Journal of Engineering Science and Technology
Special Issue on Development of Sustainability Systems - I, 131 - 143

NUMERICAL AND EXPERIMENTAL ASSESSMENT AND ANALYSIS OF A DIFFUSER-AUGMENTED WIND TURBINE

MOHANNED M. NAJI[1, *], BALASSA A. JABBAR[2], MUSTAFA A. ALMALIKI[3]

[1]School Al-Furat Al-Awsat Technical University, Al Najaf, Iraq
[2]School Al-Furat Al-Awsat Technical University, Al Najaf, Iraq
Babol Noshirvani University of Technology (NIT), Mazandaran, Iran
*Corresponding Author: mahdeali35@gmail.com

Abstract

Diffuser-augmented wind turbines (DAWT) are one of the ideas to address problems in places with low wind speeds. DAWT is a new type of horizontal wind turbine that uses a shroud as a hub to increase lower wind speeds. This study continued that effort and focused on reducing (curtailment) the overall diffuser size by implementing a numerical approach to discover optimum design parameters (L/D, θ, H/D, and γ) in 3D models to achieve the best increment in velocity ratio, ϵ value where a previous researcher's DWAT design was modified by changing the diffuser length, diffuser angle, shroud angle, flange height, and flange angle to create the ideal shape or design for small-sized wind turbines by using CFD simulations. The results of ANSYS CFX indicated that it is possible to obtain an optimum diffuser with a smaller size (compact size) than the previous researcher at a flange angle of 50, L/D = 0.6, H/D = 0.2 and get an increase in speed ratio of up to 2.2, then that diffuser was fabricated and experimentally tested. A reasonable affinity was found with the experimental results obtained from CFX. In addition, the cost has been reduced; it also concludes that the diffuser with a 50-flange angle within the optimum geometric parameters is acceptable to increase the flow velocity in the proposed location of the turbine rotor installation.

Keywords: Aerodynamic performance, CFD, Energy capture, Wind energy, Wind turbine design.

1. Introduction

One of the potential alternatives to energy from fossil fuels is wind energy. Wind energy was used in tropical regions, Europe, and America to rotate wind turbines, with an average speed is 10 m/s. Unfortunately, most of the wind turbines available in the market operate at high speeds. In contrast, the prohibitive areas lack wind speed, as the average speed reaches 5 meters per second. Therefore, it is required to design turbines that operate at low speeds by developing WORK the rotor with low speeds. One concept that emerged is DAWT, which is known as the diffuser used in horizontal-axis wind turbines [1]. Compared to a bare wind turbine, the diffuser or flanged diffuser creates separation regions behind it where low-pressure areas attract more wind into the rotors [2, 3]. When the turbine is less than 5 cm from the concentrator, and the concentrator's input-to-output ratio is 6, it performs at its best [4]. It is known that the power of wind turbines is proportional to the cube of the wind speed approaching the wind turbines. This information shows that a higher turbine power would be gained if the speed increases slightly. So, many researchers focused on finding a way to speed up the wind effectively [5-11]. Oya and others developed an efficient wind accelerator system by constructing a diffuser-shaped structure covering the wind turbines as previously proposed. Still, this design is distinguished from its predecessors by having a large flange at the outlet of the diffuser [12, 13].

Abe et al. [14] conducted experimental and numerical analyses of the flow fields of a small wind turbine with a flanged diffuser. Recent experimental and numerical research findings provided important information about the flow mechanism behind a wind turbine with a flanged diffuser. The way the tip vortex was destroyed differed noticeably between the naked wind turbine and the wind turbine with a flanged diffuser.

Owing to the significant interest among researchers in the subject matter, a comprehensive review has been undertaken to summarize and present the foremost papers on the diffuser-augmented wind turbine, as shown in Table 1.

