International Journal of Research Publication and Reviews

Journal homepage: www.ijrpr.com ISSN 2582-7421

Aerodynamic Analysis of NACA2412 Airfoil for Lift and Drag Coefficients Using Ansys

Kathula Vijay Kumar [a], Dr. Anoop Kumar Shukla [b]

[a] *M. Tech Scholar, Department of Mechanical Engineering, J.B. Institute of Engineering and Technology, Moinabad, Hyderabad-75, Telangana, India.*
[b] *Professor, Department of Mechanical Engineering, J.B. Institute of Engineering and Technology, Moinabad, Hyderabad-75, Telangana, India.*

A B S T R A C T

This project investigates the aerodynamic performance of the NACA2412 airfoil using Computational Fluid Dynamics (CFD) to determine the lift and drag coefficients at various angles of attack. The primary objective is to analyze the flow behavior around the airfoil and evaluate its aerodynamic efficiency under different conditions, using ANSYS as the simulation tool. The study involves defining the NACA2412 airfoil geometry directly in the ANSYS CFD environment, followed by meshing and applying appropriate boundary conditions to ensure realistic simulation results. Simulations are conducted at multiple angles of attack, and key outputs such as pressure contours, velocity contours, and velocity vectors are generated to visualize the flow characteristics. The lift and drag coefficients are computed for each angle of attack to understand how the airfoil's performance changes under different aerodynamic conditions. The outcome of this project includes the determination of lift and drag coefficients for the NACA2412 airfoil at various angles of attack. These coefficients will provide insights into the aerodynamic performance of the airfoil under different flow conditions. Additionally, the project will generate pressure contours, velocity contours, and velocity vectors, which will visually represent the airflow characteristics around the airfoil, which helps to understand the distribution of pressure and velocity in the flow field. This study will also highlight the changes in the angle of attack impact the aerodynamic forces acting on the airfoil.

Keywords: NACA airfoil 2412, CFD analysis, ANSYS – Fluent, Lift and Drag Coefficients.

1. INTRODUCTION

Aerodynamics is a critical field of study in engineering, particularly in aerospace, automotive, and renewable energy industries. The primary goal of aerodynamic design is to understand how objects, such as aircraft wings, vehicles, and turbine blades, interact with the surrounding fluid, typically air. By analyzing the behaviour of airflow over various shapes, engineers can optimize designs to improve performance, minimize drag, and enhance overall efficiency. Achieving these objectives relies heavily on understanding the fundamental aerodynamic forces acting on an object, most notably the lift and drag forces. These forces are influenced by various factors, such as the shape of the object, the speed of the flow, and the angle at which the object interacts with the flow. In traditional aerodynamic analysis, wind tunnel experiments and flight tests have been the primary methods for obtaining empirical data on lift, drag, and other aerodynamic parameters. While these methods are highly effective, they are often time-consuming, expensive, and limited by the available test conditions. With the advancement of computational technologies, Computational Fluid Dynamics (CFD) has emerged as a powerful alternative. CFD enables engineers and researchers to simulate fluid flow behaviour over complex geometries, providing a virtual platform for conducting detailed analyses of aerodynamic performance without the need for physical testing. The ability to perform simulations under varying conditions, such as different angles of attack, flow velocities, and pressure distributions, has revolutionized the way aerodynamic analysis is conducted. CFD relies on numerical methods to solve the governing equations of fluid flow, most notably the Navier-Stokes equations, which describe the motion of viscous fluids. These equations are complex and involve multiple variables, including velocity, pressure, and density, which vary in time and space. Through CFD, these equations are solved iteratively on a discrete grid, enabling the simulation of airflow around various shapes, including airfoils. The results provide valuable insights into flow characteristics such as pressure distribution, velocity profiles, and the occurrence of flow phenomena like separation and vortices.

1.1: Computational Fluid Dynamics (CFD)

It is the science of predicting fluid flow, heat transfer, chemical reactions, and related phenomena by solving the mathematical equations which govern these processes. CFD, also sometimes referred to as fluid flow simulation, is a numerical computer simulation that permits the fluid flow around or through any object, so the impact on that object can be analysed in detail.

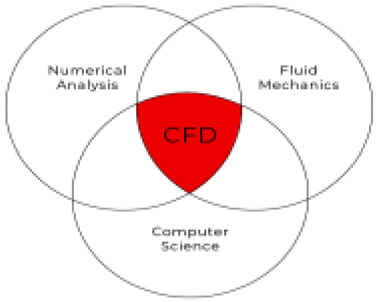


Figure 1.1: Computational Fluid Dynamics

1.2: NACA2412 Airfoil: Design and Characteristics

The NACA2412 airfoil is a widely studied profile within the NACA 4-digit series, known for its cambered design and versatile aerodynamic properties. It is characterized by the following parameters:

- Cambered airfoil with 2% maximum camber located at 40% of the chord length from the leading edge.
- 12% maximum thickness of the chord, providing a balance between aerodynamic performance and structural strength.

