International Research Journal on Advanced Engineering and Management
https://goldncloudpublications.com
https://doi.org/10.47392/IRJAEM.2025.0030
e ISSN: 2584-2854
Volume: 03
Issue:02 February 2025
Page No: 173-177

Aerodynamic Analysis of NACA 2412 airfoil using ANSYS

Namrata Deka Baruah[1], *Dimbalita Deka*[2], *Kalyan Kumar Das*[3]

[1]*PG Scholar, Dept. of ME, Assam Engineering College, Jalukbari, Assam, India.*

[2]*Assistant Professor, Dept. of IPE, Assam Engineering College, Jalukbari, Assam, India.*

[3]*Professor, Dept. of ME, Assam Engineering College, Jalukbari, Assam, India.*

Emails: *ndekab15@gmail.com*[1], *dimbalitadeka.ip@aec.ac.in*[2], *Kkdas1971@gmail.com*[3]

Abstract

Fluid flow analysis over an airfoil is crucial for understanding aerodynamic performance including lift, drag and stability. This work presents a numerical analysis and investigation into the design of airfoils focusing on the NACA 4-digit series. The research employs ANSYS simulation tools to model and analyze the aerodynamics of airfoils. NACA 4-digit airfoil generator is utilized to import the airfoil geometry of NACA 2412 series. A C-shaped computational domain is considered as the fluid domain in which the airfoil geometry is imported. The simulation results provide the variation of pressure and velocity of air while moving over the airfoil. Also, the effects of lift and drag forces are analyzed by plotting the graphs between co-efficient of lift vs time and co-efficient of drag vs time.

Keywords: *NACA 4-Digit Series, Any's Simulation, Co-Efficient of Lift and Drag.*

1. Introduction

In the realm of aerospace engineering and fluid dynamics, the study of aerodynamic analysis plays a pivotal role in understanding the behavior of air as it interacts with solid objects, particularly in the design and optimization of aircrafts. The movement of aircrafts through air, are affected by different factors such as lift, drag, angle of attack etc. Hence to analyze the performance of aerodynamic surfaces, the fundamental concepts of lift, drag, and thrust is essential. Lift is generated primarily through the interaction of airflow with the wing structure, allowing an aircraft to ascend. However, the production of lift is often accompanied by an increase in drag, which acts as a resistive force opposing the motion of the body through the air. These lift and drag forces acts due to pressure variation over the airfoil. The Lift and Drag Coefficients are expressed as follows:

$$C_l = \frac{L}{\frac{1}{2}\rho V^2 c}$$

$$C_d = \frac{D}{\frac{1}{2}\rho V^2 c}$$

Where: C_l is lift coefficient
C_d is drag coefficient
L is lift force
D is drag force
ρ is air density
V is free-stream velocity
c is chord length

The introduction of computational fluid dynamics has revolutionized aerodynamic analysis, allowing for intricate simulations that offer insights into airflow patterns.

1.1 NACA (National Advisory Committee for Aeronautics) Airfoil Series

NACA series refers to a set of standardized airfoil shapes developed by the NACA to useful for designing aircraft wings and other aerodynamic surfaces. Among various series developed by NACA, NACA 4-Digit series was the first major set of airfoils produced by NACA [2]. This is described by

International Research Journal on Advanced Engineering and Management
https://goldncloudpublications.com
https://doi.org/10.47392/IRJAEM.2025.0030
e ISSN: 2584-2854
Volume: 03
Issue:02 February 2025
Page No: 173-177

a 4-digit number.

- **First digit:** Maximum camber as a percentage of the chord length.
- **Second digit:** Position of maximum camber along the chord as a percentage.
- **Last two digits:** Maximum thickness of the airfoil as a percentage of the chord length.

NACA 2412 airfoil is one of the most studied airfoils within the NACA 4-digit series [2]. The aerodynamics analysis of NACA 4-digit series is performed by many researchers [3,4,7]. Matsson J. E. et al. [3] used a 3D printer to fabricate a wing section of the NACA 2412 airfoil and compared the results with CFD simulations using ANSYS Fluent software. In another work, Chandravadan, N [4] conducted an aerodynamic study to examine the effect of airflow around the NACA 2412 airfoil using CFD methods. The airfoil's coordinates were taken from the airfoil database and imported into a geometry modeler. The lift and drag co-efficient were compared at different angle of attack. In another research, the incompressible and compressible flow has been successfully compared through aerodynamics coefficient, velocity magnitude, static pressure, and dynamic pressure contours by exhausting the Spalart-Allmaras disorder theoretical prototypical [6].

