

CFD-Based Aerodynamic Optimization of Vehicle-Mounted Axial Wind Turbine Blades

Rufinus Nainggolan*, Husin Ibrahim, Prisca Caesa Moneteringtyas, Suadi, Baringin Sibarani, Soni Hestukoro, Abdul Razak

Politeknik Negeri Medan, Indonesia

Email: rufinus1963ng@gmail.com*, husin.19611018@polmed.ac.id,
priscacaesa@polmed.ac.id, suadisunggal@gmail.com, baringinsibarani69@gmail.com,
sony_hess@yahoo.com, abdulrazak@polmed.ac.id

Keywords:

Wind Turbine; Axial Energy;
Blades; Parametric Optimization.

Abstract

The increasing demand for sustainable energy solutions has encouraged the development of renewable energy technologies that can be integrated into various applications, including transportation systems. Wind energy generated from vehicle-induced airflow represents an alternative energy source that remains underutilized due to complex aerodynamic characteristics and unstable flow conditions. This study aimed to optimize the aerodynamic performance of vehicle-mounted axial wind turbine blades by determining the most suitable angle of attack using Computational Fluid Dynamics (CFD) simulations. The research method involved numerical analysis using CFD software to evaluate the aerodynamic behavior of turbine blades under different angle-of-attack variations of 0°, 5°, 10°, 15°, and 20°. Performance evaluation was conducted based on the lift-to-drag ratio and velocity contour distribution to identify the optimal blade configuration. The results showed that the angle of attack significantly affected aerodynamic performance, with the 10° configuration producing the highest lift-to-drag ratio and the most favorable airflow characteristics. Higher angles of attack resulted in increased flow separation and aerodynamic losses, whereas lower angles produced insufficient lift generation. The optimized turbine design consisted of six blades with a rotor diameter of 0.35 m, demonstrating the potential application of vehicle-induced airflow energy harvesting systems. In conclusion, CFD-based optimization provided an effective approach for improving the performance of vehicle-mounted wind turbine blades and supported future development of renewable energy technologies for sustainable transportation applications.

INTRODUCTION

The increasing global energy demand has accelerated the development of efficient and sustainable renewable energy technologies (Emmanuel & Ngabala, 2026; Gyamfi et al., 2025; Kumar & Rathore, 2023; Mohamed et al., 2024; Mohsin et al., 2022; Paramati et al., 2022). Among various renewable energy sources, wind energy possesses significant potential

due to its wide availability and adaptability across different scales of power generation systems (Ahmed et al., 2024; Arabzadeh & Frank, 2024; Desalegn et al., 2022; Medina et al., 2022; Roy et al., 2022). In recent years, wind turbine technology has continuously evolved through aerodynamic and structural optimization to improve energy conversion efficiency (Firoozi et al., 2024). In this context, parameters such as blade geometry, angle of attack, pressure distribution, and fluid flow characteristics are recognized as critical factors influencing wind turbine aerodynamic performance (Mansi & Aydin, 2022).

Along with advancements in computational technologies, Computational Fluid Dynamics (CFD) has become one of the most widely adopted approaches for analyzing wind turbine aerodynamic characteristics in a detailed and efficient manner (Rajamohan et al., 2022). Numerous studies have investigated the influence of blade geometry and angle-of-attack variations on the lift and drag characteristics of wind turbines (Zhao et al., 2022). However, most existing studies have primarily focused on stationary wind turbines operating under relatively stable open-environment airflow conditions.

Conversely, airflow generated by moving vehicles presents considerable potential as an alternative small-scale energy source that remains largely underutilized (Bouhentala et al., 2025; Das et al., 2025; Moreno-Pacheco et al., 2024). The airflow characteristics surrounding moving vehicles are inherently dynamic and turbulent, requiring more comprehensive aerodynamic investigations (Douvi & Douvi, 2023). Although several studies have explored vehicle-based wind turbine concepts, investigations concerning the aerodynamic optimization of axial wind turbine blades utilizing vehicle-induced airflow through CFD approaches remain limited, particularly regarding the effects of angle-of-attack variations on lift and drag characteristics, pressure distribution, and flow behavior.

