

Article

Airfoil Control of Small Wind Turbines

Dean Sesalim *  and Jamal Naser *

Department of Mechanical and Product Design Engineering, Swinburne University of Technology,
Hawthorn, VIC 3122, Australia

* Correspondence: dsesalim@swin.edu.au (D.S.); jnaser@swin.edu.au (J.N.)

Abstract: Although the latest technologies have achieved outstanding improvements in the energy capture efficiency of wind power turbines, lack of cost effectiveness remains a major disadvantage. The problem lies in the limitation of operating effectively over a wide range of wind speeds. A variety of wind turbines have implemented variable control systems such as pitch control to extend the range of wind speeds at which the turbines can operate efficiently. These systems, however, have negatively impacted on the cost of wind energy. This paper investigates the possibility of achieving variable speed control performance by means of airfoil design for an uncontrolled wind turbine. Using computational fluid dynamics (CFD), this study optimizes the maximum thickness location of the S809 airfoil profile implemented in the National Renewable Energy Laboratory (NREL) Phase VI wind turbine blades to verify the design parameter most relevant to the objective of this research. The aerodynamic characteristics of the resulting design are compared to those of the baseline model. The results do indeed indicate the ability of the airfoil design to improve the power curve without the use of control systems.

Keywords: wind turbine performance; airfoil design; wind turbine simulation; computational fluid dynamics



Citation: Sesalim, D.; Naser, J. Airfoil Control of Small Wind Turbines.

Energies **2024**, *17*, 1217. <https://doi.org/10.3390/en17051217>

Academic Editor: Oscar Barambones

Received: 31 January 2024

Revised: 29 February 2024

Accepted: 1 March 2024

Published: 3 March 2024



Copyright: © 2024 by the authors. Licensee MDPI, Basel, Switzerland. This article is an open access article distributed under the terms and conditions of the Creative Commons Attribution (CC BY) license (<https://creativecommons.org/licenses/by/4.0/>).

1. Introduction

The impact of energy capture deficiency has been a major disadvantage despite technological developments in wind power turbines, and modern wind turbines are struggling to produce enough energy for their cost due to the ever-growing size and implementation of control systems of wind turbines. The use of wind turbine technology was recorded as early as the tenth century, starting as windmills in Persia for grinding grain and pumping water. The use of wind turbines later developed in Europe for purposes such as sawing wood. In fact, windmills in Europe were the main source of power before the development of the steam engine. The number of windmills in the Netherlands alone was over 9000. The Dutch windmills further developed to implement rotating tops to keep horizontal axis windmills perpendicular to the direction of the wind. These windmills consisted of several blades that covered most of the rotor disc area. The establishment of windmill farms later spread into Africa, Argentina, Australia, Canada, and the United States [1]. Lift-based turbine blade designs implementing airfoil shapes have then become the focus in the development of wind turbines for the superior wind energy capture efficiency they provide. However, the efficiency of such turbines depends on the accessible wind conditions. For example, the average monthly wind speed in Denmark throughout the year is 7.2 m/s, as observed over a 22-year period [2]. This has led to cost-effective clean power production equal to 50% of the total energy produced in Denmark. The primary advantage in the Danish case is the lack of extensive changes in the monthly year-round average wind speed. Such constant wind speed is not the case for most geographical locations. Nevertheless, lack of sufficient performance by small-scale wind turbines in Denmark has driven the development of larger, and thus, costlier, wind turbines [3].

The problem arises at speeds outside their design speed. As the relative wind speed fluctuates, so does the angle of attack (AoA), causing drag and loss of lift due to flow separation [4]. As a result, a turbine can achieve maximum performance when operating at an optimum AoA where the lift is maximum, and the drag is minimum [5]. The unstable fluctuation of power output is the result of complicated environmental effects such as variations in wind speed and direction, atmospheric turbulence, and ground boundary layers. The fluctuation of power output becomes severe under off-design operating conditions. Consequently, control systems are implemented to expose a turbine blade to a more favorable AoA in order to maintain a maximum lift-to-drag ratio as the wind speed or direction varies. The most widely used type of aerodynamic control to maintain the optimum AoA over a wide range of velocities is a variable pitch control system. This system actively changes the AoA of a turbine blade by pitching the blade to expose it to more favorable angles of incidence against the direction of the relative velocity in order to keep the airfoil operating with an attached flow configuration and maintain maximum lift to drag coefficient over a wide range of tip speed ratios [6,7]. Early implementation of pitch control goes back to as early as 1931, with a 30 m diameter wind turbine that was built in Russia and rated as 100 kW of power [1]. However, the best pitch-controlled early wind turbine was the 53 m diameter Smith–Putnam wind turbine built in 1941 in the USA. This turbine had a 3.4 m blade chord and tower height of 38 m. It was the first megawatt power rated wind turbine, with 1.25 MW of rated power at 14 m/s wind speed. Variable turbine blades pitch controlled the turbine’s rotational speed and feathered the blade positions perpendicular to the wind at wind speeds over 35 m/s to shut down the turbine and protect the structure from aerodynamic overloading. However, one of the turbine blades failed after 1100 h of operation due to failure of the control system. Although it remained operational until 1979 as the largest wind turbine, it did not continue operating because of its excessive operational cost [1].

Advances in pitch control systems since then have reduced the risk of structural failure, but at an even higher cost. The cost of pitch control technology has also been increased by some factors; lack of start-up data, for example, causes large tracking errors. To reduce this error, adaptive variable structure control accounts for system nonlinearities and external disturbances by utilizing continuous modification of the tracking signal, allowing the controller to improve its performance over time. Eventually, stable tracking is achieved with continuous modification of the varying pitch angle, but at high computational cost [8]. Moreover, such control systems remain dependent on complex and costly electronic sensors and mechanical actuators that require substantial maintenance.

While the fixed-pitch wind turbine approach provides robustness, reliability, and a simple low-cost foundation for wind turbine design, the optimization of such design has mostly relied on determining the optimal AoA for a single operating speed [5]. In general, uncontrolled wind turbines can achieve maximum performance when operating at an optimum AoA at which the lift is at a maximum and the drag is minimum. Therefore, the optimal AoA is determined at the point of maximum lift-to-drag ratio. However, the AoA of an airfoil increases with the increase in the relative velocity. As a result, a wind turbine blade enters the stall as the AoA increases due to flow separation over the blade’s lifting surfaces. This is despite operating at a small AoA to prevent flow separation. The relative flow velocity also increases with the increase in blade span. This impacts the optimum AoA at each segment of the blade along the span, resulting in stall near the hub or excessive drag near the tip. Hence, an optimum twist angle at each segment of the blade is necessary to distribute the optimum AoA over the span of the blade [6,9,10]. Nevertheless, the problem arises at operating conditions outside the design speed when flow separation becomes excessive, causing drag and loss in lift for the given airfoil design [4]. Thus, optimization of the airfoil design has become the focus in the development of wind turbine blades. Airfoil designs such as the S-series airfoils were developed by the NREL to minimize blade soiling performance losses in stall-regulated wind turbines [11]. Still, the development relies on control systems to regulate the stall of the blade.

Fundamentally, the maximum power of a wind turbine does not exceed 59.3% of the dynamic power according to the Betz limit [6,12]. However, the wind turbine defined by the Betz's limit is an open actuator disc where flow energy is extracted. This is equivalent to an idealized horizontal propeller operating at an infinite tip speed ratio. Driven from the principles of conservation of mass and momentum, the law does not describe the open flow operations of wind turbines. In actuality, the wind imparts torque on the wind turbine blades; more energy is extracted through this torque. Here, the wind turbine blades must impart on the wind an equal and opposite reaction torque, as dictated by Newtonian physics [6,13,14]. Nevertheless, the variation in the power coefficient with the variations in the tip speed ratio is dependent upon the AoA between the relative wind speed and the plane of rotation. Hence, the value of the tip speed ratio and the produced power is controlled by the changes in the AoA, and vice versa. Given the variation in conditions in which a wind turbine operates, the power coefficient, C_P , of the extracted power, P , is used to express the proportion of the wind power that is being extracted by the wind turbine. Here, the C_P of a wind turbine is defined as

$$\frac{\text{rotor power}}{\text{dynamic power}} = \frac{P}{\frac{1}{2}\rho AV^3} = \frac{T\omega}{\frac{1}{2}\rho AV^3} \quad (1)$$

where ρ is the air density, A is the area of the rotor disc, V is the relative wind velocity, T is the torque exerted on the rotational axis of the turbine, and ω is the rotational velocity of the blade/s. The wind velocity, however, is only one of two components of the actual turbine speed. This is because the air speed at the tip of the turbine blade is much higher than that at the hub or even the middle of the blade. Therefore, the relative speed is defined as the ratio between the blade tip speed to the relative free stream wind speed, $\omega r/V$, where r is the radius of the rotor disc or the length of the blade [6,7]. Hence, the torque result for a given airspeed does directly correspond to the power coefficient if the rotational speed is constant.