2. Problem Description

In the present work, the NACA 2412 airfoil series is considered for numerical experimentation (Figure 1).

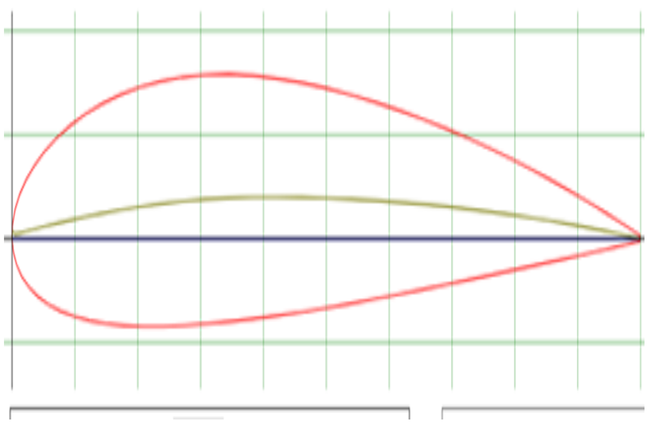

Figure 1 NACA-2421 Airfoil [1]

The analysis of NACA series airfoils involve understanding their design, characteristics, and performance through various methods. The input parameters considered for the particular series are given in Table 1.

Table 1 Input Data for NACA 2412 Airfoil

Series	NACA 2412
Geometry	Taken from NACA 4-digit airfoil generator[1]
Chord	1000mm
Maximum chamber	2%
Maximum chamber position	40%
Thickness	12%
Number of pointers	200
Trailing edge	Closed

For analysis, first the coordinates of the airfoil have to be derived, for which mathematical expressions for both the camber line and the thickness distribution are provided by the NACA 4-digit series.

Camber Line Equation: For a NACA 4-digit airfoil, the camber line equation is:

$$y_c(x) = \frac{m}{p^2}\,[\,2p\,.x - x^2] \quad for \quad 0 \le x \le p \quad (1)$$

And

$$y_c(x) = \frac{m}{(1-p)^2}\,[\,(\,1-2p) + 2p.x - x^2] \quad for\;\; p \le x \le 1 \quad (2)$$

Where: m is the maximum camber.
p is the location of maximum chamber

Thickness Distribution Equation: The thickness distribution equation defines the airfoil's upper and lower surfaces.

International Research Journal on Advanced Engineering and Management
https://goldncloudpublications.com
https://doi.org/10.47392/IRJAEM.2025.0030
e ISSN: 2584-2854
Volume: 03
Issue:02 February 2025
Page No: 173-177

$$y_t(x) = \frac{t}{0.2}[0.2969\sqrt{x} - 0.1260x - 0.3516x^2 + 0.2843x^3 - 0.1015x^4] \quad (3)$$

Where t is the maximum thickness as a percentage of the chord.

3. Methodology

The aerodynamic analysis of the given geometry showed in Figure 2, is performed using ANSYS 2024.

3.1 Geometry Creation

The airfoil geometry taken from NACA 4-digit airfoil generator is imported in Ansys.

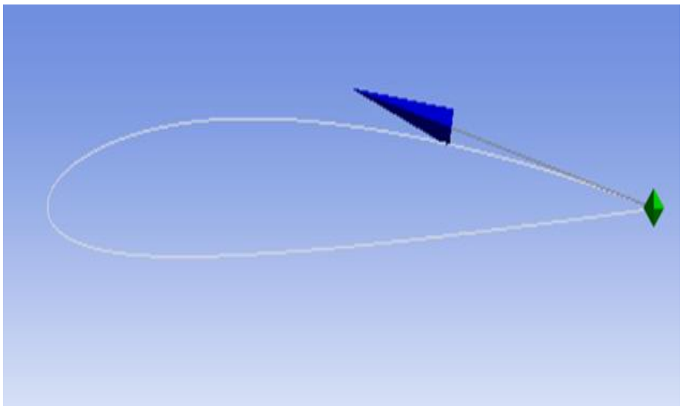

Figure 2 Airfoil Pointer

The fluid domain considered for the analysis is a C-shaped domain as shown in Figure 3. The domain also incorporated the airfoil inside the domain.

