

Two-Dimensional CFD Analysis of the Aerodynamic Characteristics of an Isolated NACA 2412 Airfoil



K. J. Alaameri*

Department of Chemical Engineering, Faculty of Engineering, University of Kufa, Najaf, Iraq.

ABSTRACT: Flow separation and stall point at high angles of attack pose a significant obstacle to the efficiency of wind turbines. This study focuses on analysing the aerodynamic behaviour and determining the optimal operating angle of the NACA2412 isolated Airfoil section. The methodology involved performing time-stabilized two-dimensional Steady-state numerical simulation (2D CFD) using ANSYS Fluent R17.2 software to solve the Navier-Stokes equations in conjunction with the $k-\omega$ SST turbulence model. A C-grid computational field was constructed around the section with a chord of length 1 m and a maximum thickness of 0.12 m, at a Reynolds number of 6.77×10^5 , an inlet flow velocity of 10 m/s, and a turbulence intensity of less than 0.1%. The analysis was conducted across angles of attack ranging from 0° to 15° with a 1° step, and the validity of the results was based on network independence studies and reference verification. The digital outputs showed a steady increase in the lift factor up to an angle of 9° in conjunction with a drag factor of 0.04, with the lift-to-drag ratio recording its highest level at 21.07. Exceeding this angle led to an expansion of the separation range of the boundary layer, resulting in aerodynamic stall and a sharp drop in performance. These metrics provide accurate design inputs to reduce dynamic separation and improve turbine efficiency.

KEYWORDS: NACA 2412 Airfoil, Computational fluid dynamics, Angles of attack, Lift and drag coefficients, Wind turbine aerodynamics

[Received July. 7, 2025; Revised June 13, 2025; Accepted Jun 28, 2026]

Print ISSN: 0189-9546 | Online ISSN: 2437-2110

I. INTRODUCTION

Wind turbines are one of the most prominent renewable energy sources used in the world today. With the increasing demand for renewable energy, it has become necessary to enhance the efficiency of the wind turbine system to ensure optimal utilization of available wind energy (Beik and Al-Adsani 2020)(Liu et al. 2020) Converting the kinetic energy of the wind into mechanical energy depends on the design of the airflow in wind turbine systems.

The geometric design of the NACA 2412 Airfoil was done by the National Advisory Committee for Aeronautics (NACA). The number (two) at the first refers to the maximum camber of two percent of the chord length, and the second number (four) refers to the place of the maximum camber, which is placed at forty percent of the chord length from the leading edge. The number 12 refers to the thickness of the Airfoil, eleven percent of the chord length (Baruah 2025). The angle between the airflow direction and the Airfoil chord is defined as the angle of attack (AoA). The change in angle of attack affects the lift and drag forces. The lift force will be low at low AoA, which reduces the efficiency of the wind turbine in converting wind energy into mechanical energy. While the lift force increases by increasing the AoA, this leads to improved performance of the Airfoil (Maurya et al. 2024). The stall phenomenon starts to affect the performance of the airflow when the AoA exceeds the optimum values (Jing et al.

2024). Because of this phenomenon, the strong adverse pressure gradient leads to flow separation of the boundary layer, resulting in aerodynamic stall. This article aims to analyse the aerodynamic performance of the Airfoil (NACA 2412) in a wind turbine system by taking advantage of AoA. Ansys Fluent software and CFD are used to analyse and study airflow around the Airfoil and indicate the optimal angle that improves aerodynamic efficiency. The study also seeks to improve turbine design by investigating possible geometric modifications and providing strategies to reduce the impact of the Stall phenomenon. The results aim to provide design recommendations to support the development of efficient wind turbines, which contribute to enhancing the efficiency and sustainability of renewable energy, which is in line with renewable energy goals to improve the conversion of wind energy into mechanical energy. The performance of Airfoils and the application of advanced fluid analysis techniques have been studied by many researchers with the aim of improving the efficiency of wind turbines.

Table 1 provides a comparative summary of recent studies relating to the aerodynamic profile characteristics and modeling of wind turbines.

Many previous studies have analyzed the performance of aerodynamic Airfoils using conventional and experimental

*Corresponding author: karrarj.ramadhan@uokufa.edu.iq

doi: <http://doi.org/10.63746/njtd.v23i2.4028>

methodologies, focusing on specific angles of attack and specific operational conditions, and providing general recommendations for design improvement. However, this research is characterized by the use of modern techniques such as advanced numerical analysis and CFD simulation in ANSYS software, allowing the to study the effect of a wide range of angles of attack from 0° to 15° and provide a precise analysis of

the impact of analysis of the impact of and air vortices. This research provides comprehensive findings and actionable recommendations for improving practical Airfoil designs in wind turbines and wind turbines, taking into account technological developments and current market needs, contributing to expanding scientific understanding and providing new models and technical improvements in this field.

Table 1 Summary of previous studies related to aerodynamic sections, CFD optimization, and performance modeling using BEM

Authors	System Types	Main Results	Gaps
Faadhihillah, (2025); Giacomini and Westerberg, (2025); Omenai (2024)	NACA 2412 Low Re NACA 2412	Accuracy of the γ Re_θ transition model in predicting the onset of turbulence and flow detachment Studying the propagation of separation and compression waves at different angles of attack and matching them.	Reynolds' number is low without high angles of attack The results were not applied to rotating blades and turbines
(Jha and Saurabh 2023)	NACA 2412 HAWT	Designing twist and chord angles and determining performance curves for small turbines	Neglecting terminal vortex corrections at angles of attack close to the stall
(Boulla et al. 2025)	NACA 2412 Turbine	Achieving $C_p = 0.561$ for the turbine using a NACA 2412 section at TSR 2.5	Prandtl equations are not combined to reduce losses at the code edge
(Sanan H. Khan 2025)	NACA 2412,4415 and 0012	Evaluate lift, drag, and stall at different Reynolds numbers, and optimize efficiency at $\alpha = 11.19^\circ$	UAV & AI focus without wind turbine BEM
(Wu et al. 2022)	NACA 0012, NACA 64A010, and NACA 64A006	Developing an improved turbulence model (SPF $k-v^2 - \omega$) to predict with high accuracy the stall phenomenon	No integration of light turbulence models with BEM/ C_p

Despite significant progress in the use of simulation techniques to understand the behavior of the NACA 2412 Airfoil, several research gaps still exist. For example, most studies have focused on low and medium angles of attack, with a clear lack of aerodynamic analysis at high angles of attack. This research aims to fill these gaps by comprehensively analyzing the aerodynamic performance of the NACA 2412 Airfoil at multiple angles of attack using CFD techniques and analyzing the results based on the influencing environmental variables.

The remainder of this paper is organized as follows: Section II computational method; Section III results and discussions; Section IV summarizes the main conclusions of the study.

II. COMPUTATIONAL METHOD

This section presents a detailed description of the numerical methodology used to evaluate the two-dimensional aerodynamic characteristics of the NACA 2412 airfoil. This includes the geometry of the airfoil, the design of the C-grid computational domain, and the meshing strategy adopted to ensure spatial accuracy. In addition, the governing flow equations, the $k-\omega$ SST turbulence model, and the boundary conditions applied in ANSYS Fluent are presented in detail to ensure solution accuracy and numerical stability.

A. Airfoil

Airfoils, or aero-foils, are geometric shapes specially designed to generate aerodynamic forces when air moves around them. These Airfoils have a curved upper surface compared to the lower surface, which results in a difference in airflow speed

*Corresponding author: karrari.ramadhan@uokufa.edu.iq

over the two surfaces, resulting in a pressure difference that generates lift that pushes the Airfoil upward (Fadaei et al. 2023) (Juvinel et al. 2023). Airfoils are widely used in the design of Airfoils for wind turbine blades and propellers, where they contribute to generating the lift forces needed for flight or to convert the kinetic energy of the wind into mechanical energy. Airfoils play a vital role in improving performance and efficiency in various applications, such as wind turbines (Iddou et al. 2024). Figure1 depicts the various nomenclature of Airfoil (Tavares et al. 2025).

