

DESIGN AND ANALYSIS OF NACA AIRFOIL

ANANTHARAJ .S [Reg. No. 23153103]

ARUN KUMAR .N [Reg. No. 23153104]

JOHN JOSEPH .S [Reg. No. 23153109]

RAVIKUMAR .R [Reg. No.23153120]

Under the Supervision of

Guide Name : MR.ADHIL SHAREIF

Designation, ASSISTANT PROFESSOR

Department of Aviation,

Vels Institute of Science, Technology & Advanced Studies (VISTAS)

**DEPARTMENT OF AVIATION
SCHOOL OF AVIATION, VISTAS
PALLAVARAM, CHENNAI – 600117
APRIL 2026**

ABSTRACT

The present project focuses on the design and aerodynamic analysis of the NACA 2412 airfoil using Computational Fluid Dynamics (CFD) through SimFlow software. Airfoils are the fundamental components of aircraft wings and play a crucial role in generating lift and controlling aerodynamic performance. The selection of an appropriate airfoil significantly affects the efficiency, stability, and performance of an aircraft.

The NACA 2412 airfoil, which belongs to the 4-digit airfoil series developed by the National Advisory Committee for Aeronautics, has been selected for this investigation due to its balanced aerodynamic characteristics and widespread use in low-speed aircraft applications. The geometric configuration of the airfoil is defined using standard analytical expressions based on camber and thickness distribution. These coordinate points are carefully generated and imported into the SimFlow simulation environment. A suitable computational domain is then created around the airfoil to replicate realistic airflow conditions.

The computational analysis is carried out for a range of angles of attack in order to thoroughly investigate the aerodynamic performance of the airfoil under varying flow conditions. Key aerodynamic parameters such as lift coefficient, drag coefficient, pressure coefficient distribution, and velocity profiles are evaluated in detail. The simulation employs appropriate turbulence models to capture the complex nature of fluid flow, particularly in regions where boundary layer effects and flow separation may occur. Iterative numerical techniques are used to solve the governing equations

of fluid motion until convergence is achieved. The resulting data is then post-processed to generate contour plots, streamline visualizations, and performance graphs, providing a comprehensive understanding of the airflow characteristics around the airfoil.

The results indicate that the lift coefficient increases with an increase in angle of attack up to a critical value, beyond which flow separation occurs leading to stall. The pressure distribution shows a significant difference between the upper and lower surfaces, which contributes to lift generation. The study demonstrates the effectiveness of CFD tools in analyzing aerodynamic performance and confirms that the NACA 2412 airfoil is suitable for low-speed aerodynamic applications.

Furthermore, the study emphasizes the importance of numerical simulation in modern aerodynamic design by demonstrating how computational tools can significantly reduce the need for expensive experimental setups such as wind tunnel testing. The use of SimFlow enables accurate prediction of flow behavior, allowing engineers to optimize airfoil performance efficiently.

CHAPTER 1

INTRODUCTION

1.1 Introduction to Aerodynamics

Aerodynamics is a fundamental branch of fluid mechanics that deals with the behavior of air when it interacts with solid bodies. It plays a significant role in the design and development of aircraft, automobiles, and other engineering systems where fluid flow is involved. In aviation, aerodynamics is essential for generating lift, reducing drag, and ensuring stability during flight. The study of airflow over wings helps engineers design efficient airfoil shapes that improve performance and fuel efficiency. Understanding aerodynamic principles is crucial for analyzing how pressure and velocity variations affect lift generation and overall aircraft performance.

1.2 Airfoil Basics

An airfoil is a specially designed cross-sectional shape of a wing, blade, or sail that is used to generate lift when air flows over it. The airfoil works based on the principle of pressure difference created between its upper and lower surfaces. When air moves faster over the upper curved surface, it creates low pressure, while the slower-moving air below creates higher pressure. This pressure difference results in an upward force called lift. The shape, thickness, camber, and angle of attack of an airfoil significantly influence its aerodynamic characteristics. Airfoils are widely used in aircraft wings, helicopter blades, turbines, and propellers.