Although it is debated that uncontrolled wind turbine blades could achieve efficient energy production, a multiple airfoil uncontrolled wind turbine blade was designed by Mehedi Hasan, Adel El-Shahat, and Musfequr Rahman in 2017 [15], with a suggestion from NREL, which incorporated three different S series airfoil profiles (the S823, the S833 and the S822 profiles) to construct a blade from the root to the tip. This design resulted in a power curve similar to that of controlled wind turbines, particularly between 5 and 7 TSR after an increase in power between 2 and 5 TSR. The CFD results show a maximum C_P of 0.42 at TSR of 6, before gradually declining with the increase in TSR. Mehedi Hasan, Adel El-Shahat, and Musfequr Rahman distributed the three airfoils along the span of the blade according to their thickness ratio, given that airfoils are generally thicker at the root of a turbine blade and gradually become thinner as they approach the tip. However, by comparing the parameters of these three airfoils with those of a higher Reynolds number airfoil such as the S809, we can see from Table 1 that the S809 has a very close thickness ratio to that of the S833 airfoil despite the S809 being classified for a much higher Reynolds number with a very close distribution to that of the S822 profile. The increase in Re seems proportional to the distribution of the thicknesses along the chord. To clarify this, this study applied the locations of the maximum thickness of the S823, the S833, and the S809 airfoils at the root, around the midsection, and at the tip of a baseline wind turbine blade model, respectively. The effect of modifications on the torque output were then investigated to determine whether a controlled power output is possible as a result of modifying the locations of the maximum thickness of these airfoils.

Table 1. NREL airfoil data [6].

NREL Airfoil	Re	C_{Lmax}	Max Thickness (%)	At (%) Chord	Max Camber (%)	At (%) Chord
S823	4.0×10^5	1.2	21.2	24.3	2.4	70.5
S833	4.0×10^5	1.1	18	36.3	2.5	78.8
S822	6.0×10^5	1.0	16	39.2	1.8	59.5
S809	2.0×10^6	1.0	21	39.5	1	82.3

2. Methodology

The method implemented for this analysis is based on evaluating the optimization results against the baseline model results extracted from numerical simulations and validated using experimental data. The use of CFD simulations can be much more reliable than undertaking experiments. This is because CFD analysis provides measurable estimates of flow phenomena for all anticipated quantities of any realistic condition at any point of time rather than the limited description of flow phenomena, operational conditions, and quantities that life experiments provide at a single point of time and restricted number of locations. CFD codes can also analyze the actual domain of a flow and are not limited to laboratory-scale models [16]. However, the accuracy of a CFD code can be affected by some errors, most of which are modelling errors. This is due to incorrect geometry, simplified boundary conditions, and input data. These errors usually propagate in the calculation once they have been created. A CFD analysis is deemed stable if the error does not become much larger during the calculation. Hence, a well-defined case of validated data is crucial in order to achieve this stability. Phase VI of the NREL experiment, for instance, provided usable data for the development and reliable validation test cases for many numerical analyses of wind turbines [17–20]. Therefore, the NREL Phase VI wind turbine was employed in this analysis as the baseline model to ensure dependable and correct predictions.

2.1. Baseline Model and Conditions

The NREL Phase VI experiment utilized a two-bladed horizontal axis wind turbine with each rotor blade consisting of 26 sections of the S809 airfoil profile of varying chord lengths and twist angles, as detailed in Table 2. Here, the blade cross-section between 0.25 span station and the tip of the blade was uniformly equivalent to the S809 airfoil. From a maximum blade chord of 0.737 m at the 0.25 span station, the blade was tapered to 0.335 m at the tip. The section twist positively decreased towards feathering from 20.040° at the 0.25 span station to -1.944° at the tip with a positive twist conversion. The standard rotor had a diameter of 10.058 m at a hub height of 12.192 m. However, the data showed no considerable aerodynamic loads of the 0.4 m diameter cylindrical tower supporting the turbine, and decreases in rotor diameter due to the flap angle were also found to be negligible [8]. Although the presence of the turbine hub had an impact even when using a simplified cylindrical representation in the absence of geometric details, such simplification could have negatively impacted on the accuracy of the model; thus, excluding the hub remained a better choice. Hence, it is a sound prediction that excluding the tower and hub in the CFD model will have negligible effect on the results.

Furthermore, a sufficiently large volume is needed to capture all important flow behaviors, particularly in the wake of the turbine 24.4 m high, 36.6 m wide, and 57.912 m long in NREL NASA Ames test section [21], where the NREL Phase VI experiment was conducted. This has been demonstrated by a number of studies [20,22–27]. However, computational accuracy was confirmed by placing particular emphasis on the lengths of the inlet and outlet of the flow domain [28]. Larger factors were used, but within the limits of the NASA Ames test section in the case of the NREL Phase VI wind turbine.

A reference frame that rotates with the rotating device is usually created to accommodate multiple rotating devices that have a variety of rotating speeds and/or axes. It was, therefore, not created since the blades in this case rotated at the same angular speed

and around the same axes. Given that only the aerodynamic behavior of the turbine was investigated in this study, the interaction of solid surfaces was eliminated by suppressing the solid bodies in the flow volume.

Table 2. Blade chord and twist distributions [8].

Radial Distance R (m)	Span Station (R/5.029)	Chord Length (m)	Twist (Degrees)	Thickness (m) (20.95% Chord)	Twist Axis (% Chord), (m)
0.0	0.0	Hub-center of rotation	Hub-center of rotation	Hub-center of rotation	Hub-center of rotation
0.508	0.101	218 (root hub adapter)	0.0 (root hub adapter)	0.218 (root hub adapter)	50 (root hub adapter)
0.660	0.131	0.218	0.0	0.218	50
0.883	0.176	0.183	0.0	0.183	50
1.257	0.250	0.737	20.040	0.154	30, (0.221)
1.343	0.267	0.728	18.074	0.152	30, (0.218)
1.510	0.300	0.711	14.292	0.149	30, (0.213)
1.648	0.328	0.697	11.909	0.146	30, (0.209)
1.952	0.388	0.666	7.979	0.139	30, (0.199)
2.257	0.449	0.636	5.308	0.133	30, (0.190)
2.343	0.466	0.627	4.715	0.131	30, (0.188)
2.562	0.509	0.605	3.425	0.126	30, (0.181)
2.867	0.570	0.574	2.083	0.120	30, (0.172)
3.172	0.631	0.543	1.150	0.113	30, (0.162)
3.185	0.633	0.542	1.115	0.113	30, (0.162)
3.476	0.691	0.512	0.494	0.107	30, (0.153)
3.781	0.752	0.482	−0.015	0.100	30, (0.144)
4.023	0.800	0.457	−0.381	0.095	30, (0.137)
4.086	0.812	0.451	−0.475	0.094	30, (0.135)
4.391	0.873	0.420	−0.920	0.088	30, (0.162)
4.696	0.934	0.389	−1.352	0.081	30, (0.116)
4.780	0.950	0.381	−1.469	0.079	30, (0.114)
5.000	0.994	0.358	−1.775	0.075	30, (0.107)
5.029	1	0.335	−1.944	0.075	30, (0.101)

Test configurations such as that of Sequence S were the least complex, implementing rigid upwind turbine rotor with a 0° tilt angle, and no tip plate. The blades were not teetered, and the yaw angle remained at 0°. The blade and the blade tip pitches were configured at 0° and 3°, respectively, in upwind locations of the rotor with a 0° cone angle. The test section employed speeds of 5, 10, 15, 20, and 25 m/s, which were maintained within close tolerance to nominal values of uniform wind velocity normal to the inlet. The 10 m/s wind speed was categorized as the onset of stall speed at which flow separation started around the mid-span of the blade and moved gradually towards the outer span of blade as wind speed increases [6,20,22–27]. This causes a challenge to the numerical reproduction of such effects, particularly at speeds higher than 10 m/s where the blades are almost entirely stalled. The rotation rate of 72 RPM (71.63 RPM synchronous speed) was constant, with uniform angular velocity and constant pitch and RPM; therefore, the test conditions resembled a steady flow. The default 1.225 kg/m³ sea level density, 101,325 Pa static pressure, and no-slip boundary conditions were used at the four side walls of the wind tunnel so that the thin boundary layers on the tunnel walls were captured [8].

