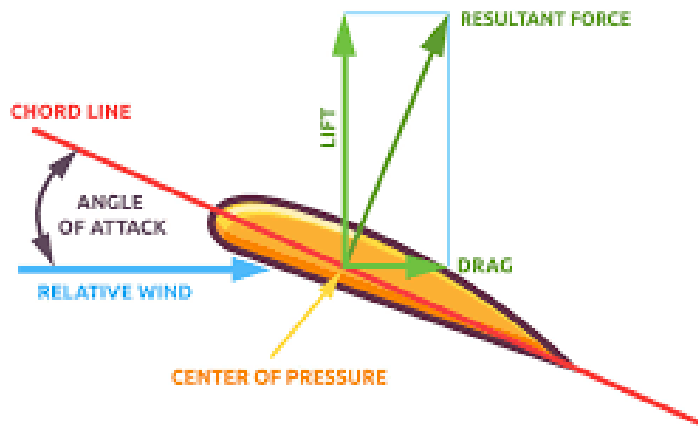


NACA 2412 WING AIRFOIL ANALYSIS

INTRODUCTION



An **airfoil** is a streamlined shape designed to generate lift when air flows over its surface. It is commonly used in the wings of aircraft, rotor blades, turbine blades, and other aerodynamic structures. The design of an airfoil plays a crucial role in determining the aerodynamic performance of a body, particularly how efficiently it can generate lift and minimize drag.

An airfoil's geometry is characterized by parameters such as:

- **Chord line:** A straight line from the leading edge to the trailing edge.
- **Camber:** The curvature of the airfoil.
- **Thickness:** The maximum distance between the upper and lower surfaces.
- **Angle of attack:** The angle between the chord line and the oncoming airflow.

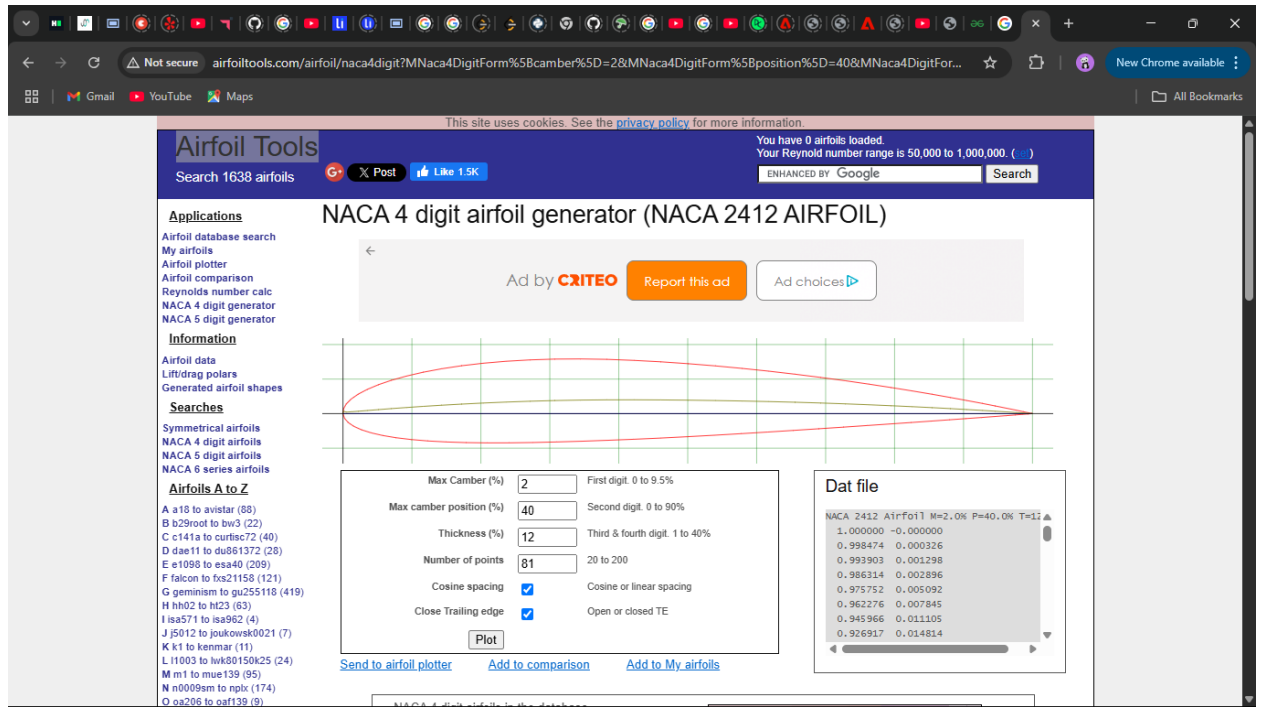
Airfoils are the cross-sectional shapes of wings, blades, or fins designed to generate lift as air flows over them. The **National Advisory Committee for Aeronautics (NACA)** developed a series of standardized airfoil shapes known as **NACA airfoils**. These airfoils are widely used in aerodynamics and aerospace engineering due to their well-documented performance characteristics.

