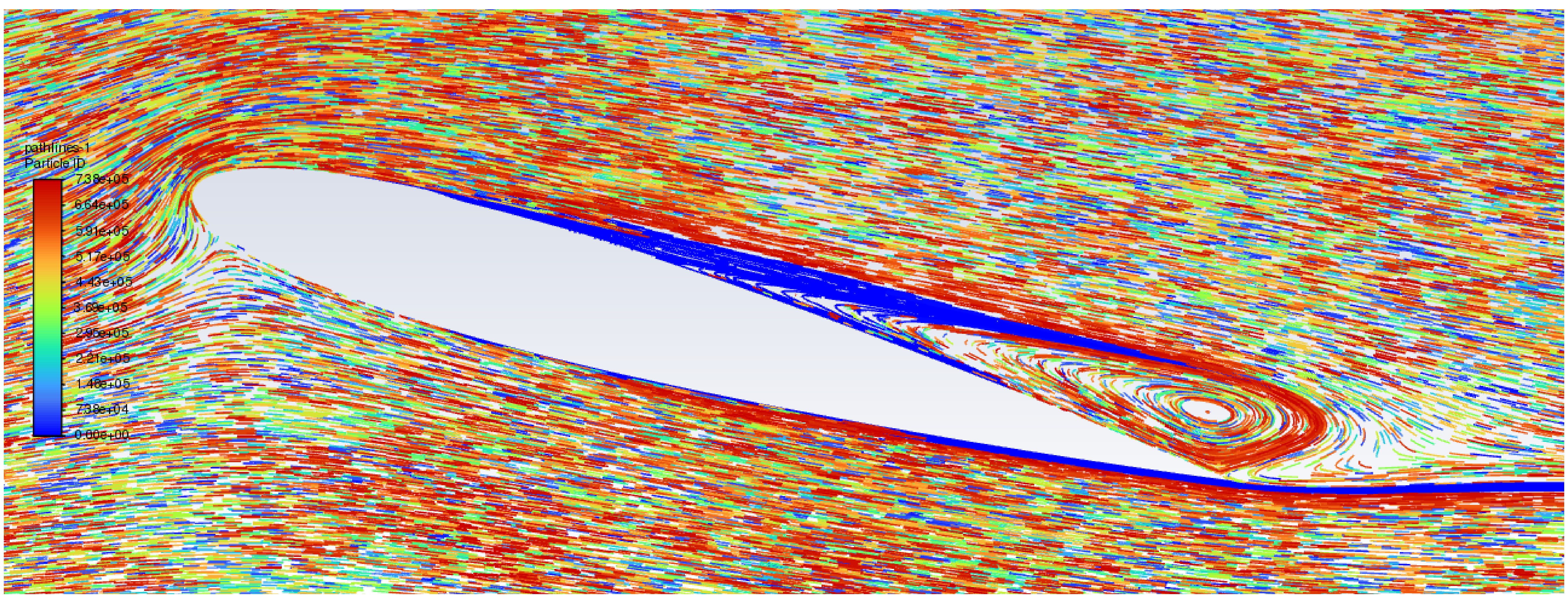
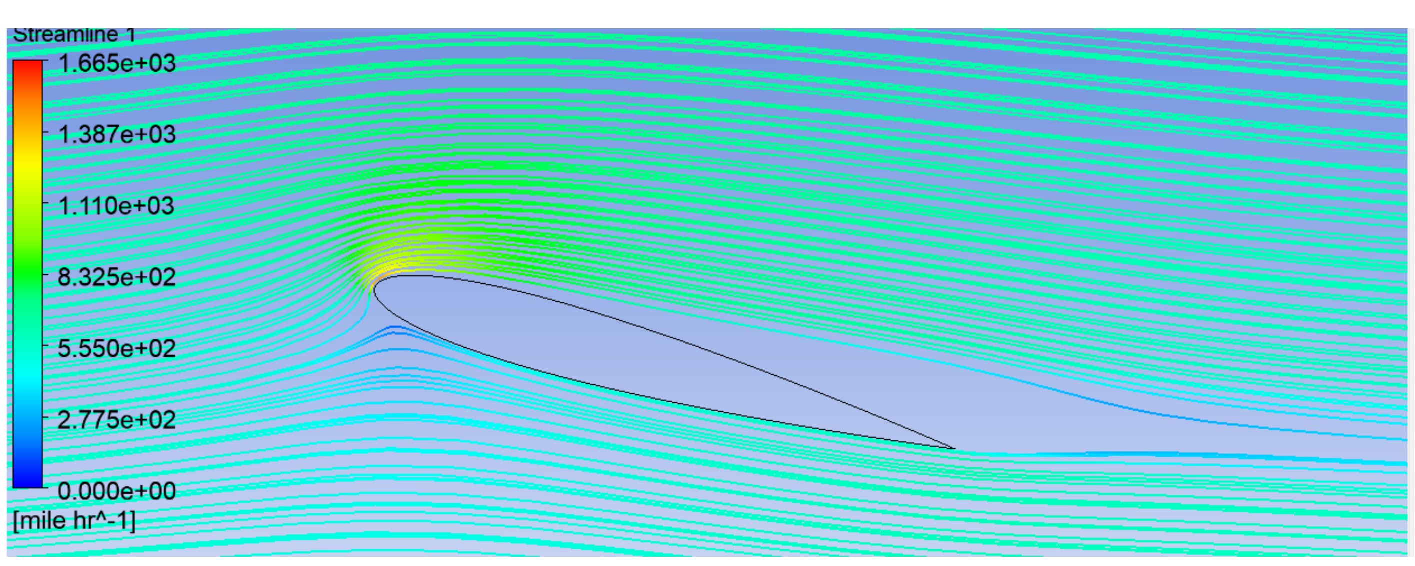
CASE STUDY