2.2. Turbulence Model

The shear stress transport *SST* $k - \omega$ two-equation turbulence model is one of the few models that can generate a laminar-to-turbulent boundary layer transition, simulating flow

separation, and obtaining the correct pressure distribution for low-Reynolds-number wind turbines such as the NREL Phase VI wind turbine, with effective trade-off between computational cost and accuracy [20,22–28]. The advantage of this model is the implementation of a blending function that activates the $k - \omega$ model at the surface and switches to the $k - \varepsilon$ model behavior in the freestream away from the surface to overcome the sensitivity to the free shear flow and boundary layer in the $k - \omega$ model and compensate for the limited ability of the $k - \varepsilon$ to capture the flow characteristics near the surface. The k and ω in the standard $k - \omega$ model are obtained using the following transport equations:

$$\frac{\partial}{\partial x}(\rho k) + \frac{\partial}{\partial x_i}(\rho k u_i) = \frac{\partial}{\partial x_j} \left(\Gamma_k \frac{\partial k}{\partial x_j} \right) + G_k - Y_k \quad (2)$$

and

$$\frac{\partial}{\partial t}(\rho \omega) + \frac{\partial}{\partial x_i}(\rho \omega u_i) = \frac{\partial}{\partial x_j} \left(\Gamma_\omega \frac{\partial \omega}{\partial x_j} \right) + G_\omega - Y_\omega \quad (3)$$

From the exact equation for the transport of k , this generation may be defined as follows:

$$G_k = -\rho \overline{u_j^i u_j^i} \frac{\partial u_j}{\partial x_i} \quad (4)$$

G_ω represents the generation of ω , and is given by

$$G_\omega = \alpha \frac{\omega}{k} G_k \quad (5)$$

The coefficient α is given by,

$$\alpha = \frac{\alpha_\infty}{\alpha^*} \left(\frac{\alpha_0 + \frac{Re_t}{R_\omega}}{1 + \frac{Re_t}{R_\omega}} \right) \quad (6)$$

where,

$$R_\omega = 2.95 \quad (7)$$

here, α^* represents the coefficient that dampens the turbulent viscosity, causing a low Reynolds number correction, and is given by,

$$\alpha^* = \alpha_\infty^* \left(\frac{\alpha_0^* + \frac{Re_t}{R_k}}{1 + \frac{Re_t}{R_k}} \right) \quad (8)$$

$$Re_t = \frac{\rho k}{\mu \omega} \quad (9)$$

$$R_k = 6 \quad (10)$$

$$\alpha_0^* = \frac{\beta_i}{3} \quad (11)$$

$$\beta_i = 0.072 \quad (12)$$

This leads to very effective simulation of the turbulences created by small wind turbines such as the NREL Phase VI wind turbine since the low-Reynolds-number effects are considerable in this case and propagate as the velocity increases due to the high viscosity.

2.3. Meshing Strategy

Considering the importance of the geometrical characteristics of the baseline and modified models in this analysis, a finer mesh was necessary in regions where there was flow circulation or large gradients and flow separation in order to capture the geometrical effects of the models, particularly their curvatures. Not necessarily by significantly increasing the mesh resolution, but by refining it where required, particularly in regions where the flow may be circulating, or where there are large gradients, such as surfaces with large curvatures. Thus, control of the curvature min size and the curvature normal angle were the two parameters adjusted to refine the mesh. The effects were localized finer mesh in regions of curvature and rapid size change. The higher the curvature, the finer the mesh. Regions with no curvature were not affected by the refinement, and the mesh at these regions remained coarse. As a result, high mesh resolution accumulated on the entire blade. This enhanced calculations in regions of interest where large pressure gradients need to be resolved for an accurate computation.

To further reduce computational cost, the tetrahedral cells were converted to polyhedral cells in Ansys Fluent, as shown in Figure 1. The advantage of a polyhedral element is the number of nodes for each element, which allows it to have more neighboring cells so the gradient can be better approximated with fewer cells. The near-wall treatment consisting of layers of hexahedral mesh elements at the wall was found to be effective at structuring the curvature that transits the mesh to the surroundings of the blades since it augmented the mesh to produce a smooth transition along the wall and ensure gradual variations and uniformity in the element height throughout the transition. The calculated torque output results in Figure 2 for the critical 10 m/s inlet velocity case with three sets of differing mesh resolutions validated mesh independence of the intermediate mesh, as the higher resolution did not yield an improvement of more than 5% in the measured results. The lower to higher resolutions are based on minimum curvature sizes of 3, 2 and 1 mm and minimum curvature angle of 3°, 2°, and 1°, respectively. The intermediate resolution converged at 1462 iterations using the ANSYS Fluent default –1000 minimum residuals convergence criterion.

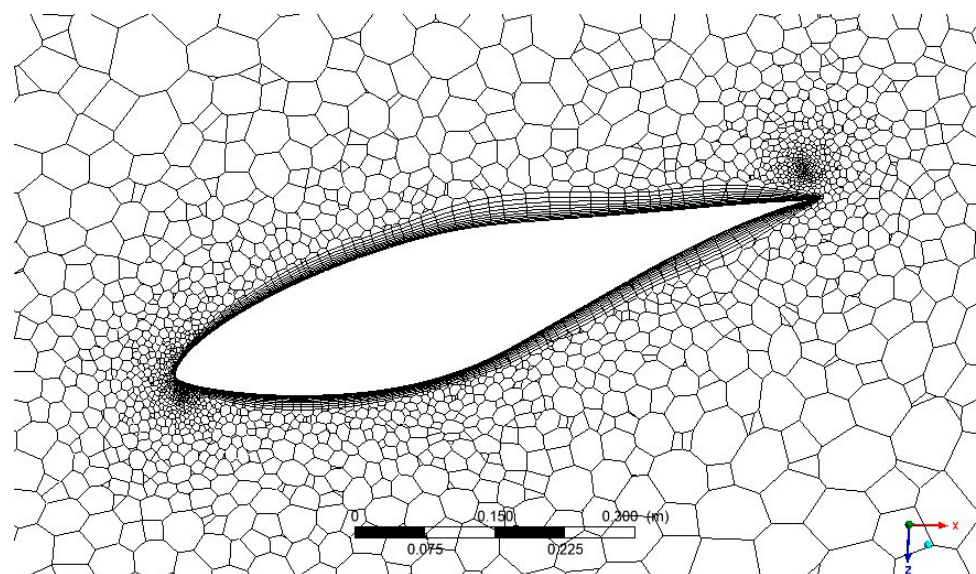


Figure 1. Profile of the S809 airfoil and the near-surface mesh.

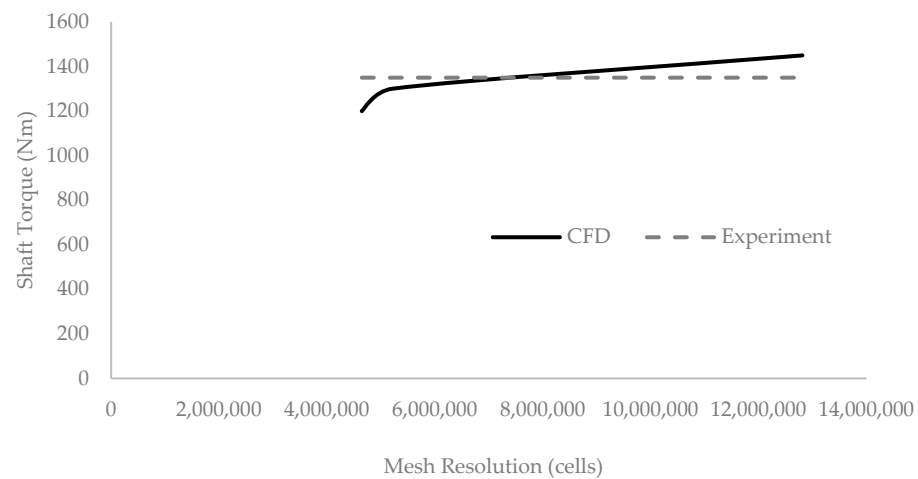


Figure 2. Mesh sensitivity study.

3. Results and Comparison

To compare the results, simulations of the original baseline model and the modified model were carried out under the exact same boundary conditions and geometrical scale of the NREL Phase VI Sequence S test parameters, validating the baseline model numerical results against the experimental data. The torque results in Figure 3 show close agreement with the data from the numerical results of the baseline model and enhanced the performance range by the modified model in the 15 and 20 m/s inlet speeds where the baseline model experienced momentary stall. The performance of the latter, however, remained higher at the lower inlet speeds of 5 and 10 m/s. This was not expected from turbine blades dominated by a nearly symmetrical airfoil with a higher Reynolds number and lower lift coefficient. It is perhaps another factor in the three-dimensional environment, such as the location of the modification along the span, which influenced these results. Nevertheless, a near-controlled power curve was the result of modifying the location of the airfoil's maximum thickness in the baseline model, as was expected. To gain in-depth understanding of the aerodynamic influences by the modified variable, the two-dimensional aerodynamic characteristics of the modified airfoil were analyzed in comparison to the original S809 airfoil for the inlet speed cases where the performance of the two models deviated the most.

