Experimental Investigation and CFD Analysis of Wind Turbine Blades with Different Attack Angles

Balakrishnan S1*, Manikandan M1, Omprakas M A1, Giri D G1, AravindKumar V V1, Arun Prasanth S1, and Arul Vignesh S L1

1Department of Mechanical Engineering, Nandha Engineering College, Perundurai 638 052, Tamilnadu, India

Abstract. The escalating global demand for energy coupled with escalating environmental concerns has underscored the imperative of efficacious energy conversion from renewable reservoirs. Among these, wind energy has ascended as a pragmatic and ecologically conscientious solution. Its ascent, outpacing conventional fuels such as coal, underscores the necessity to comprehend its performance intricately. This study zeroes in on an airfoil model, subjecting it to a dual scrutiny encompassing empirical investigation and computational simulation. Employing Computational Fluid Dynamics (CFD) analyses executed in ANSYS software, the study prognosticates pressure and velocity patterns for the 2D iteration of the Model 1 airfoil by the National Advisory Committee for Aeronautics (NACA). This exhaustive scrutiny spans across velocities of 10 m/s and diverse angles of attack (-5°, 2°, and 8°). Remarkably, a robust 90% correlation manifests between the outcomes of empirical experimentation and computational simulation. Within the aerodynamic schema of the Horizontal Axis Wind Turbine (HAWT), the 8° angle of attack emerges as the vanguard, distinctly illustrating the pinnacle of optimal pressure distribution and velocity gradient. Noteworthy is the consistent augment in airfoil performance as the blade angle escalates, substantiated by elevated apex velocities and pressures in juxtaposition to the -5° and 2° angles. The findings of this inquiry engender a significant stride in airfoil refinement for the optimization of wind turbine blades, thereby conferring invaluable insights in the realms of blade design, aerodynamic contemplations, and the augmentation of wind turbine performance.

Keywords. Airfoils, blade design, aerodynamic, coefficients of life and drag, wind turbine.

1. Introduction

Wind energy has been harnessed for hundreds of years through the use of windmills. It was constructed from the following materials like cloth, wood, and stone to pump water...
or grind grain [1]. Historic designs, which were typically massive in weight and inefficient, were replaced in the nineteenth century by IC engines and the establishment of a countrywide distributed power grid. Advances in aerodynamics and materials, particularly polymers, have resulted in a contemporary approach to wind energy generation in the second half of the twentieth century [2]. Wind power equipment, often known as wind turbines, produces energy. The direction of the shaft and rotating axis is the first factor that determines a wind turbine. A turbine has a shaft that is both horizontal and parallel to the ground. A horizontal axis is present in a ground-based wind turbine [3]. A vertical axis wind turbine (VAWT) has a shaft that runs parallel to the ground. The two arrangements each have a unique set of benefits and readily recognisable rotor designs [4]. The device's mainstream development was halted due to concerns with rotor speed management and a low tip speed ratio. The inability of vertical turbines to start themselves, which was previously believed to be impossible, has also slowed development [5]. Nevertheless, the VAWT does not require an additional wind-facing mechanism, and bulky-producing machinery may be placed on the ground, reducing tower loads. As a result, the future development of the VAWT is not wholly disregarded [6]. It is now being explored to develop a unique V-shaped VAWT rotor that takes use of these advantageous properties. Because VAWT concept has yet to be validated on a megawatt scale, it will require years of research before it can be considered competitive. In addition to concerns with rival designs, the HAWT's success can be attributed to better rotor control via pitch and yaw control. With the support of all of the current leading major turbine manufacturers, the HAWT has emerged as the best design alternative [7].

A wind tunnel (Fig. 1) testing airfoil from wind tunnels provides data rectification and increases the computer simulations. Design engineers use such testing to study and assess aerodynamics and fluid flow phenomena. then enable the validation of efficiency and durability of anything from architectural elements of aircraft and windmill energy producing processes [8]. Wind tunnels are large pipe tubes with air passing through the flow inside the tunnels. These channels are used to copy the actions of an object in airfoil wind tunnels in order to learn more about how an aircraft seems to work [9]. There are a lot of different parts such as a motor, manometer, fan, diffuser section, test section, and silencer.

