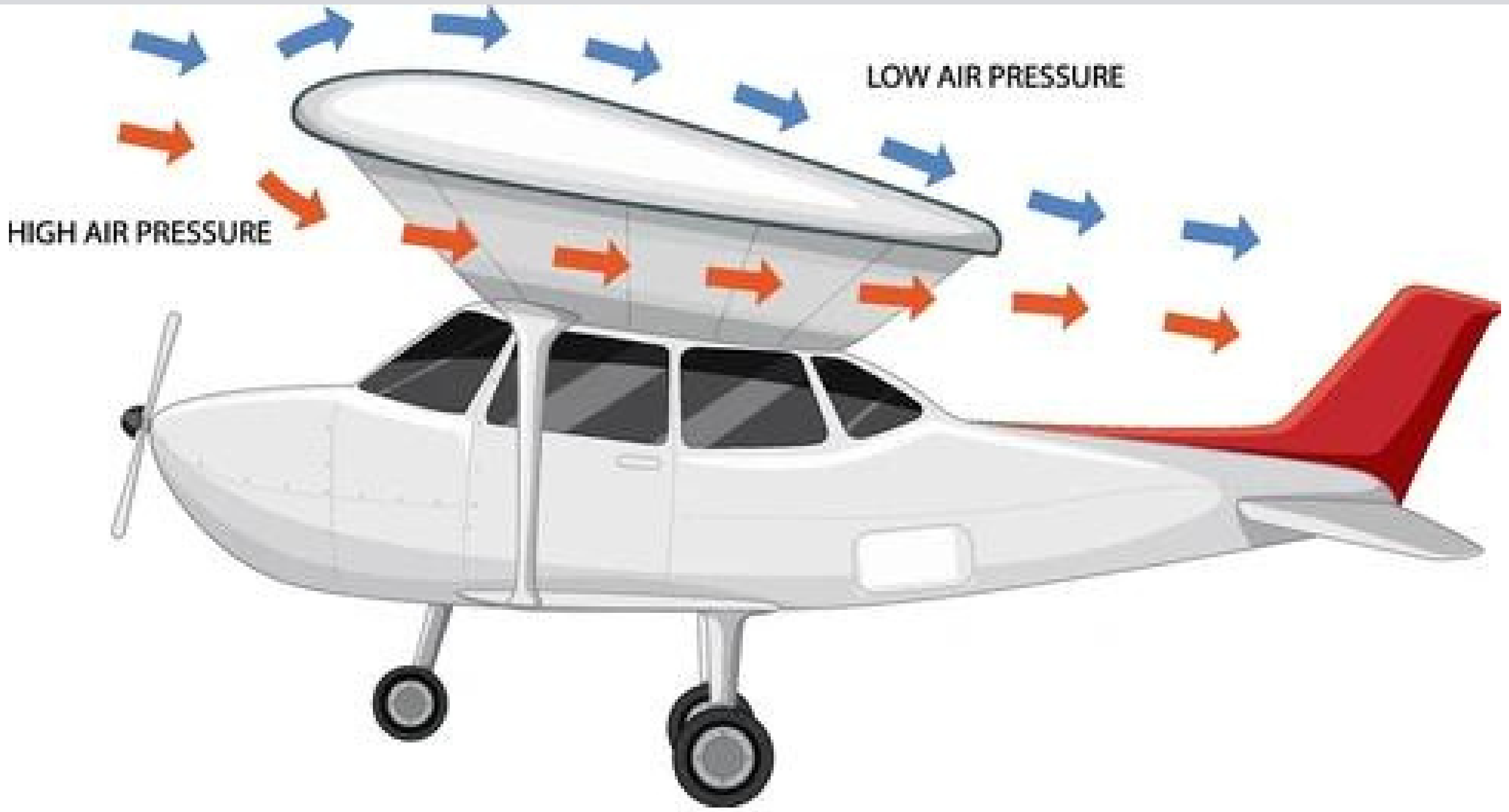
IIT Roorkee

IDEAZ

MECHANICAL ENGINEERING

COGNIZANCE



INTRODUCTION

NACA 4412

Configuration
Four-digit code denotes 4% camber, 40% back, and 12% thickness.

Symmetrical Design
Symmetrical shape balances lift for wings and control surfaces.

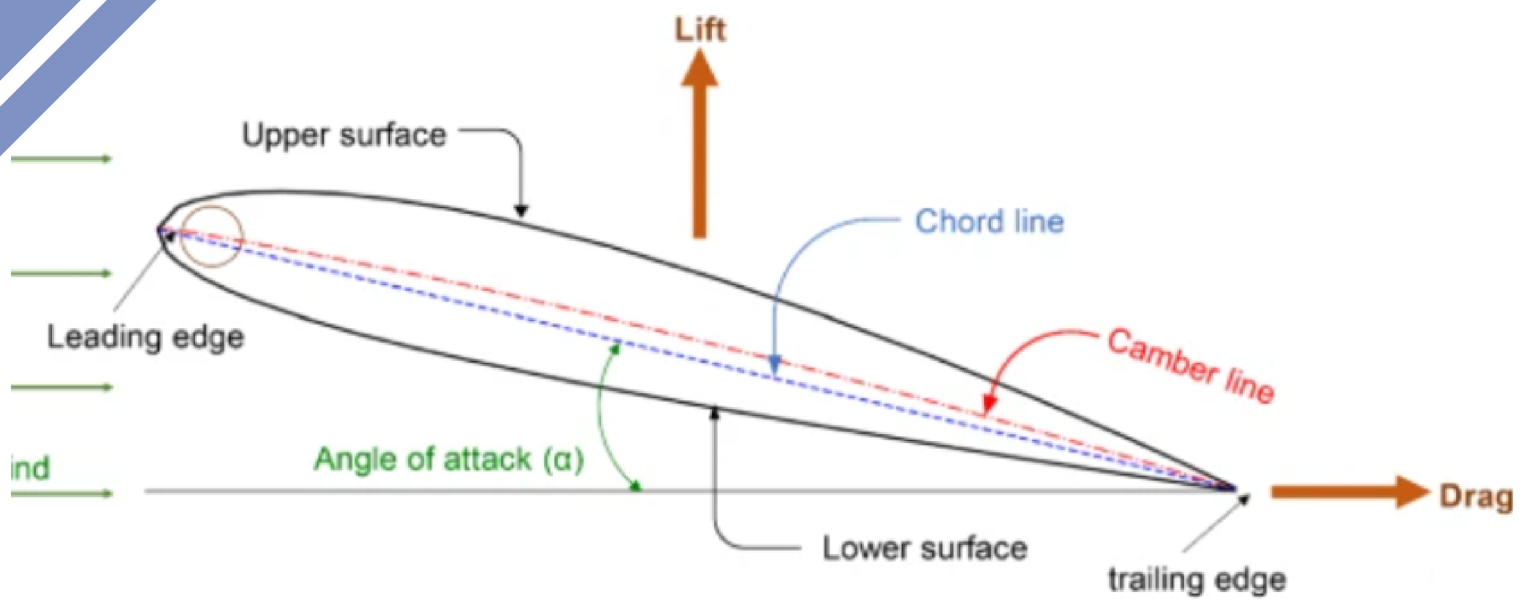
Versatility
Versatile in aviation, turbines, models due to moderate thickness and camber.

Leading-Edge Radius
Boosts lift by smoothing airflow.

Maximum Thickness Percentage
Affects lift, drag, and drag force by measuring surface differences relative to chord length.

Camber
Impacts lift at different angles; more camber means more lift but lower stalling angle.

XFLR 5



Analysis Methods

- XFLR5 utilizes the Vortex Lattice Method, Lifting Line Theory, and 3D Panel Method.
- These methods deduce the macroscopic behavior of airplanes.

Parameter Variation

- Analysis includes different Mach numbers and corresponding Reynolds numbers.
- Standard atmospheric conditions at 36,000 ft altitude are considered.

XFLR5 Analysis Tool

- XFLR5 is an open-source application.
- It employs the XFOIL program from MIT to assess airfoil parameters.

Angle of Attack Variation

- Angle of attack ranges from 2 to 5 degrees during analysis..

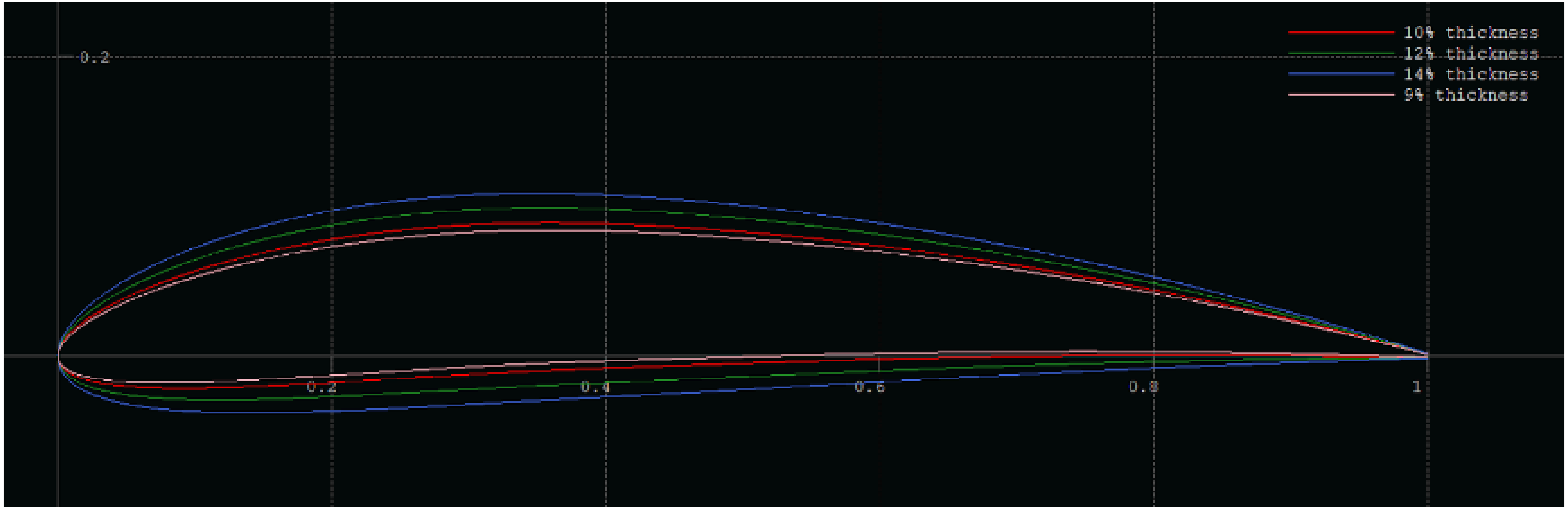
ANSYS Fluent

- ANSYS Fluent, a commercial fluid simulation software, is used.
- - It employs the RANS equation to solve various CFD problems.

