Solution Methodology

The simulation was carried out using ANSYS Fluent to analyze steady, incompressible, laminar flow over a NACA 0012 airfoil. A two-dimensional computational domain was generated using a C-type mesh topology to ensure accurate resolution of the boundary layer and wake region. The airfoil chord length was taken as 1 meter.

The boundary conditions were defined as follows: a velocity inlet with a uniform freestream velocity of 25 m/s, a pressure outlet with 0 Pa gauge pressure, symmetry boundaries at the top and bottom to mimic an unbounded domain, and a no-slip wall condition at the airfoil surface. Simulations were performed for two different angles of attack: 0° and 5°. The change in AOA was implemented by resolving the inlet velocity into horizontal and vertical components based on the angle.

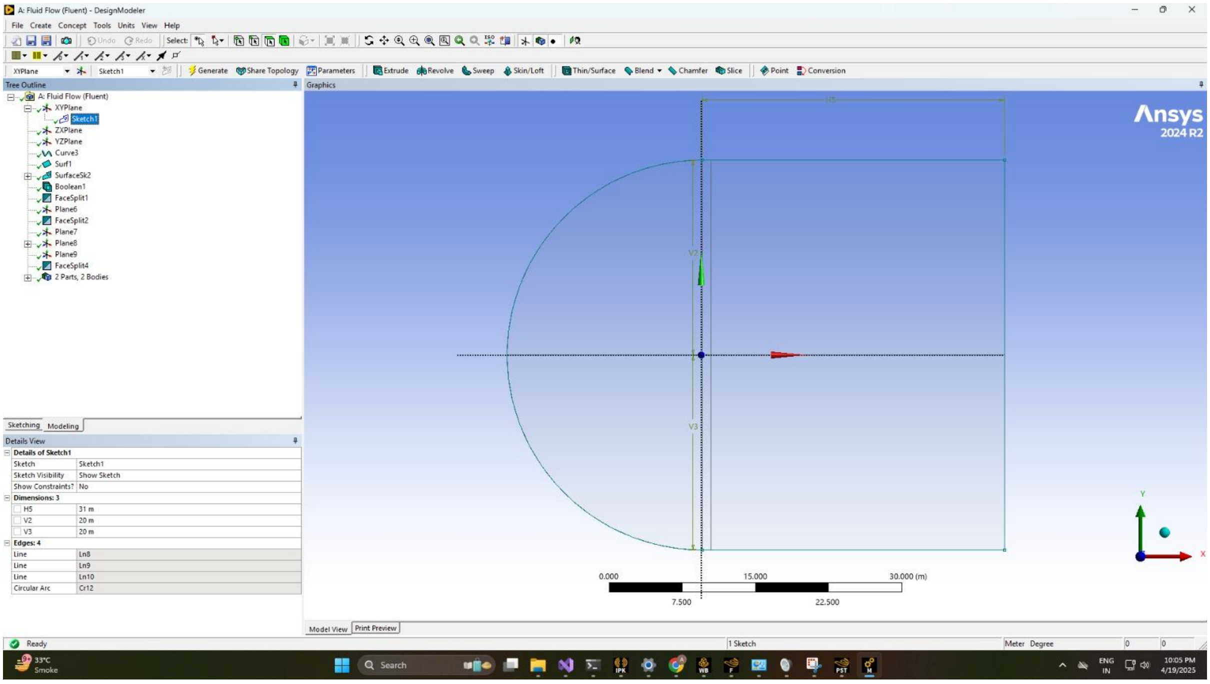
The pressure-based solver in ANSYS Fluent was used with a steady-state formulation. Pressure-velocity coupling was handled using the SIMPLE algorithm. Momentum equations were discretized using a second-order upwind scheme to ensure higher accuracy. The simulation was run until all residuals fell below 10^{-6}, indicating a converged solution.

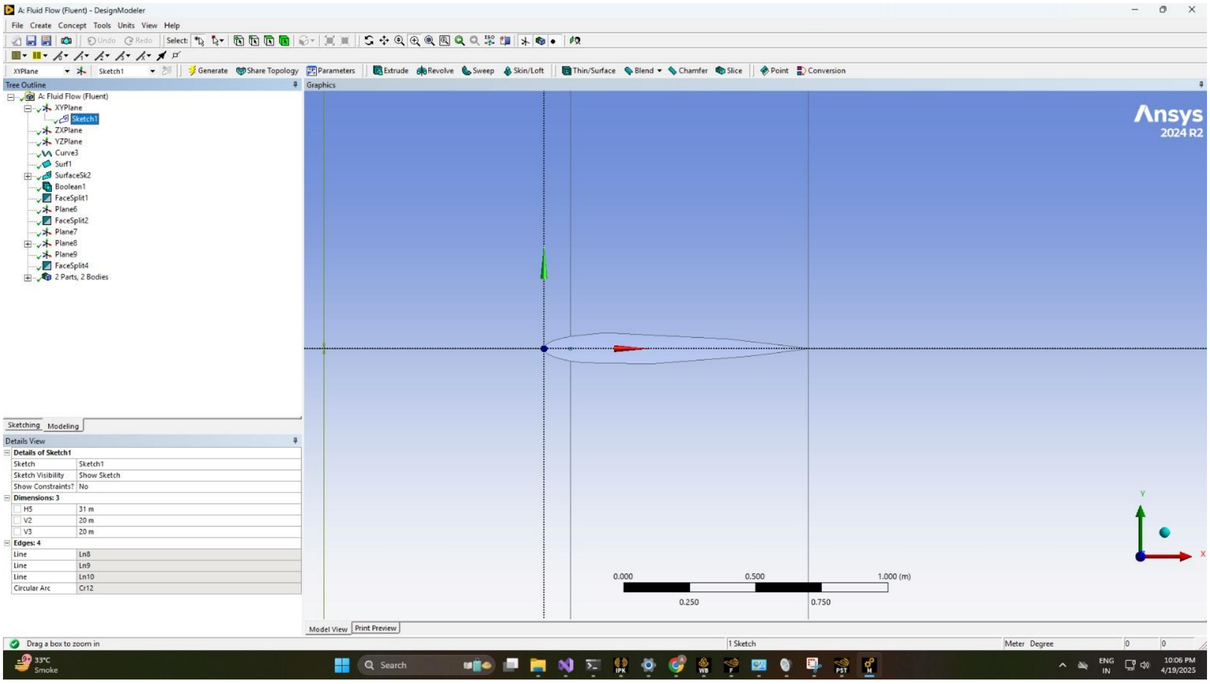
Post-processing was conducted to evaluate the aerodynamic coefficients of drag (CD) and lift (CL), and to visualize the flow behaviour using contours of velocity magnitude, static pressure, and streamlines. These contours were analyzed to identify key flow features such as pressure gradients, boundary layer development, and flow separation regions if any.

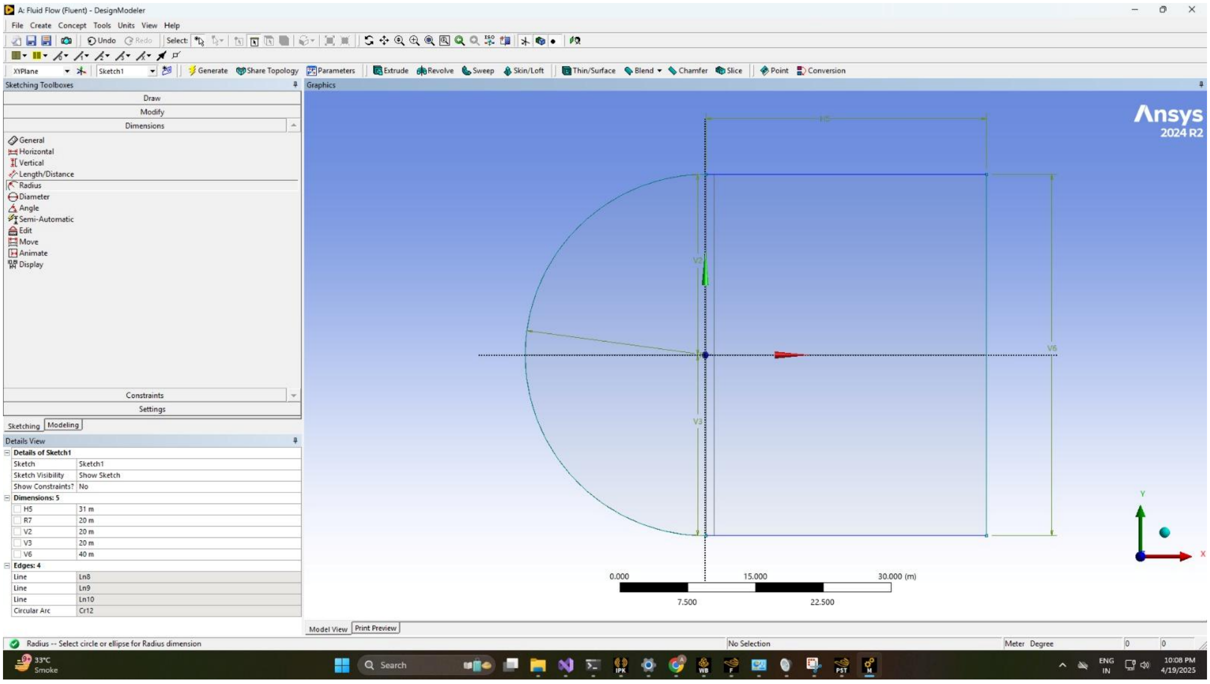
For the 5° angle of attack case, the lift and drag coefficient plots versus iteration number appeared noisy when using the laminar flow model. To obtain smoother and more stable results, the k-omega SST model was employed instead.