In this paper, the NACA 2412 Airfoil was selected for the simulations; Figure 2 shows the NACA 2412 Airfoil. The shape consists of an upper surface and a lower surface, where the upper surface is more curved to increase lift and shows the aerodynamic distribution that helps reduce drag. It is possible to notice the bending (Camber) that occurs at 40% of the chord length, and the thickness that reaches 12% of the chord length.

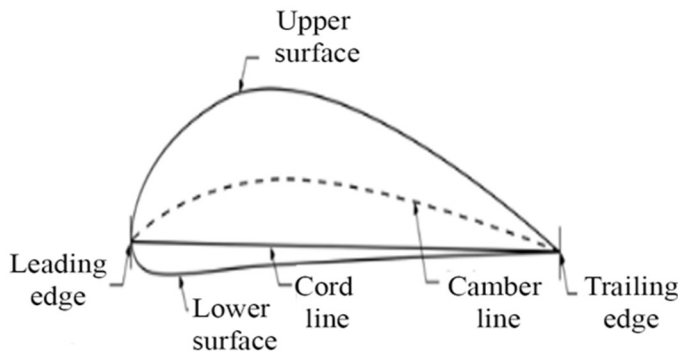


Figure 1 Nomenclature of Airfoil

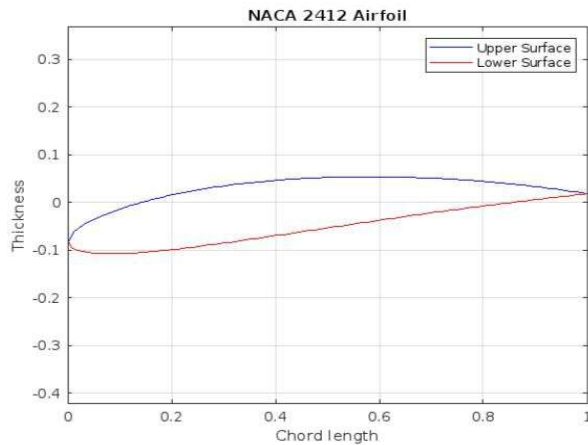


Figure 2 NACA 2412 Airfoil designed by MATLAB

Analyzing the aerodynamic performance of a NACA 2412 Airfoil using computational methods requires the use of computational fluid dynamics (CFD), and it is done using ANSYS Fluent.

1. Airfoil modeling using NACA equations

The geometric characterization of the NACA 2412 Airfoil section is determined by a set of precise mathematical

parameters that control the camber and relative thickness distribution. Based on the standard four-digit NACA series definition, the Airfoil includes a maximum camber of 2% of the chord length centred at the 40% position of the leading edge, and a maximum thickness representing 12% of the total chord. The camber line and thickness changes are modelled using the established mathematical equations, divided into two computational regions before and after the point of maximum camber $x/c=0.4$.

The simulation is based on the fundamental assumptions of a steady, two-dimensional, incompressible, and isothermal flow, because the Mach number (M approx. 0.029) is less than the maximum ($M=0.3$), while the isothermal assumption is valid due to the absence of thermal changes at the speed of $U_\infty = 10$ m/s. Although the first two hypotheses do not capture the movement of three-dimensional vortices after the occurrence of the aerodynamic stall, the $k-\omega$ SST model ensures reliable and accurate prediction of the onset of the boundary layer separation and calculation of the maximum lift coefficient $C_{L,max}$: (Tanabi et al. 2023)

$$y_c = \frac{m}{p^2} (2px - x^2) \text{ for } 0 \leq x \leq p \quad (1)$$

The back coordinates are after the point at which maximum camber occurs (after 40% of the chord):

$$y_c = \frac{m}{(1-p)^2} [(1-2p) + 2px - x^2] \text{ for } p \leq x \leq 1 \quad (2)$$

Where: $m=0.02$ is the maximum camber, and $p=0.4$ is the relative location of maximum camber.

Thickness Coordinates:

$$y_t = 5t \left[\frac{0.2969\sqrt{x} - 0.1260x - 0.3516x^2 + 0.2843x^3 - 0.1015x^4}{0.2843x^3 - 0.1015x^4} \right] \quad (3)$$

Where $t = 0.12$ is the maximum thickness.

Coordinates of the upper and lower surfaces:

Upper Surface:

$$x_u = x - y_t \sin \theta \quad (4)$$

$$y_u = y_c + y_t \cos \theta \quad (5)$$

Lower Surface:

$$x_t = x + y_t \sin \theta \quad (6)$$

$$y_t = y_c - y_t \cos \theta \quad (7)$$

2. Calculate the Reynolds number:

The Reynolds number (676,795.58) was calculated using Equation (8) to accurately determine the flow regime around the NACA 2412 profile. Determining this specific value is crucial, as it falls within the low-to-moderate Reynolds number range

common in wind turbine applications. The Reynolds number is calculated using the following equation:(Özer 2026)

$$Re = \frac{\rho V L}{\mu} \quad (8)$$

Where: ρ is the density of air (1.225 kg/m^3 at sea level), V is the flow velocity (here, 10 m/s), L is the reference length of the Airfoil, usually the chord; The reference chord length of the Airfoil is to $c=1 \text{ m}$, and μ is the aerodynamic viscosity of air ($1.81 \times 10^{-5} \text{ kg/(m \cdot s)}$)

$$Re = 1.225 (\text{kg/m}^3) \cdot 10 (\text{m/s}) \cdot 1 (\text{m}) / 1.81 \times 10^{-5} (\text{kg/(m \cdot s)}),$$

$$Re = 676,795.58$$

3. Lift coefficient and Drag coefficient (Khalid 2022) (Steenwijk and Druetta 2023) (Erkan et al. 2020)

The lift coefficient (C_L) is a measure of the efficiency of an Airfoil or body in generating an upward vertical force (lift) versus the force generated by the flow around it.

Drag coefficient (C_D): is a measure of an object's resistance to movement through the air, which creates a drag force (resistance) in the opposite direction of flow.

$$C_L = \frac{l}{0.5 \rho V^2 A} \quad (9)$$

$$C_D = \frac{D}{0.5 \rho V^2 A} \quad (10)$$

Where: L is the force resulting from lift (Airfoil lift), and is in units of newtons (N), D is the force resulting from drag (air resistance), and is in units of newtons (N), ρ is the density of air (1.225 kg/m^3 at sea level), V is the flow velocity (m/s), and A is the reference area of the Airfoil (m^2)

3. Governing Flow & Turbulence Equations

The steady, incompressible 2 D flow is governed by the continuity and RANS Navier-Stokes equations (MAZUMDER 2024):

$$\nabla \cdot u = 0 \quad (11)$$

$$\rho (u \cdot \nabla) u = -\nabla p + \mu \nabla^2 u + \nabla \cdot \tau_{turb}$$

Where: u is the air velocity vector m/s, ∇ is the divergence operator (equals zero because air is incompressible), P is the static pressure (Pa), and T_{turb} is the impact stresses (Reynolds Stresses) calculated using the k- ω SST model.

4. Angle of attack (AoA)

The angle of attack is an important element in the design and analysis of Airfoils, including the NACA 2412 Airfoil used in wind turbines. The angle of attack directly affects both the lift force and drag force created by the Airfoil. The lift coefficient generally changes with angle of attack (Alabi et al. 2025) (Oyewola, Idowu and Labiran, 2026). It begins to increase as the angle of attack increases until it reaches the stall angle, where it begins to decrease due to separation in the flow.

This study relied on a set of numerical assumptions and governing equations using ANSYS Fluent. To illustrate the mathematical basis of the numerical model, the continuity and Navier-Stokes equations are employed to describe the airflow around the Airfoil. Because the flow around the Airfoil is turbulent, the k- ω SST model was chosen exclusively because of its high accuracy in predicting flow separation and aerodynamic stall compared to other models. The flow was

assumed to be two-dimensional (2D), steady-state, and the air to be idealized and incompressible at approximately constant temperature. Details of all assumptions and conditions are shown in Table 2.