Aerodynamic Optimization of NACA 0012 Airfoil

CFD analysis with ANSYS Fluent





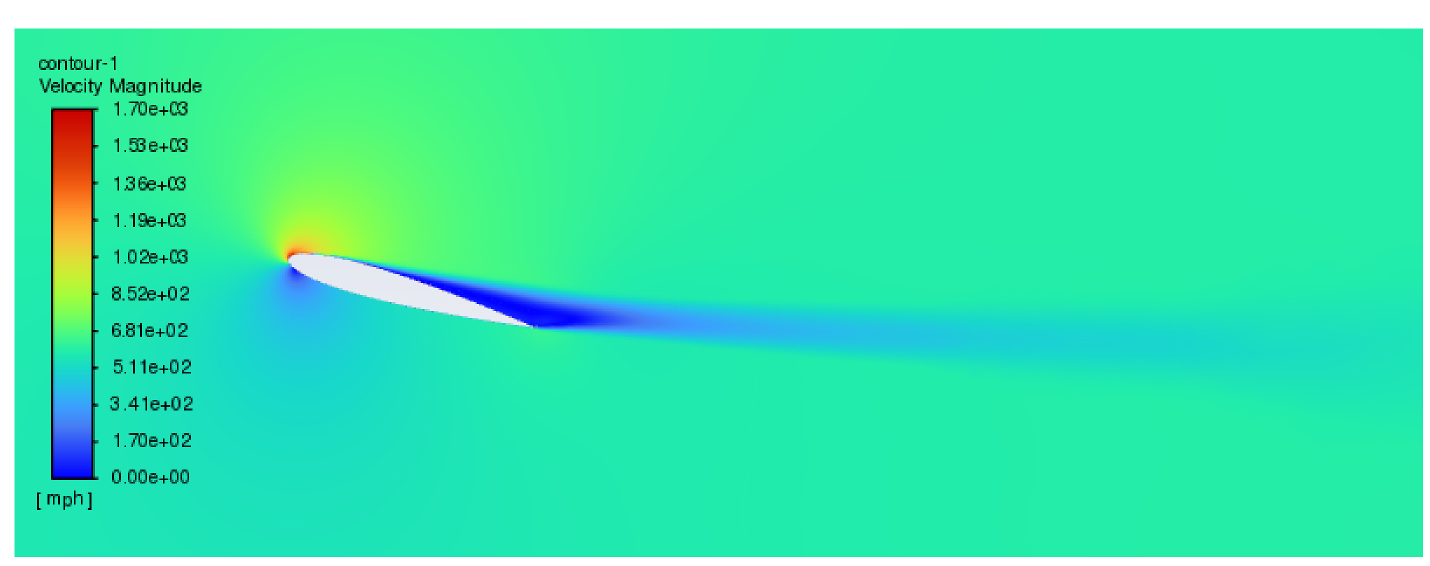


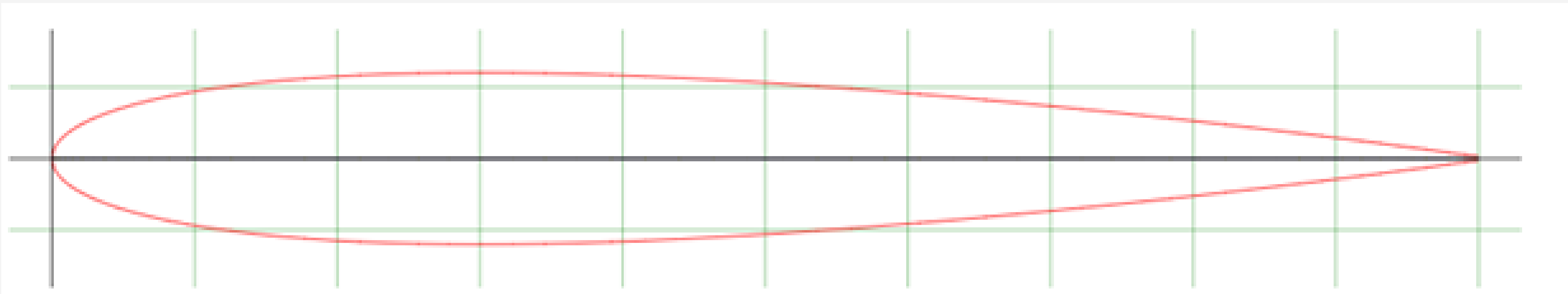
Table of contents

Table of contents 2
Abstract 2
Problem definition 3
Geometry preparation 3
Mesh generation 4
Simulation setup 4
Results 5
Conclusion 12

Abstract

This report presents a numerical analysis of the airflow around the NACA 0012 airfoil using ANSYS Fluent software. The purpose of the study is to investigate the aerodynamic characteristics of the airfoil, including lift and drag, under various flow conditions and at the stall angle where significant changes in lift and drag characteristics are expected.

Problem definition

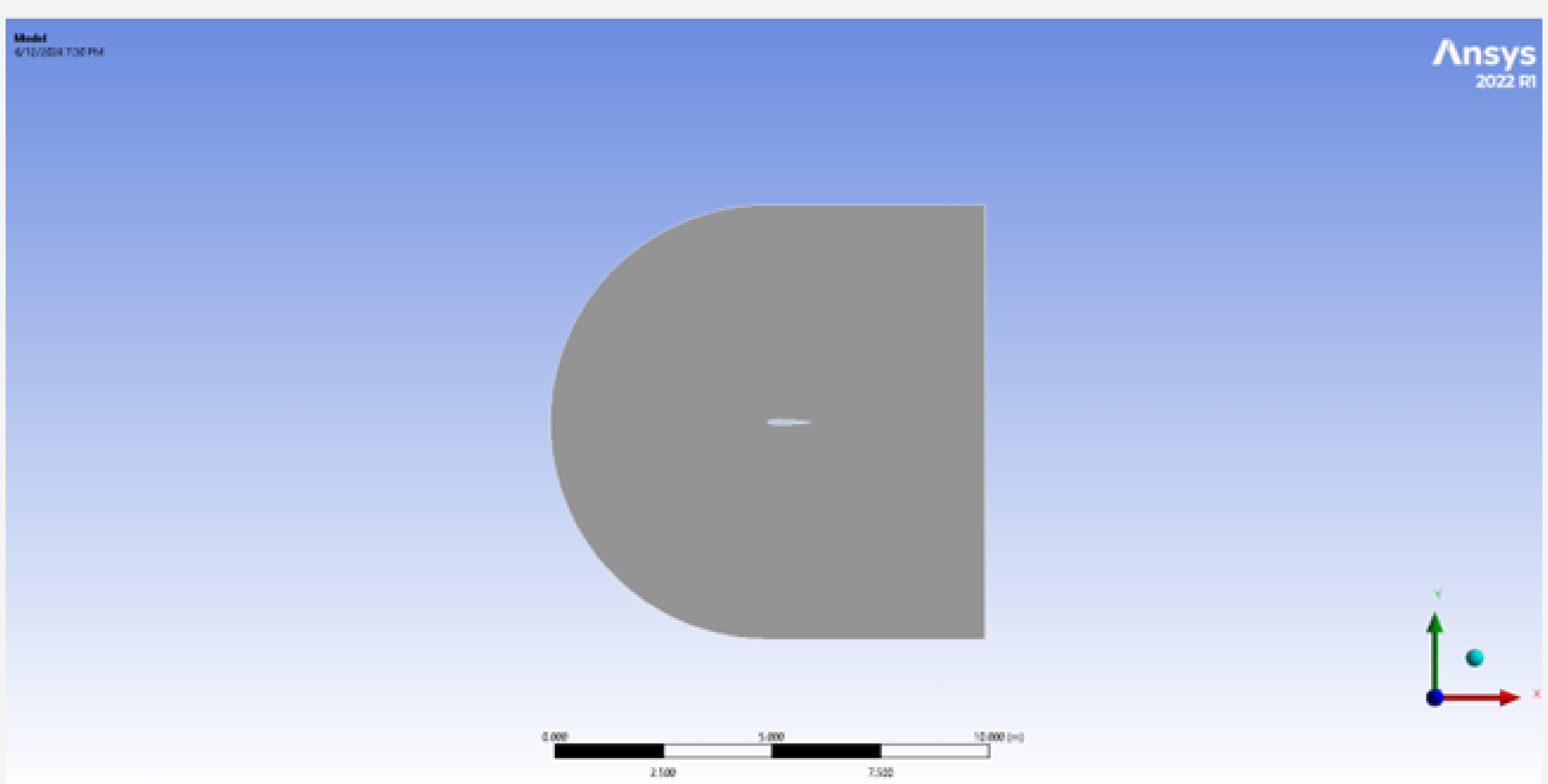


NACA 0012 profile (Figure 1)

This study aims to visualize the speed profile; determine the drag coefficient, and the lift coefficient of NACA 0012 at different angles of attack, and find the stall angle.

Geometry preparation

The NACA 0012 airfoil is a symmetrical airfoil that is widely used in aerodynamic studies and applications. The simulation of the 2D flow around the NACA 0012 airfoil was carried out based on the chord **c = 1 m** using **ANSYS Fluent** software.