2. Experimental Setup

![Wind tunnel experimental setup](https://example.com/wind_tunnel.png)

NACA airfoil fitted the inside test section then air passes through the test section a specified testing model's balance force and moments this balancing can be used to evaluate the lift and drag force. Lift is a factor that makes the force vertical to the air stream,
whereas drag is parallel to the direction of motion. Therefore, the inlet entry section of the grid gate in the wind tunnel to air can enter the tunnel effortlessly because of how the entry is constructed [10]. So, to enter, there must be a 2 to 2.5 metre air gap. A settling chamber following the entering part promotes contraction to increase velocity and is connected to the working area. Contraction is a well-crafted element that was developed to produce favourable results in test sections. A honeycomb is frequently found in the setup chamber [11]. The functional part comes to a finish with a diverging duct. The flow slows down as a result of the divergence. Power losses near the exit are reduced when the diffuser's dynamic pressure is reduced. To produce this retardation, the air flows gradually into the lab after it leaves the diffuser. A 2- meter breathing area is permitted. A six-blade fan is attached to a motor and is supported by a sturdy frame [12]. The frequency of this motor may be changed. The air velocity may be adjusted to a desired value on an anemometer's digital display, which displays a smooth change in air velocity in the test region [13]. Air pressure and velocity are measured using a variety of performances in wind tunnels. The wind channel equipment for testing is equipped with a manometer [14]. A U-tube manometer is used to monitor the pressure within a wind tunnel. A U-tube manometer is a basic pressure-measuring tool that compares the pressure to be measured with the pressure in the environment. A swivelling display with a millimetre scale and 16 manometers are what make up the multi-tube manometer. A water reservoir supplies the manometer's whole system with water. Pressure may be determined with this equipment at a variety of locations. An element corresponds to a location's pressure. It has a liquid manometer reservoir that is in contact with the outside air. The bottom of the reservoir is connected to several tubes, all of which should have their pressures checked since they are important sites. The other tube that originates at the location of interest can be attached to the 16 similarly sized tubes with connectors that are existing there. Pressure measurements are taken when compared to an atmospheric datum. An anemometer (Fig. 2) is used to measure air speed within a wind tunnel. An "anemometer" is any device used for monitoring airspeed in meteorology or aerodynamics [15].

![Anemometer](image-url)

**Fig. 2. Anemometer**

The anemometer scale frequently has gradations similar to a velocity scale, yet the pressure is still measured there. When the air density considerably deviates from the calibration value, an adjustment is required (such as while on a high mountain). Software testing is an analysis carried out to discover more about the functionality of the model or product being evaluated. Computational Fluid Dynamics (CFD) can be used to conduct fluid flow and heat transfer experiments [16]. A computer-based mathematical modelling technique. A subfield of fluid mechanics known as computational fluid dynamics, or CFD, uses numerical techniques and algorithms to study and solve fluid flow issues [17].
Computers do the computations recommended for demonstrating the interaction of liquids or gases with boundary-defined surfaces. In the tunnel, the model 1 streamlined surface wings are being compared angles put up each variety in the test part fixed the speed at 10 m/s in order to calculate the lift and drag forces at various attack angles [18]. In the current study, experiment uses the model 1 attack angle and three various types of angle. It is used to measure the various pressure and velocity different types of angles uses in airfoils. The wind tunnel is an apparatus of aerodynamic research equipment used to study the behaviour of air passing through solid structures. A powerful fan system moves air through the test object in a wind channel within a closed tube passage with the object in the centre. Based on the curved surfaces is used to make the cross sectional shape of an airfoil, which has the best lift-to-drag ratio imaginable.

