Journal of Manufacturing Engineering, March 2026, Vol. 21, Issue. 1, pp 018-024
DOI. https://doi.org/10.37255/jme.v21i1pp018-024

DESIGN AND CFD-BASED PERFORMANCE EVALUATION OF A SMALL-SCALE HORIZONTAL AXIS WIND TURBINE FOR COASTAL LOW-WIND REGIONS

Sondarangallage D.A., Sanjeewa, J.Wettasinghe

University of Vocational Technology, 100, Kandawala, Ratamalana, Sri Lanka.

Abstract

The growing environmental and economic impacts of fossil fuels have intensified demand for renewable energy. Wind energy is one of the most popular renewable energy sources, particularly in coastal regions. This study presents the design and numerical analysis of a small-scale horizontal-axis wind turbine (HAWT) intended for low-wind-speed applications. The turbine is designed for the coastal city of Hambanthota, Sri Lanka, where the average wind speed is 5 m/s. A wind turbine blade with a 5 m span was designed using NACA airfoil data and modelled in SOLIDWORKS. Computational Fluid Dynamics (CFD) analysis was performed using SOLIDWORKS flow simulation. It is used to evaluate the turbine's aerodynamic performance. Key performance parameters, including torque and efficiency. To evaluate the blade's stress, strain, and displacement characteristics, the aerodynamic loads obtained from the CFD analysis were subsequently applied in a structural simulation. The results indicate effective wind energy extraction and acceptable structural performance under the selected operating conditions. This study demonstrates the feasibility of using CFD-based simulation as a preliminary design and performance evaluation tool for small-scale wind turbines suitable for coastal and low-wind-speed regions.

Keywords. *HAWT, Computational Fluid Dynamics, Wind turbine*

1. Introduction

The world population has changed dramatically over the last few decades. Therefore, global demand for energy and concern about environmental degradation have increased [1]. With increasing demand and climate change reported worldwide, renewable energy sources have become essential alternatives to conventional fuels. These gases are linked to greenhouse gas emissions and resource depletion. There are many renewable energy sources, such as solar, wind, hydropower, and biomass. Among renewable energy sources, wind energy has emerged as one of the most promising and rapidly growing options for sustainable energy systems worldwide, thanks to its low emissions and declining costs as technology advances. As shown in Figure 01, a wind turbine converts the kinetic energy of the wind into mechanical and electrical energy. Their development has expanded from large wind farms to small-scale wind turbines.

Horizontal-axis wind turbines (HAWTs) are the most prevalent category of wind turbines due to their high energy capture efficiency and technological advancements [2]. In HAWTs, blades rotate about a horizontal axis, allowing them to extract energy from the wind efficiently. HAWTs are increasingly being considered for decentralized energy generation in coastal regions characterized by low wind speeds. Rotor blade performance is affected by airfoil geometry, wind speed, and blade dynamics [3]. For small wind turbines, accurate design and performance evaluation are essential in low-wind areas. Computation methods such as Computational Fluid Dynamics (CFD) are now commonly used to understand how air flows around the wind turbine blade and to predict aerodynamic forces with greater accuracy than traditional methods. CFD support enables designers to closely monitor complex flow effects, leading to better design selection and performance analysis under different conditions. When structural analysis is combined with CFD, it creates a complete design process. This complete design process can be used to predict both performance values and the responses of turbine parts to wind forces. This approach is useful for small HAWTs designed for coastal cities. In this study, a small-scale HAWT is designed and analysed for Hambanthota, a coastal city in Sri Lanka with an average wind speed of 5 m/s. The wind turbine blades were designed using SOLIDWORKS software and NACA airfoil data. SOLIDWORK Flow

**Corresponding Author -E-mail. sdasanjeewa@gmail.com*

Journal of Manufacturing Engineering, March 2026, Vol. 21, Issue. 1, pp 018-024
DOI. https://doi.org/10.37255/jme.v21i1pp018-024

Simulation is used to evaluate the aerodynamic performance.

