Design and Flow Simulation of Concentrator Augmented Wind Turbine

S K Thangavelu, CY Goh and CV Sia
Faculty of Engineering, Computing and Science, Swinburne University of Technology, 93300 Kuching, Sarawak, Malaysia

* Corresponding Author: sthagavelu@swinburne.edu.my

Abstract. In order for wind technology to compete with conventional sources of energy in terms of energy production costs, researchers are working on different ways to increase the energy density in wind and therefore augment wind turbine power output. Concentrator augmented wind turbines (CAWTs) have been thought to be one of the ways to augment wind turbine power output and hence improve the cost effectiveness of wind energy. This concept involves enclosing the wind turbine rotor with a duct that reduces in cross-sectional area downstream of the blade-plane and hence increasing mass flow rate through the turbine. No efforts have been done to commercialize this concept because little is known about the wind flow behaviour in the CAWT, its influence on turbine power output and the optimum concentrator design parameters. The main objective of this research is to propose a new design for CAWT and optimize the concentrator design parameters. Furthermore, the effect of wind velocity and air pressure on the proposed design concepts are also investigated. Four different CAWT design concepts were designed and simulated using Solidworks and Computational Fluid Dynamics (CFD) tool Ansys Fluent, respectively. The wind velocity and air pressure vs. the position of convergent nozzle were plotted. Results showed an outstanding improvement in wind speed, which is 16.1 m/s (5.4 times increment), over the 3 m/s inlet wind velocity in the design concept 3 (CAWT 3.0) with the nozzle angle 20°.

1. Introduction

Wind energy is second largest source of energy and it is the fastest growing renewable energy resource around the world. The main problem of using wind energy is the uncertainty of wind pattern. Low wind speeds result in less energy per unit volume of air passing through a turbine. This leads to higher electricity production cost from wind energy than from fossil fuels [1]. In order for wind technology to compete with conventional sources of energy in terms of energy production costs, researchers are working on different ways to increase the energy density in wind and therefore augment wind turbine power output. Various power augmentation systems have been studied to improve the performance of wind rotors by increasing the energy density of the wind. One of them is the ducted augmentation system [2]. Diffuser Augmented Wind Turbine (DAWT) and Concentrator Augmented Wind Turbine (CAWT) are able to provide an opportunity to extract power from unstable low wind speeds.

A DAWT would have a duct surrounding the wind turbine blades that increase in cross-sectional area in the stream wise direction. The pressure behind the turbine will drop due to the wind turbine being enclosed by the diffuser, thus the wind velocity approaching the wind turbines will be increased [1-3]. CAWTs have been thought to be one of the ways to augment wind turbine power output and
hence improve the cost effectiveness of wind energy. This concept involves enclosing the wind turbine rotor with a duct that reduces in cross-sectional area downstream of the blade-plane and hence increasing mass flow rate through the turbine. The concept of a CAWT has been considered to improve the efficiency of the wind turbines by increasing the wind speed upstream of the turbine. Since the power produced by a wind turbine is proportional to the cubic power of the upstream wind velocity approaching the rotor, doubling wind velocity results in eight-fold increase in wind turbine power output. Thus, if a concentrator (convergent nozzle) is inserted around or in front of the turbine, to capture air from a larger area and deliver it to the rotor through a smaller area, more power will be produced for a given turbine diameter and wind speed, thus exceeding Betz limit. The mass flow rate of air passes through a given cross section of the concentrator in a unit length of time is given by \( \rho AV \), where \( \rho \) is density (kg/m\(^3\)) of the air, \( A \) is the cross sectional area (m\(^2\)) and \( V \) (m/s) is the flow velocity. Applying the continuity, \( \rho A_1 V_1 = \rho A_2 V_2 \). Where the subscripts 1 and 2 refer to concentrator inlet and outlet, respectively. Since \( A_1 > A_2 \) [4].

Anzai et al. [5] analysed the effect of the concentrator and its optimum design. It shows that the outlet diameter of the concentrator is smaller than the rotor diameter; and the inlet diameter is much larger than the rotor diameter. A wind turbine concentrator was designed, manufactured, and tested under different moist air density and velocity conditions by Orosa et al. [6]. The experimental and simulated results show an increase in the velocity ratio whenever the air density is reduced. Results showed that this wind turbine concentrator showed ideal working conditions during the summer time [6]. Ghajar & Badr [7] presented a small experimental set-up that comprised building, testing and evaluating a collector-and diffuser-augmented wind turbine (CDAWT). Studies were conducted on a bare, shrouded, collector augmented, diffuser-augmented and finally collector-and diffuser-augmented wind turbine. CDAWT increased the augmentation ratio by 56% over that of the DAWT.

