

ME 207: Fluid Dynamics

CFD Simulation of NACA0012 Airfoil under Laminar Flow

Shree Hiwanj, 23110305

(19.04.2025)

Contents

1	Introduction	2
2	Tabulated Results	2
3	Contours	2
3.1	AOA = 0	2
3.2	AOA = 5	5
4	Flow Characteristics	7
4.1	Boundary Layer	7
4.1.1	0 Degree AOA	7
4.1.2	5 Degree AOA	7
4.2	Flow Separation	7
4.2.1	0 Degree AOA	8
4.2.2	5 Degree AOA	8
5	Observations	9
5.1	Observations at $\alpha = 0^\circ$ and 5°	9
5.2	Comparison with Literature and Error Sources	9
6	Geometry	9
7	Meshing	10
8	FLUENT Setup	11
8.1	Solution Method	16
9	References	16

1 Introduction

The CFD Analysis was done using the ANSYS Fluent Solver. The geometry used for the Airfoil was NACA 0012H-SA. The Geometry, Meshing and Fluent Setup settings have been described ahead in the report. The Setup and Conditions are as follows:

- The Air flow is said to be Laminar. Thus, Laminar model has been chosen in FLUENT setup.
- The Temperature of Air has been said to be 20 C. Thus, the properties of Air have been modified in the Materials panel of FLUENT to match the Dynamic Viscosity and Density of Air at 20 C.
- Since the flow temperature is kept at 20 C, the reference value of temperature at Inlet has been set at 293.15 K.
- The Boundary Condition for Velocity Inlet is set at 25m/s velocity. The components are changed depending on the angle of attack.
- The Boundary Condition for Pressure Outlet is set at 0Pa Gauge Pressure to simulate the effects of open atmosphere.
- The Airfoil is given the feature of Wall to simulate an obstruction with the No-Slip Assumption being satisfied.
- Even though attempts were made to solve the simulation using various Mesh, and Solution Methods, but the results mentioned in the document are for SIMPLE Solution Method in FLUENT with number of iterations as 1000. Considered Methods: COUPLED, SIMPLEC.

2 Tabulated Results

AOA	Parameters	ANSYS CFD Results	Literature	Error %
0	C_d	0.0061760666	0.00809	23.61%
	C_l	5.1479467e-7	0	N/A
5	C_d	0.0093571923	0.00852	9.82%
	C_l	0.51495697	0.536	3.91%