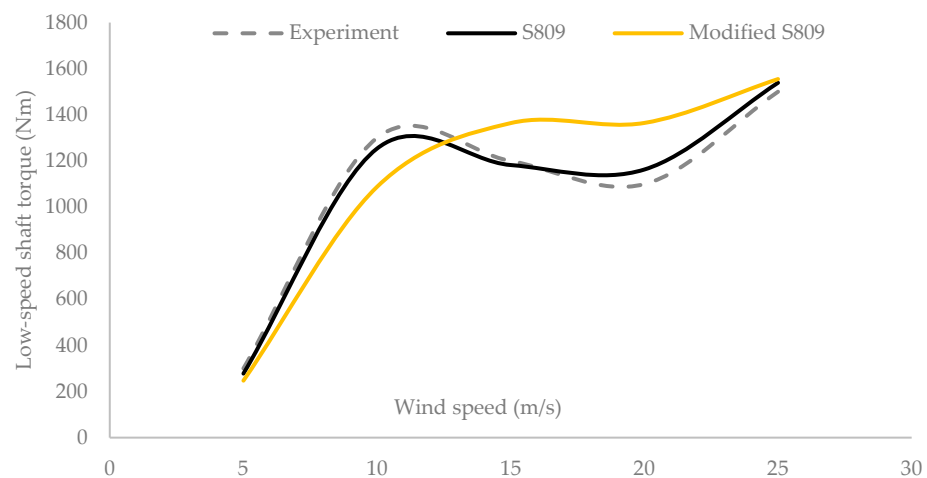


Figure 3. Torque results.

3.1. Pressure Distribution

The occurrence of stall conditions on the blade can be identified using pressure distributions. Stall prediction by the numerical models is also used here to evaluate the accuracy of the numerical results. The relative pressures are described by the dimensionless pressure

coefficient (C_p) number at the relevant points in the flow field, with each point having its own exclusive C_p near the model, independent of its size. This helps determine the critical locations where forces are mostly active at points at or near the surfaces of the models in order to investigate the leading cause of the calculated torque results. An expression was added to the Ansys CFD-Post resampling the pressure coefficient formula of

$$C_p = \frac{(P - P_{ref})}{q_{ref}} = \frac{(P - P_{\infty})}{\left(\frac{1}{2}\rho V_{\infty}^2\right)} \quad (13)$$

where P is the static pressure or normal force per at the evaluated area, determined by the impact of the flow molecules; P_{ref} is the reference pressure; q_{ref} is the reference dynamic pressure; P_{∞} is the static pressure of the freestream, equal to zero atmospheric pressure away from disturbances; ρ is the flow freestream density, defined at 1.225 kg/m^3 ; and V_{∞} is the freestream velocity, equal to the relative velocity.

In the NREL Phase VI experiment, each blade was fitted with pressure taps at 30%, 46.6%, 63.3%, 80%, and 95% span locations, instrumented toward the leading edges, to capture the most active regions of pressure distribution [23]. Evaluation of the numerical results of two-dimensional pressure distribution around airfoils at these span locations can therefore be considerable reliable since these results can be validated against the experimental data, as shown in Figures 4–6. At 10 m/s, the modified airfoil generated more negative pressure than the original S809 as the relative velocity increased along the span of the blade, as shown in Figure 5. Despite this, the original S809 generated more lift given that the higher-pressure differentials appeared to distribute between the upper and lower surfaces of its blade. Here, the original S809 seemed to generate more drag-based lift than the modified model. This gave it the ability to produce more life at lower speeds. However, stall conditions occurred as the inlet speed increased and the pressure over the lifting surface of the original blade increased significantly, which explains why it generated less lift at the 15 and 20 m/s inlet speeds. Still, the original S809 airfoil maintained higher pressure on the lower surface of the blade to generate considerable drag-based lift at these inlet speeds. In the meantime, the negative pressure increased significantly over the lifting surface of the modified blade as the inlet speed increased, as shown in the figures. This increased the pressure differential and generated the higher lift reflected on the torque results of the modified model. However, decrease in this pressure differential due to decrease in pressure over the lower surface of the blade was noticeable as we approached the tip, indicating that flow separation was accruing under the blade as the AoA increased. To verify this further, visualization of the velocity profile over the two models was deemed necessary.

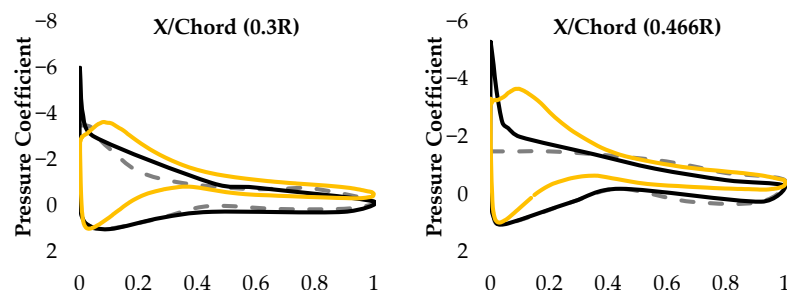


Figure 4. Cont.

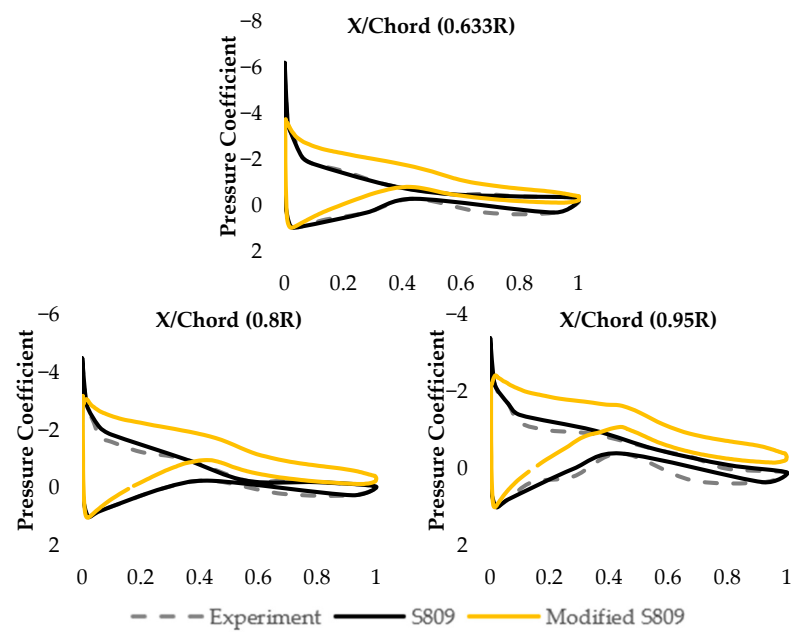


Figure 4. Pressure coefficient at an inlet velocity of 10 m/s.

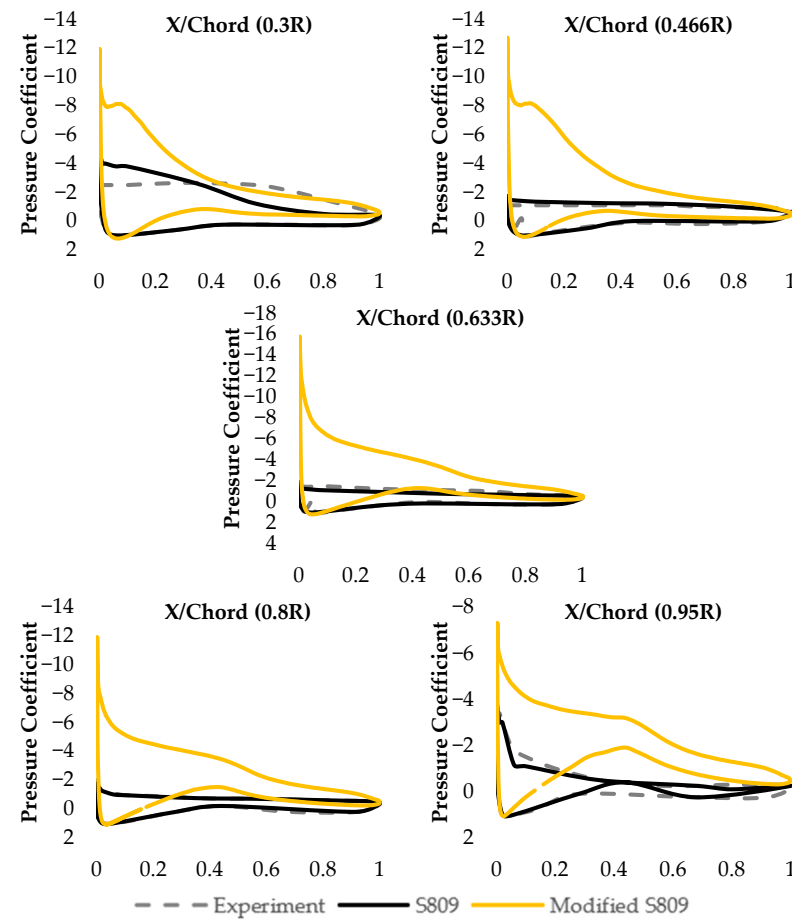


Figure 5. Pressure coefficient at an inlet velocity of 15 m/s.

