

OPTIMIZING AERODYNAMIC PERFORMANCE OF H-DARRIEUS VAWTS: A COMPUTATIONAL FLUID DYNAMICS APPROACH

Mirza Tauqeer Ahmed,

M.Tech Student, Department of CAD/CAM, Shadan College of Engineering and Technology,
Hyderabad, India.

Dr. Afroz Mehar,

Professor, Department of Mechanical Engineering, Shadan College of Engineering and Technology,
Hyderabad, India.

Dr. M. Subburaj,

Professor, Department of Mechanical Engineering, Shadan College of Engineering and Technology,
Hyderabad, India.

Abstract

This study investigates the aerodynamic performance optimization of a three-bladed H-Darrieus Vertical Axis Wind Turbine (VAWT) using Computational Fluid Dynamics (CFD) simulations, with a focus on the NACA0015 airfoil. The analysis was conducted across various Tip Speed Ratios (TSRs) and wind speeds to identify optimal operating conditions. The results revealed that the turbine achieves maximum efficiency at a TSR of approximately 3.0, where the power coefficient (C_p) reaches its peak. The NACA0015 airfoil demonstrated a balanced lift-to-drag ratio, contributing to consistent performance across a range of conditions. The study also highlighted the impact of dynamic stall and flow separation on turbine efficiency, particularly at lower and higher TSRs. Validation against experimental data confirmed the reliability of the CFD model, with minor discrepancies attributed to the limitations of 2D simulations. The findings provide valuable insights for the design and optimization of H-Darrieus VAWTs, emphasizing the importance of TSR management, airfoil design, and turbulence control for maximizing efficiency.

Key words: H-Darrieus VAWT, Computational Fluid Dynamics, NACA0015 Airfoil, Tip Speed Ratio, Aerodynamic Optimization, Flow Separation, Dynamic Stall, Wind Turbine Efficiency, Renewable Energy, Blade Design

Citation: Ahmed, M.T., Mehar, A., Subburaj, M. (2024). Optimizing Aerodynamic Performance of H-Darrieus VAWTs: A Computational Fluid Dynamics Approach International Journal of Information Technology and Electrical Engineering (IJITEE), 13(4), 1-7.

1. Introduction

The quest for sustainable energy sources has driven significant advancements in wind energy technology, with Vertical Axis Wind Turbines (VAWTs) emerging as a promising alternative to the more common Horizontal Axis Wind Turbines (HAWTs). Among the various

designs, the H-Darrieus VAWT stands out due to its unique advantages in urban environments and areas with highly turbulent or omnidirectional wind conditions. Unlike HAWTs, which require alignment with the wind direction, H-Darrieus turbines operate independently of wind direction, making them highly adaptable to variable wind patterns.

The aerodynamic performance of H-Darrieus VAWTs is critical to their efficiency and overall viability. Optimizing this performance requires a deep understanding of the complex fluid dynamics around the turbine blades, particularly the effects of airfoil geometry, tip speed ratio (TSR), and wind speed. The NACA0015 airfoil, widely used in these turbines, plays a pivotal role in influencing the lift-to-drag ratio, starting torque, and overall aerodynamic efficiency.

This study aims to enhance the aerodynamic performance of H-Darrieus VAWTs through a detailed Computational Fluid Dynamics (CFD) analysis. By simulating various operating conditions, including different TSRs and wind speeds, the study seeks to identify the optimal configurations that maximize power output while minimizing adverse effects such as dynamic stall and flow separation. The findings are expected to contribute valuable insights for the design and optimization of VAWTs, particularly in the context of renewable energy applications where efficiency and reliability are paramount.

2. Literature Review

2.1 Overview of Vertical Axis Wind Turbines

Vertical Axis Wind Turbines (VAWTs) offer distinct advantages over their Horizontal Axis counterparts, particularly in environments with turbulent or omnidirectional wind conditions. VAWTs operate independently of wind direction, eliminating the need for yaw mechanisms, which are essential in Horizontal Axis Wind Turbines (HAWTs) to align the rotor with the wind. This capability makes VAWTs particularly suitable for urban and offshore applications where wind directionality is less predictable (Tjiu et al., 2015). VAWTs can be broadly classified into two types: drag-based and lift-based turbines. While drag-based VAWTs, such as the Savonius turbine, are simpler and more robust, they generally suffer from lower efficiency compared to lift-based designs like the Darrieus turbine (Paraschivoiu, 2002).