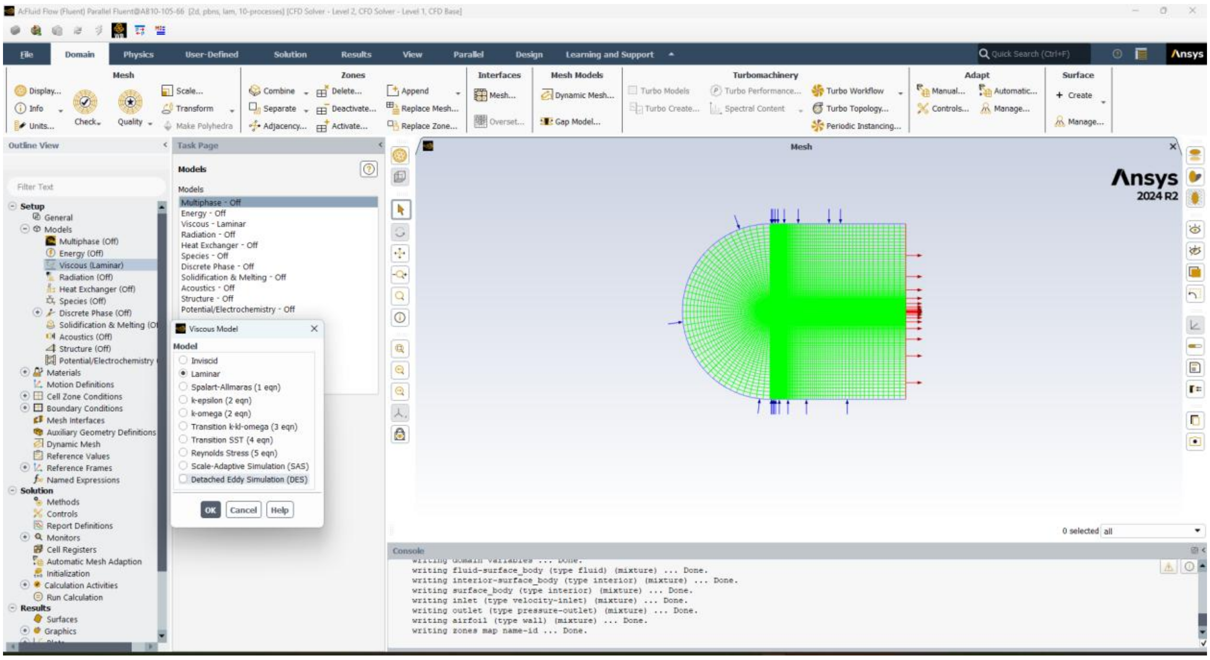
The mesh discretization procedure was iteratively refined until a sufficiently fine and appropriate grid was achieved for our analysis.

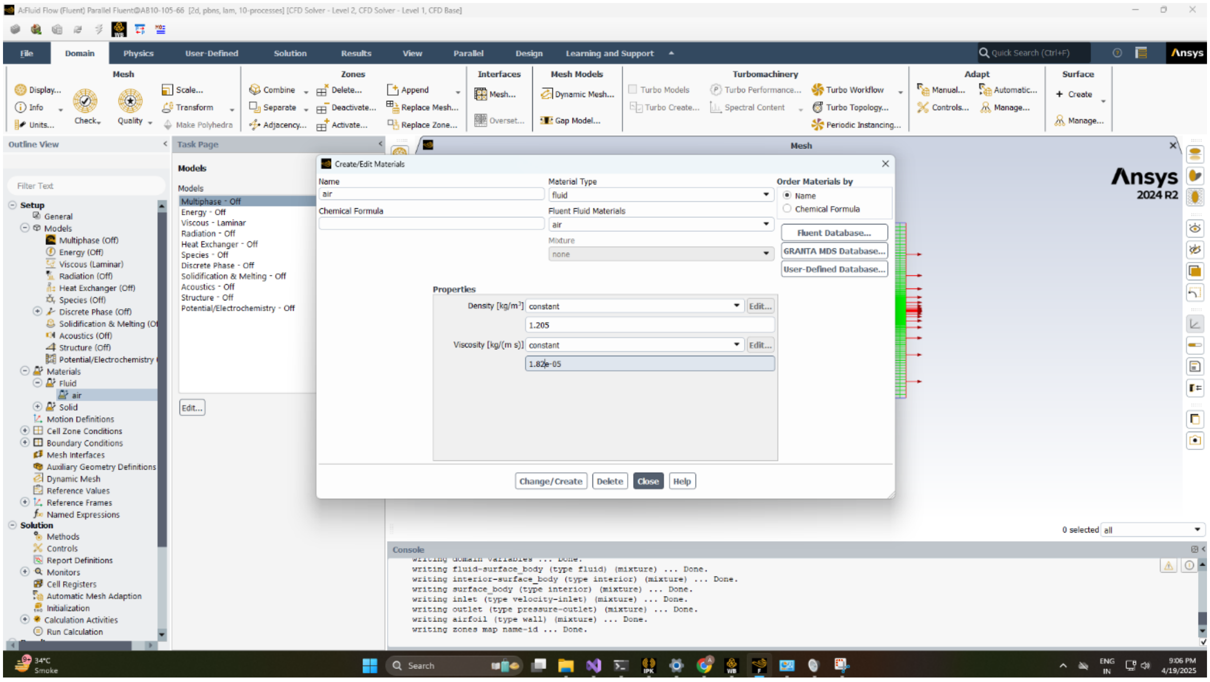
Geometry and Boundary Conditions:

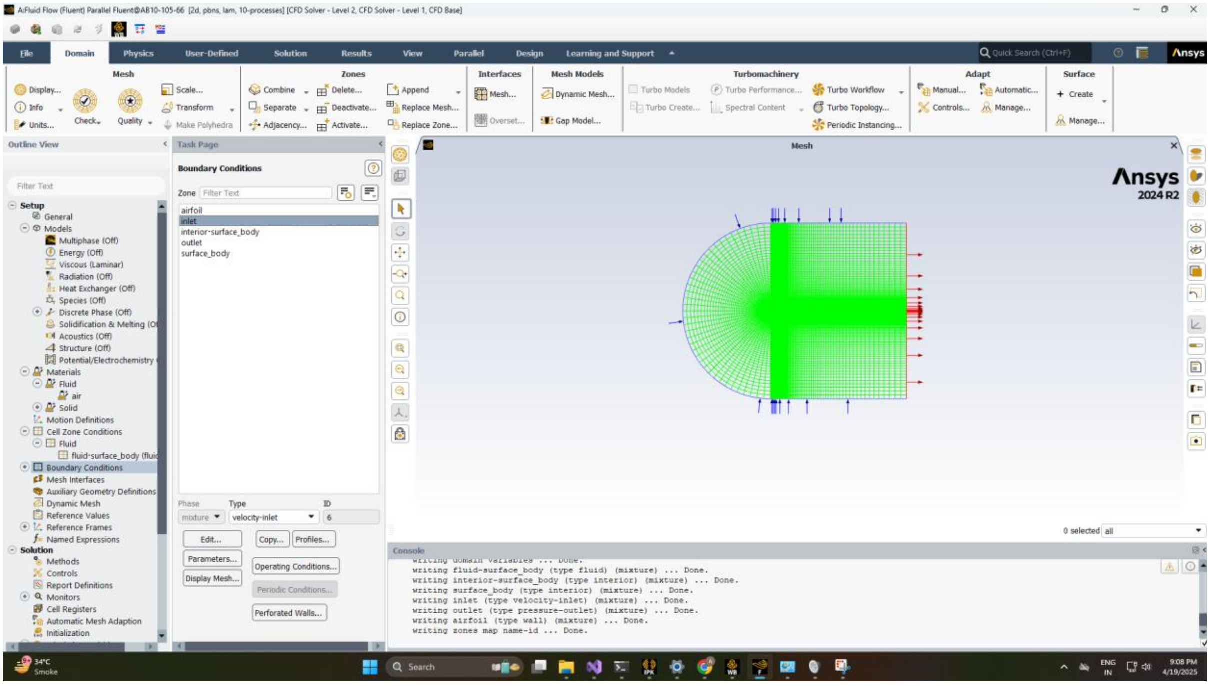

A: Fluid Flow (Fluent) - DesignModeler
Ansys
2024 R2
Details of Sketch1
H5 31 m
V2 20 m
V3 20 m
0.000
15.000
30.000 (m)
7.500
22.500