2. Design of the wind turbine

2.1 Site and design specification

In this study, the three-bladed concept, the most common for modern wind turbines, is used. The selected coastal city is Hambanthota in Sri Lanka. To analyze the pattern of wind statistical information, wind data from Hambathota are collected. Fig. 1 shows statistical information on the distribution of monthly average wind speed. According to these infromation, the average wind speed in Hambanthota is 9.42 knots = 4.85 m/s (approximately)

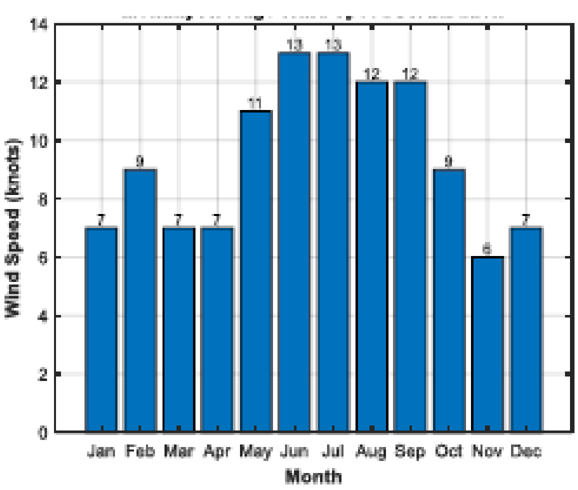


Fig. 1. Monthly average wind speed distribution

2.2 Wind Turbine Power Theory and Calculations

Wind processes kinetic energy due to the motion of air particles. When wind flows through the swept area of the wind turbine rotor, part of this kinetic energy can be converted into mechanical power. Kinetic energy of a moving mass of air is given by

$$E = \frac{1}{2}mv^2 \quad (1)$$

Where

E= Kinetic Energy(J)
m= mass of air(Kg)
V= wind velocity(m/s)

Power is defined as the rate of change of energy with respect to time.

$$P = \frac{dE}{dt} \quad (2)$$

Substituting the kinetic energy expression.

$$P = \frac{1}{2}v^2\frac{dm}{dt} \quad (3)$$

The mass flow rate of air passing through the rotor swept area is

$$\frac{dm}{dt} = \rho AV \quad (4)$$

where

ρ – air density (kg/m^2)
A- swept area of the rotor(m^2)

Substituting into the power equation

$$P = \frac{1}{2}\rho AV^3 \quad (5)$$

This equation represents the total power available in the wind flowing through the rotor area.

2.3 BETZ Law and Power Consumption

According to Betz's law, a wind turbine cannot extract all the kinetic energy available in the wind. The maximum fraction of power that is theoretically captured is,

$Cp_{,max} = 16/27 \approx 0.593$

This is the Betz limit. In real applications, wind turbines operate below this limit for several reasons, including aerodynamic, mechanical, and electrical losses. In order to account for these losses, the actual expected power of a wind turbine is expressed by using the power coefficient Cp

$$P = \frac{1}{2}\rho AV^3\text{Cp} \quad (6)$$

Table 1. Parameters of the wind turbine

Parameter	Value
Blade length (Rotor Radius), r	5m
Wind Speed, V	5 m/s
Air Density, ρ	1.23 kg/m^3
Power Coefficient, Cp	0.4

The rotor swept area is calculated as

$A = \pi r^2 = 3.14\ (5)^2 = 78.54m^2$

substituting values into the power equation

$P = ½ \times 1.23 \times 78.54 \times (4.85)^3 \times 0.4 \approx 2.20\ kW$

This value represents the available wind power and is consistent with a small-scale HAWT operating at low wind speeds.

2.4 3D Model of the Wind Turbine Rotor

Different standardized shapes of airfoils of wind turbines are available in the National Advisory Committee for Aeronautics (NACA) (now NASA). One of them is selected for this design. Initially, the airfoil profile is modeled in MATLAB to select a suitable NACA airfoil dataset.