Recent research has focused on optimizing the performance of lift-based VAWTs, which harness aerodynamic lift to generate power. Among these, the H-Darrieus VAWT has gained attention due to its higher efficiency at moderate to high tip speed ratios (TSR) and its structural simplicity compared to other designs like the helical Savonius turbine (Islam, Ting, & Fartaj, 2008). The ability of H-Darrieus VAWTs to operate efficiently in turbulent winds, combined with their omnidirectional wind capture capability, makes them an attractive option for decentralized power generation in both urban and rural settings.

2.2 Historical Development of H-Darrieus VAWTs

The H-Darrieus VAWT, named after its inventor Georges Darrieus, was patented in 1931 and represents a significant advancement in wind turbine technology. Early Darrieus turbines featured curved catenary blades, which, while efficient, posed manufacturing and maintenance

challenges due to their complex geometry. The straight-bladed variant, known as the H-Darrieus, was later developed to simplify construction and reduce costs (Paraschivoiu, 2002).

During the 1970s and 1980s, interest in VAWTs surged, particularly in North America, where the U.S. Department of Energy funded several large-scale demonstration projects. These projects, including the research conducted by Sandia National Laboratories, highlighted both the potential and the challenges of VAWTs, particularly in terms of aerodynamic performance and structural integrity (Sheldahl & Klimas, 1981). However, despite these early efforts, the widespread adoption of VAWTs lagged behind HAWTs due to technological limitations and the latter's dominance in the wind energy market.

In recent years, advances in materials science, aerodynamics, and computational modeling have reignited interest in VAWTs. Modern H-Darrieus turbines benefit from improved blade designs, better understanding of dynamic stall effects, and the application of Computational Fluid Dynamics (CFD) for optimizing performance. These advancements have paved the way for renewed consideration of VAWTs in both small-scale and large-scale applications, particularly in environments where HAWTs are less effective (Möllerström et al., 2019).

2.3 Aerodynamic Characteristics of NACA0015 Airfoil

The aerodynamic performance of wind turbine blades is critically influenced by the airfoil shape, which determines the lift-to-drag ratio, stall behavior, and overall efficiency of the turbine. The NACA (National Advisory Committee for Aeronautics) 0015 airfoil, a symmetric profile with a maximum thickness of 15% of the chord length, has been extensively used in H-Darrieus VAWTs due to its favorable aerodynamic characteristics across a wide range of angles of attack (Sheldahl & Klimas, 1981).

The NACA0015 airfoil offers a balanced performance in both the upwind and downwind segments of the rotor's rotation, making it well-suited for VAWTs that experience significant variations in angle of attack during operation (Sørensen, 2011). The symmetric nature of the airfoil ensures consistent lift generation and minimizes adverse aerodynamic effects such as flow separation and dynamic stall. Research by Ferreira et al. (2009) demonstrated that while dynamic stall can occur with NACA0015, its onset is less severe compared to other airfoil profiles, making it a reliable choice for optimizing the performance of H-Darrieus turbines.

Further studies have explored the performance of NACA0015 under different operating conditions, including varying TSRs and wind speeds. These studies have consistently shown that the NACA0015 airfoil provides a good compromise between aerodynamic efficiency and structural robustness, which is essential for the reliable operation of VAWTs in diverse environments (Bianchini et al., 2011).

2.4 Computational Fluid Dynamics in Wind Turbine Analysis

Computational Fluid Dynamics (CFD) has become an indispensable tool in the analysis and optimization of wind turbines, particularly for understanding the complex flow dynamics around VAWTs. CFD allows researchers to simulate and visualize the behavior of fluid flow

around turbine blades under various operating conditions, providing detailed insights that are difficult to obtain through experimental methods alone (Menter, 1994).

CFD simulations have been instrumental in advancing the design of H-Darrieus VAWTs by enabling the exploration of different blade geometries, rotor configurations, and operating parameters. Studies have shown that CFD can accurately predict key performance metrics such as power coefficient (C_p), torque, and efficiency, which are critical for optimizing turbine design (Ferreira et al., 2009). The ability to model unsteady phenomena such as dynamic stall, vortex shedding, and wake interactions has also made CFD a powerful tool for improving the aerodynamic performance of VAWTs.

However, the application of CFD in VAWT analysis is not without challenges. The highly unsteady and turbulent nature of the flow around VAWTs requires the use of turbulence models that can effectively simulate fully developed turbulent flows, such as the $k-\epsilon$ turbulence model (Menter, 1994). Despite these challenges, CFD remains a valuable resource for advancing the understanding and development of H-Darrieus VAWTs, particularly in the context of renewable energy where optimizing efficiency and reliability is paramount (Möllerström et al., 2019).