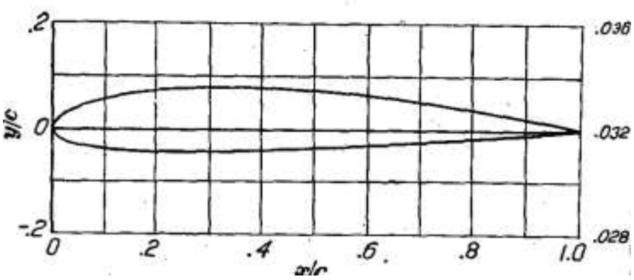


Figure 1.3: NACA2412 airfoil

2. LITERATURE REVIEW

[1]. Kulshreshtha et al. (2020) studied the performance of wings at various angles of attack using FEM and CFD techniques. The research aimed to improve fuel efficiency by optimizing the aerodynamic design of airfoil sections to enhance lift while minimizing drag. The study focused on three airfoil profiles—NACA 2412, NACA 2414, and NACA 2415—provided by the National Advisory Committee for Aeronautics (NACA). Using the standard k-epsilon viscous model, the researchers solved the governing momentum equations under steady-state conditions. The airfoil geometries were developed in ANSYS Design Modeller and analyzed in ANSYS 15 with Fluent as the solver and CFD-Post for post-processing. The analysis determined lift and drag coefficients at various angles of attack and constant air velocity, providing valuable insights into improving airfoil performance. The results revealed that the lift-to-drag ratio varied with the angle of attack for each airfoil. Among the profiles, NACA 2412 was identified as the most suitable for general applications. Although its lift-to-drag ratio decreased at higher angles of attack, it showed less variation compared to NACA 2414 and NACA 2415, where the ratio steadily declined. This suggests that NACA 2412 offers a more balanced aerodynamic performance across different conditions. The study highlights the importance of selecting appropriate airfoil designs based on operational needs, with NACA 2412 being a reliable choice for maintaining efficiency while requiring less energy at higher angles of attack.

[2]. Rao et al. (2018) conducted a numerical analysis on the NACA 0012 symmetric airfoil to investigate design modifications aimed at enhancing the lift-to-drag ratio. Using ANSYS Fluent for simulations and hybrid-structured meshing, the study explored various configurations, including flaps at different angles, slots at leading and trailing edges, and Gurney flaps with wedge supports. Validation against standard data confirmed the accuracy of the setup. Key findings revealed that adding a flap at 15° with a trailing-edge slot improved lift by approximately 1.87 times but increased drag by 2.22 times. In contrast, the Gurney flap with wedge support increased lift by 1.78 times with a smaller drag increment of 1.33 times. For higher lift needs, the flap and slot combination was preferred, while the Gurney flap was suitable for applications requiring a balance of lift and reduced drag. These findings highlight the potential of adaptive designs to optimize airfoil performance.

[3]. Rasal, S. K., & Katwate, R. R. (2017) Numerical Analysis of Lift & Drag Performance of NACA0012 Wind Turbine Aerofoil. This paper focuses on the importance of wind turbine blade design for capturing wind energy efficiently. The authors investigate the role of aerofoil shape in improving performance by controlling flow separation, a common challenge at low Reynolds numbers. The study specifically examines the NACA0012 aerofoil, which is used in wind turbine blades, and explores how adding circular dimples to the upper surface of the aerofoil can mitigate flow separation. Two models of the NACA0012 aerofoil were analyzed: one with a regular smooth surface and one with dimples placed at 70% of the chord length from the leading edge. The height of the dimples was 3 mm, equivalent to 1% of the chord length. Numerical analysis was performed using ANSYS Fluent, and the coefficient of lift-to-drag ratio was computed at various angles of attack. The results showed that the addition of circular dimples led to an improvement in the lift-to-drag ratio, indicating that surface modifications can enhance the aerodynamic performance of wind turbine blades by reducing flow separation and improving efficiency in wind energy capture.

[4]. Muhammad Ammar Qayyum Abdul Mukhti et al. (2021) investigated the aerodynamic performance of the NACA 4412 airfoil profile to evaluate its effectiveness for wind turbine applications. They emphasized the importance of selecting an appropriate airfoil section to enhance the aerodynamic efficiency of wind turbines. Using computational fluid dynamics (CFD), the authors simulated the airfoil geometry according to NACA standards and analyzed its performance across a range of angles of attack (0° to 18°) with 2° increments under low Reynolds number conditions. The results highlighted that the angle of attack significantly affects the lift and drag forces, which are critical parameters determining the aerodynamic efficiency of wind turbine blades. Their findings revealed that the NACA 4412 airfoil achieves optimal performance at an angle of attack of 13.8°, where the lift-to-drag ratio is maximized. However, the authors also noted that additional factors, such as blade backflow turbulence and blade forces, must be considered when assessing the overall aerodynamic performance of NACA airfoils. This study contributes valuable insights into the role of airfoil geometry in improving wind turbine blade efficiency, aligning with the growing focus on sustainable energy solutions.

[5]. Haci Sogukpinar and Ismail Bozkurt (2015) conducted a numerical investigation to determine the optimum angle of attack for the NACA 632-215 airfoil, which is critical for enhancing the aerodynamic efficiency of horizontal-axis wind turbines. Using the SST turbulence model, they simulated airflow around the inclined airfoil and evaluated key aerodynamic parameters such as lift coefficient, drag coefficient, lift-to-drag ratio, and power coefficient at varying wind velocities. Their findings revealed that as wind velocity increases, both lift and drag coefficients rise, leading to an initial increase in the lift-to-drag ratio, which subsequently decreases. The study identified the optimum angle of attack as approximately 4°, where the maximum lift-to-drag ratio was achieved. Additionally, the authors observed that increasing the angle of attack intensifies the pressure difference between the upper and lower surfaces of the airfoil, further influencing its aerodynamic performance. These results contribute to understanding airfoil behaviour under varying conditions, offering insights for optimizing wind turbine blade designs.