Table 1: Results

The sources for the literature value have been mentioned in the References

3 Contours

3.1 AOA = 0

Velocity Contour

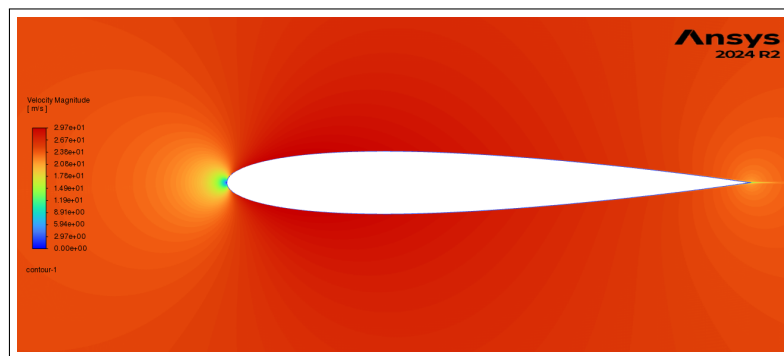


Figure 1: Velocity Contour

Pressure Contour

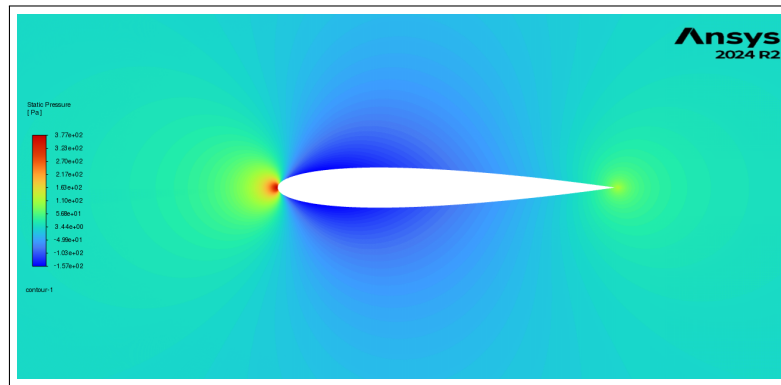


Figure 2: Static Pressure Contour

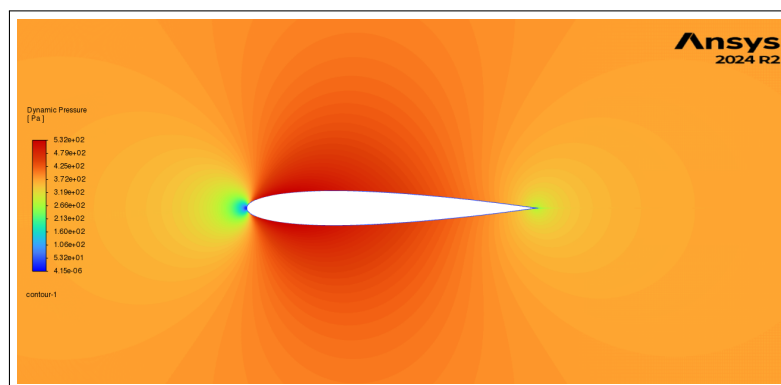


Figure 3: Dynamic Pressure Contour

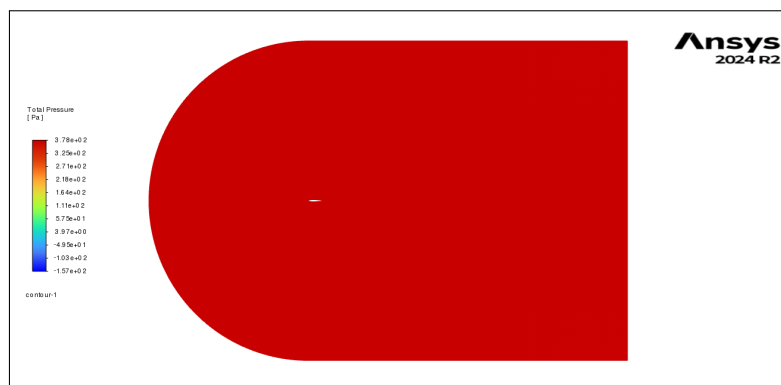


Figure 4: Total Pressure Contour

Velocity Streamline

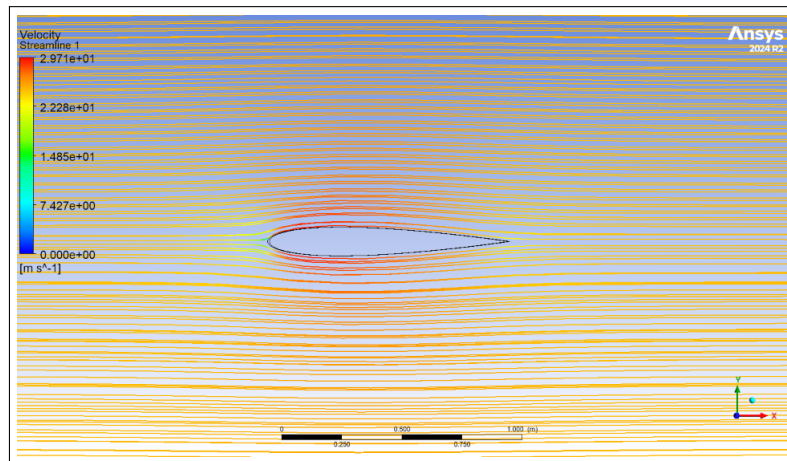


Figure 5: Velocity Streamline

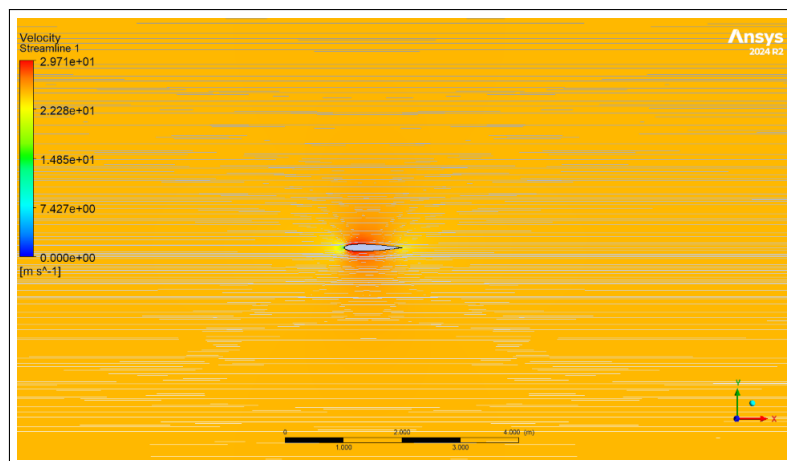