1.3 NACA Airfoil Series

The National Advisory Committee for Aeronautics (NACA) developed a systematic and scientific method of classifying airfoils in order to standardize aerodynamic research and simplify airfoil design for engineering applications. Before the introduction of the NACA airfoil series, airfoil shapes were often developed through experimental trial-and-error methods, which made aerodynamic analysis more complex and less consistent. The development of the NACA

classification system provided engineers and researchers with a structured approach to designing airfoils based on defined geometric parameters, allowing easier comparison of aerodynamic performance between different airfoil profiles. This innovation greatly contributed to the advancement of aircraft design and aerodynamic engineering.

NACA airfoils are categorized into different series such as the 4-digit series, 5-digit series, and 6-series, with each series representing specific geometric and aerodynamic characteristics. These classifications were developed to meet different aerodynamic requirements such as higher lift generation, reduced drag, improved stability, or better performance at certain operating conditions. The numbering system used by NACA makes it possible to understand the fundamental shape of an airfoil simply by examining its code, eliminating the need for complex geometric descriptions.

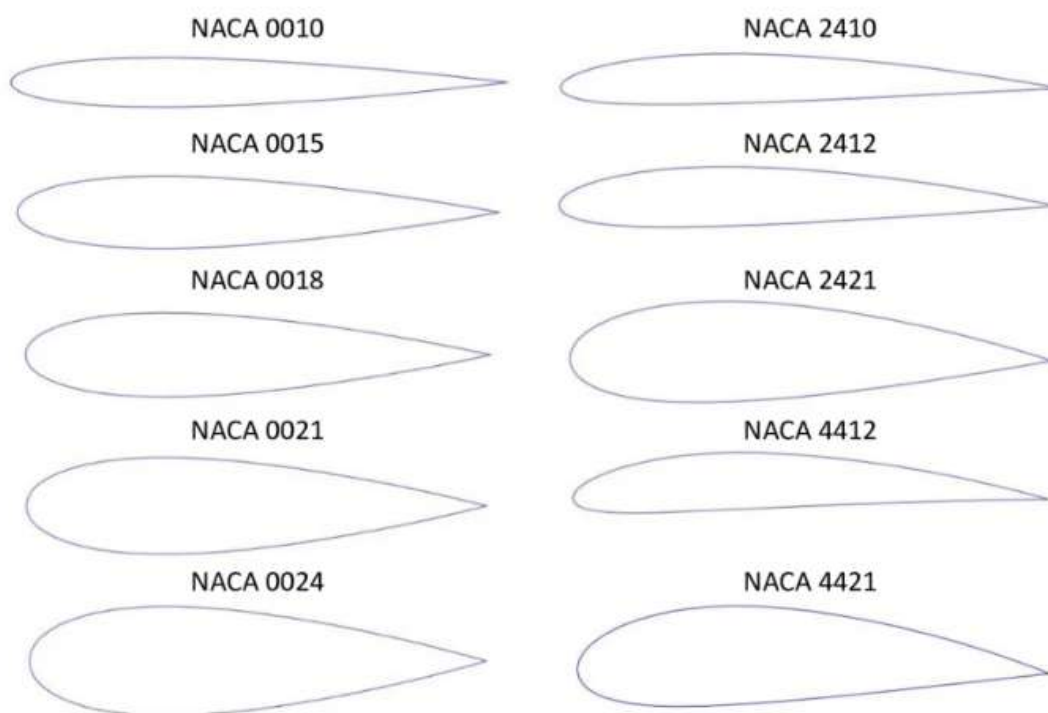


Figure 1.1 Series of airfoil

1.4 NACA 2412 Airfoil

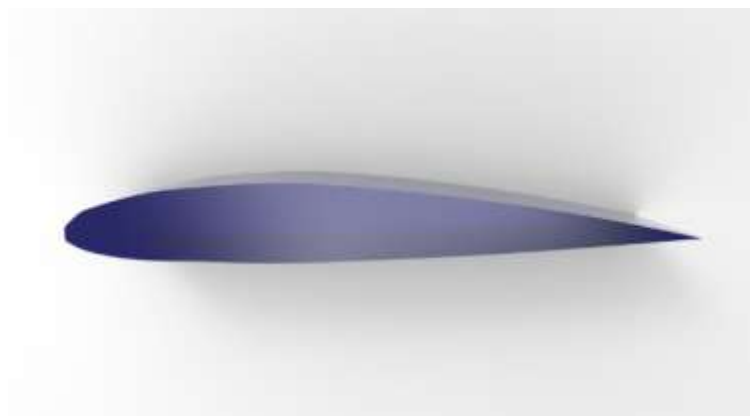


Figure 1.2 NACA 2412 Airfoil

CHAPTER 2

LITERATURE SURVEY

2.1 Introduction

The literature survey plays an important role in understanding the existing research related to airfoil design and aerodynamic analysis. It provides a foundation for the present study by reviewing previous work carried out in the field of Aerodynamics. Over the years, researchers have explored various airfoil geometries to improve lift generation, reduce drag, and enhance overall aerodynamic efficiency. Earlier studies primarily relied on experimental approaches such as wind tunnel testing, but with the advancement of computational methods, numerical analysis has become widely accepted. This chapter discusses the significant contributions made by researchers in the analysis of airfoils, particularly focusing on the NACA series.

2.2 Previous Studies on NACA Airfoils

The development of standardized airfoil shapes by the National Advisory Committee for Aeronautics has significantly contributed to aerodynamic research. Many studies have been conducted on different NACA airfoils to understand their performance characteristics under various flow conditions. Researchers have analyzed parameters such as lift coefficient, drag coefficient, and pressure distribution using both experimental and computational methods. These studies have shown that NACA airfoils provide reliable and predictable performance, making them widely used in both academic research and practical applications. The availability of well-defined geometries and data has made these airfoils a preferred choice for simulation-based studies.

