser entrance compared to other flange angles. The fundamental reason for these increments is the formation of stout vortices behind a flange, as shown in
Figure 17. As a result induces low pressure in the diffuser’s wake, sucking in more air at the diffuser’s entrance. As illustrated in Figure 16, the +10° flange angle of the curved sectional shape of the flanged shroud increases the velocity at the diffuser inlet from 5 to 8.68 m/s, but only to 8.28 m/s at the normal flange angle. As a result, the estimated generated power from a shrouded wind turbine with a +10° flange angle for a curved diffuser can be up to 15% more than a regular (0°) flange angle; this indicates that the nature of the flow separation points inside of a shroud significantly affects the acceleration of airflow through the flanged cover.

3.3. Performance analysis of the optimized shroud turbine and bare wind turbine

3.3.1. Torque variation along the blade
Figure 18 depicts the numerical torque variation generated by the shrouded and unshrouded wind turbines. The torque curve shape is similar to previous research performed by various authors (Guo
et al., 2021). In addition, it depicts the early stages of the simulation generated using a time step of 0.005 sec for optimizing the shroud wind turbine and a time step of 0.00785 sec for the unshrouded wind turbine.

Due to pressure loss and viscous effect, the torque generated by the wind turbine as a function of time tends to a steady-state value of $-10\text{Nm}$ for shrouded turbines and $-3\text{Nm}$ for the bare wind turbines, with a very tiny fluctuation for the same swept area and approaching wind speed. The power extracted by the turbine may be estimated using the calculated torque $T$ and rotor speed $\omega$, which is $P_{\text{ext}} = T\omega = 633\text{W}$ for shroud wind turbine and $P_{\text{ext}} = 120\text{W}$ for the bare wind turbine. The shrouded wind turbine increases the velocity and power by 1.58 and 5, respectively, compared to an unshrouded wind turbine with a similar swept area and wind speed. The power coefficient for a shroud wind turbine is the fraction of the developed power to the available capacity (Li, 2014). The available power for shroud wind turbine $P_{av} = \frac{1}{2} \rho A (\gamma V_0)^3 = 1557.83\text{W}$. Where $V_0 = \frac{\omega_0}{\omega}$, $\gamma = 1.58$, $D_r = 1.8m$ and $\rho = 1.204\text{kg/m}^3$, $A = |R_{out} + hf|$, but in the case of a bare wind turbine, the swept area is only the rotor of the blade $|A = R_o|$ and there is no velocity augmentation $|\gamma = 0|$ due to the absence of the shroud, therefore the available power for a bare wind turbine is $P_{av} = 336.49\text{W}$. In addition the power coefficient of a shroud wind turbine is $C_p = \frac{P_{av}}{\rho \omega^3} = 0.4065$, and for a bare wind turbine $C_p = 0.3925$ with the same rotor area and approaching wind speed. The main reason for the difference is the reduction of tip loss in shrouded wind turbines and velocity augmentation due to the presence of a diffuser.
3.3.2. Velocity augmentation in the domain
As illustrated in Figure 19, the average rotor plane velocity increases from 6 to 9.5 ms⁻¹, indicating the velocity augmentation achieved by the optimal shrouded turbine. It also shows the velocity distribution along the blade for shrouded and bare wind turbines as air passes through the turbine and its domain at $\lambda = 6$.

3.3.3. Velocity and pressure distribution in the domain
Figure 21 shows that the pressure behind the unshrouded turbine is recovered quickly to the atmospheric value. In addition, the pressure in the shrouded turbine is much lower than atmospheric pressure, and it is not uniform along with the domain for both cases. The pressure variation and velocity distribution happened with the flow due to the blade obstructing the flow, shown in Figures 20 and 21.

As shown in Figure 21, the pressure distribution in the shroud wind turbine at the front face appears higher than in bare wind turbines. Because of the considerable drop in pressure, the blade spins clockwise.

3.3.4. Velocity vectors across the blade for shrouded and bare wind turbine
As illustrated in Figure 22, the blade velocity distribution for both turbines, due to velocity augmentations, the blade velocity at the tip is higher at the root for shrouded turbines than bare wind turbines. For validation, the blade velocity calculated by hand is 57 ms⁻¹, whereas the CFD analysis finds 56.56 ms⁻¹, as indicated in Figure 22. As a result, the error is 0.77 percent, which can overlook without affecting the solution’s accuracy.
3.4. Experimental result and discussion

Table 4. Shows the output DC voltage and current measurements from the multimeter reading.

