Computational Analysis of Flow Characteristics over NACA 4412 Airfoil Using CFD simulation at Varying Angles of Attack

Ayon Sarkar[1], Vikash Kumar Mahto[2]
[1]Summer Intern, Talisha Aerospace, Durgapur, West Bengal-713201
[2]Junior Rocket Scientist, Talisha Aerospace, Durgapur, West Bengal - 713201

***Abstract*—Designing and optimizing aircraft wings, rotor blades, and control surfaces effectively requires an understanding of the aerodynamic properties of airfoils. The NACA 4412 airfoil, a popular cambered profile renowned for its smooth stall characteristics and high lift generation, is the subject of this study. ANSYS Fluent was used in a Computational Fluid Dynamics (CFD) study to examine the flow behavior over the airfoil at different angles of attack (AOA) between 0° and 20°. In order to capture lift, drag, and flow separation, the investigation involved mesh generation, solver setup, boundary condition specification, and post-processing. To guarantee accuracy in capturing flow detachment and near-wall effects, mesh quality was prioritized. Plots of velocity vectors, contours of the pressure distribution, and lift and drag coefficient data are among the simulation results. Significant flow separation beyond 15° was indicated by the obvious shift from laminar to turbulent flow and the onset of stall as the angle of attack increased. This analysis offers important information about the NACA 4412 airfoil's stall characteristics and aerodynamic performance, which can be used in aircraft wings, UAVs, and other aerodynamic devices.**

***Index Terms*—NACA 4412 Airfoil, Computational Fluid Dynamics (CFD), Angle of Attack (AOA), Lift Coefficient (Cl), Drag Coefficient (Cd), Flow Separation, Stall Prediction, ANSYS Fluent, Mesh Generation, Velocity Vectors, Pressure Contours, Boundary Layer, Turbulence Modeling, k-ε Model, Aerodynamic Performance, Structured Mesh, Simulation Convergence, Airfoil Analysis, Flow Visualization, Velocity Inlet Boundary Condition**

I. INTRODUCTION

In mechanical and aerospace engineering, airfoils are essential parts that are crucial to lift production and overall aerodynamic performance. The NACA 4412 airfoil is a cambered profile with a maximum chord thickness of 12% and a 4% camber situated 40% from the leading edge. Because of its advantageous lift-to-drag characteristics and delayed stall behavior, this particular profile has been widely used in the design of aircraft wings and blades. Detailed computational analyses are now necessary due to the growing demand for performance optimization in contemporary aerospace systems.

The aerodynamic behavior of the NACA 4412 airfoil at different angles of attack (0°, 5°, 10°, 15°, and 20°) was examined in this study using a thorough CFD-based methodology. The goal is to determine the critical point at which stall starts and assess how the flow characteristics, such as pressure distribution, velocity streamlines, and boundary layer behavior, alter as AOA increases. It is anticipated that the knowledge gained from this study will improve airfoil design and performance forecasting in real-world engineering applications.

II. OVERVIEW

The project involves using ANSYS Fluent software to run a steady-state CFD simulation. Standard coordinates were used to create the NACA 4412 airfoil's geometry, which was then extruded in a 2D fluid domain. To precisely depict the boundary layer and wake behavior, the simulation domain was discretized using a structured mesh and then fine-tuned using curvature and proximity settings. To enhance near-wall resolution, particularly in the vicinity of the airfoil's leading and trailing edges, edge sizing and inflation layers were used.

The working fluid was air at sea-level atmospheric pressure and temperature, and the flow was simulated under incompressible, steady-state conditions.

Multiple angles of attack were tested by changing the direction of the freestream velocity vector relative to the airfoil. Aerodynamic coefficients, pressure and velocity fields, and visual signs of stall, like flow separation and reversed flow regions, were extracted from the results through post-processing.

III. METHODOLOGIES

This section describes the methodical procedure used to perform the ANSYS Fluent computational analysis of the NACA 4412 airfoil. Creating the geometry, defining the domain, creating the mesh, setting up the solver, running the simulation, and post-processing the results were all sequential steps in the study. Replicating realistic aerodynamic conditions, accuracy, and mesh quality were prioritized.