Optimized Airfoil

- Results show significant potential for aerodynamic improvements.
- Findings contribute to ongoing efforts in advancing aerodynamic design.

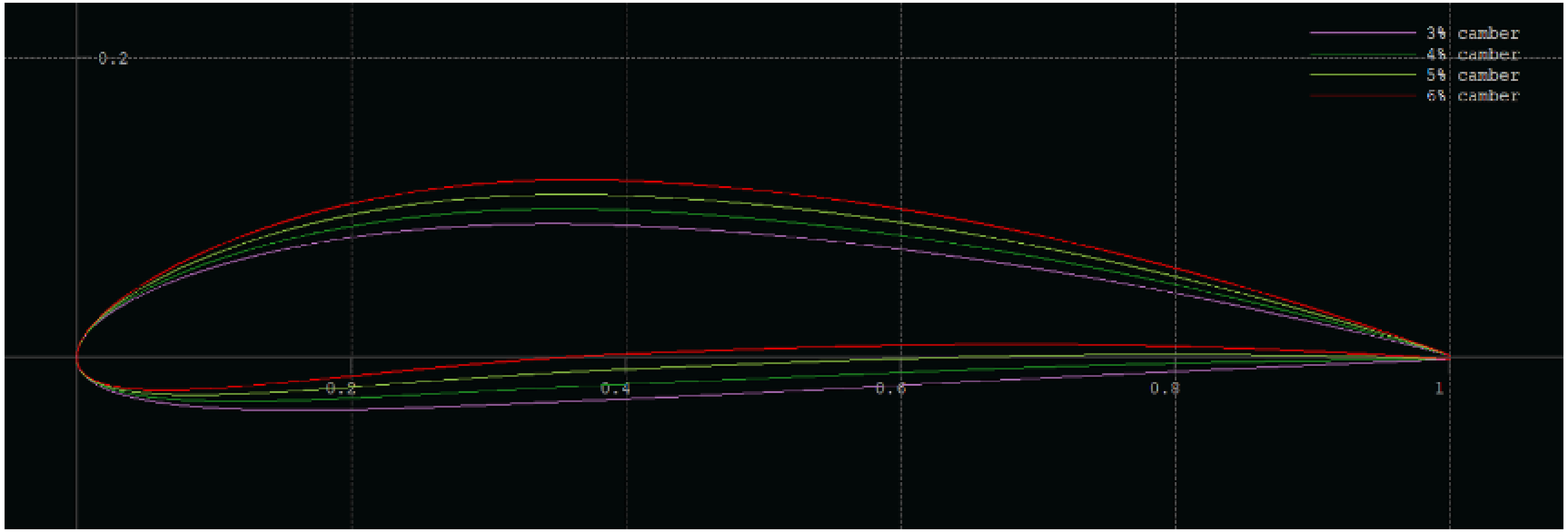
METHODOLOGY



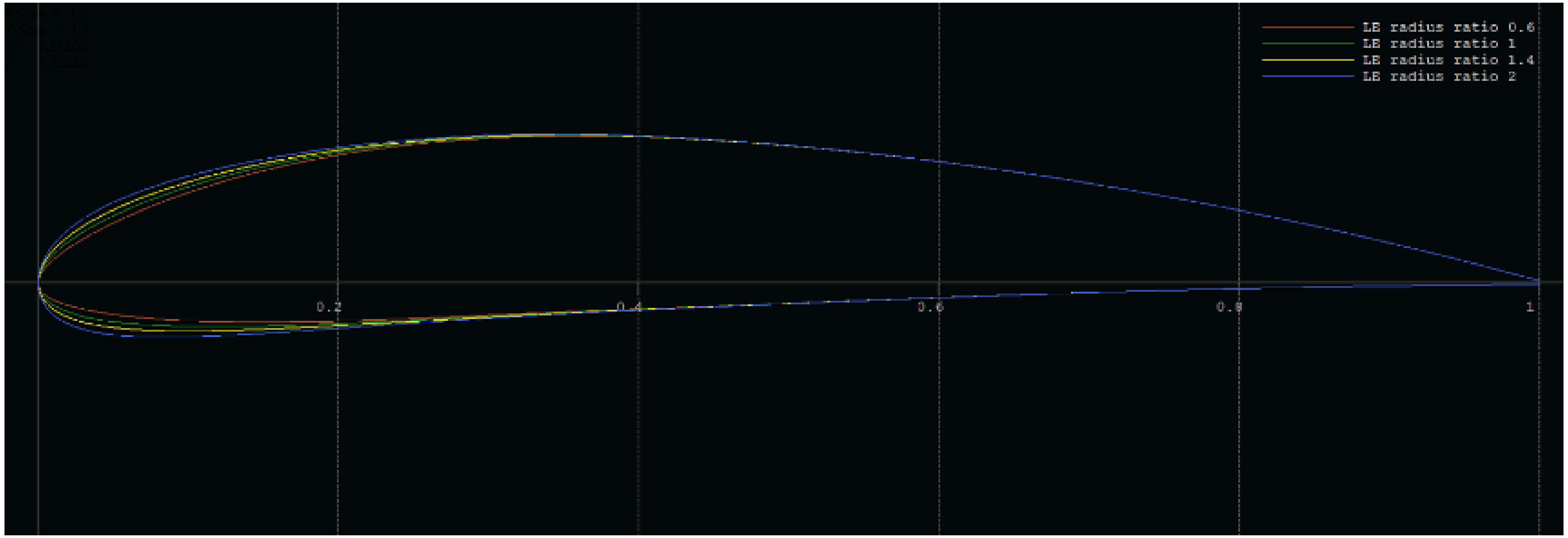
Key Parameters in Design Process and Aerofoil Optimization

- Design process involves adjusting three key parameters: Maximum camber, Maximum thickness, and Leading Edge Radius Ratio.
- These parameters are crucial for aerodynamic performance.
- Literature suggests modifying them for better results, with references backing this idea.
- Aerofoil optimization starts with the base NACA 4412 airfoil as a reference.

METHODOLOGY



Aerofoil Optimization Process



METHODOLOGY

1. Aerofoil Analysis

1. XFLR 5 Software Features:

- Boasts a comprehensive aerofoil database.
- Facilitates the selection of the base NACA 4412 airfoil from the Direct Foil Design option.
- Multiple optimizations of the NACA 4412 airfoil are generated by varying different geometrical parameters

2. Analysis Setup:

- Utilizes Direct Foil Analysis.
- Analysis conducted using Type-2 analysis.
- Mach numbers set at 0.3 and 0.7.
- Corresponding Reynolds numbers calculated for an altitude of 36,000 ft.

3. Transition Location Estimation:

- Transition location estimated using Ncrit value.
- Ncrit value set at 9 to predict disturbance in free-flowing air.

4. Analysis Parameters:

- Chord length of 1 meter.
- Angle of attack ranges from 2 to 5 degrees.
- High iteration limit of 60 ensures thorough analysis.

2. Governing Equation

XFLR 5 uses the panel method to simulate flow around two-dimensional airfoils, considering incompressible scenarios.

The basic equation used in the panel method is the integral equation of the potential flow.
This equation states that the velocity potential, Φ, is equal to the sum of the source and doublet strengths along the airfoil surface

$$\Phi = \sum_{i}^{} = N\ (S_i + K_i)$$

where S_i = source strength
K_i = doublet strengths
N = number of panels.

The source and doublet strengths are calculated by integrating the velocity potential along the airfoil surface.
The source strength is given by equation

$$S_i = \int_{c} \Phi * N_i * ds$$

where n_i = normal vector to the i-th panel
ds = differential arc length along the panel.

METHODOLOGY

2. Governing Equation

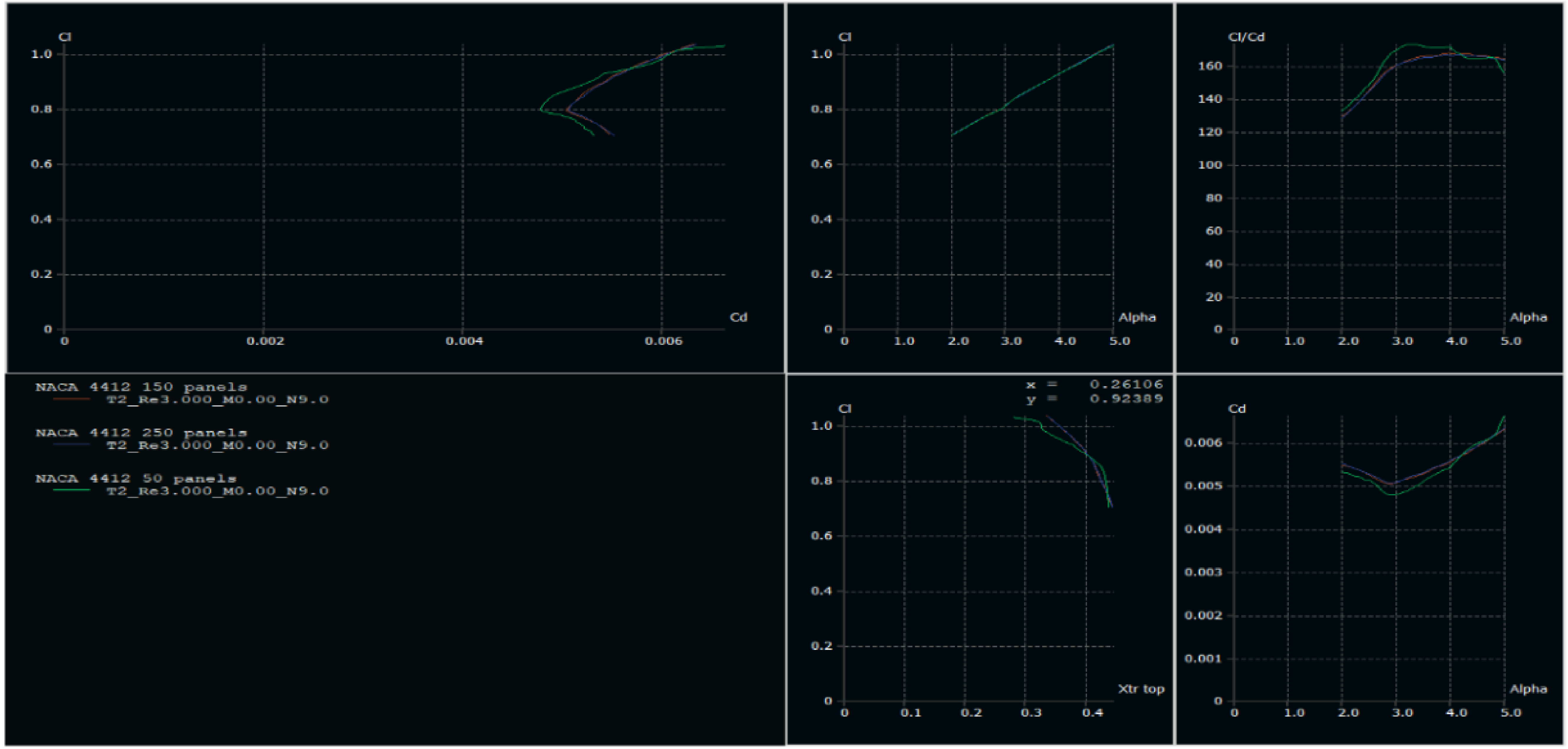
The doublet strength is given by equation