The NACA airfoil series are mathematically defined and designated by numbers. For example:

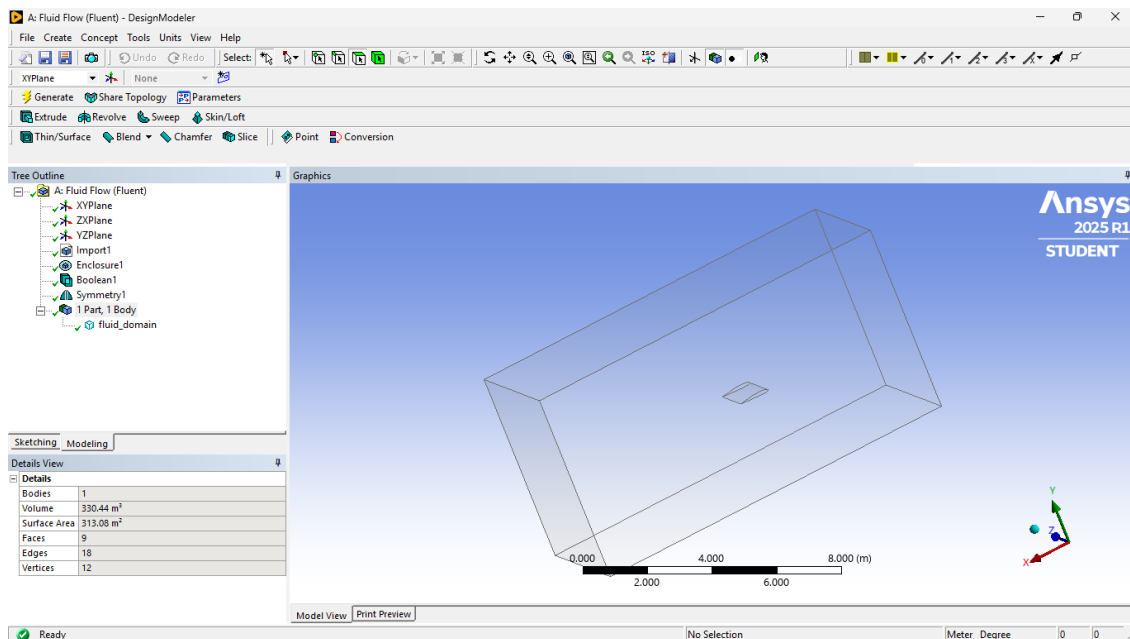
- **NACA 4-digit series** (e.g., NACA 2412) describes the camber, position of camber, and thickness.
- **NACA 5-digit and 6-digit series** offer more advanced control over lift characteristics and laminar flow.
- These airfoils are essential in the design and simulation of aircraft wings, UAVs, wind turbine blades, and other aerodynamic surfaces. By analyzing a NACA airfoil in software like ANSYS, engineers can study parameters such as **lift**, **drag**, **pressure distribution**, and **flow separation**, which are critical for efficient aerodynamic design.

a. Preprocessing

1. **Model Creation:** NACA 2412 geometry was created using data points from the website.



