Aerodynamic Analysis of NACA Airfoils Using CFD and MATLAB

A NACA Airfoil CFD Study

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 SimScale, a cloud-based computational fluid dynamics (CFD) software, alongside the Fusion product development platform, steady-state incompressible CFD simulations were conducted to visualize and analyze the airflow behavior around selected airfoils. For each simulation run, global and local forces were attained to analyze their relationships with the angle of attack. The results are 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 local force data is converted to the arc length parameter s to understand forces along the surface. 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, Fusion, 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 tear-drop cross-sectional shape 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 (C_L) and drag coefficient (C_D), among other crucial non-dimensionals.

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

Equation 1. C_L and C_D are the lift and drag coefficients, ρ is air density, V is freestream velocity, S is the airfoil's reference area, and c is the airfoil's 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.

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

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 (Massachusetts Institute of Technology).

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 like friction. Additionally, the lift creates moment, or torque, about the aircraft's center of mass, contributing to the stability of the aircraft. This rotational force about the center of mass must be stabilized with another moment of equal and opposite direction, usually provided by an additional airfoil at 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 before evening out and dropping, known as stalling. 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.

2.1 *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 is compatible with Fusion's ability to export CAD models as STEP files, allowing CAD exports of each airfoil to be moved into the simulation easily. The CAD files of each airfoil are taken from the UIUC Airfoil Database as .dat files, converted into SVG files, and extruded manually in fusion.

2.2 *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. I selected the $k - \omega$ turbulence model due to its time independence, capturing 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.

2.3 *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.

2.4 *Global Forces*

The global forces on an airfoil are the 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.

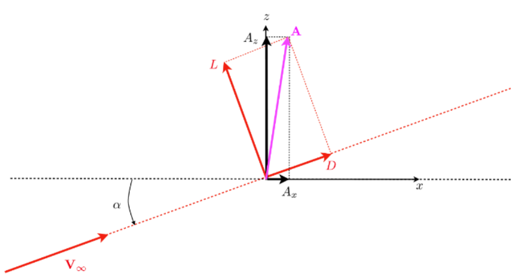

Figure 1. V_∞ refers to the freestream velocity. Note that drag is the component of aerodynamic force in the direction of freestream velocity, and the lift is the component normal to it. Common notation dictates that the airfoil geometry exists in the xz-plane, as the y-axis is reserved as the span-wise direction (Massachusetts Institute of Technology).

2.5 *Local Forces*

Local pressure and walls stress distributions along the airfoil allow a deeper understanding of the surface forces. Traditionally, data returned by the CFD are discrete values in Cartesian coordinates (x, y), where $p(x, y)$ and $\tau(x, y)$ are the pressure and shear stress at specific points on the airfoil surface.

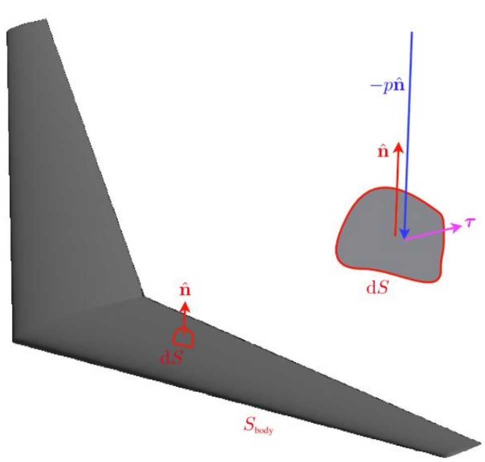

Figure 2. As seen previously, the pressure stress pushing downward and normal to the surface, combined with the shear stress working tangentially to the surface, integrated over the surface, yields the total aerodynamic force that can be separated into lift and drag (Massachusetts Institute of Technology).

For simplicity, engineers often convert stress data from Cartesian coordinates (x, y) into stress data as a function of the arc length s. This technique offers a clearer representation of the forces' effects on the surface as a function of the boundary's length.

In MATLAB, the local force data is mapped to the arc length parameter s. By definition, pressure stress is resolved as normal, while the viscous shear stress is resolved as tangential to the surface. When local forces are parameterized using s and separated into normal and tangential components, they can be compared to global body forces easily.

3. COMP. FLUID DYNAMICS

Standard NACA airfoil profiles were selected from the UIUC Airfoil Coordinates Database, titled Lockheed L-188/P-3 tip airfoil NACA 0012, NACA 2412 airfoil, and NACA 4412 respectively. The .dat files from this database contain a set of non-dimensional (x, z) coordinate points defining the upper and lower surfaces of the airfoil, where x ranges from 0 to 1, and z defines the camber/thickness. Note that the y-axis is reserved as the span-wise direction, or the direction of extrusion.