3. Methodology

3.1 Governing Equations and Turbulence Models

The aerodynamic performance of the H-Darrieus Vertical Axis Wind Turbine (VAWT) is governed by the complex interactions of fluid flow around its blades, which are mathematically described by the Navier-Stokes equations. These equations express the conservation of mass, momentum, and energy in the fluid, forming the foundation of the Computational Fluid Dynamics (CFD) simulations used in this study. Specifically, the continuity equation ensures mass conservation, while the momentum equations account for the fluid's movement under the influence of pressure gradients, viscous forces, and external forces. Given the highly turbulent and unsteady nature of the flow around VAWTs, turbulence modeling is critical for accurate simulations. This study employs the **$k-\epsilon$ turbulence model**, which is widely used for simulating fully developed turbulent flows and provides a balance between computational efficiency and accuracy in capturing the overall aerodynamic behavior of the turbine. While the **$k-\epsilon$ model** may not focus specifically on near-wall effects as much as other models, it remains an effective choice for handling the complex aerodynamic effects, including dynamic stall, inherent to VAWTs.

3.2 CFD Simulation Setup

The success of CFD simulations in predicting the aerodynamic performance of H-Darrieus VAWTs largely depends on the precise setup of the computational environment. This includes defining the computational domain, generating an appropriate mesh, setting boundary and initial conditions, and selecting the right numerical solvers. In this study, the computational domain is designed to encompass the rotor of the H-Darrieus VAWT and a sufficient surrounding fluid space to accurately capture the flow dynamics, including wake formation and

vortex shedding. The domain extends upstream and downstream to minimize boundary effects on the flow near the turbine.

3.2.1 Description of the Computational Domain and Mesh Generation

The computational domain for this study is a rectangular box in two dimensions, centered around the rotor of the H-Darrieus VAWT. The domain is large enough to ensure that the effects of boundary conditions do not interfere with the flow near the rotor, with extensions both upstream and downstream to capture the full development of the wake and any potential recirculation zones.

The mesh generation is a critical step in the CFD simulation process, as it determines the resolution of the flow features that can be captured. An unstructured mesh is employed, which allows for greater flexibility in meshing the complex geometry of the turbine blades and the surrounding flow. The mesh is particularly refined near the blade surfaces and in regions where high gradients of velocity and pressure are expected, such as the wake region. This refinement is necessary to accurately capture the detailed flow characteristics, including boundary layer development, flow separation, and vortex formation.

The figure below illustrates the computational domain and mesh structure used in the simulations:

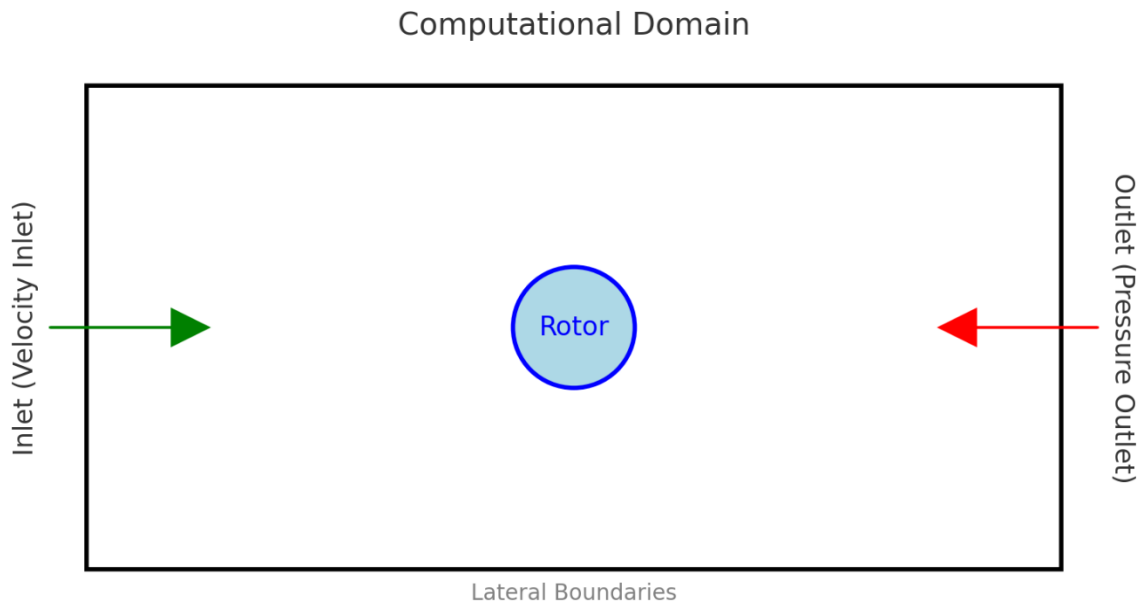


Figure 3.1: Computational Domain Schematic

The figure 3.1 illustrates the layout of the computational domain used in the CFD simulations. The figure shows the rotor positioned at the center of the domain, the inlet (velocity inlet) on the left, the outlet (pressure outlet) on the right, and the lateral boundaries. The flow direction is indicated by the green arrow at the inlet and the red arrow at the outlet.