Table 1. Summary of literature on turbines with diffuser

Name/ year	Method	parameters				Result
		L	***H***	***θ***	***α***	
[15]/ 2020	CFD	0.226D	-	7, 10, 20, 30°	-	Increase Cp by 30-60% at V = 5, and θ over 10
[16]/ 2020	CFD	2D	0.443D	12	14	Three diffuser designs were developed and modified using CFD. The diffuser with holes increases speed by 8.56 m/s over the initial velocity of 4 m/s.
[17]/ 2021	CFD	D	0.35D	10	10	The enhanced diffuser (IND-009) achieved 50% higher performance than the basic diffuser design.
[18]/ 2021	CFD	1.25D	-	4-16	-	The performance of HAWT with modified diffuser length and angle was examined using CFD. The results showed that turbines covered with a diffuser produced energy (1.4 - 2.9) times higher than those without the cover.

[19]/ 2021	CFD, EXP	0.1D, 0.137D, 0.221D, 0.37 D	0.05D0.1D 0.15D0.2D	12	-	Results revealed a 2–5 increase in energy rate compared to the bare wind turbines. The researcher created a new system for wind turbines that included a diffuser cover with a wide edge.
[20]/ 2021	CFD	(1-4) D	(0.1- 0.4) D	4, 2, 16	-	The results showed an increase in wind speed up to 1.77, as well as an increase in CP from (.76 2 - 26 .5)
[21]/ 2021	CFD	0.5D	0.25D	14.5	45	The researcher studied and analysed four different diffuser types to boost turbine power output and discovered that the flanged diffuser generates power up to 3.6 times faster than a naked turbine.
[22]/ 2022	CFD	0.25D	-	12	-	Two steps. The first step is determining the appropriate number of blades for the turbine (2, 3, 4). The second is the turbine location inside the diffuser. The turbine at the inlet increased energy by 14%-20%.
[23]/ 2022	CFD, EXP	1.98D	0.5D	10	-	Comparison between the curved and straight diffuser with edges by CFD at an angle of 10. The curved diffuser gives a higher speed than the straight diffuser by 7%.
[24]/ 2022	CFD, EXP	1.65D	-	15	-	Comparison of the perforated diffuser and the non-perforated diffuser using CFD and experimentally validated. Results showed an increase in speed by 46.8% and 27.4% compared to the un-drilled turbine.

2. Methodology

2.1. Experimental work and setup of wind tunnel

The basic parameters of an empty diffuser that are used in this paper are diffuser length L, diffuser angle Θ, flange height H, flange angle γ, and shroud angle, as shown in Fig. 1.

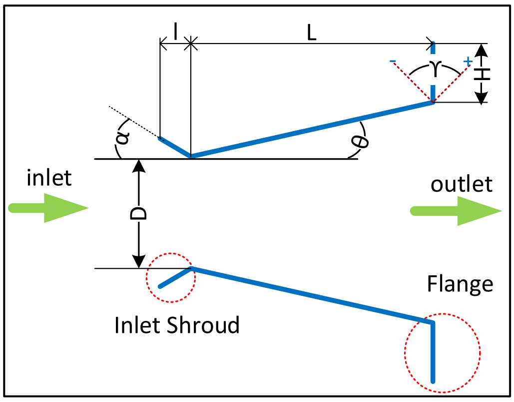


Fig. 1. The geometrical parameters of the DAWT.

The previous diffuser was manufactured experimentally and compared with numerical results by using CFD CFX R20. The basic parameters of the previous model were D = 310 mm, L/D = 1 mm, H / D = 0.35 mm, l = 4.5 mm, and Q = 10. As shown in Fig. 2, the measured velocity increase ratio (Ɛ) was calculated in 7 diffuser segments (along the diffuser). The test was carried out using an air blower manufactured in the Engineering Technical College / Najaf / Iraq/ Department of Mechanics, described in detail in Fig. 3.