1. Airfoil Geometry and Domain Setup

Using NACA-provided coordinate datasets or standard equations, the NACA 4412 airfoil geometry was produced. At 40% of the chord from the leading edge, this airfoil has a 4% camber and a maximum thickness of 12%. The geometry was created as a 2D profile in ANSYS DesignModeler. A rectangular computational domain was made in order to model the airflow surrounding the airfoil:

- Ten times the chord length is needed upstream (at the inlet) to guarantee consistent freestream conditions.
- Twenty times the chord length is needed downstream (at the outlet) to account for wake development.
- Top/Bottom Boundaries: To reduce the effects of wall interference, keep the chord distance from the airfoil at least ten times.

2. Mesh Generation:

Since mesh quality has a direct impact on the accuracy and convergence of results, mesh generation is an essential step in CFD simulation. ANSYS Meshing was used to implement the following meshing strategy:

- Mesh Type: Quadrilateral-element structured mesh for improved accuracy.
- Element Count: A total of 166,967 nodes and roughly 166,000 elements were produced.
- Inflation Layers: Added near the airfoil surface to resolve boundary layer effects with better precision. A smooth transition was applied using 2–3 inflation layers.
- Edge Sizing: To efficiently capture curvature and pressure gradients, finer sizing was applied close to the leading and trailing edges.
- Skewness Control: Skewness and orthogonal quality were maintained within acceptable limits (skewness < 0.25).
- Mapped Face Meshing: Used to create a semi-structured mesh in the domain for improved grid alignment with the flow.

3. Solver Setup in ANSYS Fluent:

The meshed geometry was imported into ANSYS Fluent, and the following physical and numerical models were defined:

3.1 Solver and Models

- Solver Type: Pressure-based, steady-state solver.
- Turbulence Model: Realizable k-ε model with enhanced wall treatment, suitable for external flow over an airfoil and capable of capturing flow separation and reattachment.
- Viscous Model: Activated based on Reynolds number and turbulence levels.

3.2 Material and Fluid Properties

- Working Fluid: Air (incompressible, Newtonian).
- Density = 1.225 kg/m^3
- Dynamic viscosity = 1.7894×10^{-5} Pa·s
- Operating conditions were set to sea-level atmospheric pressure.

3.3 Boundary Conditions

- Inlet: Velocity inlet, typically between 30–60 m/s (depending on test case), with direction adjusted to simulate various angles of attack (AOA).
- Outlet: Pressure outlet (gauge pressure = 0 Pa).
- Airfoil Wall: No-slip condition.
- Top and Bottom Boundaries: Treated as frictionless symmetry boundaries or walls, depending on domain distance.

3.4 Initialization and Convergence

- Initialization: Hybrid initialization method.
- Convergence Criteria:

Residuals: < 1e-5 for continuity, x-velocity, y-velocity, and turbulence equations.

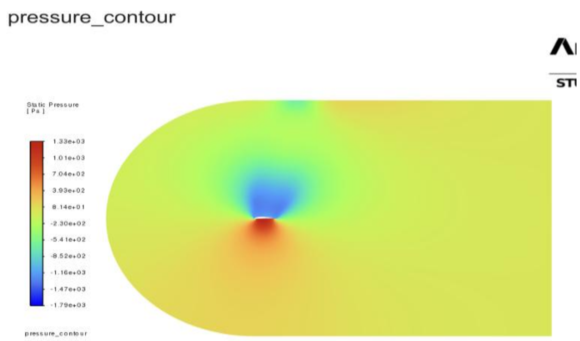
Monitored Parameters: Lift and drag coefficients.

- Solution was advanced until all residuals flattened and aerodynamic coefficients reached steady values.

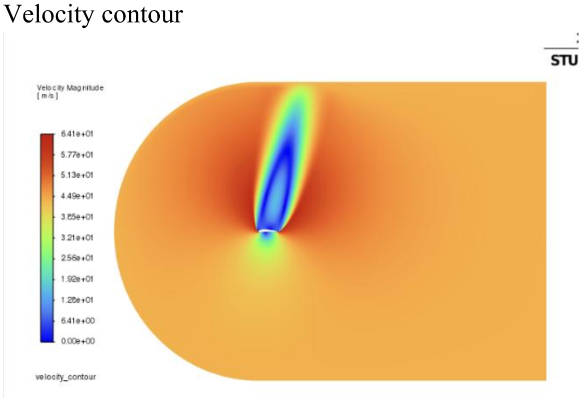
4. Post-Processing and Analysis

After convergence, flow field data were analyzed using ANSYS Fluent's post-processing tools:

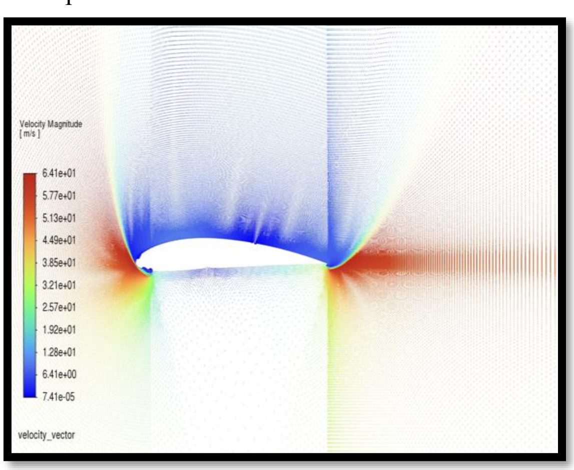
- Pressure Contours: Used to visualize pressure differences on the upper and lower surfaces of the airfoil, which directly relate to lift generation.



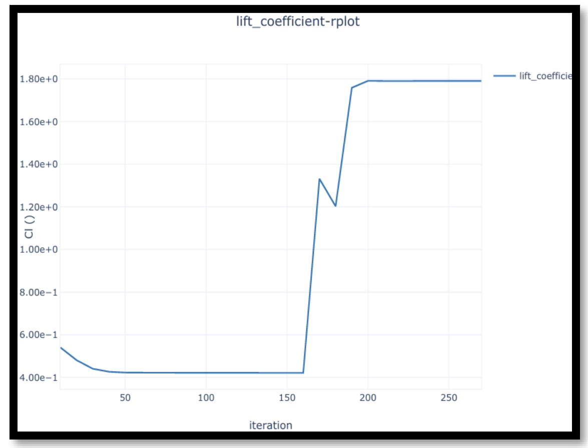
Velocity contour



- Velocity Vectors: Showed the direction and magnitude of flow, useful for detecting flow separation and wake formation.



- Streamline Plots: Highlighted areas of flow attachment and detachment, indicating boundary layer behavior.
- Lift and Drag Coefficients: Calculated using the Fluent force report and plotted against AOA to identify the stall point and performance trends. Lift coefficient 1.791092 and drag coefficient 0.3620392



The NACA 4412 airfoil's aerodynamic properties under various flight conditions were reliably and accurately represented thanks to this exacting methodology. In particular, near-wall effects and separation zones that are crucial for stall prediction were captured by the simulation setup and meshing strategies.

IV. FINAL RESULTS

The outcomes of the simulation demonstrate how different angles of attack affect the NACA 4412 airfoil's aerodynamic properties:

At 0° AOA:

- Symmetric flow with no lift generated due to the absence of camber-induced asymmetry.
- Low pressure difference between upper and lower surfaces.

At 5° and 10° AOA:

- Significant lift generated due to favorable pressure gradient.
- Boundary layer remains attached with minor flow deviation.
- Velocity vectors showed acceleration on the upper surface.

At 15° AOA:

- Clear signs of flow separation near the trailing edge.
- Lift reaches its peak while drag starts to increase.
- Pressure contours exhibit low pressure on the upper surface and stagnation near the leading edge.

At 20° AOA:

- Large-scale flow separation occurs, leading to a stall condition.
- Lift drastically decreases and drag increases.
- Reverse flow region appears in the velocity vector plot.

Graphical Representations:

- Pressure contours showed reduced pressure at the upper surface correlating with lift.
- Velocity vectors clearly showed regions of flow separation at high AOA.
- Lift vs. AOA graph: Peaks near 14–15°, followed by sharp drop.
- Drag vs. AOA graph: Gradually increases, then spikes post-stall.

For the following reasons, it is crucial to comprehend how the airfoil reacts to different angles of attack:

- Performance Optimization: Fuel efficiency and maneuverability are improved by knowing the precise AOA range for the ideal lift-to-drag ratio.
- Control and Stability: For precise reaction during flight, aircraft control surfaces depend on consistent lift behavior.
- Design Safety Margins: Pilots and autopilot systems can operate within safe bounds when stalls are accurately predicted.