The coordinates were redefined from .dat files to .DXF files, then plotted on the xz-plane in Fusion to create a closed 2D airfoil sketch, consistent with conventional notation. Each airfoil is scaled by the standard unit of millimeters, extruded along the y-axis by $10.00\,mm$, and exported in .STEP format, which is widely accepted in professional CFD software.

The resulting solids were then imported into SimScale, where the fluid domain (airflow field) and simulation settings are set up. The airfoils exist as continuous, nearly 2D solids that can be placed into the surrounding flow field.

3.1 *Domain Setup*

The flow domain, the region where the airflow field exists within the CFD, is created around the airfoil geometry. The domain, in comparison to the airfoil, is large enough in all directions to have negligible boundary effects and replicate the freestream accurately. The airfoil will be oriented at the center, while the inlet and outlet of the fluid will be above and below respectively.

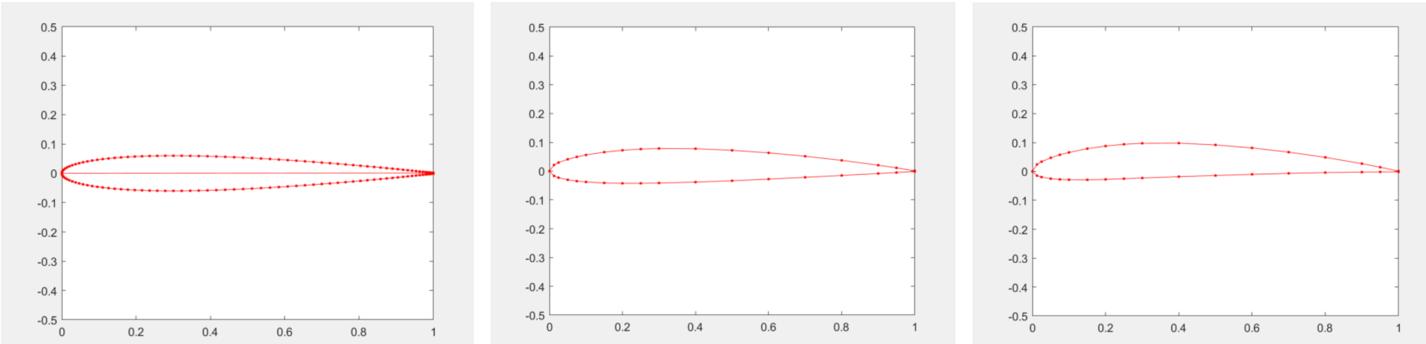

Figure 3. Using MATLAB, airfoil coordinates from the UIUC database are plotted with straight-line connectors, forming the shape of the NACA 0012, 2412, and 4412 airfoils. Note the symmetry of the first airfoil, rightly denoted by its first two digits being 0. The second and third airfoils have their maximum camber occurring at 0.4, or 4/10 of the chord line, denoted by the second digit 4. These coordinates are then opened in Fusion to be extruded.

The external flow volume is a volume of space surrounding the geometry of the object where the fluid flows. In SimScale, the external flow volume is established around the geometry, with the boundaries being the default $(x_{\min}, x_{\max}) = (-0.1, 0.4)\ m$, $(y_{\min},\ y_{\max}) = (-0.25, 0.25)\ m$, and $(z_{\min}, z_{\max}) = (-0.25, 0.25)\ m$ for NACA 0012, and $(x_{\min}, x_{\max}) = (-0.05, 0.15)\ m$, $(y_{\min},\ y_{\max}) = (-0.05, 0.05)\ m$, and $(z_{\min}, z_{\max}) = (-0.05, 0.05)\ m$ for NACA 2412 and 4412, with the airfoil geometry being located at $(0, 0)\ m$. This ensures that the geometry of the object is completely enclosed within the flow volume, while also avoiding volumes that are unnecessarily large for computation.

Using Boolean, the geometry is subtracted from the flow volume, so that air exists around but not within the airfoil. Once saved, the new model with the flow volume is used to create the simulation, while deleting the original model.

The orientation of the airfoil points the leading edge towards the negative *x*-axis, and the trailing edge towards the positive *x*-axis. This allows the velocity inlet of the airflow to point toward the positive *x*-axis. With the simulation testing subsonic flow, the inlet velocity will be $50\ m/s$ in the positive *x*-axis. To change the angle of attack without changing the velocity magnitude, the velocity inlet is altered using trigonometry.

$$\vec{U}_{inlet} = \left(50 \times \begin{bmatrix} cos(\alpha) \\ 0 \\ sin(\alpha) \end{bmatrix}\right)\ m/s.$$

Note again that the *y* direction is reserved for the span-wise direction, therefore *x* and *z*-directions are responsible for the direction of airflow. Each airfoil will have simulations run at the following angles:

$$\alpha = \{-5°, -2.5°, 0°, 2.5°, 5°, 7.5°, 10°, 12.5°\}.$$

The first two oppose the flow direction, [0, 10] shows linear increase, and the final two approach stall. With each airfoil having 8 angles, the combined runs total to 24 simulations.