The development of Computational Fluid Dynamics (CFD) has provided significant opportunities for analyzing complex aerodynamic phenomena in wind turbine systems. CFD enables researchers to simulate airflow characteristics, pressure distribution, velocity contours, turbulence behavior, and aerodynamic forces with high accuracy while reducing experimental costs. Studies conducted by Rajamohan et al. (2022) and Zhao et al. (2022) demonstrated that CFD approaches are effective tools for evaluating the influence of blade geometry and angle-of-attack variations on wind turbine performance. Through numerical simulations, researchers can identify optimal blade configurations and improve turbine efficiency under various operating conditions. However, most CFD-based wind turbine studies have focused on conventional stationary turbines operating under relatively stable atmospheric wind conditions.

Despite significant progress in stationary wind turbine research, the utilization of airflow generated by moving vehicles remains an emerging and relatively unexplored research area. Transportation systems, particularly large vehicles such as trucks, generate substantial airflow during operation that contains kinetic energy potential. This airflow has traditionally been considered a source of aerodynamic drag rather than a potential renewable energy resource. However, with appropriate turbine design and aerodynamic optimization, vehicle-induced airflow could potentially be harvested to generate additional electrical energy for auxiliary vehicle systems. The aerodynamic characteristics surrounding moving vehicles are complex due to turbulence, fluctuating velocity, vehicle geometry, and environmental

conditions, making the development of vehicle-mounted wind turbines a challenging engineering problem (Douvi & Douvi, 2023).

The primary challenge in developing vehicle-mounted axial wind turbines lies in achieving an optimal blade configuration capable of efficiently extracting energy from unstable airflow conditions. Unlike conventional wind turbines that operate under predictable wind directions and speeds, vehicle-mounted turbines experience rapidly changing airflow caused by vehicle speed variations, road conditions, and surrounding aerodynamic disturbances. Therefore, blade parameters such as the angle of attack must be carefully optimized to achieve an appropriate balance between lift generation and drag reduction. Existing research has investigated aerodynamic optimization of wind turbine blades; however, limited studies have examined axial wind turbines mounted on moving vehicles using CFD simulations to evaluate the relationship between angle-of-attack variations, lift-to-drag ratio, and airflow characteristics.

Several previous studies have investigated aerodynamic optimization methods for wind turbine blades. Zhao et al. (2022) examined aerodynamic performance variations of wind turbine blades under different angles of attack using CFD analysis and demonstrated that aerodynamic efficiency changes significantly depending on blade orientation. Similarly, Rajamohan et al. (2022) applied CFD-based optimization methods to horizontal-axis wind turbines and highlighted the importance of numerical simulations in improving blade performance. Afshari et al. (2023) combined CFD analysis with experimental investigations to evaluate airflow characteristics around NACA airfoil blades under different wind speeds and blade angles. Furthermore, Kasmaiee et al. (2024) investigated unsteady CFD simulations of rotor blades under varying wind conditions, emphasizing the importance of understanding dynamic airflow effects. Although these studies provide valuable insights into aerodynamic optimization, they primarily focus on stationary wind turbines rather than vehicle-mounted turbine applications.

Based on previous studies, a research gap can be identified regarding the limited understanding of aerodynamic optimization strategies for vehicle-mounted axial wind turbine blades operating under vehicle-induced airflow conditions. Existing studies have not sufficiently addressed how variations in blade angle of attack influence aerodynamic efficiency when turbines are exposed to airflow generated by moving vehicles. Furthermore, limited research has investigated the relationship between CFD-generated velocity contours, lift-to-drag ratio, and optimal blade configurations specifically for mobile wind energy harvesting applications. Therefore, further investigation is required to develop an aerodynamic optimization framework suitable for vehicle-mounted wind turbines that can operate effectively under dynamic airflow conditions.

The urgency of this research is associated with the increasing demand for innovative renewable energy solutions within the transportation sector. Transportation remains one of the largest contributors to global energy consumption and carbon emissions, creating a need for alternative technologies that improve energy efficiency. Vehicle-mounted wind turbines represent a potential approach for utilizing previously wasted aerodynamic energy generated during vehicle operation. Optimizing turbine blade performance is essential to ensure that harvested energy exceeds aerodynamic losses and provides practical benefits for vehicle auxiliary systems, such as battery charging, lighting, sensors, and communication equipment.

Therefore, research focusing on the aerodynamic optimization of vehicle-mounted turbines contributes to broader efforts toward sustainable transportation and renewable energy integration.

