

COMPUTATIONAL AND EXPERIMENTAL RESEARCH ON CONICAL DIVERGING DIFFUSERS FOR ENHANCING WIND TURBINE PERFORMANCE: A COMPREHENSIVE REVIEW

Mangesh R. Mahajan¹
Sanjay D. Nikhade

Received 04.12.2024.
Revised 27.05.2025.
Accepted 18.06.2025.

Keywords:

A B S T R A C T

Diffuser-Augmented Wind Turbines (DAWTs), Computational Fluid Dynamics (CFD), Conical Diffusers, Diverging Diffusers

A significant feature of a wind turbine is the possibility of improving the power output by increasing the free-stream wind velocity. This enhancement can be achieved by using a diffuser. The diffuser generates a low-pressure area, thereby drawing a larger volume of air through it. Among the different diffuser designs, the conical diverging diffuser has been widely investigated for several decades owing to its simplicity and effectiveness. Understanding its historical development, various stages of research, and present status is essential for making it acceptable. This paper presents an overall review of selected research on conical diverging diffusers for enhancing the wind turbine power output. A total of 18 research case studies were studied, and the problem statement, methodology, analysis techniques, and important outcomes were systematically presented along with important figures.



© 2025 Published by Faculty of Engineering

1. INTRODUCTION

The use of renewable energy devices for grid-connected and distributed energy generation is increasing continuously. Solar photovoltaics are now widely accepted because of their considerably lower price. In addition to solar energy acceptance, wind power is a major source of clean, renewable energy that has been spreading across the globe because of its sustainability and reduced dependency on fossil fuels (Sridhar et al., 2022). In recent decades, research on small wind turbines (SWT) has focused on the wind turbine blade aerodynamics and strength design (Alaimo et al., 2015; Kale et al., 2022; Garmode et al., 2022). However, in recent years, research on small wind turbines has shifted

to enhancing their performance using diffusers (Akhter & Omar, 2021; Agha et al., 2018). This study presents a comprehensive review of selected computational and experimental techniques applied to conical diverging diffusers to enhance wind turbine performance.

Small wind turbines used for distributed energy production are most commonly not placed at windy sites where large wind turbines are placed (Oliveira et al., 2025). Hence, SWT do not perform well because the power produced by any wind turbine is significantly reduced at low wind speeds. Researchers have used the concept of accelerated wind velocity owing to pressure reduction and surrounding the wind turbine using a basic conical diverging diffuser to accelerate the wind

¹ Corresponding author: Mangesh R. Mahajan
Email: mangesh.mahajan@mitwpu.edu.in

speed (Oman & Foreman, 1973). Figure 1 shows a wind turbine covered by a conical diverging diffuser. This basic diffuser shape is still useful and has many advantages over modern diffuser shapes (Agha et al., 2018). Hence, this review focuses on it. Most commonly, the combination of wind turbines and wind velocity-enhancing components (commonly termed diffusers) is called the Diffuser-Augmented Wind Turbine (DAWT). DAWT can achieve 1.5 to 4 a power production that associated with a bare wind turbine (BWT) depending on the diffuser's geometry, dimensional parametric combination, and operational environment.

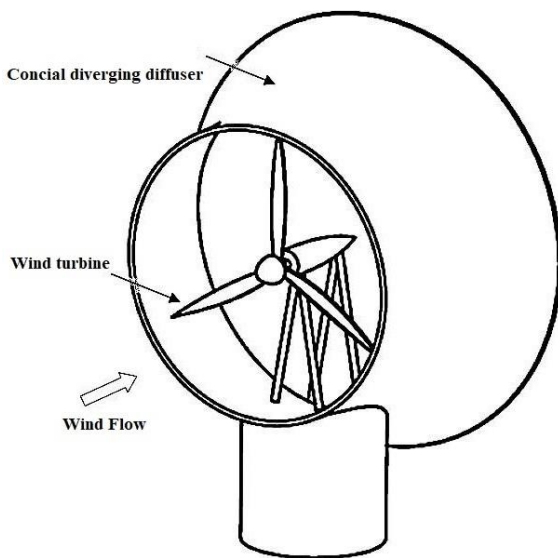


Figure 1. DAWT using a conical diverging diffuser (Teklemariyem et al., 2024).

A conical diverging diffuser is a widely studied diffuser shape in wind energy research to enhance the kinetic energy extraction from the wind. The primary principle of a conical diverging diffuser is to enhance the wind velocity through a turbine rotor by using geometric expansion and pressure gradients (Mahalakshmi et al., 2007; Prasad et al., 2020; Kale & Sapali, 2014) to obtain a higher wind flow compared to the BWT rotor. This principle makes conical divergent diffusers useful in increasing not only the velocity but also the effective power output of wind turbines. Figure 2 presents a schematic of a conical diverging diffuser that creates a low-pressure region to increase the approach wind velocity.

Figure 3 shows the working principle of a wind turbine with a flanged diffuser. The wind enters from the inlet and passes through a wind turbine placed inside the diffuser. The flange helped create strong vortices at the outlet. These vortices generated more wind inside the diffuser. Because of this arrangement, a low-pressure region was formed at the outlet. This low pressure allows extra air flow through a turbine and results in more energy capture by the DAWT compared to a BWT.

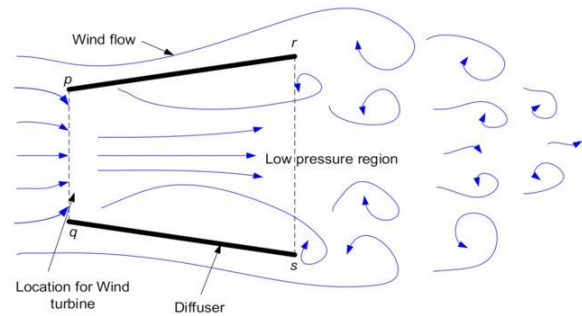


Figure 2. Wind velocity enhancement principle using conical diverging diffuser (Mohanani et al., 2021)

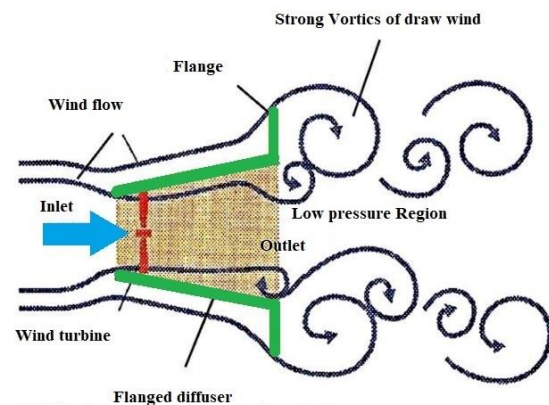


Figure 3. Wind velocity enhancement principle using flanged conical diverging diffuser (Ohya et al., 2008)

2. METHODOLOGY

The present review was conducted with the objective of compiling and analyzing the available computational and experimental studies on conical diverging diffusers used for wind turbine power enhancement. The methodology used for this review followed a structured approach to identify, classify, and evaluate the relevant literature.