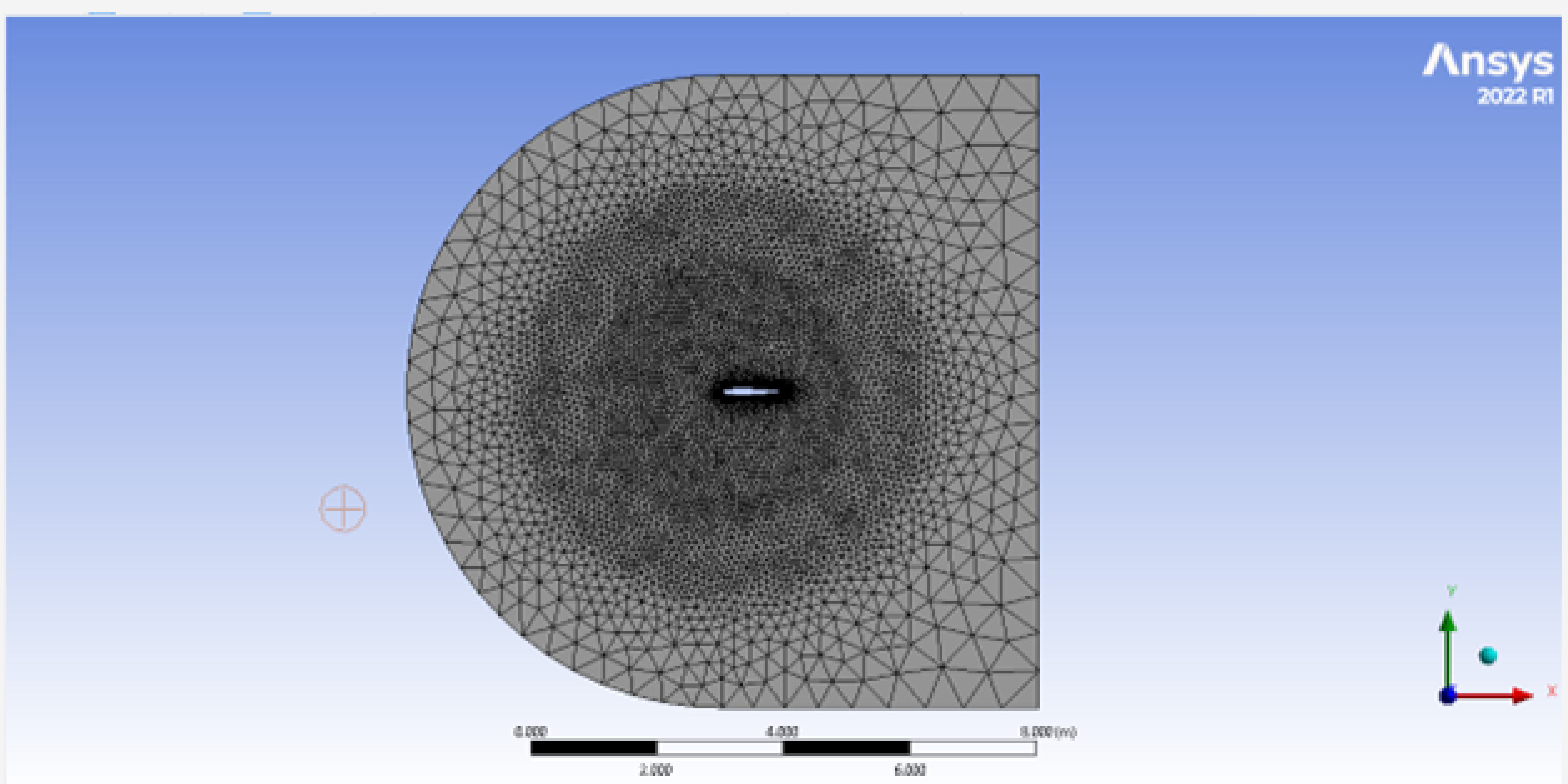


Flow domain (Figure 2)

Mesh generation

Mesh optimization and refinement may be employed to enhance the quality of the mesh and the accuracy of the simulation.

Mesh quality was evaluated based on criteria such as **aspect ratio**, **skewness**, and **orthogonality**. The generated mesh showed satisfactory quality within acceptable limits for accurate simulation.



Mesh (Figure 4)

Simulation setup

Boundary conditions: The inlet boundary condition was set to a uniform free stream velocity of 600 mph. No-slip wall boundary conditions were applied to the airfoil surface, and a far-field boundary condition was specified at the outlet.

Solver Settings: The simulations were performed using a steady-state solver with a turbulence model (k-omega model) appropriate for aerodynamic flows.

The solver is configured to run up to 10,000 iterations, automatically stopping calculation once the results converge.

Results

We carried out the simulations from different angles of attack (AOA).

The results obtained are presented in the tables below in comparison with the experimental data (Ladson experiment).

CODE	C_l $(AoA = 0^o)$	C_l $(AoA = 5^o)$	C_l $(AoA = 10^o)$	C_l $(AoA = 15^o)$
Experiment Ladson	-0.00724	0.536	1.057	1.49
SimFlow	~ 0	0.544	1.060	1.51
CFL3D (NASA)	~ 0	N/A	1.078	1.51
FUN3D (NASA)	~ 0	N/A	1.084	1.51
NTS (NASA)	~ 0	N/A	1.077	1.51

Lift coefficient Cl comparison between CFD codes and experimental data (Table 2)
Source: [2D NACA 0012 Airfoil SSTm Model Results]

CODE	C_l $(AoA = 0^o)$	C_l $(AoA = 5^o)$	C_l $(AoA = 10^o)$	C_l $(AoA = 15^o)$
Experiment Ladson	-0.00724	0.536	1.057	1.49
SimFlow	~ 0	0.544	1.060	1.51
CFL3D (NASA)	~ 0	N/A	1.078	1.51
FUN3D (NASA)	~ 0	N/A	1.084	1.51
NTS (NASA)	~ 0	N/A	1.077	1.51

Drag coefficient CD comparison between CFD codes and experimental data (Table 3)
Source: [2D NACA 0012 Airfoil SSTm Model Results]

Results (02)

CODE	Cl(AoA=0°)	Cl(AoA=5°)	Cl(AoA=10°)	Cl(AoA=15°)	Cl(AoA=20°)
Experiment Ladson	0.00724	0.536	1.057	1.49	N/A
CFD (ANSYS FLUENT)	~ 00	0.526	1.038	1.47	0.988

Lift coefficient Cl comparison between CFD (ANSYS) and experimental data (Table 4)

CODE	CD(AoA=0°)	CD(AoA=5°)	CD(AoA=10°)	CD(AoA=15°)	CD(AoA=20°)
Experiment Ladson	0.00809	0.00852	0.0119	0.0183	N/A
CFD (ANSYS FLUENT)	0.00813	0.00910	0.0180	0.0304	0.3282393

Drad coefficient Cd comparison between CFD (ANSYS) and experimental data (Table 5)

Results (03)