3.2 *Initial Conditions & Solver Settings*

The simulations use the $k - \omega$ SST turbulence model, a two-equation eddy-viscosity model that is a widely accepted industry standard turbulence model. The model accounts for near-wall effects, determining how the flow interacts with the body. Because the overall flow is subsonic and incompressible, the flow is laminar near the wall, dominated by viscous forces in the viscous sublayer (Wang et. al.).

The material, of course, will be air, which has a kinematic viscosity $\nu = 1.529 \cdot 10^{-5}\ m^2/s$, a density $\rho = 1.196\ kg/m^3$, a velocity inlet $\vec{U} = \langle 0,0,0 \rangle\ m/s$, turbulent kinetic energy $k = 3.75 \cdot 10^{-3}\ m^2/s^2$. There is a global gauge pressure of 0 *Pa* and specific dissipation rate $\omega = 3.375\ 1/s$.

For solving the velocity, turbulent kinetic energy, and the dissipation rate, the $k - \omega$ SST turbulence model features the PBiCStab solver, a Preconditioned Bi-Conjugate Gradient Stabilized algorithm that solves large systems of linear equations of the form $\boldsymbol{A}\vec{\boldsymbol{x}} = \vec{\boldsymbol{b}}$, using the inverse method.

$$\boldsymbol{A} \cdot \boldsymbol{\Psi} = \boldsymbol{b},$$
$$(\boldsymbol{M}^{-1} \cdot \boldsymbol{A}) \cdot \boldsymbol{\Psi} = \boldsymbol{M}^{-1} \cdot \boldsymbol{b}.$$

The equation uses a matrix $\boldsymbol{M}$ known as a preconditioner, similar to the original matrix $\boldsymbol{A}$, which can be inverted more easily. The original matrix includes convection, diffusion, and pressure gradient terms among its entries (Weller).

The unknowns are denoted with $\boldsymbol{\Psi}$, and $\boldsymbol{b}$ the knowns. Unknowns include pressure, temperature, components of fluid velocity, and temperature, while the known terms are derived from boundary conditions, external forces, and anything not directly related to the unknown variables (Weller).

Before running the simulation, certain result control is needed. First, forces and moments are placed throughout the flow region, except for the walls, which are slip walls. A slip wall is a surface that allows fluid to slide along it without an induced shear stress or velocity difference.

3.3 *Output Data, Figures, & Result Fields*

Upon the completion of each run, SimScale can produce outputs based on result control: (1) global outputs of lift, drag, and moment, (2) flow field data of pressure, velocity, and turbulence in the domain, and (3) local surface data of wall pressure $p(x, y)$ and wall shear stress $\tau(x, y)$, usually split into its x and y coordinates. Note that the x and y coordinates refer to the x and z coordinates of the traditional orientation of airfoils, as the y direction is reserved for the span-wise direction.

Let us observe the air's velocity magnitude about NACA 2412. This run has the inlet velocity moving purely in the positive x-direction at 50 m/s, meaning that $\alpha = 0^\circ$.