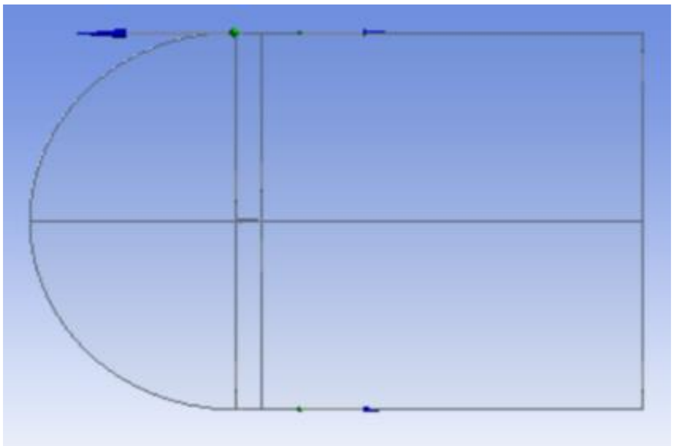

Figure 3 C-Shaped Fluid Domain

The other input parameters related to the domain are shown in the Table 2.

Table 2 Input parameters for Fluid Domain

Shape	C-shape Domain
Radius	7.5m
Length	15m downstream
Width	2r
Mesh type	Quadrilateral grids
Inflation operation	Smooth Transition
Mesh size	50mm
Number of nodes	128319
Number of elements	127625

3.2 Mess Generation

The computational domain is meshed using Ansys meshing. The quadrilateral mesh of size 50 mm is incorporated (Figure 4).

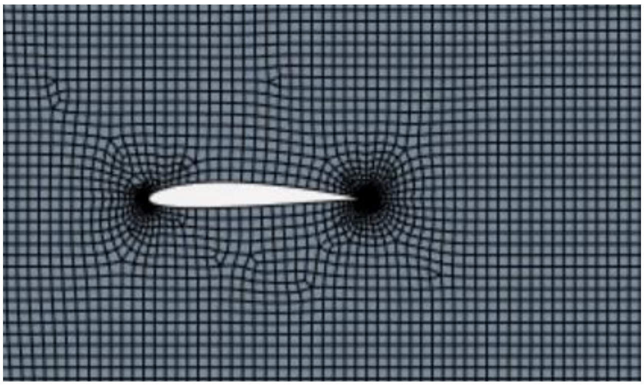

Figure 4 Generated mesh

4. Results and Discussion

4.1. Results

The pressure and velocity distribution over the airfoil is shown in Figure 5(a), (b) and Figure 6(a), (b) respectively. Figure 5(a) and 5(b) shows that the static pressure ranges from -6.64x10^2 Pa to 1.16x10^3 Pa, maximum being at the leading edge and minimum at the upper and lower surfaces of the airfoil. Similarly, the velocity of air ranges from 0 at the surface of the airfoil and maximum 54.6 m/s away from the solid surface of the airfoil as seen in the Figures 6(a) and (b). The velocity distribution over the airfoil surface can be seen more clearly from Figure 7 and Figure 8.

International Research Journal on Advanced Engineering and Management
https://goldncloudpublications.com
https://doi.org/10.47392/IRJAEM.2025.0030
e ISSN: 2584-2854
Volume: 03
Issue:02 February 2025
Page No: 173-177