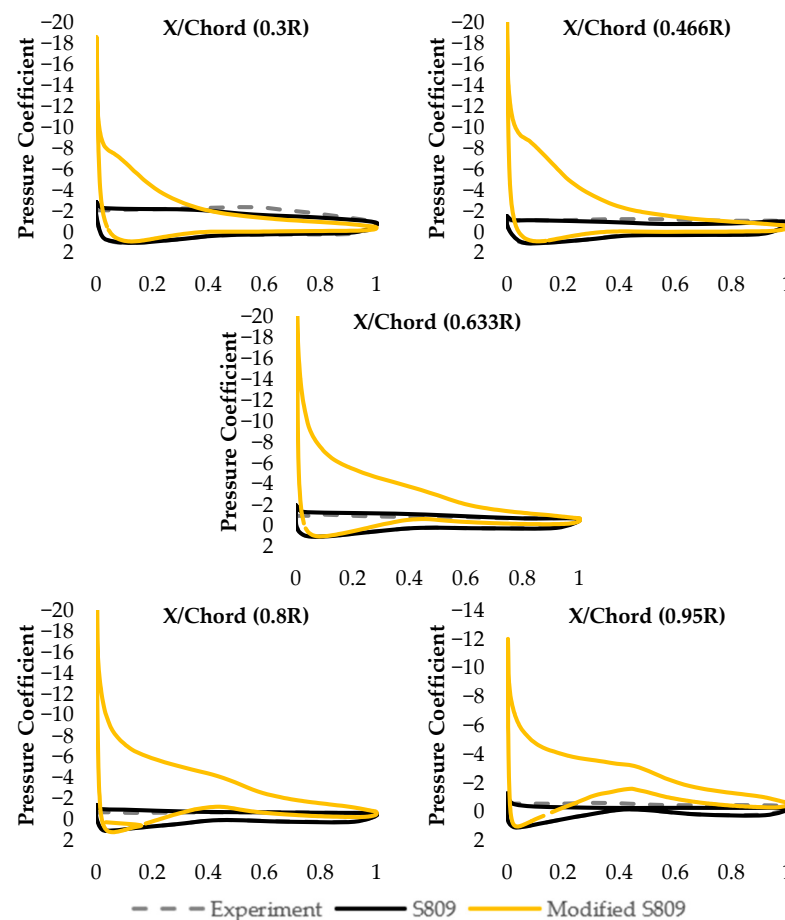


Figure 6. Pressure coefficient at an inlet velocity of 20 m/s.

3.2. Velocity Profiles

The velocity profiles as shown in Figures 7–9 from a distance of 0.5 m appear to be in agreement with the pressure distribution results. At a 10 m/s inlet speed, the flow velocity appeared higher, particularly over the lower surfaces of the modified model, which decreased the pressure distribution considerably around this blade in comparison to the baseline model, thus generating a significantly slower flow over its lower surface in comparison to what the modified model generated. This explains the lower pressure distribution calculated over the lower surface of the original blade. The deviation in the velocity profiles of the two models, however, seemed to decrease during the 15 and 20 m/s inlet speed, as shown in Figures 8 and 9. Here, the flow velocity over the lifting surface of the modified blade was somewhat higher than the original blade near the root of the blade. However, this velocity was almost the same near the tip, suggesting that shifting the location of the maximum thickness backward kept the flow attached to the lifting surface of the blade as the velocity and/or the AoA increased.

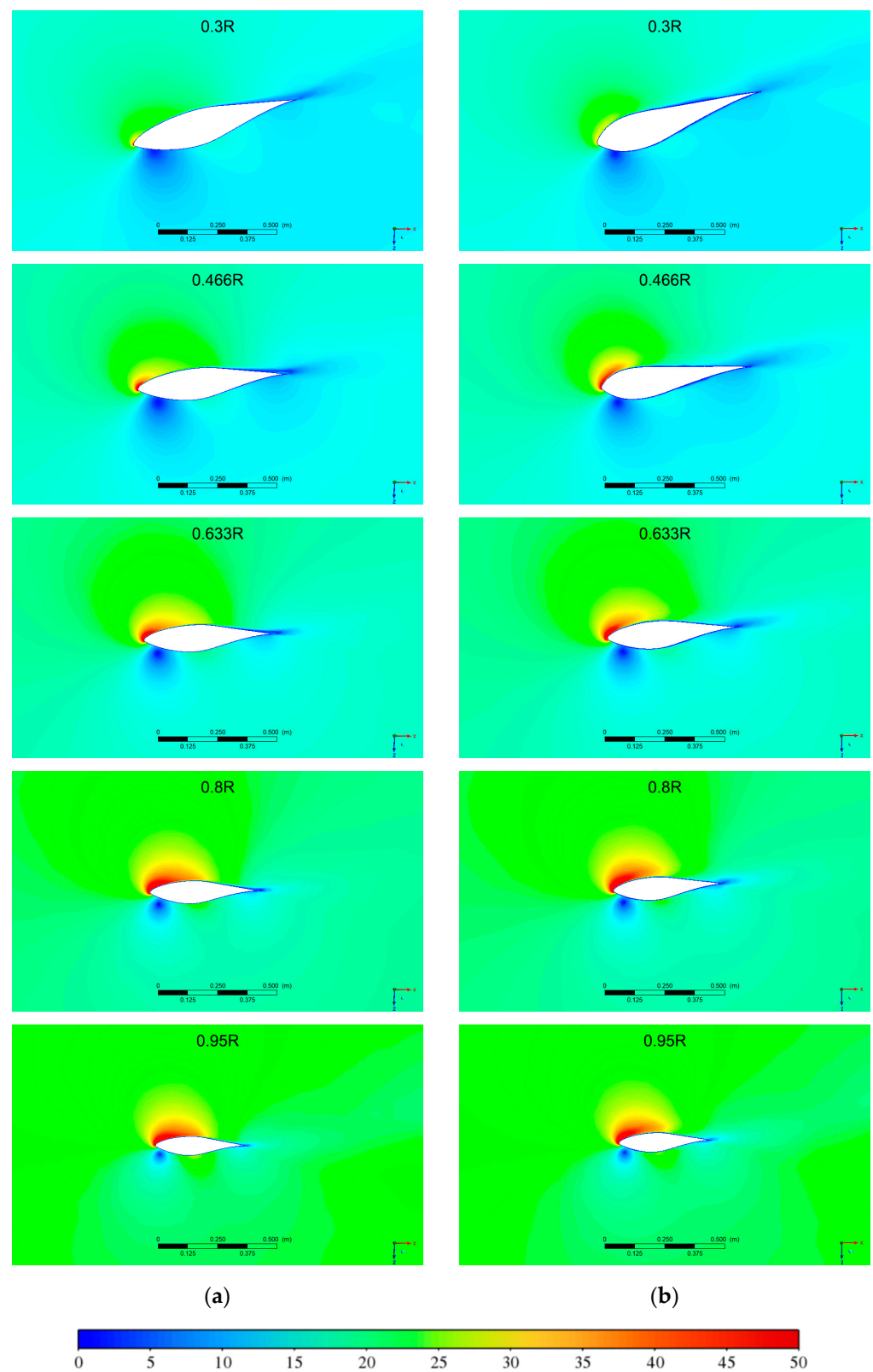


Figure 7. Velocity contours at an inlet velocity of 10 m/s: (a) S809 airfoil; (b) modified airfoil.