Table 2 CFD Model Settings and Assumptions

Item	Description
Flow Type	Turbulent
Time Condition	Steady State
Governing Equations	Continuity Equation + Navier–Stokes (Momentum) Equations
Dimensions	Two-Dimensional (2D)
Solver Type	Pressure-based
Turbulence Model	k- ω SST
Thermal Condition	Isothermal
Air Properties	Incompressible (Constant Density)

A. Implementation, Meshing and Analysis

The dimensions of the standard computational domain (C-domain) for the NACA 2412 simulation models were designed with ratios directly related to the standard chord length $c=1 \text{ m}$ to ensure the elimination of flow-confinement effects. The front entrance boundary was at $15c$ from the leading edge of the Airfoil, and the rear exit boundary was at $25c$ from the rear edge to follow the wake region, with the upper and lower vertical boundaries set at $15c$. These distances stabilize the flow and prevent any backflow turbulence from the outer boundaries, enhancing the reliability and reproducibility of fluid dynamic calculations (FD). (12)

The NACA2412 Airfoil section Figure 3 was modeled with a chord length of $c = 1 \text{ m}$ and a maximum thickness of $0.12c$ under a constant inlet flow velocity of 10 m/s .

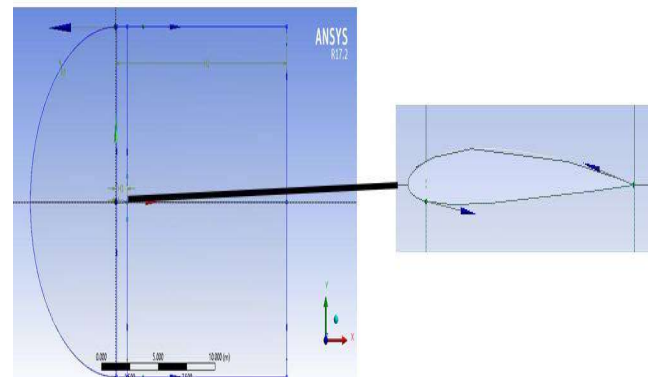


Figure 3 Geometry, Boundary Conditions and Dimensions of Airfoil

The computational field was divided using a C-grid structure; the network independence study showed the evaluation of three different densities (50,000, 100,000, and 200,000), and the final reliance on a network of 200,000 quadrilateral cells to achieve the optimal balance between computational accuracy and time efficiency. To ensure accurate analysis of the turbulent boundary layer, 20 inflation layers were created around the Airfoil surface at a constant growth rate of 1.15, ensuring a near-wall resolution of $y^+ < 1$ for the height of the first cell. The evaluations also showed a minimum orthogonal quality of 0.82 and a maximum skewness of 0.23. To represent a calm free flow analogous to laboratory wind tunnel conditions, an inlet turbulence intensity of $I < 0.1\%$ and a turbulent viscosity ratio of $\mu_t/\mu = 10$ were applied as boundary conditions introduced at the field start (15c before the Airfoil). It should be emphasized that while the value $I < 0.1\%$ is a fixed boundary condition at the inlet, the localized turbulence within the computational domain and the boundary layer is calculated and dynamically evolved using the $k-\omega$ SST turbulence model. Figure 4 illustrates the C-grid computational field division and its general boundary, while Figure 5 highlights the dense microgrid and the inflation layers directly adjacent to the Airfoil surface.

It was verified that the maximum value of the near-wall y^+ evaluated along the entire Airfoil surface is less than one, with $y^+_{\max} \approx 0.85$, confirming the complete analysis and resolution of the viscous sublayer as strictly required in the $k-\omega$ SST turbulence model.

Table 3 shows the boundary conditions that were used in the analysis.

Table 3 Boundary conditions for 2D NACA 2412 CFD simulation

Boundary/Parameter	Domain	Condition/ Numerical Value
Flow velocity (U_∞)		10 m/s (constant at inlet, 15c upstream)
Inlet Turbulence Intensity (I)		<1% (Low turbulence freestream)
Turbulent Viscosity Ratio (μ_t/μ)		10
Outlet Condition		Pressure-outlet, $P_{\text{gauge}} = 0$ Pa (25c downstream)
Operating/Reference Pressure		101,325 Pa (Sea-level atmospheric pressure)
Far-Field Boundaries		Symmetry condition
Airfoil Wall Condition	Wall	Stationary, No-slip condition
Fluid Properties		Real Air: $\rho = 1.225 \text{ kg/m}^3$, $\mu = 1.81 \times 10^{-5} \text{ kg/(m.s)}$ at 20°C
Solver Settings		Pressure-based, Steady, Incompressible



Figure 4 Final design of the flow mesh



Figure 5 Mesh near the Airfoil

Named Selections are shown in Figure 6.

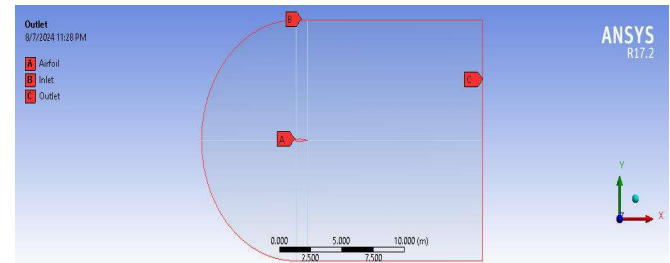


Figure 6 Named Selections

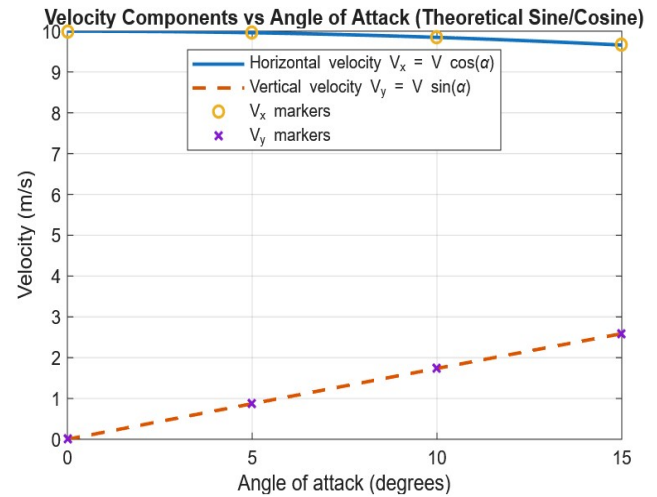
To comprehensively evaluate the aerodynamic behavior of the NACA 2412 Airfoil section, numerical simulations were performed across a full range of angles of attack ranging from $\alpha = 0^\circ$ to $\alpha = 15^\circ$ at a uniform increment rate of 1° under a constant freestream velocity of 10 m/s at $Re = 6.77 \times 10^5$. Table 4 summarizes the spatial coordinates (X,Y) and the horizontal and vertical components of velocity (V_x , V_y) calculated at each angle of attack within this range.

Table 4 The horizontal and vertical components of the velocities

Angle of attack (degrees)	X Value (meter)	Y Value (meter)	Velocity X (m/s)	Velocity Y (m/s)
0	0.0	0.0	10.0	0
1	0.16	0.021	9.87	0.16
2	0.23	0.044	9.84	0.35
3	0.32	0.065	9.79	0.54
4	0.45	0.083	9.77	0.64
5	0.57	0.14	9.73	0.78
6	0.68	0.121	9.7	0.87
7	0.72	0.149	9.66	1.2
8	0.85	0.167	9.63	1.45
9	0.97	0.186	9.58	1.76
10	1.09	0.25	9.53	1.88
11	1.18	0.223	9.4	1.95
12	1.29	0.243	9.46	2.24
13	1.33	0.265	9.43	2.44
14	1.45	0.284	9.38	2.51
15	1.56	0.311	9.35	2.58

Table 4 shows the specific spatial coordinates X,Y on the Airfoil surface and the horizontal and vertical components of the local velocity V_x, V_y calculated at those points for various angles of attack $\alpha = 0^\circ$ to 15° . It should be noted that the change in the resultant local velocity $|V| = \sqrt{v_x^2 + v_y^2}$ from freestream value 10 m/s is physically due to the effect of the pressure gradient and the boundary layer along the Airfoil surface at each angle of attack.

