External flow around a straight rectangular wing using OpenFOAM

Harshit Dhiman

July 1, 2025

1 Introduction

This report outlines the analysis of external flow over a straight rectangular wing. For this analysis, we use the free and open source software OpenFOAM. The main objective of this study is to find out the lift and drag coefficient of the wing and also find the Cp vs x/c plot of the airfoil.

2 Problem Description

We have a straight rectangular wing as the body of influence. The fluid properties and the wing dimensions are given below.

- Fluid medium → air

  Density, $\rho = 1.225 kg/m^3$

  Kinematic viscosity, $\nu = 1e-5 m^2/s$

- Wing dimensions and orientation

  Angle of attack, $\alpha = 4°$

  Span, $l_s = 1.5m$

  Chord length, $c = 0.25m$

  Cross-section → clark Y airfoil

The goal is to calcualte the force coefficients listed below and verify our results using publicly available data.

- Lift coefficient, C_l
- Drag coefficient, C_d
- Pressure coefficient, C_p

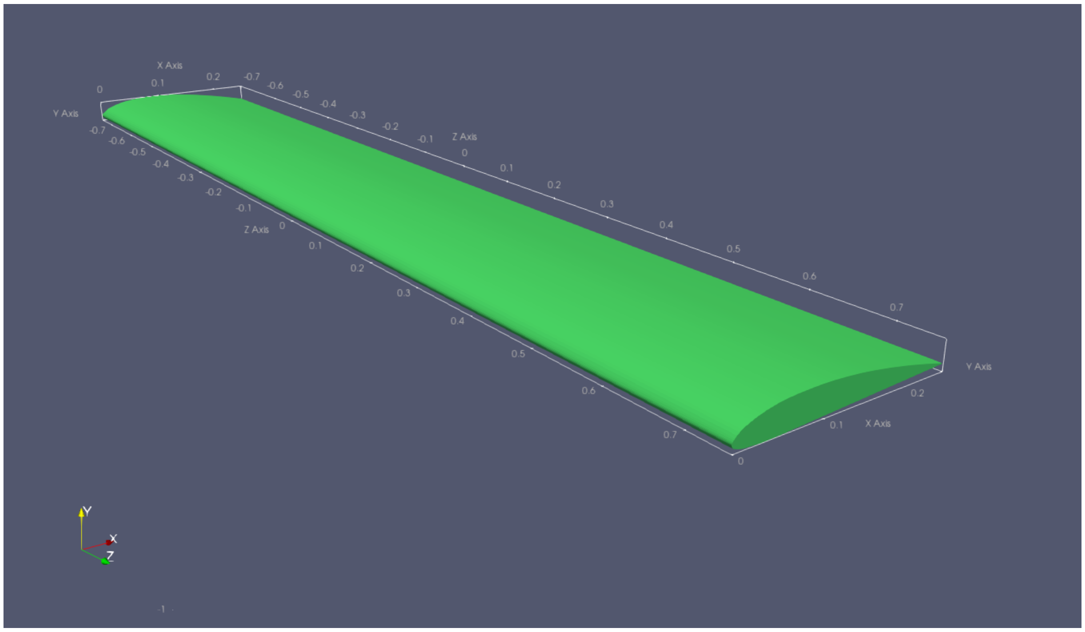

Figure 1: Wing Geometry

Table 1: Dimensions of the computational doamin for different cases

$x(m)$	$y(m)$	$z(m)$	**Distance of wing from inlet** (m)
6	4	3	3

3 Preprocessing

3.1 Geometry

The geometry has two main components, the wing and the fluid domain. The wing is a straight rectangular wing with clark Y airfoil as its cross section. The airfoil data is taken from UIUC Airfoil Coordinates Database. Figure 1 shows the wing geometry.

The fluid domain is a simple box domain with dimensions as tabulated in Table 1.

The domain for the case can be seen in Figure 2 and 3. The computation model only captures one half of the whole intended problem. The whole domain is modelled using a symmetry boundary condition on the middle parting plane of the wing. Doing this helps us reduce the amount of computation needed to solve the problem.

The geometry was created in FreeCAD and exported as .stl files to be used as base surfaces for mesh creation.

3.2 Mesh

The mesh generated is a hex mesh using OpenFOAM utility snappyHexMesh. Mesh properties are listed below:

- Maximum element size = 0.1m.
- Mesh refinement is applied on wing surface. The properties of refinement done are listed below:

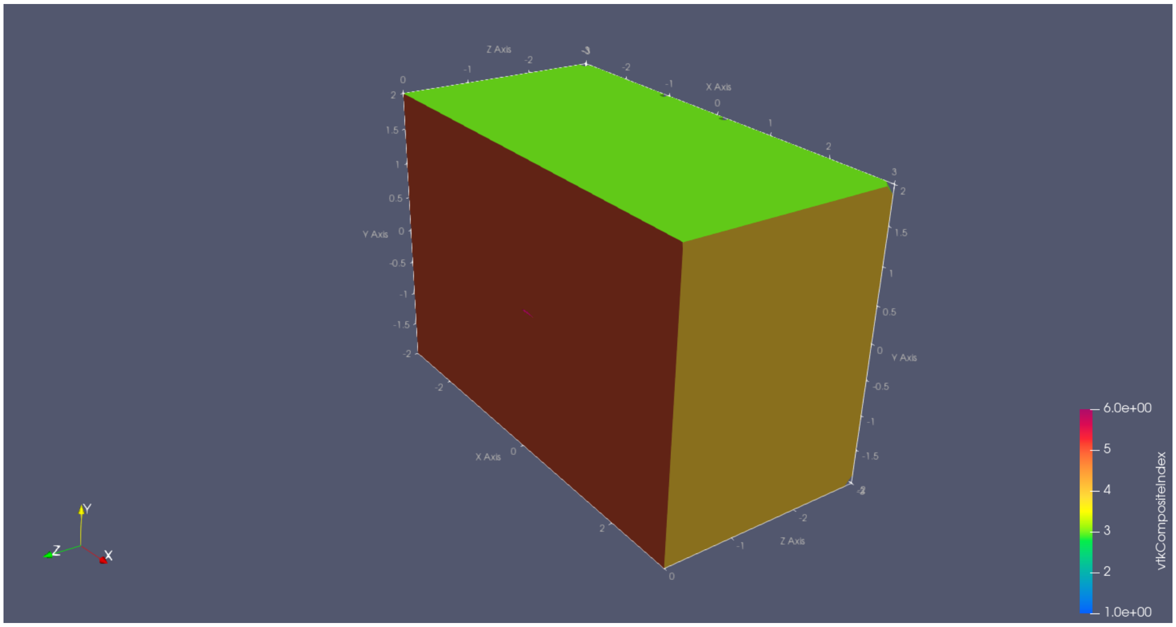

Figure 2: Fluid domain showing outlet

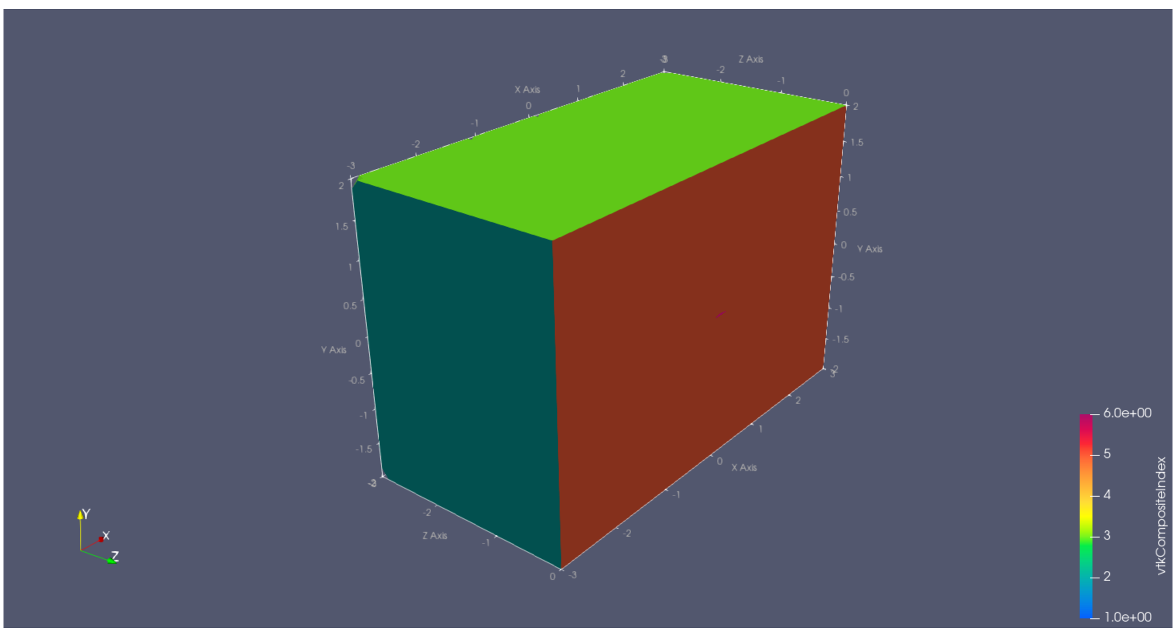

Figure 3: Fluid Domain showing inlet

- Surface refinement → level 4
- Edge refinement → level 2
- Feature refinement (to capture curvature) → level 5 with minimum feature angle to be resolved 15°
- Inflation layers are added to the wing surface to resolve boundary layer flow. Below are the properties of the layers added.
  * Final layer thickness → 0.3 times the local element size
  * Expansion ration → 1.2
  * Number of layers → 5

- A cuboidal region is defined in the wake of the wing for refinement. The level of refiment in the wake region is level 2.
- Some more refinement regions are defined for around the wing geometry as buffer zones for refinement of the wing.

