CFD Analysis of Wind Turbine Aerofoil at Various Angles of Attack

Name   :   H.S.S.M.Hewawasam

Index number   :   200221G

Table of Contents

Table of Figures

List of Tables

1   Introduction

An analysis of the NACA0012 wind turbine airfoil's aerodynamic efficiency at different angles of attack while keeping its Reynolds number constant is the main focus of the study. By improving the aerofoil design, which is essential to wind turbine performance, this research aims to increase wind turbine efficiency. The study intends to determine the effect of variations in the angle of attack on the aerodynamic properties of the aerofoil using simulations conducted with Ansys-Fluent. This research aims to establish CFD as an accurate alternative for conventional experimental methods by using a refined computational mesh around the aerofoil and validating the results with wind tunnel experiments, potentially saving time and money needed for empirical testing.
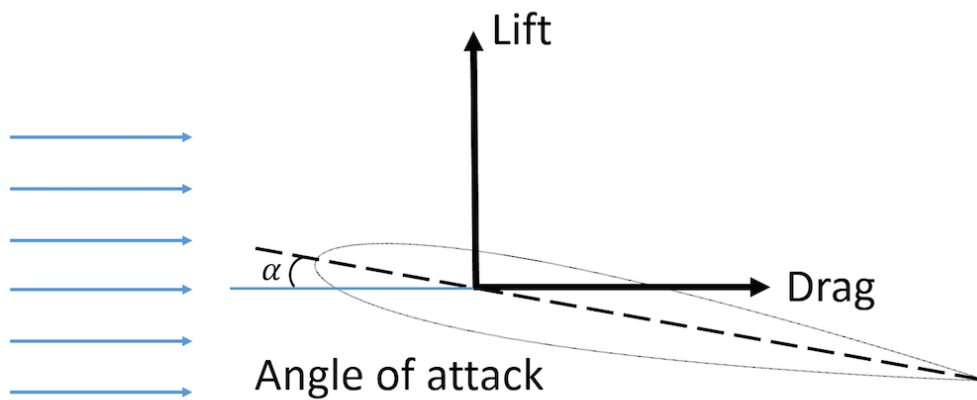
Figure 1 Lift and drag on aerofoil

NACA 0012 AIRFOILS - NACA 0012 airfoil
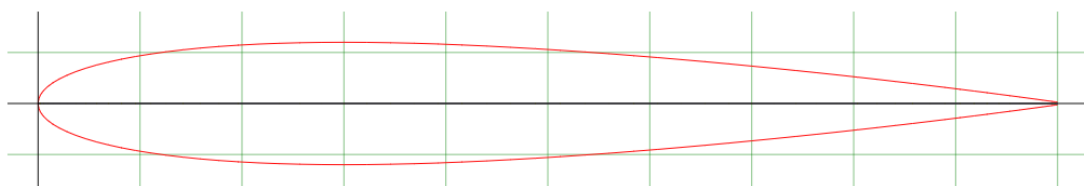
Figure 2 Geometry of NACA 0012 Airfoil

2 Assumptions and Simplifications

Steady-State Flow

It is assumed that the flow surrounding the aerofoil is constant over time. The equations required to model the flow are made simpler as a result.

Incompressible Flow

The fluid dynamics equations' density calculations are made simpler by treating the air as incompressible, specifically at subsonic speeds.

Two-Dimensional Analysis
The study probably makes the assumption that the flow can be sufficiently described in two dimensions, which lowers the need for resources and computational complexity.

Ignoring Gravity

In such aerodynamic calculations, the effects of gravity on the flow are typically insignificant and are hence frequently ignored.

Turbulence Model

The Realizable k-ε turbulence model is used in the study to simulate the impacts of airflow turbulence. Because it can manage flows with a high angle of attack, where dissociation and reattachment may occur, this model was selected.

3   Flow Domain and Computational Mesh

3.1   Information about the flow domain
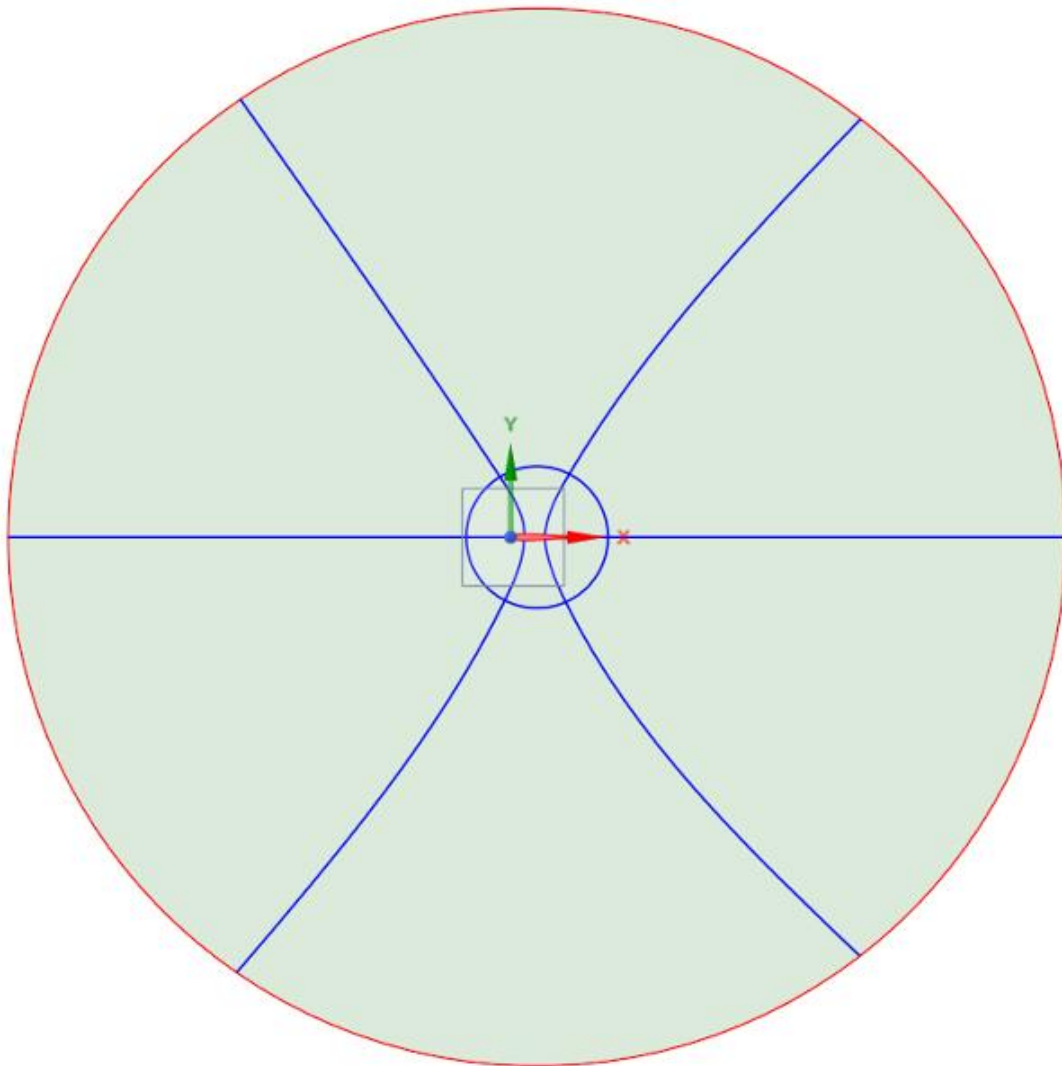
Figure 3 Geometry of NACA 0012 aerofoil

Chord length of the aerofoil – 1m

Diameter of the outer circle – 20m

Diameter of the inner circle – 6m

3.2   Inputs and boundary conditions

No	Input	Value
1	Fluid medium	Air
2	Velocity of flow	51.4496 m/s
3	Operating pressure	101325 Pa
4	Density of fluid	1.1767 kg/m^3
5	Reynolds number	10^6
6	Chord length	1 m
7	Operating temperature	277 k
8	Angles of attack	4, 6, 8, 10 degree
9	Model	Realizable $k - \varepsilon$
10	Viscosity	1.009×10^{-5} kg/m.s

Figure 4 Boundary conditions
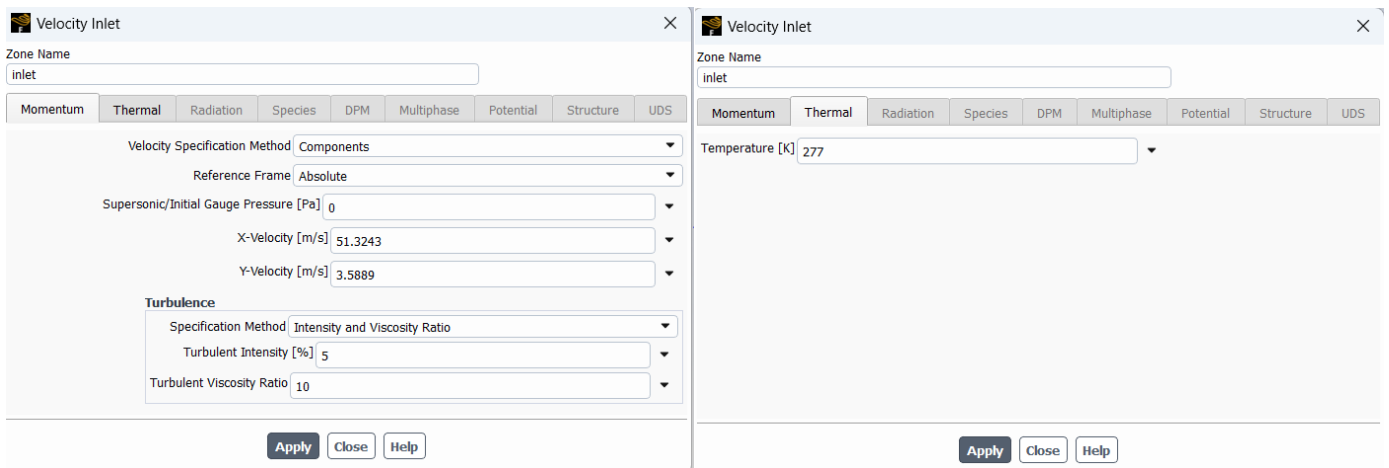
Figure 5 Inlet conditions for angle of attack 4 degree

Table 1 Velocity components with angle of attack

angle	V_act	V_x	V_y
4	51.4496	51.3243	3.5889
6	51.4496	51.168	5.378
8	51.4496	50.9489	7.16
10	51.4496	50.6679	8.9341

4

3.3 Basic mesh statistics

3.3.1 Number of elements and number of nodes
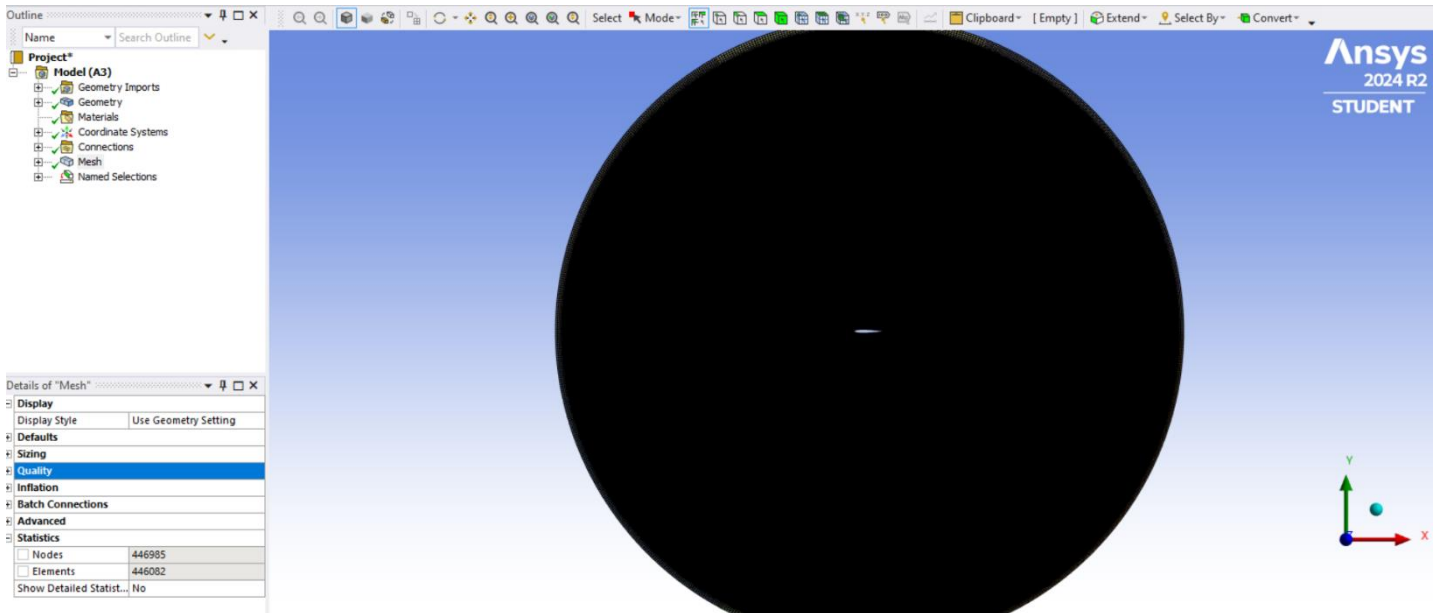
Figure 6 Number of elements and nodes