$$K_i = \int_c (\nabla^2 \Phi) * N_i * ds$$

where ∇^2 is the Laplacian operator.

3. Panel Independency

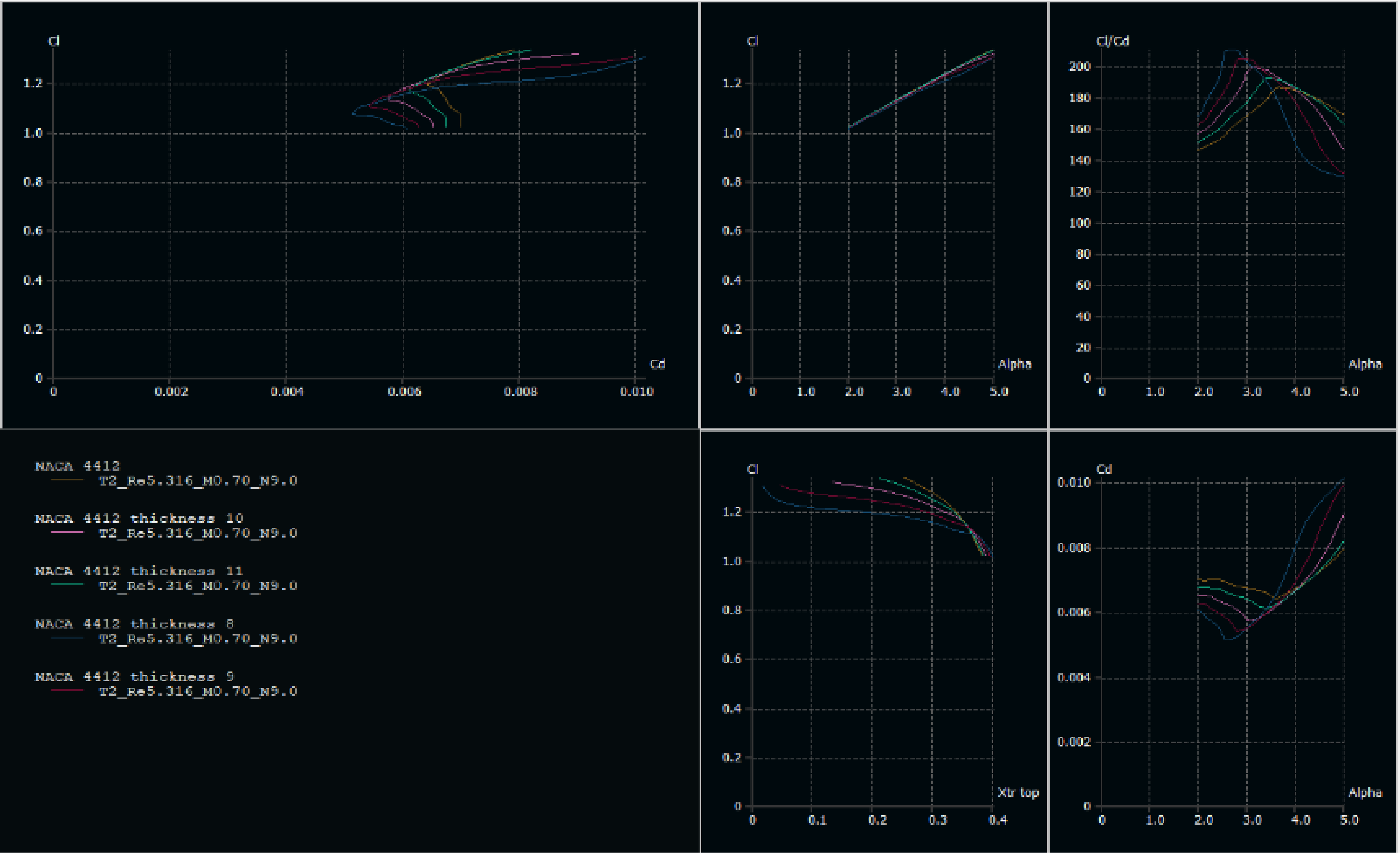
Analyzing convergence, results deviated with 50 panels but converged well with 150 and 250 panels. Hence, 200 panels were selected for simulation to balance accuracy and computational cost.



Convergence Analysis of Coeffecient of drag and coeffecient of lift of aerofoil NACA 4412:
Higher number of panels give accurate results but with increase in the no.of panels comes the computational cost which is to be kept in vision.

AEROFOIL DESIGN PROCESS

1. Maximum Thickness



1.Impact of variation of maximum thickness observed:

- Mach number: 0.7
- Corresponding Reynolds Number (at 36000 ft altitude): 5.3161e+6 1/m

2.Decreasing thickness:

Coefficient of drag:

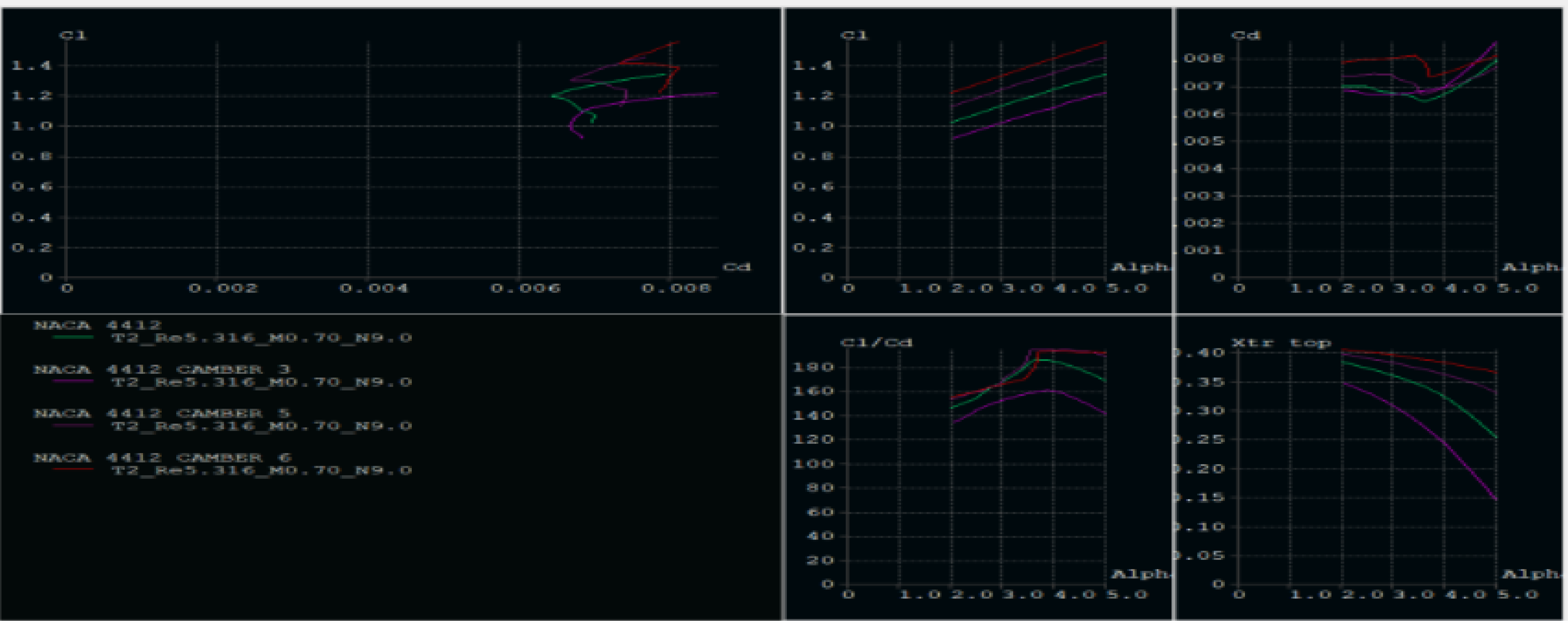
- Lower for aerofoil with lesser thickness initially
- Drastically increased, particularly in 8% and 9% thickness cases
- Sharp decrease in the value of Cl/Cd just before 3-degree angle of attack

3.Lift coefficient:

- Almost unaffected

AEROFOIL DESIGN PROCESS

2. Maximum Camber



1.Increased camber:

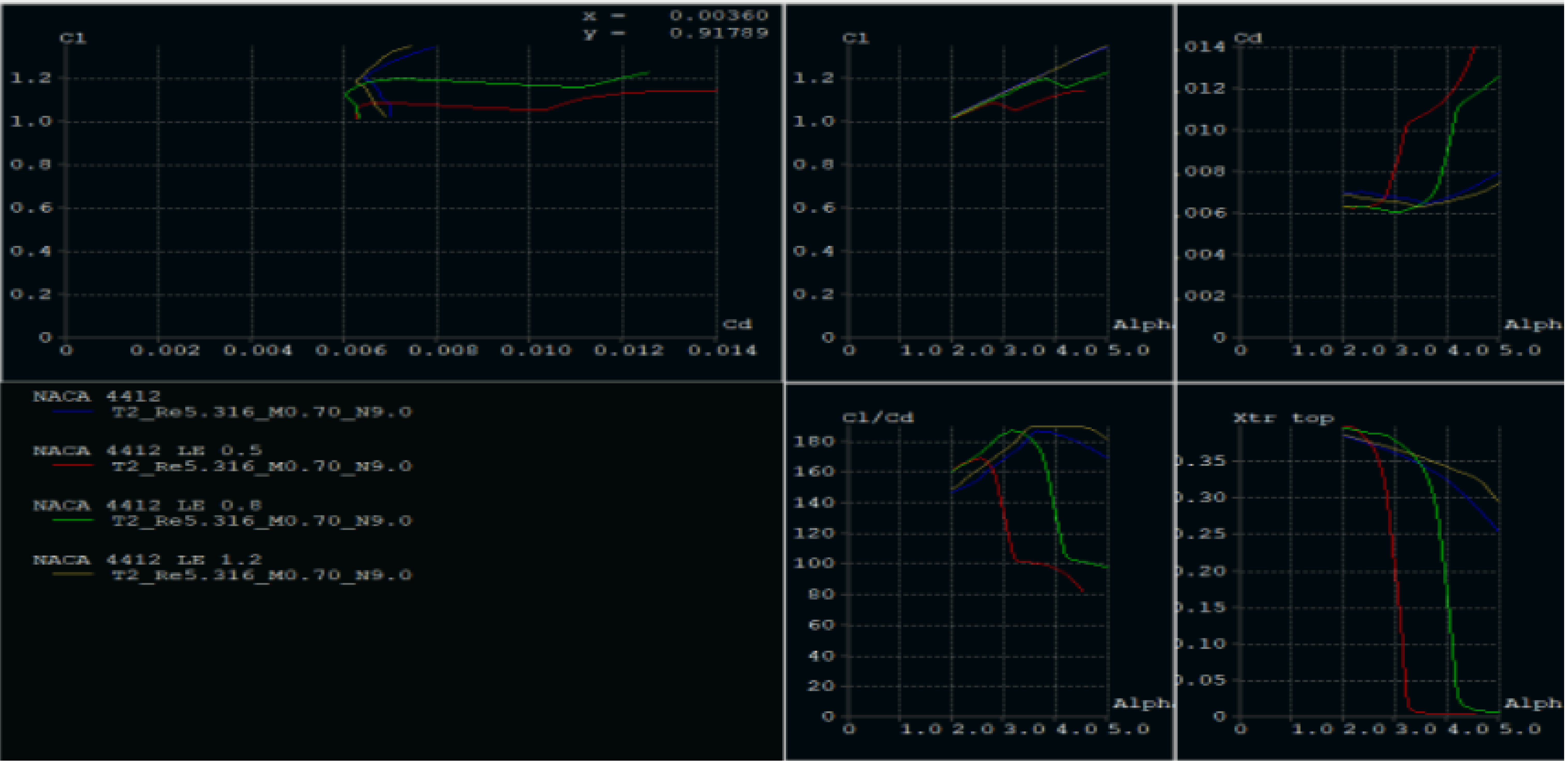
- Boosted lift and drag coefficients
- Lift increase outweighed drag increase, improving CL/CD

2.Aerofoil with 6% camber:

- Dropped due to excessive minimum CL (>1.2)
- Allowed range: 0.6-1.2

AEROFOIL DESIGN PROCESS

3. Leading Edge Radius Ratio



Decreasing leading edge (LE) radius ratio:

- Lift and drag coefficients increased
- Sharp increase in drag
- Significant decrease in CL/CD ratio
- Increase in steepness led to considerable drag increase

AEROFOIL DESIGN PROCESS

RESULT

After carrying out multiple batch analysis on XFLR5 4 aerofoils were selected for further analysis in ANSYS. The properties of these aerofoils are mentioned in the table:

Name of the Aerofoil	Maximum Thickness(%)	Maximum Camber(%)	LE radius ratio
Design 1	10	4	1
Design 2	9	5	1
Design 3	12	5	1
Design 4	9	5	0.6

ANSYS AEROFOIL ANALYSIS

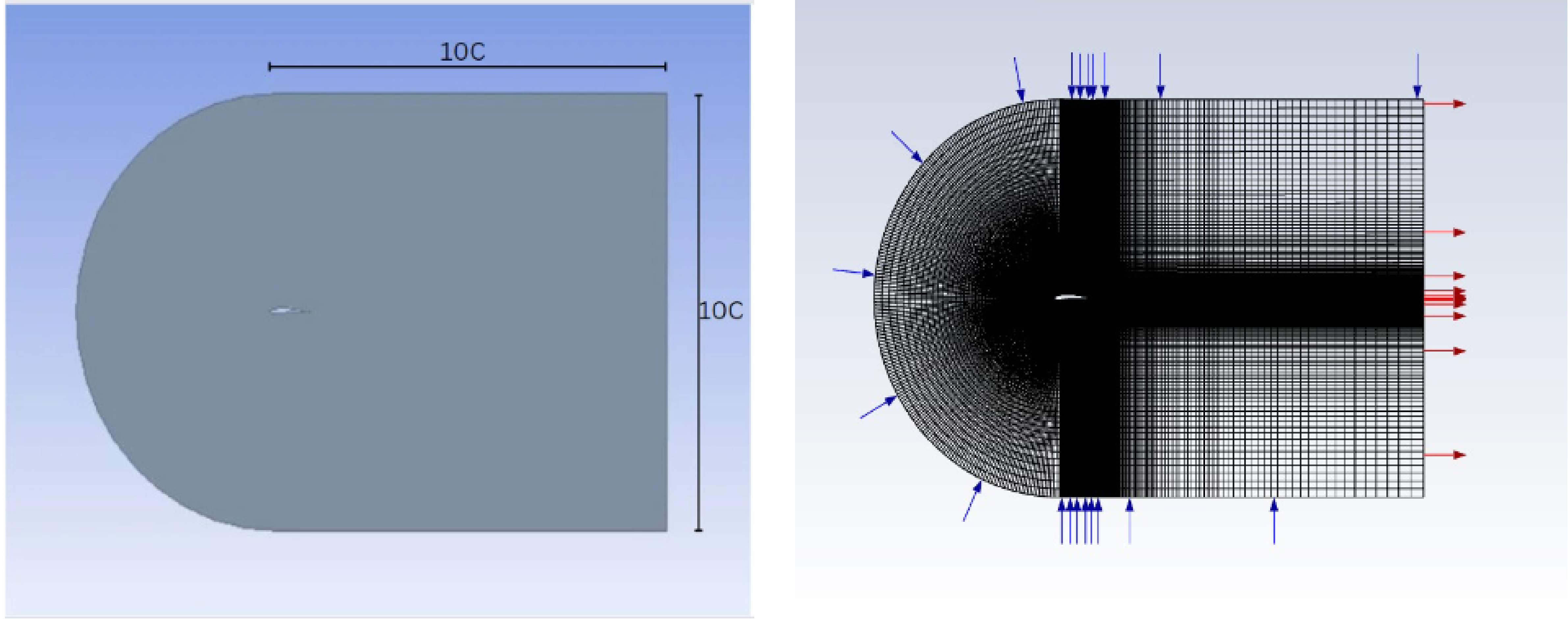


Fig1 a)C-type fluid domain b)Fluid Domain Mesh

- Studied aerodynamic performance of four airfoil profiles using ANSYS FLUENT 2022R1.
- Utilized Spalart Allmaras turbulence model for accurate results and lower memory usage.
- Airfoil coordinates from Airfoil Tools database; geometry created in ANSYS Design Modeler with 1m chord length.
- "C-type" fluid domain created around airfoil in ANSYS Design Modeler, comprising a semicircle (radius 5C) and rectangular domain (10Cx10C).
- Aimed to capture curvature effect properly from leading edge.
- Inlet and outlet boundary conditions depicted in Figure 1.

ANSYS AEROFOIL ANALYSIS

1. Governing Equation

In this study, the flow is assumed to be steady, two-dimensional, and compressible. The simulation is done for a Re number of 5.3161e+6 1/m, and the governing equations are the continuity equation and the Navier- Stokes equations. They can be written as follows:

$$\frac{\partial(\rho u)}{\partial x} + \frac{\partial(\rho v)}{\partial y} = 0$$

$$\frac{\partial(\rho u^2)}{\partial x} + \frac{\partial(\rho u v)}{\partial y} = -\frac{\partial p}{\partial x} + \mu\left(\frac{\partial^2 u}{\partial x^2} + \frac{\partial^2 u}{\partial y^2}\right)$$

$$\frac{\partial(\rho u v)}{\partial x} + \frac{\partial(\rho v^2)}{\partial y} = -\frac{\partial p}{\partial y} + \mu\left(\frac{\partial^2 v}{\partial x^2} + \frac{\partial^2 v}{\partial y^2}\right)$$

Since the flow is assumed to be compressible, the air density and kinematic viscosity are variable . The equations that are used to calculate the lift and drag coefficients are :

$$C_d = \frac{2F_d}{\rho u^2 A} \qquad C_l = \frac{2F_l}{\rho V^2 A}$$

ANSYS AEROFOIL ANALYSIS

2. Turbulence model

- Spalart-Allmaras turbulence model chosen for aerospace applications.
- Shows promise for boundary layers with adverse pressure gradients.
- Simple one-equation model solving for turbulent viscosity.
- Primarily designed for low Reynolds number situations.
- Requires proper resolution of viscous-affected boundary layer region.
- The transport equation is given as :

$$\frac{D\tilde{v}}{Dt} = c_{b1}(1 - f_{t2})\tilde{v}\tilde{s} + \frac{1}{\sigma}\left[\nabla . \left((v + \tilde{v})\nabla \tilde{v}\right) + c_{b2}(\nabla\tilde{v})^2\right] - \left(c_{w1} f_w - \frac{c_{b1}}{k^2} f_{t2}\right)\left(\frac{\tilde{v}}{d}\right)^2 + f_{t1}\Delta U^2$$

3. Boundary Conditions

- Aerofoil surface designated as "wall" enforcing no-slip condition.
- Inlet velocity: 206.61 m/s derived from Mach 0.7 and 36000 ft altitude.
- Inlet turbulent viscosity ratio: 1 for initial laminar flow.
- Outlet turbulent viscosity ratio: 10 to initiate turbulent flow.
- Higher outlet ratio assumes turbulent flow from the start, unlike the gradual transition from laminar to turbulent at the inlet.