The novelty of this study lies in applying a CFD-based aerodynamic optimization approach specifically for vehicle-mounted axial wind turbine blades by evaluating different blade angles of attack to determine the most efficient configuration. Unlike previous studies that have mainly focused on stationary turbines, this research investigates the aerodynamic behavior of turbine blades designed to utilize airflow generated by moving vehicles. The study evaluates aerodynamic performance based on lift-to-drag ratio and velocity distribution characteristics to identify the optimal blade configuration. The simulation evaluates angle-of-attack variations of 0° , 5° , 10° , 15° , and 20° to determine the configuration that produces the highest aerodynamic efficiency.

Therefore, this research aims to optimize the aerodynamic performance of vehicle-mounted axial wind turbine blades using CFD simulations by analyzing the influence of angle-of-attack variations on airflow characteristics and lift-to-drag ratio. The research contributes theoretically by expanding knowledge regarding aerodynamic optimization of mobile wind energy harvesting systems and practically by providing a foundation for developing efficient vehicle-mounted renewable energy devices. The findings are expected to support future prototype development and experimental validation under real operating conditions. The optimized turbine design may provide opportunities for utilizing vehicle-induced airflow as an alternative energy source, contributing to sustainable transportation technologies and reducing dependence on conventional electrical energy sources.

METHOD

The realization process of the proposed wind turbine design followed the research flowchart presented in Figure 5. First, the turbine specifications were determined, followed by the selection of an appropriate blade type for the wind turbine design. The selected blade design was then optimized through simulations using the Computational Fluid Dynamics (CFD) FLUENT software package.

Based on the simulation results, the optimal blade dimensions for the designed wind turbine were determined. Subsequently, the dimensions of other turbine components, including the shaft, gears, casing, and supporting components, were established. After completing the component design, the parts were assembled into a functional wind turbine prototype for testing. The completed wind turbine was then tested to evaluate its performance, and the obtained data were analyzed to draw conclusions regarding the design effectiveness. Through this process, a movable wind turbine prototype was successfully developed.

The blades of the movable wind turbine were manufactured from wood and consisted of six blades. The blades were covered with aluminum sheet metal to improve structural protection. The blade diameter was 0.35 m. The turbine developed in this study was classified as an axial-type movable wind turbine.

RESULTS AND DISCUSSION

For optimazing the turbine blades in this research, the variables used are the angles of attack which will result in the highest lift/drag ratio. While other test parameters such as air pressure, wind speed and others are held constant. In this research the angles of attack are used ranging from (0, 5, 10, 15, and 20) degrees. To obtain the optimum size of blade is used package program Computational Fluid Dynamic “FLUENT”.

The data of the tested results are shown in the figure 6 up to figure 11. The research result shows that the maximum wind speed that can be achieved by the wind turbine is 30.7 m/s on the wind turbine truck cruising speed of 80 km/h.

Results of the parametric optimization of blade are shown in figure 6. The angle of attack which gives the optimum size of blade is ten degrees. Consideration of the best angle of attack is based on the biggest ratio of the lift/drag force.

Contours of velocity magnitude for angles of attack 0, 5, 10, 15, and 20, degrees respectively, are shown in the figure 1 up to figure 5.

In inspecting the images which are shown in figure 3 up to figure 5, we observed that the best contours of velocity magnitude is figure 3 which give the angle of attack equal to ten degrees.