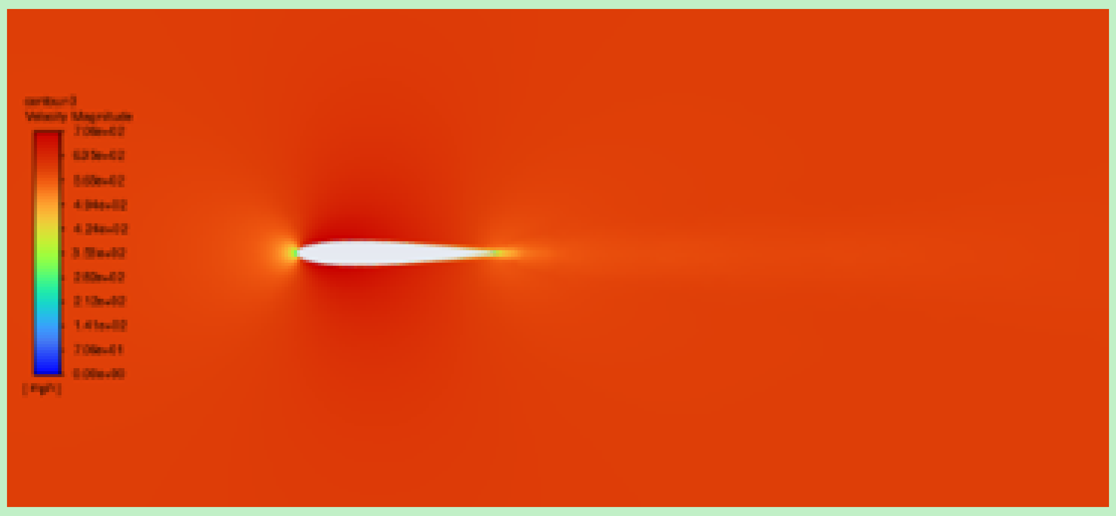
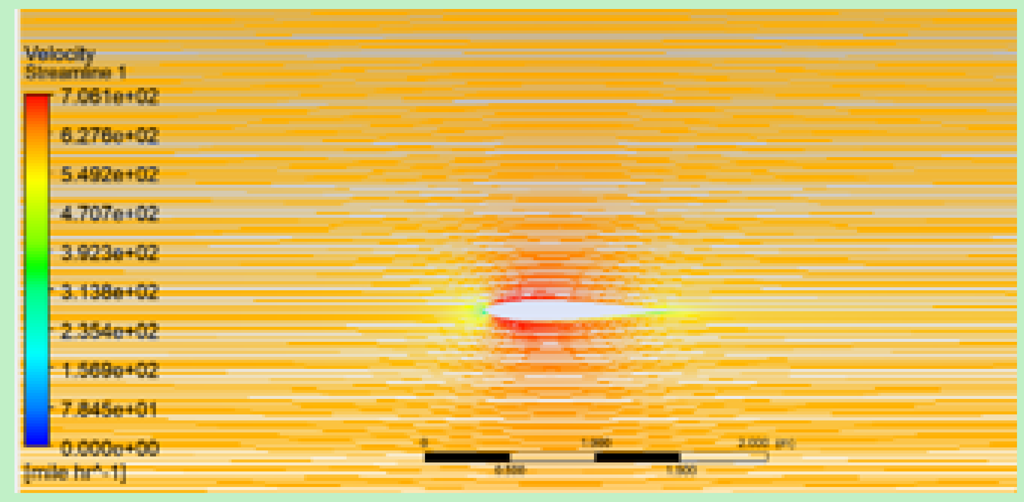
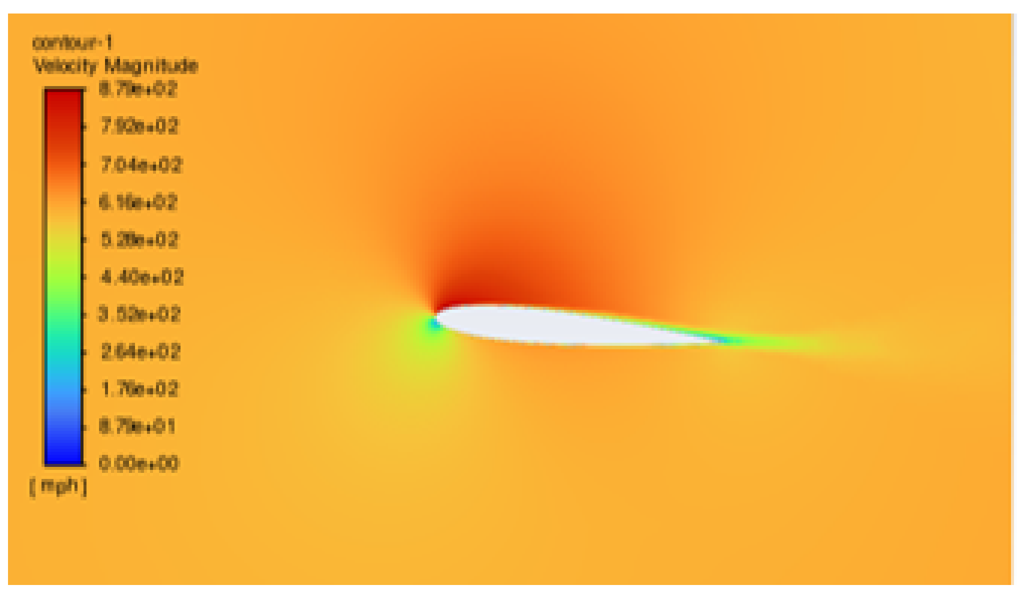
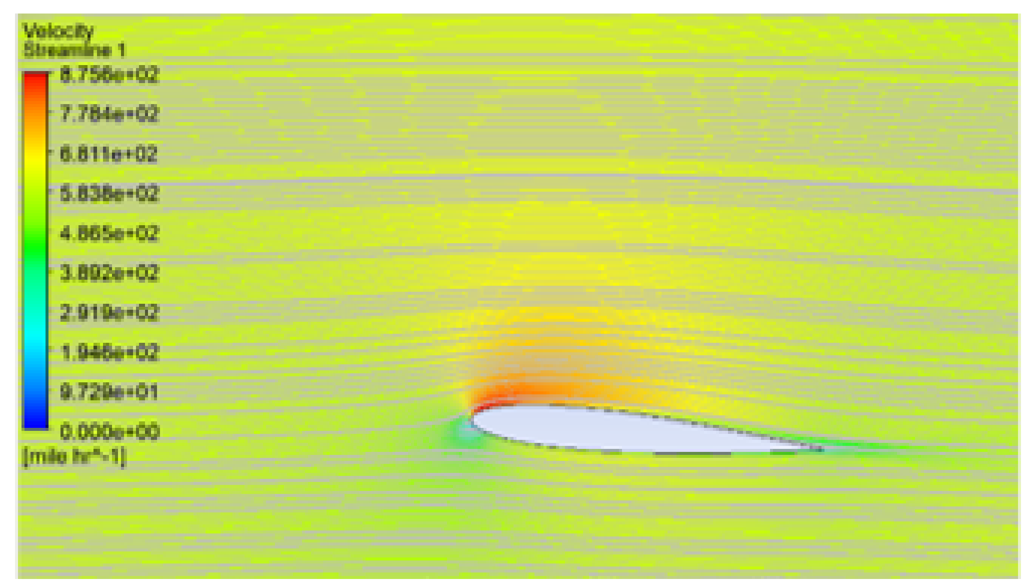
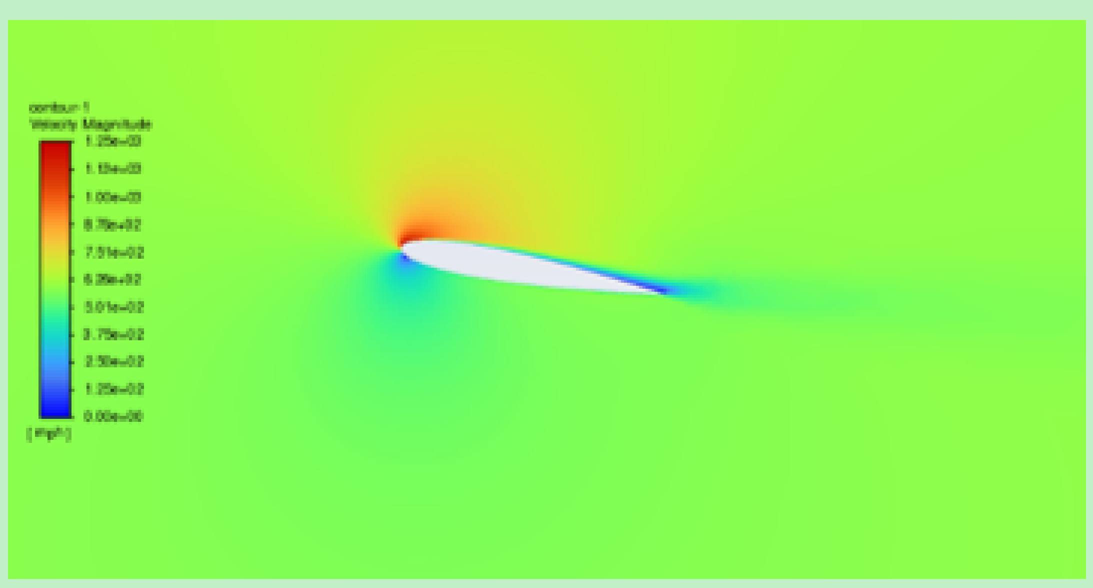