First, a search of published research papers, conference proceedings, and reports was carried out using main scientific archives, such as Scopus, Web of Science, and Google Scholar. Keywords including the “conical diffuser”, “diverging diffuser”, “flanged diffuser”, “wind turbine diffuser, and “computational fluid dynamics (CFD) analysis of diffusers” were used in different combinations. The search was limited to studies that specifically investigated diffuser-augmented wind turbines or diffusers with conical diverging shapes.

Then, the selected studies were found using computational or experimental studies for conical diffusers with and without flanges. Some researchers have studied the performance of empty diffusers (diffuser without rotor), and few have studied them by placing a rotor. During the experiment, some studies

were performed in a laboratory using a wind tunnel, and a few were performed in the field under actual working conditions. These research papers further studied diffuser dimensions and geometric parameters, including length-to-diameter ratio (L/D), divergence angle (α), and flange height (h). In addition, the inlet wind velocity used in these studies area loss was included in this study. A comparative summary of the important parameters from each study was also prepared.

Additionally, figures, flow visualizations, and CFD plots from published papers were suitably added whenever required. Wherever available, the experimental validation of the computational results was studied and added to this paper. Finally, the collected information is used to identify gaps in the existing literature and potential directions for future research.

Developments in the field of conical diverging diffusers are presented in the next section, arranged from earlier to more recent research publications.

3. RESEARCH STUDIES ON CONICAL DIVERGING DIFFUSER

3.1 Case Study 1

Abe and Ohya (2004) performed a numerical study on flanged diffusers for SWT, which consisted of a conical diverging diffuser covering a wind turbine with a large flange attached at the exit. Figure 4 shows the flanged diffuser placed in the computational domain used for the CFD analysis. They used a nonlinear eddy-viscosity model to predict complex turbulent flows. For CFD, they used boundary conditions of a Reynolds number of 20,000 and an inlet velocity of 5 m/s. During the CFD analysis, variables such as diffuser length (L), diffuser inlet diameter (D), diffuser opening angle (α), flange height (h), and loading coefficient (C_t) were varied. They also compared the CFD results with experimental results to confirm the accuracy of the model.

The study showed many important findings regarding flanged diffuser performance. The addition of a flange generates low pressure behind it and results in a more accelerated flow compared to the diffuser without a flange. The optimum loading coefficient for a flanged diffuser was lower than that for a BWT. A larger diffuser opening angle (15°) can provide a higher performance under certain conditions, despite the risk of flow separation (Abe & Ohya, 2004).

3.2 Case Study 2

Kosasih and Tondelli (2012) conducted experimental studies to compare the performance of a micro BWT and micro DAWT for two different diffuser shapes. They performed a wind tunnel test on a micro wind turbine by using two different diffusers. One conical

diverging diffuser was used with a flange, and in the other, a nozzle was placed before the diffuser. Figure 5 shows a schematic of the two diffuser designs used. Figure 6 shows an experimental testing photograph of a conical diverging diffuser with a flange.

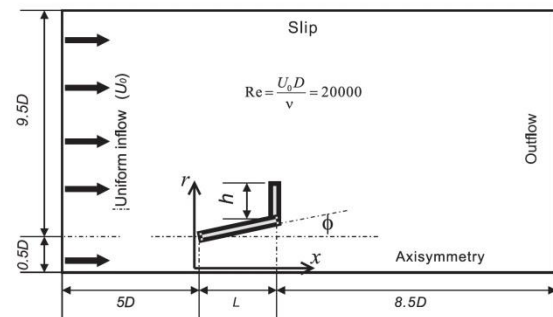


Figure 4. Computational Domain Layout used (Abe & Ohya, 2004)

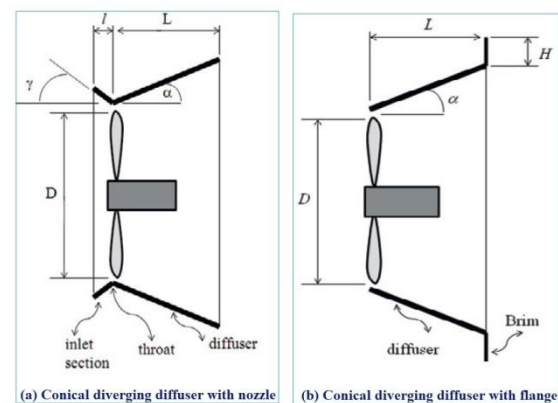


Figure 5. Diffusers Models (Kosasih & Tondelli, 2012)



Figure 6. Experimental testing of conical diverging diffuser by (Kosasih & Tondelli, 2012)

During this research work they studied three different designs: straight diffuser, conical diverging diffuser with a flange, and nozzle combined with a conical diverging diffuser. They studied diffuser length (L/D) ranging from 0.63 to 1.5, flange (brim) heights (H/D) from 0 to 0.2, and diffuser expansion angle of 12° . They observed that a diffuser in the form of only a shroud increased the performance by 60% compared with the

BWT. The nozzle combined with the conical diverging diffuser exhibited a slightly higher enhancement of 63%. A conical diverging diffuser with a flange also delivered significant performance that was compatible with the first two. Additionally, they observed that increasing the diffuser length (L/D) lifted the performance curve to higher tip speed ratios (TSR) but did not affect the optimum power coefficient (C_p). The addition of a flange at the diffuser exit increased the power and reduced the cut-in speed. Furthermore, it shifted the optimum to a higher TSR.

However, the effect of the wind tunnel blockage, particularly for heavily loaded turbines with diffuser augmentation, was not considered in this study. They suggested future computational fluid dynamics simulations to obtain more details (Kosasih & Tondelli, 2012).

3.3 Case Study 3

Kannan et al. (2013) performed different studies on three modified models of a conical diverging diffuser. Figure 7 shows the different concepts used in their research. Concept 1 adds a short nozzle to a conical diverging diffuser with a flange. Concept 2 adds a smaller inner diffuser splitter at the exit side. Concept 3 adds a cone-shaped splitter to the exit side. Concept 4 uses a double conical diverging diffuser with a cone-shaped splitter.

They prepared models using SolidWorks and completed CFD analysis using ANSYS Fluent. The major main body length, entrance diameter, expansion angle, flange height, and splitter open angle are the variable parameters. The initial wind velocity was set to 4 m/s.

After CFD analysis, they observed that Concept 1 achieved a 67.12% increase in wind velocity with a 12° open angle, which is comparable to previous studies. Concept 4 showed a 54.5% increase with $\alpha = 16^\circ$. Based on these findings, an optimized model was tested using CFD and a 61.25% increase in wind velocity was found.

They concluded that the addition of the nozzle can provide improved flow direction and streamlines, whereas flanges at the diffuser end can reduce backflow and vortices. For future work, the authors suggested further optimization of the diffuser design, considering the power coefficients and momentum transfer to the turbine blade (Kannan et al., 2013).

3.4 Case Study 4

Barbosa et al. (2013) proposed a mathematical model for predicting the velocity profile inside a conical diffuser to improve the design of a DAWT. The proposed model combines uniform flow with the flow induced by the vortex rings at the diffuser inlet and

outlet. the Biot-Savart law was used to calculate the induced velocity field considering the diffuser geometry and flow behavior. The model assumed a conical diffuser shape and modeled the suction at the outlet and flow blockage at the inlet using steady vortex rings.

