Incompressible Flow Over a 2D NACA 2412 Airfoil

Gerardo Sandoval

Michigan Technological University

ME 5210 Advanced Fluid Mechanics

Professor Dr. Cai

November 10, 2025

Abstract

This project focuses on incompressible flow (subsonic) over a 2D NACA 2412 airfoil using ANSYS Fluent. Geometry is generated from plotting data, and simulations at varying angles of attack generate lift and drag coefficients and flow contours. Results are compared with NASA data to validate CFD accuracy and demonstrate software proficiency. Sonic and supersonic flow simulations are also performed to visualize the shock interactions and turbulence flow contours.

1. Introduction

This project aims to simulate steady incompressible airflow over a NACA 2412 two-dimensional airfoil using Ansys Fluent. The goal is to analyze aerodynamic performance such as lift and drag coefficients across a range of angles of attack for 2-D Flow. The motivation behind this project is the fact that the NACA 2412 profile can be made easily or imported based on existing data online and there are existing Ansys Fluent YouTube video tutorials on how to do simulations on this specific airfoil. The expected outcomes of the CFD simulation are the drag and lift coefficients of the 2-D airfoil at different angles of attack (0, 8, and 16 degrees) and the comparison of the CFD data with NASA experimental data on the airfoil at Mach 0.13. Mach 0.13 is used since that's the air flow speed used in the NACA 2412 experimental data along with a Reynold's number of 3.1 x 10^6. Additional results are the velocity, pressure, and streamlines contours of the flowfield over the airfoil at Mach speeds of 0.13, 1.0, and 1.5. For Mach 1.0 and 1.5 air flows, the Fluent solver was set to "Density-Based" so that turbulent flow can be simulated accurately and 0 and 16° angle of attack were used.

2. Methodology

This CFD project utilized the online tutorial "NACA2412 Tutorial in Ansys Fluent (Student Version)" created by Thomas (2023) as a reference and the NACA report on airfoil data "Report No. 824 Summary of Airfoil Data" created by Abbot et. al (1945). All simulations and work presented in this paper are this paper author's own work. The book "Fluid Mechanics" by Kundu et. al. 6th edition (2015) was also used as a reference during this project.

2.1 Geometry and Airfoil Setup

The two-dimensional geometry of the NACA 2412 airfoil was made using coordinate data obtained from AirfoilTools.com. The site provides tabulated x-y coordinates useful for CFD modeling. A chord length of 1,000 millimeters, 1 meter, was used when downloading the airfoil CSV file x-y coordinates file. The following steps were taken to prepare and import the geometry into Ansys Workbench 2025:

- A new Fluent-based project was started in Ansys Workbench.

- The NACA 2412 airfoil coordinates were downloaded in CSV format and cleaned by adding two new leftmost columns, group and #points, and editing the x and y headers to only have #x and #y so that Fluent can read the column data as x and y coordinates. A column named #z was also added to the right of the column #y and the column was filled with zero values. Afterwards, the Excel file was saved as a Text file (Tab delimited format).

- The geometry was imported into Ansys Design Modeler, and the curved airfoil was turned into a solid surface, and sketches were used to create the fluid domain (arc in front of the airfoil and a rectangle fluid sketch around the airfoil).

Figures 1 and 2 illustrate the final geometry setup and includes the domain boundaries.

Figure 1 and Figure 2

*Geometry of the NACA 2412 airfoil in the 2D domain in Ansys Design Modeler*
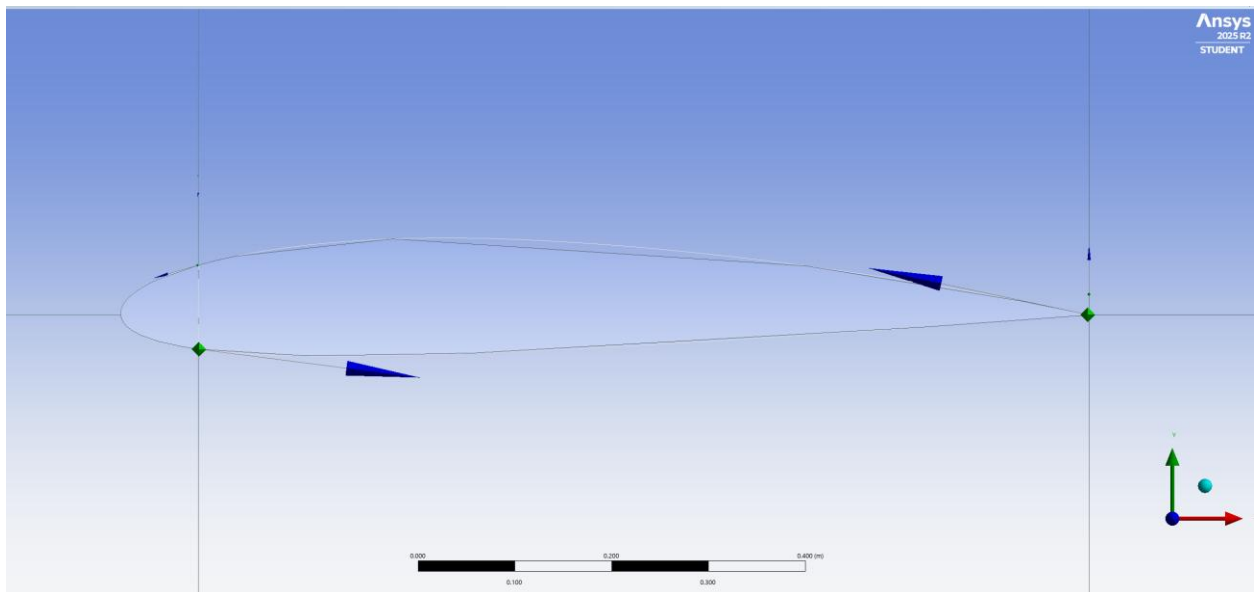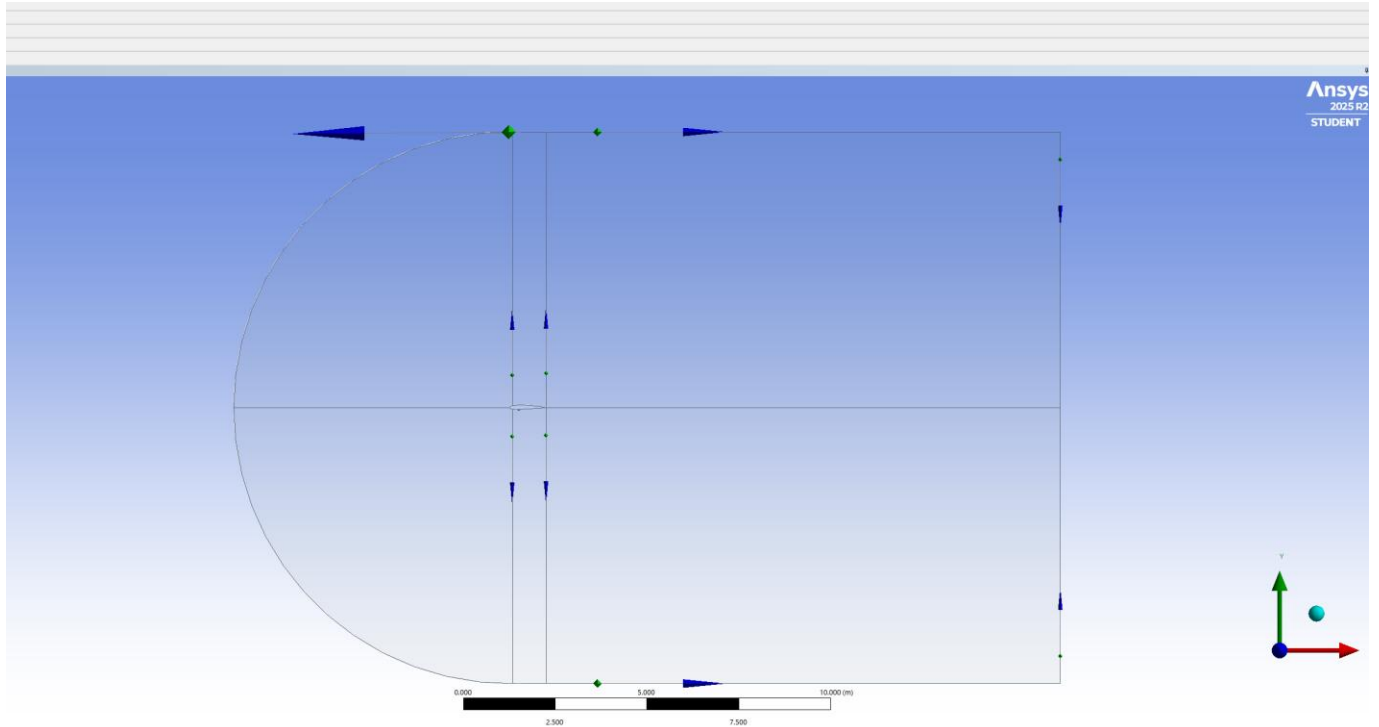
2.2 Mesh Generation