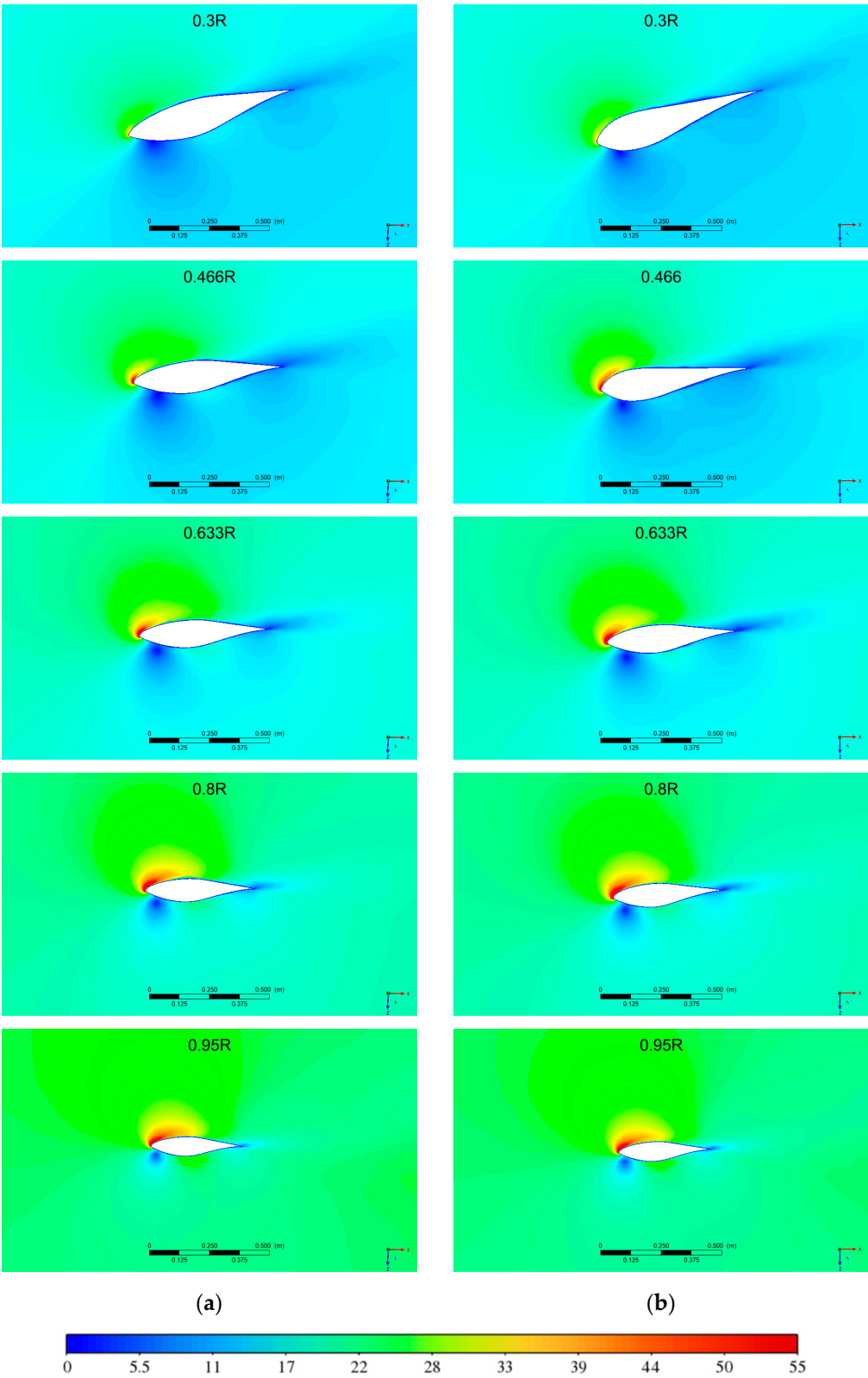


Figure 8. Velocity contours at an inlet velocity of 15 m/s: (a) S809 airfoil; (b) modified airfoil.

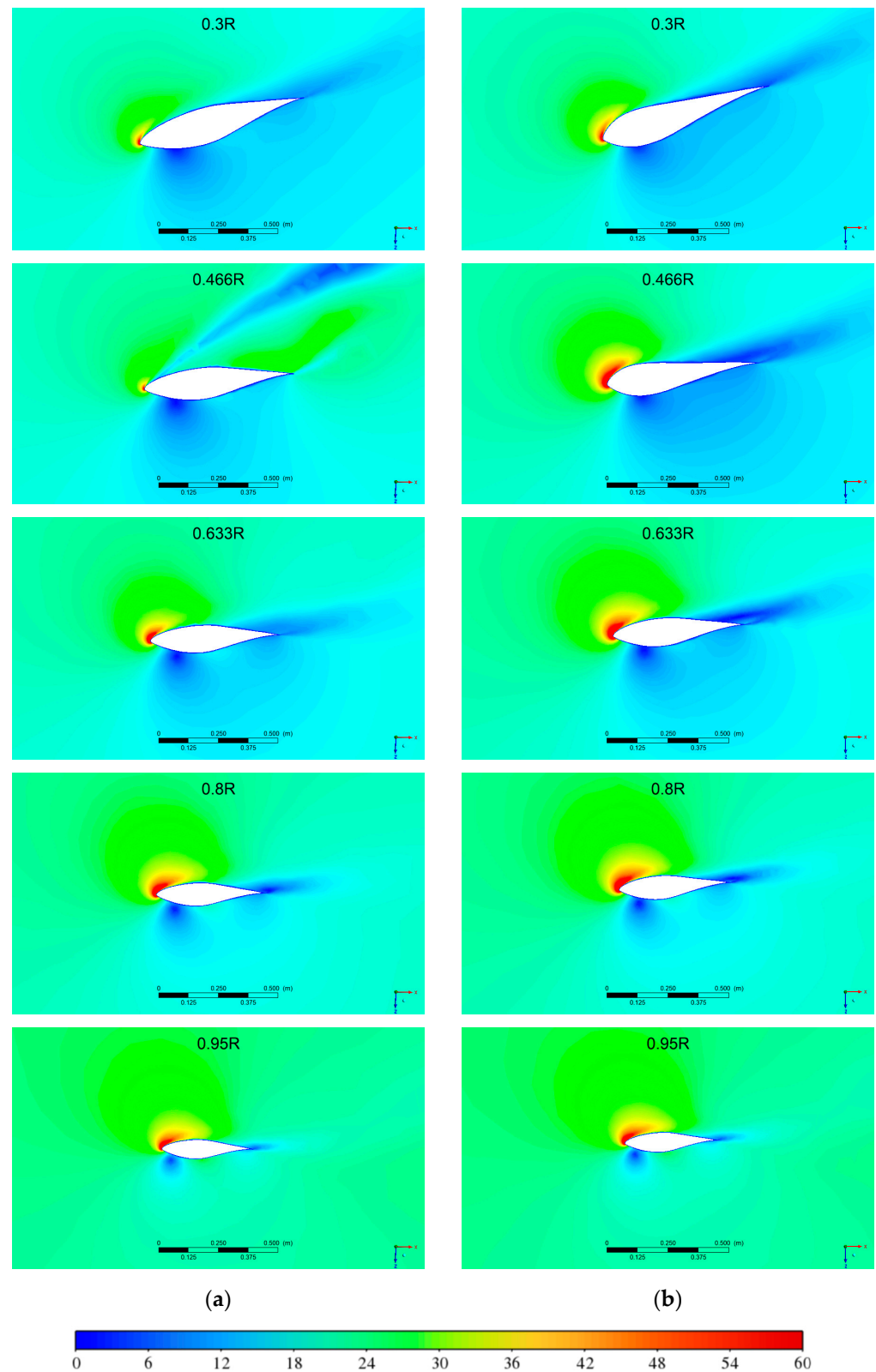


Figure 9. Velocity contours at an inlet velocity of 20 m/s: (a) S809 airfoil; (b) modified airfoil.

3.3. Streamline Analysis

The effect of the modification on the streamline characteristics of the baseline model in comparison to its original characteristics consisted of minor increases in velocity in the wake in all three inlet velocity cases shown in Figures 10–12. The three-dimensional

isometric views revealed increases in velocity over the blade surface, particularly near the leading edge, due to the modifications. This is not necessarily a disadvantage for the modification since the blue shading of the near-zero speed seen on most of the baseline blades could be due to viscous drag and the separation of flow. Drag due to flow separation seemed to be dominant on the original model near the trailing edge. Another indication of improved performance was the reduction in vortices near the root of the blades in the 20 m/s inlet velocity case due to the modification. Nevertheless, the near-tip flow behavior appeared to be the same, as the same parameters of the S809 airfoil were employed in the modification near the tip. No influence due to the modified regions seemed to appear near the tip of the blade.