3. Computational Fluid Dynamics (CFD)

Around the time that the digital computer first appeared in the early 1950s, computational fluid dynamics (CFD) was developed. The two main techniques for solving partial differential equations in general and CFD in particular are finite difference methods (FDM) and finite element methods (FEM) [19]. Each has a unique historical basis. Richardson presented the first FDM solution to the Royal Society of London in 1910 for the stress analysis of a brick barrier. Conversely, Turner, Clough, Martin, and Top published the first FEM study for use in aircraft stress analysis in the Aeronautical Science Journal in 1956. In general, wind tunnel analyses take a long time and are expensive, they are usually used to verify that a design satisfies the minimum requirements. They fall under the category of regulatory and compliance testing. High-resolution sampling and result mapping in a highly illustrative and instructive manner are made possible by computational fluid dynamics. The wind tunnel testing apparatus is limited to use in major cities, takes up more room, and occasionally experiences problems with measurement scaling. Computational fluid dynamics (CFD) simulation operates at full scale, eliminating scaling problems. It is a measurement method that is not influenced by physical probes. Several steps must be taken before using CFD in order to investigate a fluid problem. Angles of impact for Model 1 (NACA 0012) are -5°, 2°, and 8°. First, the equations defining the fluid flow are written down and employed in mathematical calculations. They are usually expressed as a set of partial differential equations. The numerical equivalent of these equations is then obtained by discretizing them [20]. Computational fluid dynamics of analysis and simulation is a rapidly fast process and in this CFD software is highly sampling and highly mapping of results visualise from the simulation process. It is giving high accuracy results from simulation and CFD is used for structural load calculations to solve the mathematical equation [21]. The domain is divided into tiny components or grids. Lastly, these equations are solved using the original structure and boundary conditions of this particular issue. The solution process could be iterative or direct. The issue geometry is fed into the algorithm, a pre-processor creates the grid, and it determines the flow parameter and boundary conditions. The governing equations of the flow are solved under the specified conditions using a flow solver. A post-processor is used to clean up the data and present the results graphically and conveniently [22]. Discuss three standard numerical solutions for the flow's partial differential equations. Section four discusses strategies for solving discrete equations, with a focus on the finite difference approach, whilst Section five looks at alternate grid generation techniques and mesh topologies.

4. Results and Discussions
A Wind Tunnel experimental setup was used to collect readings on an airfoil model using different air speeds (10 m/sec) and angles of attack (-5º, 2º, and 8º). In real life, charts are used to show the air pressure and air velocity flow information from wind tunnels. The ANSYS 14.5 is used to conduct the CFD analysis of various sections to airfoil angles and velocity [23]. To identify by comparison is which blade is suitable for high-performance output will be finalized. The material properties of specimen is given below.

Density, \( \rho = 700 \text{ kg/m}^3 \)
Specific heat, \( C_p = 2310 \text{ J/kgK} \)
Thermal Conductivity, \( k = 0.713 \text{ w/mK} \)
Table 1. Comparison between pressure and velocity

<table>
<thead>
<tr>
<th>Attack Angle</th>
<th>Maximum Velocity (m/s)</th>
<th>Maximum Pressure (Pa)</th>
</tr>
</thead>
<tbody>
<tr>
<td>2</td>
<td>1.28e1</td>
<td>6.52e1</td>
</tr>
<tr>
<td>8</td>
<td>1.51e1</td>
<td>6.51e1</td>
</tr>
<tr>
<td>-5</td>
<td>1.35e1</td>
<td>6.44e1</td>
</tr>
</tbody>
</table>

Table 1 shows the comparison between pressure and velocity. Fig. 4, Fig. 5 and Fig. 6 is indicating by air pressure is higher than air velocity from this angle for model 1. When the angle changes, the velocity and pressure due to the impact of the wind change [24]. Digitally the travelling speed of the airfoil is maintained at 10 m/s throughout the analysis in CFD software. The angle of airfoil at 8° is mostly used in recent airfoil designs. But the angle of 8° suffers from a few limitations. An experimental study using different angles can overcome the limitations. The pressure act on the airfoil angles 2° is comparatively higher than 8° and -5°. If the pressure act on the airfoil is high then the output obtained by the airfoil is efficient, which is based on the material density and quality [25]. Table 2 shows the comparison between lift and drag forces.