Table 4 shows that while there is no load, the voltage developed by the turbine is 12.1 V, and the current is virtually zero amperage. In contrast, when there is a load, the voltage produced by the turbine is 4.98 V, and the generated current in the circuit is 0.784 A. Hence the electrical power output of the turbine is 3.90432 W, and dc motor efficiency is $\eta_d = 0.8$. Therefore, the mechanical power value is 4.8804 W. However, because of the tower, shroud, and nacelle effect, the wake’s momentum deficit and turbulent kinetic energy increase (Santoni et al., 2017). In addition, there is a voltage fluctuation in the power testing due to velocity variation downstream, which directly impacts the stability and life of the micropower generation. Therefore the nacelle, shroud, and tower wake interact with the turbine blade, causing the tip vortex to break apart, affecting the wind turbine power coefficient, the flow angle, and velocity deficit at the wake (Guo et al., 2021). So it is essential to analyze the optimal shape of
| Measured Wind speed [m / s] | Loading condition | Measured DC [A] | measured DC voltage [V] | Calculated Electrical power $E_p = IV$ | Efficiency of dynamo $\eta_d$ | Mechanical power $m_p = \frac{E_p}{\eta_d}$ |
|-----------------------------|-------------------|-----------------|--------------------------|----------------------------------------|-----------------------------|----------------------------------|
| 5                           | No load           | 0.004           | 12.1                     | 0.0484                                 | 0.8                         | 0.0605                           |
| 5                           | 10 ohm            | 0.784           | 4.98                     | 3.90432                                | 0.8                         | 4.8004                           |
| Wind speed [m/s] | radius [cm] | CFD Max velocity [m/s] | Available Power [watt] | Experimental Electrical Power [watt] | CFD MP [watt] | Experimental Mechanical Power [watt] | CFD Power coefficient | Experimental power coefficient | Relative error on power coefficient |
|-----------------|-------------|-------------------------|------------------------|--------------------------------------|--------------|--------------------------------------|-----------------------|-------------------------------|--------------------------------|
| 5               | 12.5        | 8.3                     | 184549                 | 3.90432                              | 5.83832      | 4.6804                               | 0.32                  | 0.32                          | 0.32                           |
the shroud, reduce the size of the nacelle and tower, and analyze its aerodynamic shape to minimize the power fluctuation.

### 3.5. Comparison of the CFD and experimental results

Table 5 shows the experimental and numerical power coefficients for the optimum shroud wind turbine with a 15.6 percent deviation. The main variation is that the prototype does not account for various losses such as the quality of manufacturing, experimental set-up size, and human and instrumental error; this will increase the result variation between CFD and experimental. However, based on experimental results, it is evident that the turbine power output is close enough to be validated.

### 4. Conclusions

Properly investigating the shroud size and shape is critical to maximizing wind energy usage at low wind speed areas. First, this research explains how to examine the effect of the shroud flange for various sectional shapes using Ansys fluent, which is particularly valuable for shroud size reduction. Then, using EBEMT, develop a Matlab code to calculate the blade's chord and twist angle distributions in the presence of a shroud. Next, the performance of the optimized shroud turbine was analyzed, which helps install shrouded wind turbines in low wind speed regions to ensure the accuracy of the CFD simulation. The following conclusion is reached based on the previous numerical and experimental analysis and discussion.

(i) A curved diffuser increased velocity by more than 7% compared to a straight diffuser. Therefore, the velocity augmentation and reduction of the size of a flanged shroud are significant in developing a small-scale horizontal axis wind turbine.

(ii) The optimized flange angle, of the curved sectional shape of the diffuser, for maximum generated power is +10°. Due to a strong vertex formation behind the flange, a low-pressure area is created, enhancing the mass flow rate inside the shroud. As a result, a 4.83% increase in velocity is observed compared to a right-angled flange. Consequently, the estimated generated power from a +10° flange angle of a curved diffuser can be up to 15% more than a regular (0°) flange angle: this indicates that the nature of the flow separation points inside a shroud significantly affects the acceleration of airflow through the flanged cover.