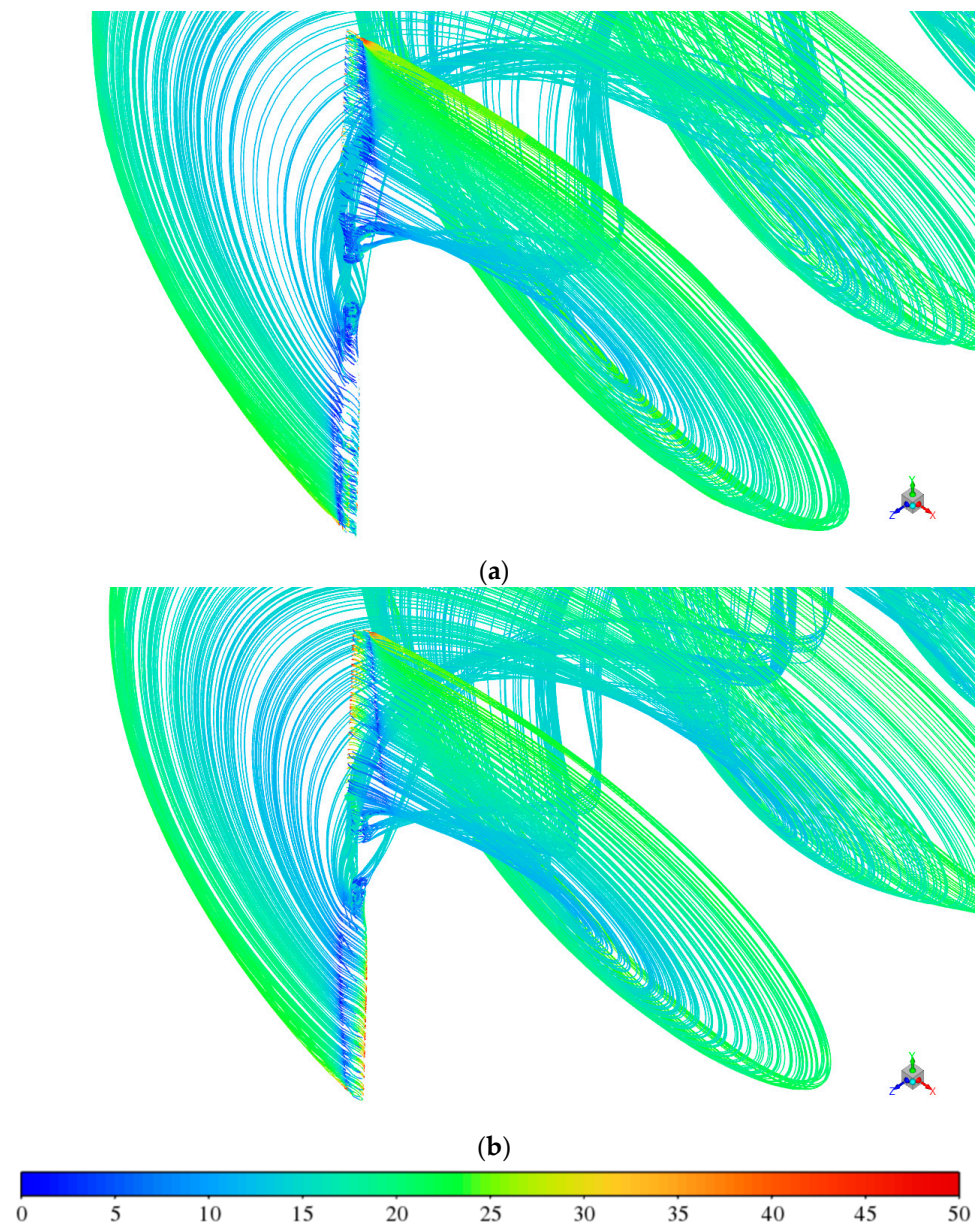


Figure 10. Three-dimensional streamline velocity at 10 m/s: (a) S809 airfoil; (b) modified airfoil.

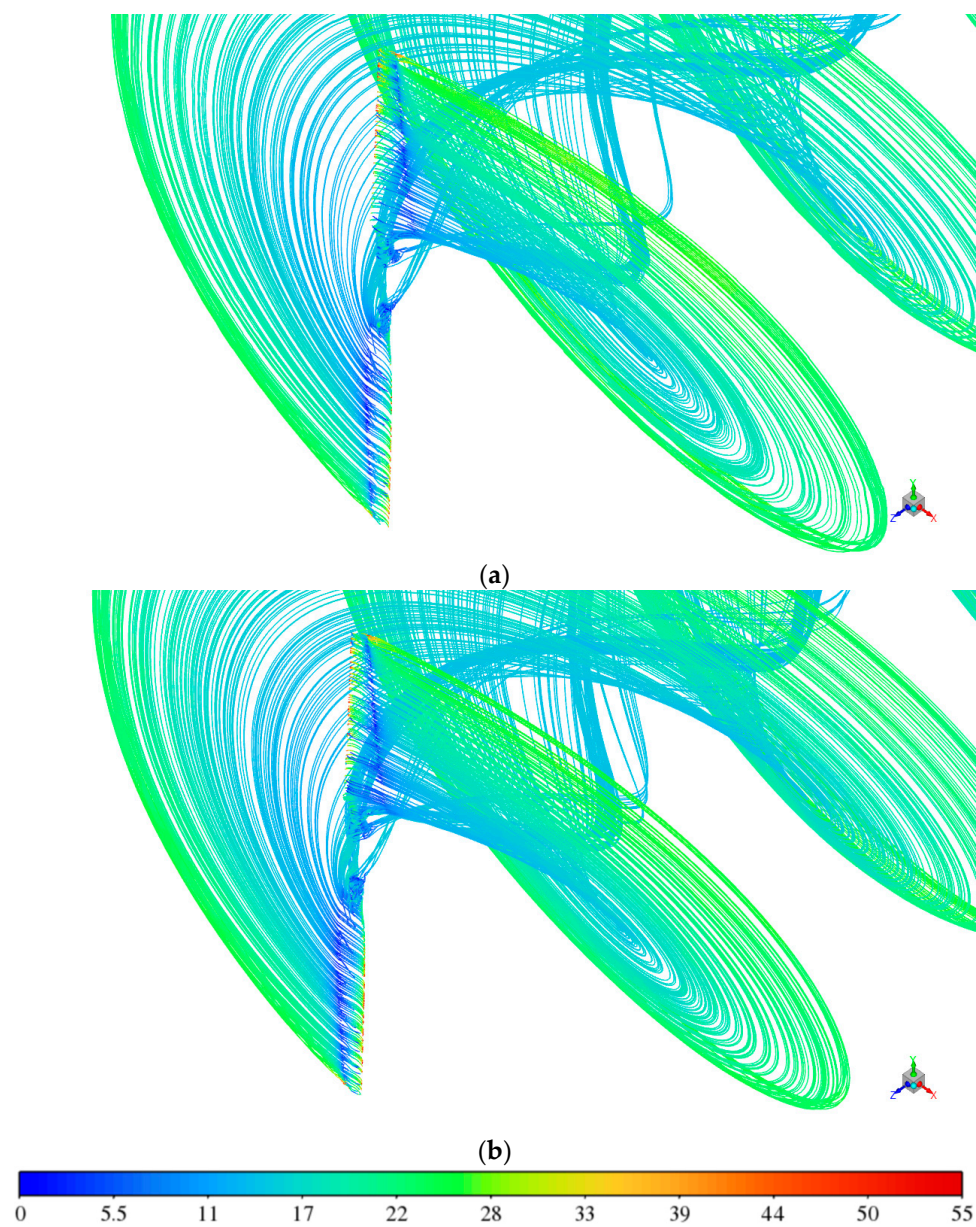


Figure 11. Three-dimensional streamline velocity at 15 m/s: (a) S809 airfoil; (b) modified airfoil.

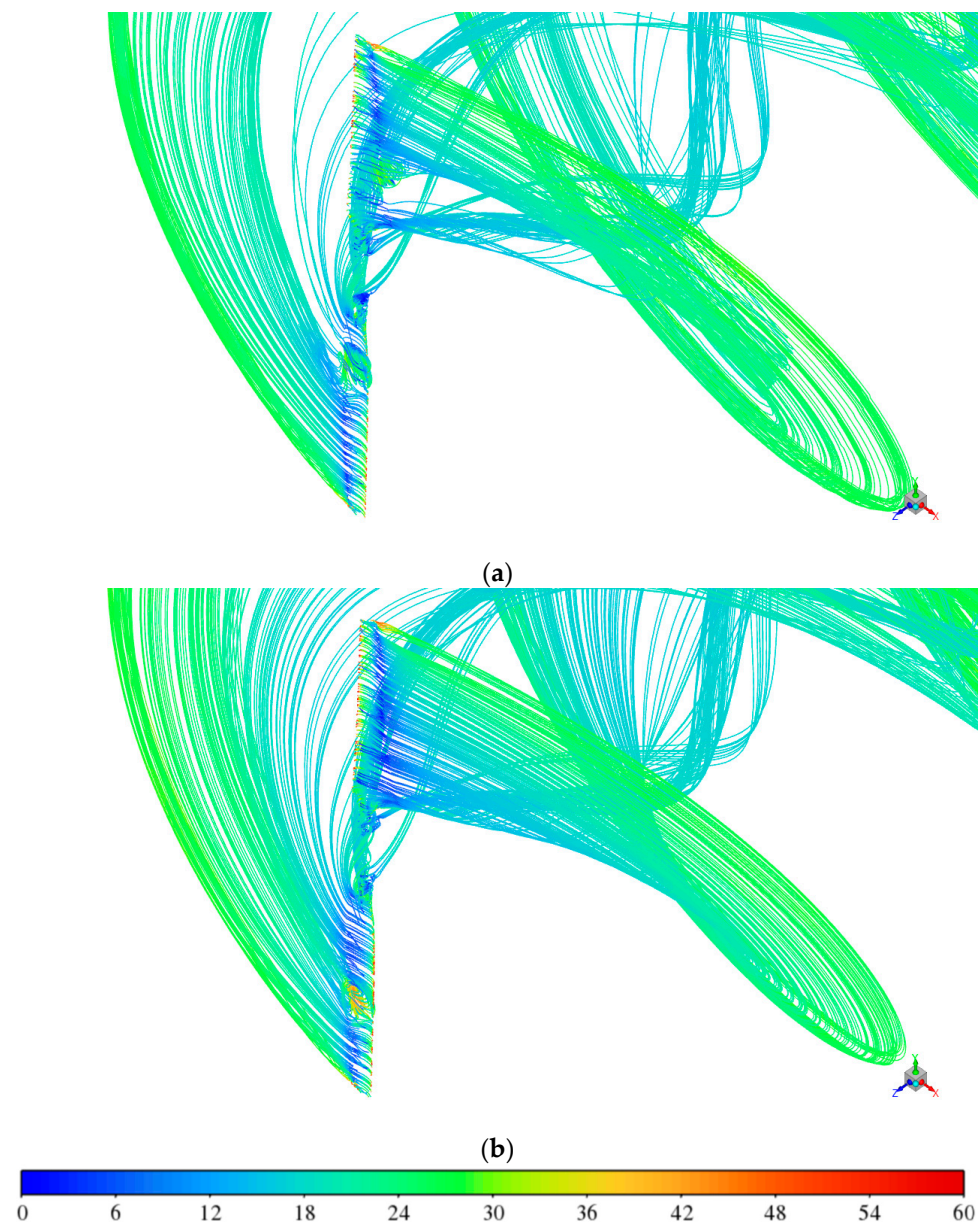


Figure 12. Three-dimensional streamline velocity at 20 m/s: (a) S809 airfoil; (b) modified airfoil.