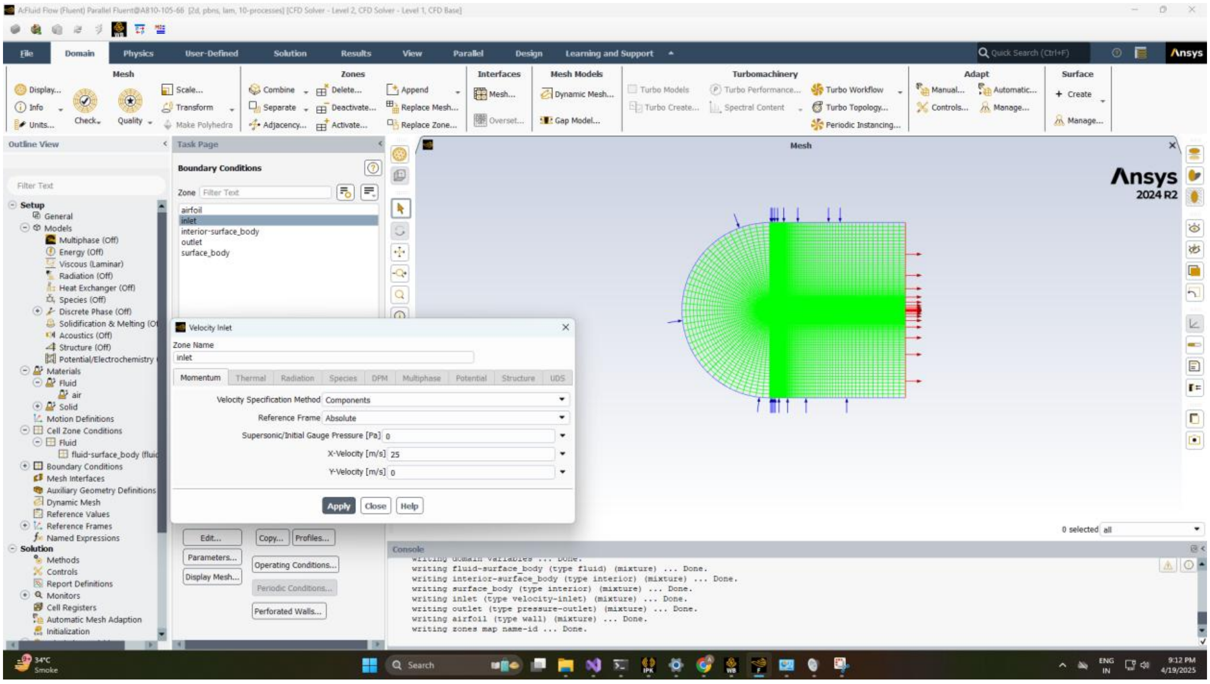

A: Fluid Flow (Fluent) - DesignModeler
Ansys
2024 R2
Details of Sketch1
H5 31 m
V2 20 m
V3 20 m
0.000
0.500
1.000 (m)
0.250
0.750

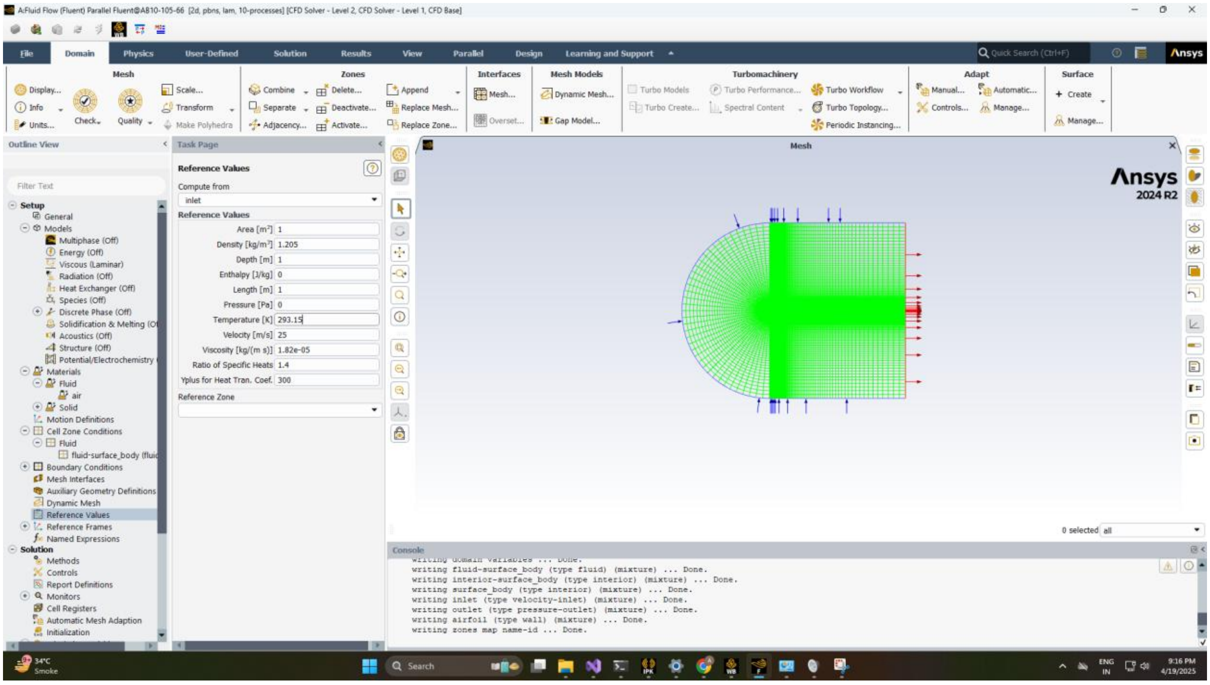


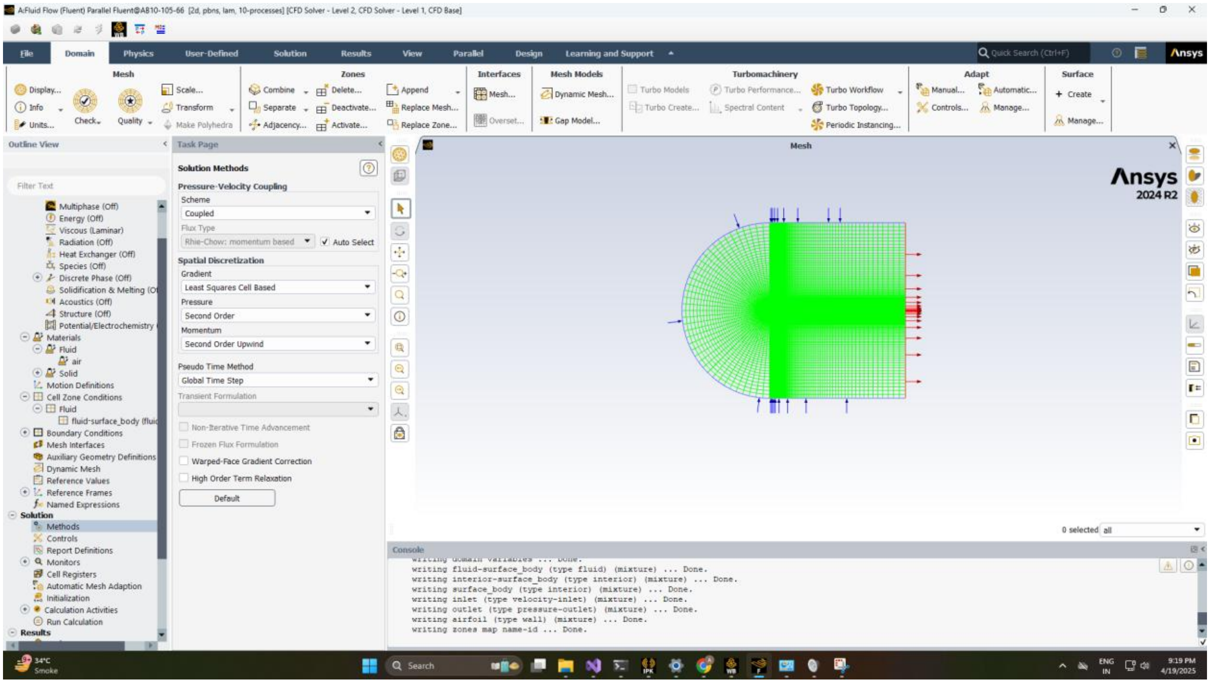


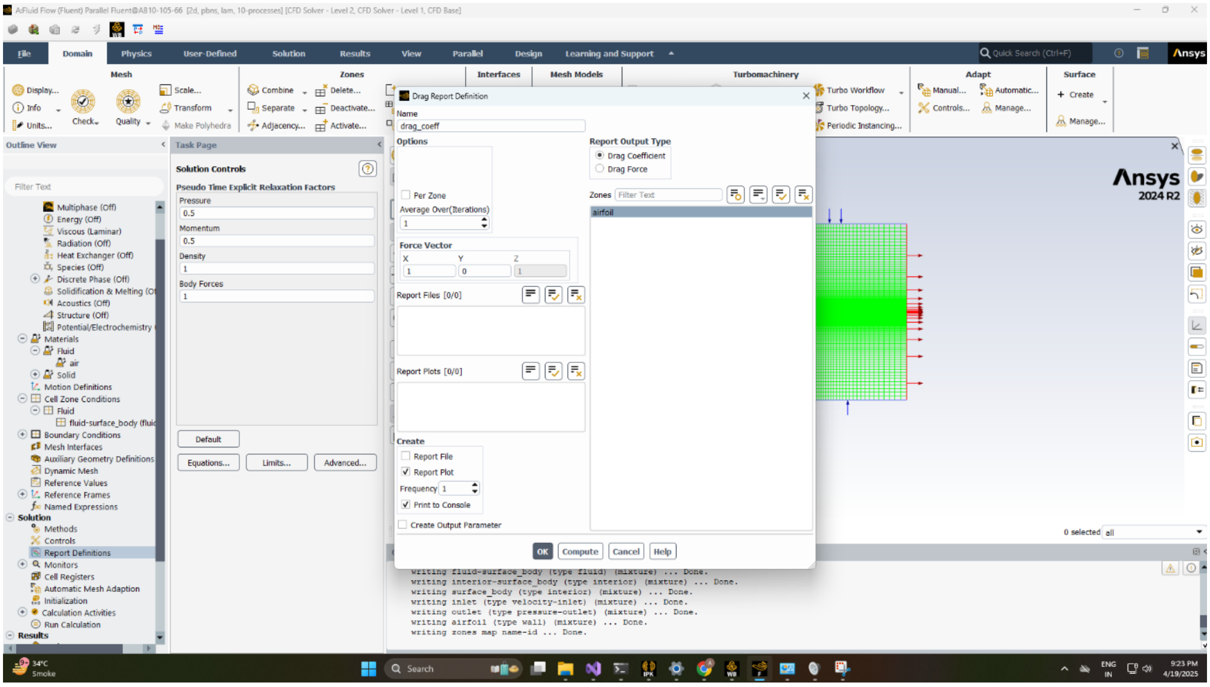

Create/Edit Materials
Name
air
Material Type
fluid
Order Materials by
Name
Chemical Formula
Fluent Fluid Materials
air
Mixture
none
Fluent Database...
GRANTA MDS Database...
User-Defined Database...
Properties
Density [kg/m³]
constant
1.205
Viscosity [kg/(m s)]
constant
1.82e-05
Change/Create
Delete
Close
Help