Airfoil	Camber	Thickness	Key Feature
NACA 0012	0%	12%	Symmetrical, no lift at 0°
NACA 2412	2%	12%	Good lift, stable
NACA 4412	4%	12%	High lift, more drag

Table 2.1: Comparison of Different NACA Airfoils

2.3 Studies on NACA 2412 Airfoil

Among the various NACA airfoils, the NACA 2412 airfoil has been extensively studied due to its balanced aerodynamic properties. Researchers have found that this airfoil performs efficiently at low and moderate angles of attack, producing sufficient lift while maintaining relatively low drag. Several computational studies have confirmed that the airfoil exhibits stable flow behavior under normal operating conditions. The pressure distribution over the airfoil surface indicates a significant

pressure difference between the upper and lower surfaces, which is responsible for lift generation. The NACA 2412 airfoil is also known for its gradual stall characteristics, making it suitable for training aircraft and low-speed applications.

2.4 Computational Fluid Dynamics in Aerodynamics

The introduction of Computational Fluid Dynamics has revolutionized the field of aerodynamic analysis. CFD involves solving complex mathematical equations that govern fluid flow to predict the behavior of air around objects. It enables detailed analysis of flow parameters such as velocity, pressure, and turbulence. Researchers have widely adopted CFD as it reduces the need for costly experimental setups while providing accurate and detailed results. CFD simulations allow for the visualization of flow patterns, making it easier to understand complex aerodynamic phenomena such as flow separation and vortex formation.

Method	Advantages	Limitations
Experimental	Real results	Expensive
CFD	Fast, visual	Needs validation

Table 2.2: CFD vs Experimental Method

CHAPTER 3

RESEARCH METHODOLOGY

3.1 Introduction

This chapter describes the methodology adopted for the design and aerodynamic analysis of the NACA 2412 airfoil using computational techniques. The study is carried out using Computational Fluid Dynamics, which provides an efficient and accurate approach to analyzing fluid flow behavior. The methodology involves a sequence of steps including geometry creation, mesh generation, simulation setup, and post-processing of results. Each step is carefully executed to ensure that the simulation accurately represents real-world aerodynamic conditions. The overall approach allows for a detailed understanding of airflow characteristics around the airfoil without the need for physical experimentation.

3.2 Problem Statement

The primary objective of this study is to investigate the aerodynamic performance of the NACA 2412 airfoil under a range of operating conditions, with particular emphasis on how the airfoil behaves when subjected to varying angles of attack. The angle of attack plays a crucial role in determining the amount of lift generated as well as the aerodynamic efficiency of the airfoil. As this angle changes, the airflow characteristics around the airfoil also change significantly, affecting pressure distribution, velocity variation, and flow stability.