Table 2. data of the airfoil of the wind turbine blade from NACA

Upper Surface x	Upper Surface y	Lower Surface x	Lower Surface y
1	0.0022	0	0
0.95	0.0211	0.0125	-0.0242
0.9	0.0385	0.025	-0.0348
0.8	0.0691	0.05	-0.0478
0.7	0.095	0.075	-0.0562
0.6	0.116	0.1	-0.0615
0.5	0.1318	0.15	-0.0675
0.4	0.1416	0.2	-0.0698
0.3	0.1427	0.25	-0.0692
0.25	0.1388	0.3	-0.0676
0.2	0.1317	0.4	-0.0616
0.15	0.1204	0.5	-0.0534
0.1	0.1035	0.6	-0.044
0.075	0.0924	0.7	-0.0335
0.05	0.0782	0.8	-0.0231
0.025	0.0584	0.9	-0.0127
0.0125	0.0445	0.95	-0.0074
0	0	1	0.0022

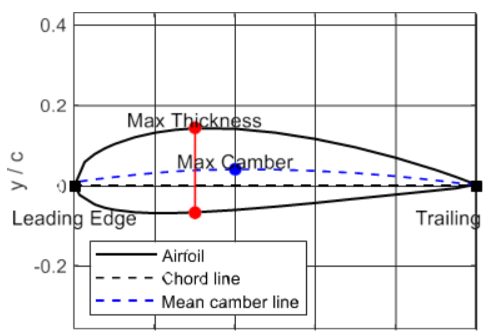


Fig. 2. Airfoil geometry with key parameters

To design a 3D model of windturbine blade, the pitch angle must be defined

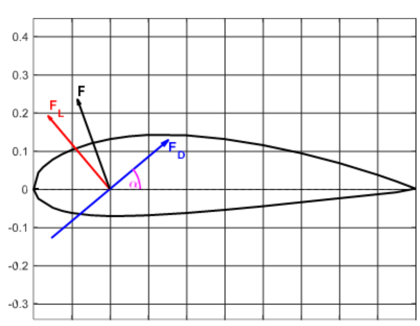


Fig. 3. Angle of Attack

The air hits the blade at an angle, which is called the angle of attack. represent the drag force and the lift force, respectively.

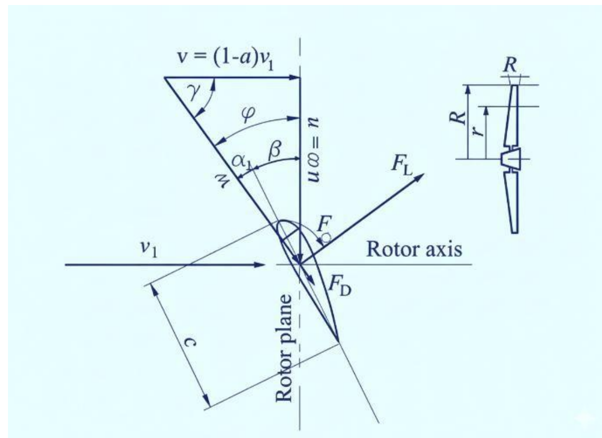


Fig. 4. Velocities and angles of the wind turbine

Where

v - wind velocity

u - tangential rotor speed w - relative wind speed

$w^2 = v^2 + u^2$

The axial velocity is adjusted by

$v = (1 - \alpha)v_1$

while

$u = \omega r$ aerodynamic performance.

To optimize the aerodynamic performance across the blade's span, the pitch angle (β) must be calculated for various radial positions(r).

Following the Betz optimization model, the angle of the relative wind to the rotor plane (φ) is determined by

$$(\varphi)r = arctan\left(\frac{2R}{3\lambda r}\right) \quad (7)$$

Final design pitch angle is derived by accounting for the design angle of the attacked $\propto_D$

$$\beta(r)_{Betz} = arctan\left(\frac{2R}{3\lambda r}\right) - \propto_D \quad (8)$$

This mathematical framework is applied to generate 10 discrete data sets for constructing the 3D blade model in SOLIDWORK.