They conducted experiments to validate their model using a conical diverging diffuser with an inlet diameter of 115 mm, an outlet diameter of 259 mm, and a length of 87 mm. The diffuser was tested in a wind tunnel at five different rotations (400-800 rpm), and measurements were taken externally to avoid wall effects. Figure 8 shows the experimental setup used by Barbosa et al. (2013).

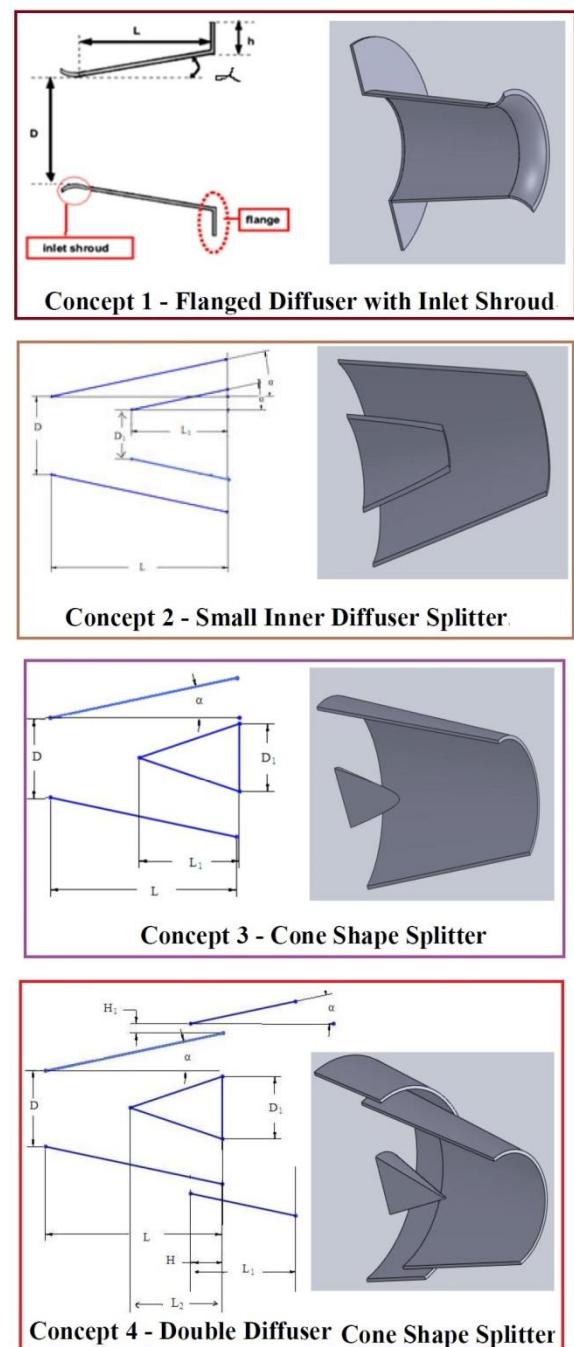


Figure 7. Four concepts studied (Kannan et al., 2013)



Figure 8. Experimental setup (Barbosa et al., 2013)

The proposed model matched the experimental data well, particularly in the internal region of the diffuser. This experiment was useful for determining the optimal turbine placement within the diffuser. The velocity distribution matched well with the experimental records from the point of minimum velocity to the maximum point but diverged in regions upstream and downstream of the diffuser. The study concludes that despite some limitations, the proposed model can be applied to DAWT. They observed that the length-to-inlet diameter ratio plays an important role in obtaining the plane of maximum velocity ratio and suggested it as an area for further research (Barbosa et al., 2013).

3.5 Case Study 5

Jafari and Kosasih (2014) used three dimensional CFD to analyze a conical diverging diffuser with a rotor placed inside. They used a commercial AMPAIR 300 W micro wind turbine with a diameter of 1.3 m for simulations. They studied diffusers with different L/D ratios and area ratios. They conducted simulations at a constant inlet wind velocity of 10 m/s and various rotor speeds to achieve different TSR. They used the SST $k-\omega$ RANS turbulence model, with additional transport equations. The Multiple Reference Frame (MRF) methodology was applied for the steady-state approximation. Figure 9 shows the computational setup used by them.

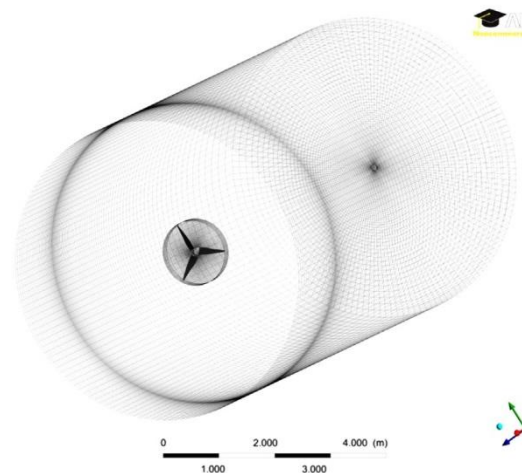
They observed that the area ratio significantly affected the backside pressure, which is the most important factor in velocity enhancement. However, the growing area ratio also causes more flow separation from the diffuser body, which reduces velocity enhancement. This effect can be controlled by varying the diffuser length. They found the optimal dimensionless parameters for effective diffuser performance (Jafari & Kosasih, 2014).

3.6 Case Study 6

Roshan et al. (2015) analyzed the effect of a stepped duct on DAWT performance in urban environments using computational and experimental methods. The objective of this research was to improve upon simple

ducted designs by adding a step inside the diffuser to increase the wind velocity and power generation. An empty diffuser was developed and tested experimentally using a wind tunnel and CFD. For CFD they used $k-\omega$ SST turbulence model for CFD. The MRF was included for the wind turbine rotor rotation.

a



b

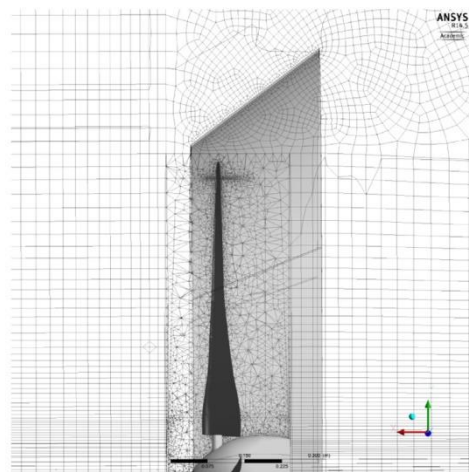


Figure 9. Computational setup (Jafari & Kosasih , 2014)

The diffuser had an entry diameter 0.12 m, an exit diameter of 0.145 m and length of 0.755 m. 1 A step with a length of 1/3 of the length was added at three positions within the diffuser. The wind turbine had a rotor radius of 1 m. Figure 10 shows the three arrangements and the experimental setup used. They also studied three turbine positions within the stepped diffuser: entry, middle, and exit.

They observed that placing the step at the exit side produced the highest wind velocity increase. The stepped diffuser design improved the pressure difference compared with a simple diffuser.

The stepped diffuser increased the wind velocity inside the diffuser more than simple conical diverging. Locating a turbine at the stepped diffuser entry resulted

in maximum power generation. The power of the DAWT using a stepped diffuser was approximately 16% higher than that of the BWT (Roshan et al., 2015).