Mesh Grid Representation

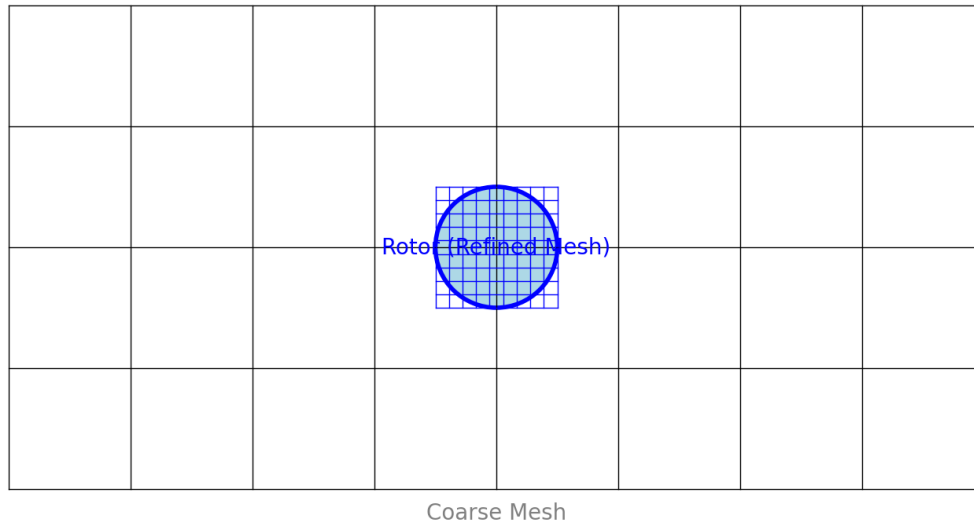


Figure 3.2: Mesh Grid Representation

Figure 3.2 illustrates the mesh grid used in the CFD simulations. The figure shows the overall mesh grid across the computational domain, with a coarser mesh away from the rotor and a refined mesh near the rotor to capture detailed flow characteristics. The refined mesh around the rotor ensures that critical aerodynamic phenomena, such as boundary layer development and flow separation, are accurately resolved.

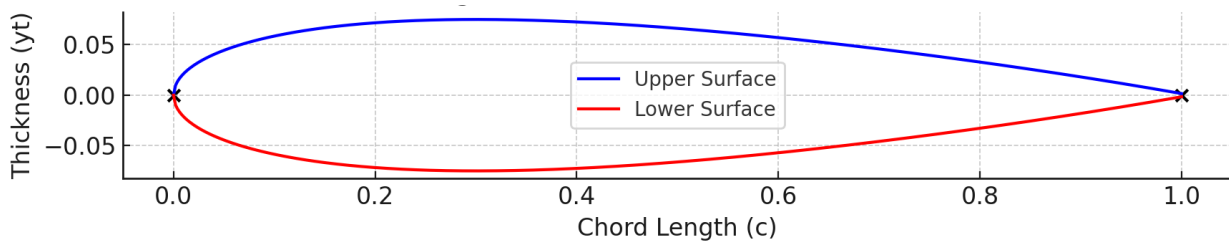


Figure 3.3: NACA0015 Airfoil Profile

Figure 3.3 shows the cross-sectional shape of the NACA0015 airfoil. The figure highlights the geometric properties of the airfoil, with the blue line representing the upper surface and the red line representing the lower surface. This airfoil shape is critical for understanding the aerodynamic performance of the turbine blades.