number of elements - 446082

number of nodes - 446985

3.4 Mesh metrics

3.4.1 Orthogonality
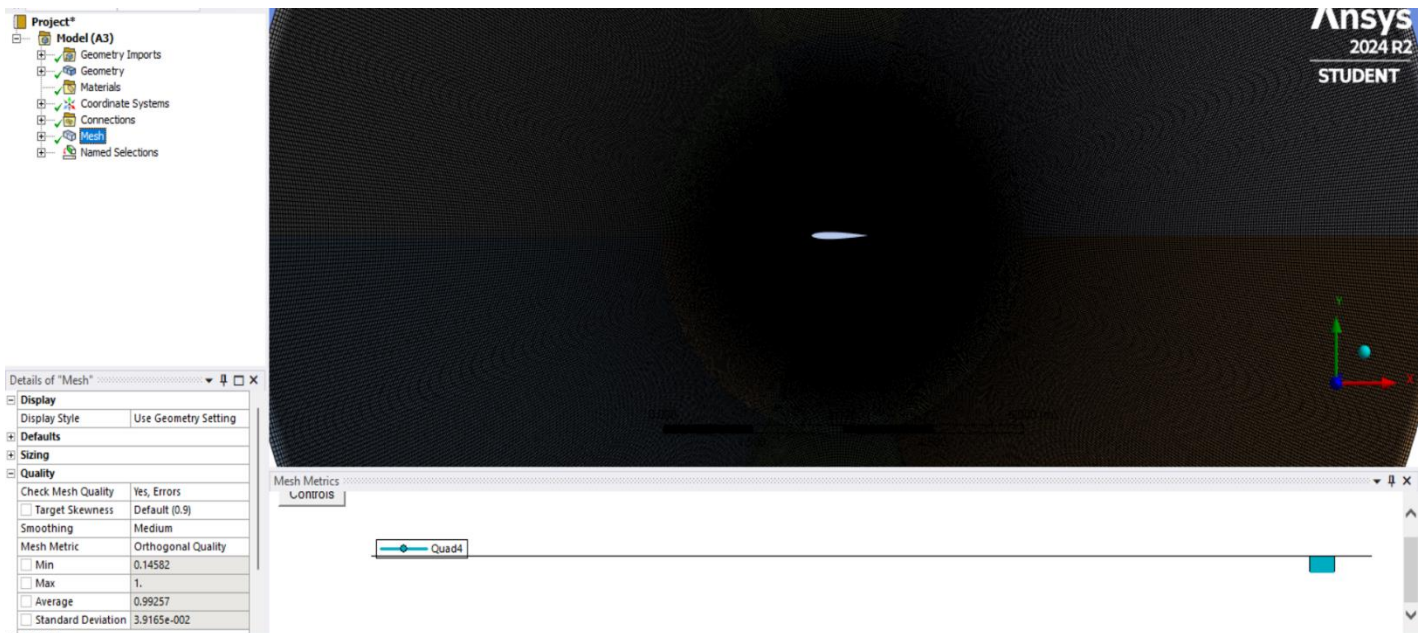
Figure 7 Orthogonality

3.4.2 Skewness

Figure 8 Skewness

6
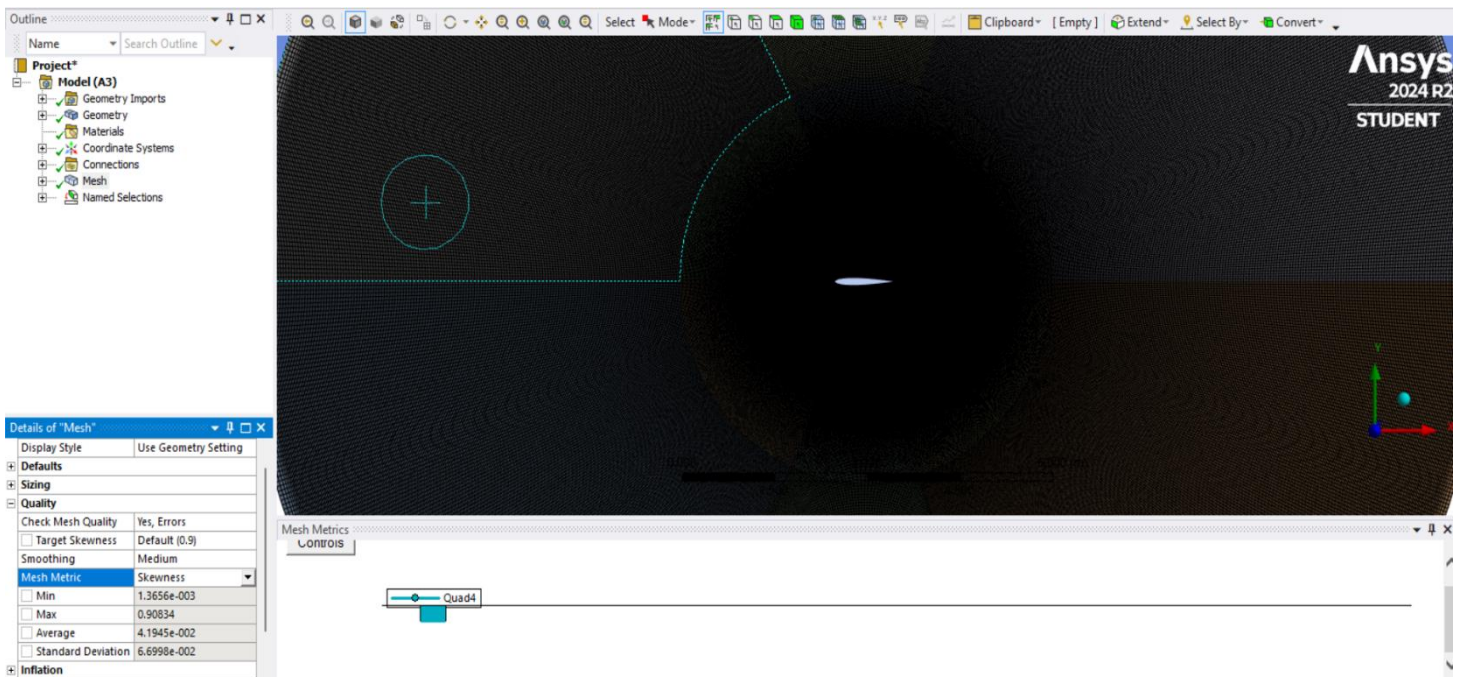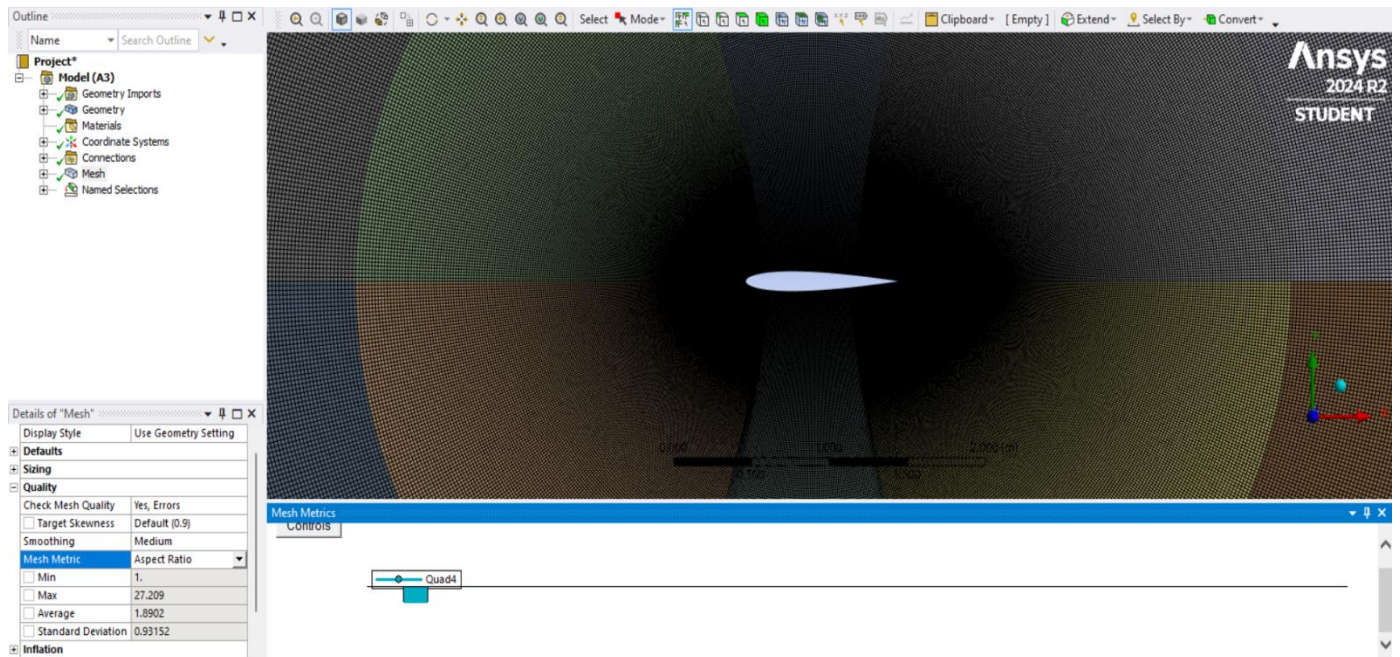
3.4.3　Aspect ratio

Figure 9 Aspect ratio

3.4.4　Element quality
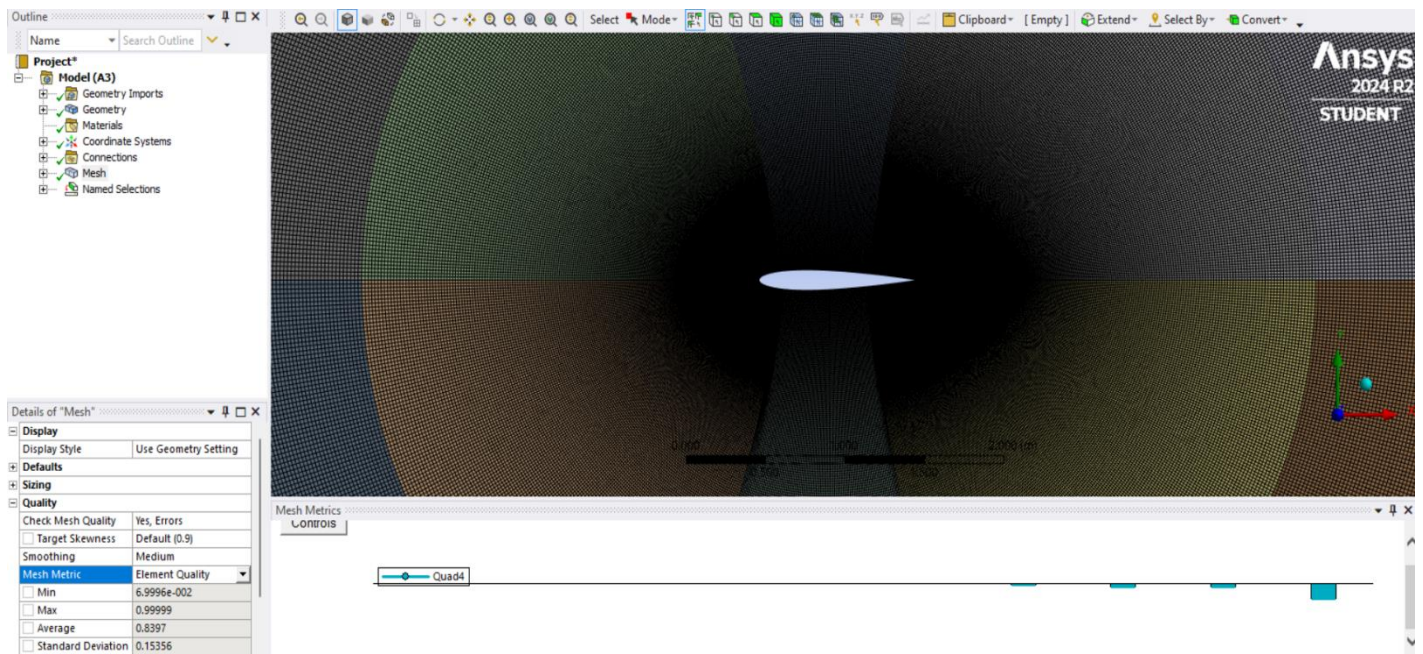
Figure 10 Element quality

Table 2 Recommended values

Parameter	Recommended values						
	Average value in the mesh	Unacceptable	Bad	Acceptable	Good	Very good	Excellent
Skewness	0.041945	0.98-1.0	0.95-0.97	0.8-0.94	0.5-0.8	0.25-0.5	0-0.25
Orthogonal quality	0.99257	0-0.001	0.001-0.14	0.15-0.2	0.2-0.69	0.7-0.95	0.95-1.0

All of the mesh's average values fall within the excellent region, as shown in the above table.

4 Simulation Settings

4.1 Convective flux calculation scheme

The convective flux calculation scheme in this simulation is configured to be "Second Order Upwind." The convective terms in the fluid flow governing equations, which are crucial for predicting how variables like velocity and pressure change within the flow field, are discretized using this technique. By better capturing the transport of flow characteristics, "Second Order Upwind" provides a higher degree of accuracy than first-order schemes, particularly in areas with steep gradients or different flow directions. This method preserves a more accurate resolution of the flow's features by reducing numerical diffusion, which is the artificial smearing of sharp interfaces and gradients. Its application is essential to guarantee that the simulation results accurately reflect the flow's physical behavior, improving confidence in the CFD model's dependability and durability as a predictive tool.

4.2 Diffusive flux calculation scheme

Diffusive flux discretization in the Ansys Fluent simulation settings is affected by the gradient calculation method selected under "Spatial Discretization." Since the gradient of scalar quantities is crucial to the computation of diffusive fluxes, the chosen approach, "Least Squares Cell Based," applies to approximate it. In order to estimate the gradient at the cell center, this method uses a least squares approach, which fits the best plane through the values in nearby cells. The simulation achieves a more accurate representation of diffusive transport processes, including molecular diffusion and thermal conduction in fluids, by utilising the "Least Squares Cell Based" method.
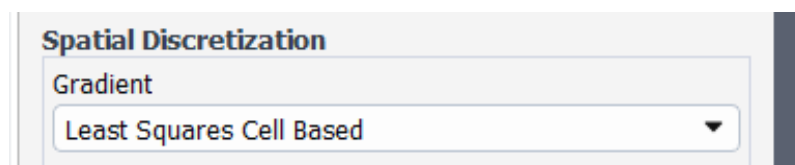

Figure 11 Spatial discretization

4.3 Turbulence modelling

The NACA0012 aerofoil is studied using the Realisable k−ϵ turbulence model because it is excellent at handling complex flow dynamics that include swirling effects and strong curvature, which are common around aerofoil structures. By more accurately predicting the airfoil's surrounding air behavior a critical component in accurately simulating the airfoil's performance at various angles. It is especially useful for studying situations where the aerofoil may stall or experience flow separation, which are frequent problems in wind turbine operations, because it can handle high angles of attack. which also makes it an excellent option for engineering applications involving aerodynamic simulations.

Figure 12 Turbulence model

4.4   Pressure velocity coupling

In CFD simulations, the coupled scheme is preferred over segregated solvers due to its capacity to improve convergence stability and speed. When accurate pressure-velocity field interaction is needed, as in aerofoil analysis at different angles of attack, this scheme is needed. It is effective for steady-state simulations where quick and accurate results are crucial because it enables faster convergence to a stable and accurate solution. The Coupled scheme solves the momentum and pressure equations simultaneously, in contrast to segregated solvers that do so step-by-step. A synchronised solution that can manage sudden shifts in flow conditions around an aerofoil is made possible by this integrated approach, which guarantees a strong interaction between the pressure and velocity fields. This is important in aerodynamic simulations, as accurate calculations of forces such as lift and drag are dependent much on the precision of these field interactions.