(iii) Power augmentation of the optimized shrouded wind turbine is a five-time increase than a bare wind turbine (wind turbine shroud), with velocity augmentation of 58% achieved with the same swept area and wind speed. Therefore, velocity and power augmentation are critical for the total usage of wind energy extraction at low wind speeds.

---

**Funding**
The authors received no direct funding for this research. This research was supported by Addis Ababa

**Author details**
Beshir Heyru1
E-mail: beshirheyru946@gmail.com

1 Mechanical Engineering, Ethiopia Space Science and Technology Institute, Addis Ababa, Ethiopia.

2 School of Mechanical and Industrial Engineering, Addis Ababa Institute of Technology, Addis Ababa, Ethiopia.

**Disclosure statement**
No potential conflict of interest was reported by the author(s).

**Citation information**
Cite this article as: Flow field analysis and testing of curved shroud wind turbine with different flange angle, Beshir Heyru & Wandwassen Bogale, Cogent Engineering (2022), 9: 2095951.

---

**References**
Abe, K. I., & Ohya, Y. (2004). An investigation of flow fields around flanged diffusers using CFD. Journal of Wind Engineering and Industrial Aerodynamics, 92(3–4), 315–330. https://doi.org/10.1016/j.jweia.2003.12.003

Agha, A., Chaudhry, H. N., & Wang, F. (2018). Diffuser augmented wind turbine (DAWT) technologies: A review. International Journal of Renewable Energy Research, 8(3), 1369–1385. http://www.ijrer.org/ijrer/index.php/ijrer/article/download/7794/pdf.

Ali, Q. S., & Kim, M. H. (2021), Design and performance analysis of an airborne wind turbine for high-altitude energy harvesting. Energy, 230, 120829. https://doi.org/10.1016/j.energy.2021.120829

Amer, A., Ali, A. H. H., Elmahgary, Y., & Boddy, M. (2012). Wind energy potential for small-scale wind concentrator turbines. Advanced Civil, Environment Materials Research, 3138–3156 http://www.i-asem.org/publication_conference12/WSA-8.pdf.