The mesh of 2412 airfoil was created by editing the mesh in Workbench -> Fluid Flow and adding four edge sizings and one face meshing. Number of divisions of 250, 150, 150, and 100 were applied to the four edge sizings and bias was added too that more cells would be displayed in the mesh. Figure 3 and Figure 4 show the structured mesh of the 2412 airfoil in Fluid Flow in the x-y plane.
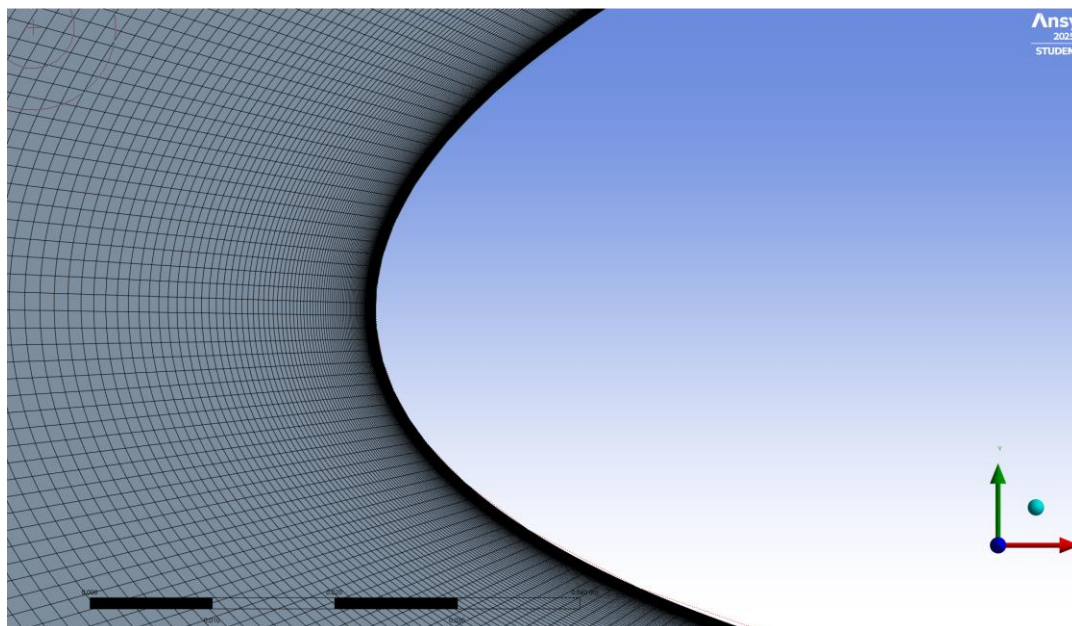
Figure 3 and Figure 4