Figure 13 4.4 Solution methods

5 Results and Validation

5.1 Grid independency analysis

Set the aerofoil to 4-degree angle of attack and test for the grid independent test. After that, note how the lift and drag coefficients alter with the number of mesh elements.
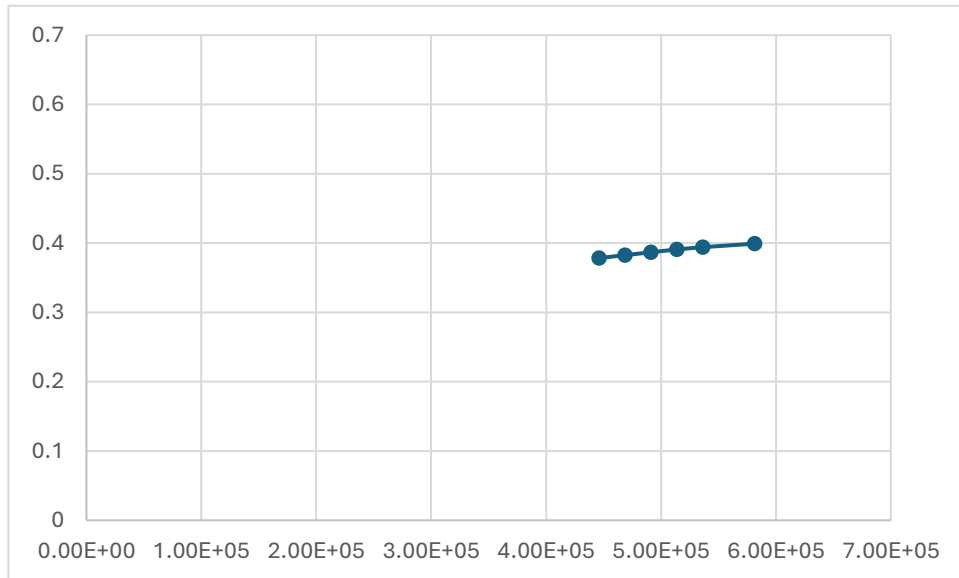

Figure 14 Lift coefficient vs number of elements

Table 3 Lift coefficient with number of elements

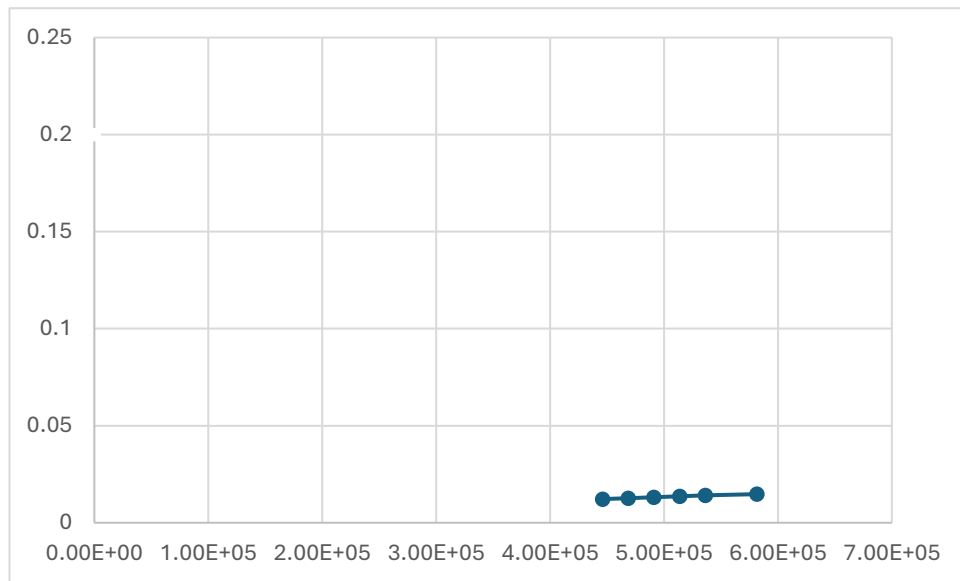| Number of elements	lift coefficient
4.46E+05	0.37834
4.69E+05	0.38263
4.91E+05	0.38685
5.14E+05	0.39081
5.36E+05	0.39417
5.82E+05	0.39918

Figure 15 Drag coefficient vs number of elements

Table 4 Drag coefficient with number of elements

Number of elements	drag coefficient
4.46E+05	0.0122
4.69E+05	0.012718
4.91E+05	0.01322
5.14E+05	0.01369
5.36E+05	0.014098
5.82E+05	0.014738

5.2   Solution convergency study

The experiment's final solutions were the lift and drag for the proper angle of attack. To get results, the inlet velocities' X and Y directions are supplied as components. This represents solution converge for angle of attack is equal to 4 degrees.
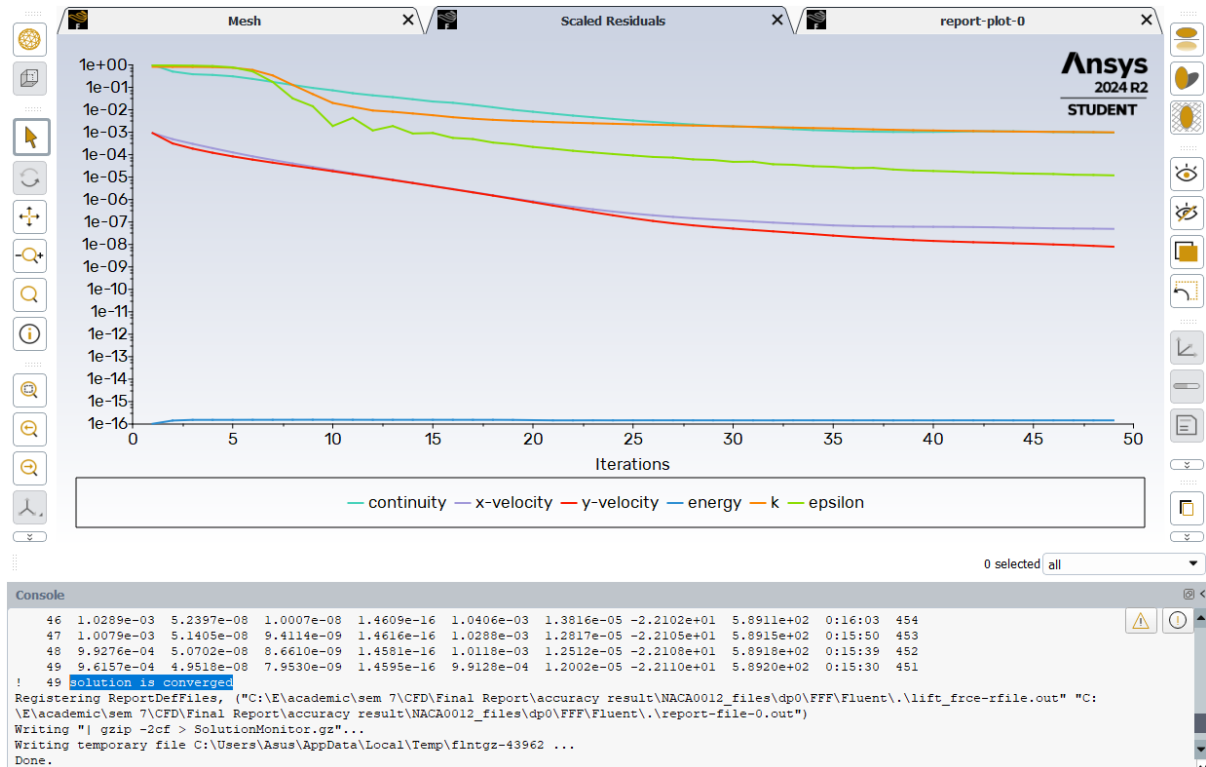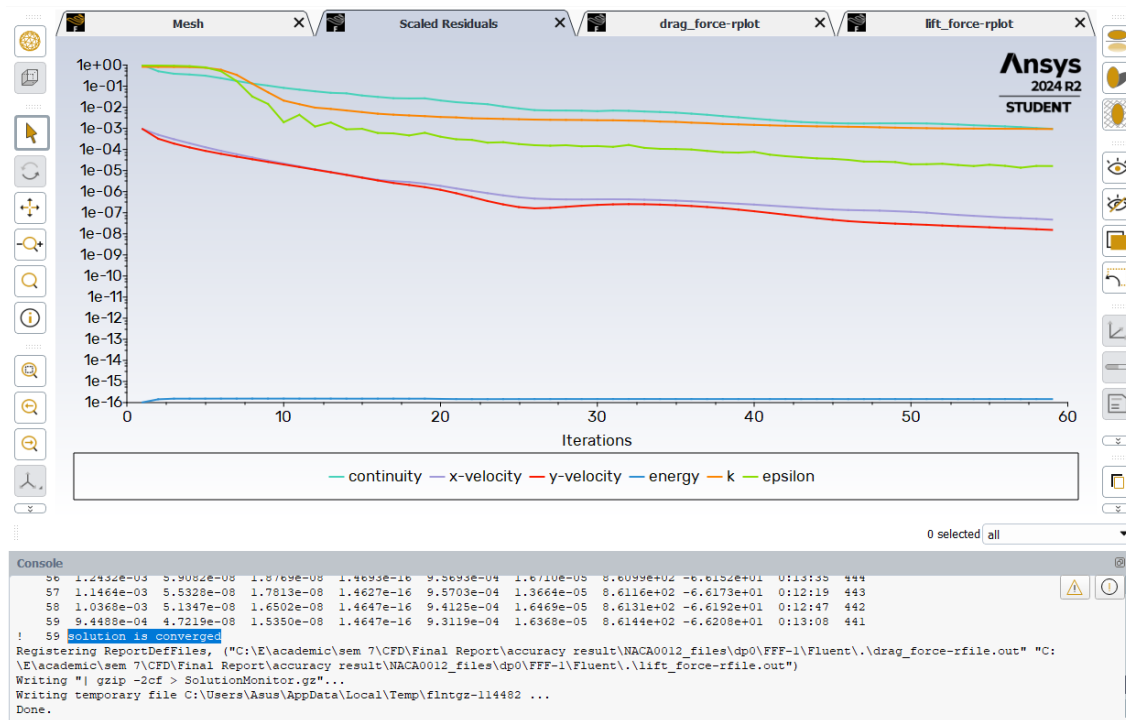

Figure 16 Scale residuals for AOA 4 degree

When angle of attack equal to 6 degrees

Figure 17 Scaled residuals for AOA 6 degrees

When angle of attack equal to 8 degrees

Figure 18 Scaled residuals for AOA 8 degrees
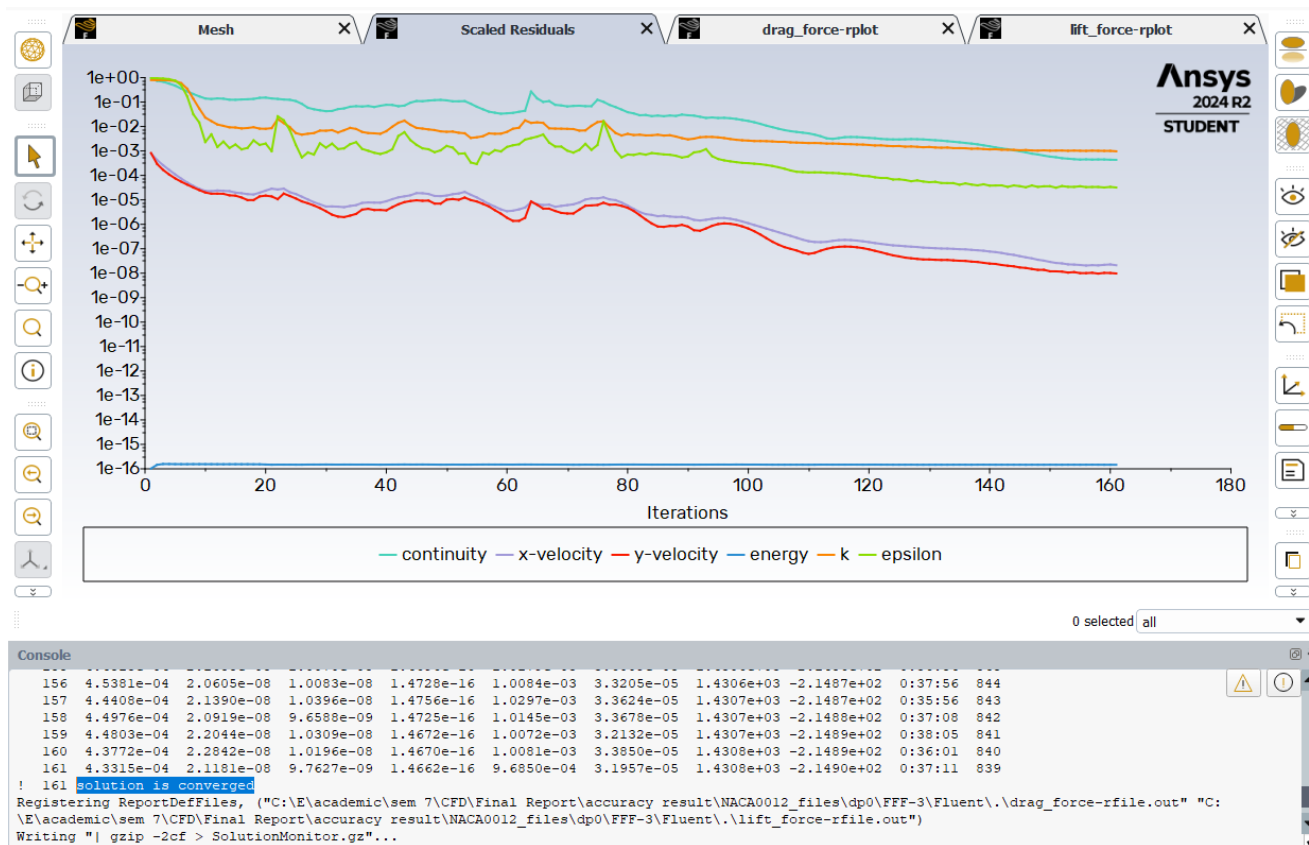
When angle of attack equal to 10 degrees

Figure 19 Scaled residuals for AOA 10 degrees

5.3   Results

The main objective of this Computational Fluid Dynamics (CFD) simulation is to calculate the lift and drag coefficients of the airfoil at a given angle of attack in order to analyze its aerodynamic performance. In order to provide a full understanding of the aerodynamic behavior, the simulation also aims to represent and determine the velocity contour and pressure contour of the airflow surrounding the airfoil. Consequently, the following are the main results and outcomes of the simulation.