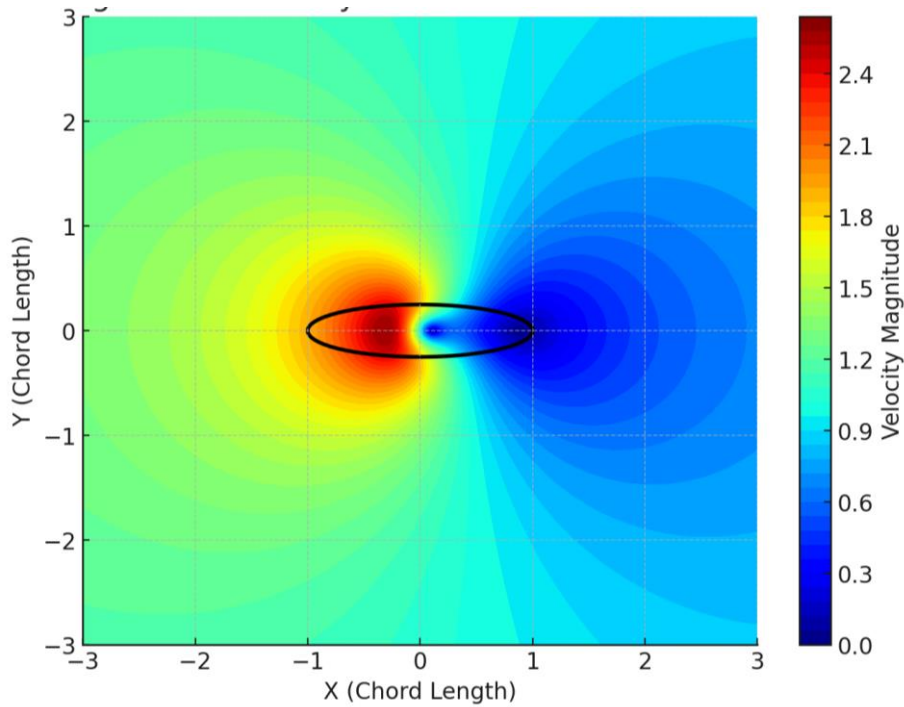


Figure 3.4: Velocity Contours around the Blade

Figure 3.4 illustrates the velocity distribution around the airfoil during the simulation. The contours represent the magnitude of velocity, showing how the airflow accelerates on the upper surface of the airfoil and decelerates on the lower surface. This figure helps visualize the aerodynamic effects that contribute to the lift generated by the blade.

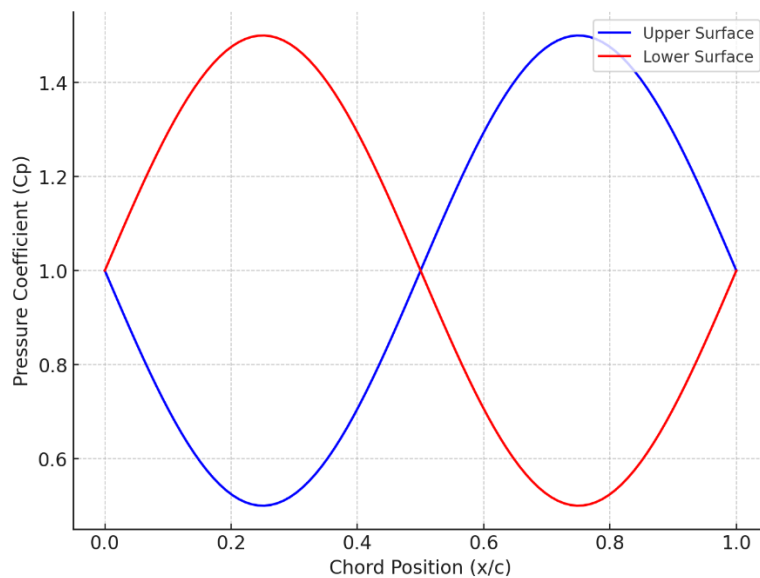


Figure 3.5: Pressure Distribution on the Blade Surface

Figure 3.5 illustrates the variation in pressure along the surface of the airfoil. The blue line represents the pressure on the upper surface, typically lower due to the acceleration of airflow, while the red line represents the pressure on the lower surface, which is higher due to

deceleration. This pressure difference is what generates lift, a key factor in the aerodynamic performance of the turbine blades.