The mesh created is shown in the Figure 4 and Figure 5. The mesh statistics are shown in Figure 6 and the mesh quality statistics are shown in Figure 7.

Limitation of Wall Resolution*: Post-processing analysis of the near-wall turbulence statistics reveals an average dimensionless wall distance (y^+) of approximately 22. This places the first cell height directly within the buffer layer ($5 < y^+ < 30$), a transition region where neither the viscous sublayer formulation ($y^+ < 1$) nor standard wall functions ($y^+ > 30$) can accurately model the shear stress.*
For the $k - \omega SST$ turbulence model used in this study, a $y^+ < 1$ is recommended to resolve the viscous sublayer and accurately predict skin friction drag. The current mesh resolution, constrained by available computational resources, is likely to introduce numerical inaccuracies, particularly in the prediction of the drag coefficient (C_d). Additionally, the downstream domain length (12 chord lengths) may not be sufficient to fully isolate the outlet boundary from the wake structure.

4 Case Setup

4.1 Numerical Model

The numerical model used is the steady state $k - \omega SST$ model with an assumed 5% turbulence intensity ($T_i = 0.05$).

The formulae used to calculate the turbulence variables is given in below eqautions.

$$k = \frac{3}{2} \cdot (U \cdot T_i)^2 \tag{1}$$

$$\epsilon = (C_\mu)^{\frac{3}{4}} \cdot \frac{k^{\frac{3}{2}}}{l} \tag{2}$$

$$l = 0.07 \cdot L \tag{3}$$

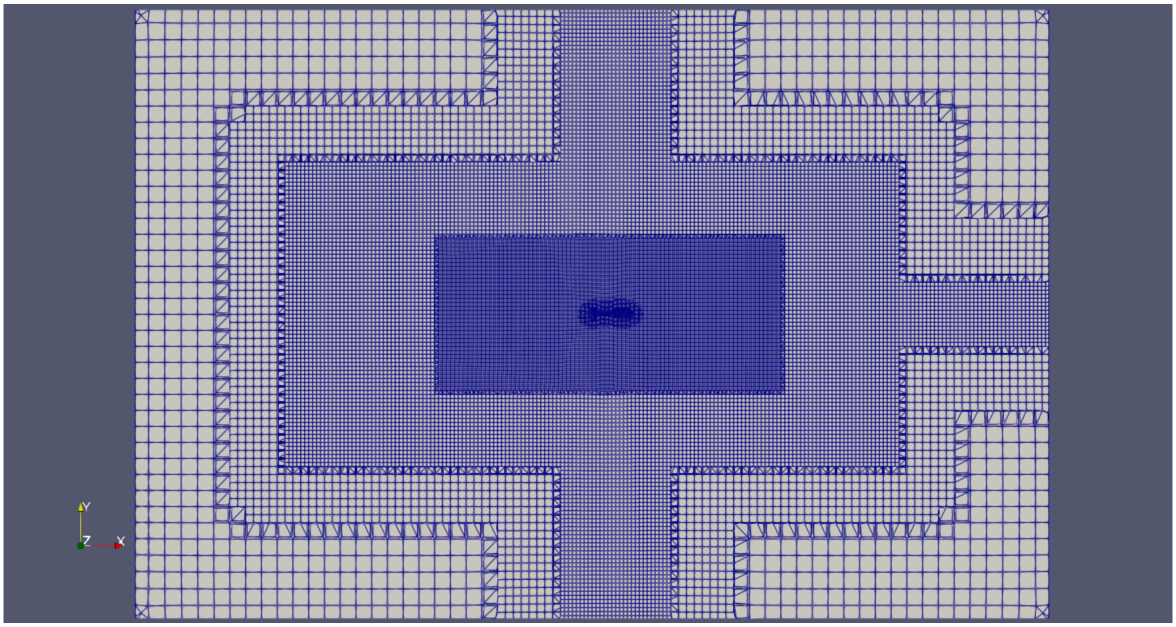

Figure 4: Whole computational domain

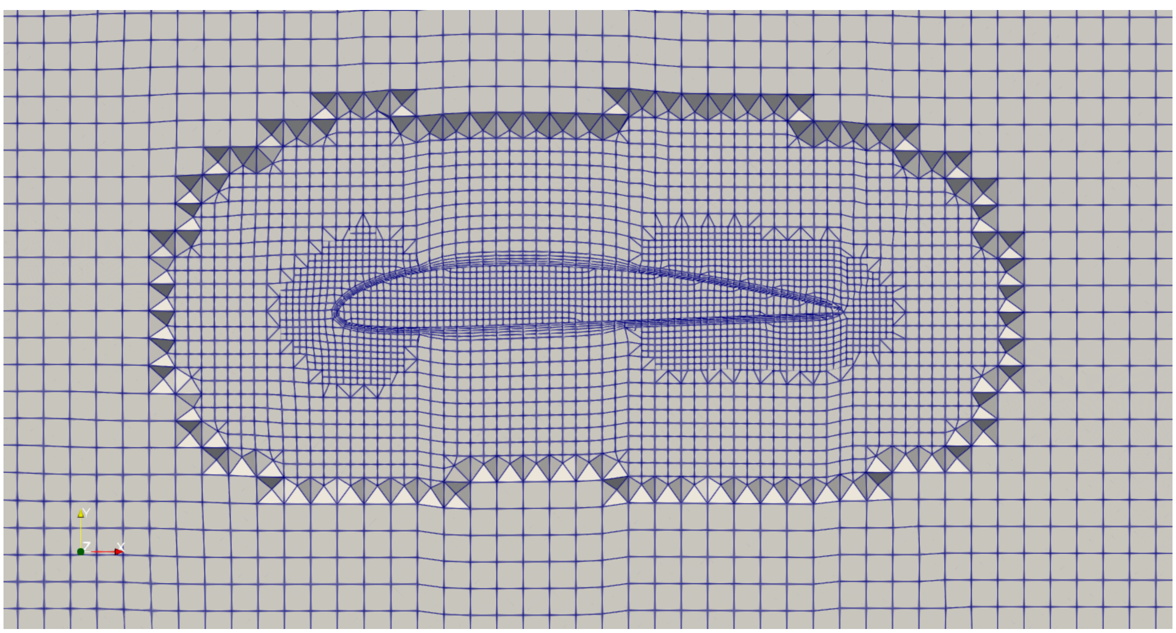

Figure 5: Mesh refinement near wing surface

```
Mesh stats
    points:           2706276
    faces:            7893478
    internal faces:   7825682
    cells:            2594057
    faces per cell:   6.05968
    boundary patches: 5
    point zones:      0
    face zones:       0
    cell zones:       0

Overall number of cells of each type:
    hexahedra:     2533052
    prisms:        3361
    wedges:        0
    pyramids:      0
    tet wedges:    1
    tetrahedra:    0
    polyhedra:     57643
    Breakdown of polyhedra by number of faces:
        faces   number of cells
            4   8
            5   29
            6   6955
            7   4028
            8   81
            9   41906
           10   1
           12   4469
           15   164
           18   2
```

Figure 6: Mesh statistics

```