Structured mesh of the computational domain.
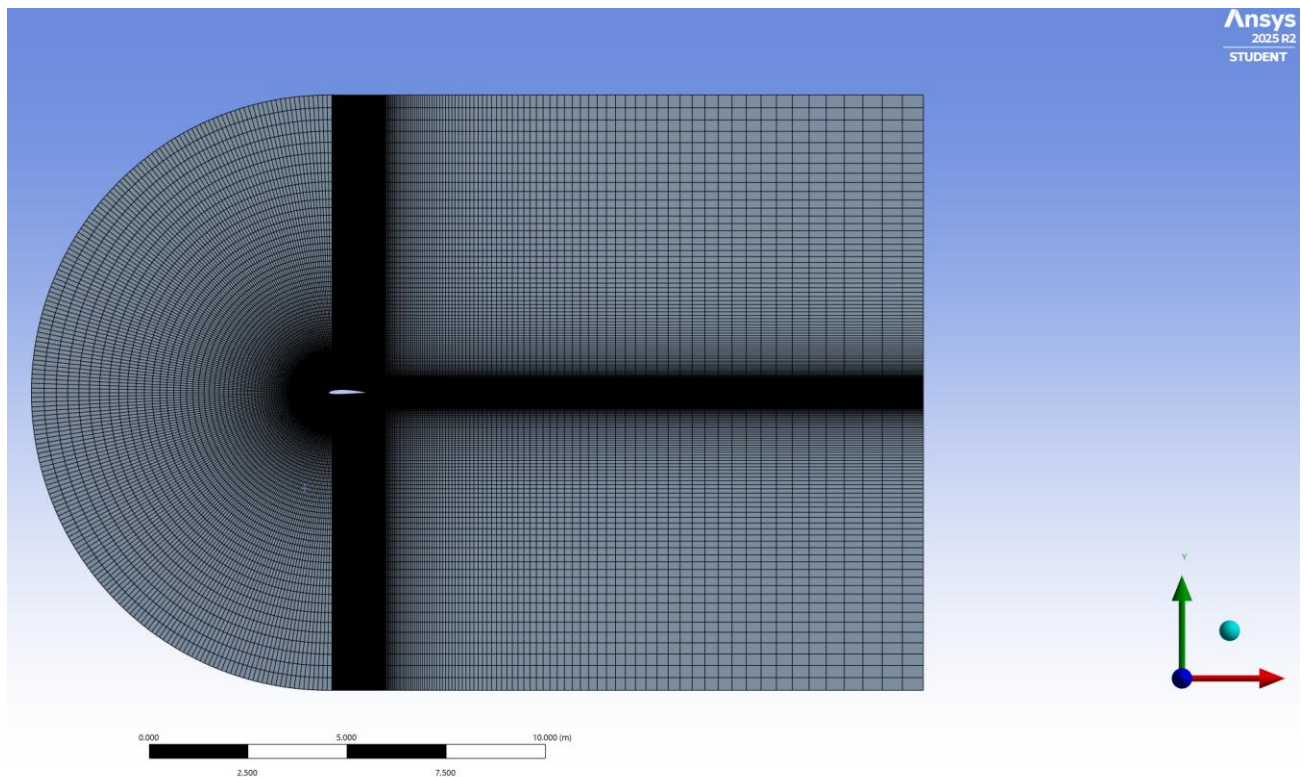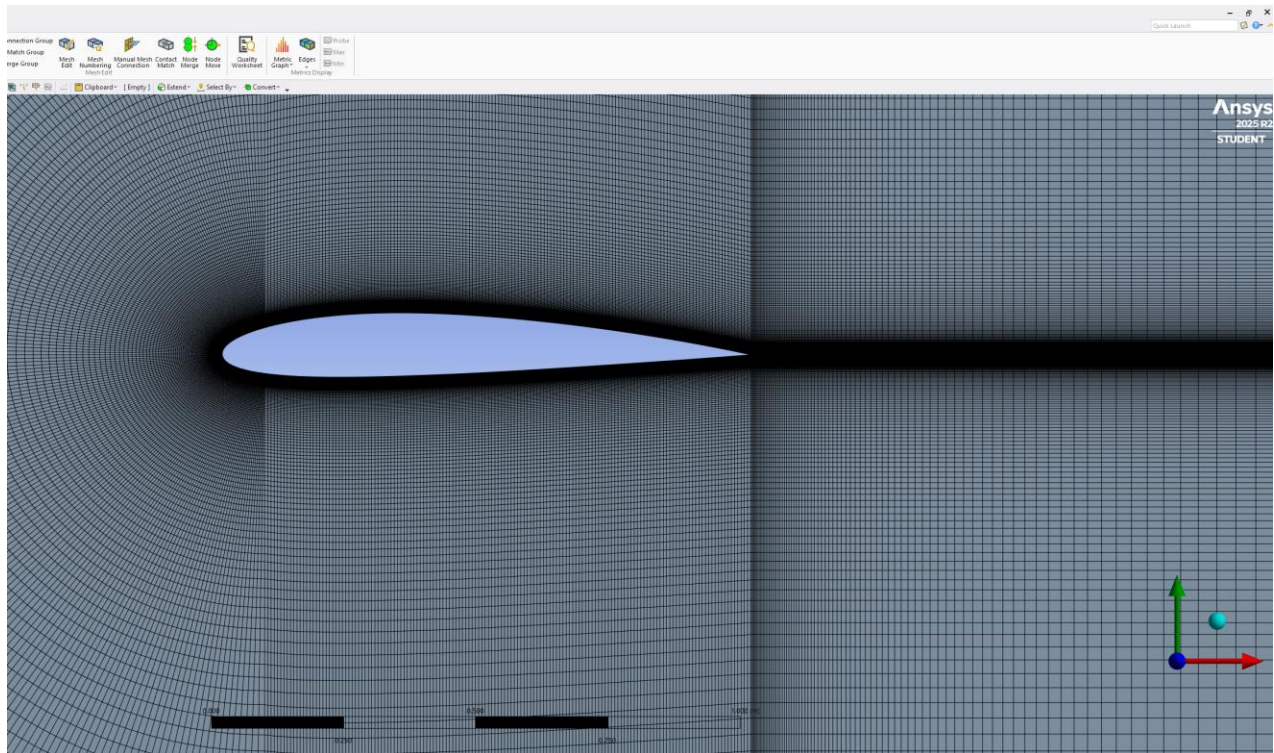
Figure 5

Refined mesh near the airfoil leading edge of the NACA 2412 airfoil.

2.3 Solver Settings

The solver settings used in Fluent were:

- Start

  - 2-D Flow

  - Four decimal points and four processors to use for the simulation.

- Fluent Setup

  - Viscous -> SST k-omega (default setting)

  - Multiphase, Energy, Ration (off) (default setting)

  - Materials

    - Fluid -> Air

      - Air density = 1.225 (default setting)

      - Viscosity set to 1.802 e-05

  - Boundary Conditions

    - Inlet -> inlet (velocity-inlet)

      - For Mach 0.13 and 0 degreees angle of attack

        - Edited -> Velocity Specification Method set to "Components" and X-velocity set to 45.6 m/s (Mach 0.13)

      - For Mach 0.13 and 8-degrees angle of attack

        - Updated X-velocity to 45.6 m/s x cosine (8 degrees) =

   - Updated Y-velocity to 45.6 m/s x sine (8 degrees) =

- Repeated for Mach 0.13 at 16-degrees angle of attack and for Mach 0.3, Mach 1.0, and Mach 1.5 at 0-degrees angle of attack.
    - Reference Values
        - Set to Compute from "inlet".
- Fluent Solution
    - Residual Monitors -> Changed absolute criteria to 1e-06 for all values.
    - Report definitions -> Created two new Force reports (drag and lift coefficients).
    - Initialization – Clicked Initialize.
    - Run Calculation -> Set the Number of Iterations to 1,000 and then clicked Calculate.
- Results
    - Added Graphics contours and streamlines by editing the Contours and adding pressure and velocity contours and one pathlines contour (created a new Surface by adding a Line/Rake in the Results tab under Surface and named it rake).
- For Mach 1.0 and 1.5
    - Applied density-based simulation and turbulent contour to simulate turbulence flow.
    - Increased the Number of Iterations to 3,000 since turbulent flow will take more calculations to simulate.

2.4 Boundary Conditions

The boundary conditions were set during the mesh creation. The inlet was set as the fluid edges in front of the airfoil and surrounding it and the outlet was set as the vertical line trailing the airfoil. Figure 6 shows the mesh boundary conditions in Fluent, the blue arrows are the inlet flows, and the red arrows represent the outlet flow.

Figure 6.

