External flow around a straight rectangular wing using OpenFOAM

Harshit Dhiman

January 8, 2026

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 $\rightarrow$ air

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

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

- Wing dimensions and orientation

  Angle of attack, $\alpha = 4^\circ$

  Span, $l_s = 1.5m$

  Chord length, $c = 0.25m$

  Cross-section $\rightarrow$ Clark Y airfoil

The goal is to calculate 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

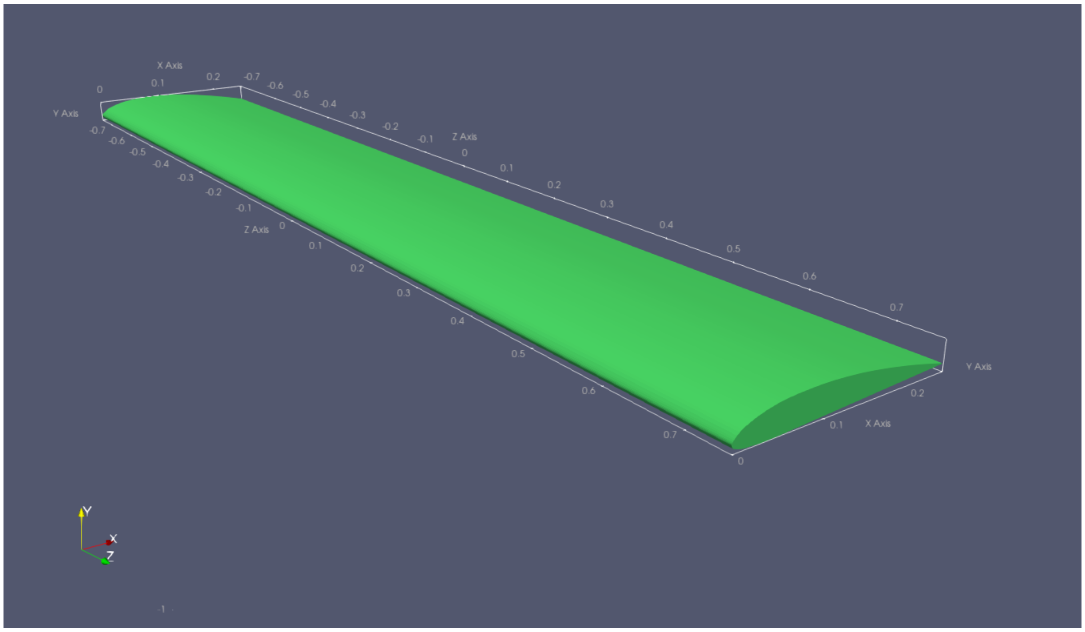

Figure 1: Wing Geometry

Table 1: Dimensions of the computational domain

$x(m)$	$y(m)$	$z(m)$	**Distance of wing from inlet** (m)
6.6	4	1.5	1.5

3 Numerical Methodology

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 = 0.05m.
- Mesh refinement is applied on wing surface. The properties of refinement done are listed below:

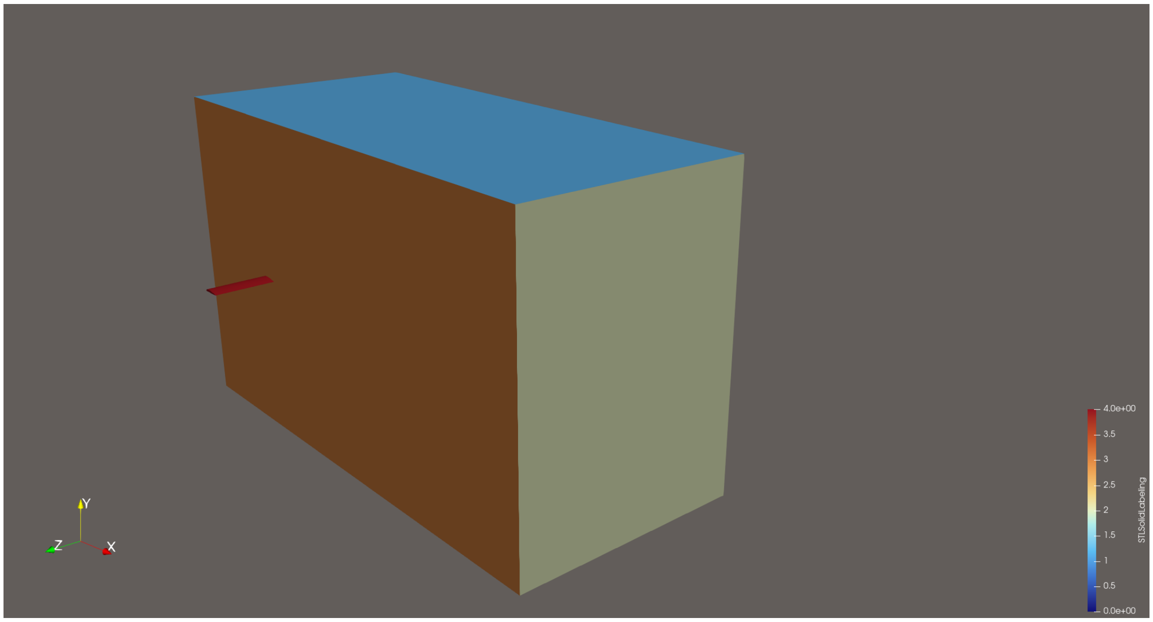

Figure 2: Fluid domain showing outlet

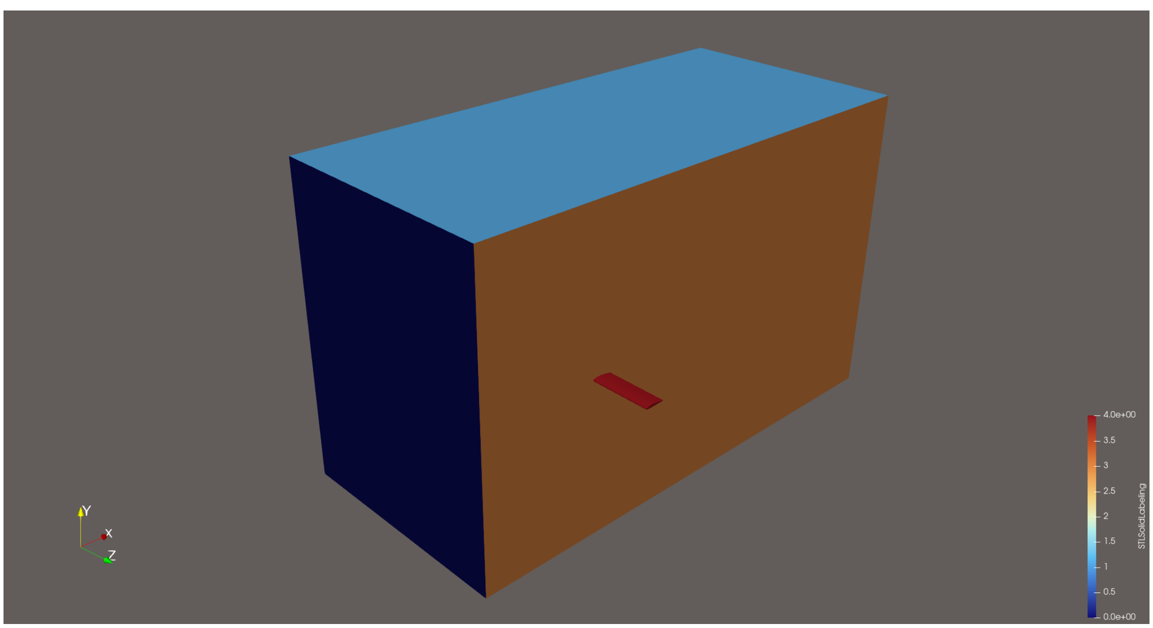

Figure 3: Fluid Domain showing inlet

- Surface refinement → level 5
- Edge refinement → level 6
- Inflation layers are added to the wing surface targeting a y^+ value of ≈ 1, to resolve boundary layer flow. Below are the properties of the layers added.
  * Final layer thickness → 0.00001766 m absolute size
  * Expansion ration → 1.3
  * Number of layers → 12

- A cuboidal region is defined in the wake of the wing for refinement. The level of refiment in the wake region is level 3.
- 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 created is $\approx 14.7M$ cells large. The mesh statistics are shown in Figure 6 and the mesh quality statistics are shown in Figure 7. Due to a small number of cells having non-orthogonality above 70, non-orthogonal correctors were used in the solver.