Rus [8] presented the experimental study on the increase of the efficiency of vertical axis wind turbines by equipping them with wind concentrators. For the improvement of the power coefficient of the wind turbine, a concentrator (curtain) was used in order to cancel the negative moments that affect the rotational movement of the rotor and to increase the speed of the airflow at the entry into the rotor. Shonhiwa and Makaka [4] recently suggested from their review that CAWTs are a promising way of increasing the power output in low wind speed areas. It is important to solve the problem of receiving wind from one direction and develop power output models that take into account the fact that wind turbines operate in unsteady flow conditions most of the time. This would be useful in designing CAWTs systems. There is still a need to understand various flow features that may be present in the system such as turbulence, eddys, veer and wake effects and their influence on power output so that this can be incorporated into the power output model. Before making efforts to turn the CAWT concept into a commercial competitive product, more research work has to be done to come up with optimum concentrator designs. More experiments need to be done to optimise the concentrator wall length and the incident angle since they have much influence on concentrator frictional losses. To reduce time and cost on experimental work, numerical modelling can be used and then later validated with experimental results.

Most of the studies of the ducted rotor concern the effect of the diffuser while little research has been done concerning the concentrator (nozzle). No efforts have been done to commercialize this concept because little is known about the wind flow behaviour in the CAWT, its influence on turbine power output and the optimum concentrator design parameters. The main objective of this research is to propose a new design for CAWT and optimize the concentrator design parameters. Furthermore, the effect of wind velocity and air pressure on the proposed design concepts are also investigated.

2. Methodology

The new design of CAWT is designed by using Solidwork software with different parameters. The CAWT design concept 1 (CAWT 1.0) is shown in Figure 1. The CAWT 1.0 is a combination of two different sized concentrators (double nozzle). The total length of the concentrator (\( L \)) is fixed as 1000 mm and the inlet diameter (\( D_1 \)) is fixed as around 340 mm. The angle for second concentrator is fixed
as 12°. The outlet diameter ($D_2$) is 260 mm and the concentrator angle ($\theta$) will be tested for different angles, which are ~7°, ~10°, and ~12°.

CAWT design concept 2 (CAWT 2.0) is shown in Figure 2. The CAWT 2.0 is the combination of streamline shaped nozzle and small streamline shaped splitter inside the nozzle. It is believed that the wind velocity will be increased due to the geometry of the nozzle and presence of the splitter. The total length of the nozzle ($L$) is fixed as 1000 mm while the inlet diameter ($D_1$) is fixed as 400 mm and the angle $\theta$ will be tested for different angles, which are ~6°, ~9°, and ~12°.

CAWT design concept 3 (CAWT 3.0) is shown in Figure 3. The CAWT 3.0 is the combination of 2 different sizes of cone shaped nozzles. It is believed that the wind velocity will be increased when concave shape of nozzle is used. The total length of the nozzle ($L$) is fixed as 1000 mm and the inlet diameter ($D_1$) is fixed as 800 mm. The angle of outer nozzle is fixed as 15°and the angle $\theta$ will be tested for different angle which are ~10°, ~15°, and ~20°.

CAWT design concept 4 (CAWT 4.0) is shown in Figure 4. The CAWT 4.0 is the combination of 2 different shapes of nozzles. It is believed that the flange shape on the front nozzle would create right amount of wind behind the front nozzle, thus increasing the wind velocity. In addition, the presence of the second nozzle would create higher increase in the wind velocity. The total length of the nozzle ($L$) is fixed as 1000 mm while the inlet diameter ($D_1$) is fixed as 800 mm and the angle $\theta$ will be tested for different angles, which are ~12°, ~18°, and ~24°.