Lift coefficient variation with angle of attack
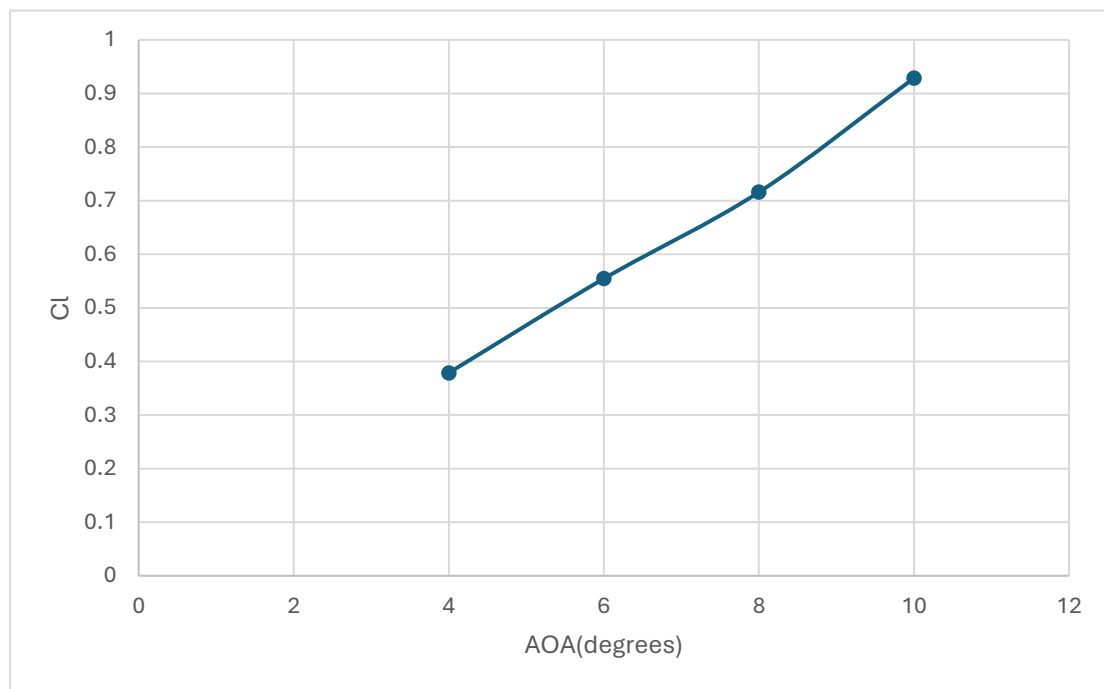
AOA(degrees)	Cl
4	0.3787
6	0.5545
8	0.7158
10	0.9287

Figure 20 Cl vs AOA

Drag coefficient variation with angle of attack

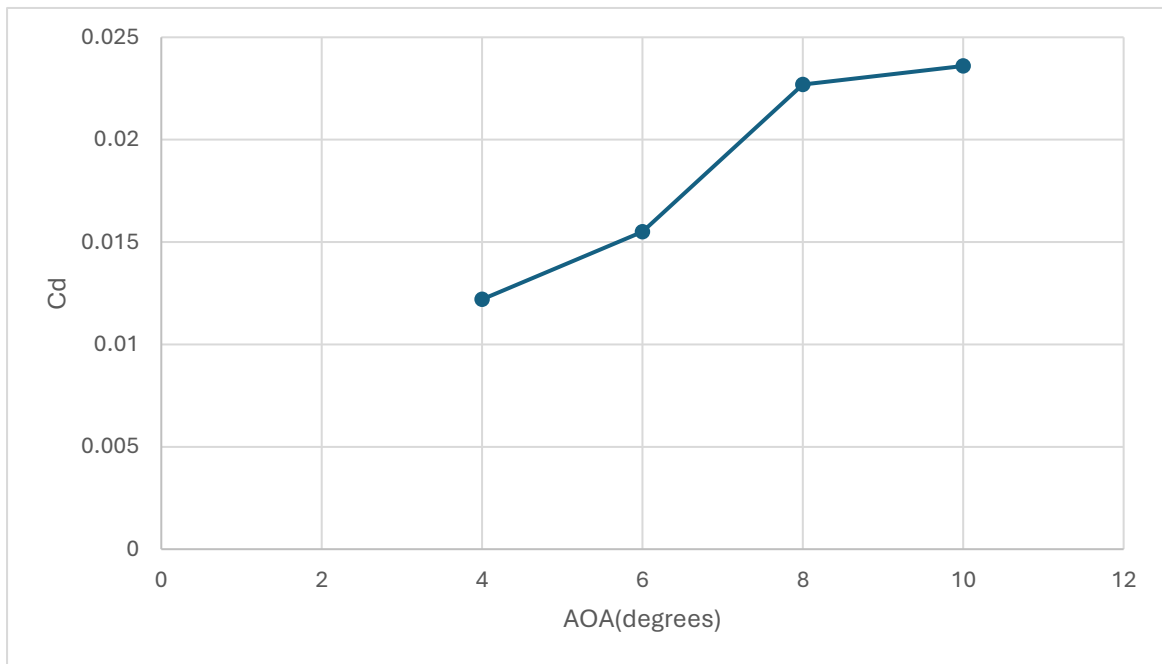| AOA(degree)	Cd
4	0.0122
6	0.0155
8	0.0227
10	0.0236

Figure 21 Cd vs AOA

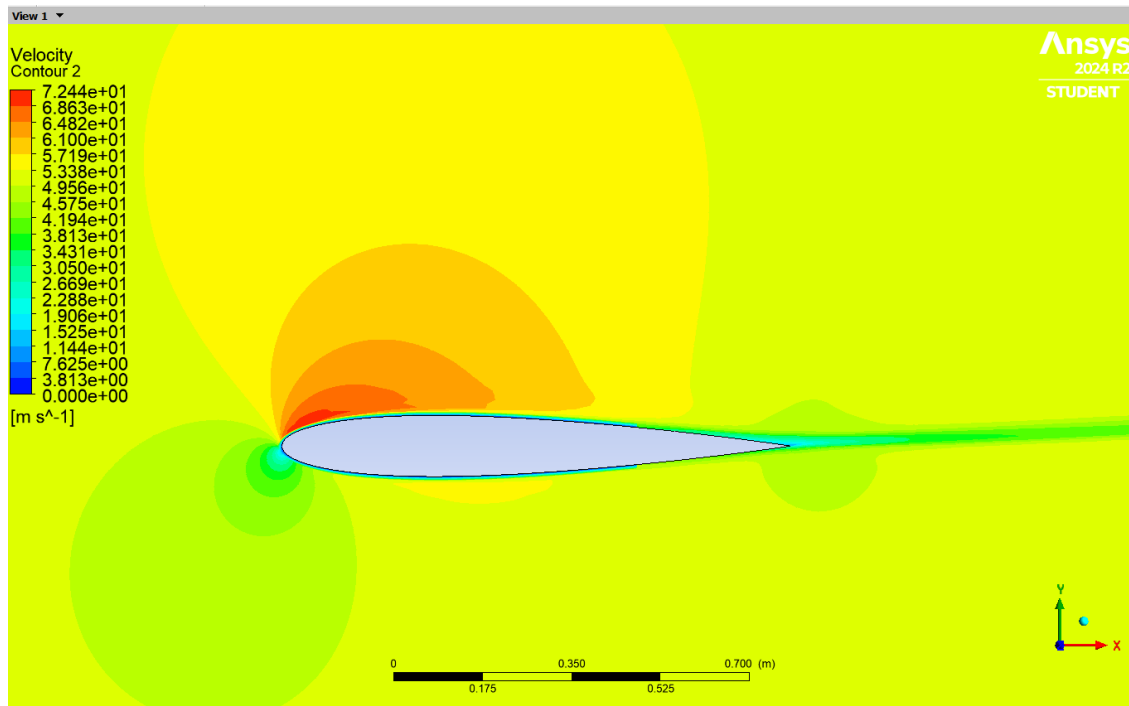Velocity contours around the NACA0012 aerofoil
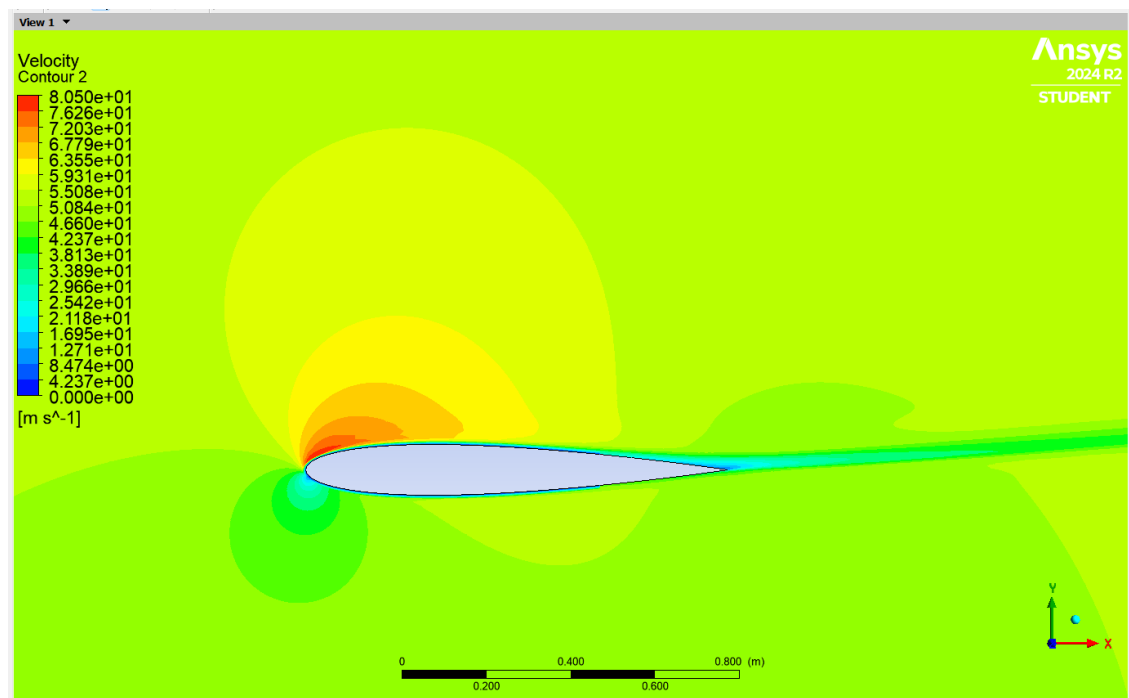

Figure 22 Velocity contour when AOA 4 degrees

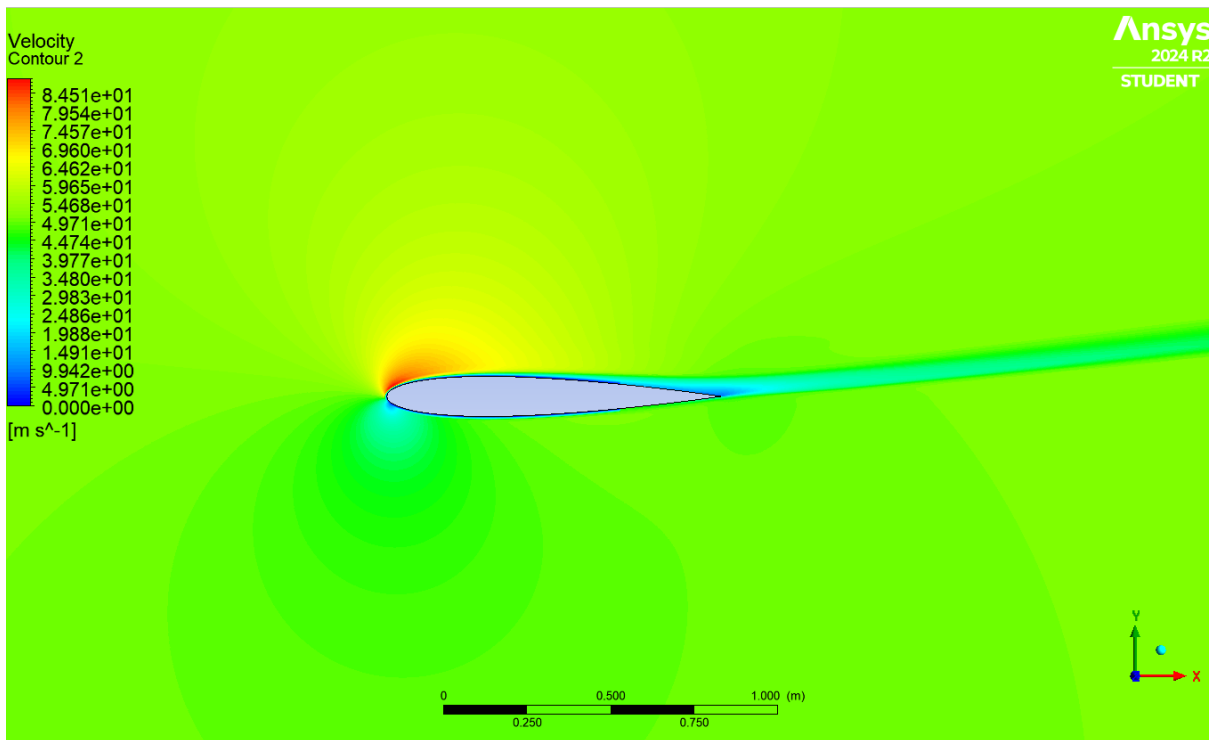Figure 23 Velocity contour when AOA 6 degrees

Figure 24 Velocity contour when AOA 8 degrees
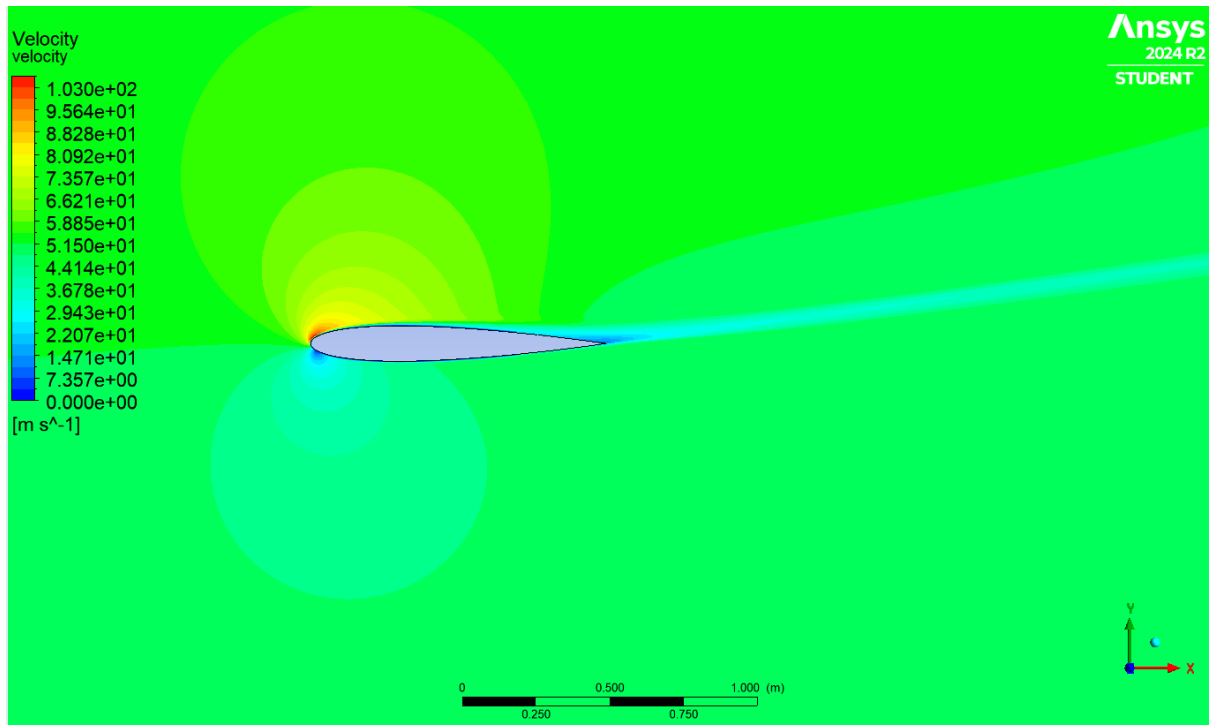