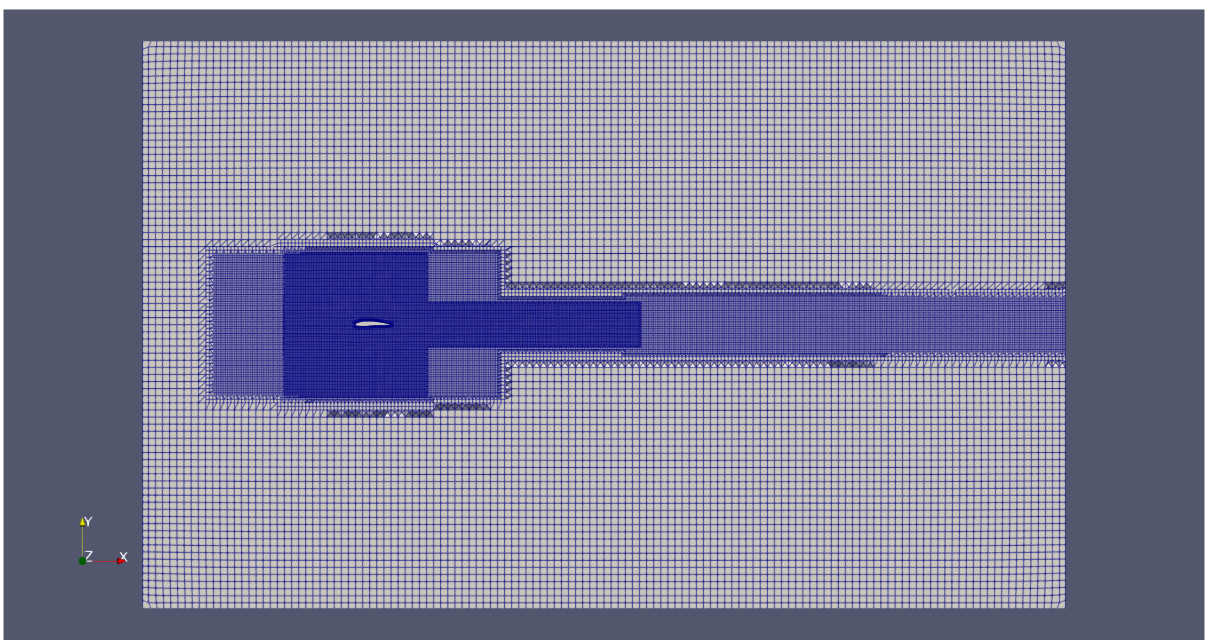

Figure 4: Whole computational domain

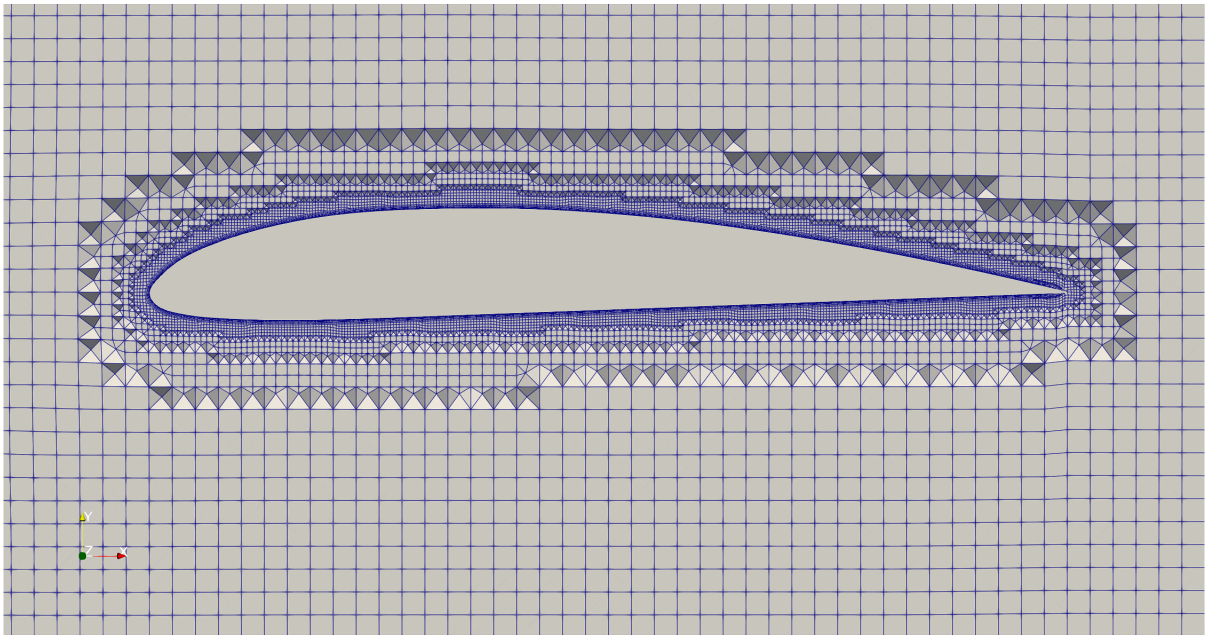

Figure 5: Mesh refinement near wing surface

```
Mesh stats
    points:           15757564
    faces:            45202554
    internal faces:   44088639
    cells:            14727620
    faces per cell:   6.062839277
    boundary patches: 20
    point zones:      0
    face zones:       0
    cell zones:       0
```

Figure 6: Mesh statistics

3.3 Numerical Model

The numerical model used is the steady state $k - \omega SST$ model with an assumed 5% turbulence intensity ($T_i = 0.05$), chosen to represent a fully turbulent freestream condition.

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

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

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

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

```
Checking geometry...
    Overall domain bounding box (-1.500001013 -2.000003332 -1.600000087) (5.000006665 2.000006758 0)
    Mesh has 3 geometric (non-empty/wedge) directions (1 1 1)
    Mesh has 3 solution (non-empty) directions (1 1 1)
    Boundary openness (1.346088862e-16 1.307477891e-16 -2.383328462e-16) OK.
    Max cell openness = 2.925720399e-15 OK.
    Max aspect ratio = 55.7007317 OK.
    Minimum face area = 2.458578999e-09. Maximum face area = 0.007329841363.  Face area magnitudes OK.
    Min volume = 7.563169067e-12. Max volume = 0.0002307535402.  Total volume = 41.59620633.  Cell volumes OK.
    Mesh non-orthogonality Max: 74.45387584 average: 4.860934628
   *Number of severely non-orthogonal (> 70 degrees) faces: 3.
    Non-orthogonality check OK.