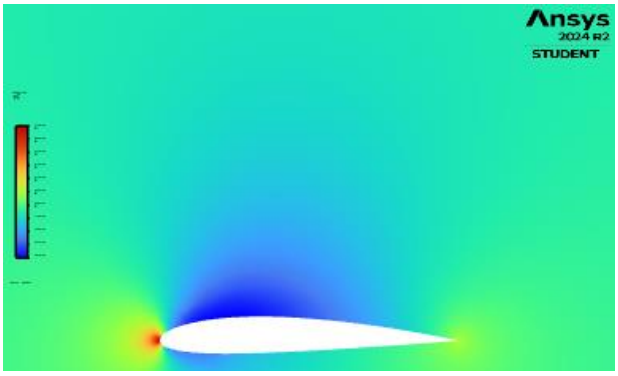

Figure 5(a) **Pressure Contour(Magnified)**

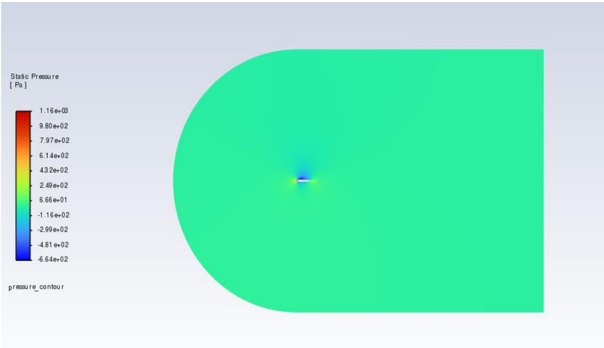

Figure 5(b) **Pressure Contour**

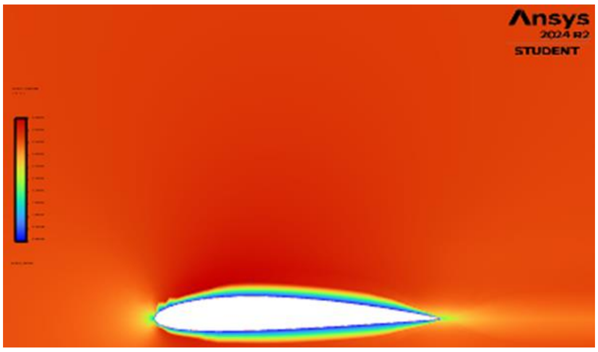

Figure 6(a) **Velocity Contour(Magnified)**

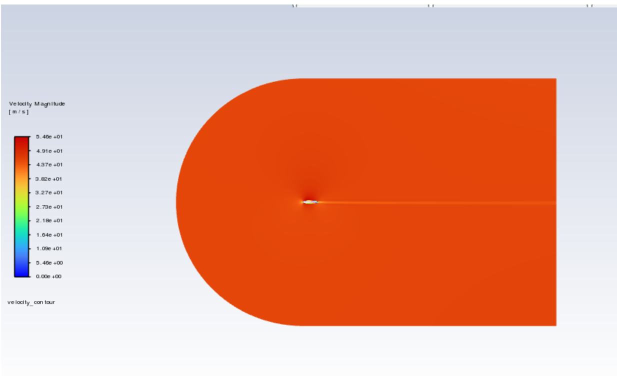

Figure 6(b) **Velocity Contour**

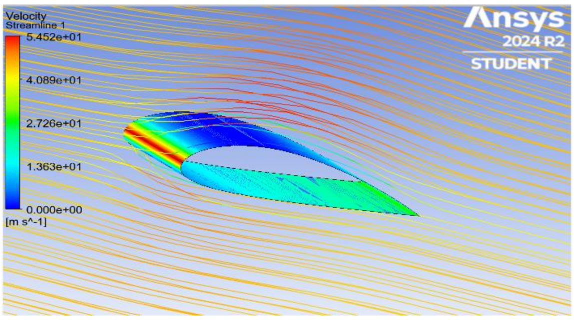

Figure 7 **Velocity Streamlines**

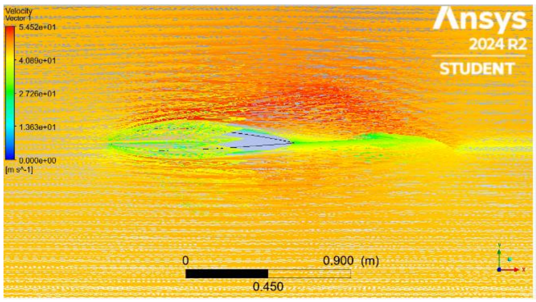

Figure 8 **Velocity Vector**

The variation of co-efficient of drag and co-efficient of lift with time is shown in Figure 9 and Figure 10 respectively, where it can be seen that C_l decreases with time, while C_d increases with time.