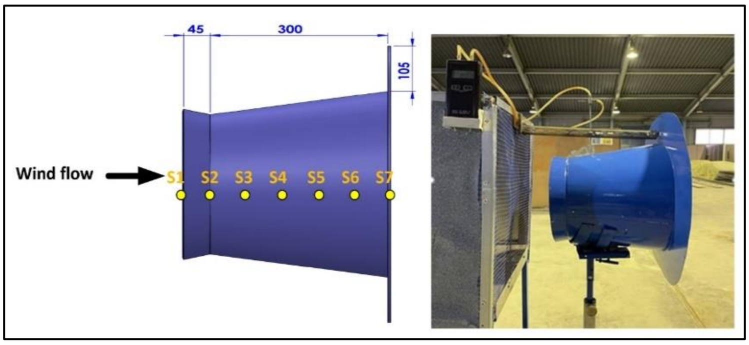


Fig. 2. The previous diffuser during the test.

A wind tunnel was constructed to study the aerodynamics of (DAWT) in the Department of Mechanical Engineering, Engineering Technical College, Najaf, Iraq. It is used for low speeds. The dimension of the tunnel is 1 meter high, 1 meter wide, and 3 meters long. The blower has a capacity of 2 horsepower, and a centrifugal fan was used to drive the airflow up to 13 m/s. An air streamlining screen has been manufactured with 1400 small square passes. The dimensions of each pass are 1.0 cm * 1.0 cm to obtain uniform air velocity. The intensity of the flow disturbance was less than 2%, and the regularity of the flow was more than 98%. Figure 3 shows the stages of manufacturing the wind tunnel.

Fig. 3. The previous diffuser during the test.

2.2. Simulation Development of diffuser and boundary condition

Several attempts were made using CFX to develop the previous diffuser. The study aims to determine the best flange angle for the diffuser, reduce the diffuser's size, and obtain the maximum possible speed ratio, as shown in Table 2.

Table 2. The diffuser specified and the variable values.

Variable	Minimum	Step	Maximum
L/D	0.3	0.1	1
h/D	0.2	0.05	0.3
***Θ* (deg)**	9	1	15
***γ* (deg)**	-10	5	60
***α* (deg)**	5	1	50

Mesh sensitivity analysis was performed to study the diffuser case considering the quality of the mesh as it gives the resolution in the solution. Irregular 3D and tetrahedral mesh were used for their flexibility in solving complex geometries. Table 3 and Fig. 4 show the mesh generation of the modelled geometry. The boundary conditions of the diffuser domain in ANSYS solver the commercial CFD code using ANSYS-CFX explored numerical simulations under steady-state conditions. The air was treated as an ideal gas with a temperature of 25 °C and a reference pressure of 1 atm throughout all domains. The flow was analysed using an isothermal steady state convergence criterion of 10-5 with turbulent model k-ω SST. The stationary inlet boundary at an inlet velocity of 5 m/s was applied, and the static pressure (0 Pa) was set at the outlet stationary boundary.

Table 3. Mish parameter and controls.

Parameter	Average Value	Standard Deviation
NO. nodes	556368	-
NO. Elements	1932359	-
Elements Quality	0.685	0.302
Aspect Ratio	9.9	036.72
Smoothing	High	-

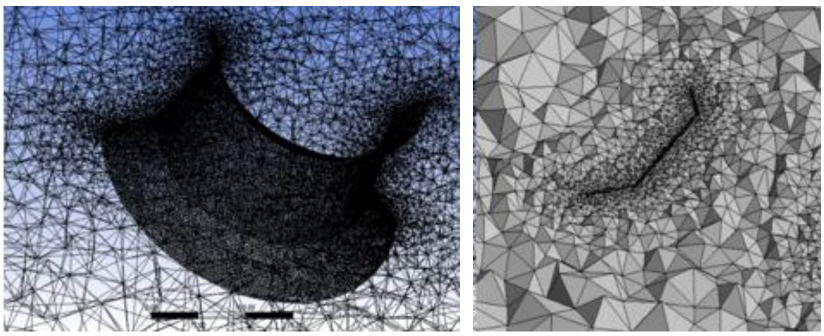


Fig. 4. Mesh generation of geometry.