  <<Writing 3 non-orthogonal faces to set nonOrthoFaces
    Face pyramids OK.
    Max skewness = 2.487262896 OK.
    Coupled point location match (average 0) OK.

Mesh OK.
```

Figure 7: Mesh quality statistics

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

3.4 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 kinematic pressure i.e.

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

OpenFOAM uses kinematic pressure 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 (Dark blue) - constant velocity inlet
2. inlet2 (Light blue) - far field velocity
3. outlet - constant pressure outlet

4. wing surface:
    - U - no-slip condition
    - ω - omegaWallFunction
    - ν_t - nutUSpaldingWallFunction
5. symmetry boundary condition.

4 Results and Discussions

4.1 Flow Visualization

The forces exerted due to external flow around a Clark Y airfoil is studied in this paper. The pressure and velocity contours are shown in Figure 8 and Figure 9 respectively.

Figure 8: Pressure contours

4.2 Aerodynamic Forces

The results of the computation are tabulated in Table 2.

Table 2: Resultant force coefficients

Force coefficient	**RANS result**	**XFOIL result (wieghted for finite wing)**
C_l	0.5024	0.58
C_d	0.0311	0.0268
Minimum C_p	-0.792	≈ -1.4

Figure 9: Velocit magnitude contours

4.3 Pressure coefficient

The pressure coefficient vs chordwise location curve obtained by the RANS simulation is given in Figure 11. The shape of the C_p vs x/c curve matches with the simulation results from XFOIL (Figure 10) with the same minima. This shows that the RANS simulation of the flow has the same behaviour as the XFoil simulation.