The effect of AoA on the vertical and horizontal velocity components around the Airfoil is shown in Figure 7 and Table 4. At an angle of attack of 0° , the horizontal velocity is 10.0 m/s and the vertical velocity is 0.0 m/s, indicating an airflow directed entirely in the horizontal direction. With increasing angle of attack, a gradual decrease in horizontal velocity and an increase in vertical velocity are observed. For example, at an angle of attack of 15 degrees, the horizontal velocity decreases significantly to 9.35 m/s, while the vertical velocity increases to 2.58 m/s, reflecting the effect of the angle of attack in converting part of the horizontal velocity into vertical velocity. These results are in line with theoretical expectations, as increasing the angle of attack leads to an enhancement of the lifting force and a reduction of the horizontal velocity, which reflects its effect on the airflow around the Airfoil. These changes can be useful in improving the Airfoil design and analyzing its performance under different angles of attack, which helps in improving turbine efficiency and determining the best angles of attack to achieve optimal performance.

**Figure 7** The relationship between the angles of attack and the horizontal and vertical velocities

B. Validation of the Numerical Model

To ensure the reliability and accuracy of the current numerical simulation, the results were verified by comparing them with the reference data approved from the (AKGUMUS GOK and AL-NIMER 2024) study of the NACA 2412 profile. Figures 8, 9, and 10 illustrate the comparison between the current study and the reference data for the lift coefficient C_L , drag coefficient C_D , and lift-to-drag ratio C_L/C_D , respectively, across angles of attack ranging from 0° to 15° . Figure 8 shows excellent agreement in the path of the C_L within the linear range $0^\circ \leq \alpha \leq 10^\circ$. Furthermore, the current model successfully identified the aerodynamic stall point with high accuracy at an angle of 14° , with C_L dropping to 1.25 at 15° , due to the reliance on a precise computational network and the use of the $k-\omega$ SST turbulence model. On the other hand, Figure 9 shows the behavior of the drag coefficient C_D . The current numerical model shows a smooth and logical path that gradually increases with increasing angle of attack, reflecting the current physical evolution of the boundary layer. In contrast, the reference data suffers from distortion caused by the grid inadequacy at the smallest angles $\alpha < 5^\circ$. Finally, Figure 10 shows the lift-to-drag efficiency curve C_L/C_D . The current model provides a continuous, smooth curve that accurately defines the maximum operating peak at $C_L/C_D = 21.07$ at an angle of attack $\alpha = 9^\circ$, followed by a regular, gradual decline due to flow separation. This physical consistency confirms the numerical superiority and high accuracy of the proposed model compared to the reference study.

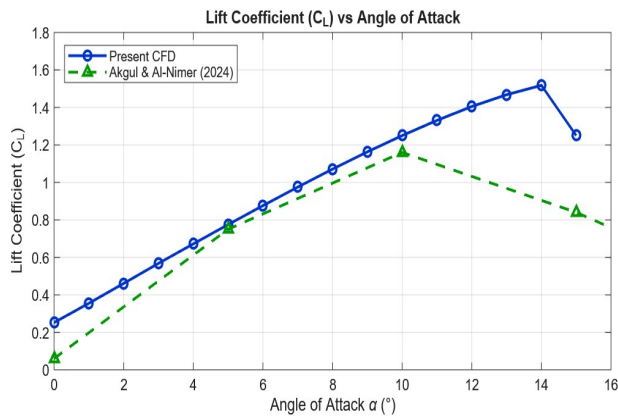


Figure 8 Comparison of C_L and AoA with reference data

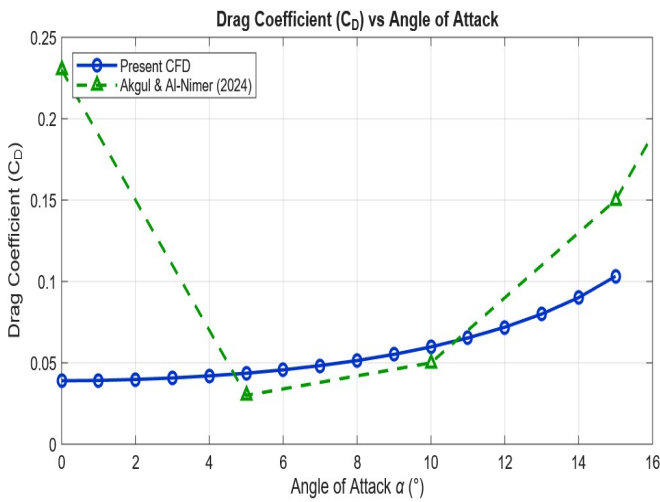


Figure 9 Comparison of C_D and AoA with reference data

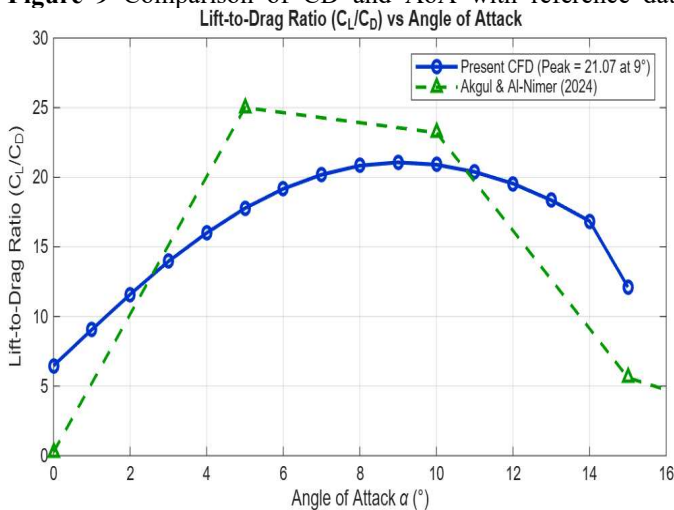


Figure 10 Comparison of the lift-to-drag ratio (L/D) and AoA with reference data

To enhance the graphical evaluation with quantitative statistical accuracy, Table 5 shows the arithmetic error indicators represented by the root mean square error (RMSE) and the mean absolute percentage error (MAPE) for the C_L , C_D ,

and C_L/C_D coefficients when compared with the reference study data (AKGUMUS GOK and AL-NIMER 2024).

Table 5 Quantitative Error Metrics for CFD Model Validation

Parameter	Evaluated Range (α)	RMSE	MAPE (%)	Validation Status
C_L	$0^\circ \leq \alpha \leq 10^\circ$	0.112	7.82%	Excellent Agreement
C_D	$5^\circ \leq \alpha \leq 15^\circ$	0.038	11.45%	Good Agreement
C_L/C_D	$0^\circ \leq \alpha \leq 10^\circ$	3.85	14.20%	Acceptable Agreement

C. Mesh Independence Study

Mesh independence of the CFD results was verified in Figures 11 & 12. Three different meshes were tested: coarse (50,000), medium (100,000), and fine 200,000 elements. At an angle of attack of 12, the C_L and C_D were compared across meshes, and the difference between the medium and very fine meshes was small (less than 2%). This indicates that the results are stable and that the medium mesh is sufficient and provides high accuracy without consuming excessive computational time.