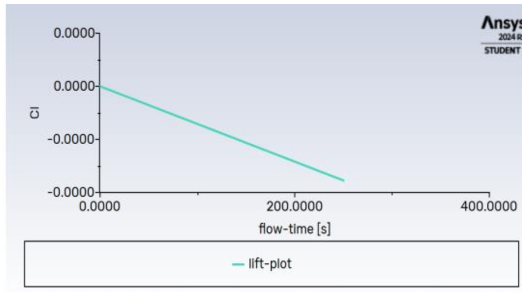

Figure 9 **Variation of Co-Efficient of Lift with Time**

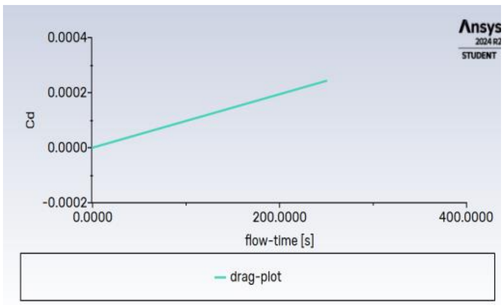

Figure 10 **Variation of Co-Efficient of Drag with Time**

International Research Journal on Advanced Engineering and Management
https://goldncloudpublications.com
https://doi.org/10.47392/IRJAEM.2025.0030
e ISSN: 2584-2854
Volume: 03
Issue:02 February 2025
Page No: 173-177

4.2.Discussion

The Ansys analysis of the NACA 2412 airfoil series demonstrated in the Section 4.1, shows that the pressure above the airfoil surface is lower compared to the edges, which helps to lift the aircraft in the upward direction. Similarly, the velocity contour shown in Figure 6., implies that the velocity at the leading edge is higher and reduces over the surface to the trailing edge due to the drag in the opposite direction. However, the co-efficient of lift decreases with time as with variation of altitude, less air flows over the airfoil, thus reducing production of lift. On the other hand, the drag co-efficient increases with time as the drag force becomes more significant when the aircraft moves faster through air.

Conclusion

The analysis of aerodynamics is important to understand the boundary layer theory of fluid mechanics along with the study of the importance of lift and drag forces of a fluid flowing past over a body. The incorporation of computational fluid dynamics in this regard helps to visualize these effects. Therefore, the present work is taken for study and analysis of the effects of lift and drag forces on an airfoil over which air is flowing. The NACA 2412 airfoil series is considered for investigation. Ansys 2024 is implemented to analysis the aerodynamics behavior of the airfoil, for which a C-shaped fluid domain is considered. The analysis shows that the pressure over the airfoil surface is less so as to enhance the lift while the velocity of air at the surface is almost zero due to no slip condition. However, the variation of lift and drag co-efficient with time also justifies the movement of aircraft through air.

References

[1]. https://airfoiltools.com/airfoil, accessed in September 2024, 2024.

[2]. Allen, Bob (31 January 2017). "NACA Airfoils". nasa.gov. NASA. Accessed in September 2024. 2024.

[3]. Matsson J. E., Voth J. A., McCain C. A., & McGraw C. (2016). Aerodynamic Performance of the NACA 2412 Airfoil at Low Reynolds Num ber. ASEE's 123rd annual conference and Exposition, New Orleans.

[4]. Chandravadan, N. (2024). Numerical Study of Flow Over NACA 2412 Airfoil at Various AOA's. International Journal of Scientific Research & Engineering Trends, 10(3), 1028-1033.

[5]. Guendaoui S., El Ouadefli L., El Akkad A., Elkhalf A., Vlase S., Scutaru M.L. (2024). Comparative Analysis of NURBS and Finite Element Method in Computational Fluid Dynamics Applications: Case Study on NACA2412Airfoil Aerodynamics. Mathematics, 12, 3211. https://doi.org/10.3390/ math12203211

[6]. Kharulaman L., Aabid A., Mehaboobali F.A.G., Khan S.A. (2019). Research on Flows for NACA 2412 Airfoil using Computational Fluid Dynamics Method. International Journal of Engineering and Advanced Technology. 9(1), 5450-5456. DOI: 10.35940/ijeat. A3085.109119

[7]. Rashid F. L., Abd H.S., and Hussein E. Q. (2022). Numerical Study of the Air Flow over Modified NACA 2412 Airfoil using CFD. AIP Conference Proceedings. 2415, 020005 (2022); https://doi.org/10.1063/5.0092303