Figure 25 Velocity contour when AOA 10 degrees

Pressure contours around the NACA0012 aerofoil
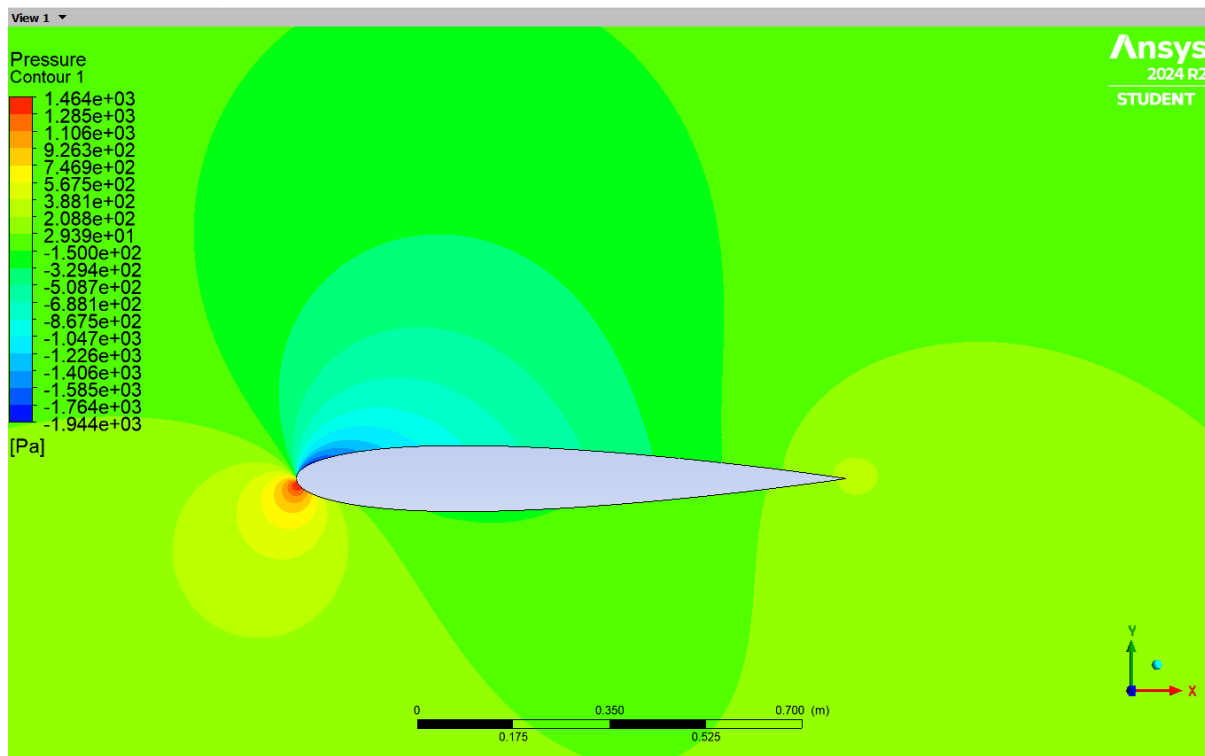

Figure 26 Pressure contour when AOA 4 degrees

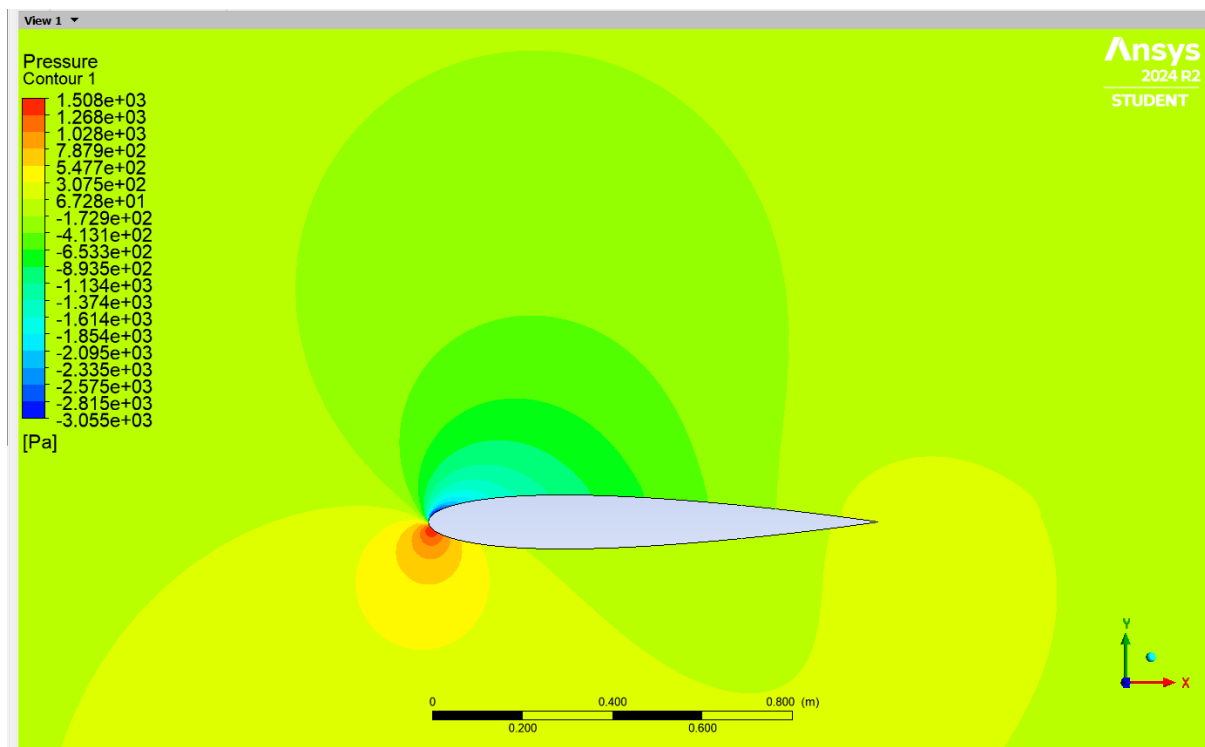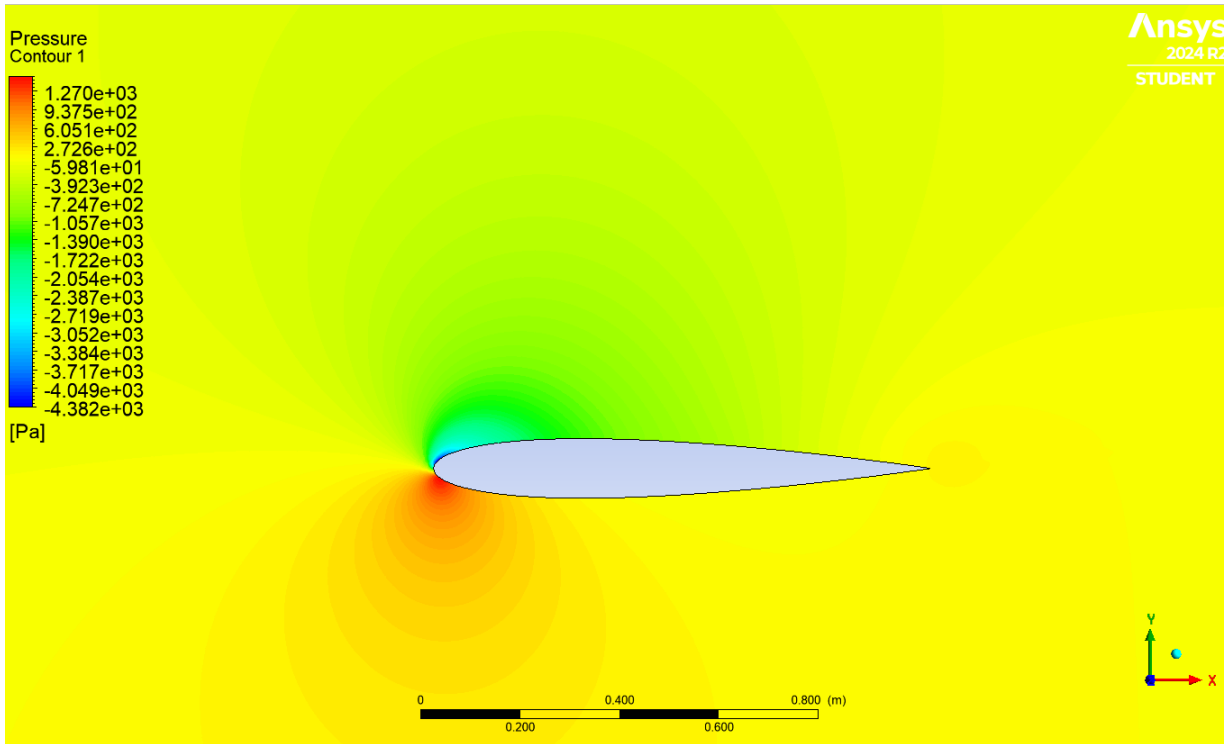Figure 27 Pressure contour when AOA 6 degrees

Figure 29 Pressure contour when AOA 8 degrees

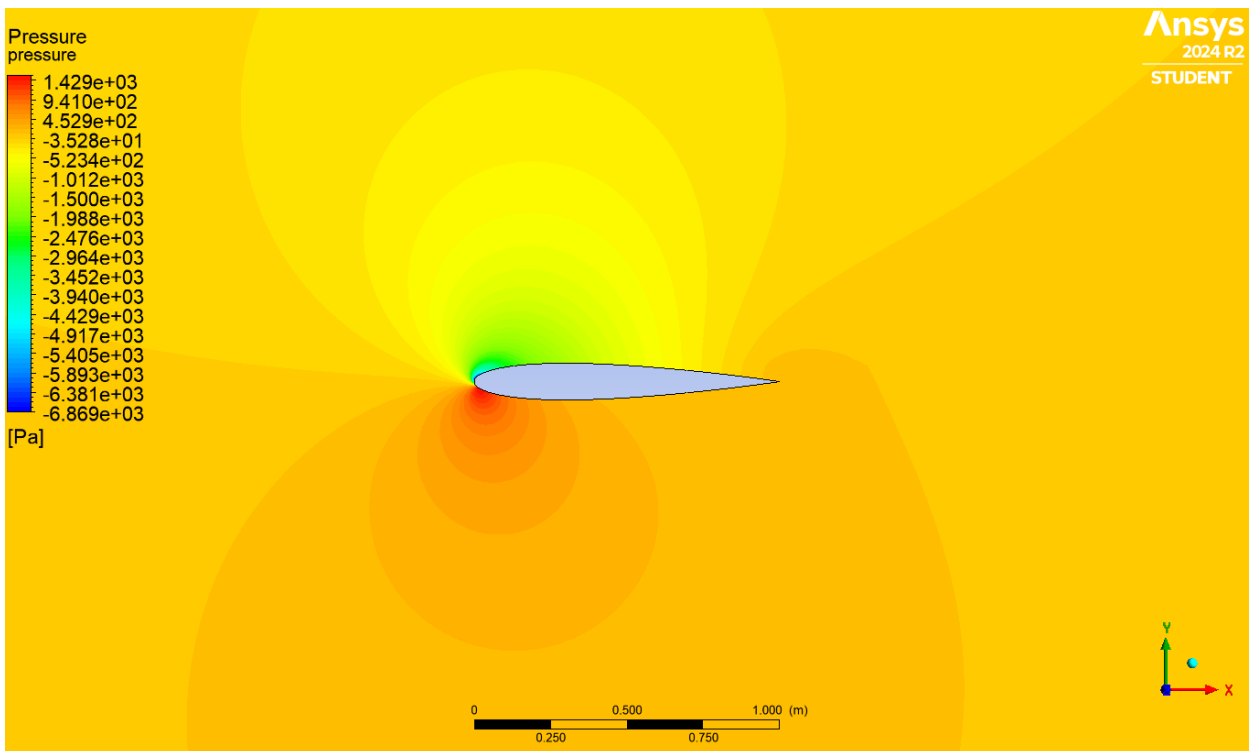Figure 28 Pressure contour when AOA 10 degrees

5.4   Result validation

comparison the results with the experimental results

<u>Lift coefficient variation with angle of attack</u>
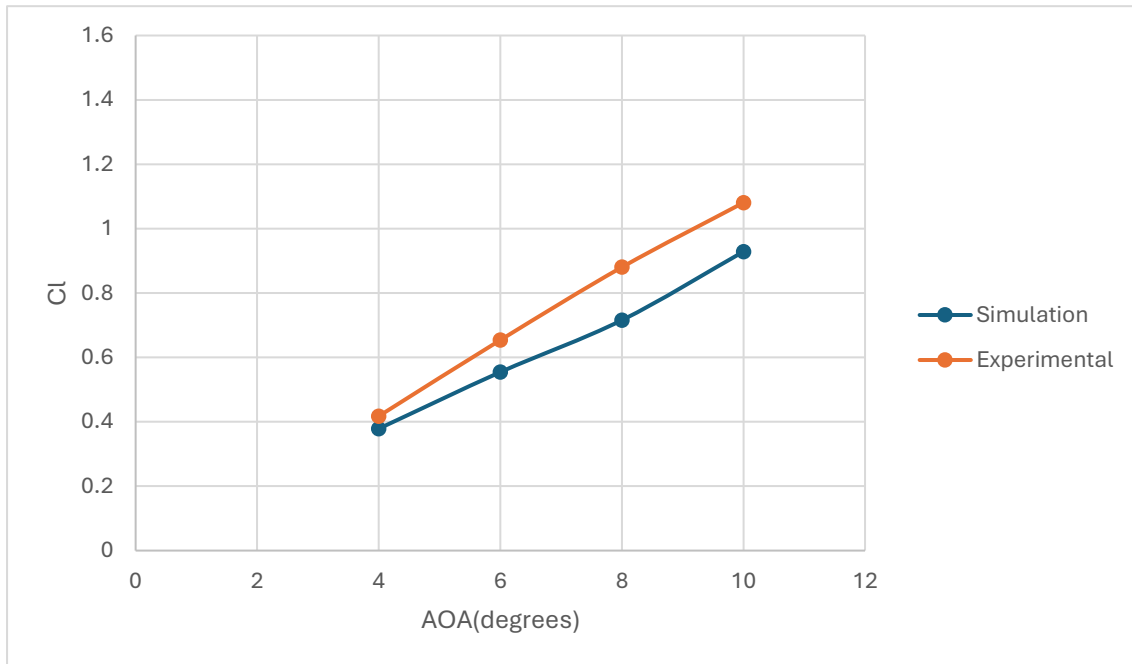
<u>Drag coefficient variation with angle of attack</u>
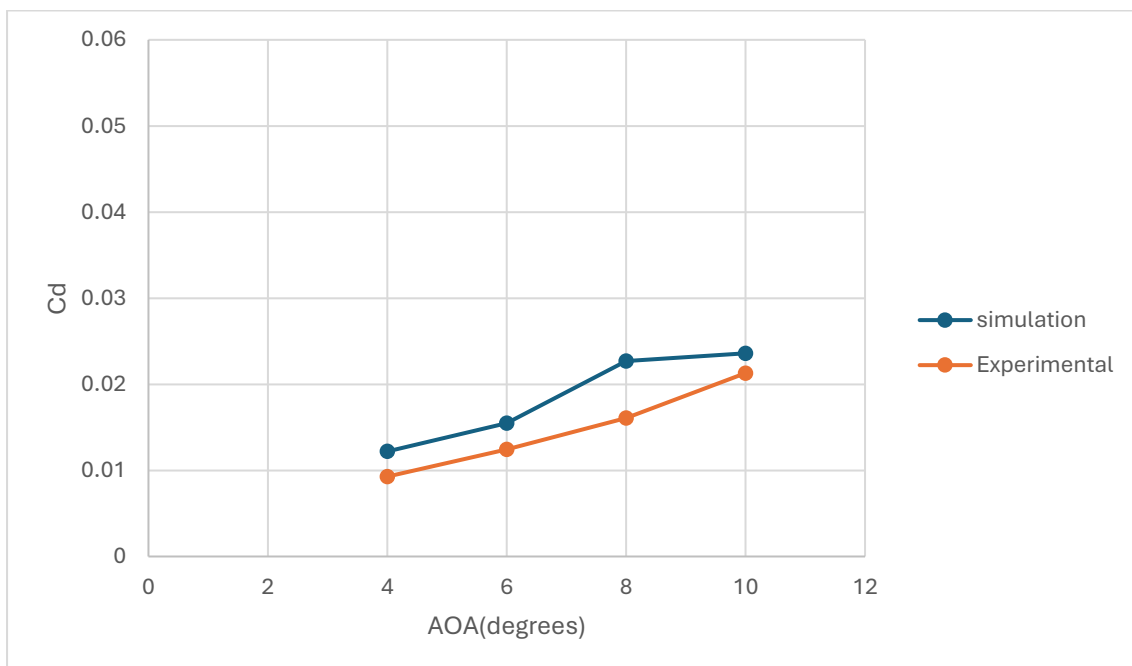
Velocity contours around the NACA0012 aerofoil