2.3. Manufacturing and Experimental Tests

After reaching the optimal design from the previous simulations, the optimal diffuser with all dimensions shown in Fig. 5 was fabricated. Initially, 2 mm thick galvanized sheets were selected based on price, toughness, and performance. The chosen material was then moulded utilizing various production processes, including cutting, bending, and spot welding. Accurate dimensions and smooth surfaces were produced using precision manufacturing techniques. The basic parameters of the previous model were D = 310 mm, L / D = 0.6 mm, H / D = 0.3 mm, l = 4.5 mm, Θ = 12. was calculated the speed increase percentage for the factory diffuser, on the 7 points along the diffuser, then experimentally tested, as shown in Figs. 6 and 7.

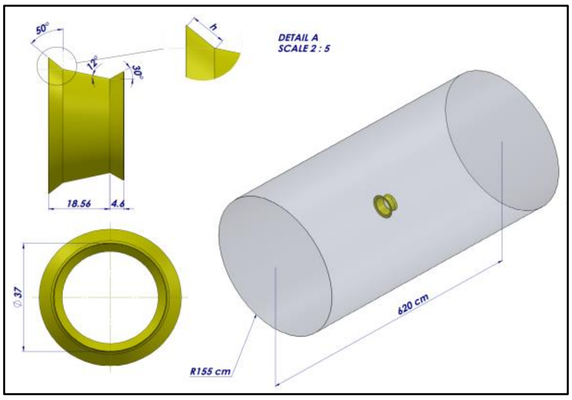


Fig. 5. Optimum design of the diffuser.

Fig. 6. Optimum diffuser manufacturing and testing.

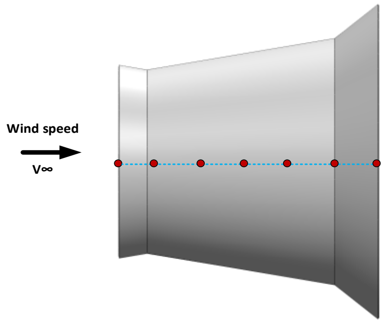


Fig. 7. The velocity ratio measured in 7 sections of the diffuser.

3. Result and Discussion

The average input velocity of the diffuser serves as the study's primary objective function. Table 4 displays the search ranges (constraints) for the variables determining the diffuser's shape. The optimization process is initiated on the geometry received from the parametric analysis by specifying the maximum number of iterations (81 Attempts) and precision (modulus = 10^{-6}). Figure 8 illustrates how the optimizer arrived at the ideal diffuser form after 81 attempts.

Using the resulting geometry, the diffuser average inflow velocity could be increased to 2.2 V_∞. Figure 9 compares the two shapes and contrasts the variables that determine the ideal diffuser shape with the output variables.

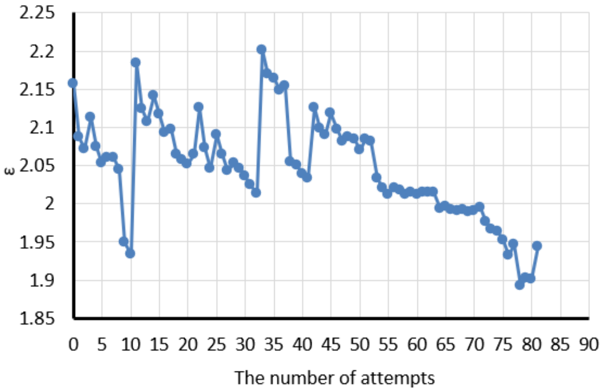


Fig. 8. History of diffuser optimization.

Table 4. Comparison of factors that distinguish two forms of the diffuser.