3. SOFTWARE TOOLS AND METHODOLOGIES

ANSYS is a comprehensive engineering simulation software suite that allows engineers and researchers to model, simulate, and analyze various physical phenomena. It provides tools for structural, thermal, fluid, electromagnetic, and multi-physics simulations. The software is modular, offering specific tools tailored to different engineering challenges.

3.1 ANSYS Modules

The key modules within the ANSYS ecosystem include:

- ANSYS Mechanical: Focuses on structural analysis, including static and dynamic loading, thermal stress, and vibration analysis.
- ANSYS Fluent: A dedicated computational fluid dynamics (CFD) tool for simulating fluid flow, heat transfer, and chemical reactions.
- ANSYS CFX: Another CFD tool, specialized for high-speed rotating machinery applications such as turbines and compressors.
- ANSYS Maxwell: For electromagnetic field simulation, used in designing electric motors, transformers, and other electrical devices.
- ANSYS HFSS: Specialized in high-frequency electromagnetic simulations, including antennas and microwave devices.
- ANSYS Discovery: Provides real-time simulation capabilities for rapid design iteration and prototyping.
- ANSYS Explicit Dynamics: Simulates highly nonlinear, transient dynamic events such as impacts, crashes, and explosions.
- ANSYS Workbench: Acts as the integration platform for all ANSYS modules, enabling a seamless workflow across different physics.

These modules can be coupled to perform multiphysics simulations, addressing complex engineering problems where interactions between different domains are significant.

3.2: Introduction to ANSYS Fluent

Computational Fluid Dynamics (CFD) is an essential discipline in engineering, allowing engineers and researchers to simulate fluid flow and thermal phenomena within systems. Unlike traditional experimental methods, CFD provides an efficient and cost-effective way to analyze complex fluid systems before implementation or construction, thus minimizing risks and optimizing designs. In industrial applications, the impact of fluid dynamics is significant, influencing the performance, efficiency, and safety of products. This computational approach has revolutionized industries like aerospace, automotive, energy, and civil engineering by enabling the virtual testing of fluid interactions.

CFD in Modern Engineering

The capability of CFD has transformed the approach to engineering problem-solving, especially when dealing with complex scenarios that are difficult or expensive to test physically. With CFD, one can simulate real-world phenomena such as airflow over an aircraft wing, fluid behaviour in pipes and ducts, or heat transfer in thermal systems. This method helps predict critical outcomes like pressure drops, temperature changes, and fluid velocities under various operating conditions. The results are highly accurate, provided the correct boundary conditions, turbulence models, and meshing is used.

Benefits of CFD

The advantages of using CFD simulations in product design and analysis include:

- Cost-Effective: By simulating fluid dynamics digitally, companies can avoid expensive physical prototypes and costly experimental testing.
- Time-Saving: Engineers can quickly modify parameters in the simulation, reducing the time required for iterative design changes.
- Optimization: CFD allows the exploration of design modifications to optimize performance without the limitations of physical testing environments.
- Safety: The predictive power of CFD helps identify potential problems in systems, such as catastrophic failure, before they occur in real-world applications.

4. EXPERIMENTAL DESIGN AND SETUP

The experimental design for this study involves a series of computational simulations conducted using ANSYS 2023 Fluid Flow (Fluent) to analyze the aerodynamic performance of the NACA 2412 airfoil at various angles of attack. The airfoil was subjected to flow conditions with a constant velocity of 45.8 m/s. The angles of attack considered were 0°, 8°, 15°, and 25°, which were chosen to explore the effects of different flow conditions on the lift and drag characteristics of the airfoil. The analysis involved the use of steady-state, incompressible flow conditions to simulate the airflow over the airfoil and evaluate key aerodynamic parameters, including pressure and velocity distributions, lift and drag coefficients, and lift-to-drag ratios. The simulation was performed under conditions that represent typical operational parameters of wind turbine blades and aerospace applications.

Table No. 4.1: Process Parameters

S. No.	Airfoil model	Angle of attack			
1	NACA 2412	0^0	8^0	15^0	25^0

4.1: Design of NACA2412 airfoil

The critical length dimension of an airfoil profile is defined in terms of its chordline; the chord is defined as the distance measured from the leading edge of the airfoil profile to its trailing edge. However, in the geometric construction of airfoil profiles, it is necessary to be more precise about how exactly the profile shape is defined, including the value and position of the maximum thickness (thickness-to-chord ratio), the value and position of the maximum camber, as well as the nose shape or radius. NACA2412 coordinates are collected and Ansys 2023 is used to generate the airfoil 3D curve.