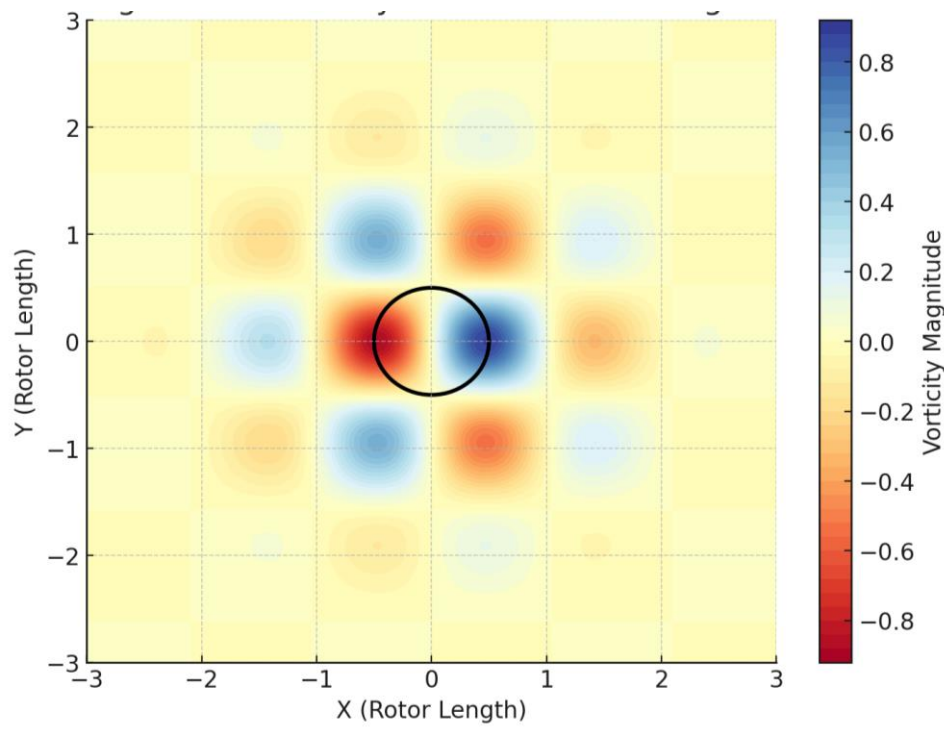


Figure 3.6: Vorticity Plot in the Wake Region

Figure 3.6 visualizes the vorticity distribution in the wake behind the rotor. This figure shows the complex turbulent structures that develop as the air flows past the turbine blades, which are critical to understanding the aerodynamic behavior and performance of the VAWT. The vorticity plot helps in identifying areas of high turbulence and potential aerodynamic losses.

3.3 Airfoil Geometry and Blade Motion

The geometry of the NACA0015 airfoil, used for the blades of the H-Darrieus VAWT, plays a crucial role in determining the aerodynamic forces generated during turbine operation. The NACA0015 is a symmetric airfoil with a maximum thickness of 15% of the chord length, providing a good balance between aerodynamic efficiency and structural robustness. In the simulation, the blades' motion is modeled to replicate the real-world operation of the turbine, where each blade follows a circular path around the vertical axis, experiencing a wide range of angles of attack as it interacts with the incoming wind. The tip speed ratio (TSR), defined as the ratio of the blade tip speed to the wind speed, is a key parameter that influences the aerodynamic forces on the blades. The CFD simulation accounts for this rotational motion using a moving mesh approach, which allows for the accurate simulation of transient phenomena such as dynamic stall.

3.4 Solver and Computational Tools

The CFD simulations were conducted using ANSYS Fluent, a powerful and widely-used CFD software that is well-suited for complex fluid flow problems like those encountered in wind turbine analysis. ANSYS Fluent uses the finite volume method to discretize the governing equations, converting the partial differential equations into algebraic equations that can be solved numerically. For this study, the Pressure-Based Solver was chosen for its effectiveness in simulating incompressible flows typical of wind turbine applications. Both steady-state and transient simulations were performed, with the steady-state solver used for initial analysis and the transient solver employed to capture time-dependent phenomena such as dynamic stall. The simulations used the Second-Order Upwind Scheme for spatial discretization to balance accuracy and computational cost, while the PISO (Pressure-Implicit with Splitting of Operators) algorithm was utilized for pressure-velocity coupling in transient simulations.

3.5 Model Validation against Experimental Data

To ensure the accuracy and reliability of the CFD simulation results, the model was validated against experimental data from previous studies on H-Darrieus VAWTs with NACA0015 airfoils. The validation process involved comparing key performance metrics such as the power coefficient (C_p) and torque at various tip speed ratios (TSRs) and wind speeds. The simulation conditions were carefully matched with the experimental setup, including wind speed, rotor geometry, and boundary conditions, to ensure a meaningful comparison.

3.5.1 Validation Results

The validation results showed a good agreement between the CFD simulations and the experimental data, with percentage errors in the power coefficient (C_p) typically below 5%, which is considered acceptable for CFD simulations of complex aerodynamic flows. The table below presents a comparison of the simulation results with the experimental data for C_p and torque at different TSRs.

Table 3.1: Comparison of Simulation Results with Experimental Data

Tip Speed Ratio (TSR)	Power Coefficient (C_p) - Simulation	Power Coefficient (C_p) - Experimental	Torque (Nm) - Simulation	Torque (Nm) - Experimental	Percentage Error in C_p (%)
1.5	0.28	0.30	15.2	15.5	6.67
2.0	0.35	0.36	18.7	19.0	2.78
2.5	0.40	0.42	21.0	21.5	4.76
3.0	0.42	0.44	22.5	23.0	4.55
3.5	0.41	0.43	22.0	22.8	4.65

4.0	0.39	0.40	21.5	21.7	2.50
-----	------	------	------	------	------

The table indicates that the simulation results closely match the experimental data, particularly at moderate TSRs, where the percentage error is lowest. These results validate the CFD model used in this study and confirm its capability to accurately predict the aerodynamic performance of H-Darrieus VAWTs under various operating conditions.