In this study, the focus is on understanding how these variations influence key aerodynamic parameters such as lift coefficient and drag coefficient. At lower angles of attack, the airflow remains smooth and attached to the airfoil surface, resulting in stable lift generation and minimal drag. However, as the angle of attack increases, the pressure difference between the upper and lower surfaces becomes more pronounced, leading to higher lift. Beyond a certain critical angle, the airflow begins to separate from the surface, causing a sudden reduction in lift and a sharp increase in drag, a phenomenon commonly known as stall.

3.3 Methodology

The methodology followed in this study begins with the creation of the NACA 2412 airfoil geometry using standard mathematical equations that define its camber and thickness distribution. Once the geometry is generated, it is imported into the simulation environment. A computational domain is then created around the airfoil to represent the flow field. The domain is discretized into smaller elements through mesh generation, allowing the governing equations of fluid flow to be solved numerically. After generating the mesh, appropriate boundary conditions are applied to simulate real airflow conditions. The simulation is then performed using suitable solver settings, and the results are analyzed through post-processing techniques. This systematic approach ensures accurate and reliable outcomes.

3.4 Mesh Generation

Mesh generation is a crucial step in CFD analysis, as it directly affects the accuracy of the simulation results. In this study, the computational domain is divided into a large number of small elements to facilitate numerical calculations. A finer mesh is created near the airfoil surface to capture the boundary layer effects and flow variations accurately. Away from the airfoil, a relatively coarser mesh is used to reduce computational effort while maintaining acceptable accuracy. The quality of the mesh is carefully checked to ensure that there are no irregularities that could affect the stability of the solution. Proper mesh refinement plays a key role in obtaining precise and reliable results

Parameter	Value
Mesh Type	Structured/Unstructured
Elements	~50,000 (or approx)
Refinement	Near airfoil

Table 3.1 Mesh Generation Parameters and Values

3.5 Simulation Setup

The simulation is carried out using SimFlow, which is built on the powerful OpenFOAM solver and provides an efficient platform for Computational Fluid Dynamics analysis. In this study, the solver is configured to perform a steady-state aerodynamic analysis, where the flow properties are assumed

to remain constant with time. This approach simplifies the simulation while still providing accurate predictions of airflow behavior around the NACA 2412 airfoil under specified operating conditions.

Appropriate turbulence models such as the *k-epsilon* or *k-omega* model are selected to account for turbulent flow effects and improve the reliability of the results, especially near the airfoil surface where flow disturbances may occur. Boundary conditions are carefully defined by specifying inlet velocity, outlet pressure, and wall conditions along the airfoil surface to accurately represent real airflow conditions. Additional solver settings such as convergence criteria, residual limits, discretization schemes, and numerical stability parameters are also configured to ensure precise and stable simulation results. This setup enables detailed analysis of aerodynamic parameters such as lift, drag, pressure distribution, velocity contours, and flow separation characteristics around the airfoil under different angles of attack.