Fluent NACA 2412 Mesh with inlet and outlet shown with blue and red arrows.
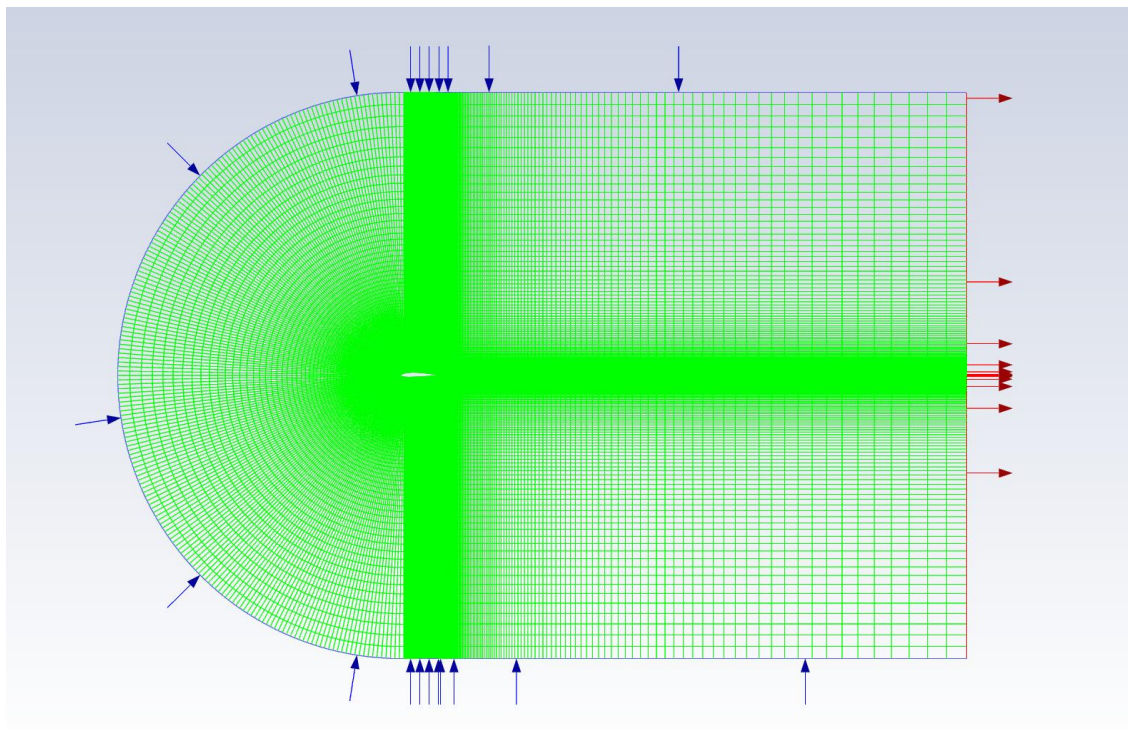
2.5 Experimental Data

Airfoil experimental data from NASA was obtained by using the online document *Report No. 824 Summary of Airfoil Data* that was published by NACA in 1945. The Fluent simulated force coefficient data on the 2412 airfoil was compared with the NASA data at Mach 0.13 (45.6 m/s). Figure 7 shows the experimental lift and drag coefficient data for the NACA 2412 airfoil.

Figure 7.

NACA 2412 Airfoil Lift and Drag Coefficient Data.

Aerodynamic characteristics of the NACA 2412 airfoil section, 24-inch chord.
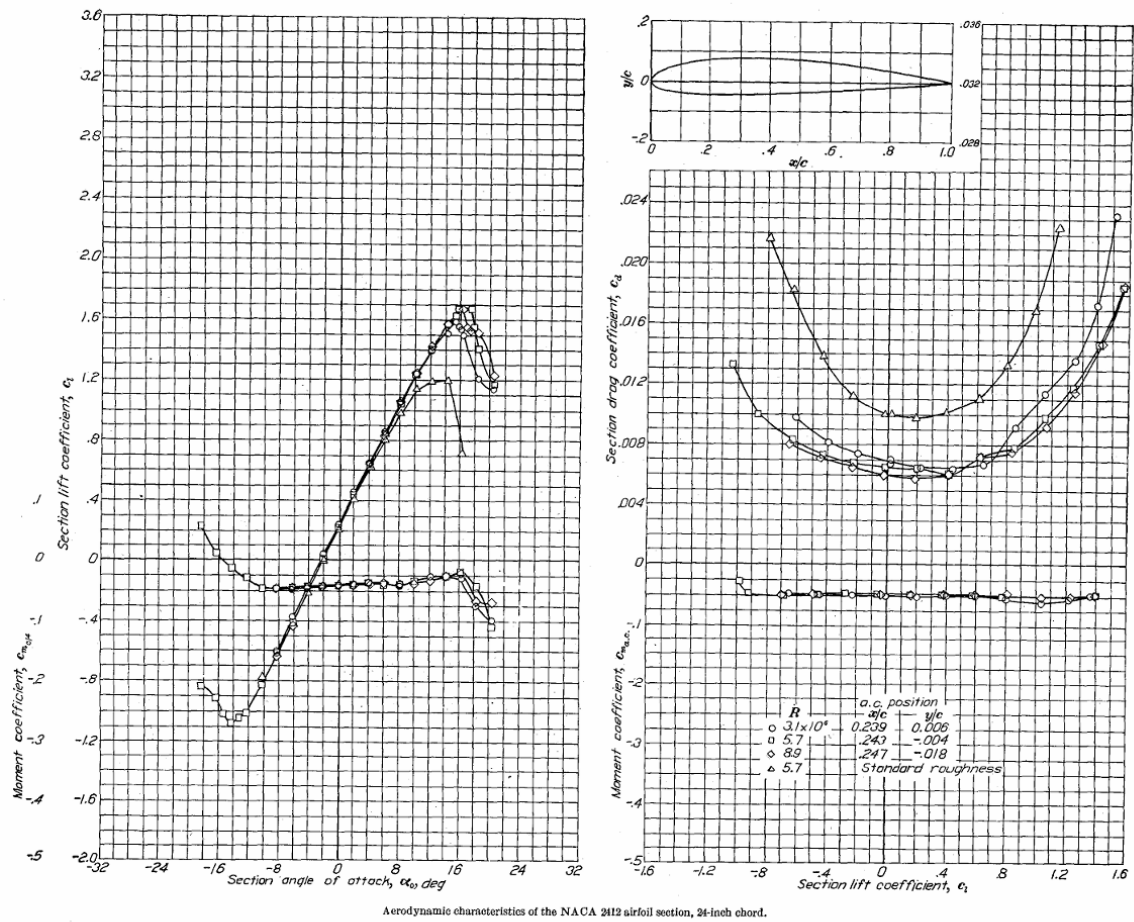
Note: Used the circle symbol guide (right side graph) in the graphs since it represents

Reynolds Number of 3.1×10^6 values.

3. Results & Discussion

3.1 Mach 0.13 (Validation Case)

To validate the CFD setup, simulations were performed at Mach 0.13 (45.6 m/s) with the

Reynolds number Re = $3.1 \ x \ 10^6$, matching NASA's experimental conditions for the NACA

2412 airfoil. The Fluent results are compared below and the lift and drag coefficients at angles of

attack of 0, 8, and 16-degrees are shown in Table 1.

At 0-degrees angle of attack, the Fluent lift coefficient closely matched the experimental

value (Fluent Cl = 0.215 vs NACA Cl = 0.25) with a 15.1% percent difference and the drag

coefficient in Fluent was much higher than the experimental value, Fluent Cd = 0.00927 vs

NACA Cd = 0.0065, resulting in a 35.1% percent difference. A larger percentage difference in

drag coefficient is expected since drag is more sensitive to mesh resolution, and the Fluent

simulation used many assumptions that may not match experimental conditions.

At 8 and 16-degrees angle of attack, the Fluent lift coefficients were close to the NACA

experimental values at 0.0095% and 8.67%, respectively. The Fluent drag coefficients deviated

by a margin of 21.0% and 59.9%. The Fluent simulations had a close accuracy of the lift

coefficients across all the angles of attack, and the aerodynamic setup was validated as being

relatively accurate even though the drag coefficients deviated significantly. Higher mesh

resolutions would have improved the drag coefficient data in Fluent. In general, the Fluent

simulations were accurate and set the foundation for compressible flow to be studied afterwards.

Pressure and velocity contours and streamlines of the flow are shown below. Air flow velocity is

45.6 m/s and decreases as it collides with the leading edge of the airfoil and increases as it

surrounds the airfoil (on top and bottom of the airfoil) leading to pressure differences that

generate lift or upward force.