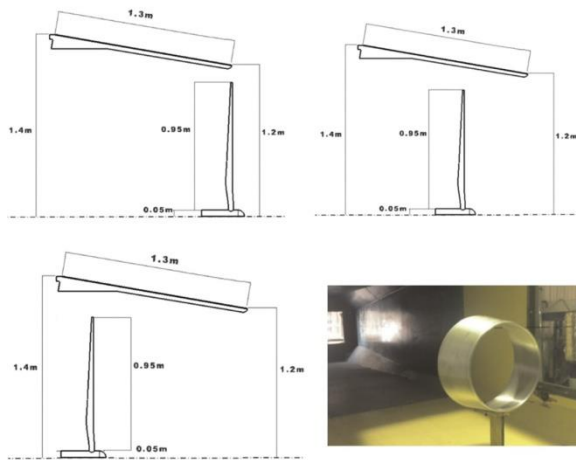


Figure 10. Three arrangements and experimental setup used by (Roshan et al., 2015)

3.7 Case Study 7

Lipian et al. (2015) optimized a conical diverging diffuser design for a 3 kW wind turbine. The study focused on two main parameters: diffuser expansion angle and flange height.

Researchers have developed a numerical model that was validated by wind tunnel testing. They used CFD with two turbulence models (SST and $k-\epsilon$) and compared the results with the experimental results. The study examined 48 geometries, covering three different flange heights (0.1D to 0.5D in the step of 0.2) and 16 diffuser angles (0° to 30° with 2° intervals). The simulated DAWT had an inlet diameter of 2.5 m, inlet velocity of 6 m/s, and 5% turbulence intensity. Figure 11 shows the velocity contour obtained for a case study by Lipian et al. (2015).

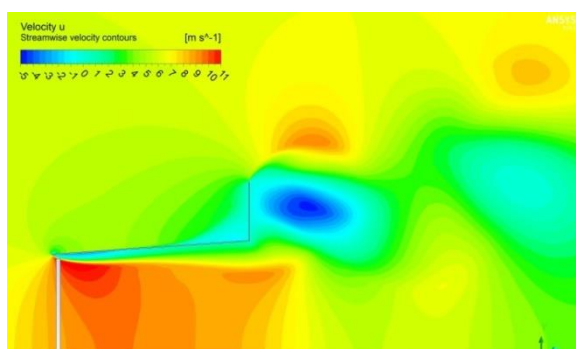


Figure 11. CFD Velocity contours (Lipian et al., 2015)

The sensitivity study showed that the flow velocity gain can be as high as 1.6 times the free stream velocity for diffuser angles of approximately 6° . Smaller diffuser angles favor higher flange heights. Angles between 18° and 26° show a satisfactory flow velocity increase but

may lead to flow instabilities. In some cases, a velocity increase of up to 2 times was observed (Lipian et al., 2015).

3.8 Case Study 8

El-Zahaby et al. (2017) investigated the effect of different flange angle on the performance of a conical diverging diffuser with a flange. The study focused on optimizing the flange angle for the maximum velocity increase at the diffuser entrance, thereby enhancing the wind turbine power output.

They used GAMBIT for 2D axisymmetric grid development and FLUENT 6.3 for flow field analysis. The diffuser model had a fixed L/D of 1.5, an outlet diameter, and a flange-to-entry diameter ratio of 0.25. The flange inclination was varied within 25° to either side of the vertical position in 5° increments. Figure 12 shows the research problem solved by El-Zahaby et al. (2017). They tested all the diffuser models at an inlet velocity 4.5 m/s. The study used the $K-\epsilon$ turbulence model and SIMPLE algorithm to solve partial differential equations.

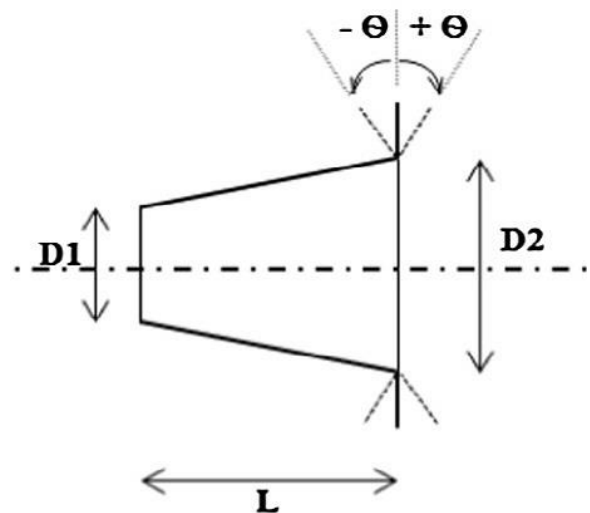


Figure 12. Research problem (El-Zahaby et al., 2017)

The numerical results matched well with previous experimental testing records, validating the CFD approach. The simulations observed vortex creation on the back side of the flange, creating a low-pressure area that accelerates fluid flow through the diffuser. The important findings from their research include the following observations: The optimal flange angle was found to be $+15^\circ$, which produced the maximum velocity at diffuser entry. At the optimal angle, the velocity at the diffuser entrance reached 5.63 m/s, compared with the inlet velocity of 4.5 m/s. The enhancement in power at the optimal flange angle was 1.953 times, and at a normal (0°) flange angle, it was 1.903 times that of the BWT. This optimization resulted

in a 5% enhancement in power generation compared with a normal flange (El-Zahaby et al., 2017).

3.9 Case Study 9

Olasek et al. (2016) studied the performance of a conical-diverging DAWT through experimental and computational techniques. They used a wind tunnel for the experimental and computational CFD results. The study focused on a flanged diffuser with a length and inlet diameter of 540 mm, exit diameter of 580 mm, flange height of 62 mm, and expansion angle of 4.25° . Three scaled models (D80, D30, and D5) were created to analyze the effect of the wind tunnel blockage. An inlet velocity of 18.6 m/s was fixed for most of the experiments. For experimental studies, they used Particle Image Velocimetry (PIV) to visualize flow patterns, pneumatic measurements to obtain velocity distributions, and wind turbine power measurements using a mechanical brake system. Figure 13 shows the experimental setup developed by Olasek et al. (2016).



Figure 13. Experimental setup (Olasek et al., 2016)

The analysis showed significant flow acceleration inside the diffuser, with velocities reaching approximately 25 m/s compared with the 18.6 m/s free stream velocity. The researchers observed a double-vortex structure behind the flange, which created a low-pressure zone that attracted more airflow inside the diffuser. The D5 model (5% blockage) showed a significant match between the computational and experimental results, with the highest velocity ratio. The D30 model was chosen for further wind turbine tests owing to its practical considerations.

The DAWT design achieved a 70-75% power enhancement compared to a bare SWT. The optimal TSR shifted from $\lambda \approx 3.8$ for the bare turbine to $\lambda \approx 4.5$ for the DAWT. However, the maximum power coefficient ($C_p = 0.37$) was lower than expected, possibly because of the scale effects and manufacturing limitations. Researchers have used 3D printing for rapid prototyping, which allows for quick modifications but introduces some surface roughness issues that may have affected performance (Olasek, et al., 2016).