Figure 30 Velocity contour when AOA 4 degrees in simulation
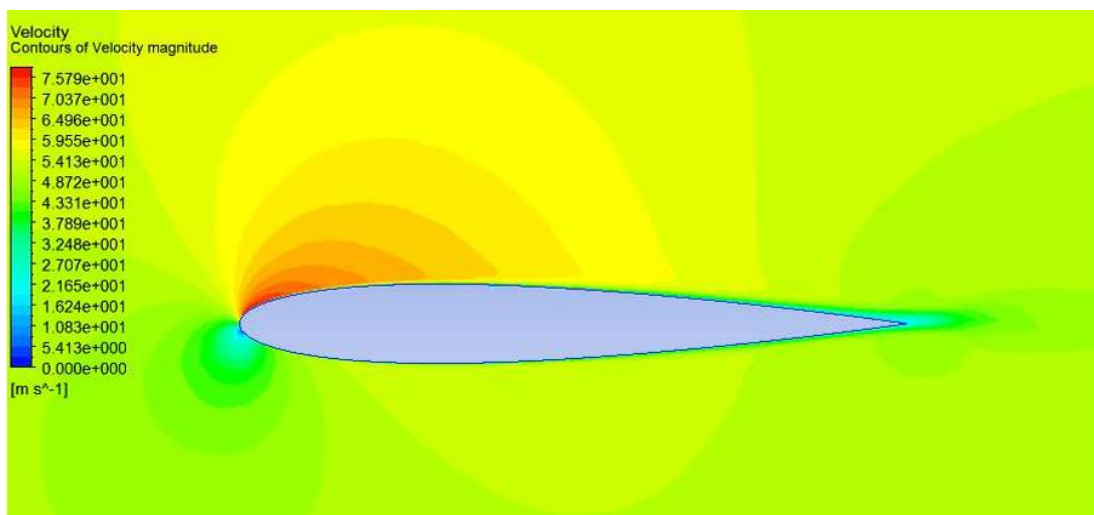
Figure 31 Velocity contour when AOA 4 degrees in research paper
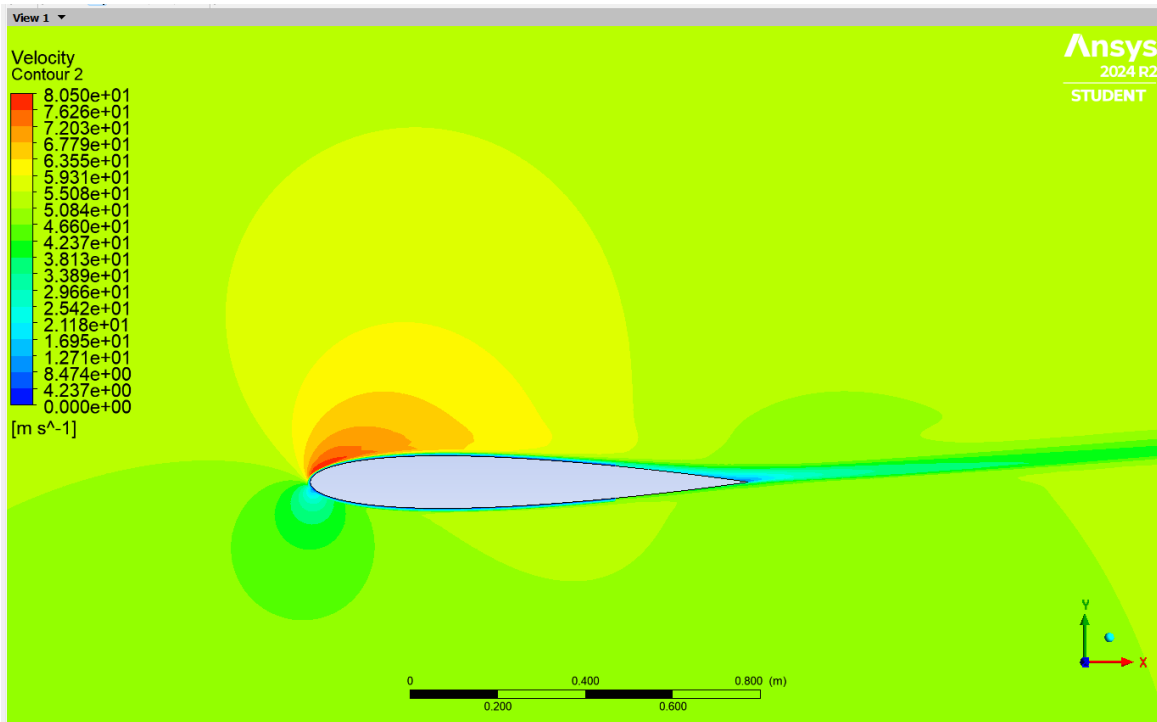
Figure 32 Velocity contour when AOA 6 degrees in simulation

Figure 33 Velocity contour when AOA 6 degrees in research paper

Figure 34 Velocity contour when AOA 8 degrees in simulation

Figure 35 Velocity contour when AOA 8 degrees in research paper

Figure 36 Velocity contour when AOA 10 degrees in simulation

Figure 37 Velocity contour when AOA 10 degrees in research paper

Pressure contours around the NACA0012 aerofoil

Figure 38 Pressure contour when AOA 4 degrees in simulation
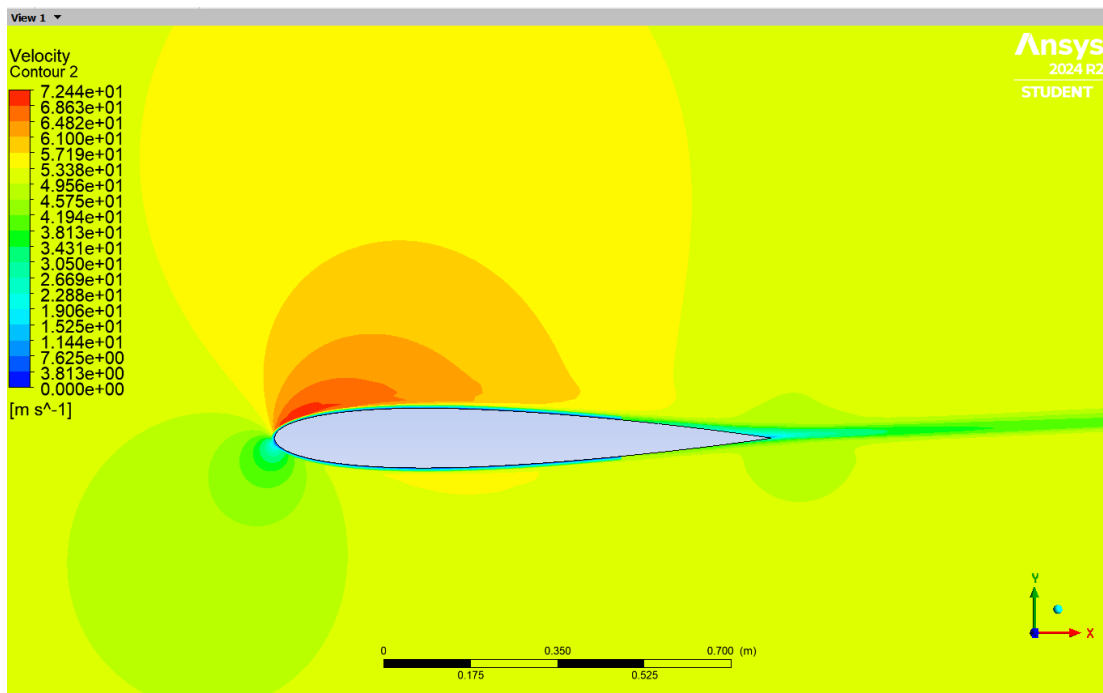

Figure 39 Pressure contour when AOA 4 degrees in research paper

Figure 40 Pressure contour when AOA 6 degrees in simulation

Figure 41 Pressure contour when AOA 6 degrees in research paper

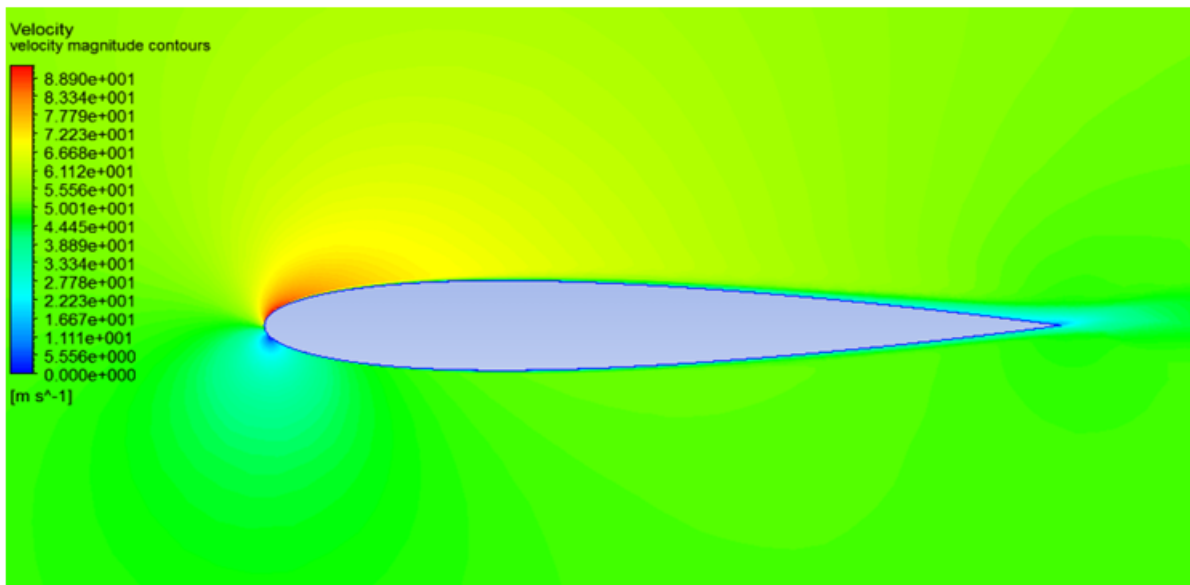Figure 42 Pressure contour when AOA 8 degrees in simulation

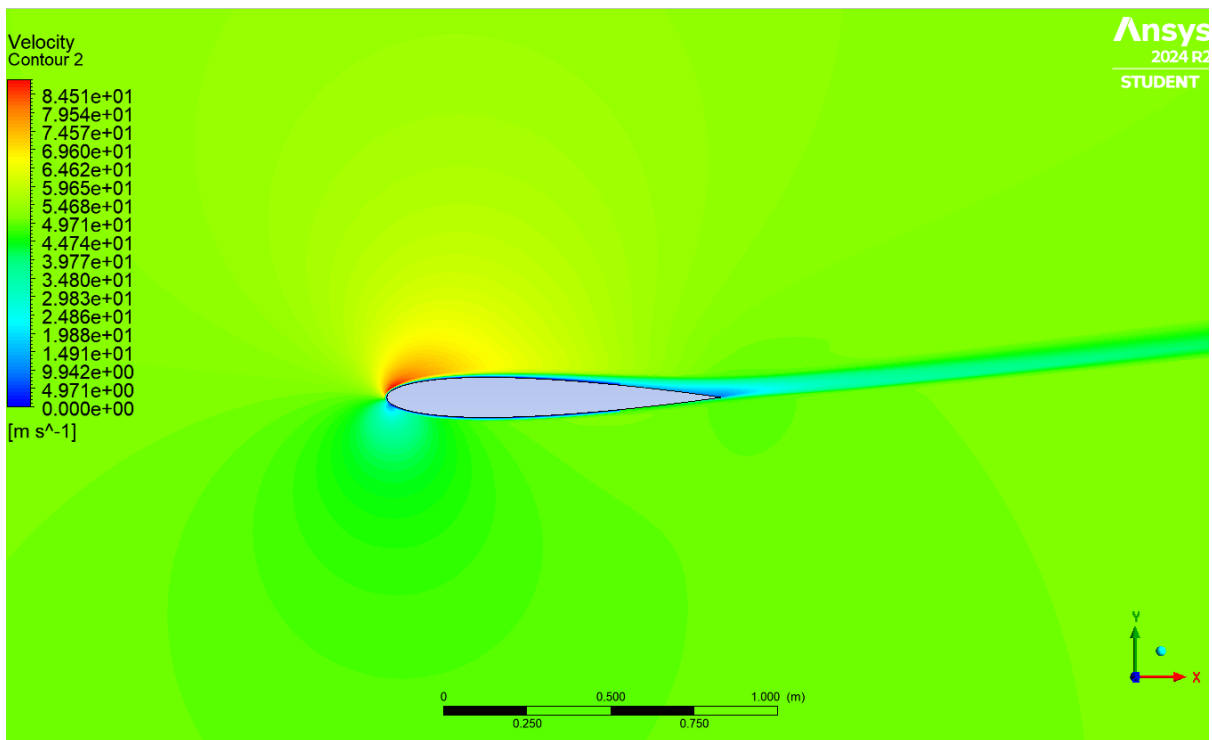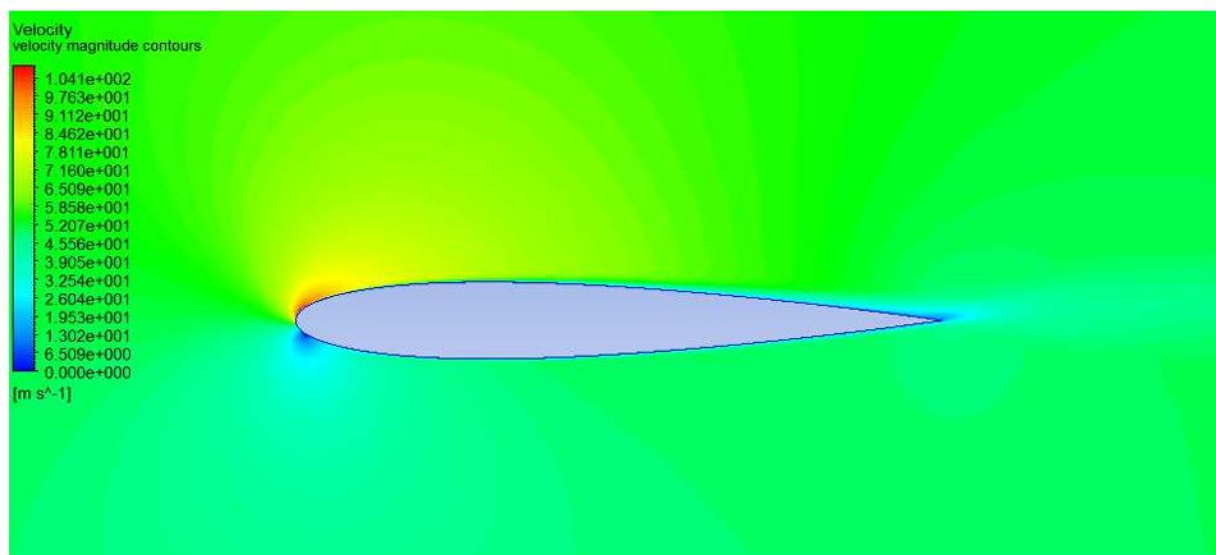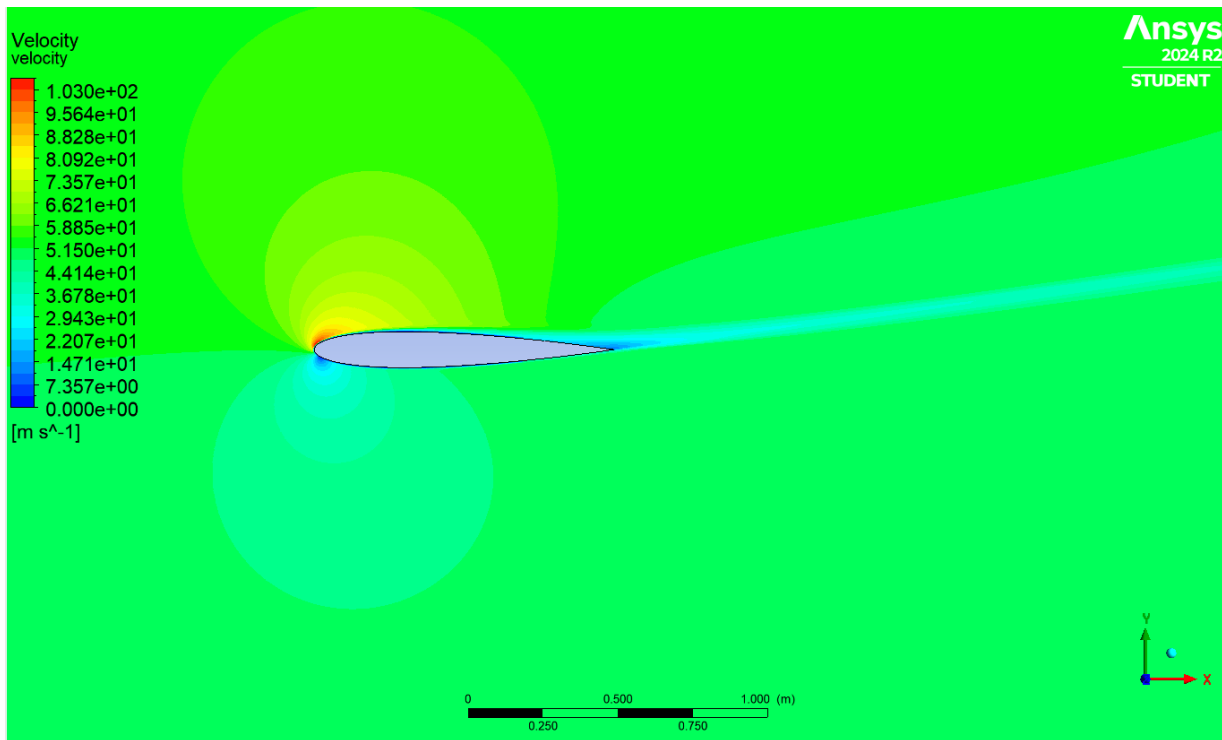Figure 43 Pressure contour when AOA 8 degrees in research paper