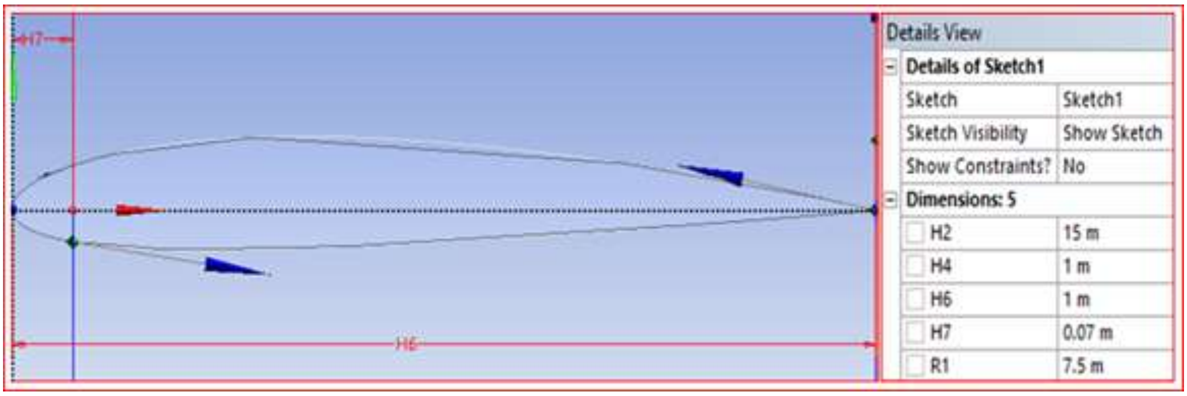


Figure 4.1: NACA2412 airfoil design

4.2: Fluid domain around the airfoil

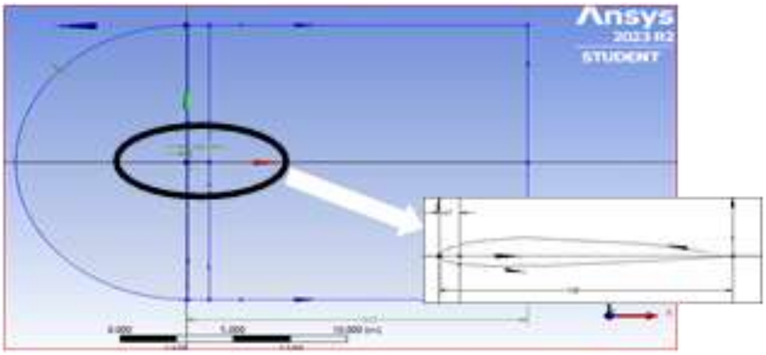


Figure 4.2: NACA2412 airfoil C-shape fluid domain

A C-shaped domain was utilized to model the fluid flow around the NACA 2412 airfoil, offering better convergence results compared to a square-shaped domain. The C-shape domain is more effective at simulating the aerodynamic flow over the airfoil, as it provides smoother boundaries that better accommodate the natural flow characteristics around curved surfaces. In contrast, a square domain often leads to sharper flow deviations and higher turbulence at the edges, resulting in slower convergence and less accurate results. The C-shape minimizes the influence of corners and sharp turns that can cause flow separation and vortices in a square domain. This streamlined geometry helps guide the fluid around the airfoil more efficiently, reducing recirculation zones and turbulence. As a result, the solver converges more quickly and accurately, leading to improved prediction of aerodynamic forces, such as lift and drag, on the NACA 2412 airfoil. The use of a C-shaped domain not only enhances computational efficiency but also ensures more reliable and precise simulations for airfoil design and performance analysis.

4.3: Generating of Mesh

A structured mesh was generated within the C-shaped domain for simulating the fluid flow around the NACA 2412 airfoil. One of the key features of this mesh is the fine resolution near the airfoil's surface, where the cell gap is set to 6.3447e-3 mm. This extremely small cell size in the near-wall region is essential for capturing the high gradients in velocity and pressure, especially in the boundary layer, where fluid flow transitions from laminar to turbulent. Such refinement ensures a highly accurate representation of the flow characteristics around the airfoil, which are crucial for obtaining reliable aerodynamic performance predictions such as lift, drag, and pressure distribution. The structured grid is carefully refined near the airfoil, focusing on areas where the flow changes most rapidly, such as the leading and trailing edges. This refinement provides a detailed resolution of the boundary layer and ensures that small-scale flow features like flow separation and vortex formation are captured effectively. The 6.3447e-3 mm cell size also helps in accurately resolving the aerodynamic forces acting on the airfoil, leading to a more precise simulation.

Outside the boundary layer, the cell size gradually increases, ensuring computational efficiency without sacrificing the accuracy needed in the critical regions. This hierarchical approach to mesh generation optimizes the use of computational resources, focusing finer resolution on areas where the flow features are most significant, while allowing for coarser cells in the far-field regions where the flow behaviour is less sensitive. Overall, this structured mesh with a fine cell gap near the airfoil enables the simulation to converge quickly and provides reliable results for aerodynamic analysis.