Table 3. Parameters of the wind turbine

Parameter	Value
No of Blades	3
Blade Length	5m
number of elements	10
width of elements	0.5m
wind velocity	4.85m/s
angular velocity	7rad/s
Tip Speed Ratio	7

3D model of the small-scale HAWT design using the SOLIDWORKS software. Initially, 10 planes are created in parallel to the front plane with a 0.5 m gap, as shown in Fig 5(*a*) and 5(b). Then insert the basic shape of the curve tool of the feather ribbon in SOLIDWORKS. The same method was used to import all cross sections of the wind turbine blade to the respective plane, as shown in Fig. 5(c). turbine rotor (whole rotor) and the final shape of the rotor.

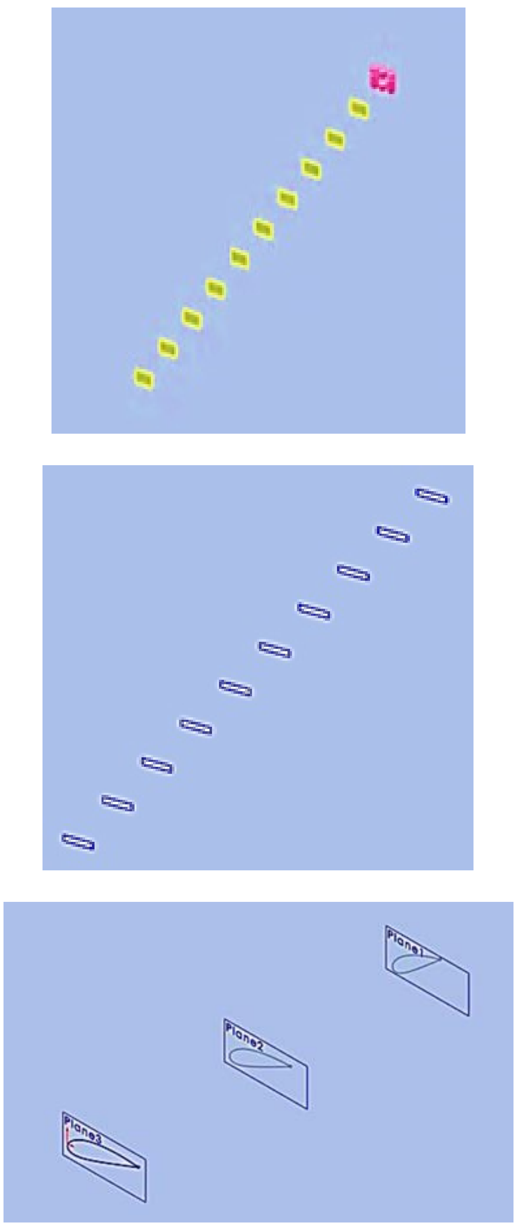


Fig. 5. Initial steps of designing the blade

After that, a 3D model of the wind turbine is created using the lofted function of SOLIDWORKS (Fig. 6). Fig. 7(*a*) and (b) show the final shape of the wind.