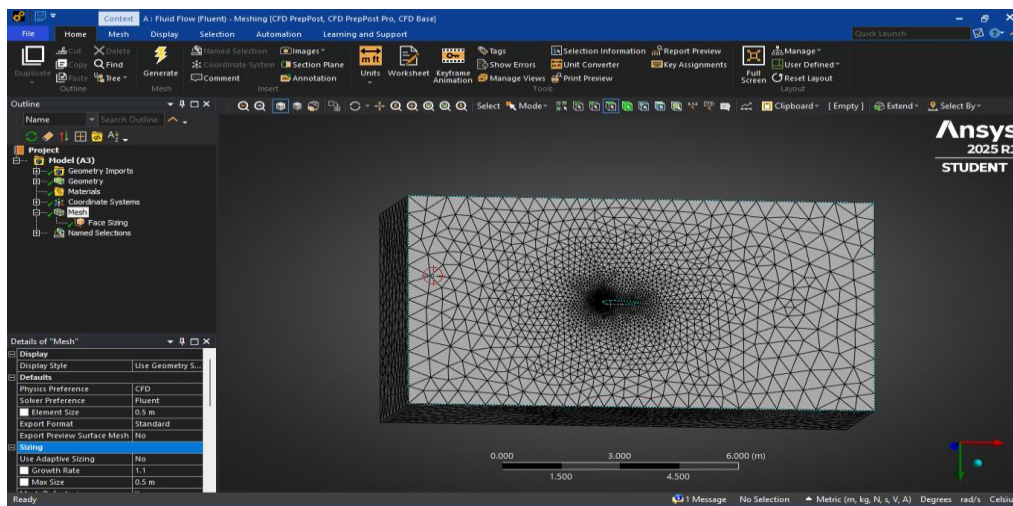
2. **Design Modelling:** Fluid domain was created in design modeler .



3. **Meshing:** The geometry was discretized into finite elements using custom mesh settings to ensure solution accuracy.

Key Parameters in Meshing

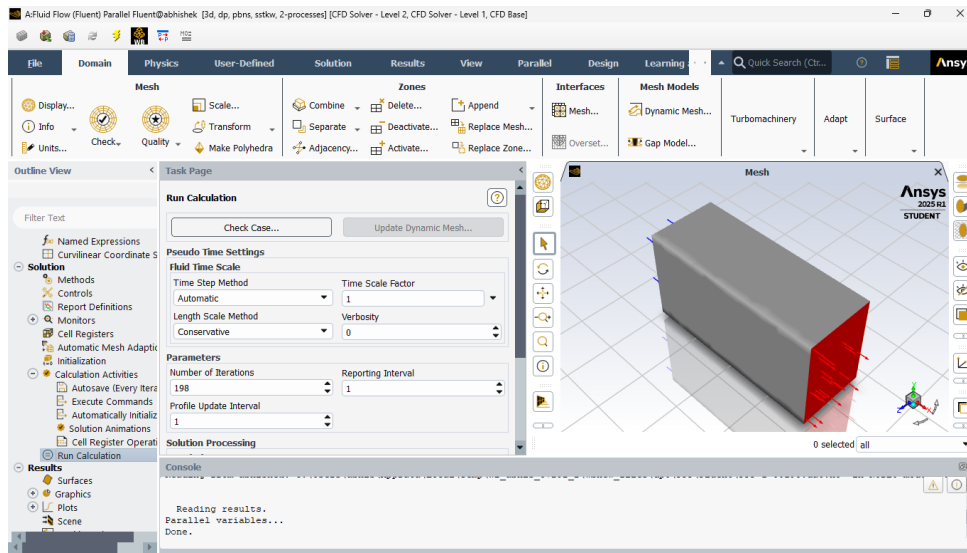
- 1) Element Size : 0.5 m
- 2) Capture Proximity : Yes (To distinguish very near surfaces)
- 3) Growth Rate : 1.1 (Rate of increase of mesh size at mesh boundaries)
- 4) Named Selections : walls , symmetry ,wing, inlet , outlet



4. **Calculation :** The Fluid domain was solved in ansys fluent using CFD .

Key Settings:

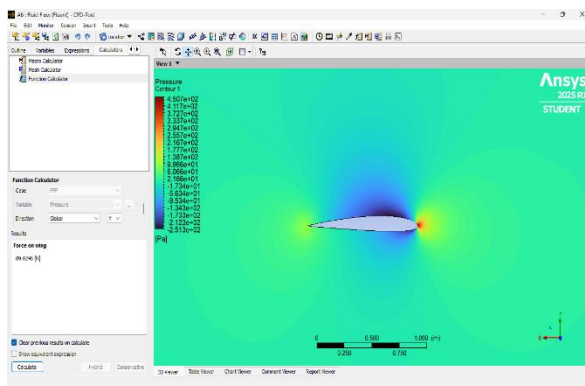
- *Fluid:* air
- *Boundary conditions:* inlet velocity = 30 m/s
- *Method :* default(coupled) (This method solves pressure and velocity equations simultaneously.)
- *Initialization :* inlet($V_x = 30$ m/s)