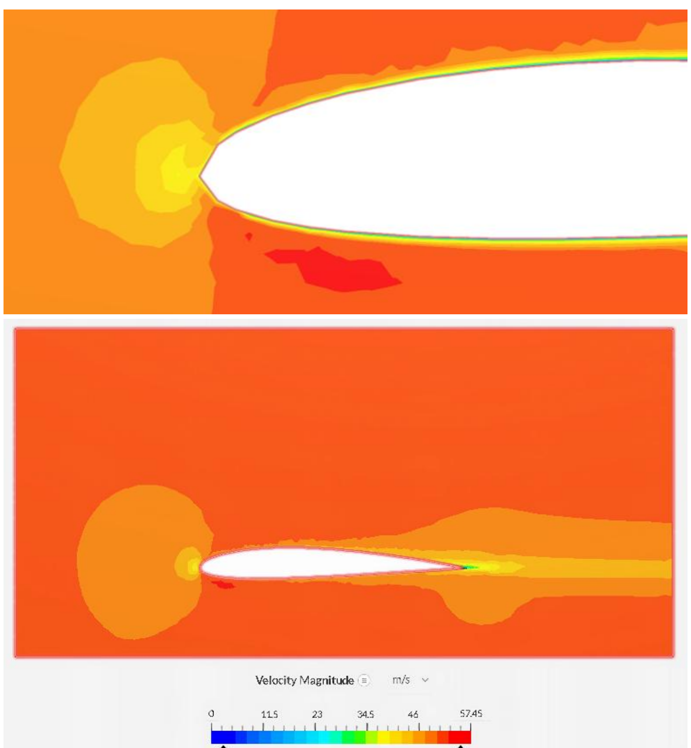

Figure 4. Most of the frame is a uniform orange red, suggesting that the velocity is a consistent 50 m/s. At the leading-edge, there is a bubble of orange becoming yellow, suggesting that air is slowing, or decelerating as it comes into its initial contact with the airfoil surface. Notice in the second close-up frame at how there is a narrow blue-green gradient stripe along the surface of the airfoil. The velocity is greater along the top surface, meaning that pressure is higher on the underside, creating lift. Furthermore, there is a wake past the trailing edge as the flow separates from the surface. These are all results of Bernoulli's principle of fluids, relating their velocity and pressure near solids.

Now, let us look at the pressure to confirm this.

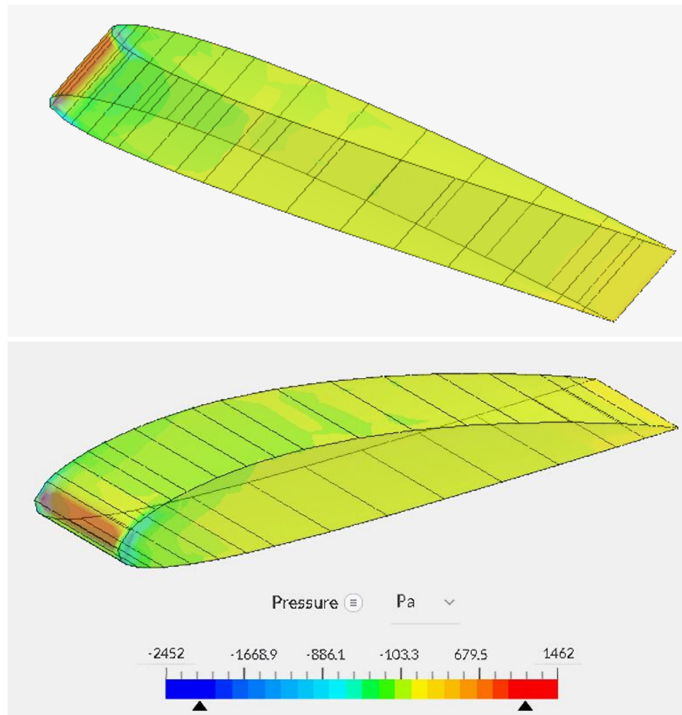

Figure 5. The leading edge undergoes very high pressure, taking the airflow head-on. The top of the airfoil is quite a bit greener, displaying the pressure difference that induces lift.

4. POST-PROCESSING

REFERENCES

CFD Online. 2011. *SST k-omega Model*. www.cfd-online.com/Wiki/SST_k-omega_model.

Computery Things. 2024. *Virtual Wind Tunnel – SimScale Tutorial*. [YouTube Channel]. www.youtube.com/watch?v=WsPy_TJotv4.

Josephy et. al. Draft 2021. *No Evidence of Galactic Latitude Dependence of the Fast Radio Burst Sky Distribution*. arXiv. arxiv.org/abs/2106.04353. Used to influence global document formatting.

Massachusetts Institute of Technology. 2018. *Introduction to Aerodynamics (16.101x)* [Online Course]. edX. learning.edx.org/course/course-v1:MITx+16.101x_2+3T2018/home.

SimScale Documentation. 2025. *Field Calculations*. SimScale. www.simscale.com/docs/simulation-setup/result-control/field-calculations/.

UIUC Applied Aerodynamics Group. *UIUC Airfoil Coordinates Database*. University of Illinois at Urbana-Champaign Department of Aerospace Engineering. m-selig.ae.illinois.edu/ads/coord_database.html.

Van Buren. 2020. *Aerodynamics* [YouTube Channel]. www.youtube.com/watch?v=5LQxe4-BEcM&list=PLe2kb8k5FJpgLfhf18VphADpX9qvaby6k.

Wang et. al. 2014. *Guts of CFD: Near Wall Effects*. Data Marine Solutions Online. www.dmsonline.us/guts-of-cfd-near-wall-effects/.

Weller, Henry. *Notes on Computational Fluid Dynamics: General Principles*. CFD Direct. doc.cfd.direct/notes/cfd-general-principles/preconditioning-and-asymmetry.

Yenni, Aryan. *Aerodynamic Analysis of NACA Airfoils Using CFD and MATLAB*. Independent researcher. github.com/aryanyenni/aerodynamics/tree/main/Project1_NACA_Airfoil_CFD.