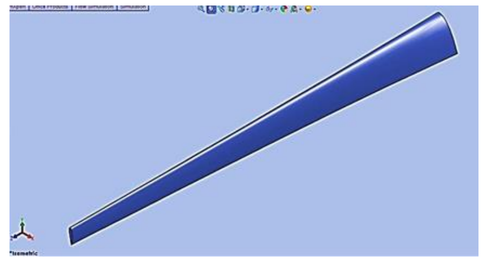


Fig. 6. 3D model of a blade

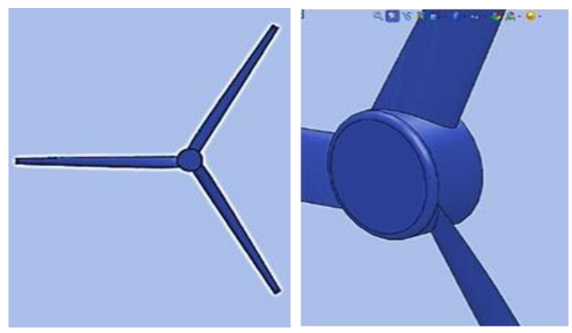


Fig. 7. final shape of the (a) wind turbine (b) rotor

3. Simulation configuration

3.1 Flow Simulation

Computational fluid dynamics (CFD) is a numerical method for analyzing fluid flow using computer-based simulation. Accurate aerodynamics prediction is necessary for effective design and performance evaluation of wind turbines. It provides practical, cost-effective wind turbine testing and full-scale experimental studies, enabling detailed analysis of airflow behaviour around turbine blades. In this study, SOLIDWORKS was used to perform the CFD analysis of the HAWT.

3.2 Design goals of the flow simulation

Four goals are defined for this flow simulation, summarized in Tables 4 and 5. After completing goal setting, the goal results can be viewed in SOLIDWORKS, as shown in the figure.

Table 4. Equation goals of the design

Goal Parameters	Formula	Goal Types
Power	{SG X - Component of Torque 1}* ω	Equation Goal
Efficiency	Power/ Calculated Power	Equation Goal

Table 5. Surface goals of design

Goal Parameters	Face	Goal Types
SG Av Total Pressure 1,	All faces that are in contact with fluid	Surface Goal
SG X-Component of Torque 1,	All faces that are in contact with fluid	Surface Goal

3.3 Static Simulation

Analysis of stress, strain, and displacement is completed using SOLIDWORKS Simulation (figure). For these analyses, static analysis is selected. The parameters used to analyze the process are listed in Table 6.

Table 6. Parameters for static analysis

Parameter	Description	Selection
Material	The Material that is used to build a wind turbine	Composite Material (Carbon/Epoxy)
Fixture	Fixed Geometry	Face1
External Load	Flow effect	Include fluid pressure effect from SOLIDWORKS flow simulation
Mesh	Create a Mesh	Standard Mesh