Table 1. Fluent CFD vs NACA 2412 Experimental Data. Note: Reynolds Number is 3.1×10^6

and velocity is 45.6 m/s.

Angle of Attack	Fluent Lift Coefficient	Fluent Drag Coefficient	NACA Lift Coefficient	NACA Drag Coefficient	Percent Difference
0	2.15 e-01	9.27 e-03	2.5 e-01	6.5 e-03	15.1% and 35.1%
8	1.049	1.42 e-02	1.05	1.15 e-02	0.095% and 21.0%
16	1.636	4.265 e-02	1.50	2.30 e-02	8.67% and 59.9%

Figure 8.

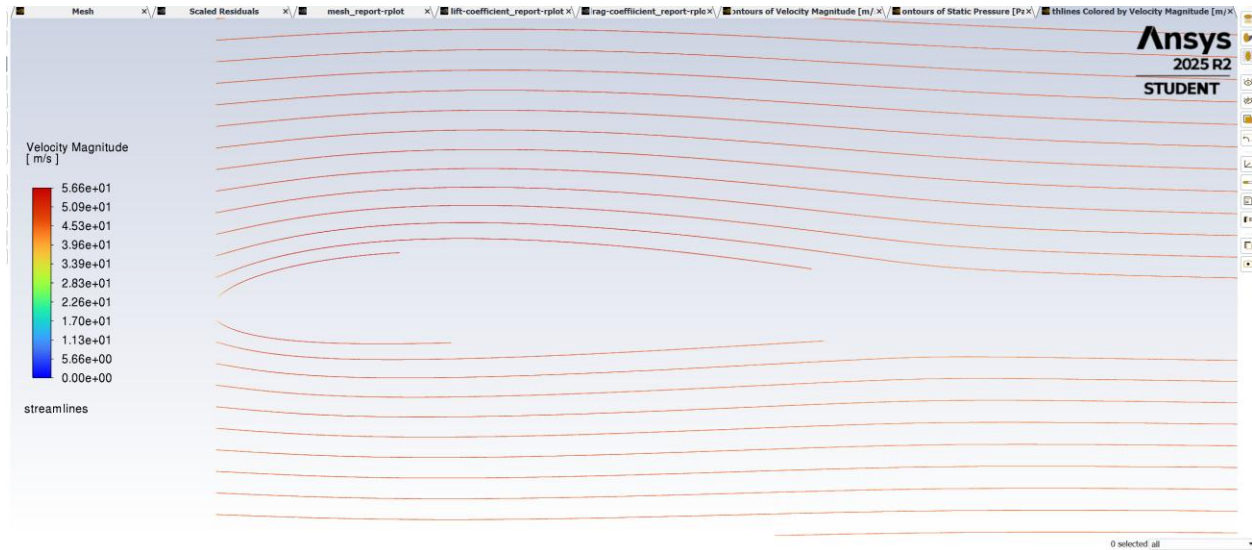Pressure Contour at Mach 0.13 (45.6 meters/second) at 0° Angle of Attack.
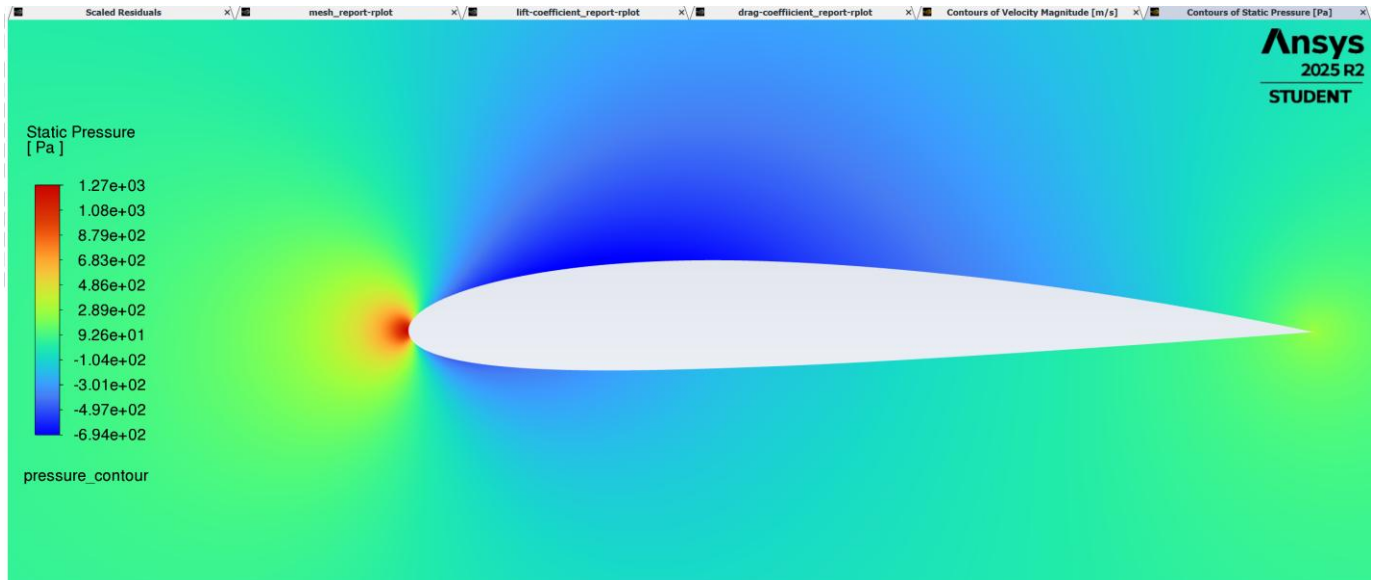
Figure 9.