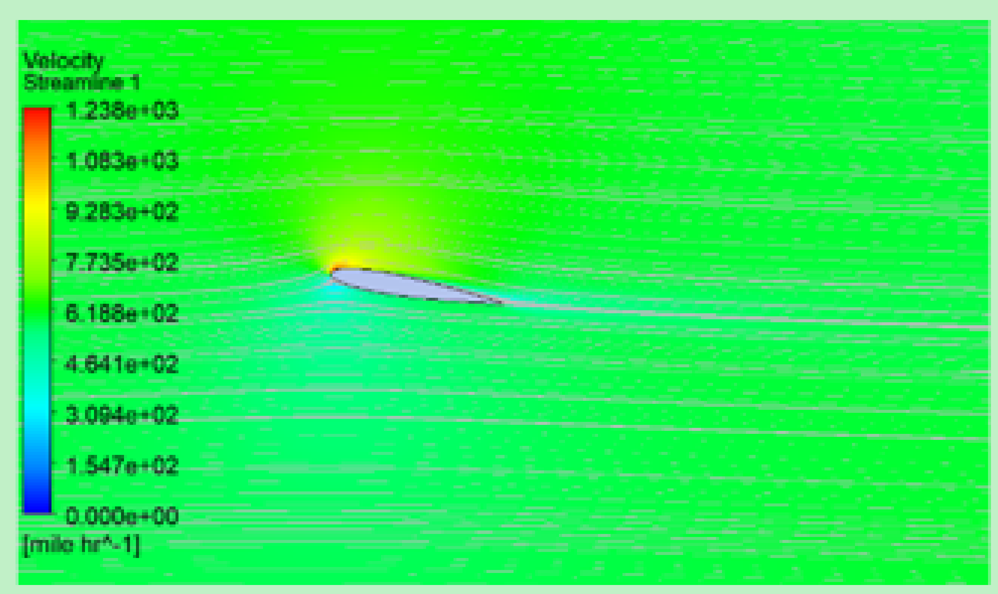
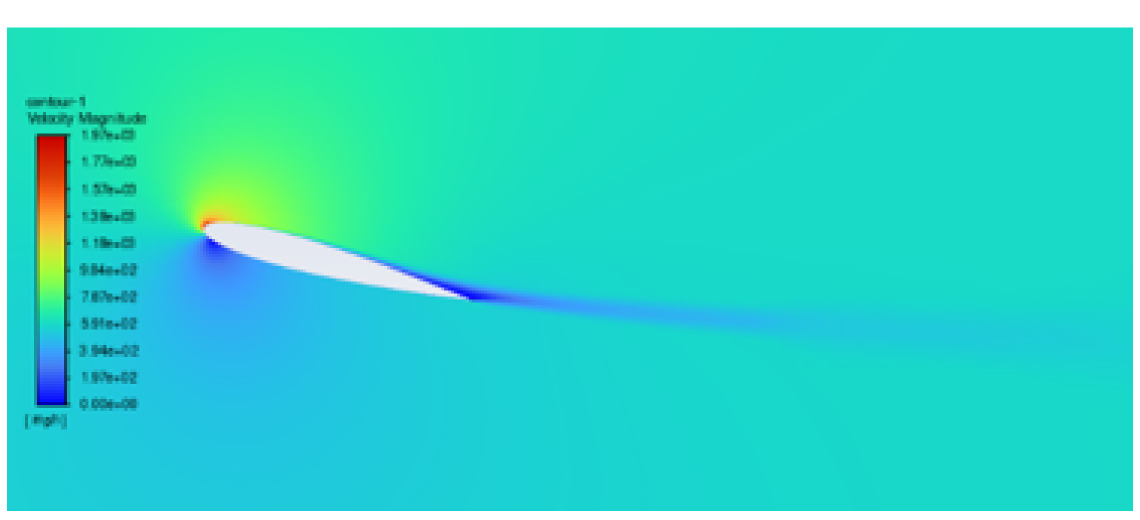
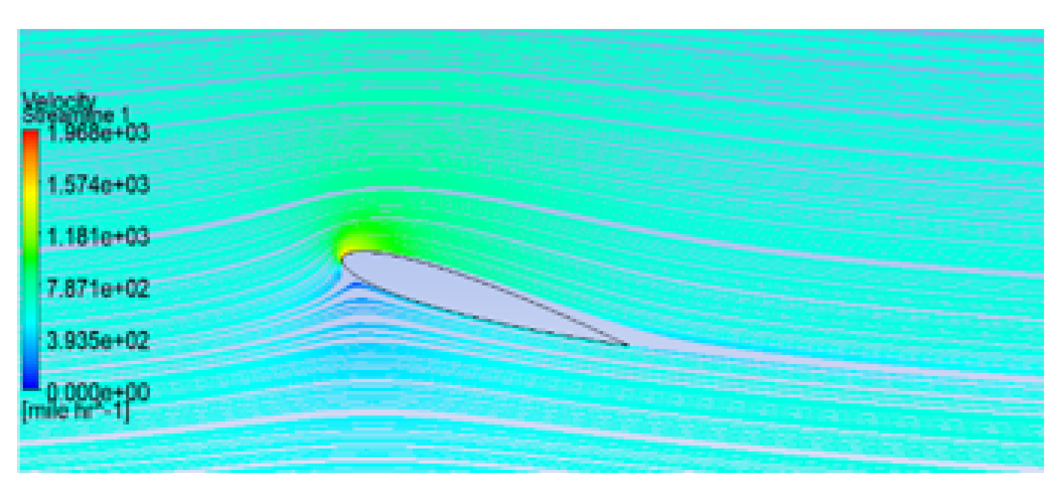
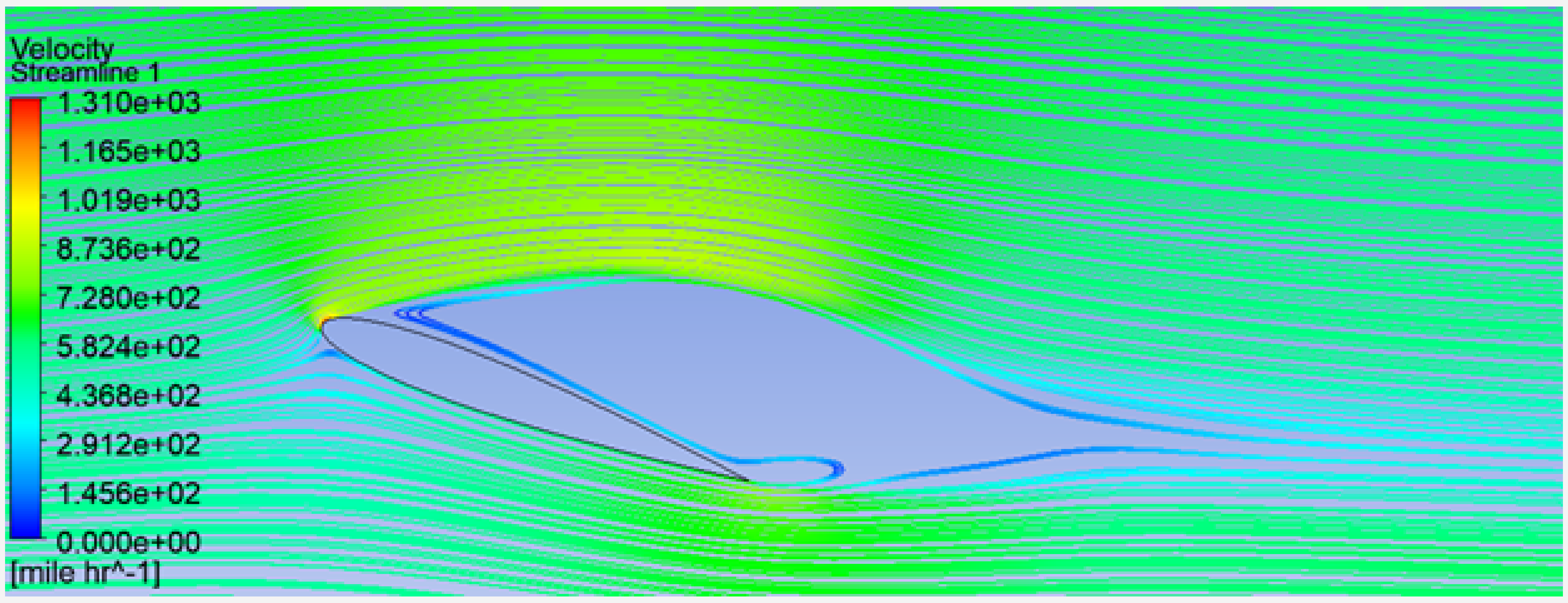
AOA(°)	Velocity contours	Velocity streamlines
0		
5		
10		
15		