Figure 6: Velocity Streamline

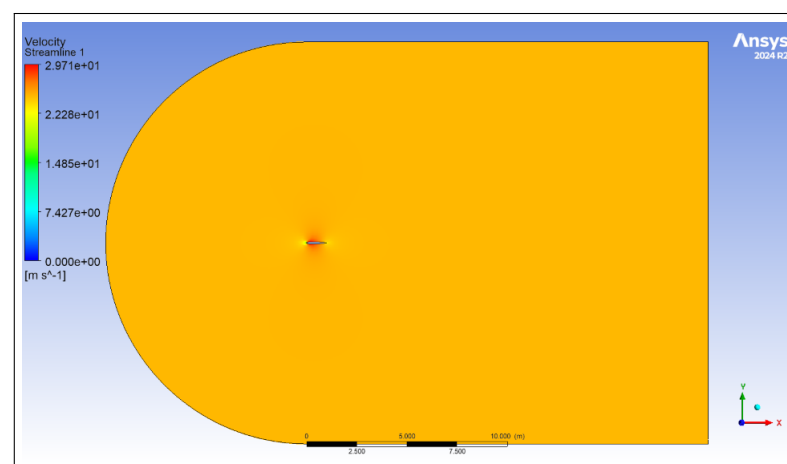


Figure 7: Velocity Streamline

3.2 AOA = 5

Velocity Contour

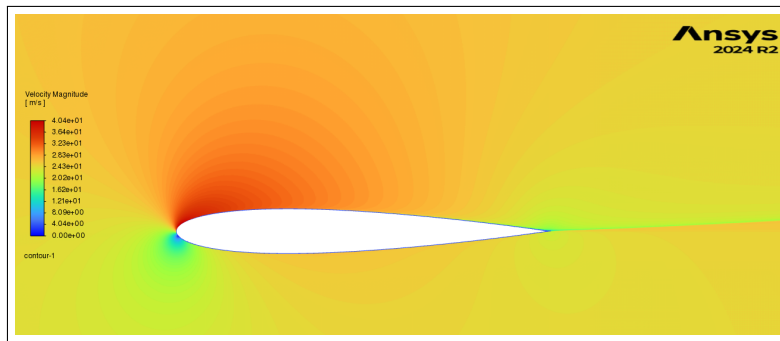


Figure 8: Velocity Contour

Pressure Contour

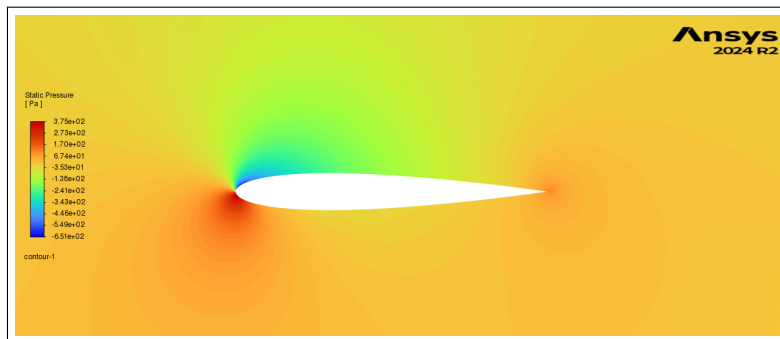


Figure 9: Static Pressure Contour

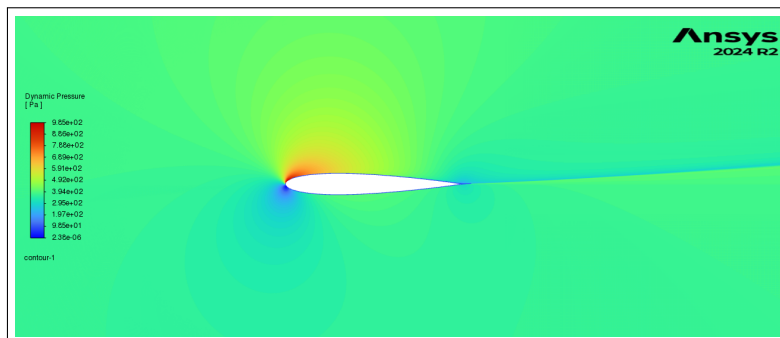


Figure 10: Dynamic Pressure Contour

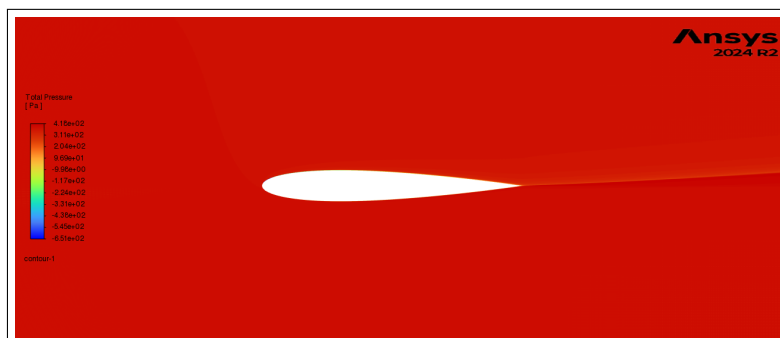


Figure 11: Total Pressure Contour

Velocity Streamline

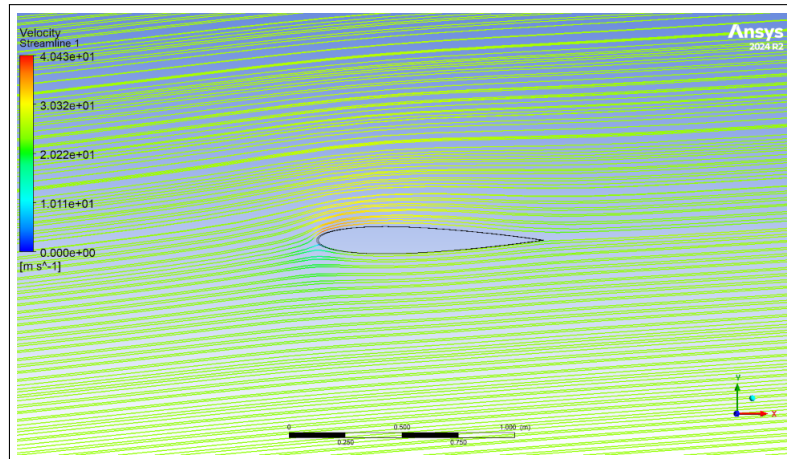


Figure 12: Velocity Streamline