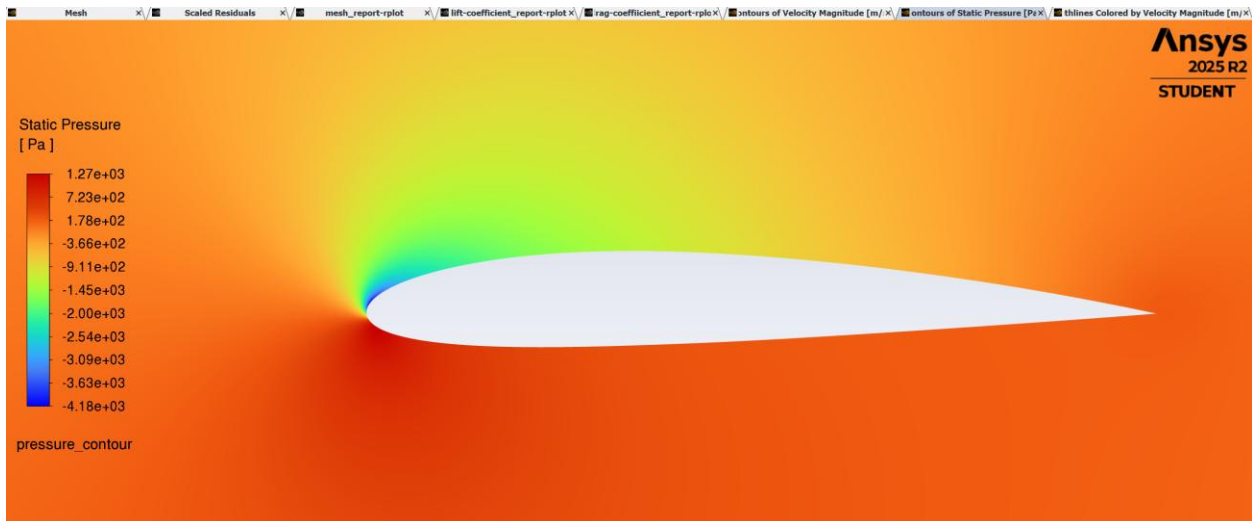Velocity Contour at Mach 0.13 (45.6 meters/second) at 0° Angle of Attack.
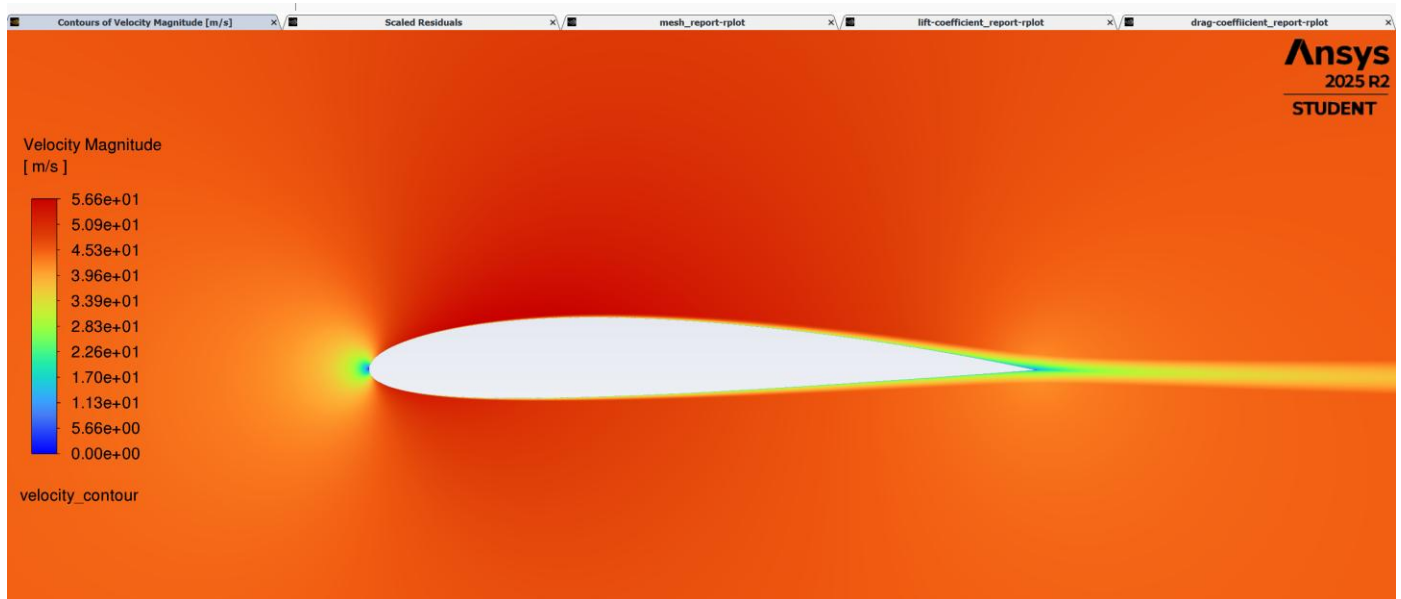
Figures 10.
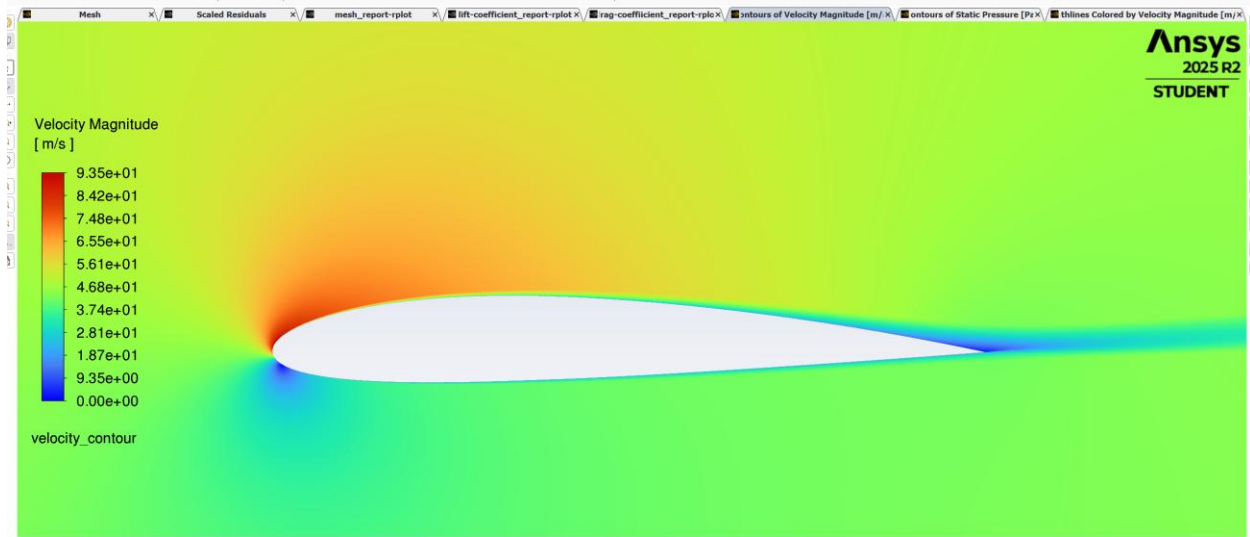
Streamlines at Mach 0.13 at 0° Angle of Attack.

Figure 11.

Pressure Contour at Mach 0.13 at 8° Angle of Attack.

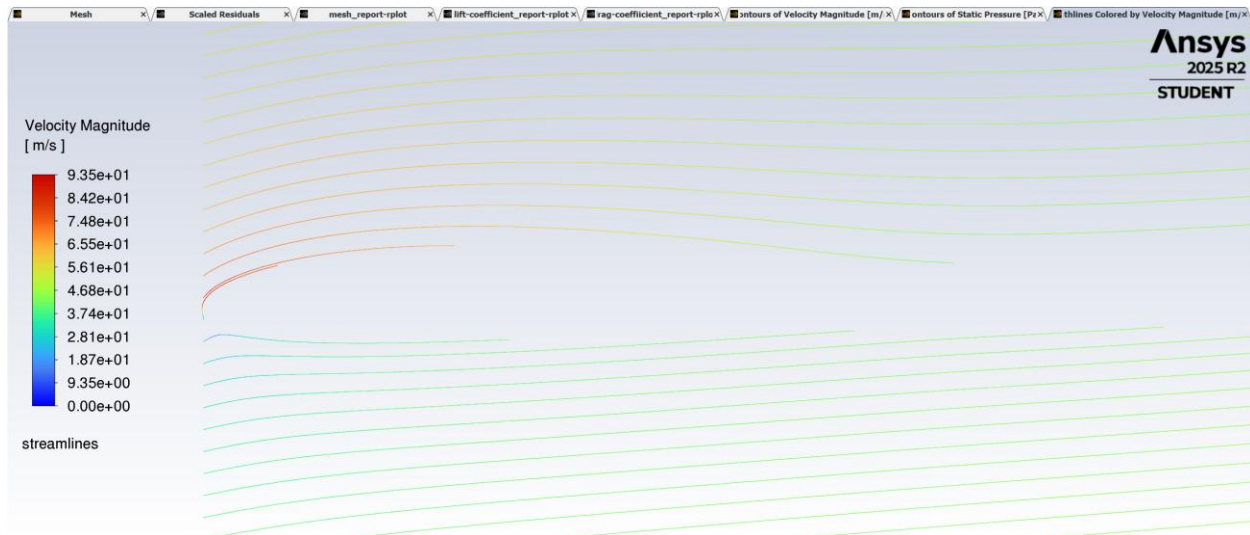**Figure 12.**

Velocity Contour at Mach 0.13 at 8° Angle of Attack.