Checking topology...
    Boundary definition OK.
    Cell to face addressing OK.
    Point usage OK.
    Upper triangular ordering OK.
    Face vertices OK.
    Number of regions: 1 (OK).

Checking patch topology for multiply connected surfaces...
                   Patch    Faces   Points                  Surface topology
                  inlet1     1140     1207  ok (non-closed singly connected)
                  inlet2     8556     8843  ok (non-closed singly connected)
                 outlet1     2094     2215  ok (non-closed singly connected)
               symmetry1    29472    30117  ok (non-closed singly connected)
                   wall1    26534    26856  ok (non-closed singly connected)

Checking geometry...
    Overall domain bounding box (-3 -2 -3.00004) (3 2 0.000117371)
    Mesh has 3 geometric (non-empty/wedge) directions (1 1 1)
    Mesh has 3 solution (non-empty) directions (1 1 1)
    Max cell openness = 6.65268e-16 OK.
    Max aspect ratio = 12.8332 OK.
    Minimum face area = 1.97419e-07. Maximum face area = 0.0181088.  Face area magnitudes OK.
    Min volume = 3.64149e-10. Max volume = 0.00136085.  Total volume = 71.9935.  Cell volumes OK.
    Mesh non-orthogonality Max: 62.1022 average: 4.30248
    Non-orthogonality check OK.
    Face pyramids OK.
    Max skewness = 2.67084 OK.
    Coupled point location match (average 0) OK.

Mesh OK.
```

Figure 7: Mesh quality statistics

$$\nu_t = C_\mu \cdot \frac{k^2}{\epsilon} \tag{4}$$

4.2 Initial and boundary Conditions

The initial conditions are given below:

- $U = (24.94\hat{\imath} + 1.75\hat{\jmath} + 0\hat{k})m/s$
- $p = 0m^2/s^2$
- $\nu_t = 0m^2/s$
- $k = 2.34m^2/s^2$
- $\omega = 13.31s^{-1}$

Here p is pressure per unit density. i.e

$$p = \frac{\bar{p}}{\rho} \tag{5}$$

OpenFOAM uses this quantity instead of pressure in incompressible cases. The inlet velocity has a magnitude of 25m/s in the direction 4° from the X-axis in the XY plane. The angle simulates the angle of attack of the wing.

There are 5 boundaries defined in the geometry. The area highlighted in green and blue in Fig 2 and Fig 3 are the inlets, the one in yellow is outlet and the one in red is the symmetry boundary..

1. inlet1 (blue) - constant velocity inlet
2. inlet2 (green) - far field velocity
3. outlet - constant pressure outlet
4. wing surface - No slip condition with wall functions for k, ω and ν_t.
5. symmetry boundary condition.