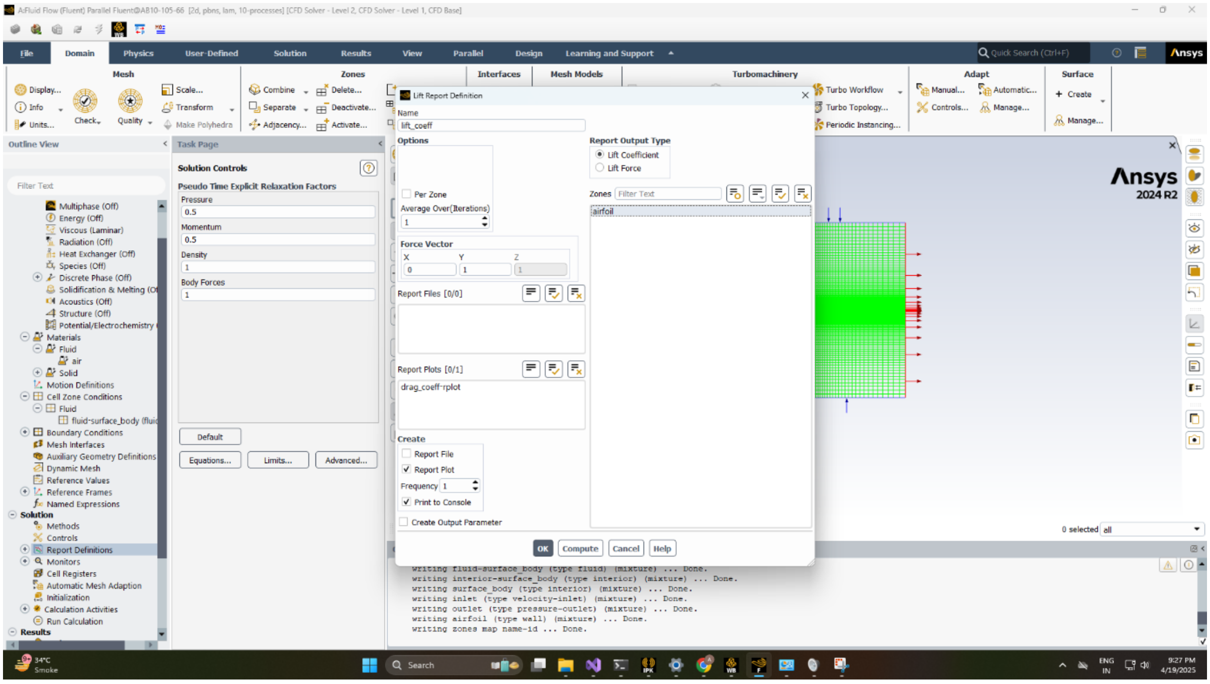

Boundary Conditions
Zone
airfoil
inlet
interior-surface_body
outlet
surface_body
Phase
mixture
Type
velocity-inlet
ID
6
Edit...
Copy...
Profiles...
Parameters...
Operating Conditions...
Display Mesh...
Periodic Conditions...
Perforated Walls...
Mesh
Ansys
2024 R2

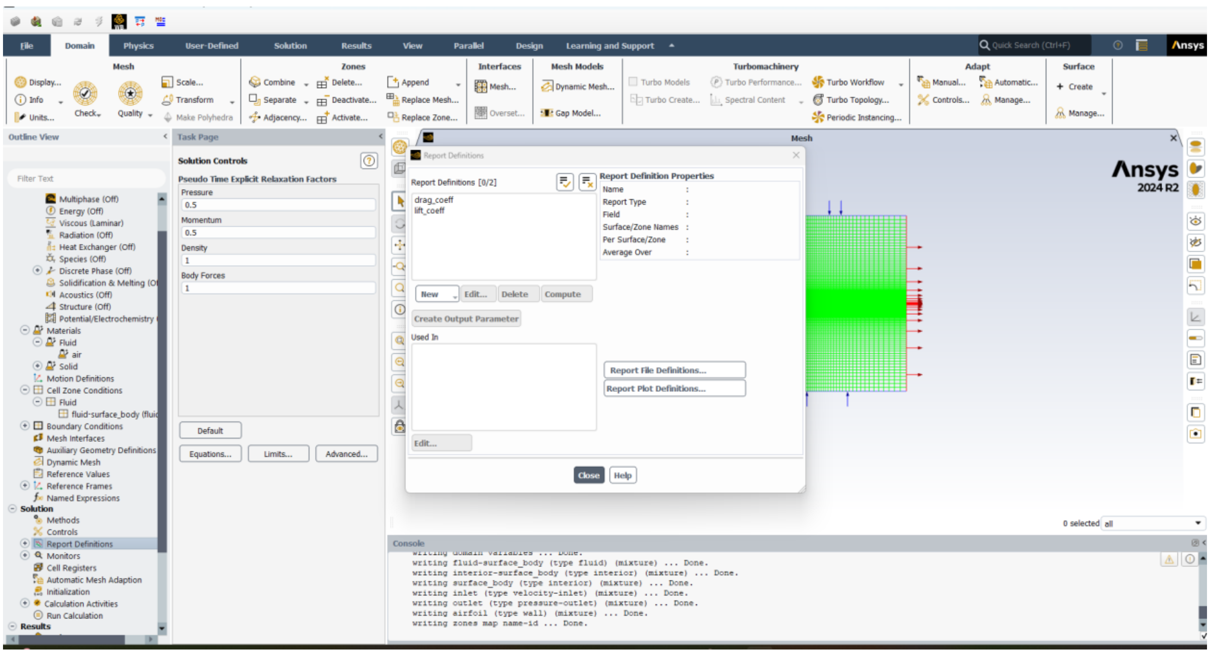

Velocity Inlet
Zone Name
inlet
Velocity Specification Method Components
Reference Frame Absolute
Supersonic/Initial Gauge Pressure [Pa] 0
X-Velocity [m/s] 25
Y-Velocity [m/s] 0
Apply
Close
Help

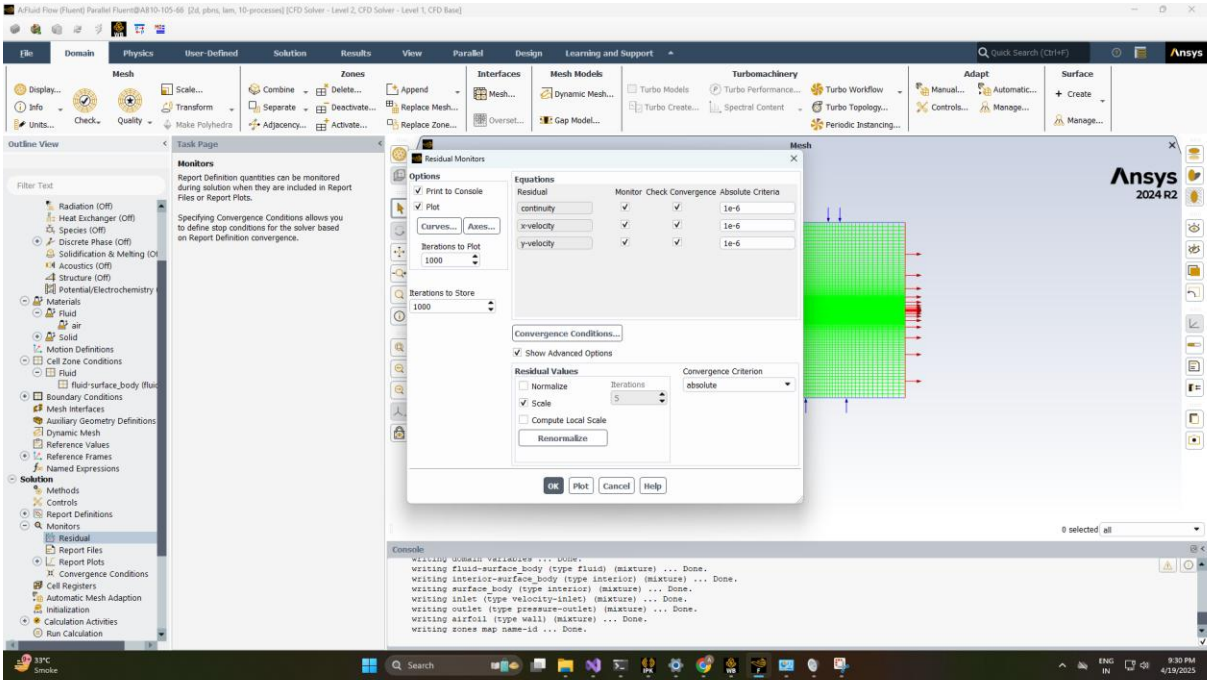

Reference Values
Compute from
inlet
Area [m²] 1
Density [kg/m³] 1.205
Depth [m] 1
Enthalpy [J/kg] 0
Length [m] 1
Pressure [Pa] 0
Temperature [K] 293.15
Velocity [m/s] 25
Viscosity [kg/(m s)] 1.82e-05
Ratio of Specific Heats 1.4
Yplus for Heat Tran. Coef. 300
Reference Zone


Solution Methods
Pressure-Velocity Coupling
Scheme
Coupled
Flux Type
Rhie-Chow: momentum based
Auto Select
Spatial Discretization
Gradient
Least Squares Cell Based
Pressure
Second Order
Momentum
Second Order Upwind
Pseudo Time Method
Global Time Step
Transient Formulation
Non-Iterative Time Advancement
Frozen Flux Formulation
Warped-Face Gradient Correction
High Order Term Relaxation
Default
Mesh
Ansys
2024 R2
Console
writing fluid-surface_body (type fluid) (mixture) ... Done.
writing interior-surface_body (type interior) (mixture) ... Done.
writing surface_body (type wall) (mixture) ... Done.
writing inlet (type velocity-inlet) (mixture) ... Done.
writing outlet (type pressure-outlet) (mixture) ... Done.
writing airfoil (type wall) (mixture) ... Done.
writing zones map name-id ... Done.