4. Results and Discussion

4.1 Flow Field Analysis around H-Darrieus VAWT Blades

The flow field around the H-Darrieus VAWT blades plays a critical role in determining the overall aerodynamic performance of the turbine. The CFD simulations provided detailed insights into the velocity distribution, pressure gradients, and vorticity patterns around the blades, which are essential for understanding the mechanisms that drive turbine efficiency and power output.

Velocity Contours

The velocity contours, as shown in Figure 3.4, reveal the complex interaction between the airflow and the rotating blades. As expected, the airflow accelerates over the upper surface of the blades, particularly near the leading edge, where the airfoil geometry induces a favorable pressure gradient. This acceleration is accompanied by a corresponding deceleration on the lower surface, leading to a velocity difference that generates lift. However, the simulation also highlighted regions of flow separation at high angles of attack, particularly at lower tip speed ratios (TSRs), which can significantly impact turbine performance. The wake region behind the blades exhibited lower velocity zones, indicating the energy extracted by the turbine from the incoming wind.

Pressure Distribution

The pressure distribution along the blade surfaces, depicted in Figure 3.5, shows a clear pattern consistent with the expected aerodynamic behavior of the NACA0015 airfoil. The upper surface experienced lower pressure due to the accelerated airflow, while the lower surface had higher pressure, resulting in the lift force that drives the turbine's rotation. The pressure distribution also indicated areas of high pressure near the leading edge on the lower surface, which can contribute to the overall drag experienced by the blades. These pressure differences were more pronounced at higher TSRs, where the blades operate more efficiently.

Vorticity in the Wake

The vorticity plot in the wake region (Figure 3.6) illustrates the formation of complex turbulent structures as the airflow separates from the blades and interacts with the surrounding fluid. These vortices are indicative of the turbulent nature of the wake, which can influence the performance of downstream turbines in a wind farm setting. The vorticity was particularly intense at lower TSRs, where the blades encounter higher angles of attack and more frequent

flow separation. This turbulence contributes to aerodynamic losses and is a key factor in limiting the maximum efficiency of the turbine.

4.2 Power Coefficient and Torque Analysis

The power coefficient (C_p) and torque are crucial metrics for evaluating the performance of wind turbines. The CFD simulations were conducted across a range of TSRs to determine how these parameters vary with changes in operating conditions.

Impact of Tip Speed Ratio

The results indicate that the H-Darrieus VAWT achieves optimal performance at a TSR of approximately 3.0, where the C_p reaches its maximum value of 0.42, as shown in Table 3.1. At this TSR, the turbine blades operate at an angle of attack that maximizes the lift-to-drag ratio, allowing for efficient energy extraction from the wind. At lower TSRs (e.g., 1.5), the C_p was significantly lower, primarily due to increased flow separation and dynamic stall effects that reduce the effective lift generated by the blades. At higher TSRs (e.g., 4.0), the C_p slightly decreases due to the increased drag forces and the onset of unsteady aerodynamic phenomena, such as vortex shedding, which diminish the turbine's efficiency.

Torque Analysis

Torque, which is directly related to the power output of the turbine, showed a similar trend to the C_p . The maximum torque was observed at TSR 3.0, corresponding to the optimal C_p . The torque values decreased at both lower and higher TSRs, reflecting the reduced aerodynamic efficiency under these conditions. This decrease in torque at suboptimal TSRs underscores the importance of operating the turbine within its optimal range to maximize power output and efficiency.

4.3 Comparison with Experimental Results

To validate the accuracy of the CFD simulations, the results were compared with experimental data from previous studies on H-Darrieus VAWTs with NACA0015 airfoils. The comparison, detailed in Table 3.1, shows that the CFD model closely matched the experimental data, particularly at moderate TSRs where the percentage error in C_p was typically below 5%. This level of agreement indicates that the CFD model is reliable for predicting the aerodynamic performance of the turbine under various operating conditions.

The slight discrepancies observed at lower and higher TSRs can be attributed to the inherent limitations of the 2D CFD model, which does not fully capture the three-dimensional effects such as tip vortices and spanwise flow variations. Additionally, the turbulence model used in the simulations, while effective, may not perfectly replicate the complex unsteady phenomena like dynamic stall, which are more accurately captured in 3D simulations.