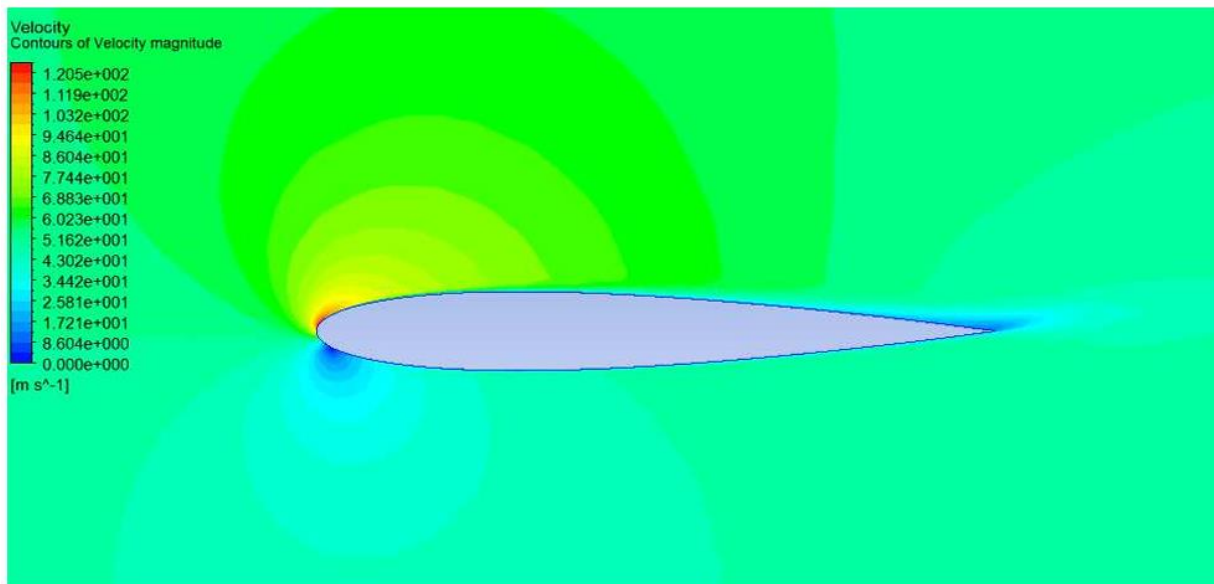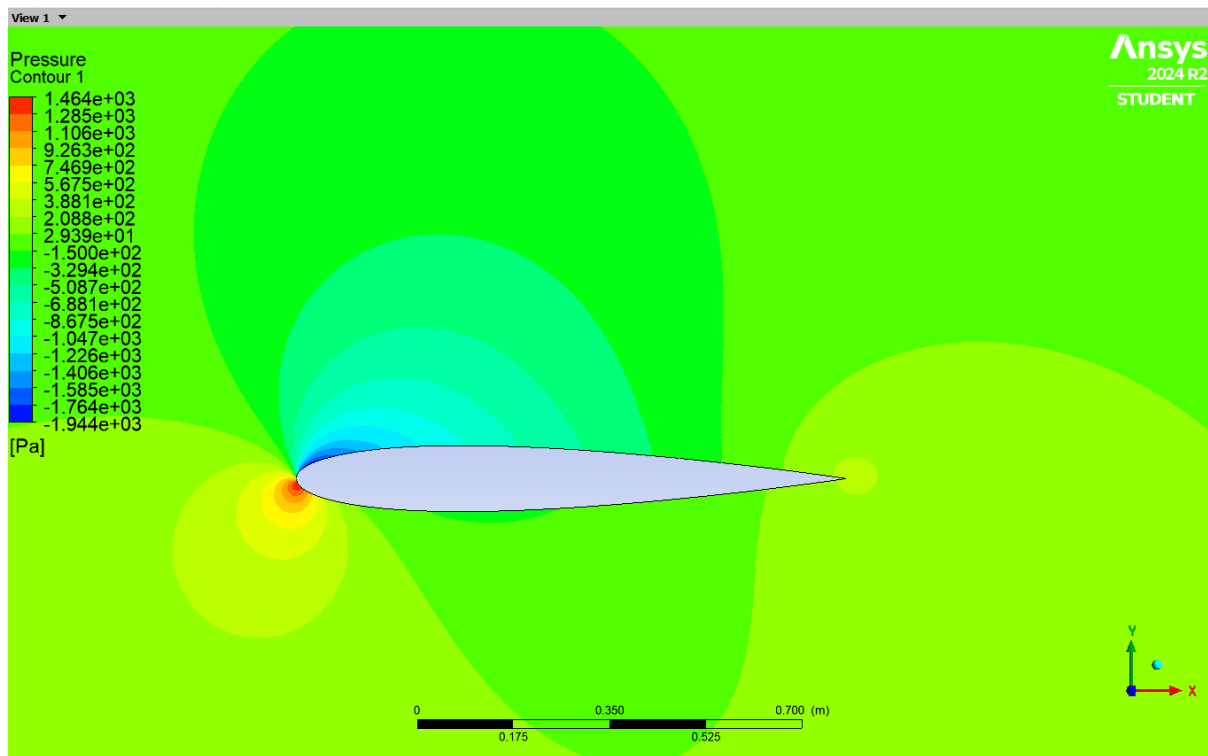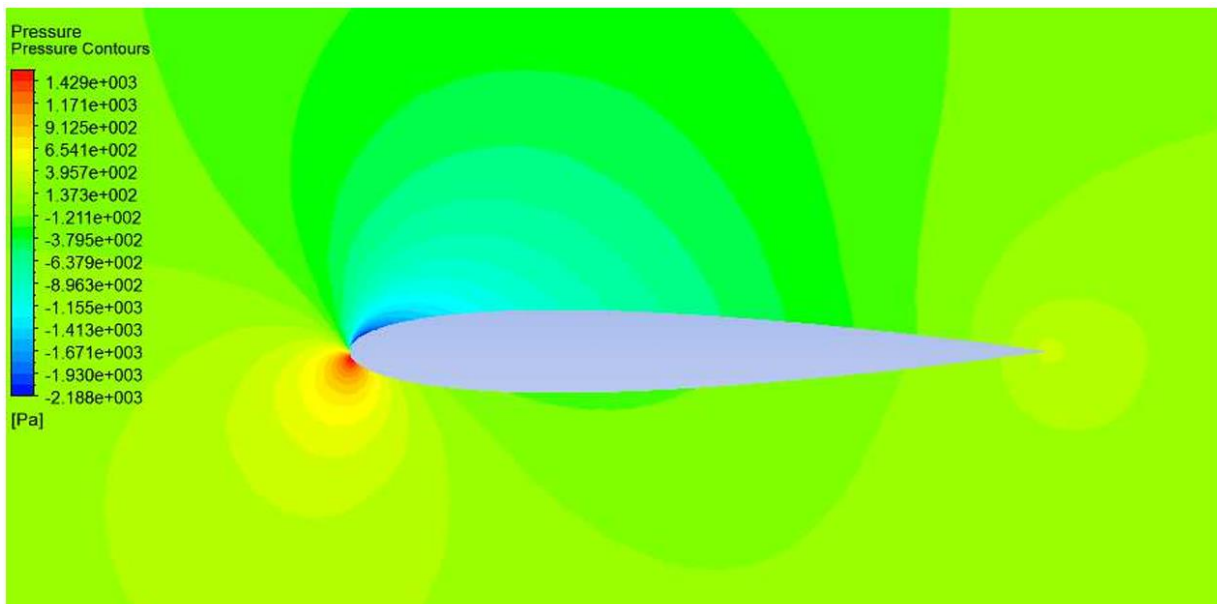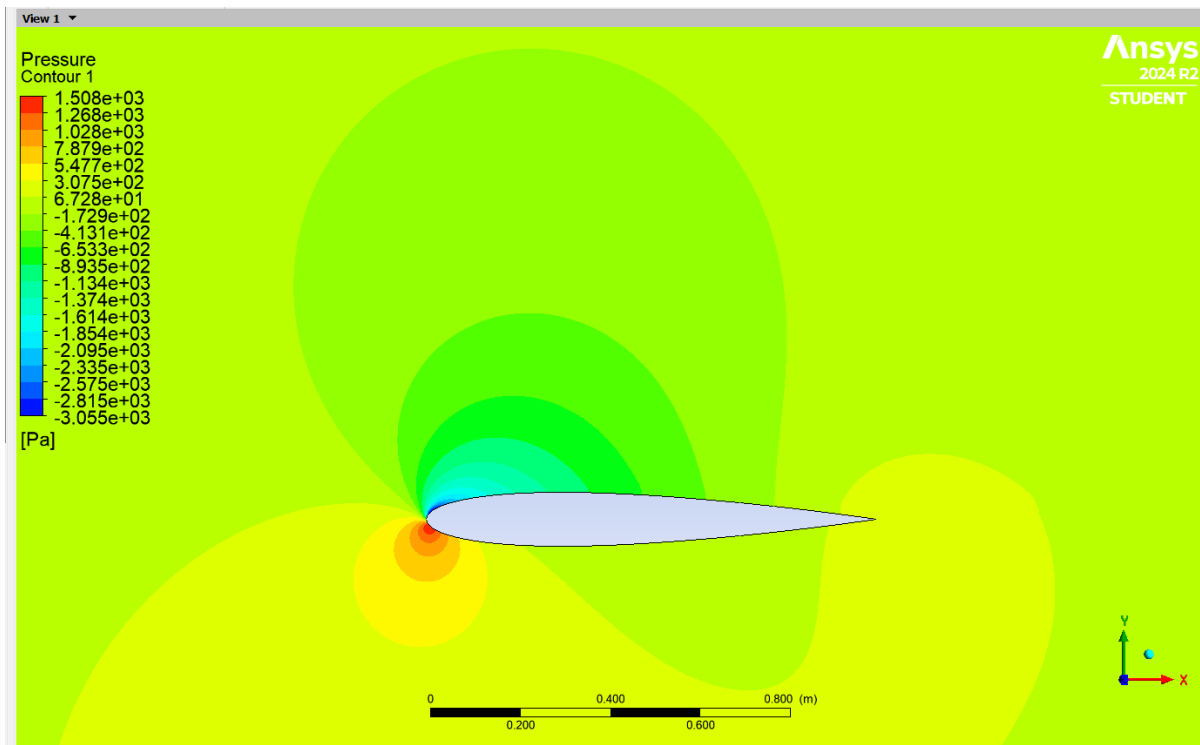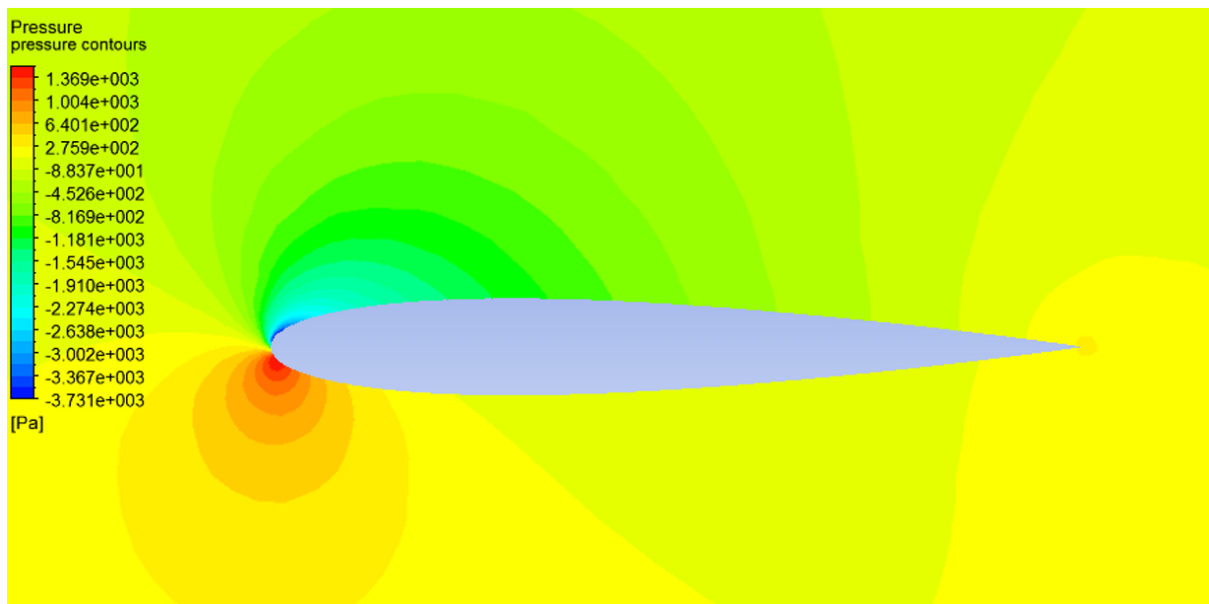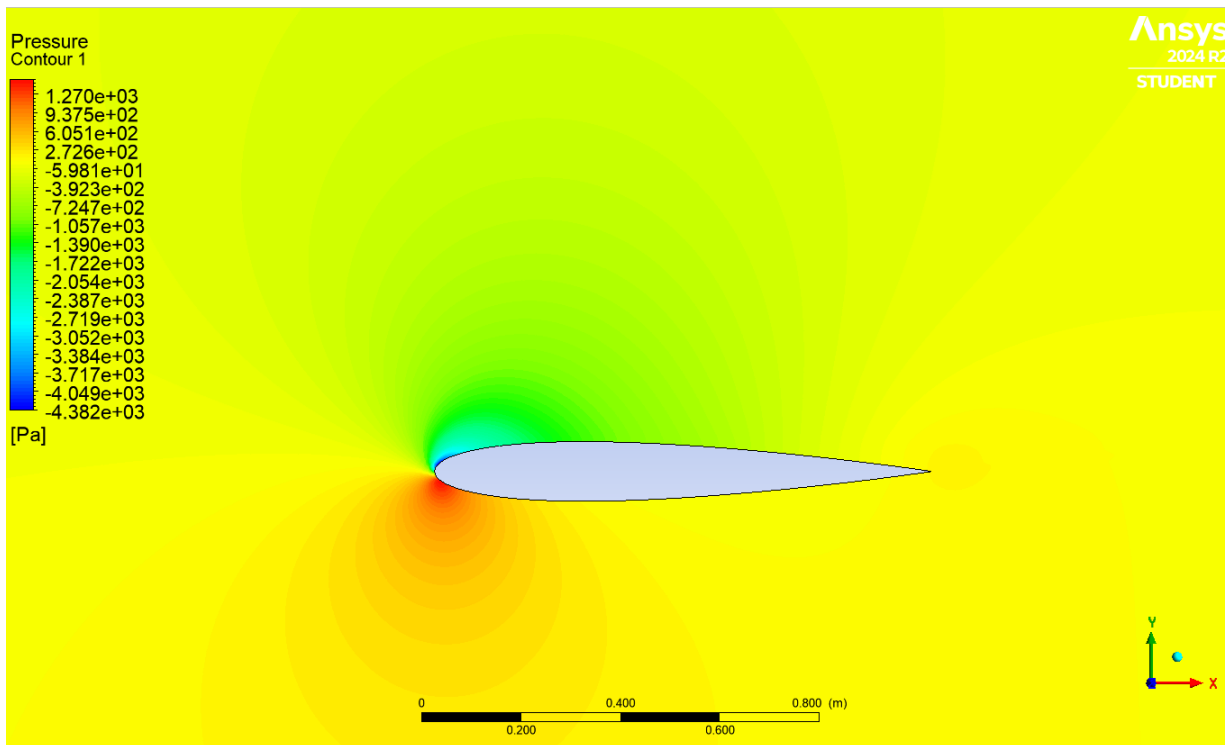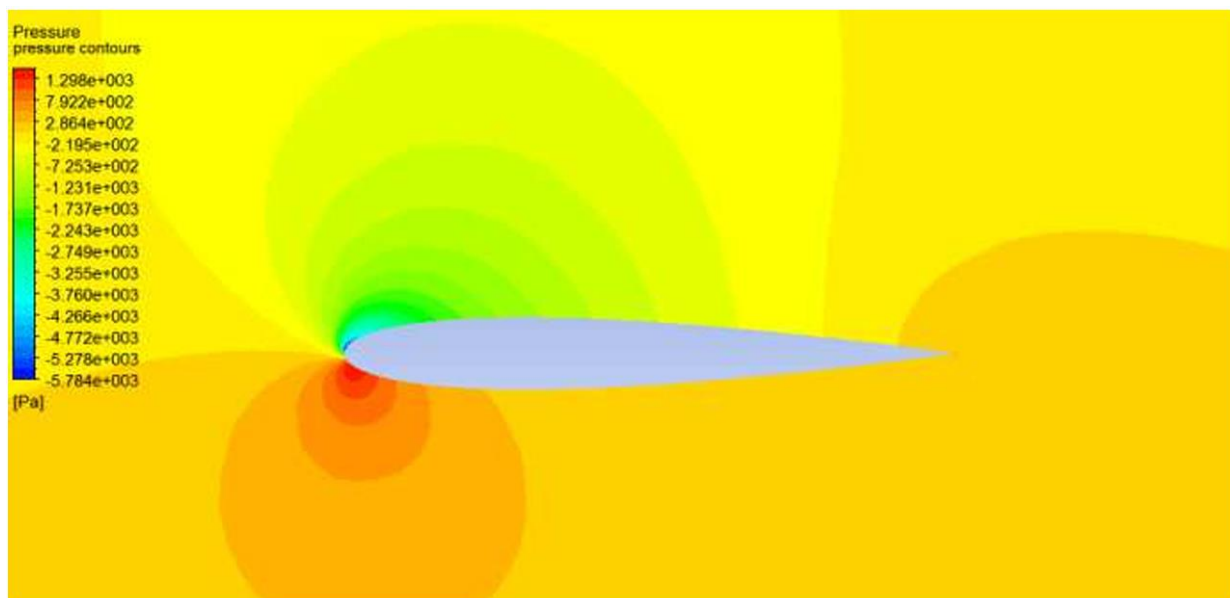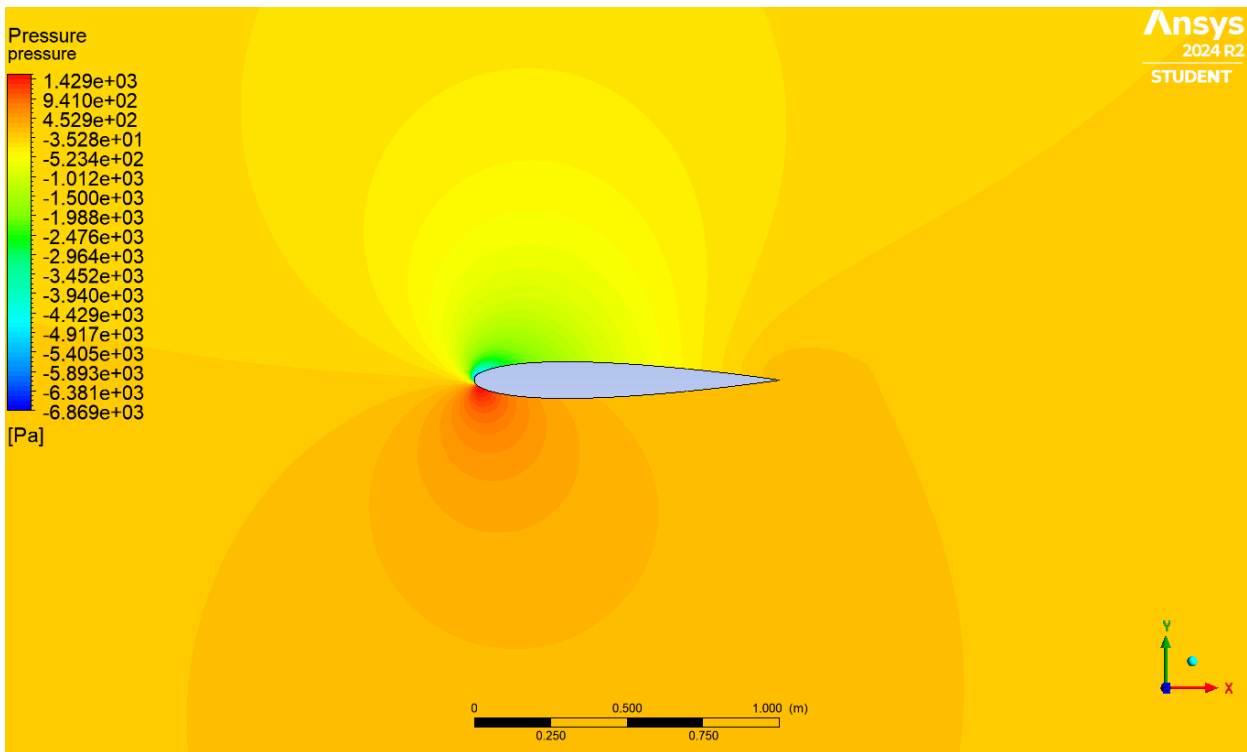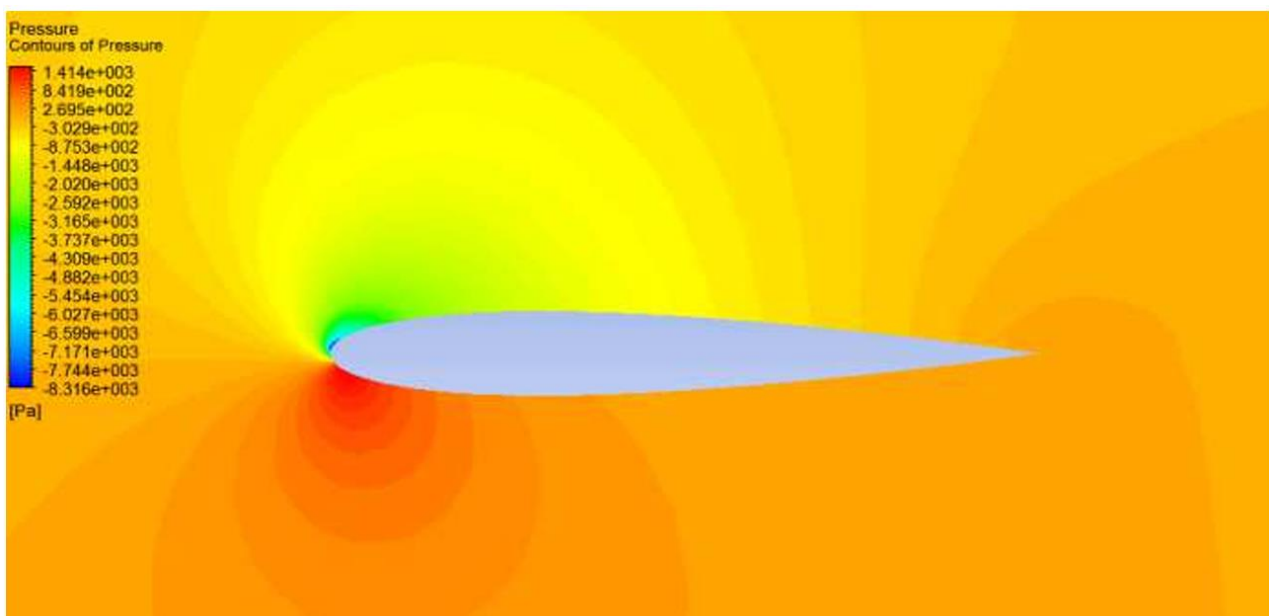Figure 44 Pressure contour when AOA 10 degrees in simulation