4. Methods

- Pressure-based coupled algorithm
- Solves compressible RANS equations
- Enhanced convergence rate
- Second-order pressure discretization
- Second-order upwind for momentum and turbulence viscosity

RESULTS AND DISCUSSION

1. Mesh Validation

- Mesh validation performed against experimental study by Abbott aerospace.
- Inlet velocity set to 43.9 m/s.
- Reynolds number of 3,000,000.
- Angle of attack: 0 degrees.
- Plot of CL (lift coefficient) and CD (drag coefficient) versus iterations provided.
- Validation ensures accuracy and reliability of mesh for simulation.

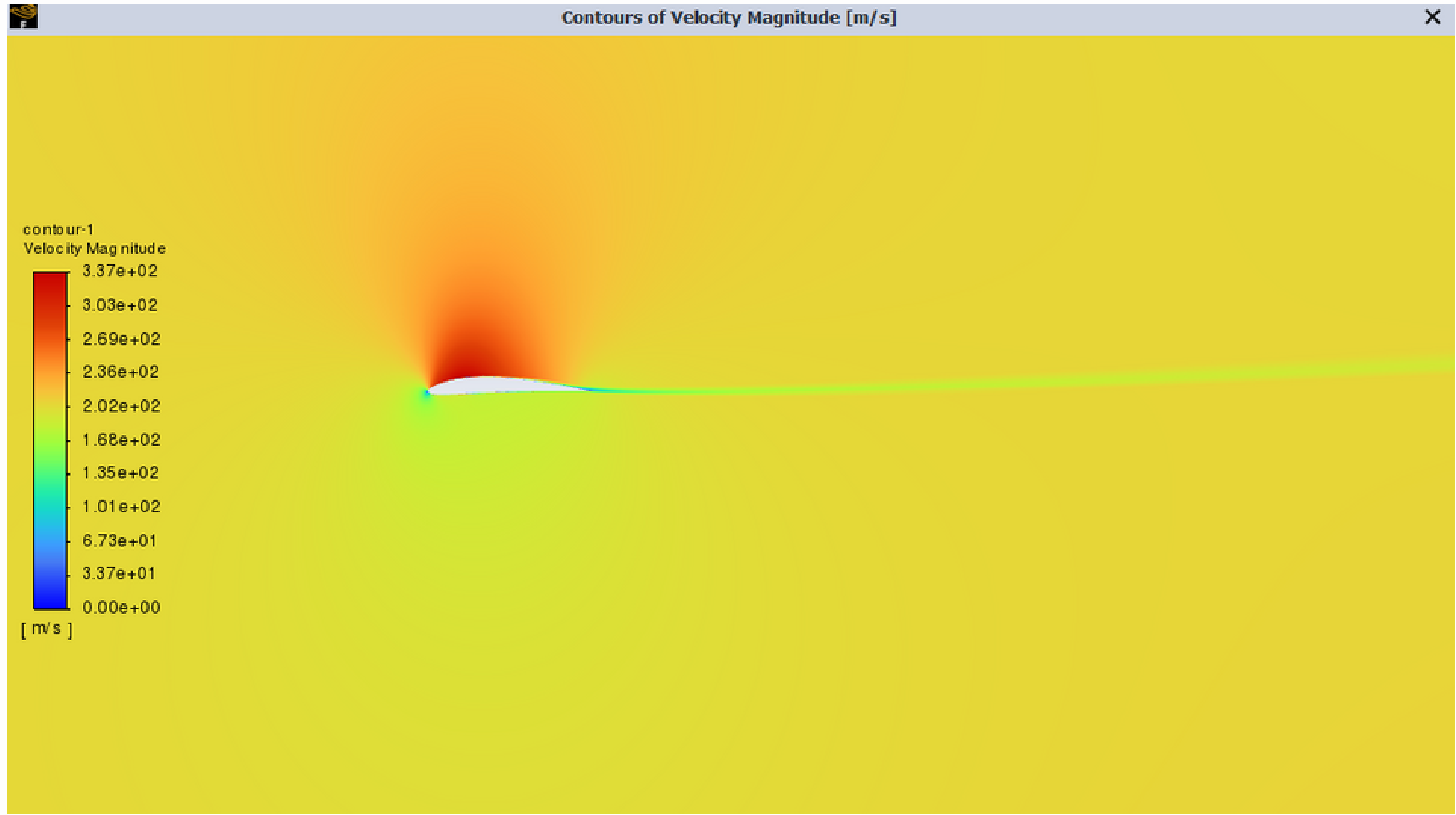
2. Required Data for CFD Simulation

After the analysis on XFLR5 The maximum CL/CD value was observed and correspondingly the optimum angle of attack for the 4 aerofoils was chosen as shown in the table

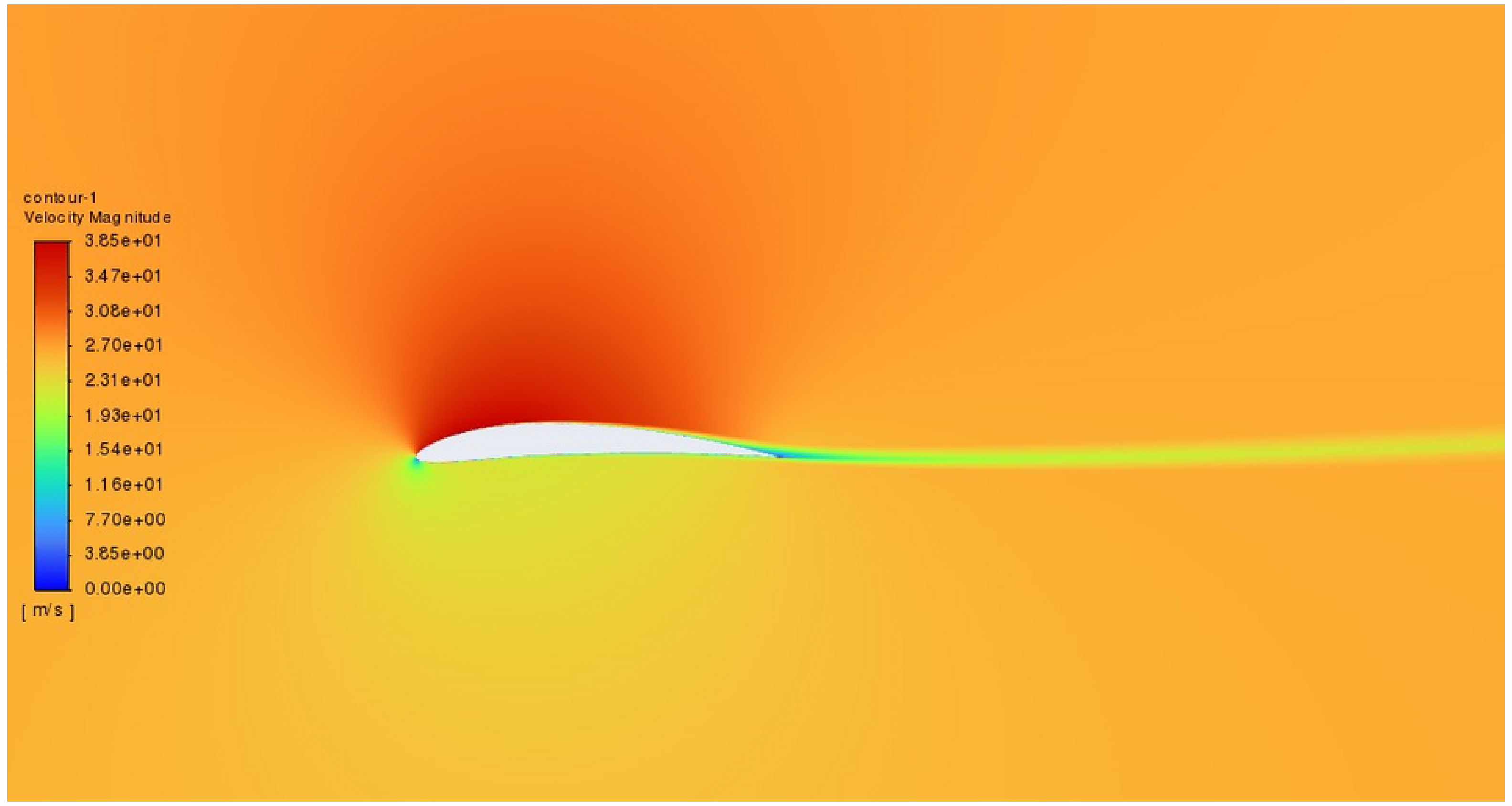
Name of the Aerofoil	Optimum angle of attack
Design 1(10% thickness)	3
Design 2(9% thickness and 5% camber)	4
Design 3(5% camber)	4
Design 4(9% thickness, 5% camber and 0.6 LE radius ratio)	3.5