Table 4 & 5

Results (04)

NACA 0012, AOA = 20°

At an angle of attack of 20 degrees, there is an increase in flow separation on the upper surface of the airfoil. This flow separation causes a significant increase in pressure and disrupts the pressure distribution compared to attached flow conditions.



Velocity streamline (AOA 20°) (Figure 5)

<u>**NB**</u>**:** At AOA = 20° we have total flow separation along the upper surface of the airfoil. This means that the **stall point has already been reached**. We run the simulation from AOA 12.5°, increasing it by 1° until we find the accurate stall angle.

Results are below :

AoA (°)	Cl	Cd
12.5	1.1478	0.0443
13.5	1.3673	0.0329
14.5	1.4329	0.0372
15.5	1.1749	0.0711

Lift and drag (Table 6)

Results (05)

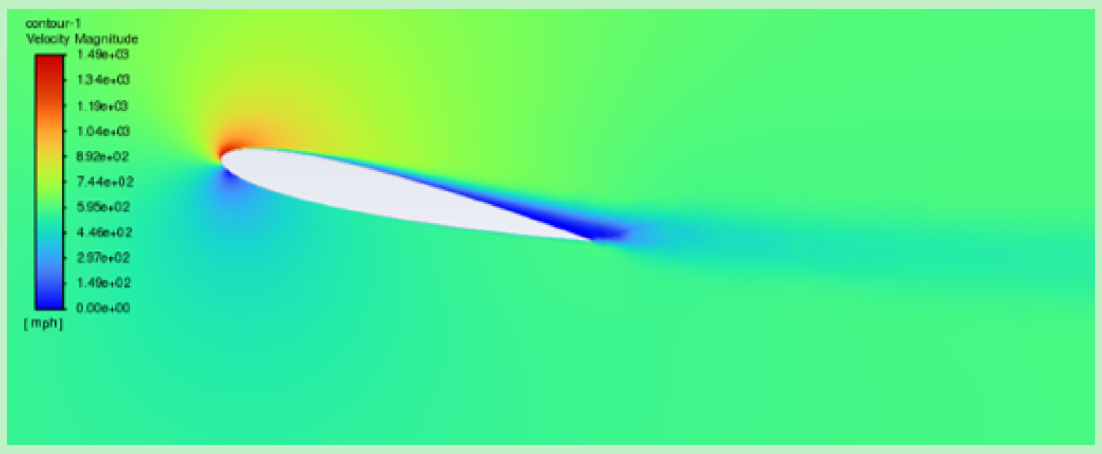
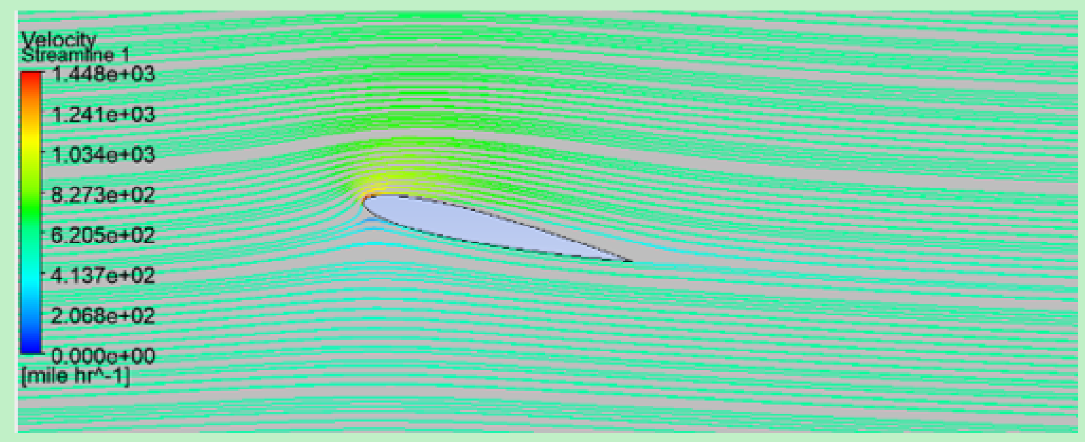
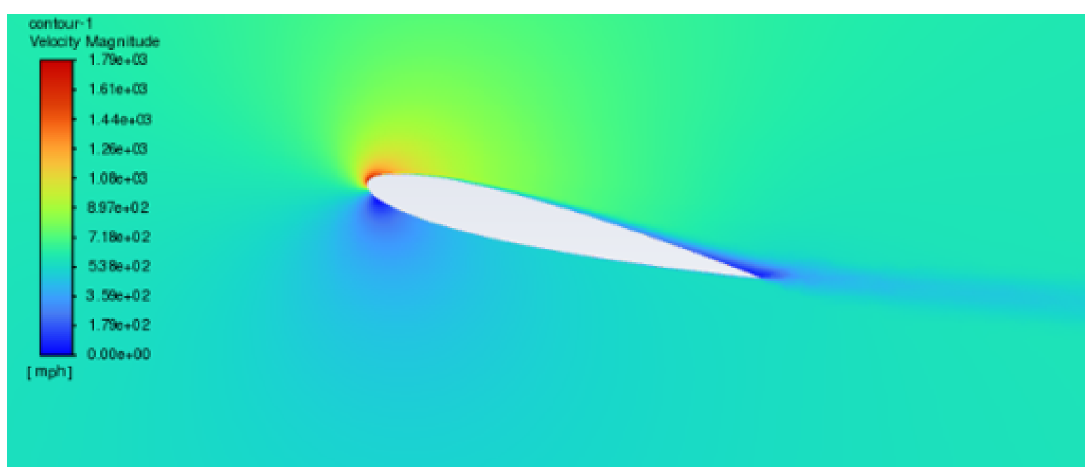
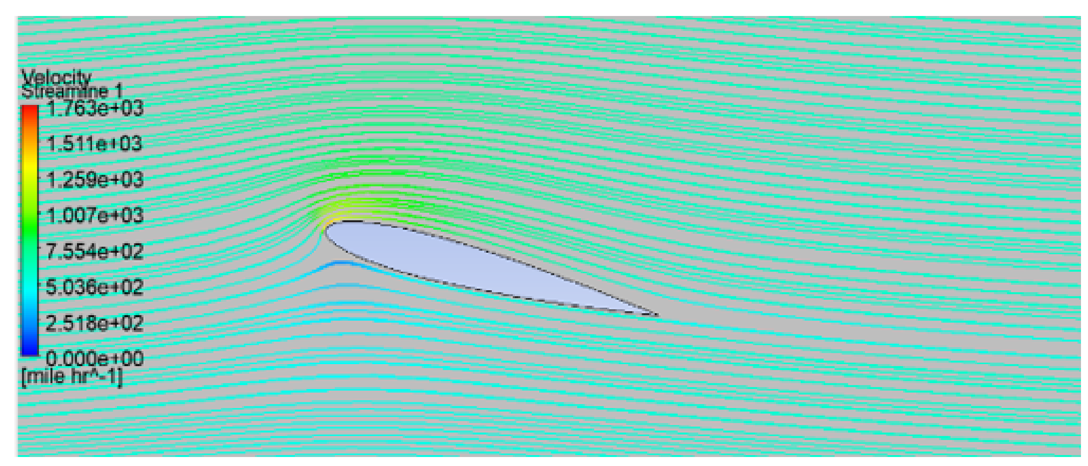
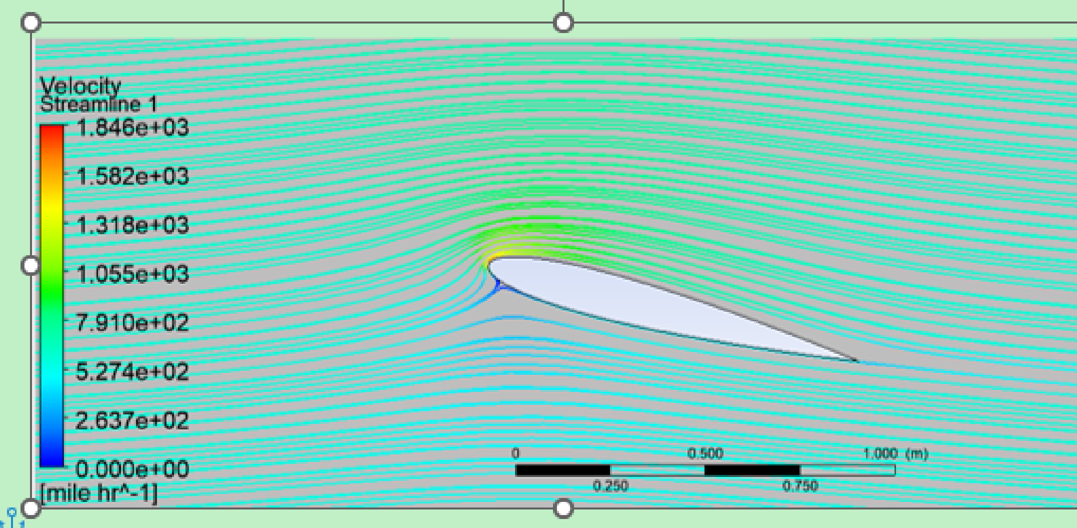
AOA (°)	Velocity contours	velocity streamlines
12.5		
13.5		
14.5		