3.10 Case Study 10

Masukume et al. (2016) focused on determining the optimal dimensional parameters for maximum wind velocity enhancement in diffusers without placing wind turbines. During the performance analysis, they considered the diverging angle (θ) and the nondimensional ratio of the length and diameter. Figure 14 shows the experimental setup used for testing conical diffusers with varying L/D ratios (0.5–3) and expansion angles (1° – 24°) at a fixed wind speed of 3 m/s. Flow speed measurements were performed using an L-type pitot tube along the central diffuser axis, and further calculations were performed to obtain the velocity enhancement.



Figure 14. Experimental setup (Masukume et al., 2016)

The study found that the wind speed is improved with the expansion angle up to a certain limit, after which it decreases. Each length-to-diameter ratio depends on a unique angle value for improved velocity augmentation. The maximum value of augmentation improved with the length-to-diameter ratio, from 1.48 to 1.55 for various ratios, showing a 4.7% increase. The optimal expansion angle decreases as the length-to-diameter ratio increases, from 14.5° to 3.5° . The research determined that optimizing the dimensional parameters of conical diffusers can significantly enhance wind velocity. However, the study notes that while larger L/D ratios provide greater wind speed gains, they also increase costs and reduce responsiveness to wind direction changes owing to increased inertia. Therefore, a balance must be achieved between wind speed augmentation and system practicality when designing diffuser-augmented wind-turbine systems (Masukume et al., 2016).

3.11 Case Study 11

Masukume et al. (2018) extended their previous research and conducted field tests on an optimized conical diverging DAWT. The optimized conical-diverging diffuser was manufactured with an L/D of 0.5 and $\alpha = 14.5^\circ$. This study determined the optimization of the dimensional parameters of conical diffusers.

Figure 15 shows the field-testing setup in which one Dolphin Z 300-WB SWT was placed without a diffuser and the same model in the optimized diffuser. Data acquisition was used to record the various input and output parameters.

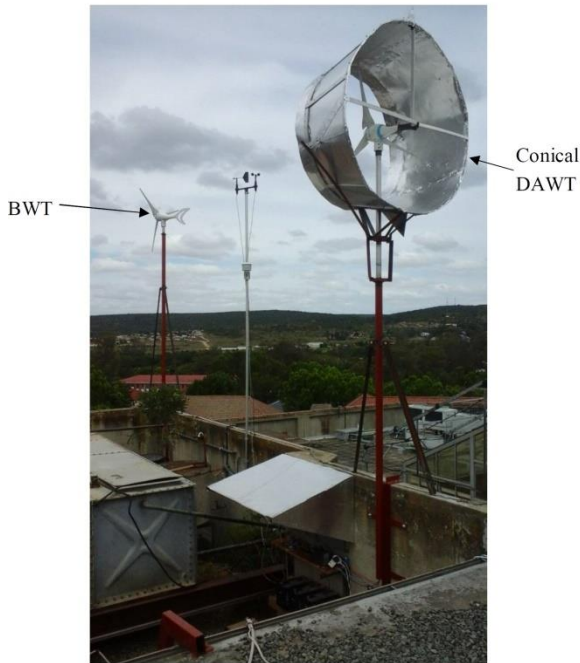


Figure 15. Field testing setup (Masukume et al., 2018)

The results indicated that the DAWT started at a lower cut-in speed of 1.6 m/s whereas the BWT starts at 2.5 m/s. This indicates the proven ability of the developed diffuser to improve the wind velocity. The DAWT power was considerably higher than that of the BWT. Figure 16 shows the power curve obtained by the (Masukume et al., 2018).

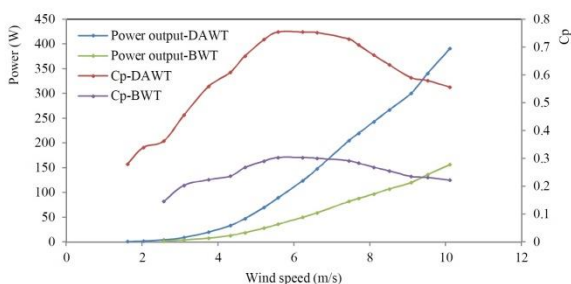


Figure 16. Power output curve (Masukume et al., 2018)

3.12 Case Study 12

Pambudi et al. (2019) in their study investigated the performance of a horizontal axis SWT with different diffuser geometries at wind speeds of 1 to 5 m/s. This study examined two conical diffuser models with and without an inlet nozzle or shroud. Wind tunnel experiments were performed by placing a wind turbine inside the diffuser models at the wind speeds stated earlier.

It is observed that for the diffuser without a nozzle and with a nozzle, the wind speeds are increased by 10% and 13.3%, respectively. The BWT and DAWT using a diffuser without a nozzle produced 9.7240 and 13.6523 W power, respectively, at selected higher speed values, as mentioned earlier. SWT using a diffuser without a nozzle and with a nozzle produced 81.7% and 151.3% power, respectively (Pambudi et al., 2019).

3.13 Case Study 13

Takey et al. (2020) investigated the performance of different diffuser configurations, including a conical-diverging diffuser, using a computational technique. They attempted to replace a conventional conical diverging flanged diffuser with a new design concept that uses a circular flange ring as an alternative to a conventional diffuser. Figure 17 illustrates the research framework used by the authors. They used an HWAT with a rotor of 0.6 m diameter. As shown in the figure, they examined the turbine performance in three configurations: flanged diffuser, flangeless diffuser, and flange. During the CFD process, the upstream wind velocity was fixed at 5 m/s, and the rotor speed was varied from 200 to 1100 rpm.

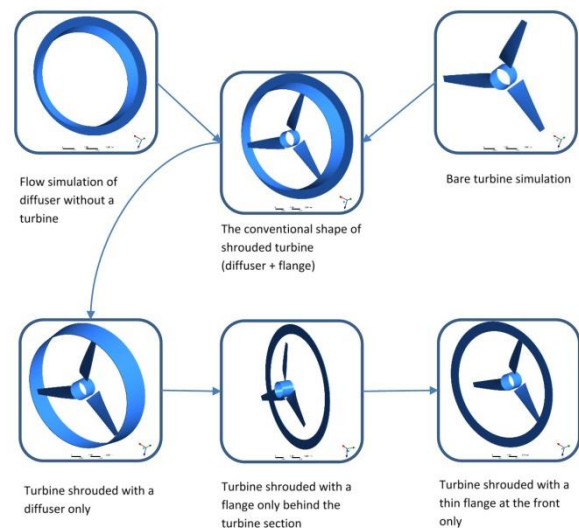


Figure 17. Research framework of (Takey et al., 2020)

They reported important findings from their research. The flanged diffuser configuration significantly improved the turbine performance, increasing the torque by up to 107% compared to the bare turbine. The flange played a crucial role in the performance enhancement, as its removal resulted in negligible improvement. A new configuration with only a ringed flange placed in front of the turbine showed a 33% performance improvement over a bare turbine (Takey et al., 2020).

3.14 Case Study 14

Mohan et al. (2021) studied the use of a conical converging nozzle and conical diverging diffusers to improve the free stream wind velocity and the resulting

power output of the SWT. They termed the nozzle the concentrator in their research paper. In their study They focused on conical sections with a 1-meter length and 1-meter diameter at the larger end. For diffusers, divergence angles from 2° to 16° were tested, whereas for concentrators, convergence angles in the same range were examined. They performed detailed CFD simulations. The inlet velocities varied from 1 to 15 m/s during the simulated analysis using ANSYS Fluent software. Furthermore, they carried out a trend analysis and derived empirical equations. Figure 18 shows the prototype developed by Mohanan et al. (2021).