3. CFD Simulation Results

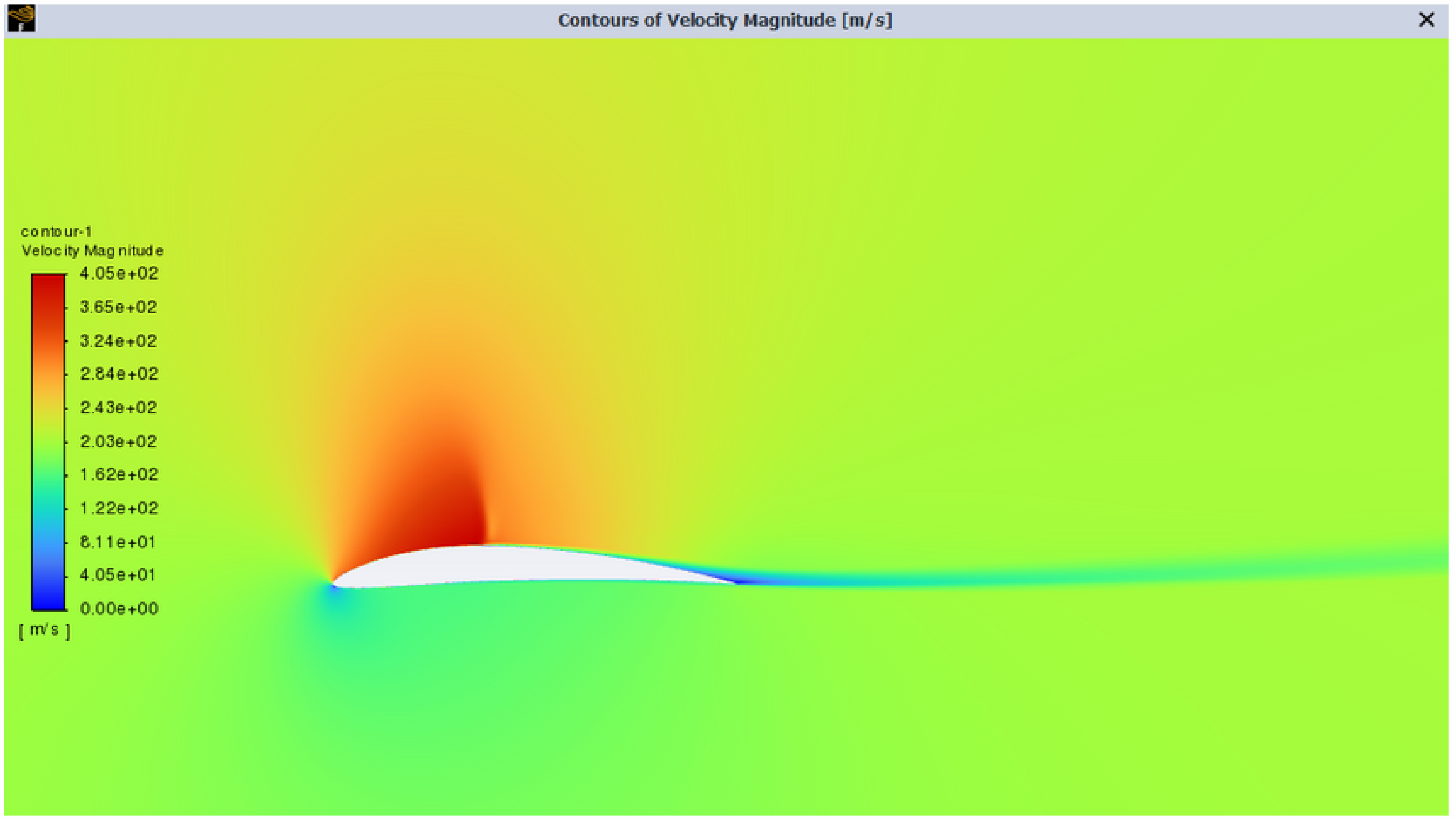
1. In this aerodynamic study the values of lift and drag coefficient are computed alongwith the velocity and pressure contours for the 4 aerofoil profiles mentioned previously.
2. The results obtained were as follows



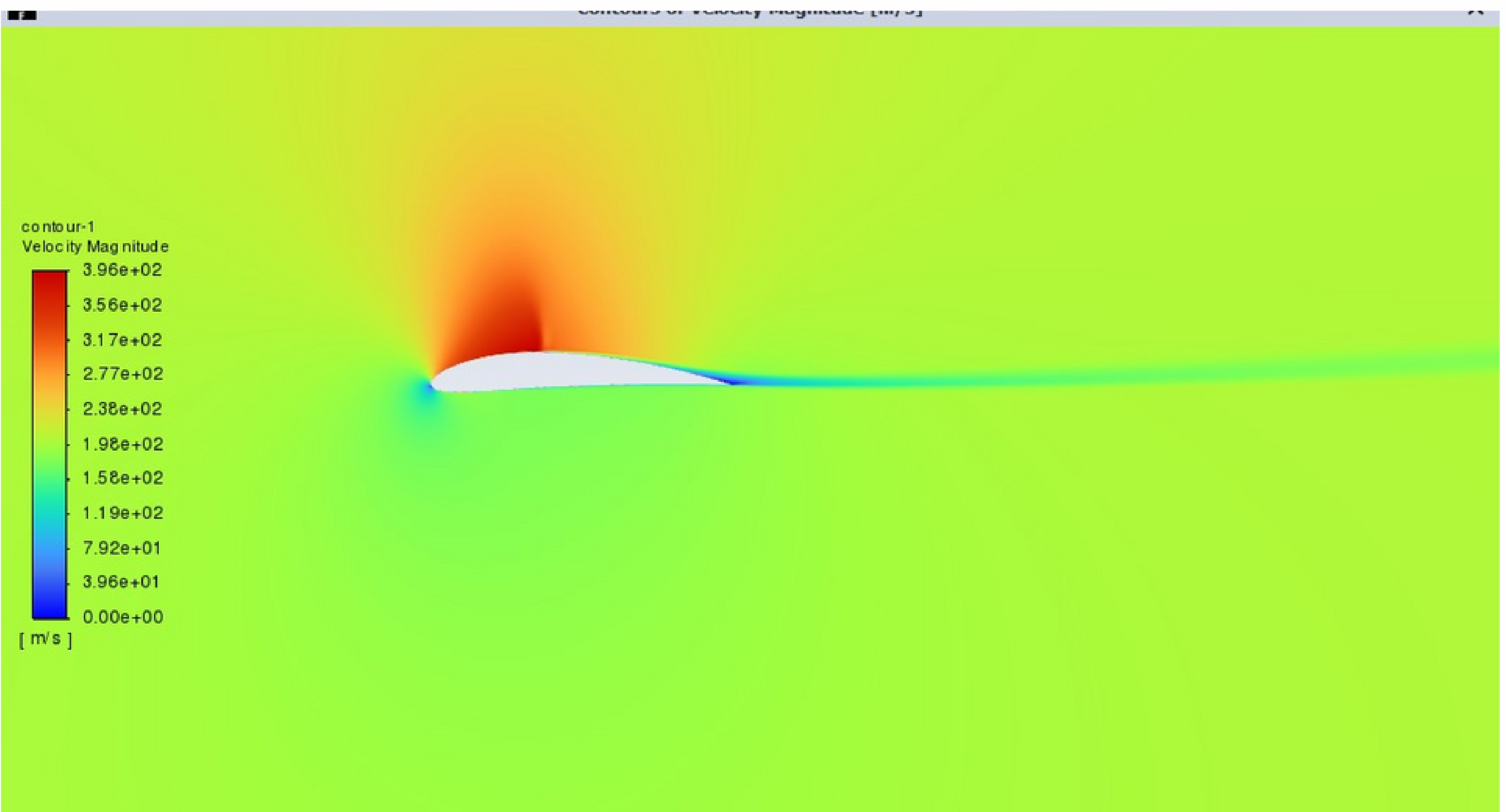
Velocity Contour of Design 1(Thickness 10%)



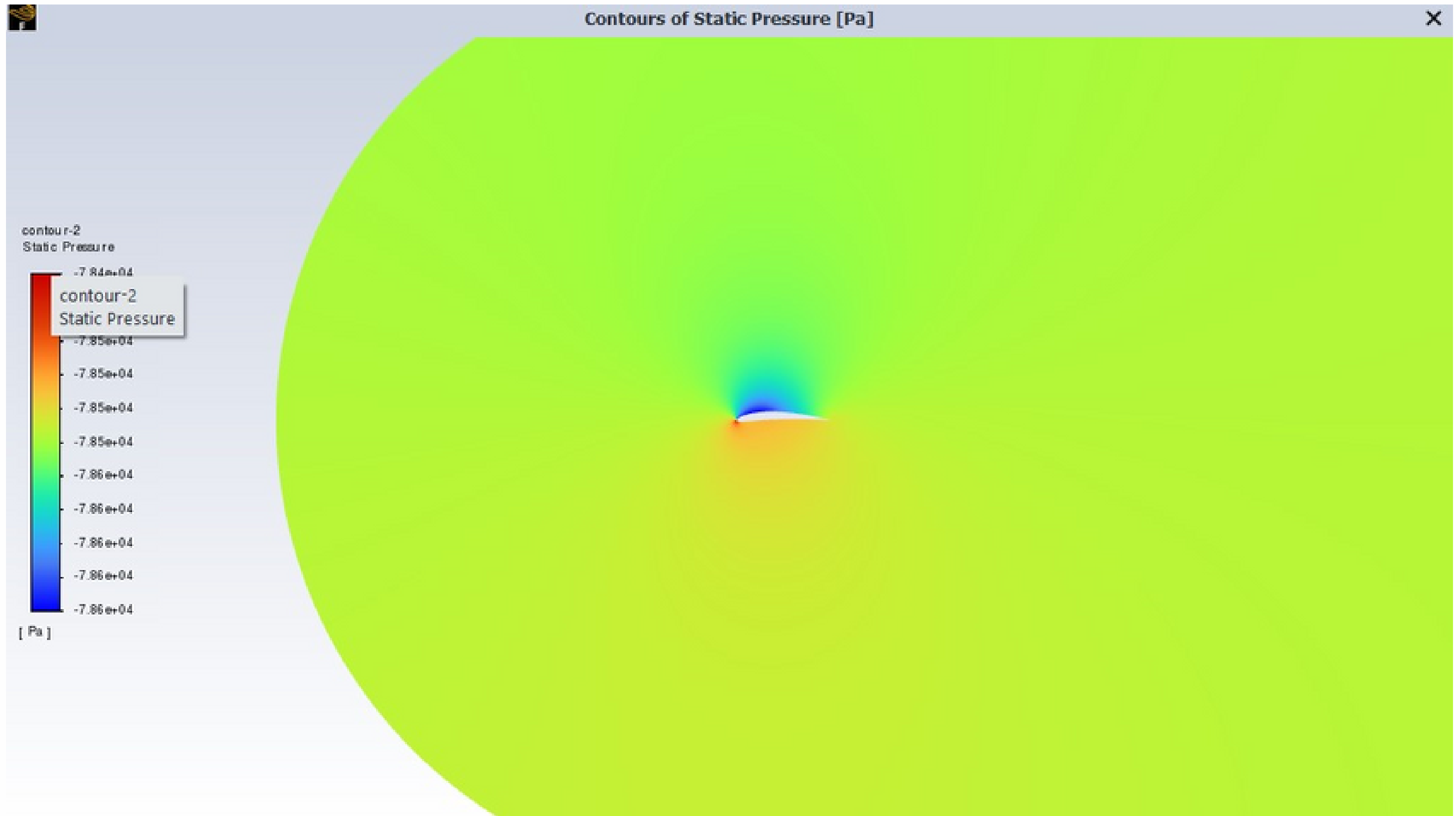
Velocity Contour of Design 2(Thickness 9% and Camer 5%)