Parameter study	Attempts	*L/D*	*h/D*	*Θ* (deg)	*γ* (deg)	*α* (deg)	*ε*
Previous diffuser [17]	15	1	0.35	10	0	10	2
Optimum Diffuser	81	0.65	0.3	12	50	30	2.2

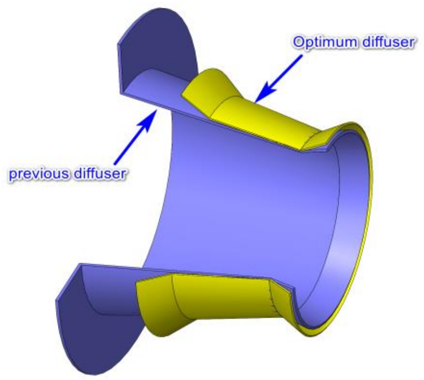


Fig. 9. Comparison between the two diffuser shapes.

Figure 10 illustrates the diffuser flange angle's impact on the diffuser's average entrance velocity. The results demonstrate that a forward flange angle can significantly enhance the average entrance velocity, surpassing the value obtained with zero flange angle, with a peak of 2.2 V∞ achieved when utilizing a 50° forward flange angle. The flange angle can increase the average velocity at the throttle due to the creation of eddies behind the diffuser, which increases the pressure drop and thus increases the velocity flow.

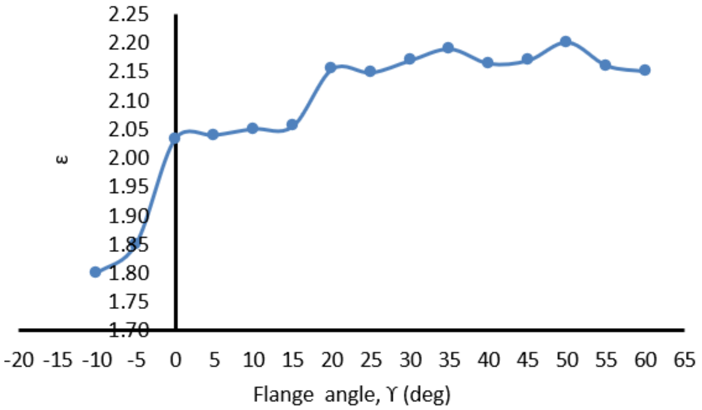


Fig. 10. Effect of flange angle on velocity ratio.

The velocity was measured using a Pitot tube in each of the diffuser's seven portions during the experimental phase, as illustrated in Fig. 11. The experimental phase and the simulation results revealed a reasonable convergence.

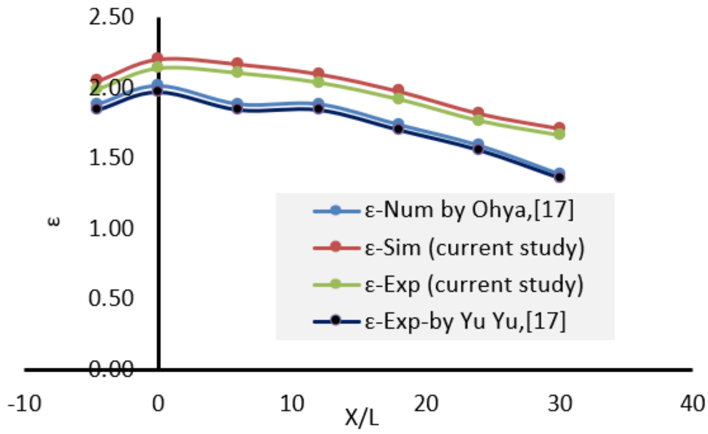


Fig. 11. Numerical and experimental results comparison of the current study's diffuser and the previous researcher's study.

Using CFD CFX Modelling, air flows may be represented visually and with precise information. These results support the notion that velocity increases when flanges are used at diffuser exits and help identify the ideal flange angle that produces the greatest velocity increase at the diffuser entrance. Pitot tubes were placed in each diffuser section throughout the experimental phase to measure the velocity, as shown in Fig. 10. The CFX data include the velocity magnitude at each point inside the physical domain and may easily extract velocity contours. The velocity contour and streamline inside and around the diffuser with a flange angle of β = + 50∘ are shown in Figs. 7 and 8. The data above show an increase in velocity at the diffuser entrance and a decrease in velocity in the area of low pressure behind the flange.