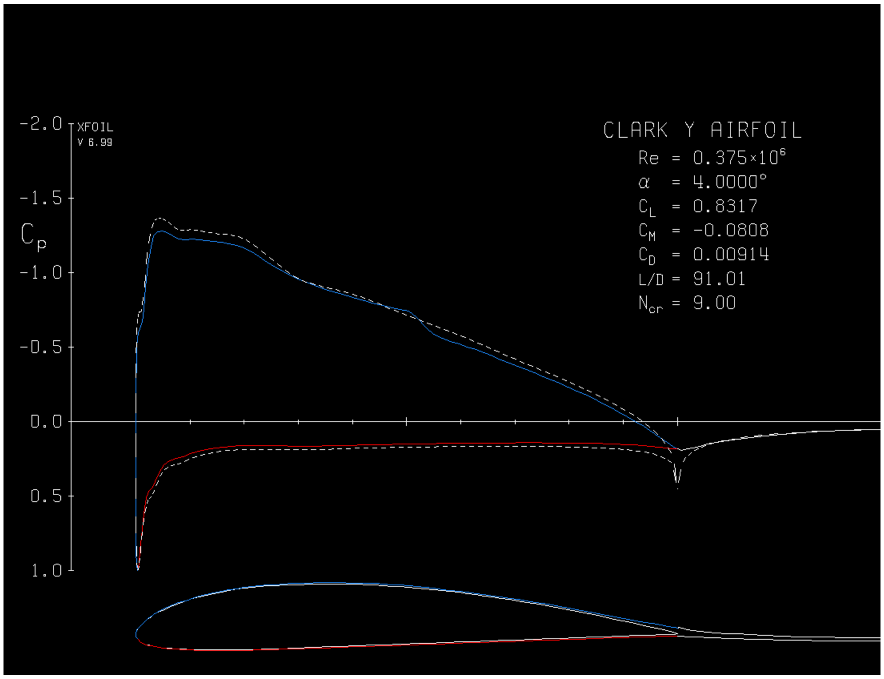

Figure 10: Pressure coefficient vs chordwise location curve exported from xfoil (Re = 375000)

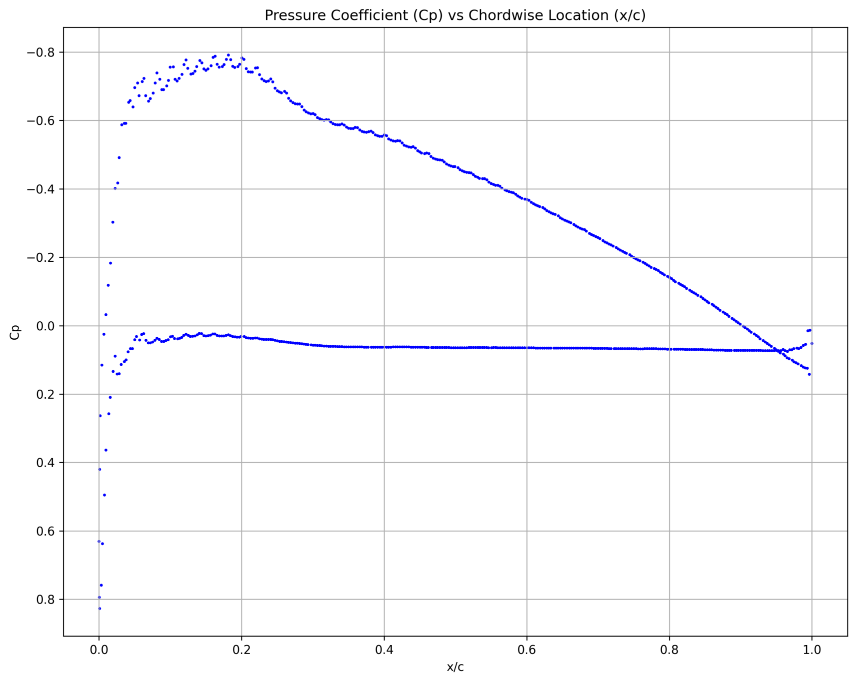

Figure 11: Pressure coefficient vs chordwise location curve

4.4 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 compared directly. This is due to the formation of wing tip vortices, downwash and induced drag. Wingtip vortices 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 significant 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 theoretical values which accounts for these effects in a finite wing.

4.4.1 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:

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

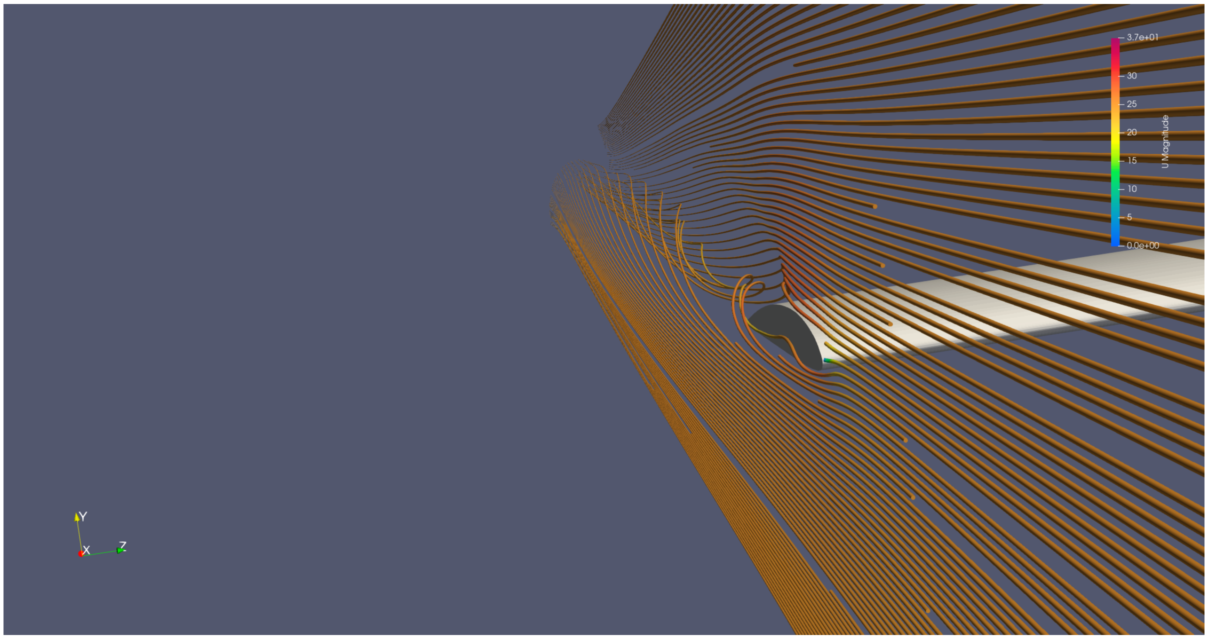

Figure 12: Wing tip votrtices produced in RANS simulation

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:

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

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
- 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 0.75(estimate for a rectangular wing with moderate aspect ratio), we can estimate the 3D lift coefficient to be:

$$C_{l_{th}} = \frac{0.83}{1 + \frac{6.1}{\pi \cdot 0.75 \cdot 6}} \Rightarrow C_{l_{th}} = 0.58$$

Similarly we can find the total drag coefficient as induced drag + profile drag(0.009 as per the XFOIL simulation):

$$C_{d_{th}} = \frac{0.502^2}{\pi \cdot 0.75 \cdot 6} + 0.009 \Rightarrow C_{d_{th}} = 0.0178 + 0.009 = 0.0268$$

5 Conclusions

This study analyzed the external aerodynamic flow over a finite rectangular wing with a Clark Y airfoil using a steady-state RANS approach ($k-\omega$ SST turbulence model). The simulation results were verified against theoretical predictions derived from XFOIL data and Prandtl's Lifting-Line Theory.

- **Lift Coefficient(C_l)**: The RANS simulation predicts a C_l of 0.502 which is a $\approx$ 13.5% under-prediction compared to the theoretical prediction of 0.58 using XFOIL. The discrepancy can be attributed to the below reasons:
  - The RANS simulation assumes a fully turbulent flow, making the boundary layer at the trailing edge much larger compared to the XFOIL model which assumes laminar flow for the initial part of the wing.
  - The RANS simulation is a fully 3D model, which captures the wing tip vortices much more realistically compared to the lifting line theory which uses a clumped value of the span efficiency.
- **Drag Coefficient(C_d)**: The RANS simulation predicts a C_d of 0.0311, compared to the theoretical prediction of 0.0268 (derived from XFOIL with induced drag corrections). This represents an over-prediction of approximately 16%.
  - The deviation compared to the XFOIL approximation is low, showing that the viscous sublayer has been resolved correctly. This is also backed by the fact that the average y^+ value for the simulation is $\approx$ 1.48 which is squarely in the linear region of the boundary layer.
  - This over-predictions also shows that the RANS simulation predicts a larger profile drag component ($C_{d,profile} = C_{d,actual} - C_{d,induced} = 0.0311 - 0.0178 = \mathbf{0.0133}$) than the XFOIL simulation. This is consistent with the fully turbulent assumption for the $k-\omega SST$ RANS simulation, which generates higher skin friction coefficients compared to the laminar/transitional flow regimes often present in 2D XFOIL estimations.

In conclusion, the OpenFOAM simulation successfully captures the fundamental aerodynamics of a finite wing. While the fully turbulent $k-\omega$ SST model yields conservative estimates for lift and drag compared to 2D transitional codes, the results are physically consistent and valid for a fully turbulent flight regime.