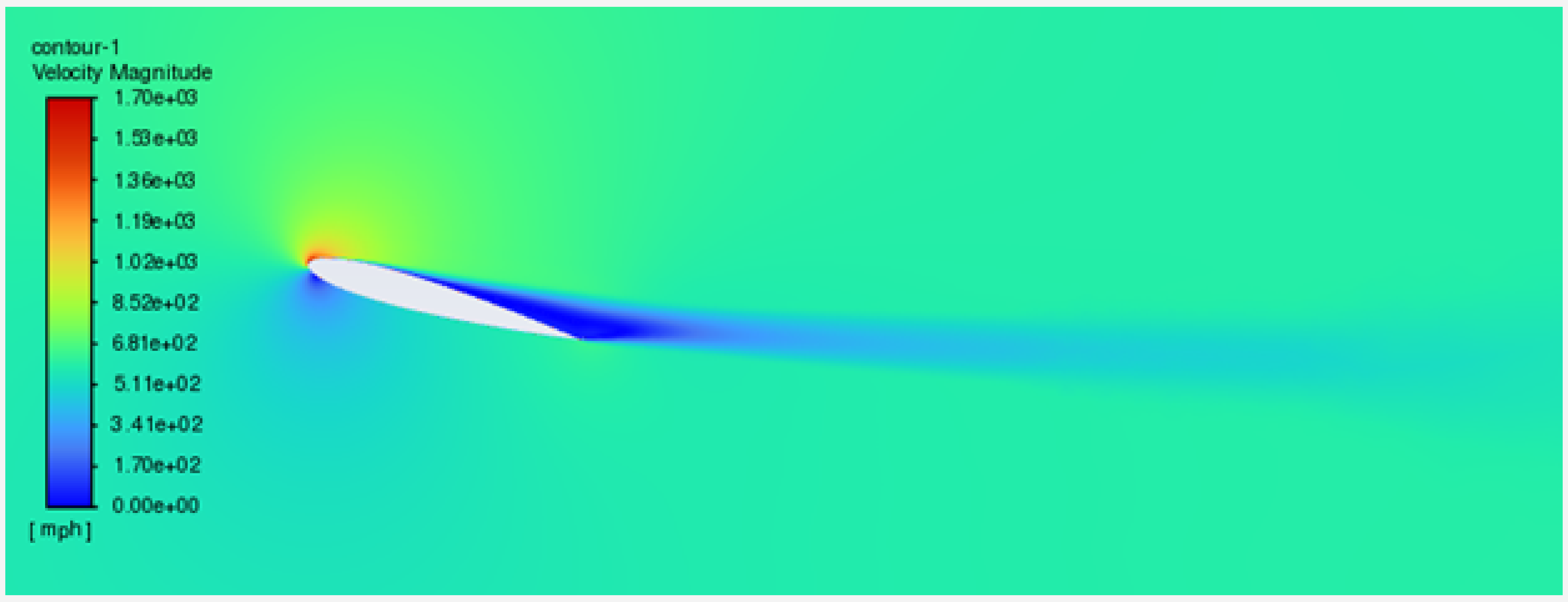
Table 6 (suite)