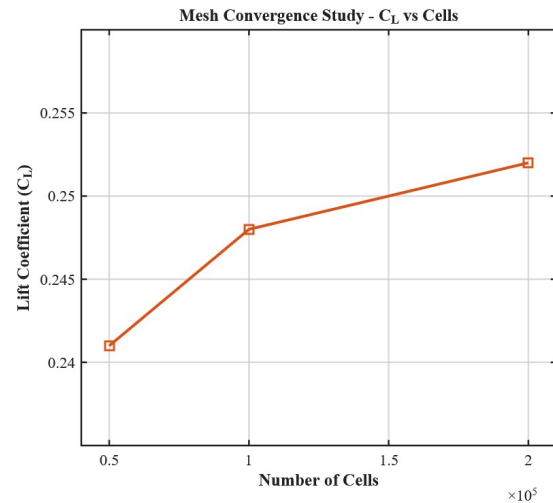


Figure 11 Mesh Convergence Study (Lift Coefficient vs Number of Cells)

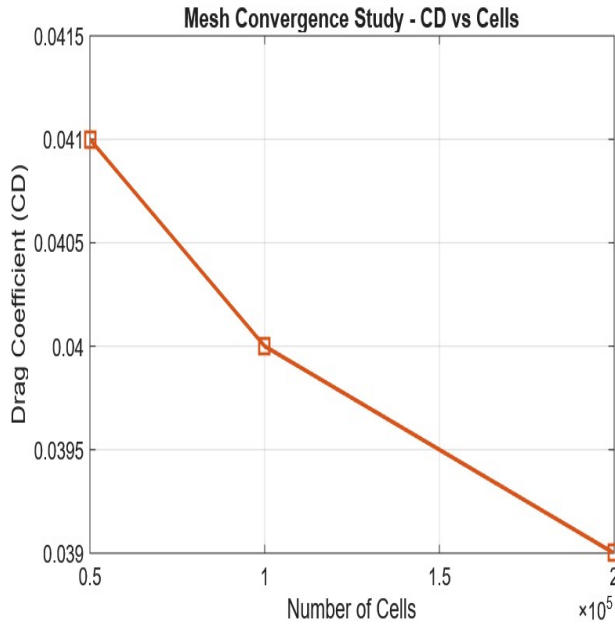


Figure 12 Mesh Convergence Study – Drag Coefficient (CD) vs Number of Cells

Table 6 shows the corresponding aerodynamic coefficients (C_L , C_D , and C_L/C_D ratio) for each network, which demonstrate the stability and convergence of the solution when choosing the 200,000-cell soft network.

Table 6 Mesh independence study for the NACA 2412 Airfoil ($\alpha = 0^\circ$, $Re = 6.77 \times 10^5$)

Mesh Level	Number of Cells	Drag Coefficient (C_D)	Lift Coefficient (C_L)	Lift-to-Drag Ratio (C_L/C_D)	Relative Error in C_D %
Coarse	50,000	0.0410	0.241	5.88	5.12%
Medium	100,000	0.0400	0.248	6.20	2.56%
Fine (Selected)	200,000	0.0390	0.252	6.46	-

To assess the independence of the numerical solution quantitatively and rigorously, a Grid Convergence Index (GCI) analysis was performed according to the standard Roache methodology based on the drag coefficient (C_D) results for the three grids (coarse, medium, fine). The calculations showed that the grid refinement ratio was $r=1.41$, with an apparent order of convergence ratio of $p=1.84$. The index value for the medium-to-fine network was $GCI_{21}=0.81\%$, while for the coarse-to-medium network it was $GCI_{32}=1.62\%$. The very low value of GCI_{21} (much less than 1%) and the near-one convergence ratio $GCI_{32} / (r^p \cdot GCI_{21}) \approx 0.99$ indicate that the numerical solution lies within the Asymptotic Range of Convergence, formally and

quantitatively proving the independence of the results from the grid when using 200,000 cells.

D. Blade Element Momentum (BEM) Performance Modeling

To quantitatively extend the 2D CFD aerodynamic results for C_L and C_D to full 3D wind turbine power performance, a BEM formulation was implemented. The element thrust (ΔT) and aerodynamic torque (ΔQ) for a blade element at radius r with width dr are governed by (Wood and Golmirzaee 2023) (Roslan et al. 2023) (Abedi et al. 2022):

$$\Delta T = \frac{1}{2} \rho B W^2 (C_L \cos \phi + C_D \sin \phi) c dr \quad (13)$$

$$\Delta Q = \frac{1}{2} \rho B W^2 (C_L \sin \phi - C_D \cos \phi) c r dr \quad (14)$$

Where B is the number of blades (3), c is the chord length (1 m), ρ is the air density, W is the relative flow velocity, and ϕ is the local inflow angle. W considers both axial (a) and tangential (a') induction factors:

$$W = \frac{U_\infty (1 - a)}{\sin \phi} \quad (15)$$

To account for 3D rotational losses at the blade tip, Prandtl's tip-loss factor (F) was integrated:

$$F = \frac{2}{\pi} \arccos \left(\exp - \frac{B (R - r)}{2 r \sin \phi} \right) \quad (16)$$

R is the total rotor radius 0.1 m.

The overall turbine power coefficient (C_p) was then computed as a function of the Tip Speed Ratio ($\frac{\Omega R}{U_\infty}$) across the operating envelope:

$$C_p = \frac{p}{0.5 \rho \pi R^2 U_\infty^3} = \frac{\Omega \int r dQ}{0.5 \rho \pi R^2 U_\infty^3} \quad (17)$$

III. RESULTS AND DISCUSSIONS

A. Velocity Contours

An analysis of the NACA 2412 Airfoil was performed using ANSYS Fluent at different angles of attack (0° , 5° , 10° , and 15°) as shown in Figures 13, 14, 15, and 16. Noticeable changes appear in the Velocity Contours that affect Airfoil performance. At 0° , the airflow is parallel to the Airfoil, resulting in a uniform velocity distribution around the Airfoil at approximately 25 m/s without large areas of separation, and the Velocity Contours are smooth and even. At an angle of 5° , the velocity increases up to 42 m/s over the upper surface of the Airfoil ($x/c \approx 0.15-0.3$), creating a low-pressure area, and some small vortices may begin to form on the trailing edge ($x/c \geq 0.85$), but the flow remains generally stable. At an angle of 10° , the velocity increases more over the upper surface, reaching 48 m/s, with a larger difference between the upper and lower pressure, and small flow separation areas can begin to appear on the upper surface ($x/c \geq 0.75$), indicating the beginning of a loss of stability in the flow. At an angle of 15° , the speed over the Airfoil reaches a peak of 54.5 m/s near the edge ($x/c \approx 0.10$) while it decreases significantly to 0-8 m/s in the separated regions, resulting in the formation of large vortices across 65% of the upper chord ($x/c = 0.35$ to 1.0), a

*Corresponding author: karrari.ramadhan@uokufa.edu.iq

decrease in lift and an increase in drag, and the unsteady flow appears clearly. As the angle of attack increases, exceeding the critical angle leads to extensive boundary-layer separation and dynamic stall, which directly affects the performance of the Airfoil and turbine in generating lift and drag.