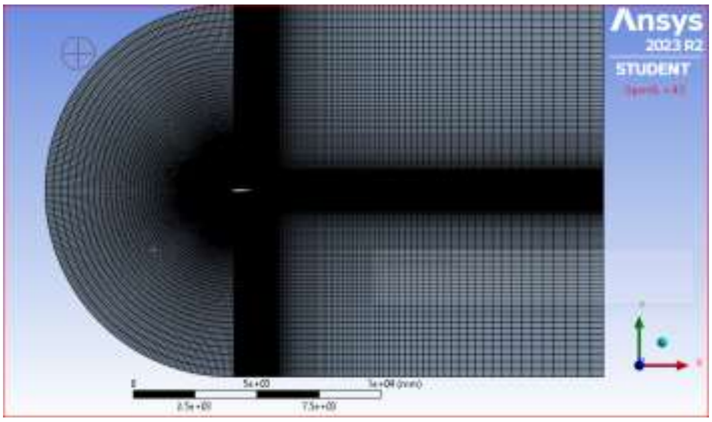


Figure 4.3: Structured mesh of NACA2412 airfoil

4.4: Setting of boundary conditions

For the simulation of fluid flow around the NACA 2412 airfoil, appropriate boundary conditions were set to ensure accurate and realistic results. The inlet boundary condition was defined as a uniform velocity profile, specifying the incoming flow velocity. This represents the free-stream conditions approaching the airfoil. The airfoil surface itself was set as a no-slip wall, meaning the fluid velocity at the surface is zero, ensuring the correct interaction between the airfoil and the boundary layer. The outer domain boundaries were defined as the outlet, where the flow is allowed to exit with a specified static pressure, typically set to atmospheric conditions, ensuring smooth flow exit without causing backflow. Finally, the surrounding domain walls, except for the airfoil surface, were defined as walls with no-slip conditions, preventing fluid from passing through the boundaries. These boundary conditions collectively enable a realistic simulation of the airflow around the NACA 2412 airfoil, ensuring the solver converges to accurate aerodynamic results.

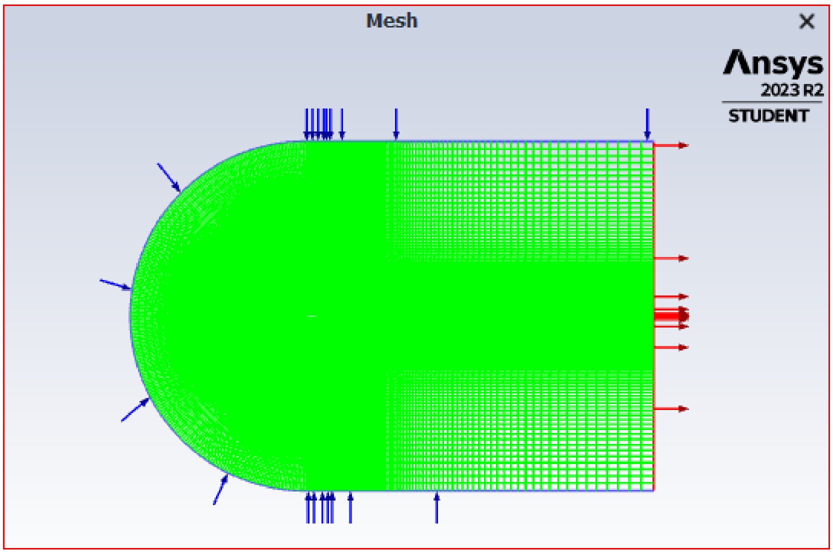


Figure 4.4: Setting of boundary conditions of NACA2412 airfoil

4.5: Pressure contours at different Angle of air attack

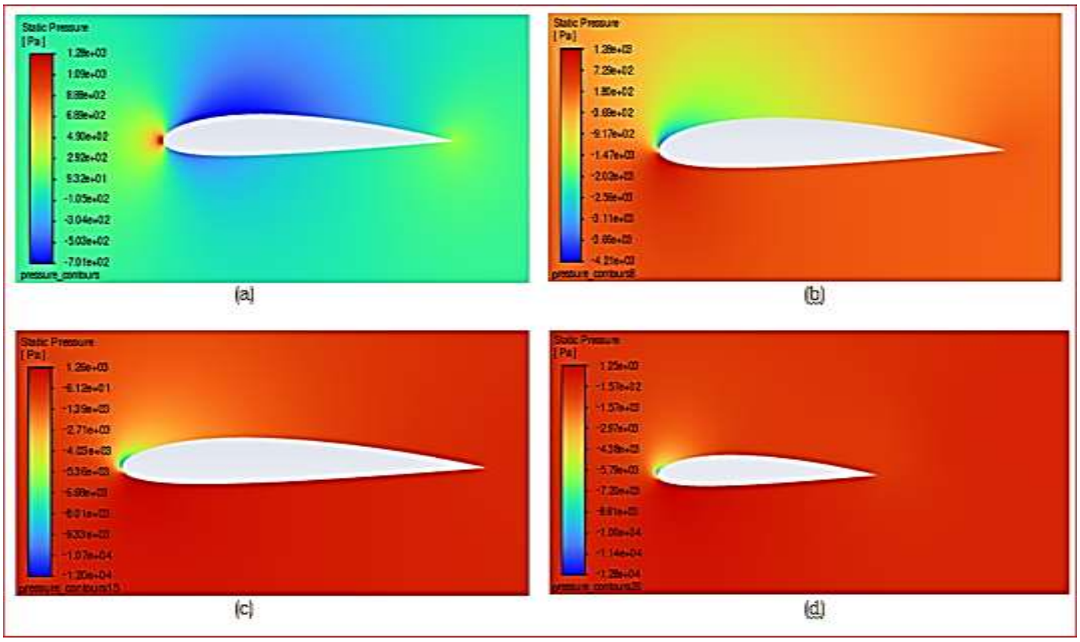


Figure 4.5: Static pressure (a) at 0^0 angle (b) at 8^0 angle

(c) at 15^0 angle (d) at 20^0 angle.