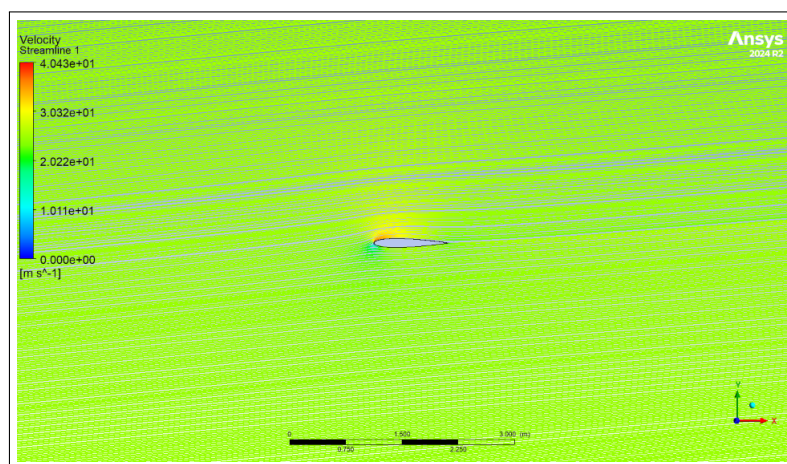


Figure 13: Velocity Streamline

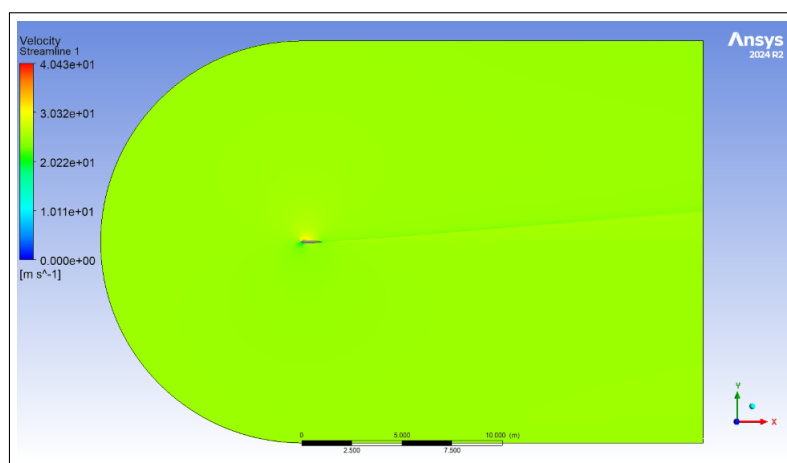


Figure 14: Velocity Streamline

4 Flow Characteristics

4.1 Boundary Layer

In the magnified Velocity Magnitude contour plots near the tail of the airfoil, a velocity gradient can be visualised near the outer boundary of the airfoil. This gradient arises due to No Slip condition at the Airfoil wall and shear forces acting over the subsequent upper layers of airflow. This region of gradience where the velocity is gradually increasing is the Boundary Layer