Figure 13.

Streamlines at Mach 0.13 at 8° Angle of Attack.

3.2 Compressible Flow at 0 degrees Angle of Attack

At Mach 1.0 (340.29 m/s) and 0° angle of attack, there is limited turbulence at the leading edge of the 2412 airfoil and the lift and drag coefficients are 1.2643 x 10 and 4.111, respectively. The zero-degree angle of attack results in no major pressure differences compared to subsonic flows at the same angle of attack. At Mach 1.5 (497 m/s) and a 16° angle of attack, turbulence and pressure differences are very visible. The shock interactions could be visible as sudden changes of pressures.

| Mach Number	Reynolds Number (calculated)	Flow Regime	Lift Coefficient (CL)	Drag Coefficient (Cd)	Notes
1.0	2.314 x 10^7	Turbulent	1.2643 e+01	4.111	The turbulence contour shows limited turbulence contour near the leading edge, especially at the top surface. The zero degrees angle of attack causes no shock interaction to be seen.
1.5	3.471 x 10^7	Turbulent	1.748 e+02	5.83	Turbulence is very visible, especially behind the trailing edge of the airfoil.

3.2.1 Mach 1.0 at 0° Angle of Attack.

Figure 14.

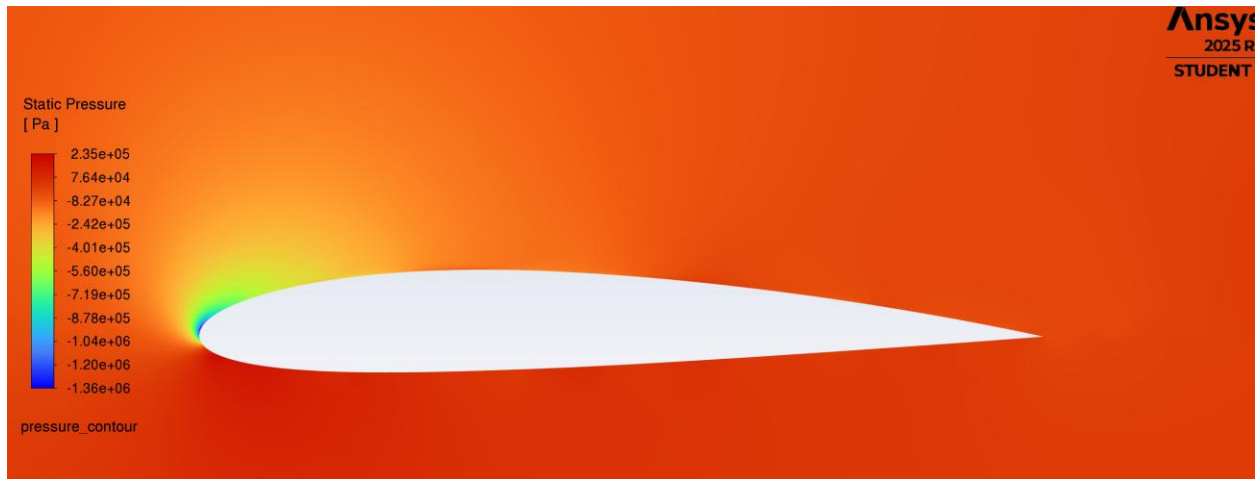Pressure contour at Mach 1.0 (340.29 m/s) at 0° Angle of Attack.
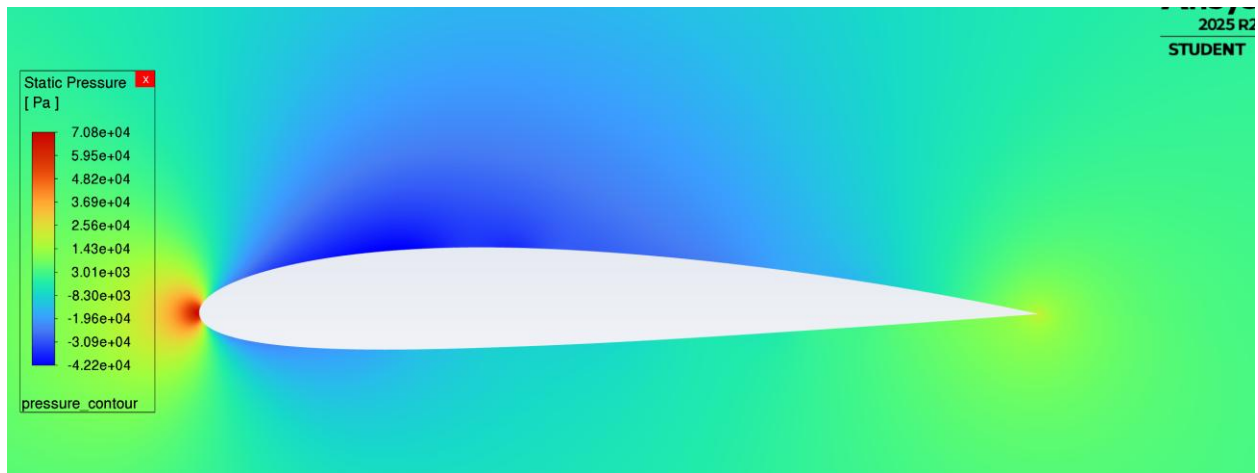
Figure 15.