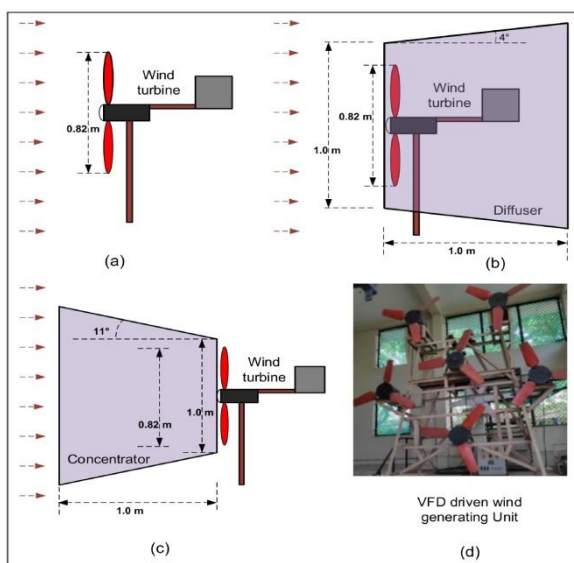


Figure 18. Experimental prototype (Mohanan et al., 2021)

Based on computational and experimental research, they identified several important outputs. A diffuser with an angle of 4.5° showed an air speed improvement of 23.3%. The same diffuser increased the power output by 65.1% at a divergence angle of 3.6° . The concentrator mode was less effective, with a maximum velocity improvement of 9.8% at a convergence angle of 11.5° . Empirical equations were derived to calculate the velocity and power variations based on convergence/divergence angles (Mohanan et al., 2021).

3.15 Case Study 15

Rahmatian et al. (2023) mainly focused on the use of various statistical tools for the analysis of diffusers with flanges and diffusers with nozzles. These include the Design of Experiments for selecting the experiments to be performed. They used the Response Surface Method (RSM) and Genetic Algorithm (GA). Here, focused details about diverging diffusers are added. In this study, this diffuser was called the primary duct. In the current review, it is discussed as a primary diffuser. The primary diffuser was studied using computational fluid dynamics (CFD) to understand its effect on wind speed. The results were confirmed by experimentation, showing an average difference of less than 2%, to

confirm the correctness of the computational model. At an inlet of 5 m/s, the maximum velocity at its throat was observed as 7.9 m/s. When the primary diffuser was fitted around a small HAWT, turbine performance was significantly enhanced. The presence of the primary diffuser increased the turbine output torque by up to 1.71 times compared to the BWT. Therefore, the C_{pmax} increased from 0.33 (BWT) 0.68. The addition of a diffuser enhanced the optimum operation of the turbine. The C_{pmax} for the DAWT using the primary diffuser observed at a TSR of 3.7, compared to 3.1 BWT. Figure 19 shows that the diffuser broke up the helical vortices that formed behind the rotor. This effect reduces the aerodynamic forces and noise levels, making it feasible to install an auxiliary rotor or other structure behind the main turbine (Rahmatian et al., 2023).

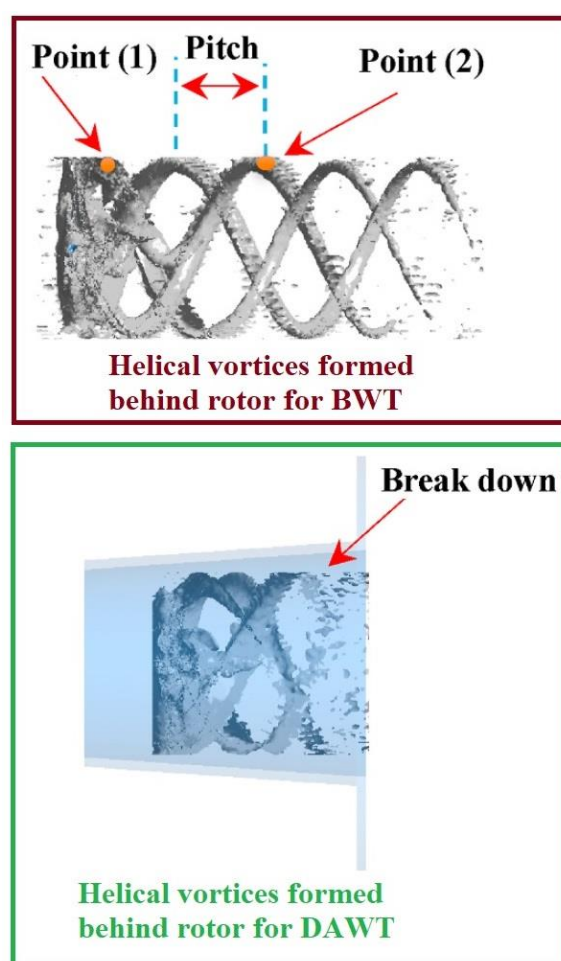


Figure 19. Wake formation for BWT and DAWT (Rahmatian et al., 2023).

3.16 Case Study 16

Jauhar et al. (2023) performed studies on conical diverging diffusers for wind turbines with variable input parameters, including the divergence angle and flange height. This study was performed for a rotor with a diameter of 1 m. The divergence angles were considered to be 2° , 4° , and 7° , and the optimized flange height was 0.25 m. These models were tested computationally at

inlet velocities of 2–10 m/s (in 2 m/s increments). Furthermore, these results were validated by previous results.

They observed that the maximum increase in velocity was reached at α a divergence angle of $= 4^\circ$. The optimal location for turbine placement varies with the inlet velocity and divergence angle. For $\alpha = 2^\circ$, the rotor should be positioned at 35.6% of length, which increases to 36.6% for $\alpha = 4^\circ$. The diffuser was more effective at lower free-stream velocities (Jauhar et al., 2023).

3.17 Case Study 17

İlhan et al. (2024) presented a study on the design parameters of conical diverging diffusers using CFD. The study used the BEM theory and k- ϵ turbulence model to analyze 17 different DAWTs. Figure 20 shows two primary configurations of the linear shrouding profiles: one without a nozzle and the other with a nozzle at entry.

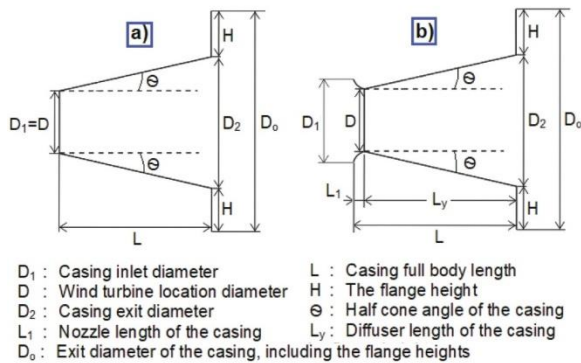


Figure 20. Diffuser models studied (İlhan et al., 2024)

The study analyzed four cases (Cases 1-4) with h/D ratios of 0, 0.2, 0.3, and 0.4, while maintaining L/D at 1.40 and $\alpha = 9^\circ$. The addition of a flange significantly increased the wind speed. The maximum wind speed enhancement was observed at $h/D = 0.3$, which reached a value of 1.5. In addition, the maximum pressure drop occurred at $h/D = 0.3$, with a dimensionless pressure coefficient of -1.284.