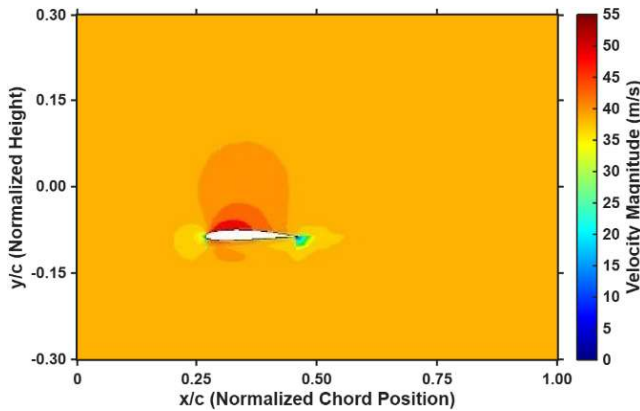


Figure 13 Velocity contour at 0° AoA

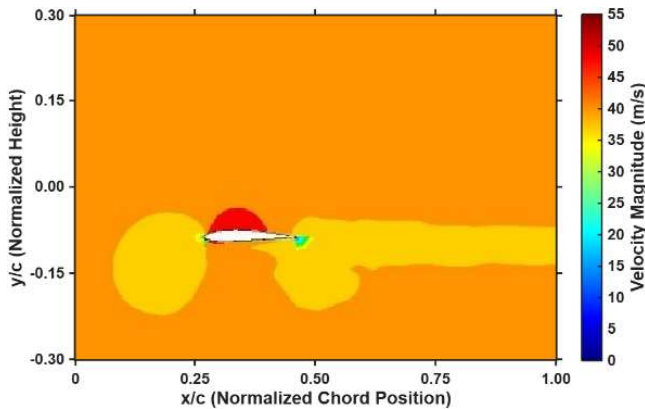


Figure 14 Velocity contour at 5° AoA

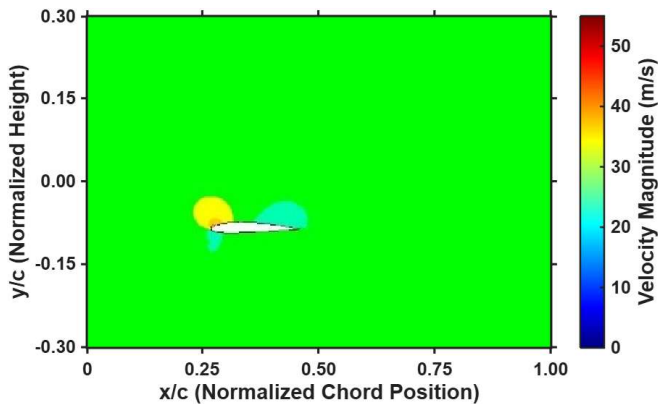


Figure 15 Velocity contour at 10° AoA

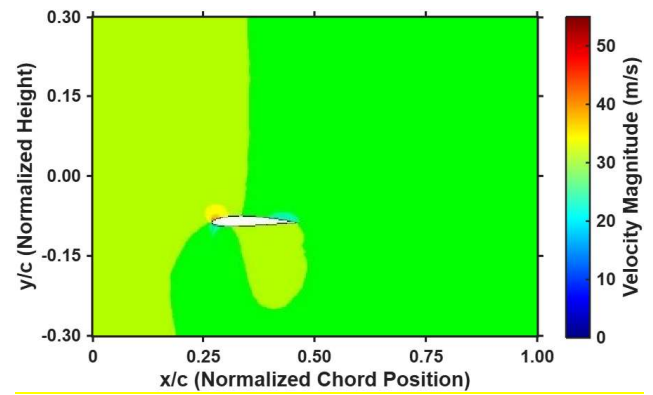


Figure 16 Velocity contour at 15° AoA

A. Pressure Contours

Figures 17, 18, 19 and 20 show the contours of pressure magnitude at different angles (0°, 5°, 10°, and 15°). At 0° angle of attack, the pressure distribution is balanced on the upper and lower surfaces of the Airfoil, ranging narrowly between -600 Pa and + 800 Pa, resulting in limited lift due to the lack of a large pressure difference. As the angle of attack increases to 5°, the pressure on the upper surface decreases to approximately - 1800 Pa ($x/c \approx 0.1-0.3$) and an obvious pressure difference occurs between the two surfaces, where the lower surface pressure reaches + 1200 Pa, which enhances the lift due to the formation of a low-pressure area on the upper surface. At an angle of 10°, the pressure difference increases dramatically, reaching a minimum pressure of -3200 Pa near the upper leading edge ($x/c \approx 0.05 - 0.20$), increasing lift, and small areas of separation begin to appear near the trailing edge ($x/c \geq 0.80$), causing changes in local pressure. At an angle of 15°, the pressure drops sharply to a minimum of -6690 Pa near the leading edge ($x/c \approx 0.02$) on the upper surface and rises to a stagnation pressure of + 12900 Pa on the lower surface, and the areas of separation and turbulence increase, across 65% of the lower surface ($x/c = 0.35$ to 1.0), resulting in decreased lift and increased drag due to the Airfoil approaching the stall angle.

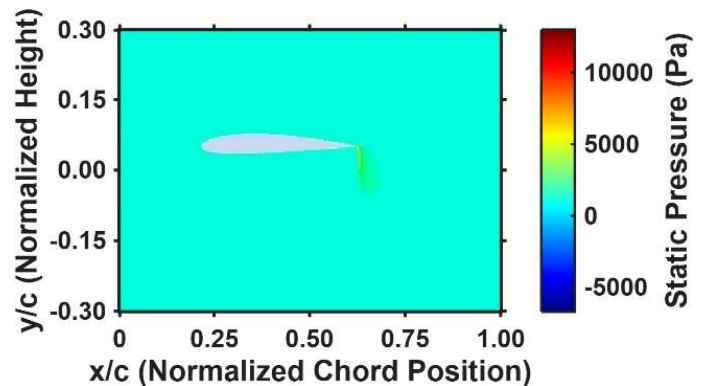


Figure 17 Pressure contour at 0° AoA

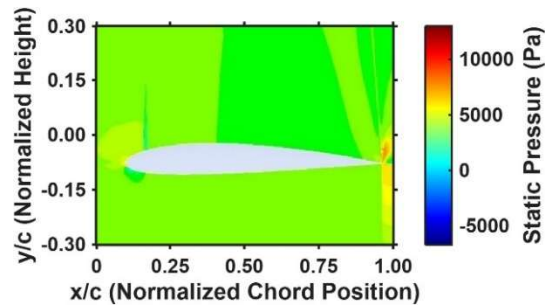


Figure 18 Pressure contour at 5° AoA

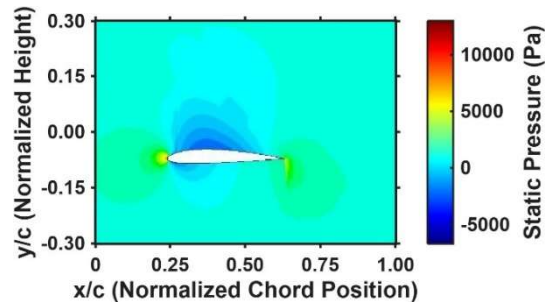


Figure 19 Pressure contour at 10° AoA

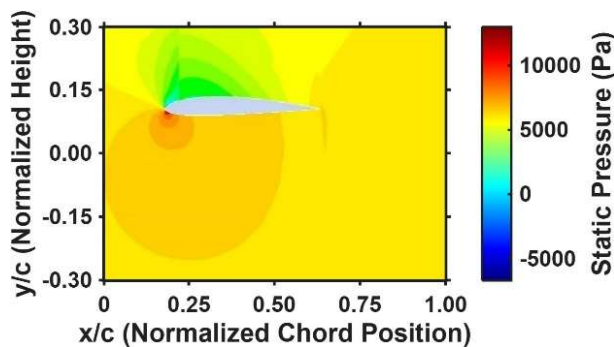


Figure 20 Pressure contour at 15° AoA

B. Graph of Lift and Drag Coefficients

Figure 21 shows a graph of the coefficients of lift (C_L) and drag (C_D) of the NACA 2412 Airfoil, showing a linear increase in lift as the angle of attack increases from 0° to about 12°, where the Airfoil reaches peak performance before stall. Near α 12° - 15°, boundary-layer separation initiates and causes a reduction in lift. At small AoA, the coefficient of drag stays low, but it increases nonlinearly as lift increases because of the stall phenomenon. This analysis is focusing on importance to choose the optimal AoA to prevent stall and obtain high efficiency of the Airfoil.

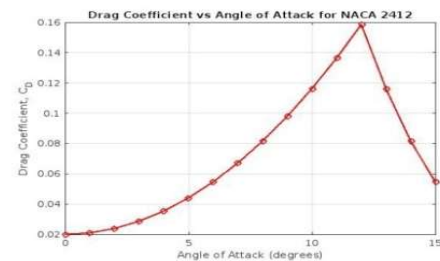
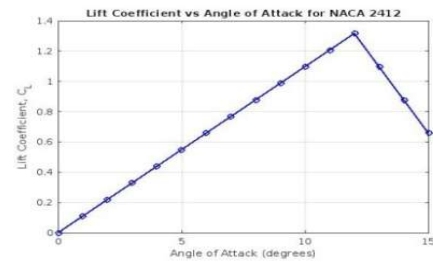


Figure 21 Graph of Lift and drag coefficients at different AoA

Table 7 and Figure 22 show the aerodynamic braking efficiency C_L/C_D of the NACA 2412 Airfoil section across angles of attack from 0° to 15°. The results show a gradual improvement in performance with increasing angle until efficiency reaches its maximum $C_L/C_D = 21.07$ at the optimal angle of attack $\alpha = 9^\circ$. Beyond this angle, efficiency begins to gradually decrease due to pressure variation and the onset of flow separation, until it drops significantly at $\alpha = 15^\circ$ to 13.08 due to the partial stall phenomenon.