5 Results and Discussions

The forces exerted due to external flow around a clark Y airfoil is studied in this paper. The results of the computation are tabulated in Table 2. The pressure and velocity contours are shown in Figure 8 and Figure 9 respectively. The pressure coefficient vs chordwise location curve is given in Figure 10.

Table 2: Resultant force coefficients

Force coefficient	**RANS result**	**XFOIL result**
C_l	0.556	0.83
C_d	0.0409	0.009
Minimum C_p	-0.78	≈ -1.4

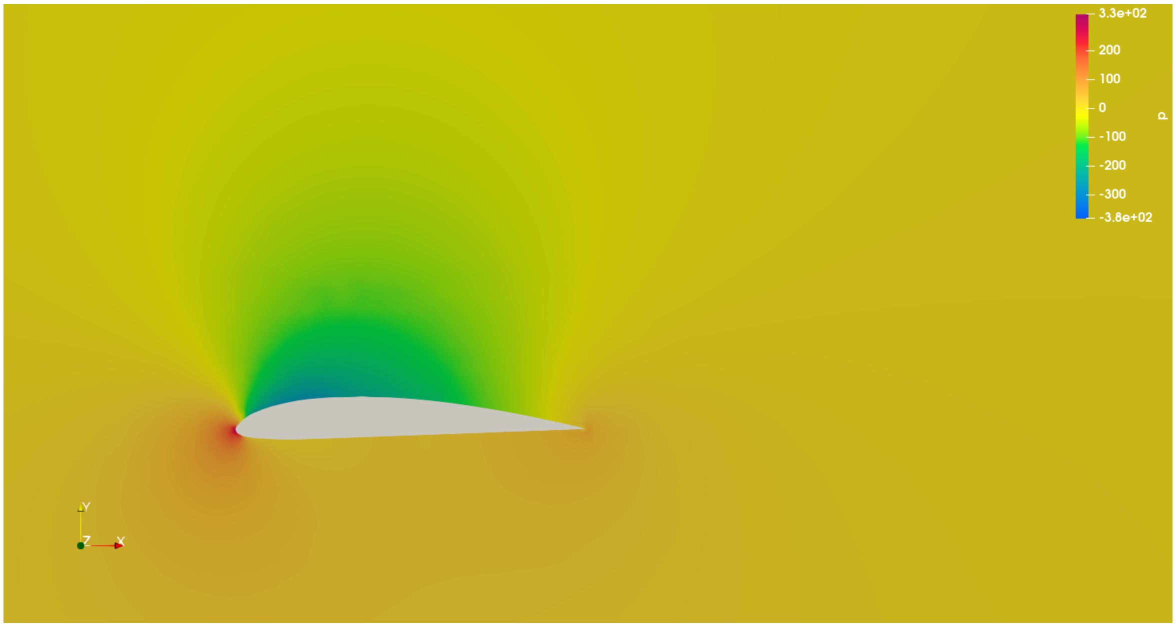

Figure 8: Pressure contours

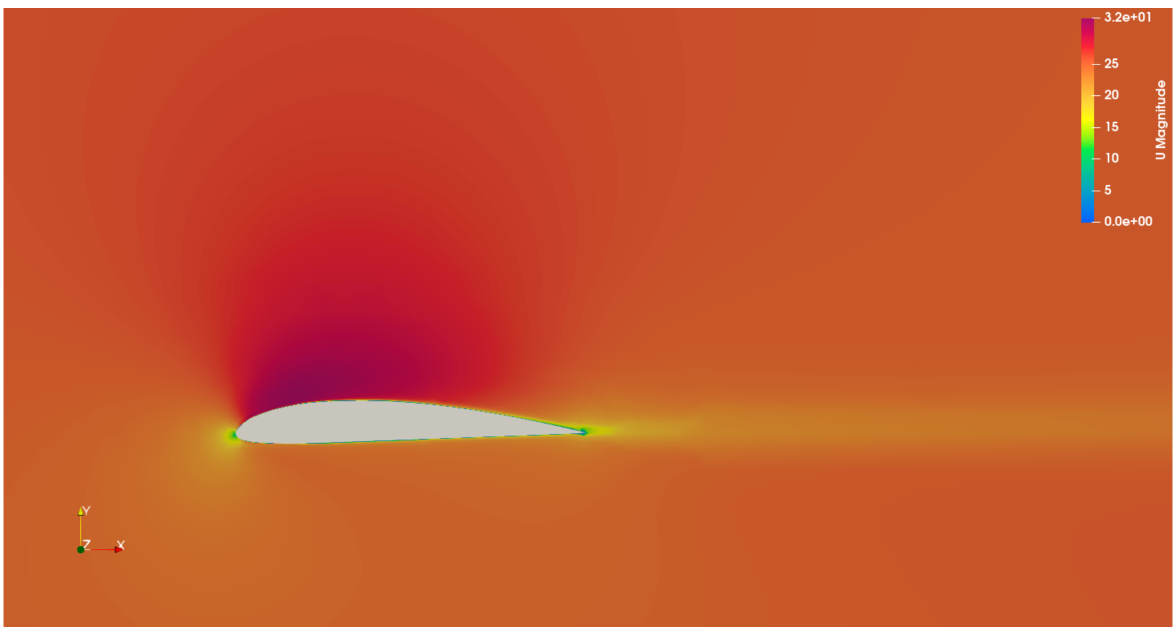

Figure 9: Velocit magnitude contours