Figure 12 shows the impact of the diffuser shape (central body length with flange angles) on the diffuser's average entrance velocity. The diffuser entrance velocity is improved at various flange heights and open angles by lengthening the diffuser. Where there is no considerable pressure difference, the entrance velocity only marginally changes for diffuser length.

The wind speed at the diffuser inlet will also increase by installing a large flange surrounding the diffuser outlet. The wind speed increases near the diffuser entrance when a large flange is attached to the outside periphery of a diffuser exit. The flow is left to right, and vortices like the Karman vortex street can be seen downstream of the flange, as illustrated in Fig. 12 shows how the flange height affects the magnitude of the velocity. When the flange height is increased to h/d = 0.3, neither the exit pressure nor the exit velocity significantly changes. According to the diffuser length, the entrance velocity increased sharply for angles greater than or equal to 11° up to a specific angle.

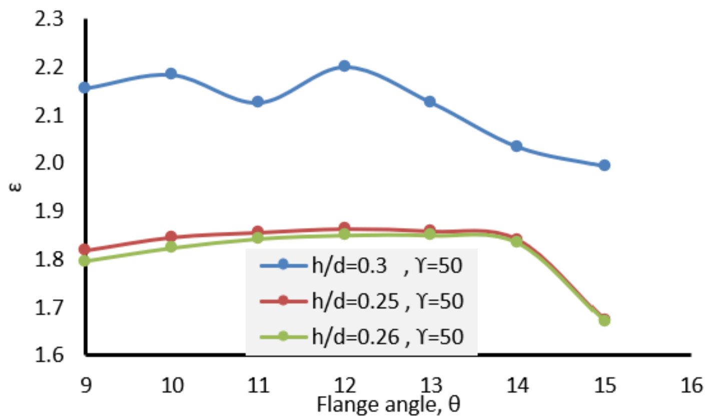
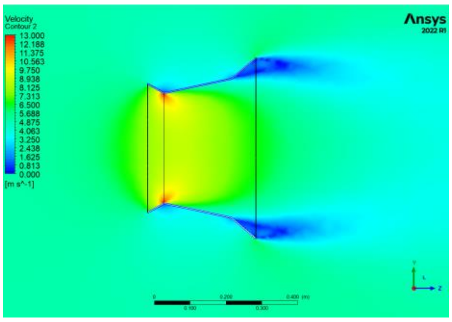
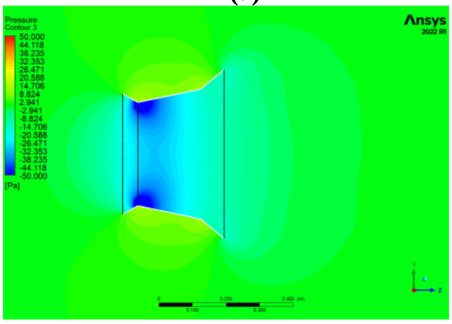


Fig. 12. Effect of diffuser shape on the mean entry velocity at a flange angle of 50 degrees

The simulated visualization for the optimum diffuser type was shown in terms of velocity contour, pressure contour, and velocity streamlines at a specified wind velocity of 5 m/s. The streamlines flow smoothly inside the diffuser, and the streamline velocity does not fluctuate considerably in the radial direction, Except for the boundary-layer region near the diffuser wall. As a result, the flow inside a diffuser is strongly connected to the surrounding flow, resulting in the formation of low pressure at the diffuser wake, which works as a vacuum, sucking in and speeding up the wind. In this regard, a flange placed at the diffuser outlet is critical for accelerating the oncoming flow. As seen in Figs. 13 and 14, a flange generates a massive separation behind it, where a very low-pressure region.