4.6: Velocity contours at different Angle of air attack

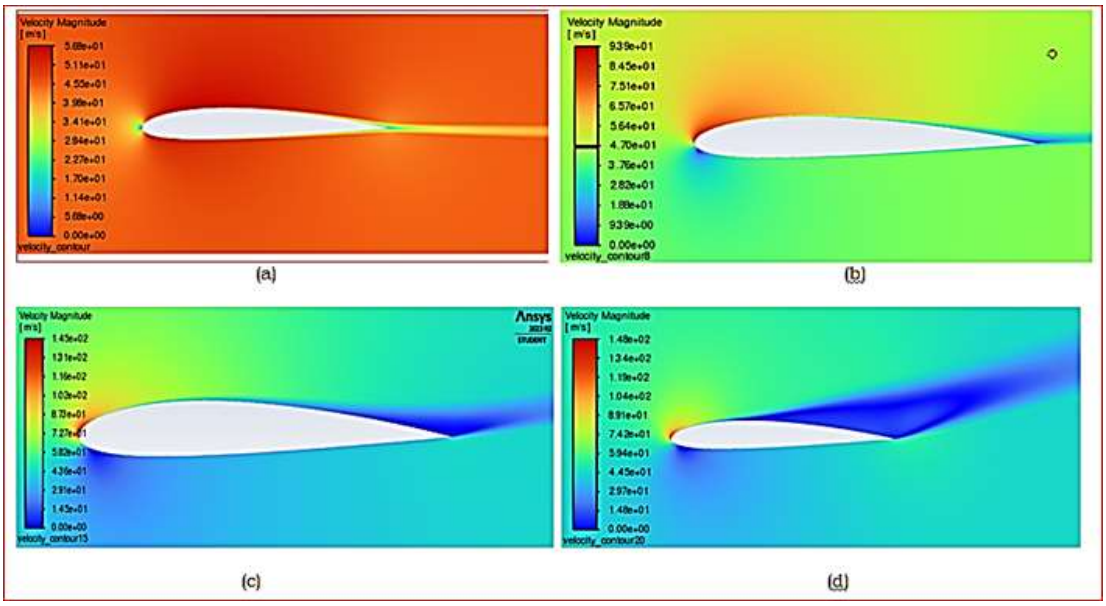


Figure 4.6: Velocity magnitude (a) at 0^0 angle (b) at 8^0 angle

(c) at 15^0 angle (d) at 20^0 angle.

4.7: Velocity vector at different Angle of air attack

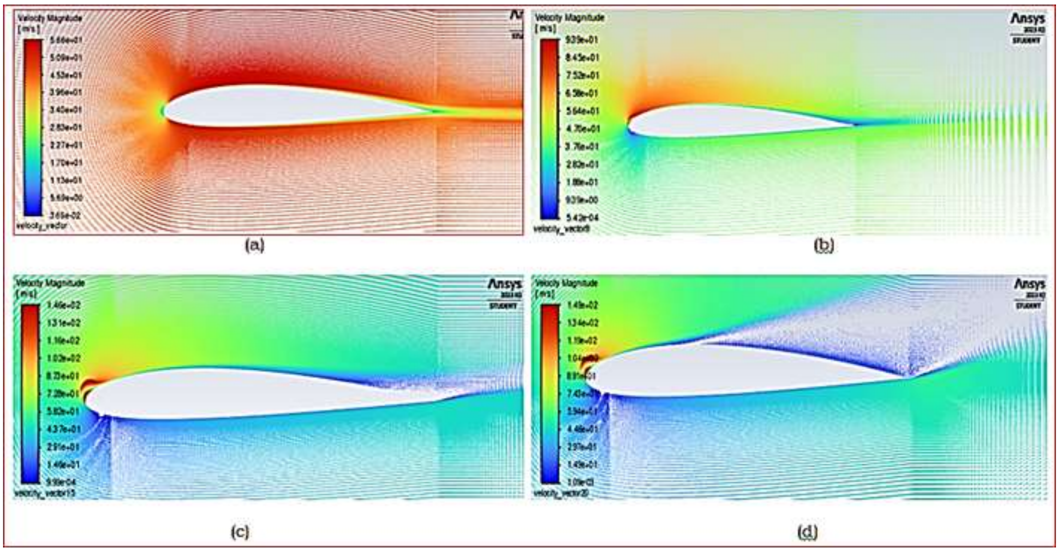


Figure 4.7: Velocity vector (a) at 0^0 angle (b) at 8^0 angle

(c) at 15^0 angle (d) at 20^0 angle

5. DATA ANALYSIS

The National Advisory Committee for Aeronautics (NACA) has published comprehensive data sheets that provide lift and drag coefficient values for airfoils at different angles of attack under various fluid flow conditions. These data sheets serve as a valuable resource for aerodynamic analysis, enabling researchers and engineers to predict the performance of airfoils across a range of operating conditions. By using this information, it is possible to evaluate the effects of changes in angle of attack on lift, drag, and overall aerodynamic efficiency. The data also plays a crucial role in validating experimental results and optimizing airfoil designs for improved performance.

Table 5.1: Percentage error at 0^0 degrees angle of attack

Parameter	True Value (Data Sheet)	Observed Value (Measured)	Absolute error	Percentage error
Lift Coefficient (C_l)	0.25	0.218	0.032	12.8%
Drag Coefficient (Cd)	0.0065	0.0092	0.0027	41.54%

Table 5.2: Percentage error at 8^0 degrees angle of attack

Parameter	True Value (Data Sheet)	Observed Value (Measured)	Absolute error	Percentage error
Lift Coefficient (C_l)	1.05	1.0495	0.0005	0.047%
Drag Coefficient (Cd)	0.012	0.0142	0.0022	18.33%

Table 5.3: Percentage error at 15^0 degrees angle of attack

Parameter	True Value (Data Sheet)	Observed Value (Measured)	Absolute error	Percentage error
Lift Coefficient (C_l)	1.5	1.626	0.126	8.4%
Drag Coefficient (Cd)	0.022	1.73×10^{-2}	0.0047	21.36%