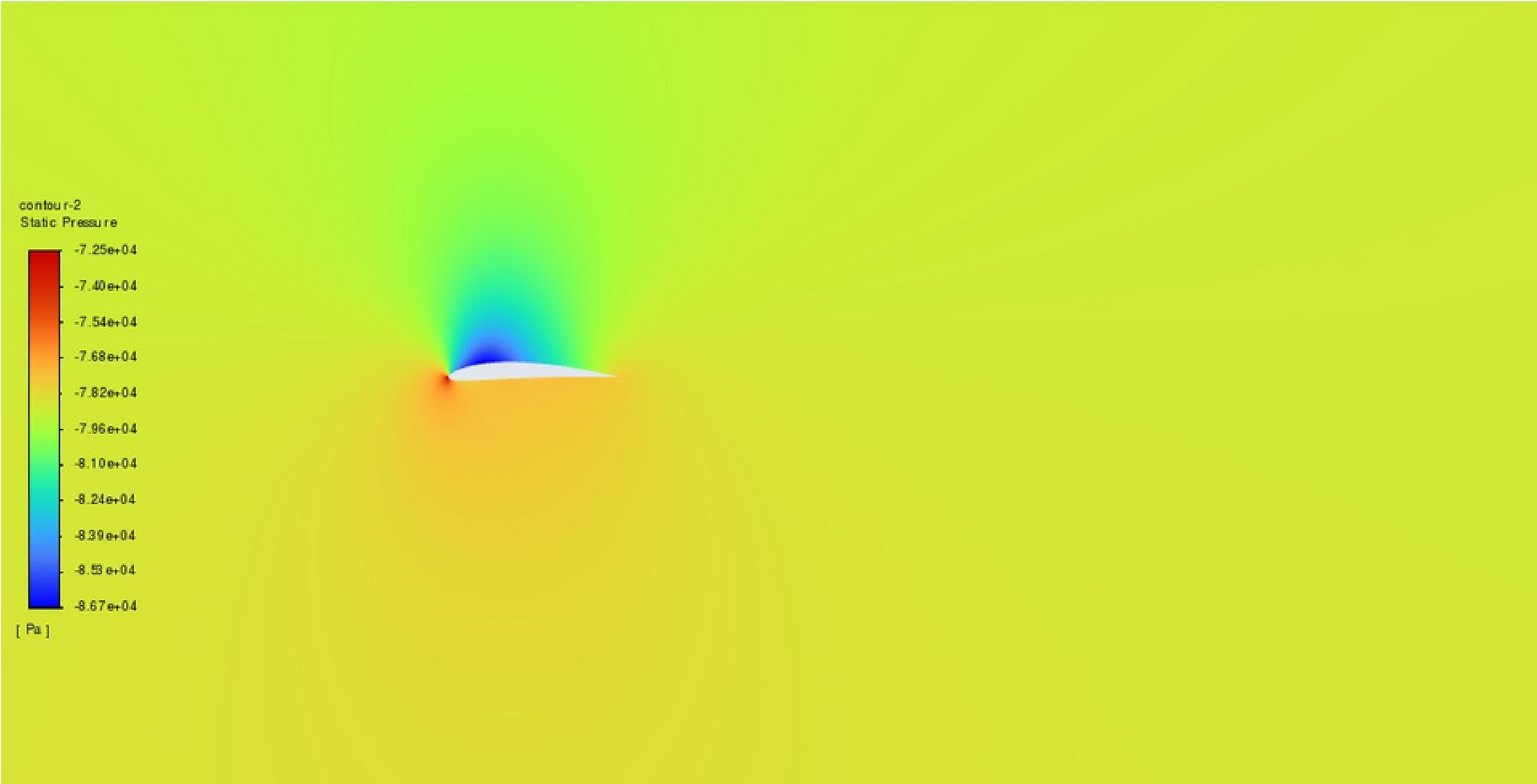
Velocity Contour of Design 4(Thickness 9% camber 5% and LE radius ratio 0.6)



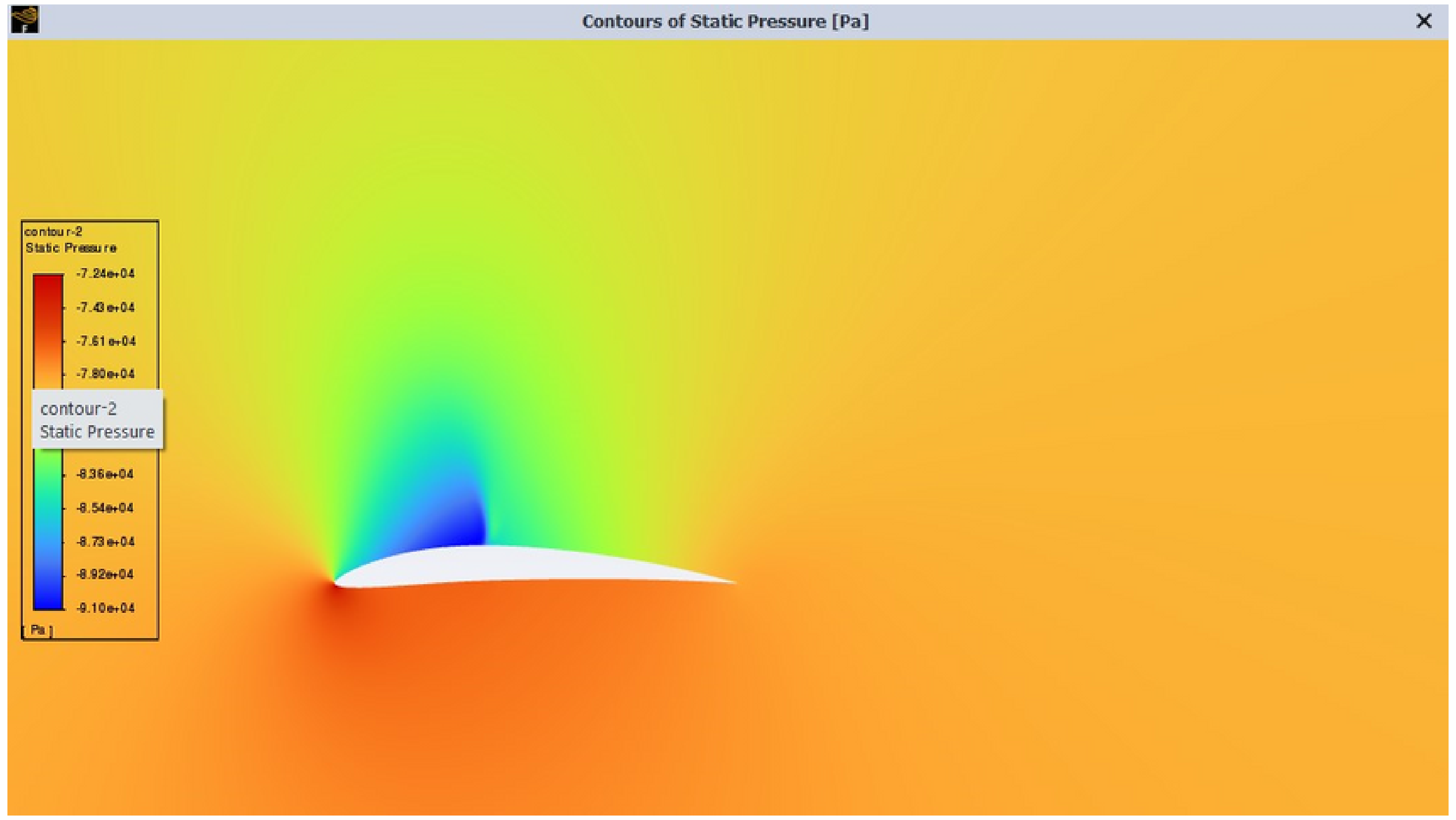
Velocity Contour of Design 3(camber 5%)



