Aerodynamic Analysis of NACA Airfoils Using CFD and MATLAB

Aryan Yenni

3 July 2025

ABSTRACT

I present computational analysis of the aerodynamic performance of 2-dimensional NACA airfoils across a range of angles of attack. Using the SimScale platform, alongside the Onshape product development platform, steady-state incompressible computational fluid dynamics (CFD) simulations were conducted to visualize and quantify the airflow behavior around selected airfoils. For each configuration, certain key aerodynamic characteristics, such as stall behavior and efficiency, were evaluated through attaining the lift and drag coefficients. The results were then processed and plotted using MATLAB, producing C_L vs. α, C_D vs. α, and the drag polar (C_L vs. C_D) plots to compare performance across different angles and airfoil geometries. The study demonstrates how airfoil shape and angle of attack influence aerodynamic response, particularly to generate lift and minimize drag. The project displays the effectiveness of combining modern CFD tools with computational analysis to investigate classic airfoil designs and support engineering decision-making in early aerodynamic design stages.

Key words: Computational fluid dynamics (CFD), lift and drag coefficients, pressure and shear stress, SimScale, MATLAB

1. INTRODUCTION

Aircraft and airfoils depend on the behavior of moving air, or airflow, over their surfaces to generate the forces needed for flight. The interactions between fluid flow and solid bodies mark the foundation of aerodynamics. Understanding how these forces arise and change with shape and orientation is crucial in developing models for wings and entire aircraft.

The key to the interaction is the airfoil, a two-dimensional cross section of a wing, whose geometry determines how air flows along its surface, dictating the forces of lift, drag, and induced moment. These forces are often reported with non-dimensional coefficients, allowing engineers to determine the performance of an airfoil purely from its geometry, and independent of scale, speed, and air density. These include the lift coefficient (C_L), drag coefficient (C_D), and the moment coefficient (C_M), among other crucial non-dimensionals.

$$C_L = \frac{L}{\frac{1}{2}\rho V^2 S_{ref}}, \quad C_L = \frac{D}{\frac{1}{2}\rho V^2 S_{ref}}, \quad C_L = \frac{M}{\frac{1}{2}\rho V^2 c S_{ref}}.$$

Equation 1. L, D, and M are the lift, drag, and pitching moment, ρ is air density, V is freestream velocity, S is the reference area, and c is the chord length. The freestream velocity refers to the difference between the air velocity and the aircraft's velocity, or the aircraft's velocity relative to the surrounding air.

The pressure and shear viscous force that happen locally on the surface of the airfoil generate the global body forces, known as lift, drag, and pitching moment.

$$\text{Pressure: } p(x, y)$$
$$\text{Shear Stress: } \tau(x, y)$$

$$\text{Aerodynamic Force}$$
$$A = \iint_{S_{body}} (-p\hat{n} + \tau)\ dS$$

Equation 2. The total aerodynamic force over the surface can be found using the surface integral, where $-p\hat{n}$ is the pressure stress normal and downward into the surface, τ is the wall shear stress tangential to the surface, and dS is the infinitesimal surface element.

The lift is characterized as a force normal to the freestream velocity, typically pointing upward on an aircraft airfoil to oppose the force of gravity. The drag force directly opposes the direction of motion, acting as a sort of friction. Finally, the moment, or torque, contributes to the stability of the aircraft. The lift and drag forces acting on the airfoil means that there is rotational force about the aircraft's center of mass, which must be stabilized with another airfoil near the aircraft's tail.

To quantify the effects, engineers keep certain parameters constant, such as the freestream velocity. At a low Mach number, where $M_\infty < 0.3$, the air is virtually incompressible, meaning that its air density is constant, rather than changing with position.

$$M_\infty = \frac{v}{a}.$$

Equation 3. The Mach number is a non-dimensional parameter determining the behavior of flow, giving the ratio of the freestream velocity to a_∞, the speed of sound in the freestream. While keeping the freestream velocity and airfoil geometry constant, engineers change the angle of attack, or orientation of the airfoil's chord line with respect to the incoming airflow.