V. IMPLEMENTATIONS OF THE SOLUTION

The outcomes of this computational study provide a solid foundation for multiple real-world applications:

a) Aircraft Design: Helps in selecting the optimal angle of attack for maximum lift during takeoff and landing.
b) Control Surface Design: Insights into stall behavior are essential for sizing flaps and ailerons.
c) UAV and MAV Optimization: Can be used for airframe design under varying mission profiles.
d) CFD Validation: Offers an academic benchmark for validating future CFD studies and software.
e) Engineering Education: A valuable teaching tool for students learning about fluid dynamics, CFD, and aerodynamics.

By carefully defining boundary conditions, fine-tuning the mesh, and modeling turbulence, the simulation accurately simulates real-world conditions. The results are intended to provide high-fidelity flow predictions by utilizing fine mesh settings and guaranteeing mesh quality (skewness, orthogonality).
The NACA 4412 airfoil's aerodynamic behavior is thoroughly examined in this work, and the knowledge acquired can be applied both practically and academically to the design of aerodynamic surfaces in the aerospace sector.

VI. FUTURE SCOPE

Machine Learning Integration**:** To speed up simulations and increase prediction accuracy, CFD may be combined with AI and machine learning. By quickly reconstructing flow fields for novel airfoil designs or situations using neural networks trained on existing CFD data, the computational cost can be decreased by orders of magnitude.

Aeroelasticity: Investigating the interaction between the aerodynamic forces and the structural dynamics of the airfoil. This is crucial for understanding phenomena like flutter and divergence, especially for flexible wings.

Three-Dimensional Analysis: The majority of foundational research is two-dimensional. By taking into consideration wingtip vortices and other three-dimensional effects, a comprehensive 3D analysis of the NACA 4412 airfoil would yield more accurate results.

Fluid-Structure Interaction (FSI): To represent how the airfoil deforms under aerodynamic stresses and how that deformation impacts the airflow, a thorough FSI study would combine a fluid dynamics simulation with a structural simulation.
Shape optimization: It is the process of automatically identifying the best airfoil designs that optimize the lift-to-drag ratio or satisfy other performance requirements by combining optimization methods, such as genetic algorithms, with CFD.

VII. CONCLUSION

The following is a typical result drawn from a CFD analysis of a NACA 4412 airfoil at different angles of attack:

Drag and Lift: At the critical or stall angle, the coefficient of lift achieves its highest value after increasing with the angle of attack. The lift coefficient falls after this point. Though frequently more drastically than the lift coefficient. The angle of attack at which the lift coefficient is at its highest is known as the stall angle. Usually, this is between 14 and 16 degrees for the NACA 4412 airfoil. It might be risky to fly past this angle because of flow separation and lift reduction.

Pressure and Velocity: The cambered airfoil's bottom surface has a larger static pressure than its upper surface when the angle of attack is at zero degrees. This pressure differential becomes more noticeable as the angle of attack rises, producing more lift. On the other hand, lower pressure is associated with a higher airflow velocity across the top surface.

VIII. ACKNOWLEDGMENT

I would like to express my sincere gratitude to all those who supported and guided me throughout the completion of this project titled "Computational Analysis of Flow Characteristics over NACA 4412 Airfoil Using CFD simulation at Varying Angles of Attack".
First and foremost, I would like to thank Mr. Utsav Chakraborty, my project supervisor, for their continuous encouragement, insightful suggestions, and valuable technical guidance, which were instrumental in shaping the direction and outcome of this work.
I am also thankful to the Department of Research and Development, Talisha Aerospace, for providing access to essential resources and facilities, including the ANSYS Fluent software, which played a crucial role in performing the CFD simulations for this study.
Special thanks to my fellow teammates and friends for their cooperation, discussions, and feedback throughout the course of the project.
Lastly, I am grateful to Talisha Aerospace company for their constant motivation and unwavering support during this work.

REFERENCES

[1] $Chakraborty, U.$ and Paul, S., 2024. Influence of Surface Roughness on Symmetric Airfoil Flow Properties and Reynolds Number.
[2] author = {Chakraborty, Utsav and Marwaha, Kunwarpreet and Vero, Rhoto},
year = {2024},
month = {04},
pages = {},
title = {Effects of Simulated Surface Roughness on Flow Characteristics in Symmetrical Airfoil},
doi = {10.55041/IJSREM30823}
[3] NASA NACA Airfoil Series Technical Reports. https://ntrs.nasa.gov.
[4] Anderson, J.D. (2010). *Fundamentals of Aerodynamics*, 5th Ed., McGraw-Hill Education.
[5] Exergy based modeling and optimization of solar thermal collector provided with impinging air jets