NACA 0012, AOA = 15.5°

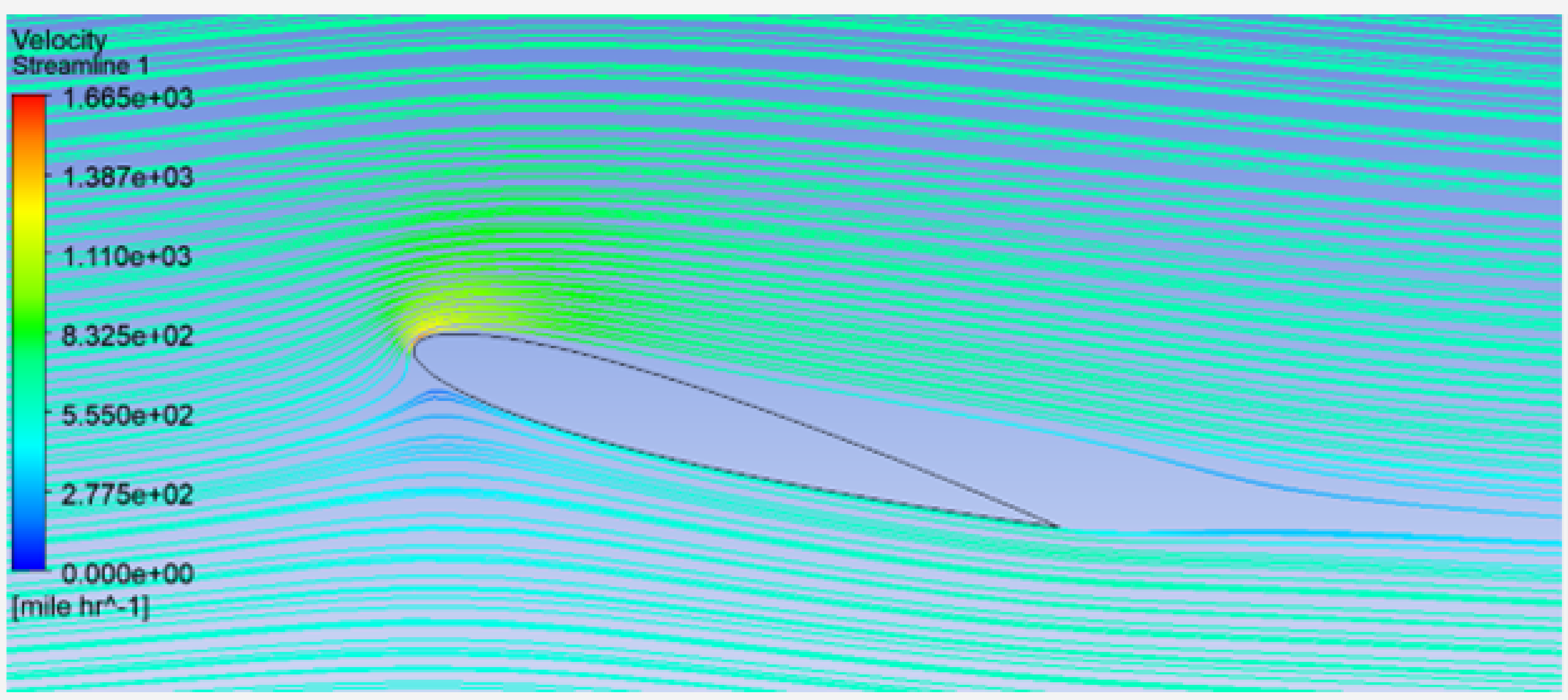
At an angle of attack of 15.5 degrees, the pressure contour over the surface of a NACA 0012 airfoil exhibits significant changes compared to lower angles of attack. The flow experiences boundary layer separation, especially near the trailing edge.

Velocity Contours: *(see next page)*

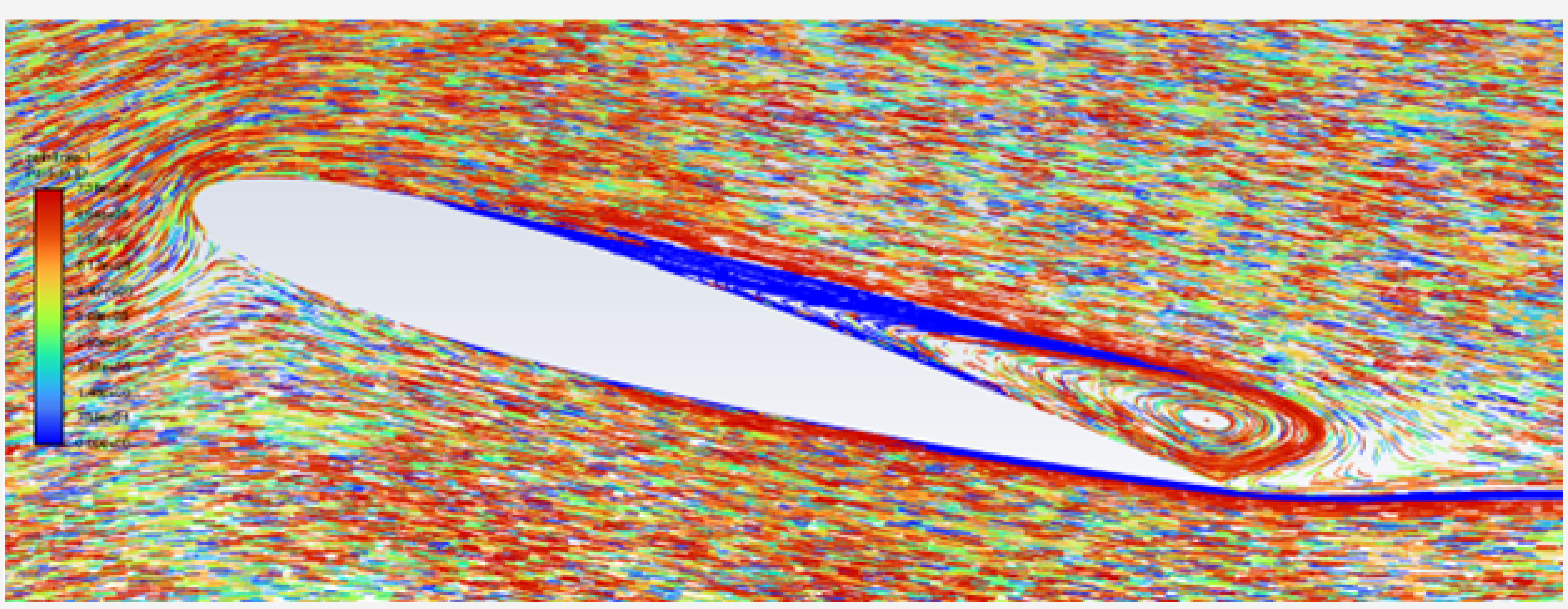
Results (06)



Velocity (AOA 15.5°) (Figure 6)



Velocity streamline (AOA 15.5°) (Figure 7)



Velocity streamline (AOA 15.5°) (Figure 8)

Results (07)

Observations

The simulation results from the above AOA show high velocity on the upper surface of the airfoil and low velocity on the upper surfaces, resulting in **high-pressure distribution on the lower faces** and **low pressure on the upper faces**.

The flow separation begins slightly at 15° AOA.

It appears at 15.5° AOA; a sharp decrease of the Cl, while the Cd increases and a flow separation where the boundary layer of air separates from the surface of the airfoil.

We can conclude that a stall point is reached at 15.5°.

AoA (°)	Cl	Cd
0	~ 00	0.00813
5	0.526	0.00910
10	1.037	0.01831
12.5	1.147	0.04438
13.5	1.367	0.03292
14.5	1.432	0.03725
15	1.472	0.03046
15.5	1.174	0.07111
20	0.987	0.32823

Table 7

Conclusion

Based on simulations here are the conclusions drawn about the NACA 0012 airfoil:

- **At low angles of attack** (0-10 degrees), the NACA 0012 airfoil typically exhibits smooth flow characteristics with attached flow over both the upper and lower surfaces.
- **As the angle of attack increases towards stall**, the NACA 0012 airfoil experiences flow separation, especially on the upper surface.
- **At post-stall**, the NACA 0012 airfoil exhibits reduced lift and increased drag due to turbulent flow and separation.

In conclusion, we have determined the stalling angle (15.5°) of NACA 0012 airfoil. It should be noted that the difference between experimental data of Ladson experiment and our results (ANSYS) are not larger than 2% for lift coefficient Cl and 5% for drag coefficient CD in most of the case.

Further Reading

- ***Introduction to CFD Simulation*** | *Practical Examples*
- ***Introduction to CAE*** | *Computer-Aided Engineering*

Let's talk about your next project! *Contact our team* now and claim a free quote.