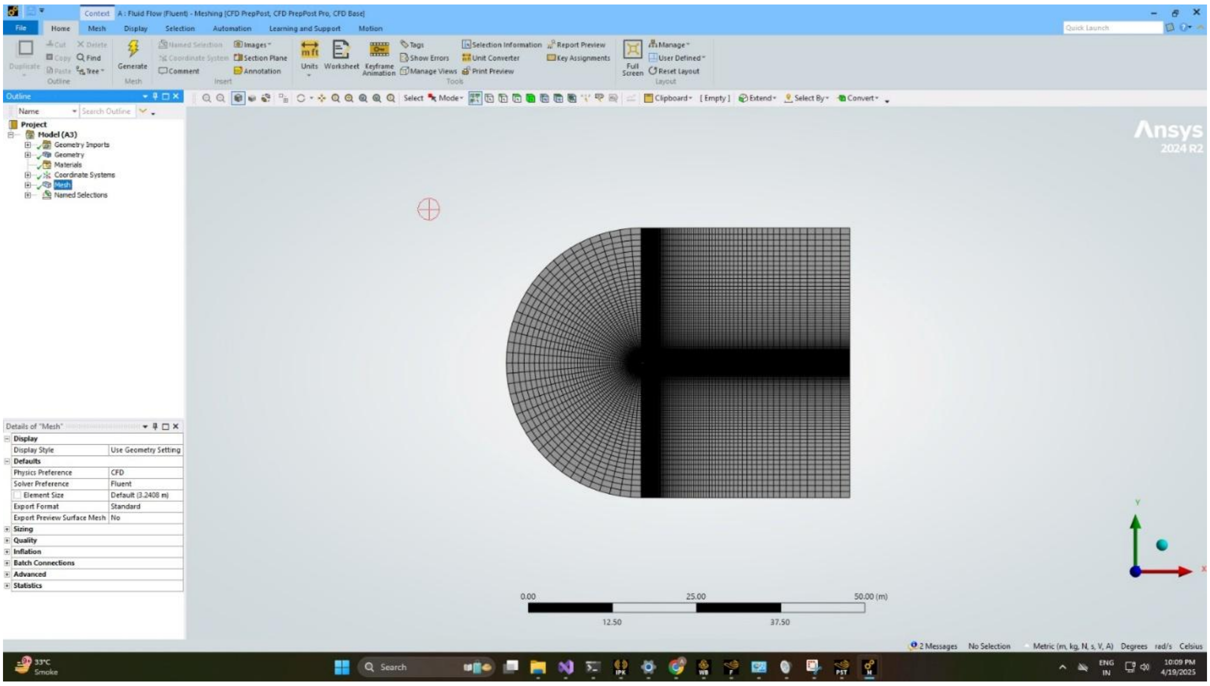

Solution Controls
Pseudo Time Explicit Relaxation Factors
Pressure
0.5
Momentum
0.5
Density
1
Body Forces
1
Default
Equations...
Limits...
Advanced...
Drag Report Definition
Name
drag_coeff
Options
Per Zone
Average Over(Iterations)
1
Report Output Type
Drag Coefficient
Drag Force
Zones
airfoil
Force Vector
X
Y
Z
1
0
1
Report Files [0/0]
Report Plots [0/0]
Create
Report File
Report Plot
Frequency
1
Print to Console
Create Output Parameter
OK
Compute
Cancel
Help

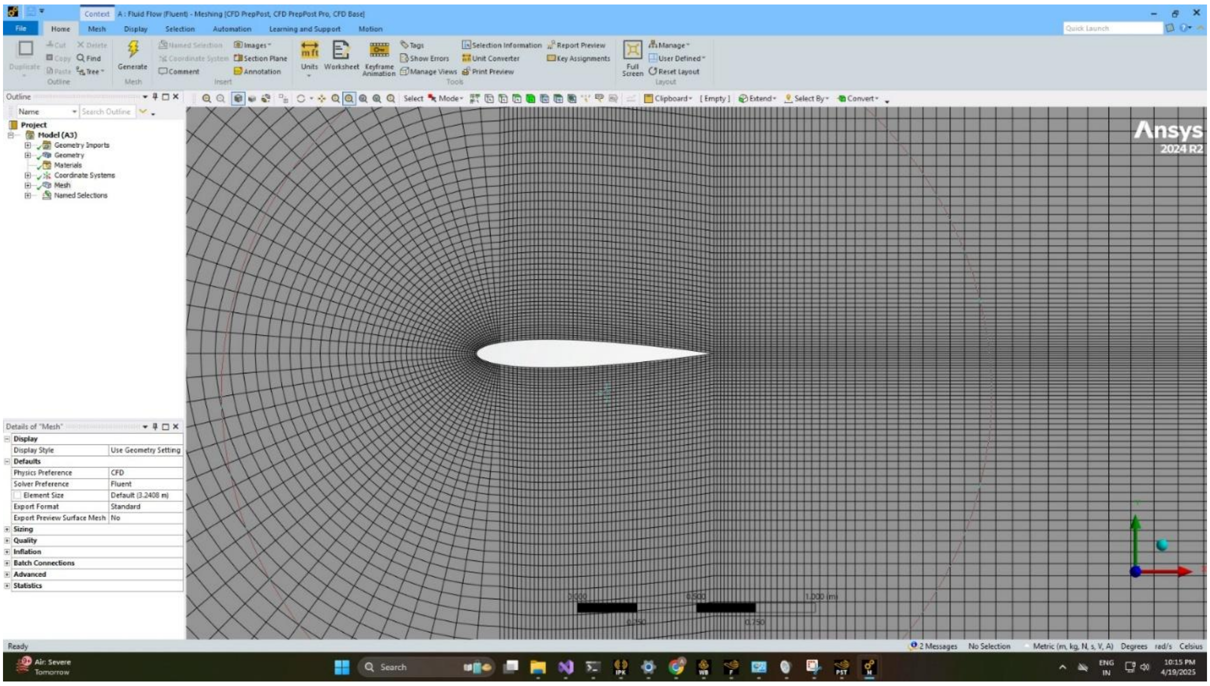


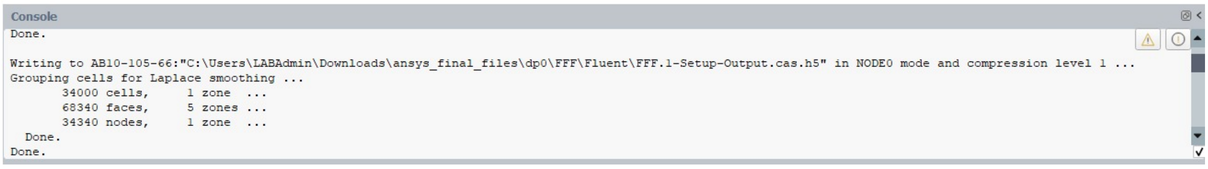




Mesh statistics and figure:







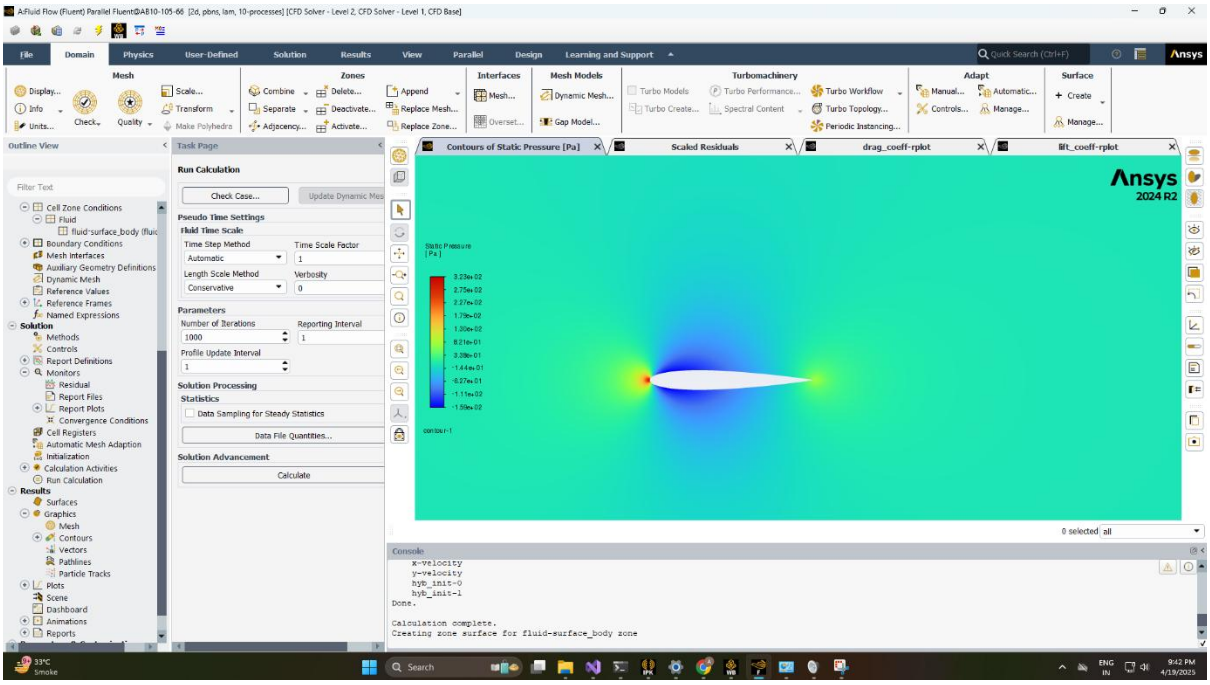
```
Console
Done.

Writing to AB10-105-66:"C:\Users\LABAdmin\Downloads\ansys_final_files\dp0\FFF\Fluent\FFF.1-Setup-Output.cas.h5" in NODE0 mode and compression level 1 ...
Grouping cells for Laplace smoothing ...
       34000 cells,     1 zone  ...
       68340 faces,     5 zones ...
       34340 nodes,     1 zone  ...
  Done.
Done.
```