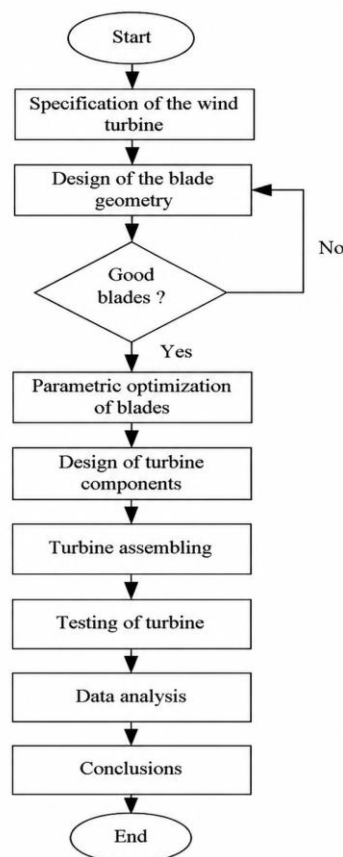


Figure 1. Mobile Wind Turbine Design Process Flow Diagram

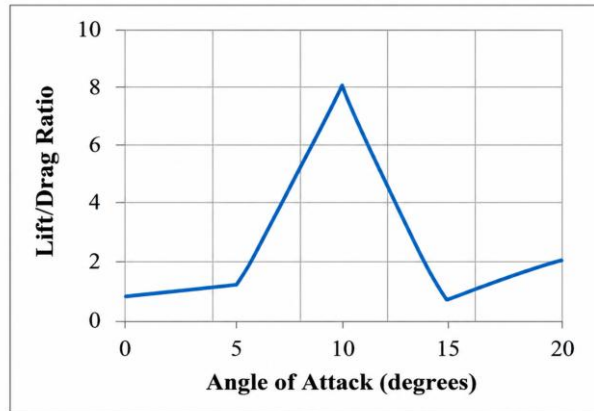


Figure 2. Relation between Lift/Drag ratio versus angle of attack of turbine wind blades

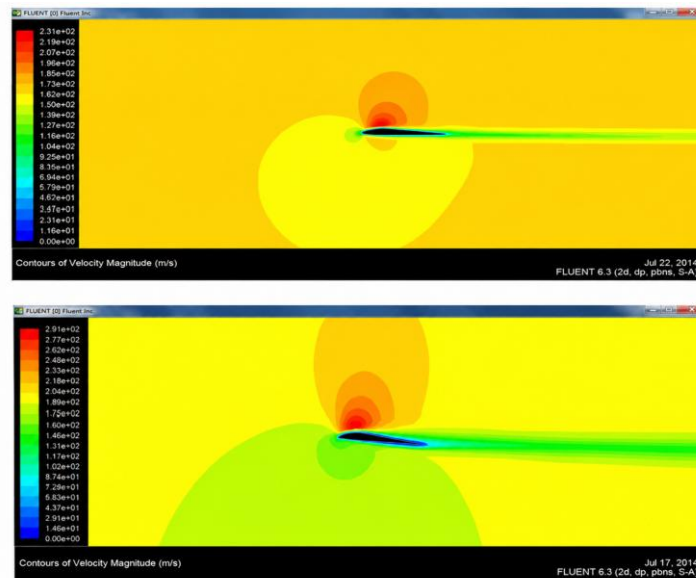


Figure 3. Contours of velocity magnitude for turbine wind blade at different angles of attack (0° and 5°)

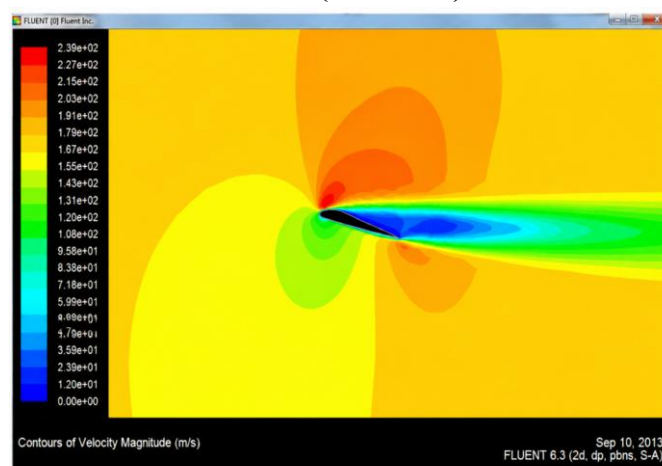


Figure 4. Contours of velocity magnitude for turbine wind blade at an angle of attack of 15°