Table 5.4: Percentage error at 20^0 degrees angle of attack

Parameter	True Value (Data Sheet)	Observed Value (Measured)	Absolute error	Percentage error
Lift Coefficient (C_l)	1.15	1.21	0.06	5.22%
Drag Coefficient (Cd)	0.012	0.015	0.0025	20%

Table 5.5: Lift Coefficient (C_l) with Error Percentages for Various Angles of Attack

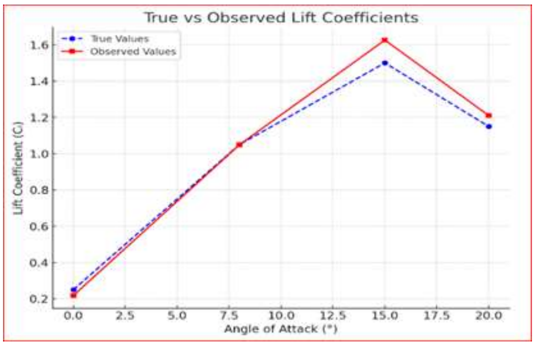
Angle of Attack (°)	Lift Coefficient (C_l) (True)	Lift Coefficient (C_l) (Observed)	Absolute Error (C_l)	Percentage Error (C_l)
0°	0.25	0.218	0.032	12.8%
8°	1.05	1.049	0.0005	0.047%
15°	1.5	1.626	0.126	8.4%
20°	1.15	1.210	0.06	5.22%

Table 5.6: Drag Coefficient with Error Percentages for Various Angles of Attack

Angle of Attack (°)	Drag Coefficient (C_d) (True)	Drag Coefficient (C_d) (Observed)	Absolute Error (C_d)	Percentage Error (C_d)
0°	0.0065	0.0092	0.0027	41.54%
8°	0.0120	0.0142	0.0022	18.33%
15°	0.0220	0.0173	0.0047	21.36%
20°	0.0125	0.0150	0.0025	20%

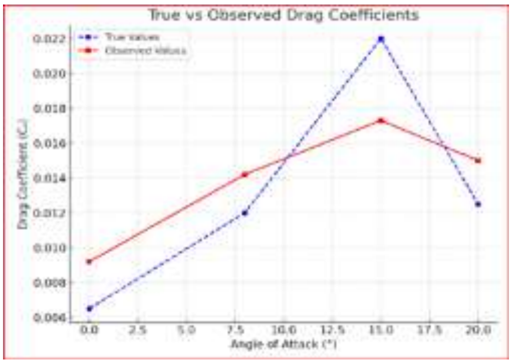
6. RESULTS AND DISCUSSIONS

6.1: Lift Coefficient



The graph shows that the lift coefficient increases steadily with the angle of attack, peaking at 15^0 before slightly decreasing at 20^0. This behaviour is consistent with the airfoil's aerodynamic performance, where the lift typically increases until reaching the stall angle, beyond which efficiency decreases. Discrepancies between the true and observed values are apparent. For instance, the observed values are slightly lower than the true values at 0^0 and 8^0, while they are higher at 15^0 and 20^0. The largest percentage error occurs at 0^0 (12.8%), and the smallest error occurs at 8^0 (0.047%).

6.2: Drag Coefficient



The x-axis represents the angle of attack, ranging from 0^0, while the y-axis shows the drag coefficient, which quantifies the airfoil's resistance to airflow. The true drag coefficient, represented by a blue dotted line with circular markers, and the observed drag coefficient, represented by a red solid line with square markers, exhibit similar trends, with the drag coefficient increasing as the angle of attack rises, peaking at 15^0. However, notable deviations occur between the true and observed values. At 0^0, the observed drag coefficient is significantly higher than the true value, resulting in a percentage error of 41.54%, the largest among all angles. At 8^0, the observed value slightly exceeds the true value, with an error of 18.33%. Conversely, at 15^0, the observed value is lower than the true value, with an error of 21.36%, while at 20^0, the observed value again exceeds the true value, with a 20% of error. These discrepancies may result from experimental limitations, measurement inaccuracies, or simplifications in theoretical modeling. Despite the differences, the overall trend is consistent, with the drag coefficient increasing with the angle of attack. This analysis highlights the need for refining experimental methods to reduce errors and improve alignment with theoretical predictions.

7. CONCLUSIONS

After the successful completion of the project, the following conclusions can be drawn.

- The computational analysis of the NACA2412 airfoil using ANSYS Fluent provided critical perceptions into its aerodynamic performance at varying angles of attack.
- The results reveal that the airfoil performs optimally at an 8° angle of attack. At this angle, it achieves a high lift-to-drag ratio due to the noticeable pressure difference between the upper and lower surfaces and largely attached airflow. This balance is ideal for operations like take-off, climbing, and efficient travelling, making it a favourable condition for many aerodynamic applications.
- At 0°, the airfoil experiences a symmetric pressure distribution, resulting in negligible lift and low drag. This condition is suitable for steady motion requiring minimal aerodynamic forces, such as travelling with low resistance.
- For higher angles of attack, particularly 15° and 20°, increased drag and significant flow separation lead to reduced efficiency and eventual stall. These conditions demonstrate the airfoil's limitations and highlight the need for careful angle optimization in practical applications.