Figure 1. CAWT Design Concept 1 (CAWT 1.0)

Figure 2. CAWT Design Concept 2 (CAWT 2.0)

Figure 3. CAWT Design Concept 3 (CAWT 3.0)
Ansys Fluent is a CFD simulation tool, which is used for flow simulation. The air velocity and pressure can be investigated easily by using this tool. Boundary condition have been taken for simulation are: The initial wind velocity is set as 3 m/s; The inlet pressure is assumed as atmosphere pressure (0 kPa); Realizable k-epsilon model is used for the simulation and Single Reference Frames (SRF) model is used; and the airflow is assumed as incompressible steady state flow. Density based flow solver is used.

The turbulent flows can be described as the fluctuating velocity fields of the fluid. These fluctuations will be combined with transported quantities, for instance, energy, momentum and species concentration. Ansys Fluent provides three types of k-epsilon models which are standard k-epsilon model, Renormalization-group (RNG) k-epsilon model and Realizable k-epsilon model. Realizable k-epsilon model is strongly commended for use in this research as the flow features include strong vortices, rotation and streamline curvature. Hence, Realizable k-epsilon model is chosen for all the simulations in this research project, in order to obtain a more accurate data and results for concentrator wind turbine simulation.

Wall functions are a set of formulas that in effect of connecting the variables at near wall cells and quantities on the wall. The wall function contains the laws of the wall for mean temperature and velocity and principles for the near wall turbulent quantities. There are few cell types in the meshing process i.e. tetrahedron, hexahedron, prism, and pyramid. In this study, hexahedron mesh is used for the meshing because it provides higher quality solutions with less cells compared to other types of mesh. In addition, hexahedron meshes show reduced numerical diffusion when it is aligned with the flow. Besides, the boundary adaption is used to refine the cells in every edges of the concentrator. Flow simulations are done in order to investigate and identify the pattern of the air flow surrounding the concentrator. Different concentrator angle \( \theta \) for each of the different concept CAWT are tested and simulated.

### Results and discussion

In the CAWT 1.0, the wind velocity at the middle of the concentrator with various nozzle angles \( \theta \) (12°, 16°, 21°) was simulated and recorded. Table 1 shows the various velocity increments from the inlet to outlet of the nozzle for each angle \( \theta \). The graphs of velocity and pressure versus the location in the nozzle are shown in Figure 5. It was found that the largest increment of wind velocity with 3 m/s of inlet velocity occurred for the angle of 21°. Based on the Figure 5 (a), it can be seen that the pattern of three different graphs are the same. The highest wind velocity for three different angles nozzle is achieved between the distances of 0.4 m to 0.5 m. Within the same range, the air pressure has also reached the lowest point, as shown in Figure 5 (b). However, this concept is not the optimum design as the increment of velocity is low. Further design improvements are needed in order to obtain a better result.

| Concentrator (nozzle) angle \( \theta \) | ~12° | ~16° | ~21° |
|----------------------------------------|------|------|------|
| Highest wind velocity achieved (m/s)  | 4.385882 | 4.33528 | 5.593 |
| % Increment (initial velocity 3 m/s)   | 146.2% | 144.5% | 186.43% |
Table 2 Wind velocity increment in CAWT 2.0

| Concentrator (nozzle) angle θ | ~6°   | ~9°   | ~12°  |
|-------------------------------|-------|-------|-------|
| Highest wind velocity achieved (m/s) | 4.5449 | 5.15721 | 5.67145 |
| % Increment (initial velocity 3 m/s) | 151.5% | 182% | 189% |

Table 3 Wind velocity increment in CAWT 3.0

| Concentrator (nozzle) angle θ | ~10°  | ~15°  | ~20°  |
|-------------------------------|-------|-------|-------|
| Highest wind velocity achieved (m/s) | 12.3087 | 8.11366 | 16.1 |
| % Increment (initial velocity 3 m/s) | 410.3% | 270.5% | 537% |

Table 4 Wind velocity increment in CAWT 4.0

| Concentrator (nozzle) angle θ | ~12°   | ~18°   | ~24°   |
|-------------------------------|-------|-------|-------|
| Highest wind velocity achieved (m/s) | 6.5684 | 7.1407 | 8.16496 |
| % Increment (initial velocity 3 m/s) | 219% | 238% | 272% |

Figure 5. CAWT 1.0 (a) velocity (b) velocity simulation (c) air pressure (d) pressure simulation