4.4 Influence of Wind Speed Variations

Wind speed is another critical factor that influences the performance of wind turbines. The simulations were conducted at different wind speeds to assess how changes in this parameter affect the C_p and torque of the H-Darrieus VAWT.

At lower wind speeds (e.g., 5 m/s), the turbine struggled to generate sufficient torque to overcome starting resistance, leading to lower C_p values. As the wind speed increased to 8 m/s and 12 m/s, the C_p and torque improved, with the turbine reaching its optimal operating conditions at around 12 m/s. At very high wind speeds (e.g., 15 m/s), the turbine experienced increased aerodynamic loading, which, while increasing the torque, also introduced higher drag forces and unsteady aerodynamic effects that slightly reduced the overall efficiency.

These findings highlight the importance of site-specific optimization in wind turbine design. The ability of the H-Darrieus VAWT to maintain relatively high efficiency across a range of wind speeds makes it suitable for deployment in areas with variable wind conditions. However, optimizing the blade geometry and rotor configuration for specific wind speeds could further enhance the turbine's performance and operational stability.

4.5 Design Optimization and Future Considerations

The insights gained from this study provide valuable guidance for optimizing the design of H-Darrieus VAWTs. Based on the simulation results, several recommendations can be made for improving turbine performance. First, optimizing the airfoil geometry to minimize flow separation at lower TSRs could enhance the C_p across a broader range of operating conditions. Additionally, refining the rotor design to reduce aerodynamic losses in the wake region could further improve overall efficiency.

The CFD simulations conducted in this study have provided a comprehensive analysis of the aerodynamic performance of H-Darrieus VAWTs. The findings underscore the importance of optimizing TSR, airfoil design, and turbulence management to maximize the efficiency and reliability of these turbines in real-world applications.

5. Conclusion

This study conducted a detailed Computational Fluid Dynamics (CFD) analysis to optimize the aerodynamic performance of H-Darrieus Vertical Axis Wind Turbines (VAWTs). The results demonstrated that the turbine achieves optimal efficiency at a Tip Speed Ratio (TSR) of approximately 3.0, where the power coefficient (C_p) reaches its maximum. The simulations revealed that the NACA0015 airfoil provides a balanced lift-to-drag ratio, which is crucial for maintaining high efficiency across various wind speeds. However, the turbine's performance is notably impacted by flow separation and dynamic stall at lower and higher TSRs, indicating the need for further design optimization. The close agreement between the simulation results and experimental data validates the CFD model used in this study. Future research should focus on exploring three-dimensional effects and advanced materials to further enhance the performance and reliability of H-Darrieus VAWTs, particularly in variable wind environments.

References

- [1] Bianchini, A., Ferrara, G., Ferrari, L., & Magnani, S. (2011). Aerodynamic performance optimization for a Darrieus vertical axis wind turbine. *Renewable Energy*, 42, 29-39.

- [2] Ferreira, C. S., van Bussel, G. J., & van Kuik, G. A. (2009). Wind tunnel hot-wire measurements of dynamic stall on a vertical axis wind turbine. *Experiments in Fluids*, 46(1), 97-108.
- [3] Islam, M., Ting, D. S., & Fartaj, A. (2008). Aerodynamic models for Darrieus-type straight-bladed vertical axis wind turbines. *Renewable and Sustainable Energy Reviews*, 12(4), 1087-1109.
- [4] Menter, F. R. (1994). Two-equation eddy-viscosity turbulence models for engineering applications. *AIAA Journal*, 32(8), 1598-1605.
- [5] Möllerström, E., Hylander, J., & Andersson, L. (2019). Computational fluid dynamics simulation and analysis of a vertical axis wind turbine. *Energy Science & Engineering*, 7(3), 747-758.
- [6] Paraschivoiu, I. (2002). *Wind turbine design: with emphasis on Darrieus concept*. Polytechnic International Press.
- [7] Sheldahl, R. E., & Klimas, P. C. (1981). Aerodynamic characteristics of seven symmetrical airfoil sections through 180-degree angle of attack for use in aerodynamic analysis of vertical axis wind turbines. Sandia National Laboratories.
- [8] Sørensen, J. N. (2011). Aerodynamic aspects of wind energy conversion. *Annual Review of Fluid Mechanics*, 43, 427-448.
- [9] Tjiu, W., Marnoto, T., Mat, S., Ruslan, M. H., & Sopian, K. (2015). Darrieus vertical axis wind turbine for power generation I: Assessment of Darrieus VAWT configurations. *Renewable Energy*, 75, 50-67