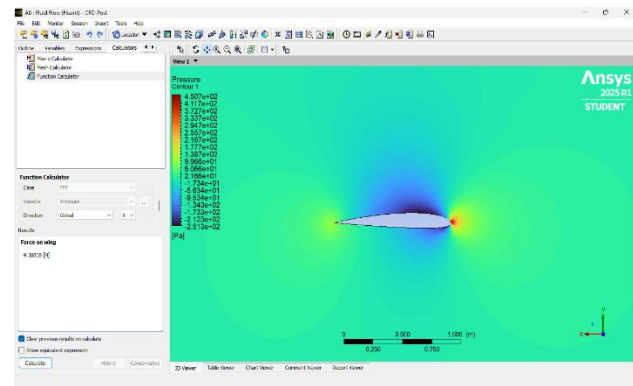
5. **Force Calculation** : Lift force and drag force was calculated using function calculator in the result option.

Calculated Drag Force = 9.58 N

Calculated Lift Force = 89.02 N

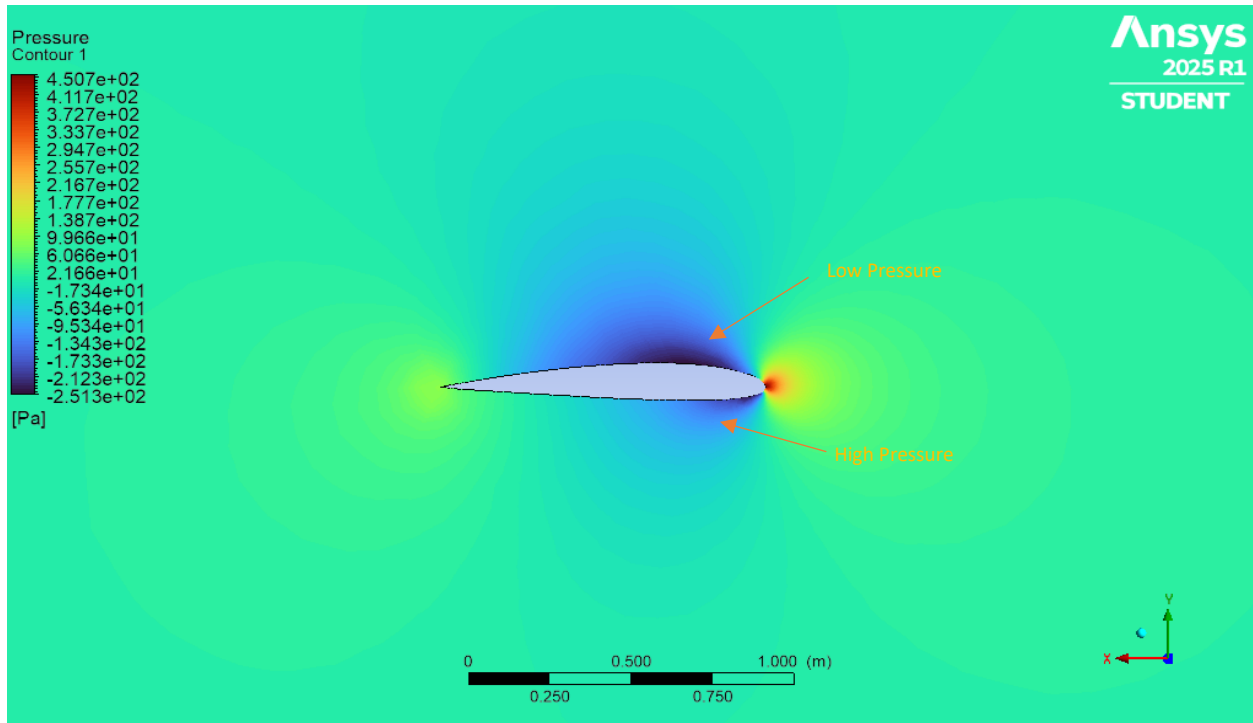


LIFT FORCE



DRAG FORCE

INSIGHTS



1. Pressure Distribution

- The pressure was found to be **lower on the upper surface** of the airfoil and **higher on the lower surface**, creating a net upward force (lift).

2) Lift and Drag Performance

- The airfoil generated a **lift force of 89.02 N** and a **drag force of 9.58 N**, resulting in a **Lift-to-Drag Ratio (L/D) of approximately 9.29**.