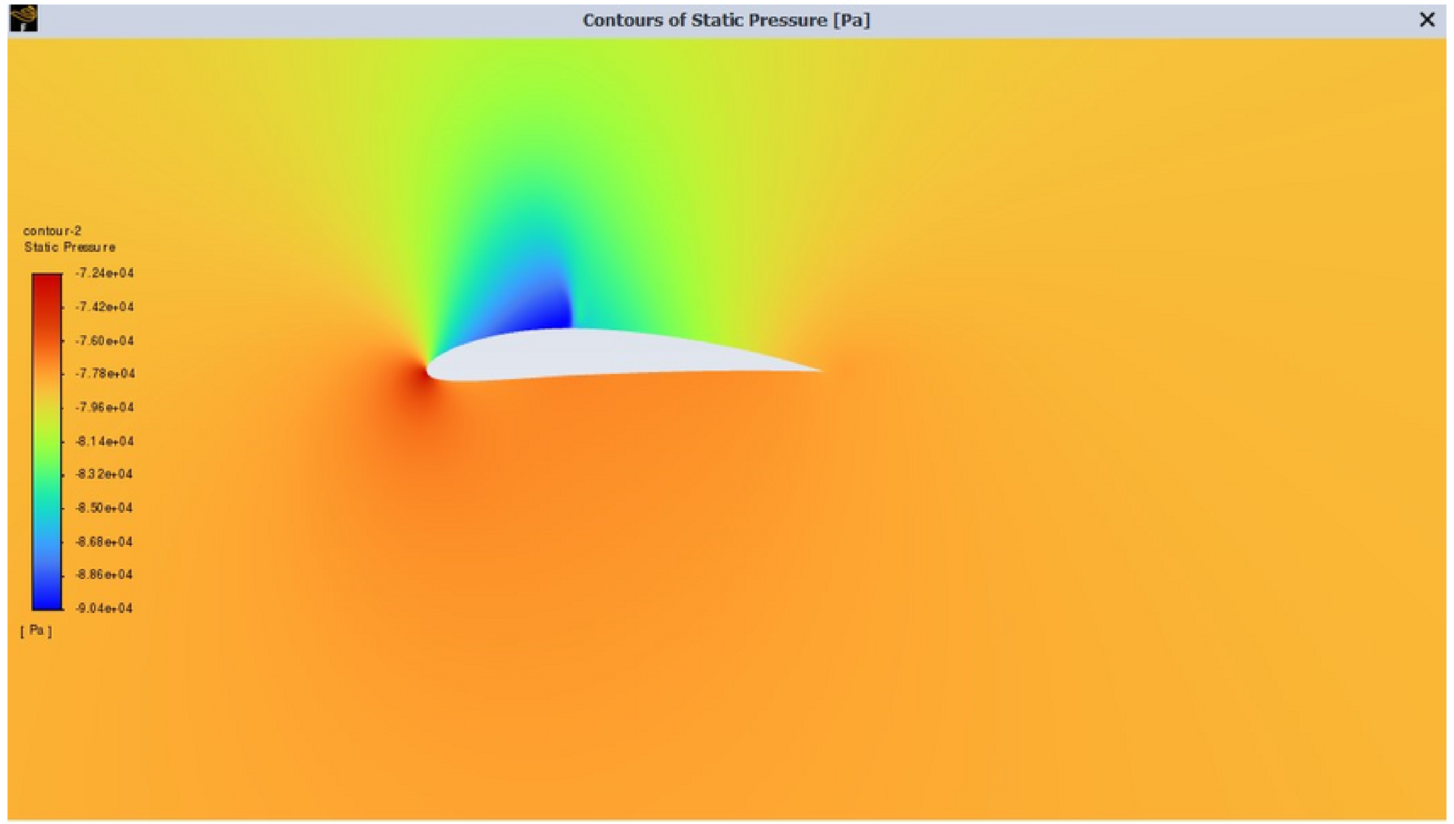
Pressure Contour of Design 1(Thickness 10%)



Pressure Contour of Design 2(Thickness 9% and Camber 5%)



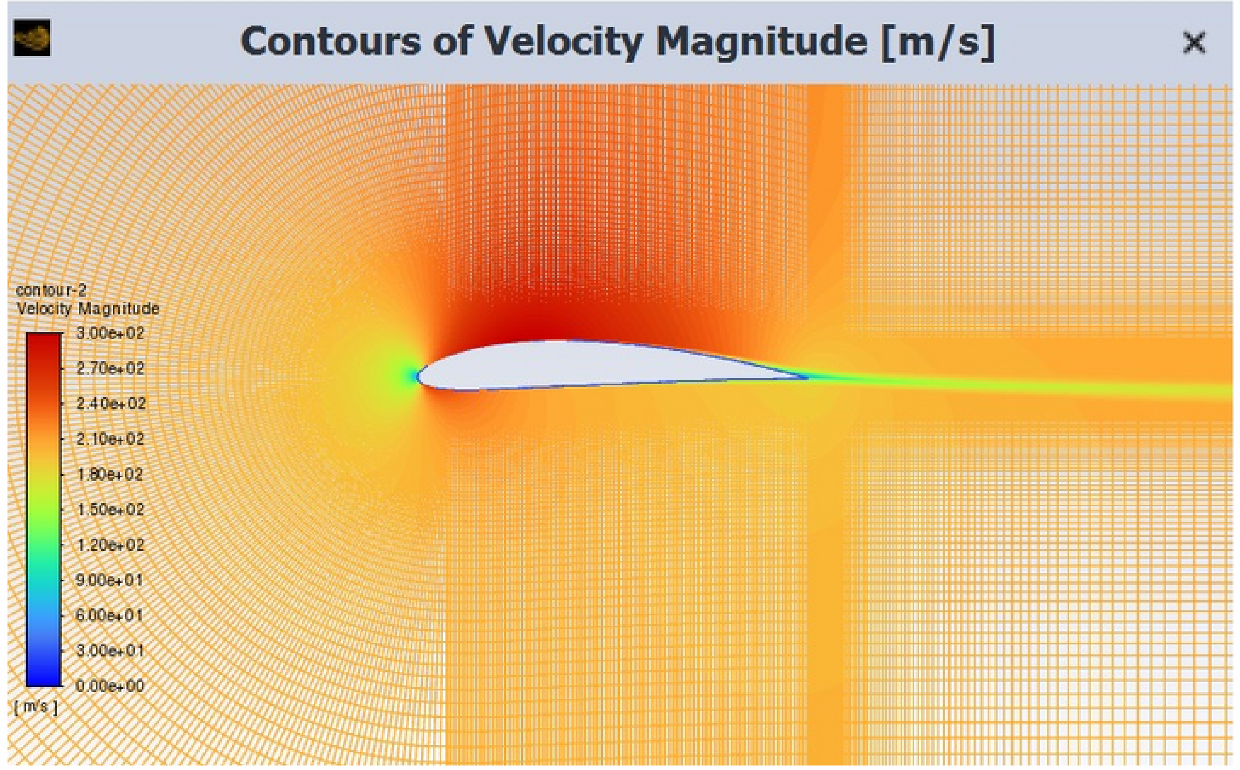
Pressure Contour of Design 4(Thickness 9%, camber 5% and LE radius ratio 0.6)



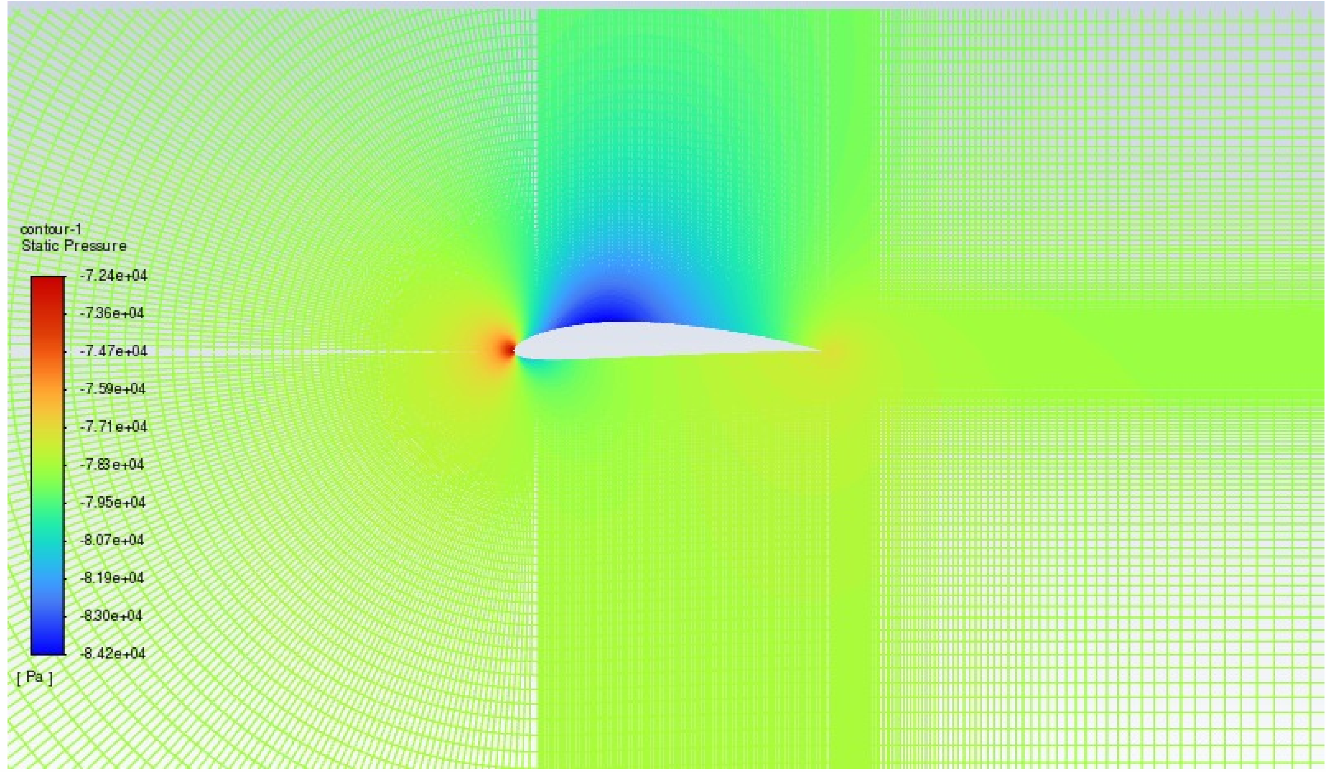
Pressure Contour of Design 3(Thickness 9% and Camber 5%)

Name of the Aerofoil	Cl value	Cd value
Design 1(Max Thickness 10%)	9.7518e-01	-3.8682e-02
Design 2(Max Camber 5%)	1.1903	-6.4822 e-02
Design 3(Max Thickness 9% and Max Camber 5%)	9.5067e-01	-4.9987e-02
Design 4(Max Thickness 10%, Max Camber 5% and LE Radius 0.6)	1.3541e+00	-9.7570e-02

4. Conclusion



Velocity Contour of NACA 4412



Pressure Contour of NACA 4412

- After comparison of the results with the original NACA 4412 results we found that increase in thickness had a much positive impact on the Cl/Cd value as it considerabely reduced the Cd value while keeping a decent value of Cl hence reflecting the best Cl/Cd value among the other 3 aerofoils
- The Cl value for NACA 4412 was 0.42 and the Cd value was 0.01

THANK YOU!!

TEAM MEMBERS:

AARUSH GOYAL

NAMAN KANKARIYA

AYANSH BHARDWAJ

KUSH AGGRAWAL

KISHAN SONI