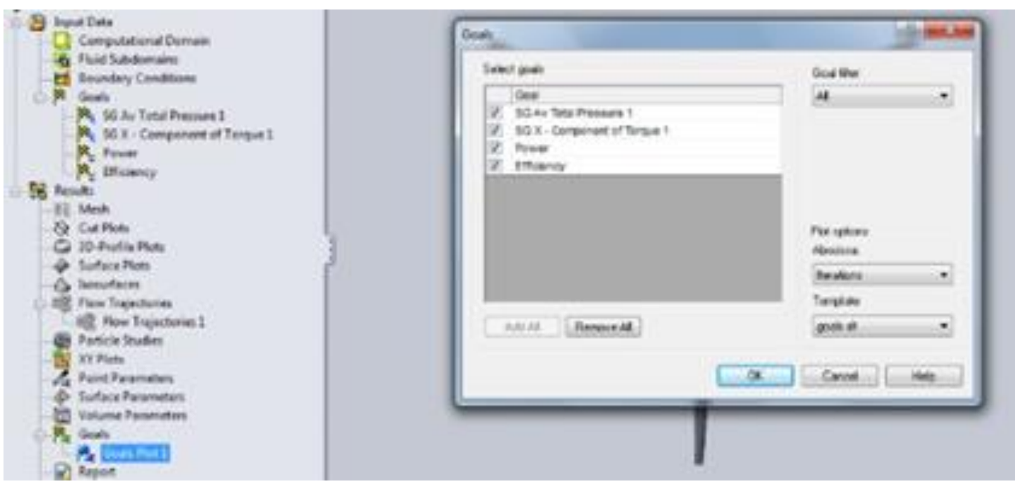


Fig. 8. Goal Creating

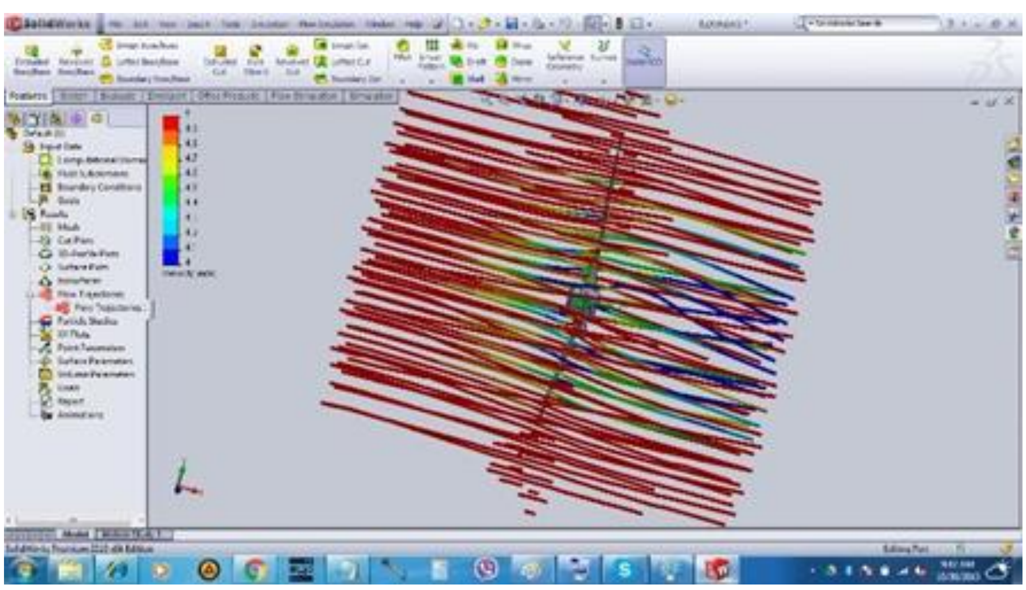


Fig. 9. Velocity analysis (Isometrix view)

Journal of Manufacturing Engineering, March 2026, Vol. 21, Issue. 1, pp 018-024
DOI. https://doi.org/10.37255/jme.v21i1pp018-024

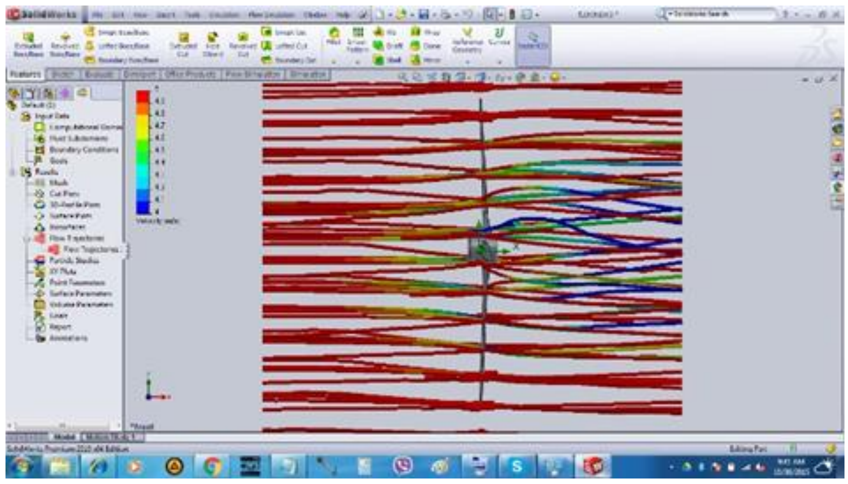
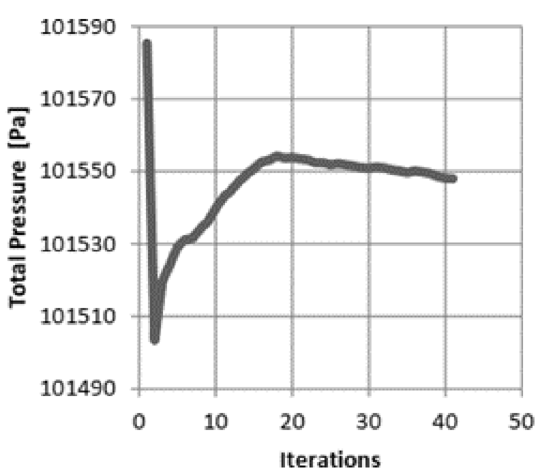