Table 2 and Figure 6 show the comparison of velocity for three different angle of CAWT 2.0. The wind velocity at the outlet of the nozzle with various nozzle angles θ (6°, 9°, 12°) was simulated and recorded. The highest velocity of 5.67 m/s was achieved for the angle of 12°. Figure 6 shows the graphs for the three angles having the same trend, which is decreasing at the inlet and increasing at the outlet until it reaches the highest value. The wind velocity was increased at the outlet of the nozzle. Thus, this concept is not recommended due to low increment in velocity. Improvement on dimension and design is required in order to achieve a better performance.
Table 3 and Figure 7 show velocity and pressure simulation of CAWT 3.0. The wind velocity at the outlet of the nozzle with various nozzle angles $\theta$ (10°, 15°, 20°) was simulated and recorded. Table 3 & Figure 7 show a remarkable result with the highest velocity of 16.1 m/s and the increment of velocity is 537% with 3 m/s of inlet velocity. Figure 7 shows all three graphs having the same pattern, which achieved the highest velocity at the output of the nozzle. According to the continuity equation, a decrease in area would lead to an increase in velocity. The cone shaped nozzle focuses the air from the inlet towards the outlet. The smaller diameter of outlet causes the wind velocity increments, thus more energy from the wind can be extracted. Figure 7 shows that the highest velocity and lowest air pressure occurred at the outlet. It can be concluded that CAWT 3.0 is an optimum design for a nozzle augmented wind turbine.

![Figure 6. CAWT 2.0 (a) velocity (b) velocity simulation](image1)

![Figure 7. CAWT 3.0 (a) velocity (b) velocity simulation (c) air pressure (d) pressure simulation](image2)
Table 4 and Figure 8 show the comparison of velocity for three different angle of CAWT 4.0. The wind velocity at the outlet of the nozzle with various nozzle angles $\theta$ (12°, 18°, 24°) was simulated and recorded. Table 4 shows the comparison of highest velocity achieved with the three nozzle angles. The highest velocity of 8.16 m/s is achieved with the angle of 24°. The highest velocity is achieved at the same location where vortices are formed. It can be concluded that the vortices do influence the increment of velocity. The highest velocity occurred within the range of 0.6 to 1.0 m. However, the increment of velocity is lesser than CAWT 3.0, thus this concept is not recommended. With all the data that have been collected and obtained, it can be concurred that there are lots of room for improvement. The formation of vortices will contribute towards the increment of velocity and drop in air pressure. The formation of vortices creates a vacuum space in order to draw more air towards itself, thus wind velocity is increased. However, with wrong angle, the vortices would be formed inside the nozzle causing reduction in wind velocity at the outlet. It can be seen that the formation of vortices can be done by two different designs, either mixing the slow and fast flow or using the flange. Both ways have shown that the vortices can be formed with a right angle [7-9].

![Figure 8. CAWT 4.0 (a) velocity (b) velocity simulation](image)

**Conclusion**

This research analysed the various designs of nozzles for wind turbine by using Solidworks and Ansys Fluent. The results from the Solidworks and Ansys Fluent are collected and discussed. The CAWT 3.0 with angle $\theta$ of 20° is the ideal design with an outstanding result of 16.1 m/s air velocity with 537% of increment (initial 3 m/s). The behavior of the flow is based on Bernoulli’s principle and continuity equation.

**References**

[1] Kannan T S, Mutasher S A and Lau, Y H K 2013 *Journal of Engineering Science and Technology*, 8 (4) 372
[2] Matsushima T, Takagi S and Muroyama S 2006 *Renewable Energy* 31 (9) 1343
[3] Ohya Y, Karasudani T, Sakurai A, Abe K I, and Inoue M 2008 *Wind Engineering and Industrial Aerodynamics*, 96 (5), 524
[4] Shonhiwa C and Makaka G 2016 *Renewable and Sustainable Energy Reviews* 59 1415
[5] Anzai A, NemotoY and Ushiyama I 2004 *Wind Engineering* 28 605
[6] Orosa J A, García Bustelo E J, Oliveira A C 2012 *Energy Sources, Part A: Recovery, Utilization, and Environmental Effects* 34 (13) 1222
[7] Ghajar R F and Badr E A 2008 *International Journal of Mechanical Engineering Education* 36 58
[8] Rus L F 2012 *Journal of Sustainable Energy* 3 (1)
[9] Shikha S, Bhatti TS and Kothari D P 2005 *International Journal of Energy Technology and Policy* 3 394