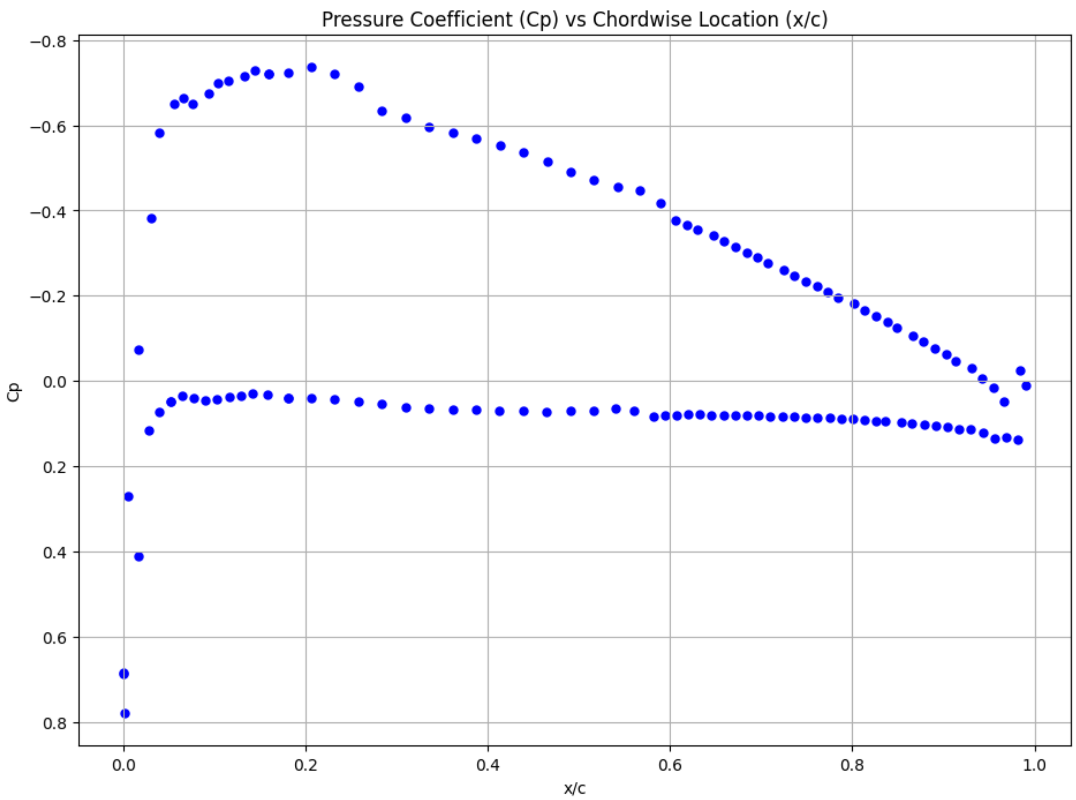

Figure 10: Pressure coefficient vs chordwise location curve

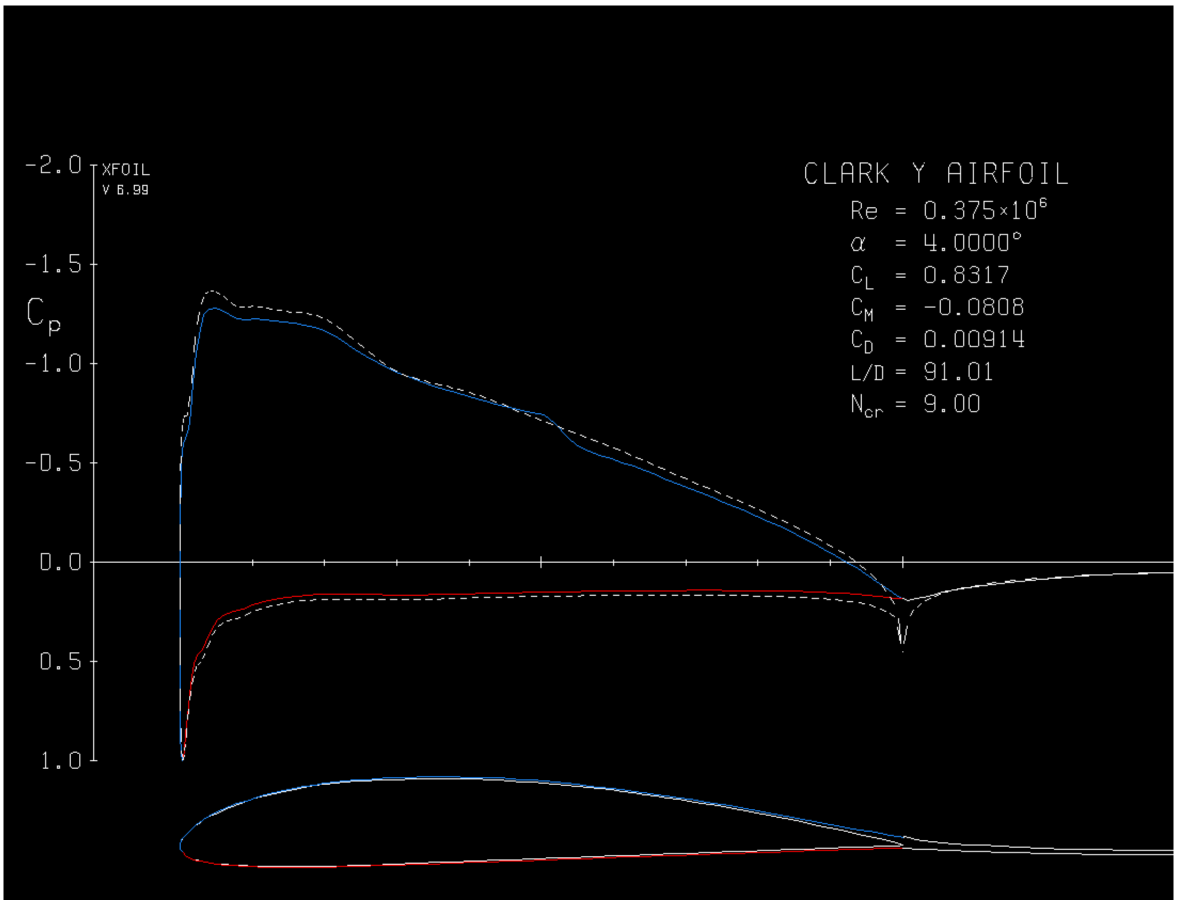

Figure 11: Pressure coefficient vs chordwise location curve exported from xfoil (Re = 375000)

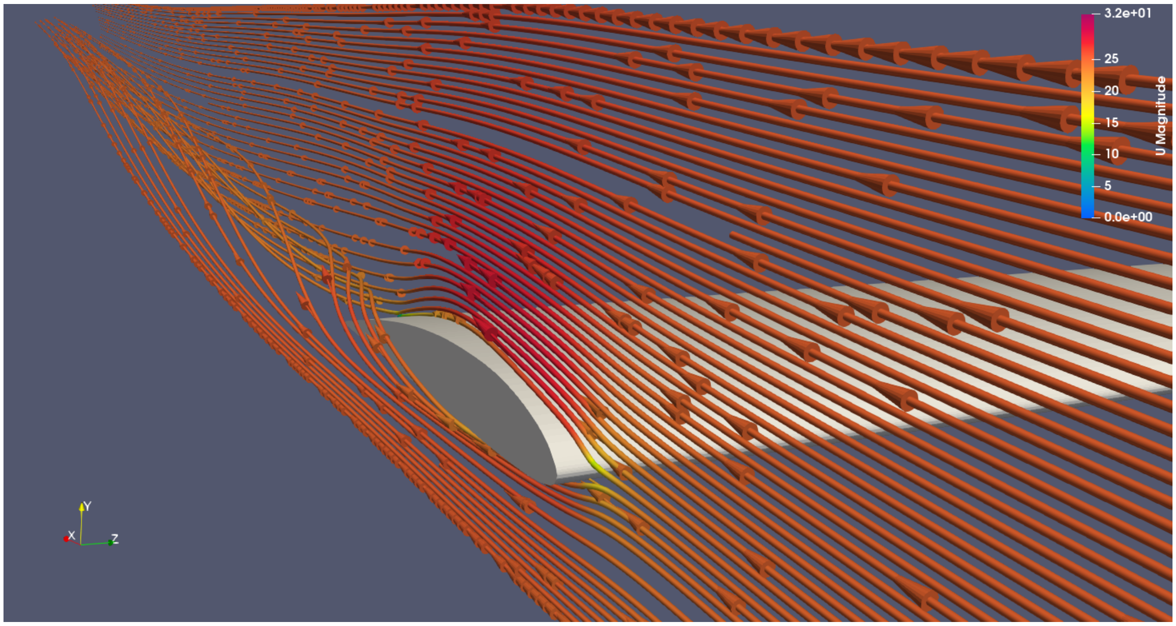

Figure 12: Wing tip votrtices produced in RANS simulation

5.1 Verification of results

To verify our results we compare them to the XFOIL simulation results which are considered as a benchmark. XFOIL is a software for design and analysis of subsonic airfoils.
XFOIL assumes the wing to be for infinite length and hence the results to the RANS simulation cannot be compaired directly. This is due to the formation of wing tip vortices, downwash and induced drag. These are vortices formed at the tip of the wing due to the pressure differential between the upper and lower part of the wing. They have a significan impact on the lift and drag of the wing and are major contributors in loss of lift. The visualization of wing tip vortices produced in the RANS simulation is shown in Figure 12

To compare the results, we need to first estimate the lift coefficient for a finite length wing using the xfoil data. We use the Prandtl's Lifting-Line Theory for the estimation of theoretcial values which accounts for these effects in a finite wing.