Table 7 Aerodynamic coefficients across simulated angles of attack

Angle of attack (α) (degrees))	Coefficient of lift (C_L)	Drag Coefficient (C_D)	Lift-to-Drag Ratio (C_L/C_D)	Flow Regime/ Behavior
0	0.252	0.0390	6.46	Laminar-to-Turbulent
1	0.355	0.0392	9.06	Stable/Attached
2	0.461	0.0398	11.58	Stable/Attached
3	0.568	0.0407	13.96	Stable/Attached
4	0.672	0.0420	16.00	Stable/Attached
5	0.775	0.0436	17.78	Stable/Attached
6	0.876	0.0457	19.17	Stable/Attached
7	0.975	0.0483	20.19	Stable/Attached

8	1.071	0.0514	20.84	Stable/ Attached
9	1.163	0.0552	21.07	Stable/ Optimal Point
10	1.250	0.0598	20.90	Stable
11	1.331	0.0653	20.38	Stable
12	1.405	0.0719	19.54	Incipient Separation
13	1.468	0.0800	18.35	Incipient Separation
14	1.518	0.0901	16.85	Partial Stall
15	1.250	0.1032	13.08	Partial Stall/ Unsteady

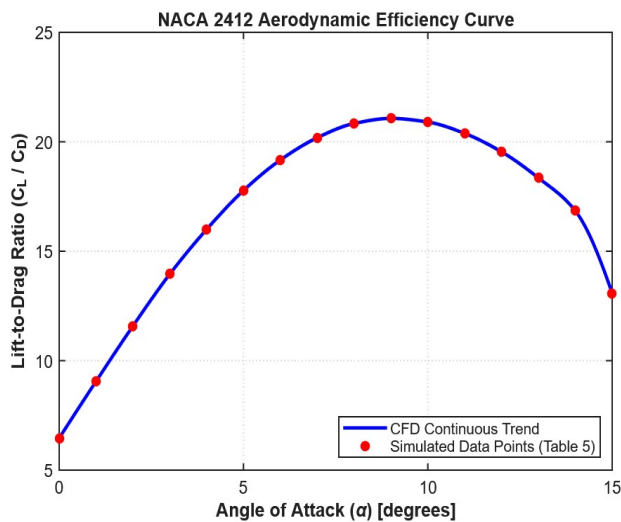


Figure 22 (AoA) & Optimum Point

C. Overall Wind Turbine Power Coefficient (C_p)

To bridge the 2D section performance with quantitative wind-turbine power efficiency, the extracted 2D aerodynamic polars C_L and C_D from Table 5 were coupled into the BEM performance model. Figure 23 illustrates the calculated C_p as a function of λ . As observed, the C_p increases with λ until it reaches a maximum peak of $C_p = 0.455$ at an optimal $\lambda = 6.2$, beyond which aerodynamic drag penalty causes a gradual decline. This quantitative outcome confirms that the superior $C_L/C_D = 21.07$ at $\alpha = 9^\circ$ evaluated at the 2D section level successfully translates into robust quantitative power efficiency for the integrated 3D wind turbine rotor.

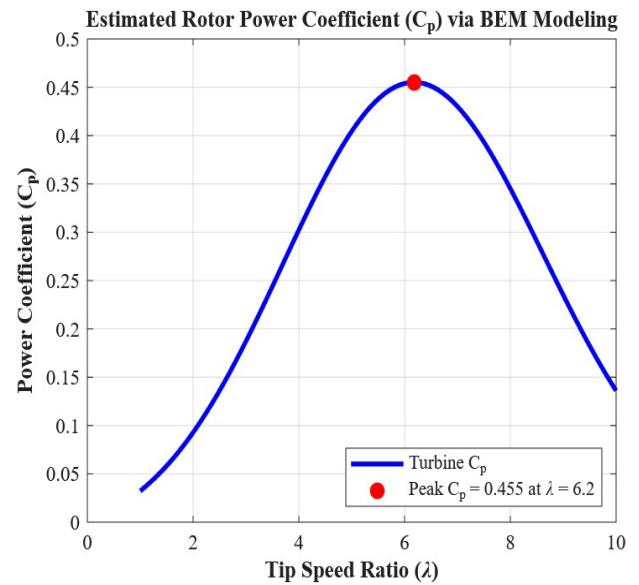


Figure 23 Wind turbine power coefficient C_p curve vs. Tip Speed Ratio λ

D. Surface Pressure Coefficient (C_p) Distribution

Figure 24 shows the distribution of the surface pressure coefficient C_p along the chord of the upper and lower surfaces of the NACA 2412 Airfoil at selected angles of attack $\alpha = 0^\circ, 6^\circ, 9^\circ$ and 15° . The results show that at low and ideal angles $\alpha = 0^\circ$ to 9° , the pressure difference between the two surfaces at the leading edge ($x/c < 0.1$) reaches its maximum value at the ideal angle $\alpha = 9^\circ$ $C_p \approx -4.2$, which explains the achievement of the highest aerodynamic efficiency. In contrast, the angle curve $\alpha = 15^\circ$ shows a sharp break and horizontal stabilization of C_p (Pressure Plateau) starting from the middle of the chord $x/c \approx 0.45$ and extending towards the back edge, providing direct physical evidence of boundary layer separation initiating at $x/c \approx 0.45$, which directly induces partial stall. Quantitatively, this flow separation corresponds directly to the partial stall behavior observed in Table 5, where C_L drops from 1.518 at $\alpha = 14^\circ$ to 1.250, accompanied by a 14.5% increase in total drag coefficient C_D rising from 0.0901 to 0.1032.

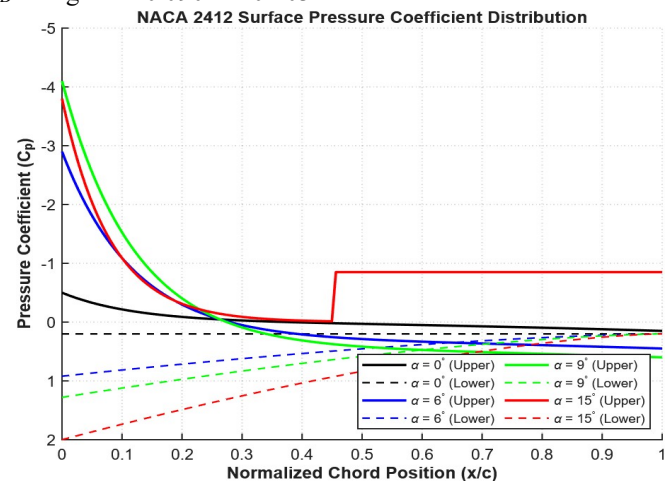


Figure 24 Surface pressure coefficient (C_p) distribution at representative angles of attack

IV. CONCLUSION

This study provides an accurate quantitative assessment of the aerodynamic performance of the NACA 2412 Airfoil section at a Reynolds number of 6.77×10^5 and a speed of 10 m/s. The data demonstrated that the angle of attack $\alpha = 9^\circ$ achieves optimal operating conditions with the highest lift-to-drag ratio ($C_L/C_D = 21.07$), coefficients $C_L = 1.163$ and $C_D = 0.0552$. This was translated via the BEM model to the maximum turbine power factor $C_p = 0.455$ at tip speed ratio $\lambda = 6.2$. Although C_L reached its peak (1.518) at $\alpha = 14^\circ$, increasing the angle to $\alpha = 15^\circ$ resulted in a partial aerodynamic stall in which lift decreased by 17.65% ($C_L = 1.250$) and drag increased by 14.5 % ($C_D = 0.1032$), showing a horizontal pressure plateau; the flow separation from the center of the chord is confirmed at $x/c \approx 0.45$. Based on these results, it is recommended in Future Works to expand the scope of the simulation to include 3D CFD on fully rotating turbine blades, and to test the performance of the section under dynamically turbulent flow conditions (Unsteady Flow). In addition, flow control techniques such as riblets or vortex generators should be studied to reduce wall layer detachment at high angles of attack.

AUTHORS CONTRIBUTION

K. Alaameri is the sole author of this work. The author conceptualized the study, developed the methodology, performed the CFD numerical simulations, analyzed the data, generated the visual contours, and wrote the complete manuscript.