André, L., & Maia, B. (2014, December). Experimental and numerical study of a diffuser augmented wind turbine
- dawt renewable energies and energy efficiency experimental and numerical study of a diffuser augmented wind turbine - DAWT,” Integrity-Reliability-Failure, 2018, no. July, pp. 1085–1100, [Online]. Available: https://core.ac.uk/reader/153412820. Avalon, F., Rogni, D., & Casalino, D. (2020). On the effect of the tip-clearance ratio on the aerocoustics of a diffuser-augmented wind turbine. Renewable Energy, 152(June), 1317–1327. https://doi.org/10.1016/j.renene.2020.01.064 Bet, F., & Grossmann, H. (2001). Upgrading conventional wind turbines. Renewable Energy, 28(1), 71–78. https://doi.org/10.1016/S0960-1481(01)00187-2. Bontempo, R., & Mann, M. (2020a). On the potential of the ideal diffuser augmented wind turbine: An investigation using a momentum theory approach of a free-wake ring-ruable energy, 87, 154–167. https://doi.org/10.1016/j.apenergy.2020.112794. Bontempo, R., & Mann, M. (2020b). Diffuser augmented wind turbines: Review and assessment of theoretical models. Applied Energy, 280(June), 115867. https://doi.org/10.1016/j.apenergy.2020.115867. Bontempo, R., Carandente, R., & Mano, M. (2021). A design of experiment approach as applied to the analysis of diffuser-augmented wind turbines. In Energy Convers. Manag. Vol. 235(n). October 2020, p. 113924. https://doi.org/10.1016/j.enconman.2021.113924. Dige, V. V., Avalon, F., & von Bussel, G. (2019 November). Effects of yawed inflow on the aerodynamic and aerocoustic performance of ducted wind turbines. Journal of Wind Engineering and Industrial Aerodynamics, 201, 104174. doi: https://doi.org/10.1016/j.wwpe.2020.104174. Foreman, K. (1981). Preliminary design and economic investigations of Diffuser Augmented Wind Turbines (DAWT). Final Report, 15 May 1979–31 March 1980, p. 28. Available. https://digital.library.unr.eduARK:/ 167531/metad1092691. Göltenbott, U., Ohy, Y., Yoshido, S., & Jamieson, P. (2017). Aerodynamic interaction of diffuser augmented wind turbines in multi-rotor systems. Renewable Energy, 112, 25–34. https://doi.org/10.1016/j.renene.2017.05.014. Guo, T., Guo, X., Gao, Z., Li, S., Zheng, X., Gao, X., Li, R., Wang, T., Li, Y., & Li, D. (2021, August). Nacelle and tower effect on a stand-alone wind turbine energy output—A discussion on field measurements of a small wind turbine. Applied Energy, 303, 117590. https://doi.org/10.1016/j.apenergy.2021.117590. Hansen, M. O. L., Sørensen, N. N., & Flay, R. G. J. (2000, October). Effect of placing a ducted wind turbine on wind turbine. Wind Energy, 3(4), 207–213. https://doi.org/10.1002/we.37 Harrington, C. (2006, October). Real-time urban picture sought. Jane's Defects Wkyd, 87, 1116–1123. https://doi.org/10.1016/eniconeman.2014.03.064). Hekal, H. A., Abu-Elyazeed, O. S. M., Nawar, M. A. A., Attai, Y. A., & Mohamed, M. M. S. (2018). On the actual power coefficient by theoretical developing of the diffuser flange of a wind-lens turbine. Renewable Energy, 125, 295–305. https://doi.org/10.1016/j.renene.2018.02.100. Hirahara, H., Hossain, M. Z., Kawahashi, M., & Nonomura, Y. (2005). Testing basic performance of a tiny wind turbine designed for multi-purposes. Renewable Energy, 30(8), 1279–1297. https://doi.org/10.1016/j.renene.2004.10.009. Hu, J. F., & Wang, W. X. (2011). Upgrading a shrouded wind turbine with a self-adaptive flanged diffuser. Energies, 8(6), 5319–5337. https://doi.org/10.3390/ en8065319. Jafari, S. A. H., & Kasosih, B. (2014a). An investigation on performance augmentation of diffuser shrouded micro wind turbine with computational fluid dynamics simulations (2.; Vol. 19. Jafari, S. A. H., & Kasosih, B. (2014b). Journal of wind engineering flow analysis of shrouded small wind turbine with a simple frustum diffuser with computational fluid dynamics simulations. Journal of Wind Engineering and Industrial Aerodynamics, 125, 102–110. https://doi.org/10.1016/j.jweia.2015.10.013. Kasosih, B., & Soleh Hudin, H. (2016). Influence of inflow turbulence intensity on the performance of bare and diffuser-augmented micro wind turbine model. WCE 2010 – World Congress Engineers, 2(Wce), 988–993. https://www.researchgate.net/publication/ 28974954. Li, Y. (2016). On the definition of the power coefficient of tidal current turbines and efficiency of tidal current turbine farms. Renewable Energy, 68, 868–875. https://doi.org/10.1016/j.renene.2013.09.020. Liu, Y., & Yoshida, S. (2015). An extension of the generalized actuator disc theory for aerodynamic analysis of the diffuser-augmented wind turbines. Energy, 93, 1852–1859. https://doi.org/10.1016/j.energy.2015.09.114. Lubitz, W. D., & Shomer, A. (2016). Wind loads and efficiency of a diffuser augmented wind turbine (DAWT). Proceedings Con Society Mechanical Engineers International Conference, 1–5. http://www.soec. uoguelph.ca//webfiles/wlubitz/ LubitzShomerCSME2014DAWTs.pdf Memon, Z. A., Sahito, A. A., Leghari, Z. H., & Shaikh, P. H. (2016). Output voltage characteristics of wind energy system considering wind speed and number of blades. Sindh University Research. Journal-SURJ (Science Series), 48(2). Nikolić, V., Petković, D., Shamshirband, S., & Ćobaslić, Ž. (2015). Adaptive neuro-fuzzy estimation of diffuser effects on wind turbine performance. Energy, 89, 328–333. https://doi.org/10.1016/j.energy.2015.05.126. Owis, F., Badawy, M. T. S., Abed, K. A., Fawaz, H. E., & Elfeky, A. (2015). Numerical investigation of loaded and unloaded diffuser equipped with a flap. International Journal of Scientific and Engineering Research, 6(11), 312–341. https://www.citefactor.org/journal profesional of loaded and unloaded diffuser equipped 12825–12831. Ożęb, O., & Ożębner, L. (2007). Exergy and reliability analysis of wind turbine systems: A case study. Renewable and Sustainable Energy Reviews, 11(8), 1811–1826. https://doi.org/10.1016/j.rser.2006.03. 004. Ranjbar, M. H., Rafie, B., Nasrasonadi, S. A., Gharali, K., Soltani, M., Al-Haq, A., & Nathwani, J. (2021). Power enhancement of a vertical axis wind turbine equipped with an improved duct. Energies, 14(18), 1–19. https://doi.org/10.3390/en14185780. Rivorolo, M., Freda, A., & Traverso, A. (2020). Test campaign and application of a small-scale ducted wind turbine with analysis of yaw angle influence. Applied Energy, 279(September), 115850. https://doi.org/10. 1016/j.apenergy.2020.115850.
Santoni, C., Corrasquillo, K., Arenas-Navarro, I., & Leonardi, S. (2017). Effect of tower and nacelle on the flow past a wind turbine. *Wind Energy*, 20(12), 1927-1939. https://doi.org/10.1002/we.2130