4. Conclusions

This study investigated the possibility of achieving controlled wind turbine performance characteristics by means of an airfoil design. The location of the maximum thickness of the baseline airfoil was the main variable in the optimization. Three different modification values were applied across the span of the baseline blade as appropriate, taking into account the increase in velocity along the span of the blade. This was as a result of a data review suggesting a relation between the maximum airfoil thickness location and their designated Reynolds numbers. The results do indeed show that as the maximum thickness location of an airfoil moves forward along its chord, its Reynolds number decreases. This was not due to the maximum thickness, as has previously been reported. The methodology started with a detailed study of the relevant issues and fundamentals. The cost of wind turbine development and operation was found to be the main issue, and the blade airfoil design was found to be the key subject in obtaining a solution. The torque results were used to evaluate the performance of the optimization against the original performance of the baseline model. The results show potential towards the research objective, and a near-controlled power curve was achieved. This achievement is also relevant to applications

of all lift-based rotary blades, including aircrafts rotors and propellers. In-depth analysis showed the ability of the modified model to cope better with the change in velocity and AoA. However, other factors seemed to have influenced the results. For example, although it was the original S809 profile that was implemented near the tip of the modified blade, the velocity profile, and the pressure distributions accordingly, near the tip of the modified blade was influenced by modifications of the airfoil section at various span stations. However, it remains to be investigated why the lower Reynolds number airfoil with a higher lift coefficient generated reduced performance on lower inlet speeds. Nevertheless, the results show the ability of the modified model to better cope with the increase in speed, thus producing higher torque over a wider range of wind speeds.

Author Contributions: Conceptualization, D.S.; methodology, D.S.; software, D.S.; validation, D.S. and J.N.; formal analysis, D.S.; investigation, D.S.; resources, D.S.; data curation, D.S.; writing—original draft preparation, D.S.; writing—review and editing, J.N.; visualization, D.S.; supervision, J.N.; project administration, D.S.; funding acquisition, J.N. All authors have read and agreed to the published version of the manuscript.

Funding: This research received no external funding.

Data Availability Statement: The data presented in this study are available on request from the corresponding author.

Conflicts of Interest: The authors declare no conflicts of interest.

References

- Nelson, V. *Wind Energy: Renewable Energy and the Environment*, 2nd ed.; Vestas: Sweetwater, TX, USA, 2013.
- Fornarelli, R.; Shahnia, F.; Anda, M.; Bahri, P.A.; Ho, G. Selecting an economically suitable and sustainable solution for a renewable energy-powered water desalination system: A rural Australian case study. *Desalination* **2018**, *435*, 128–139. [\[CrossRef\]](#)
- Nelson, A. *Wind Hybrid Systems Technology Characterization: Report for National Renewable Energy Laboratory*; West Texas A&M University: Canyon, TX, USA; New Mexico State University: Las Cruces, NM, USA, 2001.
- Anderson, J. *Fundamental of Aerodynamics*; McGraw-Hill: Boston, MA, USA, 2001.
- Jureczko, M.; Pawlak, M.; Męzyk, A. Optimisation of wind turbine blades. *J. Mater. Process. Technol.* **2005**, *167*, 463–471. [\[CrossRef\]](#)
- Hubbard, H.H.; Shepherd, K.P. *Wind Turbine Technology: Fundamental Concepts of Wind Turbine Engineering*, 2nd ed.; ASME: New York, NY, USA, 2009.
- Manwell, J.F.; McGowan, J.G.; Rogers, A.L. *Wind Energy Explained: Theory, Design and Application*, 2nd ed.; John Wiley and Sons: Hoboken, NJ, USA, 2009.
- Hand, M.M.; Simms, D.A.; Fingersh, L.J.; Jager, D.W.; Cotrell, J.R.; Schreck, S.; Larwood, S.M. *Unsteady Aerodynamics Experiment Phase VI: Wind Tunnel Test Configurations and Available Data Campaigns*; National Renewable Energy Laboratory: Golden, CO, USA, 2001.
- Thumthae, C.; Chitsomboon, T. Optimal angle of attack for untwisted blade wind turbine. *Renew. Energy* **2008**, *34*, 960–1481. [\[CrossRef\]](#)
- Eggleston, D.M.; Stoddard, F. *Wind Turbine Engineering Design*; Van Nostrand Company: New York, NY, USA, 1987.
- Buhl, M. Wind Turbine Airfoil Families, National Renewable Energy Laboratory, 6 July 2012. Available online: <https://wind.nrel.gov/airfoils/AirfoilFamilies.html>. (accessed on 11 July 2021).
- Burton, T. *Wind Energy Handbook*; John Wiley and Sons: Hoboken, NJ, USA, 2001.
- Van Kuik, G.A. The Lanchester-Betz-Joukowsky Limit. *Wind Energy* **2007**, *10*, 289–291. [\[CrossRef\]](#)
- Schaffarczyk, A.P. *Introduction to Wind Turbine Aerodynamics, Green Energy and Technology*; Springer: Berlin/Heidelberg, Germany, 2014.
- Hasan, M.M.; El-Shahat, A.; Rahman, M. Performance investigation of three combined airfoils bladed small scale horizontal axis wind turbine by BEM and CFD analysis. *J. Power Energy Eng.* **2017**, *5*, 14–27. [\[CrossRef\]](#)
- Tu, J.; Yeoh, G.H.; Liu, C.; Tao, Y. *Computational Fluid Dynamics: A Practical Approach*; Butterworth-Heinemann: Oxford, UK, 2012.
- Song, Y.; Perot, B. CFD Simulation of the NREL Phase VI Rotor. *Wind Eng.* **2015**, *39*, 299–310. [\[CrossRef\]](#)
- Yelmule, M.M.; Vsj, E.A. CFD predictions of NREL Phase VI Rotor Experiments in NASA/AMES Wind tunnel. *Int. J. Renew. Energy Research.* **2013**, *3*, 261–269.
- Suárez, J.M.; Doerffer, P.; Szulc, O. *CFD Validated Technique for Prediction of Aerodynamic Characteristics on Horizontal Axis Wind Energy Turbines*; EWEA Offshore: Copenhagen, Denmark, 2015.
- Elfarra, M.A.; Sezer-Uzol, N.; Akmandor, I.S. NREL VI rotor blade: Numerical investigation and winglet design and optimization using CFD. *Wind Energy* **2014**, *17*, 605–626. [\[CrossRef\]](#)

21. 40- BY 80-/80- BY 120-Foot Wind Tunnels, NASA Ames Research Center. Available online: <https://rotorcraft.arc.nasa.gov/Research/Facilities/windtunnels.html> (accessed on 4 January 2024).
22. Sorensen, N.N.; Michelsen, J.A.; Schreck, S. Navier–Stokes predictions of the NREL phase-VI rotor in the NASA Ames 80 ft × 120 ft wind tunnel. *Wind Energy* **2002**, *7*, 151–169. [[CrossRef](#)]
23. Johansen, J.; Sørensen, N.N.; Michelsen, J.A.; Schreck, S. Detached-eddy simulation of flow around the NREL Phase VI blade. *Wind Energy* **2002**, *5*, 185–197. [[CrossRef](#)]
24. Chao, D.D.; Van Dam, C.P. Computational aerodynamic analysis of a blunt trailing-edge airfoil modification to the NREL Phase VI rotor. *Wind Energy* **2007**, *10*, 529–550. [[CrossRef](#)]
25. Rikui, Z.; Wu, V.D.J.-Z. Aerodynamic characteristics of wind turbine blades with a sinusoidal leading edge. *Wind Energy* **2012**, *15*, 407–424.
26. Hsu, M.C.; Akkerman, I.; Bazilevs, Y. Finite element simulation of wind turbine aerodynamics: Validation study using NREL Phase VI experiment. *Wind Energy* **2014**, *17*, 461–481. [[CrossRef](#)]
27. Simms, D.; Schreck, S.; Hand, M.; Fingersh, L.J. *NREL Unsteady Aerodynamics Experiment in the NASA-Ames Wind Tunnel: A Comparison of Predictions to Measurements*; National Renewable Energy Laboratory: Golden, CO, USA, 2001.
28. Wei, L.; Yu, L.; Leilei, J.; Haoming, L.; Shuo, L.; Yunfei, C.; Qiaoyue, Y. Effect of circumferential spokes on the rotating stall flow field of mixed-flow pump. *Energy* **2024**, *290*, 130260.

Disclaimer/Publisher’s Note: The statements, opinions and data contained in all publications are solely those of the individual author(s) and contributor(s) and not of MDPI and/or the editor(s). MDPI and/or the editor(s) disclaim responsibility for any injury to people or property resulting from any ideas, methods, instructions or products referred to in the content.