Fig. 10. Velocity analysis (front view)

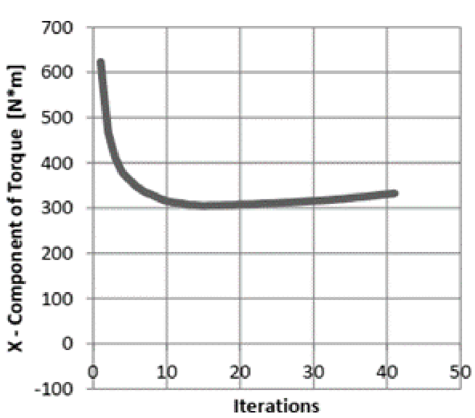
4. Simulation results and analysis

4.1. Flow Simulation Results



5.

6. Fig. 11. Convergence plot-total presure



7.

8. Fig. 12. Convergence plot-torque

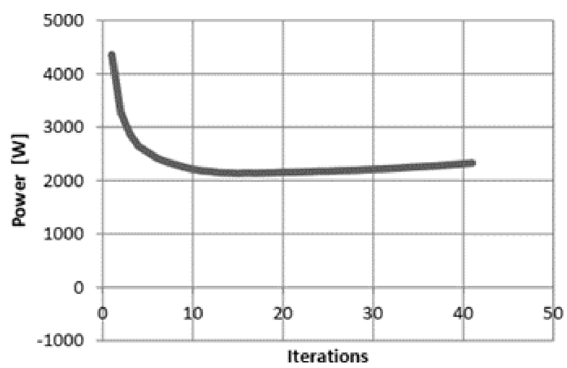


Fig. 13. Convergence plot-power

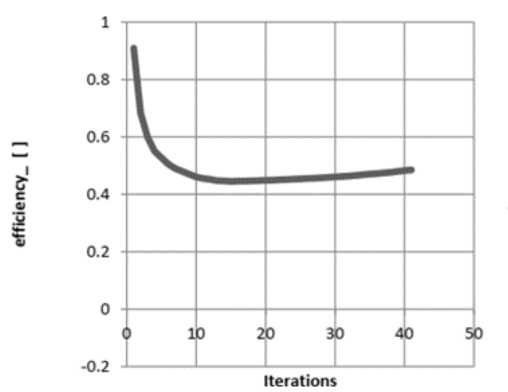


Fig. 14. Convergence plot-efficiency

Fig. 9 and Fig. 10 show velocity flow trajectories obtained from the SOLIDWORKS flow simulation of the designed wind turbine. The color range indicates the magnitude of the velocity. The red region indicates high velocities, and the blue-colored region indicates low velocities. The results show notable velocity reduction and flow disturbance of wind due to interaction with the wind turbine. It is evidence for energy extraction by the wind turbine. Beside the turbine, a wake region is formed where the airflow is slower and more disturbed. This wake is important because it affects the performance of turbines placed downstream. The CFD simulation was conducted over 41 iterations with an analysis interval of 20. The results (Fig. 11-Fig. 14) show the numerical stability and reliability of the model. The average total pressure at the inlet stabilizes at approximately 101.55 kPa. It is closely aligned with standard atmospheric conditions. The aerodynamic torque on the rotor is 332.34 N · m (final value) and 318.49 N · m (average). It shows consistent dynamic loading. Simulation results show that the turbine generates 2.3 kW with 48.5% efficiency under the simulated conditions.