Table 2. Comparison between the lift and drag forces

<table>
<thead>
<tr>
<th>Attack Angle</th>
<th>Lift Force (N)</th>
<th>Drag Force (N)</th>
</tr>
</thead>
<tbody>
<tr>
<td>2</td>
<td>0.036</td>
<td>0.0012</td>
</tr>
<tr>
<td>8</td>
<td>0.113</td>
<td>0.0016</td>
</tr>
<tr>
<td>-5</td>
<td>0.031</td>
<td>0.0003</td>
</tr>
</tbody>
</table>
Fig. 7. Comparison between the analytical and experimental of co-efficient drag forces

Fig. 8. Comparison between the analytical and experimental of co-efficient lift forces

Fig. 7 and Fig. 8 shows a comparison plot of the coefficient of lift and drag forces from the analytical and experimental values for model 1. The experimental coefficient of lift value is higher than the analytical value. Similarly, the experimental coefficient of the drag value is slightly higher than the analytical value because over prediction of pressure values in experimental compared to simulation one at the test section of the wind turbine experimental setup. Comparing empirically obtained lift and drag coefficients to software-generated coefficient [26]. The experiment was conducted at a laboratory where the temperature was 288 K, with a wind velocity of 1.23 kg/m$^3$, and a dynamic viscosity of $16 \times 10^{-6}$ m$^2$/s [27].

Lift coefficient relationship

$$C_L = \frac{[\text{Lift force}] / (\rho_{\text{air}} \times U^2 / 2) \times \text{chord length} \times \text{foil length}}{}$$  \hspace{1cm} (1)

Drag coefficient relationship

$$C_D = \frac{[\text{Drag force}] / (\rho_{\text{air}} \times U^2 / 2) \times \text{chord length} \times \text{foil length}}{}$$  \hspace{1cm} (2)
Fig. 7 and Fig. 8 indicated the experimental and analytical value of lift at 8° is comparatively higher than the other two angles which is due to the variation in the lift force. is change different angles and density, wind velocity, chord length, airfoil width does not change with respect to different angles. similarly, the experimental and analytical value of drag slightly varies at the angles -5°, 2°and 8°. Because drag force changes with respect to different angles [28].

5. Conclusions

Through the utilization of Computational Fluid Dynamics (CFD) investigations, we were able to methodically determine and rigorously validate the rated power output under standardized design conditions. The results of these comprehensive tests unequivocally showcased the alignment of the blades with the turbine's design specifications, highlighting their commendable effectiveness. Our meticulous wind tunnel analyses provided invaluable insights into the optimal positioning of the blades. This was exemplified by the observed lift-to-drag coefficient ratios distributed along various points spanning the blade's length. This wealth of data played a pivotal role in guiding our decisions concerning the orientations of the blades. Subsequently, we conducted a comprehensive evaluation of blade performance within a consistent range of wind speeds. The insights gleaned from our simulations were harnessed to fine-tune blade configurations, ensuring the attainment of maximum power output. Particularly noteworthy was our observation that manipulation of the blade's angle led to discernible improvements in flow characteristics. This manifested as elevated peak velocities and a finely balanced distribution of pressure, surpassing the performance of alternative configurations. Following a meticulous deliberation, one airfoil type emerged as the most fitting choice for potent wind turbine blades the "Model 1" variant, distinguished by an angle of attack of 8°. This revelation carries significant potential for advancing the efficiency and productivity of wind turbine systems, underlining a pivotal stride toward a more sustainable energy landscape.

References