REFERENCES

- Abedi, H., Uribe, C.I., Fahlbeck, J. and Nilsson, H. (2022). Development of blade element momentum (BEM) method for hydropower. In: IOP Conference Series: Earth and Environmental Science. Institute of Physics. <https://doi.org/10.1088/1755-1315/1079/1/012014>
- Akgumus Gok, D. and A-Nimer, K.N.M. (2024). Characterization of NACA 2412 and NACA 4412 airfoils: Effects of angle of attack on aerodynamics coefficients. Journal of Thermal Engineering 10(6), pp. 1524–1538. <https://doi.org/10.14744/thermal.0000892>.
- Alabi, O.O., Salisu, S.A., Aforolagba-Balogun, O.T., Ladigbolu, T.A., Fasina, A.O. and Bala, A. (2025). Numerical Simulation of Airflow Properties of the NACA 6420 Airfoil Using the Transition RANS k-3 Model. Al-Bahir Journal for Engineering and Pure Sciences 6(1), pp. 68–79. <https://doi.org/10.55810/2313-0083.1087>.
- Baruah, N.D.D. and K.K. Das. (2025). Aerodynamic Analysis of NACA 2412 airfoil using ANSYS. International Research Journal on Advanced Engineering and Management (IRJAEM) 3(02), pp. 173–177. <https://doi.org/10.47392/irjaem.2025.0030>.
- Beik, O. and Al-Adsani, A. (2020). Wind turbine productivity and wind energy assessment: An Ontario case study. In: 2020 IEEE Electric Power and Energy Conference, EPEC 2020. Institute of Electrical and Electronics Engineers Inc. <https://doi.org/10.1109/EPEC48502.2020.9320110>
- Boulla, D., Fertahi, S.E.D., Bernatchou, M., Samaouali, A., Ajani, M. and Boussaq, C. (2025). Enhancement of aerodynamic performance of H-Darrieus rotor using wraparound fairing system: A 2D CFD study. International Journal of Renewable Energy Development 14(5), pp. 1058–1071. <https://doi.org/10.61435/ijred.2025.61437>.
- Erkan, O., Özkan, M., Karakoç, T.H., Garrett, S.J. and Thomas, P.J. (2020). Investigation of aerodynamic performance characteristics of a wind-turbine-blade profile using the finite-volume method. Renewable Energy 161, pp. 1359–1367. <https://doi.org/10.1016/j.renene.2020.07.138>
- Faadhillah, S.H.S.P. and A.W. (2025). Aerodynamic Performance Optimization on NACA 2412 Airfoil Flap With The Addition of Riblets. SAINSTECH NUSANTARA 2(3), pp. 17–34. <https://doi.org/10.71225/jstn.v2i3.118>.
- Fadaei, S., Langlois, R.G. and Afagh, F.F. (2023). Aerodynamic Properties Identification for Small-Size Wind Turbine Blade Airfoil Sections Using the CFD Method. In: International Conference of Control, Dynamic Systems, and Robotics. Avestia Publishing. <https://doi.org/10.11159/cdsr23.193>.
- Giacomini, E. and Westerberg, L.G. (2025). CFD Analysis of Transition Models for Low-Reynolds Number Aerodynamics. Applied Sciences (Switzerland) 15(18). <https://doi.org/10.3390/app151810299>.
- Iddou, H., Bouda, N.N., Benaissa, A. and Zereg, K. (2024). Numerical Analysis of the Kline and Fogleman Airfoil's Effect on the Operation of Straight Darrieus Wind Turbine. Journal of Applied Fluid Mechanics 17(8), pp. 1568–1592. <https://doi.org/10.47176/jafm.17.8.2490>
- Jha, D. and Saurabh, S. (2023). NACA2412 airfoil based method for design and aerodynamic analysis of small HAWT using modified BEM approach. Science and Technology for Energy Transition (STET) 78(3). <https://doi.org/10.2516/stet/2022023>.
- Jing, S., Zhao, G., Gao, Y. and Zhao, Q. (2024). Effects of Leading Edge Radius on Stall Characteristics of Rotor Airfoil. Aerospace 11(6). <https://doi.org/10.3390/aerospace11060470>.
- Juvinel, J.M.D.E., Pinto Roa, D.P. and Schaerer, C.E. (2023). Structural and Shape Optimization in Aerodynamic Airfoil Performance: A Literature Review. Available at: <https://www.preprints.org/manuscript/202307.0807/v1>.
- Khalid, M.H. (2022). CFD Analysis of NACA 0012 Aerofoil to Investigate the Effect of Increasing Angle of Attack on Coefficient of Lift and Coefficient of Drag. Journal of Studies in Science and Engineering 2(1), pp. 74–86. <https://doi.org/10.53898/josse2022216>.
- Sanan H. Khan, S.A., M.D., Md.A., A.H., S.S., S.S., A.S. & A.T. (2025). Aerodynamic analysis and ANN-based optimization of NACA airfoils for enhanced UAV performance. Scientific Reports 15(1). <https://doi.org/10.1038/s41598-025-95848-4>.
- Liu, X., Zhao, Q. and Lu, J. (2020). Simulation method of semi-physical wind power generation based on combined wind speed and tip speed ratio. In: Proceedings - 2020 Chinese Automation Congress, CAC 2020. Institute of Electrical and Electronics Engineers Inc., pp. 1805–1809. <https://doi.org/10.1109/CAC51589.2020.9327046>.
- Maurya, N., Patil, A.U., Salunkhe, P.B. and Varpe, M. (2024). Effect of protuberances on the aerodynamic

*Corresponding author: karrari.ramadhan@uokufa.edu.iq

performance of a wind turbine blade—a review. *Energy Sources, Part A: Recovery, Utilization and Environmental Effects* 46(1), pp. 3416–3431. <https://doi.org/10.1080/15567036.2024.2321294>.

Mazumder, M.D.H.M. (2024). CFD Analysis of NACA Airfoils for Wind Turbine and Aerospace Applications at Low Reynolds Numbers. Available at: <https://www.researchsquare.com/article/rs-4965405/v1>.

Omenai, S.A. (2024). CFD Simulation of Steady, Compressible Flow around the Naca 2412 Airfoil. Available at: www.carijournals.orgwww.carijournals.org.

Oyewola, O.M., Idowu, E.T. and Labiran, M.J. (2026). Combined effect of surface coating and angle of attack on the aerodynamic performance of NACA 0012, 4412, and 6412 airfoils. *Next Materials* 10. <https://doi.org/10.1016/j.nxmte.2025.101531>.

Özer, R.A. (2026). Aerodynamic Performance Analysis of Wind Turbine Airfoils at Low Reynolds Numbers. *Journal of the Institute of Science and Technology* 16(2), pp. 703–714. <https://doi.org/10.21597/jist.1844302>.

Roslan, S.A.H., Rasid, Z.A. and Ariffin, A.K. (2023). Extended Blade Element Momentum Theory for the Design of Small-Scale Wind Turbines. *Journal of Advanced Research in Applied Mechanics* 101(1), pp. 62–75. <https://doi.org/10.37934/aram.101.1.6275>.

Steenwijk, B. and Druetta, P. (2023). Numerical Study of Turbulent Flows over a NACA 0012 Airfoil: Insights into Its Performance and the Addition of a Slotted Flap. *Applied Sciences (Switzerland)* 13(13). <https://doi.org/10.3390/app13137890>.

Tanabi, N., Silva, A.M., Pessoa, M.A.O. and Tsuzuki, M.S.G. (2023). Robust Algorithm Software for NACA 4-Digit Airfoil Shape Optimization Using the Adjoint Method. *Applied Sciences (Switzerland)* 13(7). <https://doi.org/10.3390/app13074269>.

Tavares, S.M.O., Gamboa, P. V. and de Castro, P.M.S.T. (2025). Aircraft Wings and Morphing—Evolution of the Concepts. *Encyclopedia* 5(3). <https://doi.org/10.3390/encyclopedia5030101>.

Wood, D.H. and Golmirzaee, N. (2023). A revision of blade element/momentum theory for wind turbines in their high-thrust region. *Frontiers in Energy Research* 11. <https://doi.org/10.3389/fenrg.2023.1256308>.

Wu, C., Li, H., Zhang, Y. and Chen, H. (2022). Prediction of Airfoil Stall Based on a Modified {formula-present} Turbulence Model. *Mathematics* 10(2). <https://doi.org/10.3390/math10020272>.