5.1.1 Prandtl's Lifting-Line Theory

According to Prandtl's Classical Lifting-Line Theory, the lift curve slope of a finite wing (a) is related to the 2D infinite airfoil slope (a_0) by eq 6:

$$a = \frac{a_0}{1 + \frac{a_0}{\pi \cdot e \cdot AR}} \tag{6}$$

and since:

$$C_l = a \cdot \alpha_{eff} \tag{7}$$

we get:

$$C_{l_{th}} = \frac{C_{l_{2D}}}{1 + \frac{a_0}{\pi \cdot e \cdot AR}} \tag{8}$$

The formula for drag coefficient due to induced drag is given by eq 9:

$$C_{d_i} = \frac{C_{l_{th}}^2}{\pi \cdot e \cdot AR} \tag{9}$$

Where:

- C_{d_i}: Drag coefficient due to induced drag
- $C_{l_{th}}$: Theoretical 3D lift coefficient
- $C_{d_{th}}$: Theoretical 3D drag coefficient
- $C_{l_{2D}}$: 2D lift coefficient
- a: Lift curve slope of the finite wing
- a_0: The theoretical 2D lift curve slope
- e: Span efficiency factor (≈ 1 for wings with ideal elliptical lift distribution)
- AR: aspect ratio
- α_{eff}: effective angle of attack w.r.t. the zero lift angle ($\approx -3.5° to - 4.0°$ considered $-3.8°$)

The aspect ratio of the wing can be found by eq 10.

$$AspectRatio(AR) = \frac{span}{chord} = \frac{1.5}{0.25} = 6 \tag{10}$$

Using the data from XFOIL, the lift slope for Clark Y airfoil is calculated to be 6.1. And considering the span efficiency to be 1, we can estimate the 3D lift coefficient to be:

$$C_{l_{th}} = \frac{0.83}{1 + \frac{6.1}{\pi \cdot 1 \cdot 6}} \Rightarrow C_{l_{th}} = 0.627$$

Similarly we can find the drag coefficient due to induced drag:

$$C_{d_{th}} = \frac{0.63^2}{\pi \cdot 1 \cdot 6} + 0.009 \Rightarrow C_{d_{th}} = 0.03$$

Here we can see that:

- The lift coefficient of the wing is 0.556 which is close to the theoretical prediction of 0.63 using XFOIL. The discrepancy is likely due to mesh resolution and non-ideal span efficiency.
- The drag coefficient obtained from the RANS simulation is 0.0409, compared to the theoretical prediction of 0.03 (derived from XFOIL with induced drag corrections). This represents an over-prediction of approximately 36%.

  This discrepancy is primarily attributed to the mesh resolution at the wall ($y^+ \approx 22$). By failing to resolve the viscous sublayer ($y^+ < 1$) or target the log-law region ($y^+ > 30$), the simulation likely overestimated the wall shear stress, leading to an inflated skin friction component of the total drag. The non-ideal span efficiency and potential domain confinement effects are secondary contributors to this error.

5.1.2 Pressure coefficient

The shape of the C_p vs x/c curve matches with the simulation results from XFOIL (Figure 11) with the same minima. This shows that the RANS simulation of the flow has the same behaviour as the XFoil simulation.

The higher value of C_p at the minima compared to the XFOIL simulation can be attributed to the low resolution of the mesh used as discussed earlier in section 3.2.

Nomenclature

α	Angle of Attack
α_{eff}	effective angle of attack
$\bar{p}$	Static pressure
ϵ	Turbulence dissipation rate
μ	Model constant for standard k-ϵ model
ν	Air kinematic viscosity
ν_t	Turbulent viscosity
ρ	Density
a	Lift curve slope of the finite wing
a_0	The theoretical 2D lift slope
AR	aspect ratio
C_d	Drag coefficient
C_l	Lift coefficient
C_p	Pressure coefficient
C_{d_0}	Profile Drag
C_{d_i}	Drag coefficient due to induced drag
$C_{d_{th}}$	Theoretical 3D drag coefficient
$C_{l_{2D}}$	2D lift coefficient
$C_{l_{th}}$	Theoretical 3D lift coefficient
e	Span efficiency factor
k	Turbulent kinetic energy
L	Characteristic length scale
l	Turbulence length scale

l_s	Wing span
p	Pressure per unit density
T_i	Turbulence intensity
U	Inlet flow velocity
c	Chord Length