Shives, M., & Crawford, C. (2016). Computational analysis of ducted turbine performance. 3rd International Conference Ocean Energy, 6, 1-6. https://pic.s.uvic.ca/sites/default/files/uploads/publications/shives_cao_2010.pdf

Shukla, I., Tupkori, S. S., Raman, A. K., & Mullick, A. N. (2015 May). Wall Y* approach for dealing with turbulent flow through a constant area duct. *AIP Conference Proceedings*, 1440, 144–153. https://doi.org/10.1063/1.4704213

Siva Sankara Raju, R., Venkata Siva, B., & Srinivasa Rao, G. (2020). Advances in applied mechanical engineering. In *Lecture Notes in Mechanical Engineering*, 2020, pp. 381-388. https://doi.org/10.1007/978-981-15-1201-8

Sorribes-Palmer, F., Sanz-Andres, A., Ayuso, L., Sant, R., & Franchini, S. (2017). Mixed CFD-1D wind turbine diffuser design optimization. *Renewable Energy*, 105, 386–399. https://doi.org/10.1016/j.renene.2016.12.065

Toshimitsu, K., Nishikawa, K., & Ohya, Y. (2005). PIV measurement of flow field of the wind turbine with a brimmed diffuser. *Nihon Kikai Gakkai Ronbunshu, B Hen Transactions Japan Society Mechanical Engineers, B Part B*, 72(5), 1236-1240.

Vaz, J. R. P., & Wood, D. H. (2018). Aerodynamic optimization of the blades of diffuser-augmented wind turbines. *Energy Conversion and Management*, 123, 35–45. https://doi.org/10.1016/j.enconman.2016.06.015

Vaz, J. R. P., & Wood, D. H. (2018). Effect of the diffuser efficiency on wind turbine performance. *Renewable Energy*, 126, 969-977. https://doi.org/10.1016/j.renene.2018.04.013

© 2022 The Author(s). This open access article is distributed under a Creative Commons Attribution (CC-BY) 4.0 license.

You are free to:
Share — copy and redistribute the material in any medium or format. 
Adapt — remix, transform, and build upon the material for any purpose, even commercially. 
The licensor cannot revoke these freedoms as long as you follow the license terms.

Under the following terms:
Attribution — You must give appropriate credit, provide a link to the license, and indicate if changes were made.
You may do so in any reasonable manner, but not in any way that suggests the licensor endorses you or your use.
No additional restrictions
You may not apply legal terms or technological measures that legally restrict others from doing anything the license permits.

*Cogent Engineering (ISSN: 2331-1916)* is published by Cogent OA, part of Taylor & Francis Group.

Publishing with Cogent OA ensures:
- Immediate, universal access to your article on publication
- High visibility and discoverability via the Cogent OA website as well as Taylor & Francis Online
- Download and citation statistics for your article
- Rapid online publication
- Input from, and dialog with, expert editors and editorial boards
- Retention of full copyright of your article
- Guaranteed legacy preservation of your article
- Discounts and waivers for authors in developing regions

Submit your manuscript to a Cogent OA journal at www.CogentOA.com