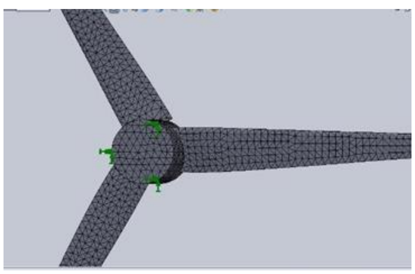


Fig. 15. Wind turbine after creating the mesh

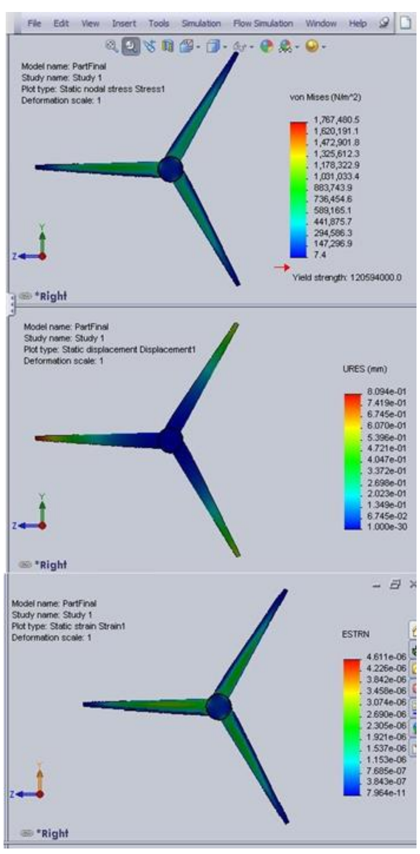


Fig. 15. Static simulation results

4.2 Static Simulation Results

Figure 14 illustrates the computation mesh and rotating region of the designed wind turbine modelled in SOLIDWORKS. Fig. 15 shows the structural analysis of the 3-blade HAWT rotor obtained from the SOLIDWORKS statics study. According to the results, the highest stress concentrations occur near the blade root and hub region. Lower stress levels can be observed along the blade span towards the blade tip. The strain result also shows the same pattern. The displacement plot shows that the largest deformation appears at the blade tip. The hub remains almost rigid. Overall, the results confirm that the wind turbine blades behave as expected under the given operating conditions.

5. Conclusion

This study presented a CFD-based performance evaluation of a small-scale horizontal-axis wind turbine for coastal low-wind regions. The blades of the wind turbine are designed using the NACA aerofoil data. It was analysed using SOLIDWORKS flow simulation to predict the pressure distribution, torque, power output and efficiency. The convergence of all the aforementioned goals was achieved within 41 iterations. The final solution satisfies the predicted convergence criteria. It confirms the numerical stability. The predicted power output is 2.3 kW and an efficiency of about 48.5%. Stress analysis shows that von Mises stress and strain are concentrated near the blade root. Maximum displacement occurred at the blade tips. The results confirmed the aerodynamic feasibility and structural integrity of the proposed design for coastal wind energy applications, providing a reliable basis for preliminary turbine development and future experimental versions.

References

1. *H. M. Robin, M. M. Ahmed, Md. R. I. Sarker, Md. N. Hasan, and M. H. Masud, "Aerodynamic performance investigation of a biomimetic-based small-scale horizontal-axis wind turbine: An experimental analysis," Wind Energy, vol. 28, no. 7, 2025, doi: 10.1002/we.70031.*

2. *F. E. Kan, V. Demirci, M. Seyhan, and M. Sarioğlu, "Performance analysis of small-scale horizontal axis wind turbines with outward leading-edge tubercles," Sustainable Energy Technologies and Assessments, vol. 82, p. 104531, 2025, doi: 10.1016/j.seta.2025.104531.*

3. *C.-J. Bai and W.-C. Wang, "Review of computational and experimental approaches to analysis of aerodynamic performance in horizontal-axis wind turbines (HAWTs)," Renewable and Sustainable Energy Reviews, vol. 63, pp. 506–519, 2016, doi: 10.1016/j.rser.2016.05.078.*