As the angle increases, the lift generally increases up to a certain point, known as stall. An even greater angle causes the air to detach from the upper surface of the airfoil, resulting in a stall. Understanding this behavior is key in assessing an airfoil's performance, defining its ability to create lift, minimize drag, and maintain stability throughout flight.

This project uses computational tools to simulate these behaviors numerically. SimScale, a cloud-based CFD platform, is used to solve the flow field around several airfoils at different angles of attack. The resulting lift and drag coefficients are extracted and visualized in MATLAB, providing a quantitative comparison of how airfoil shape influences aerodynamic performance.

2. METHODOLOGY

This study uses computational fluid dynamics (CFD) to analyze aerodynamic performance of two-dimensional airfoils across a range of angles or attack. All simulations were conducted using SimScale, a cloud-based platform capable of solving the Navier-Stokes equations for various airflow regimes. The simulation results were processed using MATLAB, where coefficients and force distributions were visualized and analyzed.

1. AIRFOIL GEOMETRY

A selection of standard NACA 4-digit airfoils was used to compare performance across different shapes.

Used throughout aerodynamics, NACA 4-digit airfoils give the maximum camber as a percentage of the chord (first digit), the position of the maximum camber as a tenth of the chord (second digit), and the maximum thickness as a percentage of the chord (final two digits).

The three airfoils to be used in the study will be NACA 0012, 2412, and 4424. Note that NACA 0012 does not have a maximum camber, therefore it cannot have a position for the maximum camber, meaning that it is symmetrical across its chord line.

SimScale's compatibility with Onshape, a cloud-based CAD and product development platform, allows CAD exports of each airfoil to be moved into the simulation easily. The CAD files of each airfoil is taken from the UIUC Airfoil Database and extruded manually.

2. SIMULATION SETUP

The simulations were configured as state-ready, incompressible flows, appropriate and accurate for low-speed subsonic conditions, namely $M_\infty < 0.3$ where compressibility is negligible. The $k - \omega$ turbulence model was selected with steady-state time independence to capture both attached and detached airflow regions with good near-wall resolution.

The boundary conditions are crucial. There is a velocity inlet with flow directed at varying angles of attack, adjusted by changing the inlet vector. A pressure outlet is set to 0 Pa, and no-slip walls on the surface. A no-slip wall is a boundary condition enforcing a viscous fluid to attain zero flow velocity when adjacent to a solid boundary. Finally, there are slip, or symmetry conditions, at the top and bottom surface.

In the various simulations, the angle of attack is altered by tweaking the inlet vector rather than physically rotating the airfoil within the flow field. This ensures that the mesh along the surface remains consistent throughout trials, reducing variability in results from due to grid formation.

3. MESH GENERATION

The computation mesh along the airfoil surface was automatically generated using SimScale's SIMPLE meshing algorithm for steady-state time independence. There are local surface refinements so that the mesh is density depends on the geometry rather than refining the mesh consistently over the entire airfoil.

Boundary layer inflation is added around the airfoil, simulating thin, inflation layers adjacent to the boundary wall of the airfoil in the mesh. This is done to accurately depict the behavior of the boundary layer, a thin region where shear viscous forces dominate. The combination of these two creates an accurate resolution of pressure stress and near-wall viscous forces which are critical in understanding skin friction and separation behavior.

4. GLOBAL FORCES

The global forces on an airfoil are the aforementioned lift, drag, and moment forces that are caused from the local pressure stress and viscous shear that happen when the airfoil interacts with airflow. The lift and drag are components of the aerodynamic force, derived from integrating the pressure and viscous shear across the surface of the airfoil. SimScale computes this lift and drag forces, extracting their coefficients. These extracted coefficients, along with their corresponding angles of attack, are exported to MATLAB for further analysis.