Figure 45 Pressure contour when AOA 10 degrees in research paper

6   Discussion

6.1   Accuracy of the results

The accuracy of CFD results is evaluated by contrasting the simulation data with research experimental data. The high accuracy of the simulations using the Realisable k−ϵ turbulence model is indicated by the close agreement between these data sets. Because it affects the prediction of flow separation and reattachment, two critical elements in figuring out the aerodynamic properties of airfoils, this model selection is significant. The resolution of the mesh surrounding the airfoil, particularly in close proximity to crucial areas like the leading and trailing edges where flow separation may occur, may have an additional impact on accuracy.

6.2   Possible errors in the simulation and what can be done to reduce the errors

CFD simulation errors may come from different sources,

- Mesh Quality and Density: In areas with significant gradient changes, such as those close to the airfoil surfaces, a low mesh density may lead to an insufficient resolution of the flow field.

- Turbulence Modelling: The simulation results can be impacted by the turbulence model selection, particularly for flows with separation and high angles of attack.

Strategies to Reduce Errors,

- Mesh refinement: By better capturing the flow dynamics, increasing the mesh density around the airfoil, especially in the boundary layer, can improve simulation accuracy. Also increase the size of the flow domain and increase the number of elements in the mesh.

- Advanced Turbulence Models: Predictive errors may be decreased by using hybrid or more advanced turbulence models that are better able to represent the flow physics at various regimes.

6.3 Difficulties faced

The analysis was conducted with a number of difficulties. The ANSYS Student Version's restriction on the number of elements that could be included in the mesh was one of the main challenges. This limitation had a major effect on the simulations' accuracy and refinement, especially when a higher mesh density was needed for in-depth analysis. Additionally, some critical information necessary for accurate modeling was not provided in the research paper being referenced. It was challenging to completely validate the results against accepted findings because of the gaps in the analysis and uncertainties brought about by the lack of this data.

6.4 Things learned from the simulation results

- Effect of Angle of Attack: The findings make it evident how changing the angle of attack impacts the lift and drag forces, focusing on the crucial equilibrium required for the best possible airfoil performance.

- Performance Predictions: Before physical prototypes are tested, CFD offers a reliable platform for estimating and improving airfoil designs, potentially saving time and money in the development of wind turbines.

- Effects of Turbulence: Knowing how various turbulence models affect the outcomes aids in choosing the best model depending on the particular needs of the flow procedure under consideration.

These insights may be crucial in creating wind turbine airfoils that are more effective, which could improve performance and lessen mechanical strain on the blades.

7   Suggestions for Future Improvement

Studying Computational Fluid Dynamics (CFD) has been helpful thanks to this module. It has offered a strong basis for choosing suitable solver parameters, highlighting the significance of understanding the fundamental ideas rather than carelessly depending on the software's default settings. The abilities we have gained from this module are extremely valuable and will surely help us in work environments, enabling us to succeed in our careers. It would be advantageous to include sample problems, like previous paper questions, in the lectures as an enhancement.

8   References

[1]
Airfoil Tools, "NACA 4 digit airfoil generator (NACA 2412 AIRFOIL)," *Airfoiltools.com*, 2019. http://airfoiltools.com/airfoil/naca4digit

 [2]
"How Do I Compute Lift and Drag?," *COMSOL*, 2015. https://www.comsol.com/blogs/how-do-i-compute-lift-and-drag

 [3]
A. Dash, "CFD Analysis ofWind Turbine Airfoil at Various Angles of Attack," *IOSR Journal of Mechanical and Civil Engineering*, vol. 13, no. 04, pp. 18–24, Apr. 2016, doi: https://doi.org/10.9790/1684-1304021824.