4.1.1 0 Degree AOA

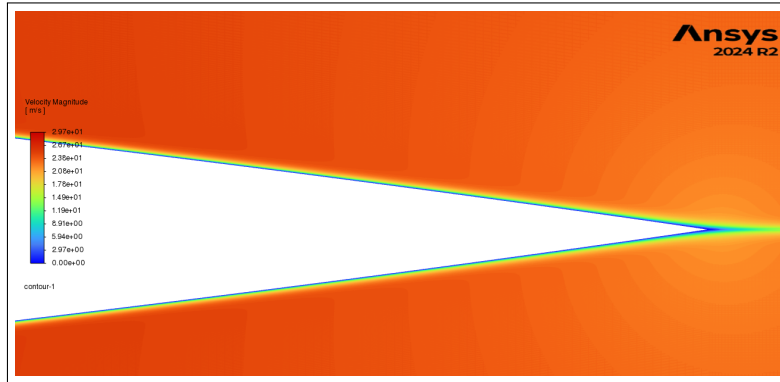


Figure 15: Velocity Contour depicting Boundary Layer

4.1.2 5 Degree AOA

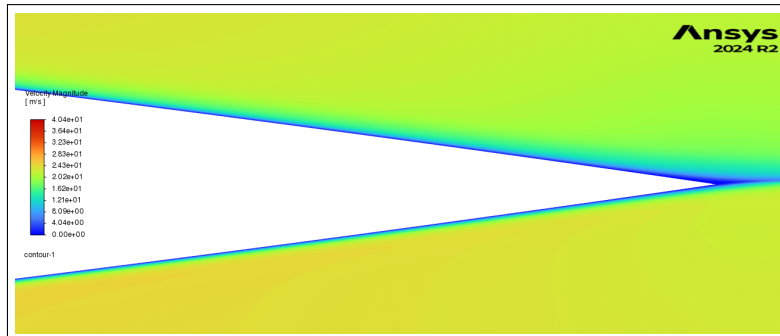


Figure 16: Velocity Contour depicting Boundary Layer

4.2 Flow Separation

The flow separation can be visualised through the X/Y Velocity and Pressure Contours. We can see in the contours that as the Airflow comes in contact with the nose of the Airfoil, the X Velocity becomes negative, depicting backflow of the air at the nose of the airfoil. This can be seen in Figure 17 and 19.

The flow separation can also be visualised through the Static Pressure contour where the Pressure instead of Dropping, increases in the direction of the flow as can be seen in Figure 18 and 20