Six different half-cone angles (3° , 4° , 5° , 6° , 9° , and 12°) were analyzed (Cases 5-10) with $L/D = 1.35$ and $h/D = 0.20$. The highest performance was achieved with a half-cone angle of 5° , which resulted in a maximum wind speed ratio of 1.567 and a maximum pressure drop coefficient of -1.456.

Five cases (Cases 11-15) with L/D of 1.30, 1.35, 1.40, 1.45, and 1.50 were examined. The influence of casing length was found to be less significant. However, the longest casing ($L/D = 1.50$) yielded the highest maximum wind speed ratio of 1.584 and highest pressure drop ratio of -1.509.

This study compared a case without an inlet shroud (Case 16) to one with an optimally designed inlet shroud (Case 17). The addition of a properly designed inlet shroud results in a significant increase in performance. This increased the maximum wind speed enhancement by approximately 16% (from 1.602 to 1.853) and caused a substantial pressure drop of 55%. This optimally designed system, featuring an inlet shroud and flange, confirmed the highest wind speed improvement and enhanced wind mass flow rate through the rotor disc (İlhan et al., 2024).

3.18 Case Study 18

Teklemariyem et al. (2024) optimized various geometrical shapes and parameters for a HAWT using CFD. They studied nine diffuser models. However, as per the scope of this review, only two models, A and B, are discussed. Model A was a conical diverging diffuser without a flange, and Model B was a conical diverging diffuser with a flange. Model A was used to analyze the effect of the opening angle (varied from 2° to 22°), and Model B was used to analyze the effect of flange height (0.1 to 0.8 of the throat diameter). They created 3D models in SOLIDWORKS and ANSYS FLUENT software. They used a tetrahedral mesh for discretization and the SST k- ω turbulence model for the analysis. All analyses were performed at 4.5 m/s.

It was observed that the maximum velocity was found when the $\alpha = 8^\circ$ for model A. This configuration resulted in a 46% increase in velocity to 6.57 m/s inside the diffuser. The optimum flange height was found to be 0.3 times the throat diameter, which contributed to a 57.4% velocity increment. This configuration achieved a velocity of 7.083 m/s, which is significantly higher than that of the inlet (Teklemariyem et al., 2024).

4. DISCUSSION

A comprehensive review of 18 case studies on conical diverging diffusers showed a significant potential for enhancing wind turbine performance. The primary principle behind creating a low-pressure region has been studied in detail by many researchers. This can be significantly improved by adding a flange at the exit, which generates strong vortices that draw more air into the diffuser.

An important aspect identified in the reviewed literature is the optimization of the diffuser geometry. Parameters such as the divergence angle, length-to-diameter (L/D) ratio, flange height, and flange inclination are interdependent and important for maximizing the performance.

It is noteworthy that there is an optimum value for the diverging angle to obtain the maximum power output. Beyond a certain limit, flow separation can occur and reduce the diffuser performance. Some studies have also

shown that this optimum angle tends to decrease as the L/D ratio increases. Lipian et al. (2015) found that the flow velocity could be increased up to 1.6 times for angles around 6°, while Masukume et al. (2016) identified an optimal angle of 14.5° for an L/D ratio of 0.5. This indicated that a unique optimum geometry did not exist. However, it must be designed from the specific operational requirements and constraints. During this design, appropriate weights should be assigned to practical considerations such as cost and structural inertia.

The addition of inlet nozzles (or shrouds) has also been reported in some studies. A nozzle can help direct the flow and provide a modest increase in performance, and the flange remains the main parameter for performance enhancement. Kannan et al. (2013) showed that adding a nozzle could increase the wind velocity by 67.12%, while İlhan et al. (2024) found that an optimized inlet shroud boosted the wind speed enhancement by approximately 16% over a flanged-only design.

The methodologies used in these studies were primarily CFD and experimental wind tunnel testing. CFD simulations using models such as k- ϵ , k- ω SST, and non-linear eddy-viscosity models have been used in parametric analysis. The flow visualization, which is useful for exploring a wide range of geometries efficiently. Experimental validations using techniques such as Particle Image Velocimetry (PIV) are important for confirming the accuracy of these computational models.

However, challenges such as wind tunnel blockage effects and inconsistencies between scaled models and full-sized field tests remain issues that require careful consideration in future research.

5. CONCLUSION

This review systematically studied 18 important studies on conical diverging diffusers to enhance wind turbine performance. The important findings from these computational and experimental reviews are presented here.

Conical diverging diffusers are a proven and effective technology for augmenting the wind turbine power output. By enhancing the wind velocity through a rotor, DAWTs can generate significantly more power, ranging from 1.5 to 4 times than of the BWT, and can operate effectively at lower wind speeds.

The addition of a flange is the most important aspect of performance enhancement. This creates a low-pressure wake region that considerably increases the mass flow rate through the turbine.

The performance of a diffuser depends considerably on geometric parameters, including the divergence angle, length-to-diameter ratio, and flange height. There is no single optimal diffuser configuration. The combined use of CFD simulations and experimental testing is important for the diffuser design. CFD allows for wide parametric studies, while wind tunnel and field tests provide the necessary validation and assure the actual wind turbine performance characteristics. To achieve optimum performance, a diffuser should be designed using CFD, a wind tunnel, and field testing before commercialization.

Future work should focus on analyzing wind tunnel blockage effects, scaling from models to full-size wind turbines, and further optimizing the designs. Additionally, research on the aerodynamic interaction between the rotor and diffuser and the impact of the diffuser on wake characteristics and noise reduction will be helpful in making DAWTs a more commercially viable and widely accepted technology.

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. <https://doi.org/10.20508/ijrer.v8i3.7794.g7436>
- Akhter, M. Z., & Omar, F. K. (2021). Review of Flow-Control Devices for Wind-Turbine Performance Enhancement. *Energies*, 14(5), 1268. <https://doi.org/10.3390/en14051268>
- Alaimo, A., Esposito, A., Messineo, A., Orlando, C., & Tumino, D. (2015). 3D CFD Analysis of a Vertical Axis Wind Turbine. *Energies*, 8, 3013-3033. <https://doi.org/10.3390/EN8043013>
- Barbosa, D. L. M., Vaz, D. A. T. D. R., Vaz, J. R. P., Figueiredo, S. W. O., da Silva, M. O., Blanco, C. J. C., & Mesquita, A. L. A. (2013). A Proposed Mathematical Model For The Velocity Profile Internally To A Conical Diffuser. *Revista de Engenharia Térmica*, 12(2), 69. <https://doi.org/10.5380/reterm.v12i2.62050>
- El-Zahaby, A. M., Kabeel, A. E., Elsayed, S. S., & Obiaa, M. F. (2017). CFD analysis of flow fields for shrouded wind turbine's diffuser model with different flange angles. *Alexandria Engineering Journal*, 56(1), 171–179. <https://doi.org/10.1016/j.aej.2016.08.036>