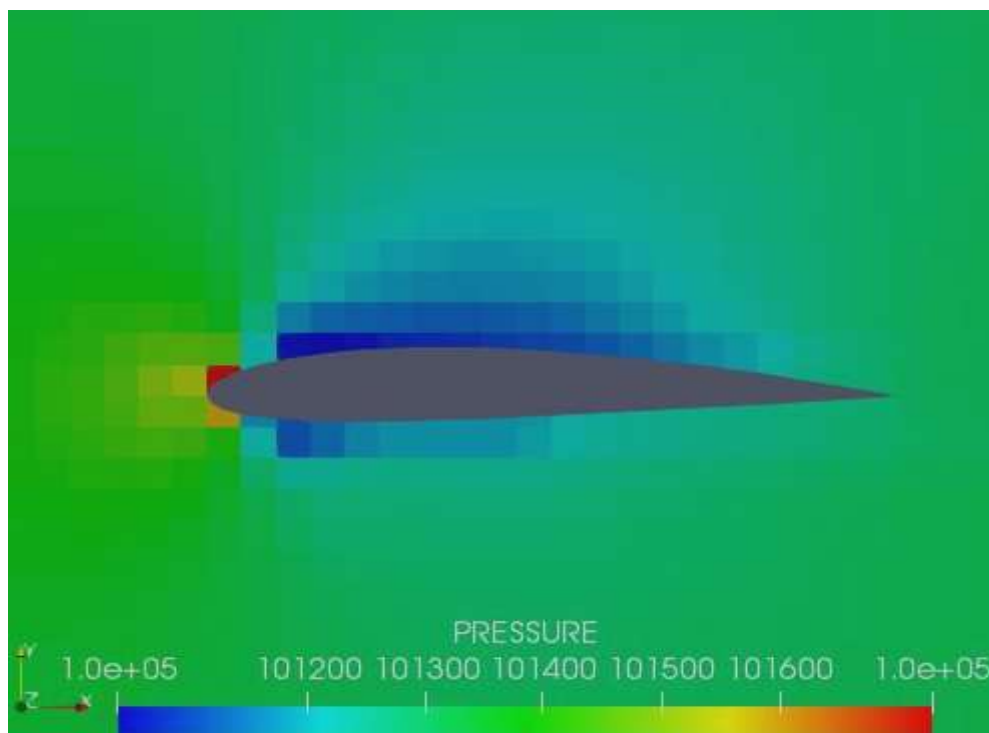


Figure 3.1 Simulation Setup

3.6 Parameters Used

The simulation in this study is carried out under carefully controlled and consistent conditions in order to accurately evaluate the aerodynamic performance of the NACA 2412 airfoil. Maintaining controlled conditions is important because it ensures that any observed changes in the results are due to variations in the angle of attack rather than external disturbances or inconsistent input parameters.

CHAPTER 4

RESULT AND DISCUSSION

4.1 Introduction

This chapter presents the results obtained from the aerodynamic analysis of the NACA 2412 airfoil and discusses the variation of key parameters such as lift coefficient and drag coefficient with respect to the angle of attack. The analysis is performed using Computational Fluid Dynamics through SimFlow, which enables accurate prediction of airflow characteristics. In addition to contour visualization, numerical values of lift and drag are used to plot graphs, providing a clearer understanding of aerodynamic performance.

4.2 Lift Characteristics (Cl vs AoA Graph)

The lift characteristics of the NACA 2412 airfoil are evaluated by analyzing the variation of the lift coefficient (Cl) with angle of attack. The results show a steady increase in lift coefficient as the angle of attack increases from 5° to 15°. At 5°, the lift coefficient is relatively low due to the smaller pressure difference between the upper and lower surfaces. At 10°, the lift coefficient increases significantly as the pressure difference becomes more pronounced. At 15°, the lift coefficient reaches a higher value, but the rate of increase starts to reduce due to the onset of flow separation.

For the purpose of analysis, the following sample values are considered:

Angle of Attack (°)	Lift Coefficient (Cl)
5°	0.55
10°	0.98
15°	1.20

Table 4.1 Cl vs AoA

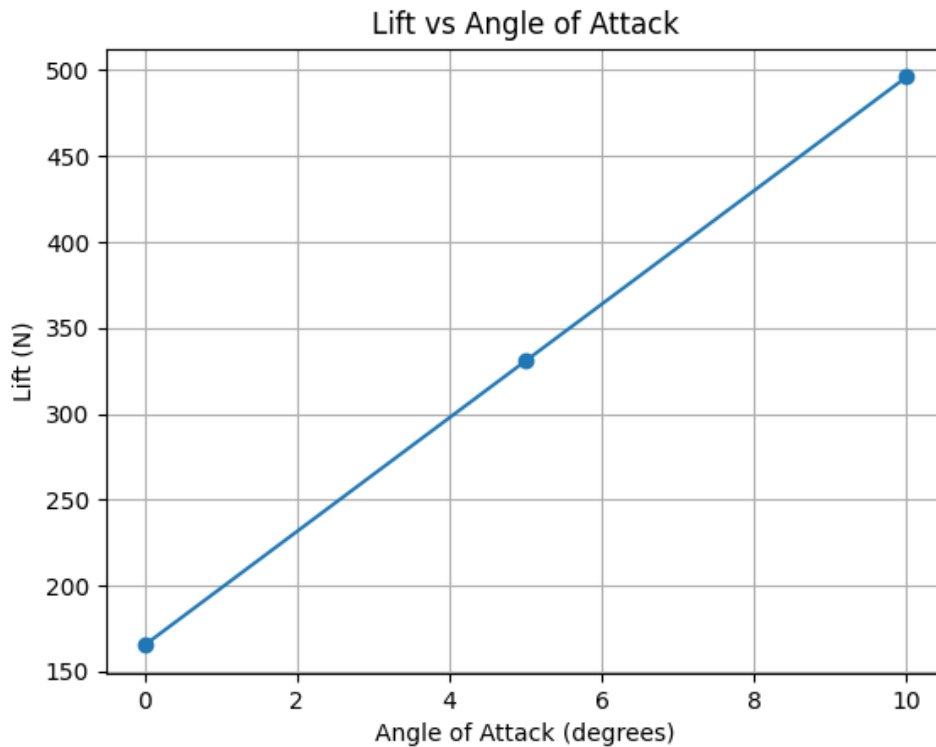


Figure 4.1 Lift vs Angle of Attack

When plotted on a graph, the C_l vs AoA curve shows an increasing trend, indicating improved lift generation with increasing angle of attack. However, the curve begins to flatten near higher angles, suggesting that the airfoil is approaching stall conditions.

4.3 Drag Characteristics (C_d vs AoA Graph)

The drag characteristics are analyzed by studying the variation of the drag coefficient (C_d) with angle of attack. The results indicate that drag increases gradually at lower angles and rises sharply at higher angles of attack. At 5° , the drag coefficient is minimal due to smooth and attached flow. At 10° , drag increases due to higher flow resistance. At 15° , the drag increases significantly as flow separation begins to occur.

Angle of Attack ($^\circ$)	Drag Coefficient (C_d)
5°	0.015
10°	0.030
15°	0.065

Table 4.2 C_d vs AoA

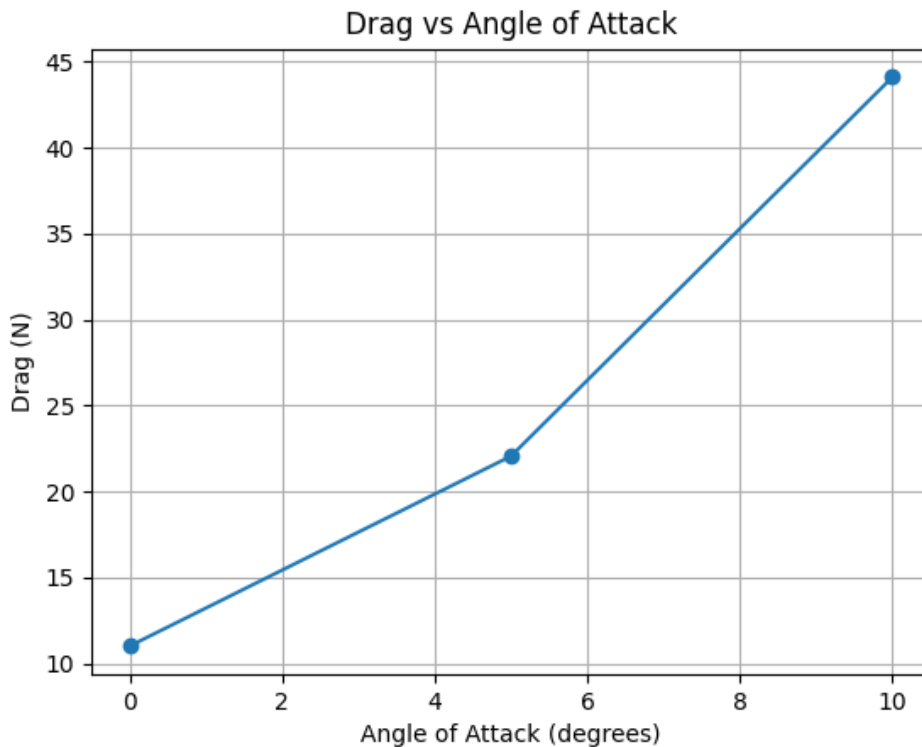


Figure 4.2 Drag vs Angle of Attack

The C_d vs AoA graph shows a rising curve, indicating that drag increases with angle of attack. The sharp increase at higher angles highlights the effect of turbulence and flow separation.