- Pressure contours at 8° shown smooth pressure distribution and attached airflow, while velocity contours highlight streamlined airflow with minimal separation. Velocity vector analysis clearly shows that the airflow remains smooth and well-aligned at 8°, with minimal turbulence. In contrast, at higher angles, the airflow becomes messy and detaches from the surface.
- The study emphasizes the importance of understanding flow behaviour through visual and numerical analyses to optimize airfoil design and performance. Insights gained from this analysis are valuable for aeronautical and wind energy applications, particularly in optimizing lift and minimizing drag.
- Future investigations could explore the influence of Reynolds numbers, airfoil surface modifications, and three-dimensional effects to enhance the analysis. These extensions would provide a more comprehensive understanding of the NACA2412 airfoil's performance across broader operating conditions.

Acknowledgement

It is my pleasure to acknowledge, with deep appreciation to all those who have helped and guided me throughout the project work.

I would like to express my sincere gratitude to my Project Guide ***Dr. Anoop Kumar Shukla***, Professor and HOD of Department of Mechanical Engineering, JBIET, for his invaluable guidance, constant support, and cooperation throughout the project.

I am also deeply thankful to **Mr. G. Gopinath**, Assistant Professor, and P.G Coordinator, Department of Mechanical Engineering, JBIET, for his timely guidance, technical inputs and suggestions despite of his busy schedule.

My heartfelt thanks to **Dr. Anoop Kumar Shukla**, HOD, Department of Mechanical Engineering, JBIET, for his continued encouragement and technical guidance throughout the course of this project.

I would also like to extend my special thanks to **Dr. P.C. Krishnamachary**, Principal of JBIET, and the Management of JBIET for providing the necessary facilities to carry out this project.

References

1. A. Kulshreshtha, S. K. Gupta, and P. Singhal, "FEM/CFD Analysis of Wings at Different Angles of Attack," Materials Today: Proceedings, 2020.
2. Talluri Srinivasa Raoa, Trilochan Mahapatraa, Sai Chaitanya Mangavellib, “Enhancement of Lift-Drag characteristics of NACA 0012”, Materials Today: Proceedings, p:p- 5328–5337, 2018.
3. S. K. Rasal and R. R. Katwate, "Numerical Analysis of Lift & Drag Performance of NACA0012 Wind Turbine Aerofoil," International Research Journal of Engineering and Technology (IRJET), vol. 4, no. 6, pp. 2892–2895, June 2017.
4. M. A. Q. A. Mukhti, D. H. Didane, M. Ogab, and B. Manshoor, "Computational Fluid Dynamic Simulation Study on NACA 4412 Airfoil with Various Angle of Attacks," Journal of Design for Sustainable and Environment, vol. 3, no. 1, pp. 1–8, Mar. 2021.
5. H. Sogukpinar and I. Bozkurt, "Calculation of Optimum Angle of Attack to Determine Maximum Lift to Drag Ratio of NACA 632-215 Airfoil," Journal of Multidisciplinary Engineering Science and Technology (JMEST), vol. 2, no. 5, pp. 1103–1108, May 2015.
6. A. Agriss, M. Agouzoul, and A. Ettaouil, "Drag Reduction of a NACA Aerodynamic Airfoil: A Numerical Study," Journal of Fluid Flow, Heat and Mass Transfer (JFFHMT), vol. 10, pp. 97–104, 2023.
7. D. A. Gok and K. N. M. Al-Nimer, "Characterization of NACA 2412 and NACA 4412 Airfoils: Effects of Angle of Attack on Aerodynamics Coefficients," Journal of Thermal Engineering, vol. 10, no. 6, pp. 1524–1538, Nov. 2024.
8. M. Pane, "Wing Simulation Using NACA 2412 and 2415 Airfoils with Variations in Angle of Attack for Lift and Drag," VANOS Journal of Mechanical Engineering Education, vol. 8, no. 2, pp. 190–199, Nov. 2023.
9. Moses Omolayo Petinrin and Vincent Adah Onoja, “Computational Study of Aerodynamic Flow over NACA 4412 Airfoil”, British Journal of Applied Science & Technology, pp: 1-11, Article no.BJAST.31893 ISSN: 2231-0843, 2017.
10. MD. Hasib Mahmud Mazumder, “CFD Analysis of NACA Airfoils for Wind Turbine and Aerospace Applications at Low Reynolds Numbers”, Bangladesh University of Engineering and Technology (BUET), August, 2024.
11. J. D. Anderson, Jr., “Fundamentals of Aerodynamics”, 6th edition. New York: McGraw-Hill, 2016.
12. E. L. Houghton and P. W. Carpenter, “Aerodynamics for Engineering Students”, 6th edition. Oxford: Butterworth-Heinemann, 2013.
13. A. Pope and K. L. Goin, “High-Speed Wind Tunnel Testing”, 2nd edition. New York: Wiley, 2000.
14. T. Cebeci and J. Cousteix, “Modeling and Computation of Boundary-Layer Flows”, 2nd edition. New York: Springer, 2005.
15. National Aeronautics and Space Administration (NASA), “Summary of Airfoil Data”, NASA Technical Memorandum 82482, 1989.

16. Ira H. Abbott, Albert E. Von Doenhoff, and Louis S. Stivers, Jr. "National Advisory Committee for Aeronautics", Langley Memorial Aeronautical Laboratory, Report No. 824. 1945.
17. ANSYS Inc., ANSYS Fluent User's Guide, Release 2023, ANSYS Inc., Canonsburg, PA.