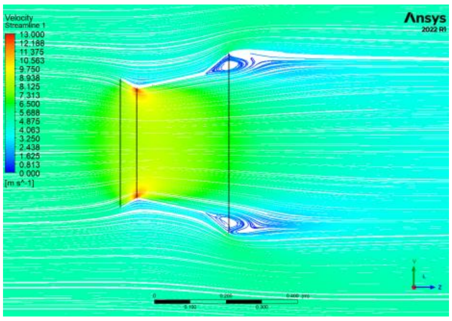


(a)

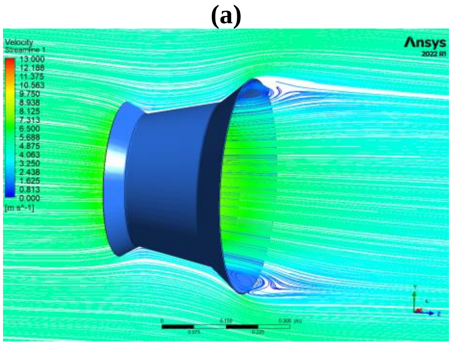


(b)

Fig. 13. (a) velocity contour of optimum design.
(b) pressure contour of optimum design.



(a)



(b)

Fig. 14. (a) 2D streamline of optimum design.
(b) 3D streamlining of optimum design.

4. Conclusion

The current study finds that the flow characteristics within and around the diffuser model for different flange angles are shown by the CFX program's numerical analysis. From the research, the following conclusions were made:

- The optimal flange angle of β = 50 has been identified, achieving the largest increase in inlet air velocity.
- The computed value of ε is (2.2) for the optimal flange angle (+50∘), and the change in flange angle is expected to result in a rise in power.
- The current numerical results support the existence of vortices behind the diffuser flange, which create an area of low pressure and ultimately increase the air velocity entering the diffuser.
- One of the most important benefits of the diffuser is that it helps to increase the airflow at the inlet, thus obtaining an increase in power.
- DAWTs produce more noise due to their vertical blade design, causing turbulent airflow and causing disturbances in residential areas. Additionally, DAWTs experience higher structural loads due to drag-based design, requiring stronger materials and higher construction and maintenance costs.

Nomenclatures

C_p	Power coefficient
D	Diffuser diameter, m
H	Flange high
$V\infty$	Upstream wind velocity, m/s

Greek Symbols

α	Shroud angle, deg.
γ	Flange angle, deg.
θ	diffuser angle, deg.
H	Flange high
ε	Ratio of maximum axial to upstream wind velocity in the diffuser

Abbreviations

BHAWT	Bare Horizontal Axis Wind Turbine
CFD	Computational Fluid Dynamic
DAWT	Diffuser Augmented Wind Turbine
EX	Experimental
RM	Research Method

References

1. Maulana, M.I.; Syuhada, A.; and Nawawi, M. (2018). Analysis of diffuser augmented wind turbine (DAWT) with flange and curved interior using CFD. In *AIP Conference Proceedings*, 1, 020025.
2. Bet, F.; and Grassmann, H. (2003). Upgrading conventional wind turbines. *Renewable Energy*, 28(1), 71-78.