4.2.1 0 Degree AOA

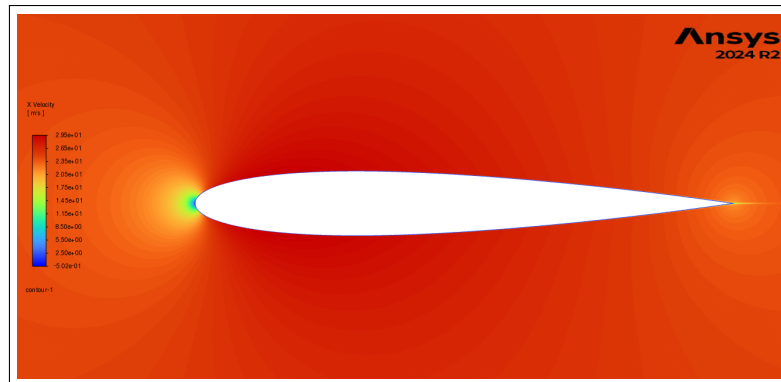


Figure 17: X Velocity Contour depicting Flow Separation

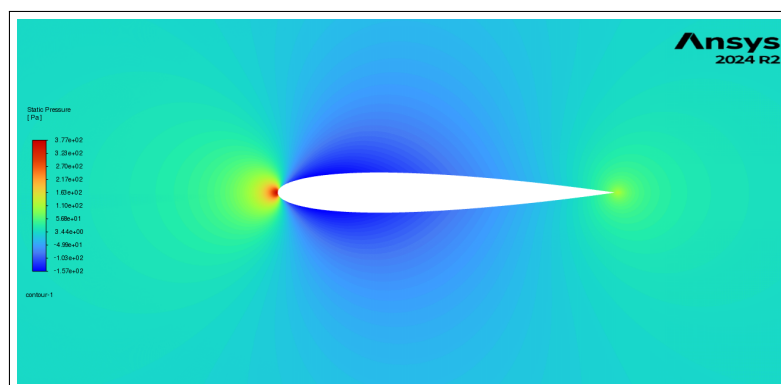


Figure 18: Static Pressure Contour depicting Flow Separation

4.2.2 5 Degree AOA

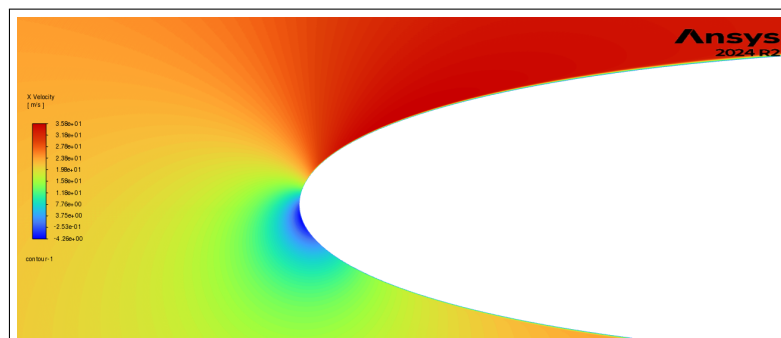


Figure 19: X Velocity Contour depicting Flow Separation

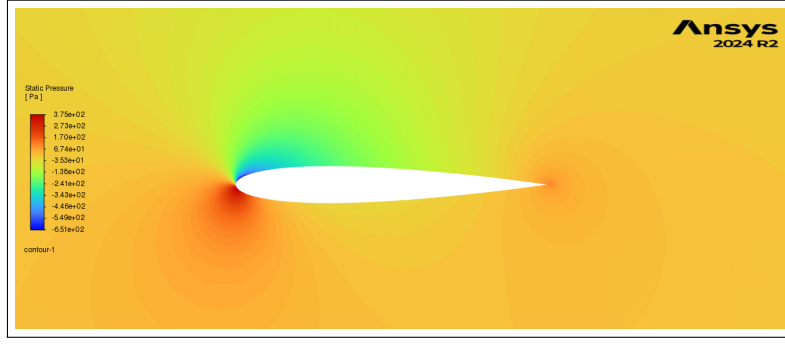


Figure 20: Static Pressure Contour depicting Flow Separation

5 Observations

5.1 Observations at $\alpha = 0^\circ$ and 5°

At $\alpha = 0^\circ$, the airfoil generated negligible lift, as expected from its symmetric profile under aligned flow. The lift coefficient (C_L) was approximately zero, while the drag coefficient (C_D) remained lower than experimental calculations.

At $\alpha = 5^\circ$, an increase in lift was observed. The simulation produced a lift coefficient of $C_L \approx 0.51$, nearly consistent with thin airfoil theory and published experimental results. The drag coefficient also increased slightly due to the rise in pressure drag, with $C_D \approx 0.009$. The increase in lift coefficient was also expected due to asymmetric contact of air with the airfoil.

5.2 Comparison with Literature and Error Sources

The simulation results showed good agreement with literature for both C_L and C_D at low angles of attack. Minor deviations (within 5–15%) were observed and can be attributed to the following factors:

- **2D Flow Assumption:** The simulation was carried out in 2D, neglecting 3D effects like spanwise flow or tip vortices present in wind tunnel experiments.
- **Numerical Method:** While the SIMPLE scheme is stable and widely used, its segregated nature may slightly underperform compared to coupled solvers in resolving pressure gradients, though it was found more reliable in this case.
- **Mesh Resolution:** The Mesh could be resolved into a finer version to allow for better convergence.
- **Iterations:** The significant computational time taken for simulations hindered the use of higher extent of iterations to converge the solution. The iterative method thus had certain percentage of error associated with it.

Overall, the laminar simulation successfully captured the aerodynamic performance of the NACA 0012 airfoil at low angles of attack, and the results validate well against benchmark data.

6 Geometry

The chord length for the NACA 0012 Airfoil was chosen to be 1m. It was understood from various sources and tutorials that convergence for Airfoil simulation occurs better when a C-Shaped Inlet is chosen in the Airfoil's upstream while a Rectangular Region is chosen in its downstream. Moreover, since the Length of the chord is 1m, the suggested C-shaped inlet was suggested to have radius $\geq 7.5 \times \text{Chord Length}$. Also, the rectangular region's length was suggested to be $\geq 15 \times \text{Chord Length}$. These conditions were sourced from Tutorials for better convergence. After dimensioning the geometry, it was divided into 6 parts. 2 parts were completely upstream of the flow, 2 parts were above

and below the airfoil while 2 parts were downstream of the flow. The part above and below the airfoil was created to help in higher mesh refinement and for better capturing of Fluid Behaviour near the airfoil.

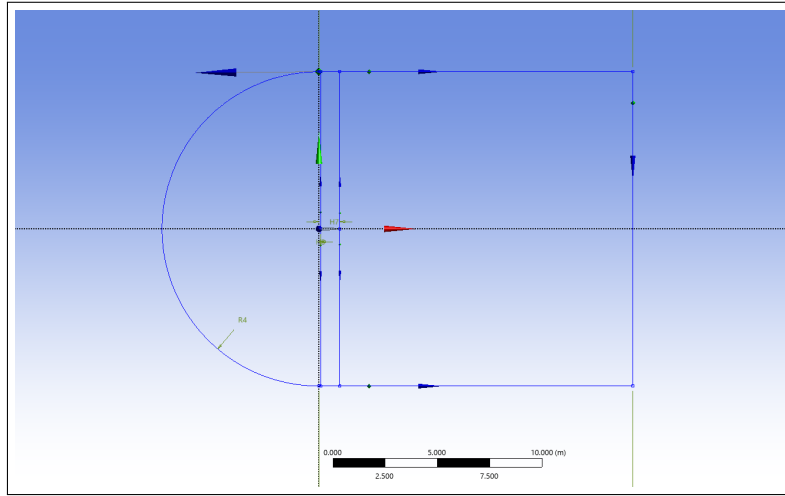


Figure 21: Geometry

2 geometries were created.