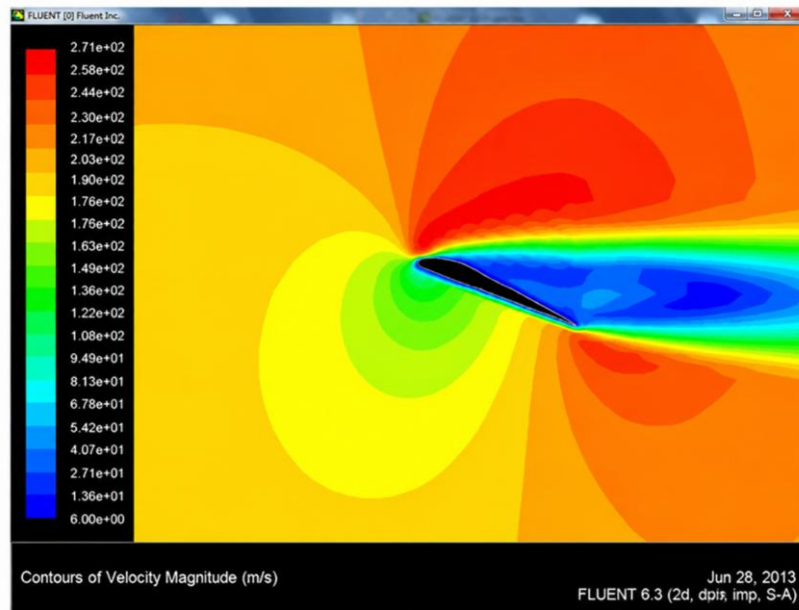


Figure 5. Contours of velocity magnitude for turbine wind blade at an angle of attack of 20°

Lift-to-Drag Ratio Analysis

Based on the CFD simulation results presented in Figure 6, the aerodynamic performance of the blade is strongly affected by the angle of attack. The lift-to-drag ratio (L/D), which is commonly used as an indicator of aerodynamic efficiency, shows a significant variation as the angle of attack changes from 0° to 20°. An increase in the angle of attack from 0° to 10° results in a progressive improvement in the L/D ratio, indicating a more favorable aerodynamic condition for energy extraction.

At low angles of attack, namely 0° and 5°, the airflow interacts with the blade surface without generating a substantial pressure difference between the upper and lower surfaces of the airfoil. Consequently, the generated lift force remains relatively low, resulting in a lower lift-to-drag ratio. Although drag force is also relatively small in this region, the aerodynamic efficiency of the blade is still limited due to insufficient lift generation.

The highest lift-to-drag ratio was obtained at an angle of attack of 10°, indicating the optimum aerodynamic configuration among all tested cases. This result suggests that the blade geometry at this angle is capable of producing a favorable balance between lift generation and drag reduction. The increased pressure difference around the blade surface contributes to higher lift forces while maintaining aerodynamic stability. Therefore, the 10° configuration provides the most effective condition for converting the kinetic energy of airflow into useful rotational energy.

When the angle of attack was further increased to 15° and 20°, the lift-to-drag ratio decreased. This reduction indicates that the increase in drag force becomes more dominant than the increase in lift force. Higher blade inclination tends to disturb the airflow around the blade, reducing aerodynamic efficiency. Similar trends have been reported in aerodynamic studies of wind turbine airfoils, where excessive angles of attack result in increased flow disturbances and reduced aerodynamic performance. Therefore, the CFD results confirm that

an angle of attack of 10° represents the optimum operating condition for the proposed vehicle-mounted axial wind turbine blade.

Velocity Contour Analysis

The velocity contour distributions presented in Figures 7–11 provide further insight into the aerodynamic behavior of the blade under different angle-of-attack conditions. The contour plots illustrate the interaction between the incoming airflow and the blade surface, allowing the evaluation of flow acceleration and potential flow disturbances around the blade.

For the 0° angle of attack, the velocity distribution remains relatively uniform around the blade profile. Only a limited acceleration of airflow can be observed on the upper surface of the blade, indicating a relatively small aerodynamic loading. As a result, the generated lift force is expected to be lower than that of the other tested configurations.

At an angle of attack of 5° , the velocity contours indicate a stronger acceleration region over the blade surface. This condition suggests an increase in the pressure difference between the upper and lower surfaces of the blade, leading to improved lift generation. However, the aerodynamic performance has not yet reached its optimum level.

The most favorable velocity distribution is observed at an angle of attack of 10° . The airflow accelerates smoothly along the blade surface and maintains a relatively stable flow pattern around the airfoil. Compared with the other configurations, the velocity contour at this angle exhibits the most balanced flow behavior, which is consistent with the highest lift-to-drag ratio obtained from the CFD simulations. This result indicates that the blade operates under a highly efficient aerodynamic condition.

For the 15° and 20° angle-of-attack configurations, the velocity contours exhibit increasingly complex flow patterns. Localized low-velocity regions and flow disturbances become more noticeable near the downstream side of the blade. These characteristics suggest the possibility of flow separation and increased turbulence intensity, which contribute to additional aerodynamic losses. Consequently, the aerodynamic efficiency decreases despite the larger blade inclination.

Overall, the velocity contour analysis supports the lift-to-drag ratio results and confirms that the 10° angle of attack provides the most favorable aerodynamic characteristics among the investigated configurations.