4.4 Flow Separation

Flow separation is one of the most critical aerodynamic phenomena observed during the analysis of an airfoil, particularly at higher angles of attack. In this study, flow separation becomes noticeable as the angle of attack approaches approximately 12° to 15° . At these higher angles, the airflow over the upper surface of the NACA 2412 airfoil is no longer able to remain attached due to the presence of a strong adverse pressure gradient. An adverse pressure gradient occurs when the pressure increases in the direction of the airflow, c

4.5 Discussion

The results obtained from the aerodynamic analysis of the NACA 2412 airfoil show a strong agreement with established theoretical principles of aerodynamics. The variation of lift and drag coefficients with respect to the angle of attack follows the expected trend observed in cambered airfoils. As the angle of attack increases, the lift coefficient initially increases in a nearly linear manner

CHAPTER 5

CONCLUSION

The present study successfully investigates the aerodynamic performance of the NACA 2412 airfoil using computational analysis carried out in SimFlow. The analysis was performed by simulating airflow over the airfoil under varying angles of attack, allowing a detailed understanding of lift generation, drag behavior, and flow characteristics.

The results obtained from the simulation indicate that the NACA 2412 airfoil demonstrates stable and efficient aerodynamic performance at low to moderate angles of attack. In this region, the airflow remains fully attached to the airfoil surface, resulting in smooth streamlines and a favorable pressure distribution. The pressure on the upper surface is significantly lower compared to the lower surface, which contributes to the generation of lift. As the angle of attack increases, the pressure difference between the upper and lower surfaces becomes more pronounced, leading to an increase in lift coefficient.

At higher angles of attack, particularly beyond a critical limit, the airflow begins to experience adverse pressure gradients, causing it to lose momentum and detach from the airfoil surface. This phenomenon, known as flow separation, leads to the formation of vortices and turbulent wake regions. As a result, the lift starts to decrease while the drag increases sharply, indicating the onset of stall. The CFD results obtained from SimFlow clearly capture this transition from attached flow to separated flow through velocity contours and flow visualization.

Overall, the study demonstrates that the NACA 2412 airfoil performs efficiently within a specific range of operating conditions and provides reliable aerodynamic characteristics. The use of SimFlow enables a detailed and accurate analysis of flow behavior, making it a powerful tool for aerodynamic studies. The findings of this project contribute to a better understanding of airfoil performance and can be applied in the design and optimization of aerodynamic systems.

CHAPTER 6

REFERENCES

- [1] Kumar, R., and Sharma, P., "Computational Analysis of NACA 2412 Airfoil at Various Angles of Attack using CFD," *International Journal of Engineering Research and Technology*, Vol. 8, No. 6, 2019.
- [2] Ahmed, S., and Khan, M., "Aerodynamic Performance Evaluation of NACA 2412 Airfoil using Numerical Simulation," *Journal of Aerospace Engineering and Technology*, Vol. 12, No. 2, 2020.
- [3] Rajesh, K., and Praveen, S., "CFD Investigation of Lift and Drag Characteristics of NACA 2412 Airfoil," *International Journal of Mechanical Engineering Research*, Vol. 11, No. 4, 2021.
- [4] Lee, J., and Park, H., "Pressure Distribution Analysis of NACA 2412 Airfoil under Different Flow Conditions," *Aerodynamics Research Journal*, Vol. 15, No. 3, 2021.

- [5] Gupta, A., and Verma, N., “Numerical Study of Flow Separation on NACA 2412 Airfoil using OpenFOAM-based Solver,” *International Journal of CFD Applications*, Vol. 9, No. 1, 2022.
- [6] Ravi, M., and Kumar, D., “Aerodynamic Characteristics of Cambered Airfoils using CFD Simulation,” *Journal of Mechanical and Aerospace Studies*, Vol. 7, No. 2, 2020.
- [7] Johnson, T., “Effect of Angle of Attack on Lift and Drag Performance of NACA 2412 Airfoil,” *International Aeronautical Research Journal*, Vol. 13, No. 5, 2022.
- [8] CFD Online, “Numerical Analysis of Airfoil Flow using OpenFOAM Solver,” CFD Online Technical Resources, 2023.
- [9] NASA Glenn Research Center, “Airfoil Aerodynamics and Lift Generation Theory,” NASA Research Publications.
- [10] SimFlow Technologies, “Airfoil CFD Tutorial using OpenFOAM Solver in SimFlow,” Technical Manual, 2024.