Geometry 1:
 Radius = 7.5m
 Length = 15m
 Geometry 2:
 Radius = 10m
 Length = 20m

The results have however been shown for Geometry 2 as better and closer to experimental value results were produced for this geometry.

This can be attributed to the fact that a larger area study around the airfoil would have allowed for better capturing of the Fluid Flow phenomenon and Boundary Effects.

7 Meshing

Continuing with Geometry 2, 2 meshes have been created. The Simulation results came to be better for Mesh 2 with higher nodes and cells, and thus, the results have been shown for the finer mesh (Mesh 2)

Primarily, edge sizing was used for providing mesh divisions to edges of the 6 parts. Bias was kept in the sizing to allow for better capturing of the fluid behaviour. Moreover, Hard Meshing was used when the mesh cells were needed to be rectangular/quadrilateral. At last, face meshing was utilised to keep uniform, quadrilateral shaped cells in the Mesh.

Y+ Value: It's important that your mesh near the wall is properly sized to ensure accurate simulation of the flowfield. The Y+ value is an indicator of this accuracy. It is the height of the first mesh cell off the wall required to achieve a desired Y+. Generally Y+ ≥ 1 allows for a good simulation.

The Y+ value calculated for the given flow conditions came to be : **1.464e-4 m**

Mesh 1:
 Nodes: 273941

Elements: 272750

Distance between Cells near the Airfoil: 6.37×10^{-6} m

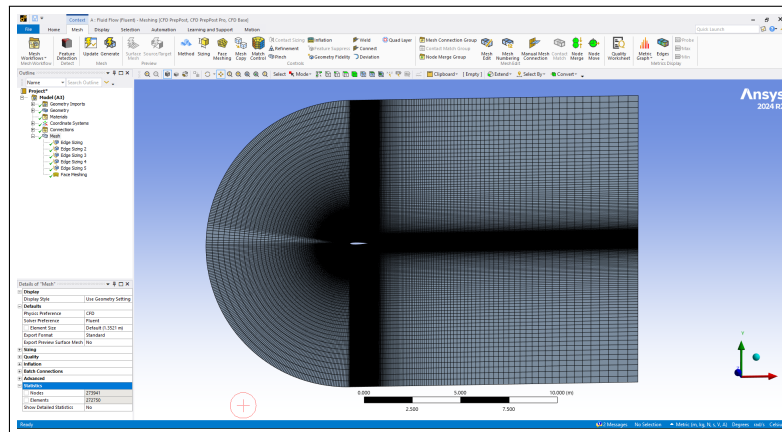


Figure 22: Mesh 1

Mesh 2:

Nodes: 1442880

Elements: 1440000

Distance between Cells near the Airfoil: 4.26×10^{-6} m

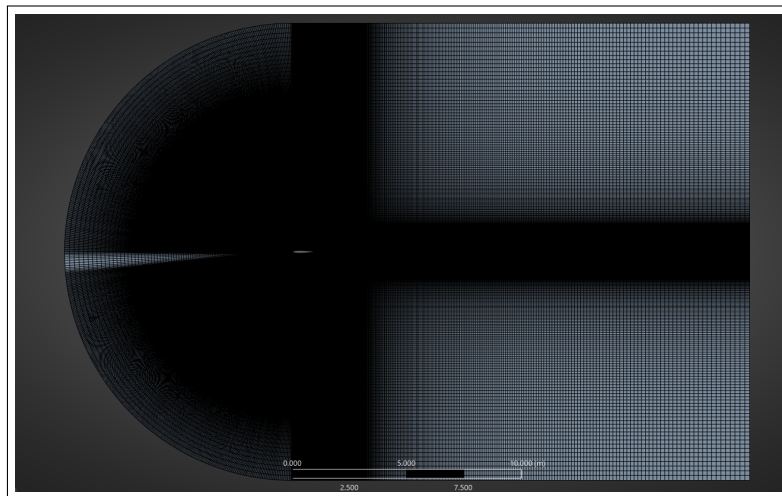


Figure 23: Mesh 2

8 FLUENT Setup

The Solver settings have been picturised as follows:

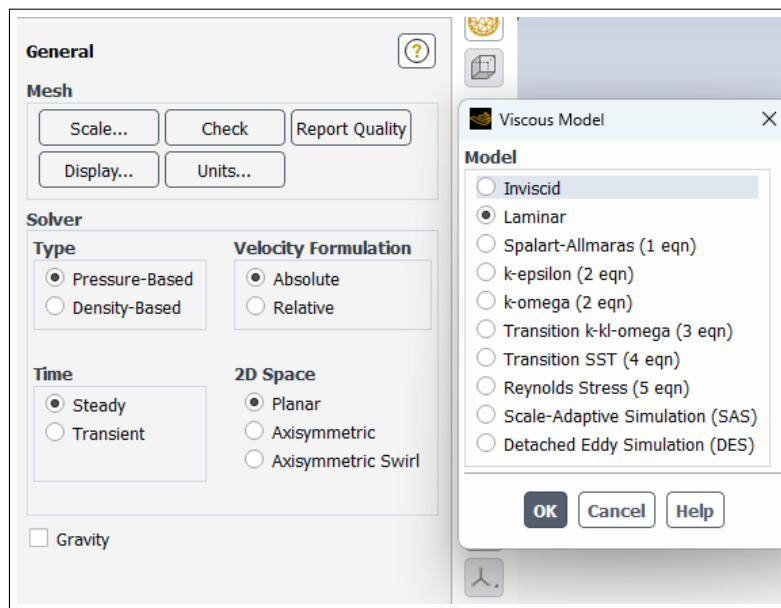


Figure 24: Model

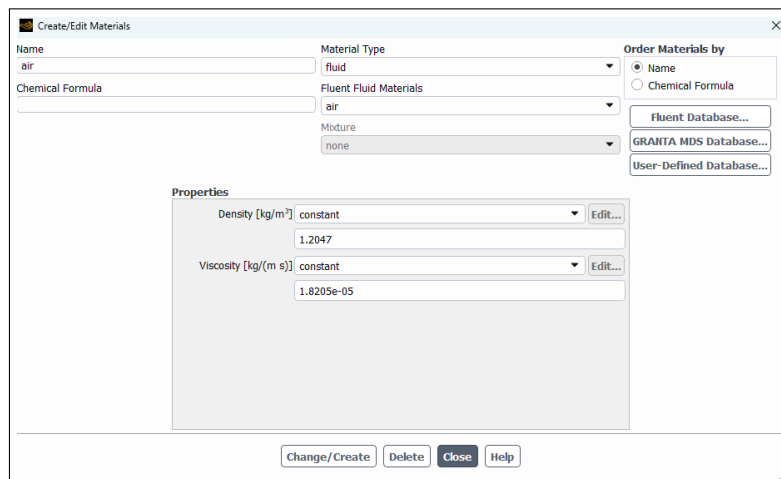


Figure 25: Material

Solution Methods

?

Pressure-Velocity Coupling

Scheme

SIMPLE

Flux Type

Rhie-Chow: momentum based

☒ Auto Select

Spatial Discretization

Gradient

Least Squares Cell Based

Pressure

Second Order

Momentum

Second Order Upwind

Pseudo Time Method

Off

Transient Formulation

☐ Non-Iterative Time Advancement

☐ Frozen Flux Formulation

☐ Warped-Face Gradient Correction

☐ High Order Term Relaxation

Default

Figure 28: SIMPLE Solution Method

Solution Methods

?

Pressure-Velocity Coupling

Scheme

Coupled

Flux Type

Rhie-Chow: momentum based

☒ Auto Select

Spatial Discretization

Gradient

Least Squares Cell Based

Pressure

Second Order

Momentum

Second Order Upwind

Pseudo Time Method

Global Time Step

Transient Formulation

☐ Non-Iterative Time Advancement

☐ Frozen Flux Formulation

☐ Warped-Face Gradient Correction

☐ High Order Term Relaxation

Default

Figure 29: COUPLED Solution Method

?

Reference Values

Compute from

inlet

Reference Values

Area [m²]1

Density [kg/m³]1.2047

Depth [m]1

Enthalpy [J/kg]0

Length [m]1

Pressure [Pa]0

Temperature [K]293

Velocity [m/s]25

Viscosity [kg/(m s)]1.8205e-05

Ratio of Specific Heats1.4

Yplus for Heat Tran. Coef.300

Reference Zone

fluid-surface_body

Figure 30: Reference Values

8.1 Solution Method

In this simulation, two common pressure-velocity coupling schemes used in ANSYS Fluent—SIMPLE (Semi-Implicit Method for Pressure-Linked Equations) and COUPLED—were considered. The SIMPLE algorithm solves the momentum equations and then corrects the pressure and velocity fields iteratively. It is stable and widely used for steady-state, incompressible flows due to its lower computational cost. In contrast, the COUPLED scheme solves the momentum and continuity equations simultaneously in a fully coupled manner. This method provides faster convergence for high-speed compressible flows or transient simulations but is more computationally expensive. For the laminar, low-speed airflow over the NACA 0012 airfoil, the SIMPLE method provided accurate results with efficient convergence, making it suitable for this study.

9 References

1. SimFlow CFD Software, “NACA 0012 Airfoil Validation Case,” SimFlow Documentation, 2023. [Online]. Available: <https://help.sim-flow.com/validation/naca-0012-airfoil>. [Accessed: Apr. 19, 2025].
2. “NACA2412 Tutorial in ANSYS Fluent (Student Version) - Lift, Drag, Angle of Attack” *YouTube*, 2020. [Online]. Available: <https://youtu.be/3i9Ryq-m1HA>. [Accessed: Apr. 19, 2025].