Engineering Implications for Vehicle-Mounted Applications

The findings obtained in this study have important implications for the development of vehicle-mounted wind energy harvesting systems. Unlike conventional wind turbines that operate in relatively stable atmospheric conditions, the proposed turbine utilizes airflow generated by the motion of a moving vehicle. Therefore, achieving an optimal blade configuration is essential to maximize energy extraction while maintaining aerodynamic efficiency.

The identification of a 10° optimum angle of attack demonstrates that proper blade design can significantly improve the capability of the turbine to convert vehicle-induced airflow into mechanical energy. A higher lift-to-drag ratio allows more effective utilization of the available airflow energy while minimizing aerodynamic losses. This characteristic is particularly important for mobile applications where the available airflow is directly related to vehicle speed and operating conditions.

Furthermore, the optimized blade configuration can contribute to improved power generation performance for auxiliary electrical systems installed on trucks and other large vehicles. The harvested energy may be utilized for battery charging, lighting systems, sensors, communication devices, or other low-power electrical loads. Such applications could reduce the dependence on conventional electrical power sources and support the implementation of sustainable transportation technologies.

Despite the promising results obtained from the CFD simulations, several limitations should be acknowledged. The present study was conducted under a constant airflow velocity of 22 m/s and focused primarily on blade aerodynamic characteristics. In actual operating conditions, airflow around a moving vehicle is influenced by vehicle geometry, traffic conditions, crosswinds, and road environments. Therefore, future studies should incorporate transient CFD analyses and experimental validation using a full-scale prototype installed on a moving vehicle to obtain a more comprehensive assessment of system performance.

CONCLUSIONS

The results of this study demonstrate that Computational Fluid Dynamics (CFD)-based aerodynamic optimization is an effective approach for determining the optimal configuration of vehicle-mounted axial wind turbine blades. The simulation results indicate that variations in the angle of attack significantly influenced the aerodynamic performance of the turbine blades, as reflected by changes in the lift-to-drag ratio and velocity distribution characteristics. Among the tested configurations, an angle of attack of 10° produced the most favorable aerodynamic performance, providing the highest lift-to-drag ratio and the most stable airflow pattern compared with angles of 0° , 5° , 15° , and 20° . The optimized blade configuration, consisting of six blades with a rotor diameter of 0.35 m, demonstrated the potential for utilizing vehicle-induced airflow as an alternative energy source to support sustainable transportation systems. This research contributes to the development of mobile wind energy harvesting technology by providing an aerodynamic optimization framework for vehicle-mounted turbine applications.

Future research should focus on validating the CFD simulation results through experimental testing using a physical prototype installed on an actual moving vehicle. Further investigations are also recommended to incorporate more complex operating conditions, including variations in vehicle speed, crosswind effects, transient airflow behavior, vehicle geometry influences, and real-world environmental conditions. In addition, future studies may evaluate the complete energy conversion performance of the system, including turbine rotational speed, electrical power generation, energy conversion efficiency, and the aerodynamic impact on vehicle fuel consumption. These improvements will provide a more comprehensive understanding of the practical feasibility of vehicle-mounted wind turbines and support their potential implementation as an innovative renewable energy technology in the transportation sector..