- Garmode, R. K., Gaval, V. R., Kale, S. A., & Nikhade, S. D. (2022). Comprehensive evaluation of materials for small wind turbine blades using various MCDM techniques. *International Journal of Renewable Energy Research*, 12(2), 981-992.
- İlhan, A., Zontul, H., Tümse, S., Bilgili, M., & Şahin, B. (2024). Flow analyses of diffuser augmented wind turbines. *Energy Sources, Part A: Recovery, Utilization and Environmental Effects*, 46(1), 1102–1126. <https://doi.org/10.1080/15567036.2020.1861130>
- Jafari, S. A. H., & Kosasih, B. (2014). 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.2013.12.001>
- Jauhar, T. A., Hussain, M. I., Kiren, T., Arif, W., Miran, S., & Lee, G. H. (2023). Effect of flanged diffuser divergence angle on wind turbine: A numerical investigation. *PLoS ONE*, 18(6 June). <https://doi.org/10.1371/journal.pone.0287053>
- Kannan, T. S., Mutasher, S. A., & Lau, Y. K. (2013). Design and flow velocity simulation of diffuser augmented wind turbine using CFD. *Journal of Engineering Science and Technology*, 8(4), 372-384. <https://www.researchgate.net/publication/265089318>
- Kosasih, B., & Tondelli, A. (2012). Experimental study of shrouded micro-wind turbine. *Procedia Engineering*, 49, 92–98. <https://doi.org/10.1016/j.proeng.2012.10.116>
- Lipian, M., Karczewski, M., & Olasek, K. (2015). Sensitivity study of diffuser angle and brim height parameters for the design of 3 kW Diffuser Augmented Wind Turbine. *Open Engineering*, 5(1), 280–286. <https://doi.org/10.1515/eng-2015-0034>
- Mahalakshmi, N. V., Krithiga, G., Sandhya, S., Vikraman, J., & Ganesan, V. (2007). Experimental investigations of flow through conical diffusers with and without wake type velocity distortions at inlet. *Experimental Thermal and Fluid Science*, 32(1), 133–157. <https://doi.org/10.1016/j.expthermflusci.2007.02.008>
- Masukume, P. M., Makaka, G., & Mukumba, P. (2018). Optimization of the power output of a bare wind turbine by the use of a plain conical diffuser. *Sustainability (Switzerland)*, 10(8). <https://doi.org/10.3390/su10082647>
- Masukume, P.-M., Makaka, G., & Tinarwo, D. (2016). To cite this article: Peace-Maker Masukume, Golden Makaka, David Tinarwo. Optimum Geometrical Shape Parameters for Conical Diffusers in Ducted Wind Turbines. *International Journal of Energy and Power Engineering*, 5(6), 177–181. <https://doi.org/10.11648/j.ijepe.20160506.11>
- Mohanani, J. N., Sundaramoorthy, K., & Sankaran, A. (2021). Performance improvement of a low-power wind turbine using conical sections. *Energies*, 14(17). <https://doi.org/10.3390/en14175233>
- Ohya, Y., Karasudani, T., Sakurai, A., Abe, K. ichi, & Inoue, M. (2008). Development of a shrouded wind turbine with a flanged diffuser. *Journal of Wind Engineering and Industrial Aerodynamics*, 96(5), 524–539. <https://doi.org/10.1016/j.jweia.2008.01.006>
- Olasek, K., Karczewski, M., Lipian, M., Wiklak, P., & Józwik, K. (2016). Wind tunnel experimental investigations of a diffuser augmented wind turbine model. *International Journal of Numerical Methods for Heat and Fluid Flow*, 26(7), 2033–2047. <https://doi.org/10.1108/HFF-06-2015-0246>
- Oliveira, T. D., Tofaneli, L. A., & Santos, A. Á. B. (2025). Aerodynamic optimization of small diffuser Augmented Wind Turbines: A differential evolution approach. *Energy Conversion and Management: X*, 26. <https://doi.org/10.1016/j.ecmx.2025.100891>
- Oman, R. A., & Foreman, K. M. (1973, December). Advantages of the diffuser-augmented wind turbine. In *Wind Energy Conversion Systems: Workshop Proceedings, Washington, DC* (pp. 103-106).
- Kale, S. A., Rady, M. H., Garmode, R. K., & Gooroochurn, M. (2022). Effects of Blade Root Dimensions on Physical and Mechanical Characteristics of a Small Wind Turbine Blade. *International Journal of Renewable Energy Research (IJRER)*, 12(3), 1339-1346.
- Rahmatian, M. A., Nazarian Shahrabaki, A., & Moeini, S. P. (2023). Single-objective optimization design of convergent-divergent ducts of ducted wind turbine using RSM and GA, to increase power coefficient of a small-scale horizontal axis wind turbine. *Energy*, 269. <https://doi.org/10.1016/j.energy.2023.126822>
- Prasad, K. R., Kumar, V. M., Swaminathan, G., & Loganathan, G. B. (2020). Computational investigation and design optimization of a duct augmented wind turbine (DAWT). *Materials Today: Proceedings*, 22, 1186-1191. <https://doi.org/10.1016/j.matpr.2019.12.116>
- Pambudi, N. A., Febriyanto, R., Wibowo, K. M., Setyawan, N. D., Wardani, N. S., Saw, L. H., & Rudiyanto, B. (2019). The performance of shrouded wind turbine at low wind speed condition. *Energy Procedia*, 158, 260-265. <https://doi.org/10.1016/j.egypro.2019.01.086>

- Roshan, S. Z., Alimirzazadeh, S., & Rad, M. (2015). RANS simulations of the stepped duct effect on the performance of ducted wind turbine. *Journal of Wind Engineering and Industrial Aerodynamics*, 145, 270–279. <https://doi.org/10.1016/j.jweia.2015.07.010>
- Kale, S., & Sapali, S. N. (2014, November). Development and field testing of an inclined flanged compact diffuser for a micro wind turbine. In *ASME International Mechanical Engineering Congress and Exposition* (Vol. 46521, p. V06BT07A070). American Society of Mechanical Engineers. <https://doi.org/10.1115/IMECE2014-37883>
- Sridhar, S., Mohammad Zuber, Shenoy B., S., Kumar, A., Ng, E. Y. K., & 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. <https://dx.doi.org/10.1016/j.rser.2022.112316>
- Takey, M., Lazim, T. M., Ishak, I. S., Nik Mohd, N. A. R., & Othman, N. (2020). Computational investigation of a wind turbine shrouded with a circular ring. *CFD Letters*, 12(10), 40–51. <https://doi.org/10.37934/cfdl.12.10.4051>
- Teklemariyam, D. A., Yimer, E. T., Ancha, V. R., & Zeru, B. A. (2024). Parametric study of an empty diffuser geometric parameters and shape for a wind turbine using CFD analysis. *Heliyon*, 10(5). <https://doi.org/10.1016/j.heliyon.2024.e26782>

Mangesh R. Mahajan

Research Scholar, School of
Engineering and Technology, Sandip
University, Nashik, India
mangesh.mahajan@mitwpu.edu.in
ORCID 0000-0003-1269-4663

Sanjay D. Nikhade

School of Engineering and Technology,
Sandip University, Nashik, India
sanjay.nikhade@sandipuniversity.edu.in
ORCID 0000-0001-8110-4140