3. Van Bussel, G.J. (2007). The science of making more torque from wind: Diffuser experiments and theory revisited. *Journal of Physics: Conference Series*, 75, 012010
4. Shonhiwa, C.; and Makaka, G. (2016). Concentrator augmented wind turbines: A review. *Renewable and Sustainable Energy Reviews*, 59, 1415-1418.
5. Igra, O. (1981). Research and development for shrouded wind turbines. *Energy Conversion and Management*, 21(1), 13-48.
6. Gilbert, B.L.; and Foreman, K.M. (1983). Experiments with a diffuser-augmented model wind turbine. *Journal of Energy Resources Technology*, 105(1), 46-53.
7. Rauh, A.; and Seelert, W. (1984). The Betz optimum efficiency for windmills. *Applied Energy*, 17(1), 15-23.
8. Phillips, D.; Richards, P.; and Flay, R. (2002). CFD modelling and the development of the diffuser augmented wind turbine. *Wind & Structures*, 5(2), 267-276.
9. Hansen, M.O.L.; Sørensen, N.N.; and Flay, R. (2000). Effect of placing a diffuser around a wind turbine. *Wind Energy*, 3(4), 207-213.
10. Nagai, M.; and Irabu, K. (1987). Momentum theory of diffuser augmented wind turbine. *Nippon Kikai Gakkai Ronbunshu, B Hen*, 53(489).
11. Ushiyama, I. (1997). *Introduction of wind turbine*. Sanseido Press, Tokyo.
12. Ohya, Y.; Karasudani, T.; and Sakurai, A. (2003). Development of high-performance wind turbine with brimmed diffuser. *Japan Society of Aeronautical Space Sciences*, 50(587), 477-482.
13. Ohya, Y.; Karasudani, T.; Sakurai, A.; and Inoue, M. (2004). Development of High-Performance Wind Turbine with a Brimmed-Diffuser - Part 2. *Japan Society of Aeronautical Space Sciences*, 52(604), 210-213.
14. Abe, K.; Nishida, M.; Sakurai, A.; Ohya, Y.; Kihara, H.; Wada, E.; and Sato, K. (2005). Experimental and numerical investigations of flow fields behind a small wind turbine with a flanged diffuser. *Journal of wind engineering and industrial aerodynamics*, 93(12), 951-970.
15. Takeyeldein, M.; Lazim, T.M., Ishak, I.S.; and Nik Mohd, N.A.R. (2020). Wind lens performance investigation at low wind speed. *Evergreen*, 7(4), 481-488.
16. Thangavelu, S.K.; Wan, T.G.L.; and Piraiarasi, C. (2020). Flow simulations of modified diffuser augmented wind turbine. in *IOP Conference Series: Materials Science and Engineering*, 886, 012023.
17. Maw, Y.Y.; and Tun, M.T. (2021). Sensitivity Analysis of Angle, Length and Brim Height Of The Diffuser For The Small Diffuser Augmented Wind Turbin. *ASEAN Engineering Journal*, 11(4), 280-291.
18. Susandi, A.; Arifin, F.; and Kusumanto, R. (2021). Simulation of Diffuser Parameters in the Performance of Horizontal Axis Wind Turbine using Computational Fluid Dynamics. *Technology Reports of Kansai University*, 63(06).
19. Rezek, T.; Camacho, R.G.R.; Filho, N.M.; and Limacher, E.J. (2021). Design of a hydrokinetic turbine diffuser based on optimization and computational fluid dynamics. *Applied Ocean Research*, 107, 102484.

20. Elsayed, A.M.; (2021). Design Optimization of Diffuser Augmented Wind Turbine. *CFD Letters*, 13(8), 45-59.
21. Chhetri, RK; Upadhyay, D.B.; and Ghimire, N. (2021). Comparative Analysis of Diffusers for Micro Wind Turbine. *Nepal Journal of Science and Technology*, 20(2), 103-112.
22. Shkhair, M.M.; Jaber, O.K.; and Al-Absi, S.M. (2022). Effect of Rotor Blades Number and Rotor Position on the Performance of a Diffuser Augmented Wind Turbine. *International Journal of Mechanical Engineering and Robotics Research*, 11(11).
23. Heyru, B.; and Bogale, W. (2022). Flow field analysis and testing of curved shroud wind turbine with different flange angle. *Cogent Engineering*, 9(1), 2095951.
24. Sridhar, S.; Zuber, M.; Shenoy B.S.; Kumar, A.; Ng, E.Y.K.; and Radhakrishnan, J. (2022). Aerodynamic comparison of slotted and non-slotted diffuser casings for Diffuser Augmented Wind Turbines (DAWT). *Renewable and Sustainable Energy Reviews*, 161, 112316.