REFERENCES

- Afshari, F., Khanlari, A., Sözen, A., Tuncer, A. D., Ateş, I., & Şahin, B. (2023). CFD analysis and experimental investigation to determine the flow characteristics around NACA 4412 airfoil blades at different wind speeds and blade angles. *Proceedings of the Institution of Mechanical Engineers, Part E: Journal of Process Mechanical Engineering*, 237(5), 1915–1925. <https://doi.org/10.1177/09544089221128421>
- Ahmed, M. M. R., Mirsaeidi, S., Koondhar, M. A., Karami, N., Tag-Eldin, E. M., Ghamry, N. A., El-Sehiemy, R. A., Alaas, Z. M., Mahariq, I., & Sharaf, A. M. (2024). Mitigating uncertainty problems of renewable energy resources through efficient integration of hybrid solar PV/wind systems into power networks. *IEEE Access*, 12, 30311–30328.
- Arabzadeh, V., & Frank, R. (2024). Creating a renewable energy-powered energy system: Extreme scenarios and novel solutions for large-scale renewable power integration. *Applied Energy*, 374, 124088.
- Bouhentala, B., Navid, A., & Eckart, S. (2025). The role and potential of alternative fuels in driving the energy transition. *Journal of Thermal Analysis and Calorimetry*, 150(22), 18007–18042.
- Das, N., Sarkar, D., Hoque, N. A., Saikh, M. M., Das, S., & Ray, P. P. (2025). Upcycling agricultural and packaging waste: A spacer-free, all-porous triboelectric sensor for bidirectional breeze energy harvesting and real-time on-road vehicular over-speed surveillance. *Chemical Engineering Journal*, 166938.
- Della Posta, G., Leonardi, S., & Bernardini, M. (2023). Large eddy simulations of a utility-scale horizontal axis wind turbine including unsteady aerodynamics and fluid-structure interaction modelling. *Wind Energy*, 26(1), 98–125. <https://doi.org/10.1002/we.2789>
- Desalegn, B., Gebeyehu, D., & Tamrat, B. (2022). Wind energy conversion technologies and engineering approaches to enhancing wind power generation: A review. *Heliyon*, 8(11).
- Douvi, E., & Douvi, D. (2023). Aerodynamic characteristics of wind turbines operating under hazard environmental conditions: A review. *Energies*, 16(22), 7681. <https://doi.org/10.3390/en16227681>
- Emmanuel, J. K., & Ngabala, F. J. (2026). Global energy demand and the role of technological innovation towards renewable energy generation for economic growth and a cleaner environment: A review. *Sustainable Environment*, 12(1), 2633481.
- Gyamfi, B. A., Adebayo, T. S., Agozie, D. Q., Bekun, F. V., & Koy, A. (2025). Is sustainable energy consumption, technological advancement and urbanization fast addressing South Asia's green energy expansion deficits? *Environment, Development and Sustainability*, 27(8), 19969–19998.
- Kasmaiee, S., Kasmaiee, S., & Farshad, A. (2024). Unsteady CFD simulation of a rotor blade under various wind conditions. *Scientific Reports*, 14, 19176. <https://doi.org/10.1038/s41598-024-70350-5>
- Kumar, S., & Rathore, K. (2023). Renewable energy for sustainable development goal of clean and affordable energy. *International Journal of Materials Manufacturing and Sustainable Technologies*, 2(1), 1–15.
- Mansi, A., & Aydin, M. (2022). Numerical investigation of aerodynamic characteristics of wind turbine airfoils. *Energy Conversion and Management*, 253, 115174.
- Medina, C., Ana, C. R. M., & González, G. (2022). Transmission grids to foster high penetration of large-scale variable renewable energy sources: A review of challenges, problems, and solutions. *International Journal of Renewable Energy Research*, 12(1), 146–169.
- Mohamed, N. M. A., Kamal, K. M. M., & Binsuwadan, J. (2024). The adoption of renewable energy technologies by oil-producing countries: An inevitable outcome at a time of global challenges and demand for sustainable development. *Sustainability*, 16(8), 3155.

- Mohsin, M., Taghizadeh-Hesary, F., Iqbal, N., & Saydaliev, H. B. (2022). The role of technological progress and renewable energy deployment in green economic growth. *Renewable Energy*, 190, 777–787.
- Moreno-Pacheco, L. A., Sánchez-Hueto, L. J. L., Barbosa-Saldaña, J. G., Martínez-Trinidad, J., Toledo-Velázquez, M., & García-León, R. A. (2024). Assessment of wind energy potential generated by vehicles: A case study in Mexico. *Designs*, 8(6), 126.
- Paramati, S. R., Shahzad, U., & Doğan, B. (2022). The role of environmental technology for energy demand and energy efficiency: Evidence from OECD countries. *Renewable and Sustainable Energy Reviews*, 153, 111735.
- Rajamohan, S., Prakash, M., & Kumar, V. (2022). CFD-based aerodynamic optimization of horizontal axis wind turbine blades. *Energy Reports*, 8, 592–604.
- Roy, P., He, J., Zhao, T., & Singh, Y. V. (2022). Recent advances of wind-solar hybrid renewable energy systems for power generation: A review. *IEEE Open Journal of the Industrial Electronics Society*, 3, 81–104.
- Zhao, X., Wang, Y., Li, Z., & Chen, H. (2022). Aerodynamic performance analysis of wind turbine blades under varying angles of attack using CFD. *Renewable Energy*, 189, 812–825.