Turbulence Contour at Mach 1.0 (340.29 m/s) at 0° Angle of Attack.
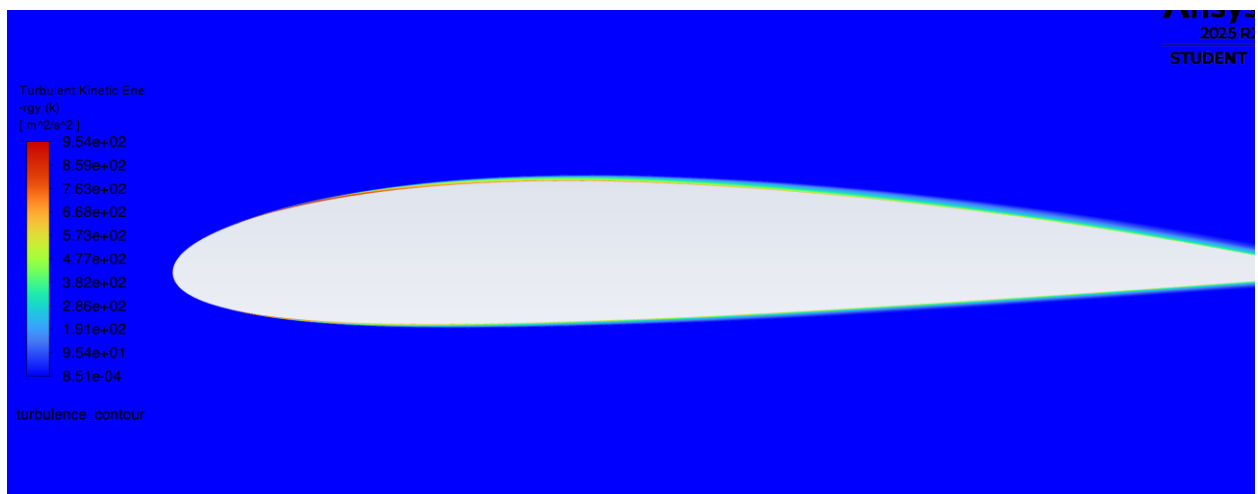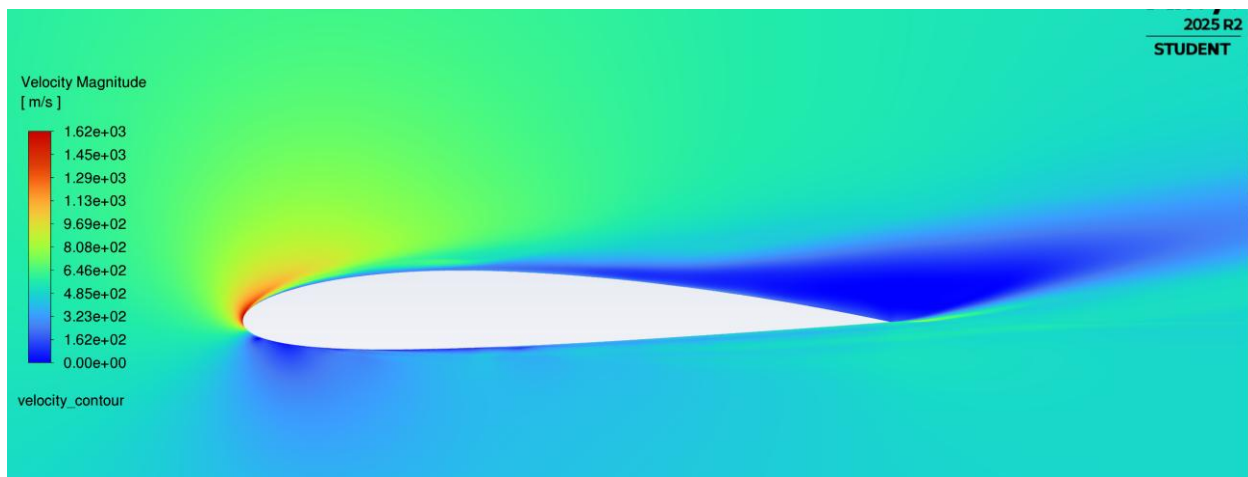
3.2.2 Mach 1.5 (16 degrees angle of attack)

Figure 16.

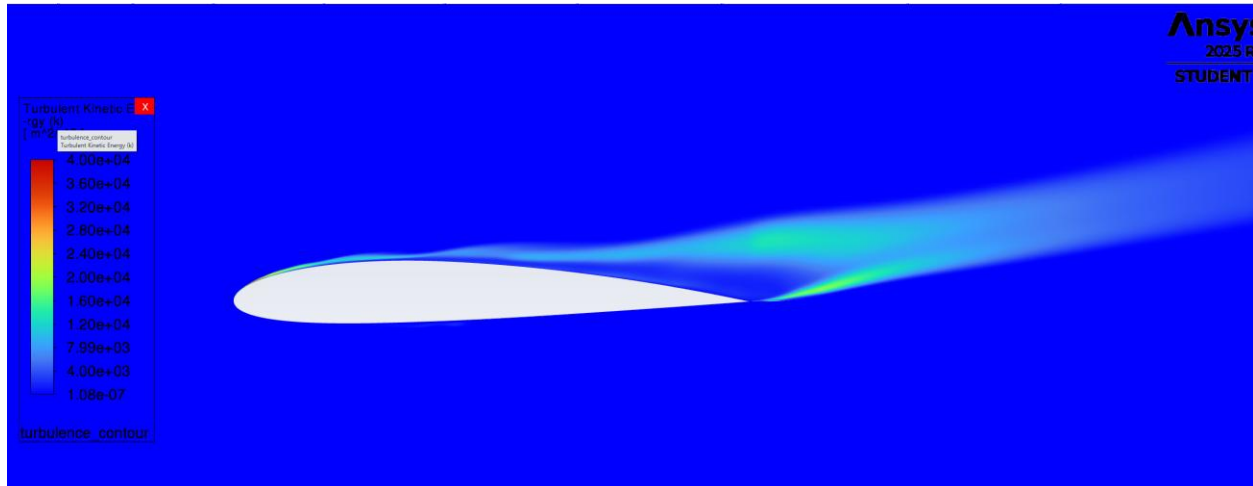Pressure contour at Mach 1.5 (497 m/s) at 16° Angle of Attack.

Figure 17.

Velocity contour at Mach 1.5 (497 m/s) at 16° Angle of Attack.

Figure 18.

Turbulence contour at Mach 1.5 (497 m/s) at 16° Angle of Attack.

4. Conclusion

This computational fluid dynamics (CFD) project successfully analyzed the aerodynamic performance of the NACA 2412 airfoil in a wide range of flow conditions using Ansys Fluent. Validation at Mach 0.13 proved that the simulated force coefficients were close to NASA experimental data especially when predicting the lift coefficients. Simulated drag coefficients were not as close to the NASA experimental values due to issues with mesh resolution and limitations with using many assumptions in steady flow fluid simulations. Compressible flow at Mach 1.0 and Mach 1.5 using density-based solver settings resulted in turbulent flow visualization and signs of shock interactions at Mach 1.5 with a 16-degree angle of attack as visible turbulent contour was generated and there were sudden changes in pressure and velocity in the leading edge. The project demonstrated basic capability of using Fluent to model subsonic and compressible turbulent flows over a simple airfoil. The biggest challenges during the project were generating the mesh without any odd swirls (caused by not reversing the bias during meshing) and choosing the right settings for turbulent flow and generating the streamlines.

Future work for this CFD project could explore unsteady simulations and simulations of the 2412 and more complex airfoils in three-dimensions to explore complex flow behavior that are common in fluid system design such as wing design and fluid research problems.

5. References

Abbott, I.H., Doenhoff, A. E., & Stivers, L. S. (1945). Report No. 824 Summary of Airfoil Data National Advisor of Committee For Aeronautics (NACA).

Cillian Thomas. NACA2412 Tutorial in ANSYS Fluent (Student Version). (2023). YouTube. https://www.youtube.com/watch?v=3i9Ryq-m1HA.

Kundu, P. K., Cohen, I. M., & Dowling, D. R. (2015). Fluid Mechanics (6th edition). Academic Press.