Plots:

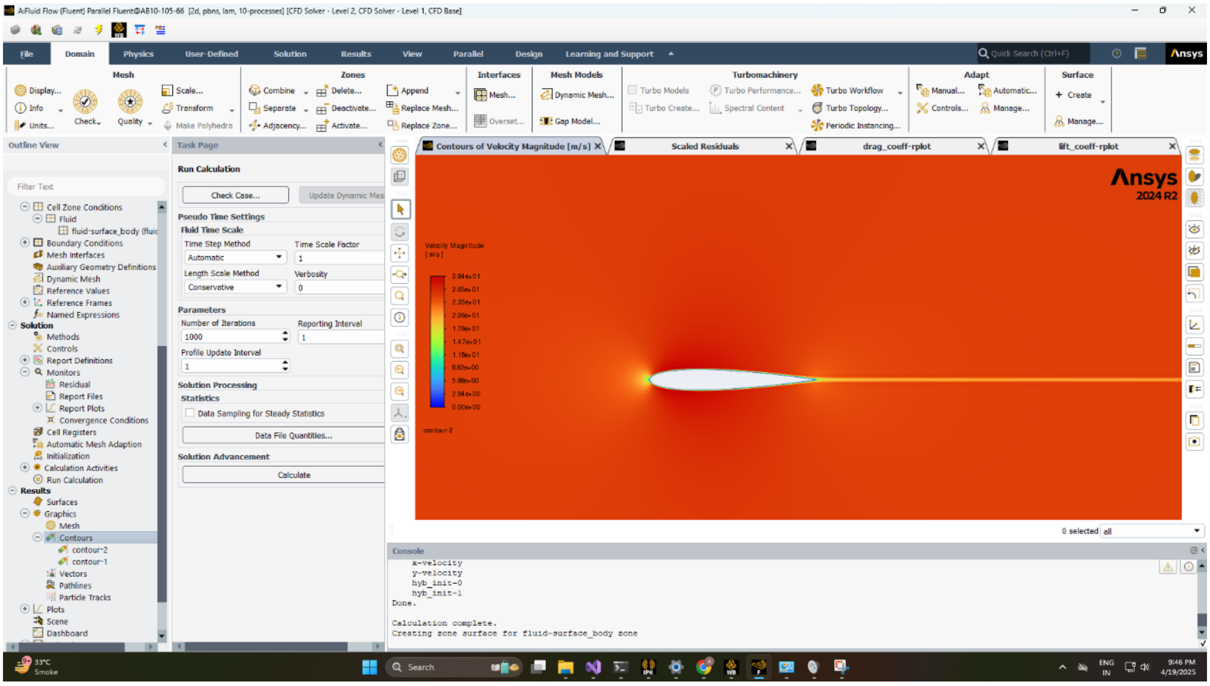
For 0-degree AOA:

Pressure Contour:



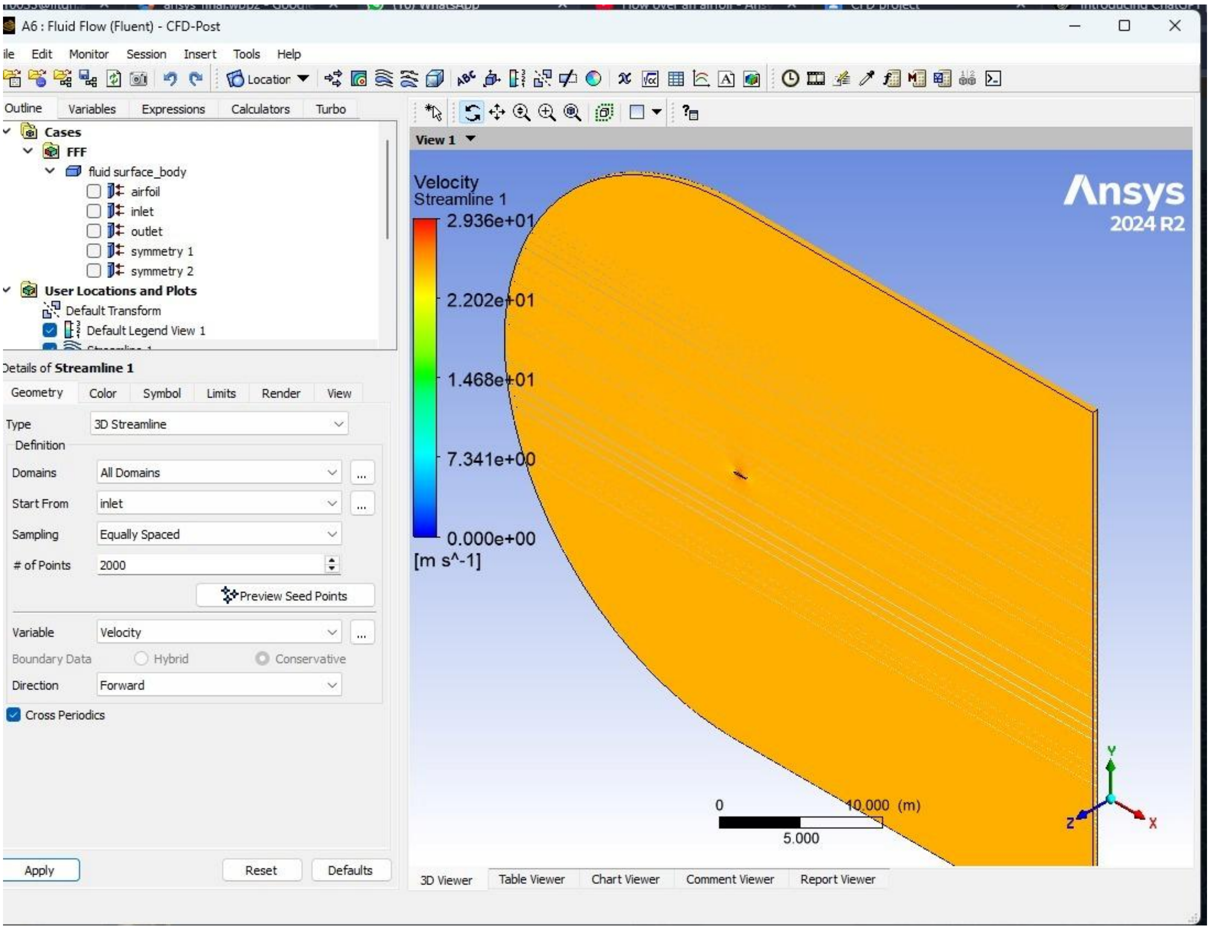
At 0° angle of attack, the pressure contour around the airfoil is nearly symmetric, indicating minimal lift generation. A stagnation point is observed at the leading edge with high pressure, followed by a drop on both upper and lower surfaces. The pressure gradient is nearly equal on both sides, confirming negligible aerodynamic asymmetry. The boundary layer remains mostly attached along the surface, with no signs of separation due to the absence of significant adverse pressure gradients. Flow behavior is predominantly laminar near the leading edge, transitioning gradually downstream.

Velocity Contour:

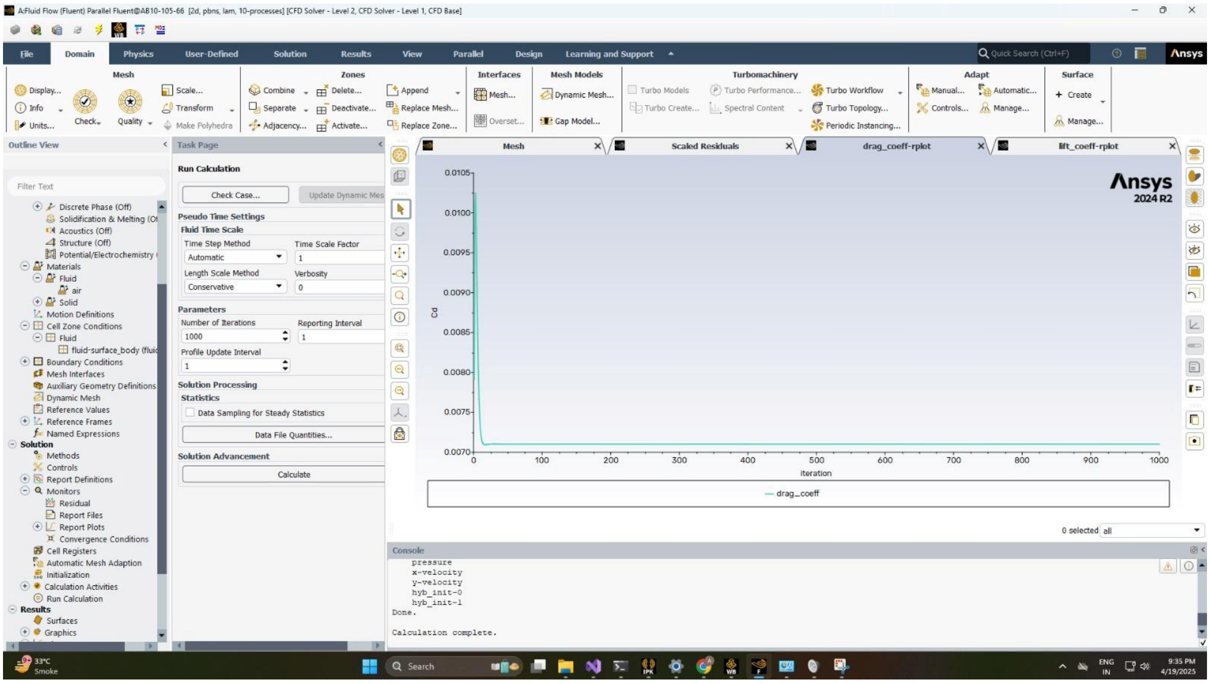


The velocity contour at 0° AOA shows a symmetric pattern with accelerated flow over both upper and lower surfaces, peaking near the leading edge. The velocity magnitude increases smoothly as the flow moves around the airfoil and returns to freestream downstream. This uniform acceleration indicates an attached boundary layer with minimal separation. The absence of strong velocity gradients or recirculation zones confirms steady laminar flow, typical for symmetric profiles at zero angle of attack.

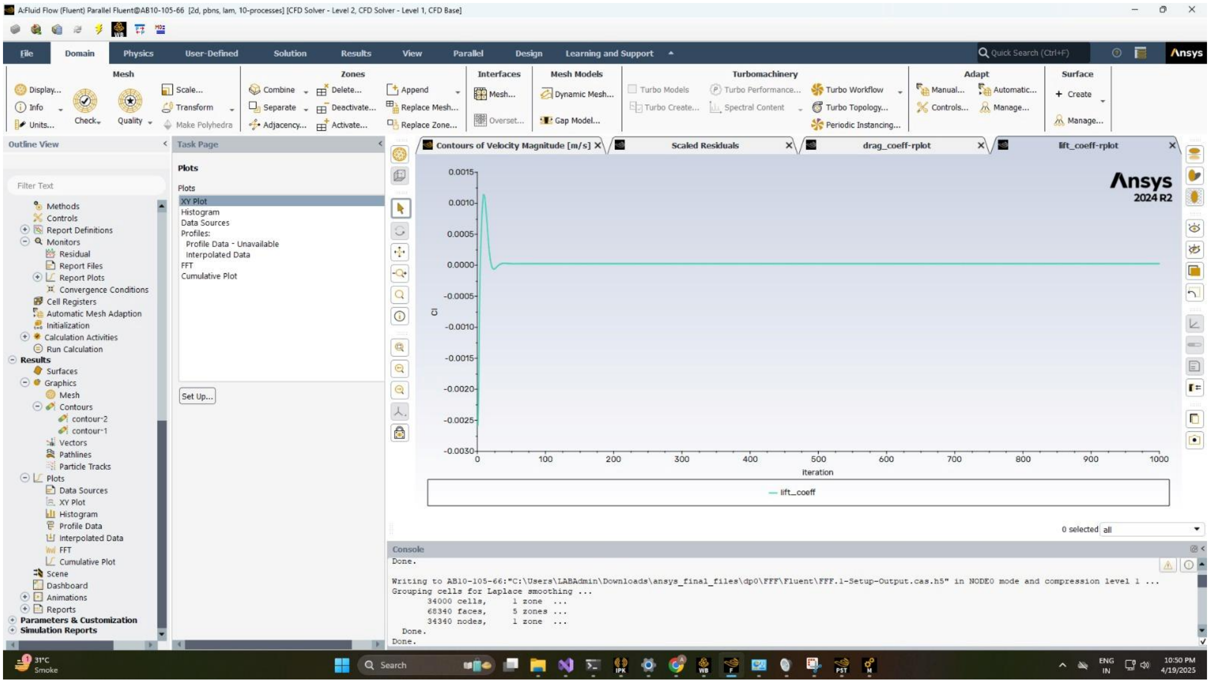
Streamline Contour:



Drag Coefficient v/s Number of iterations:



Lift Coefficient v/s Number of iterations:

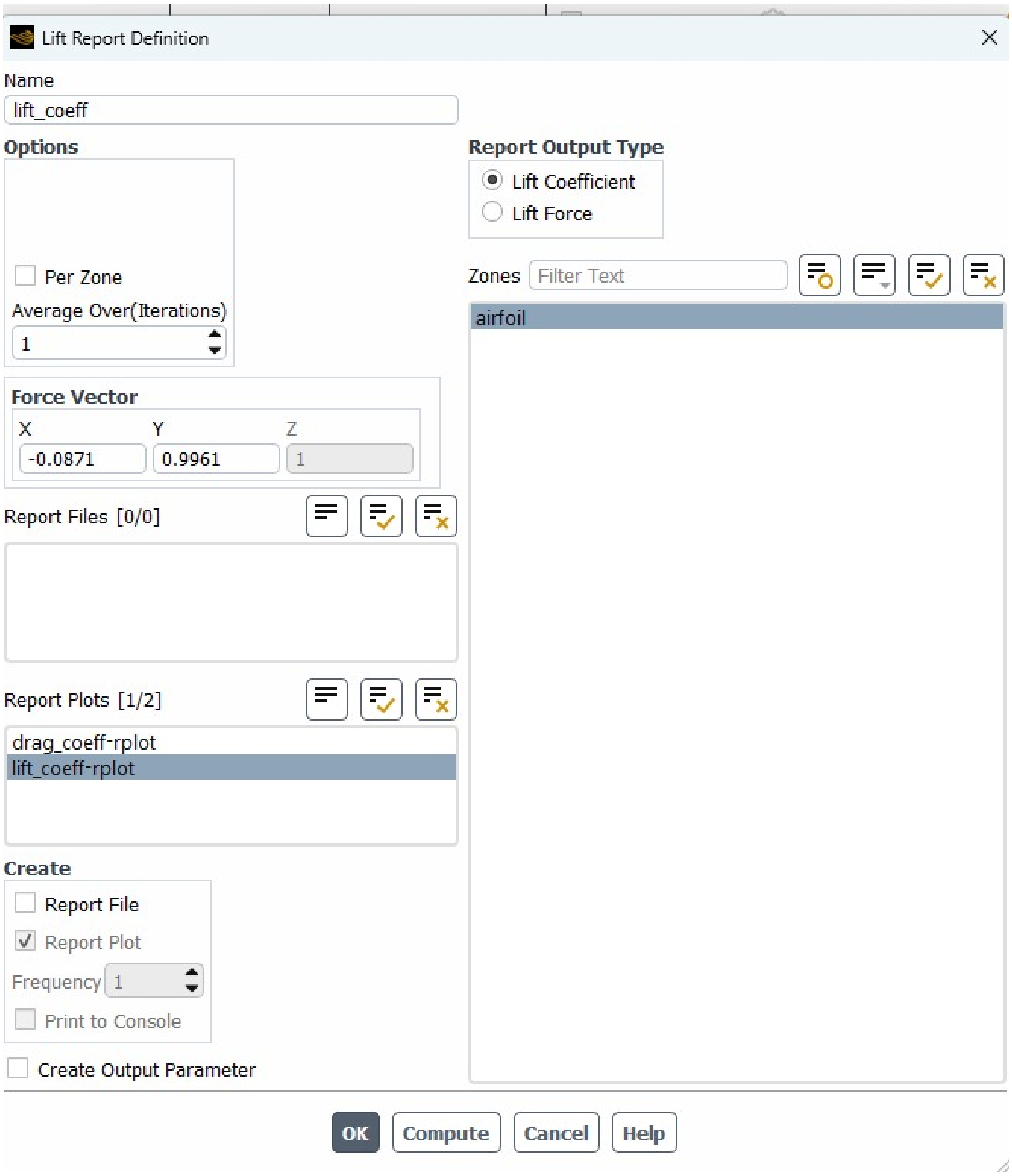


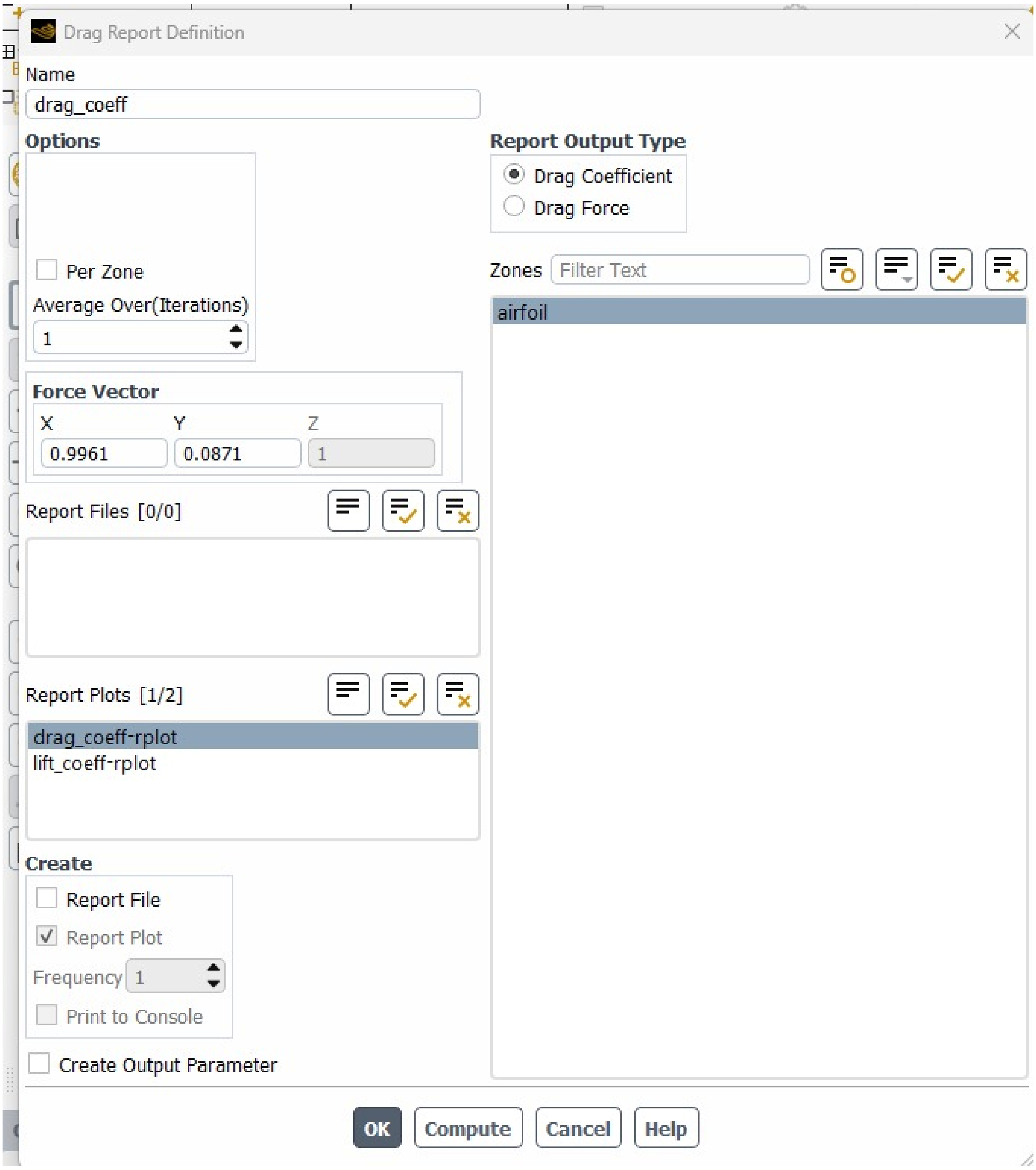
Final Drag Coefficient: 2.603E-5

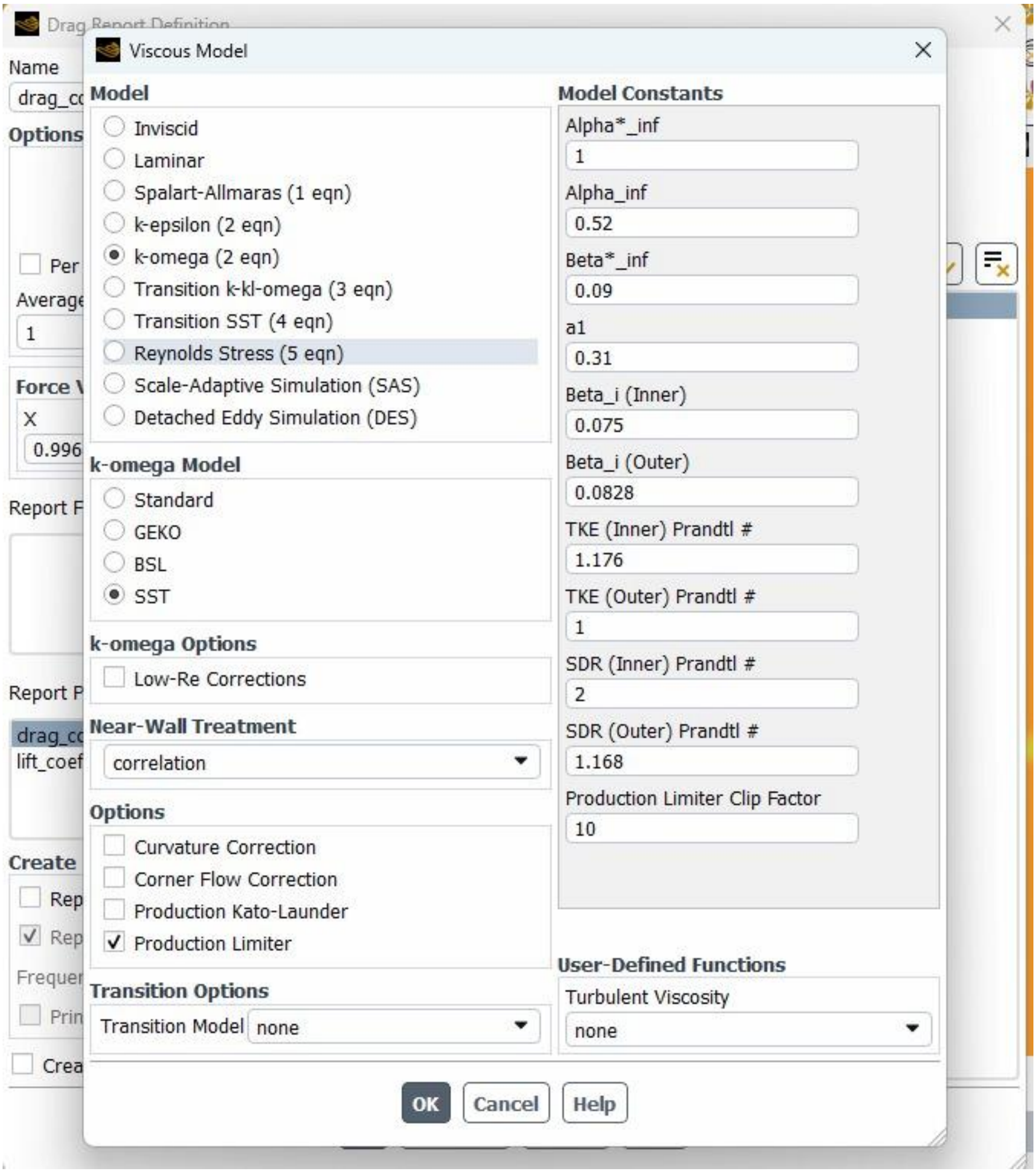
Final Lift Coefficient: 0.00724

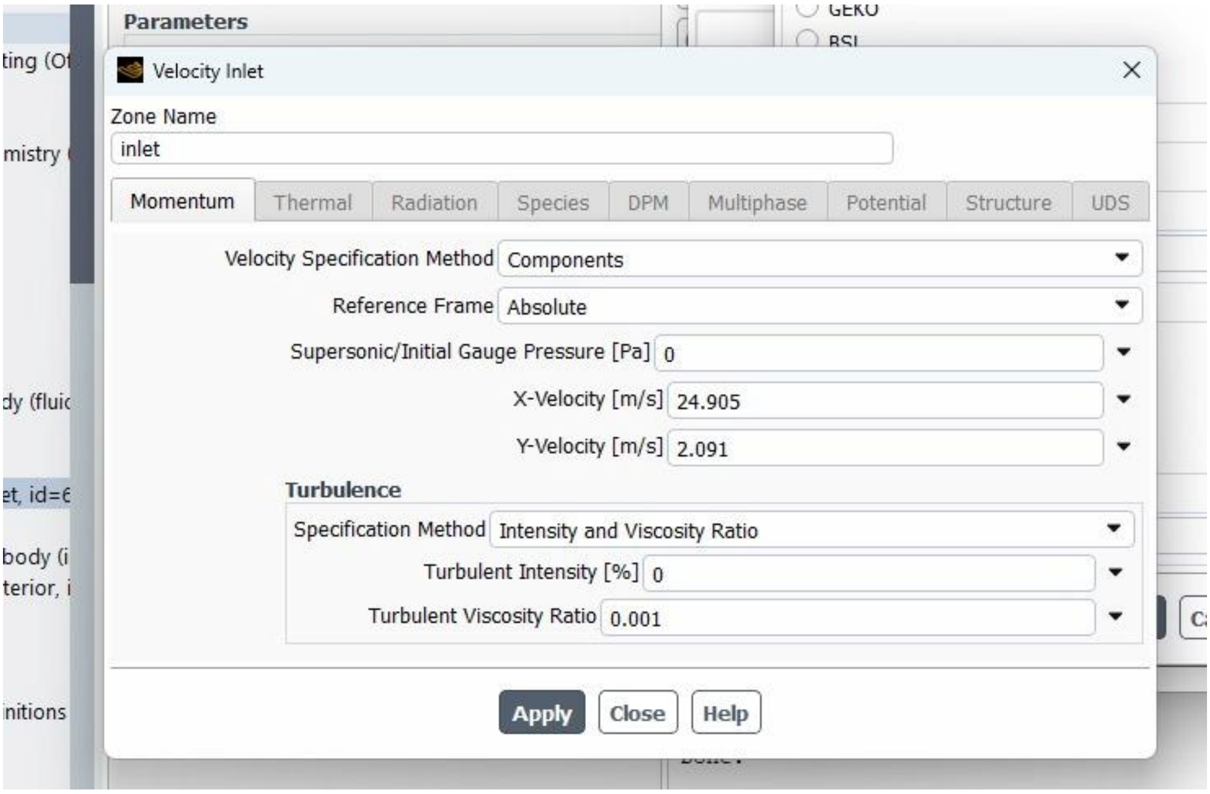
For 5-degree AOA:

Boundary Conditions:

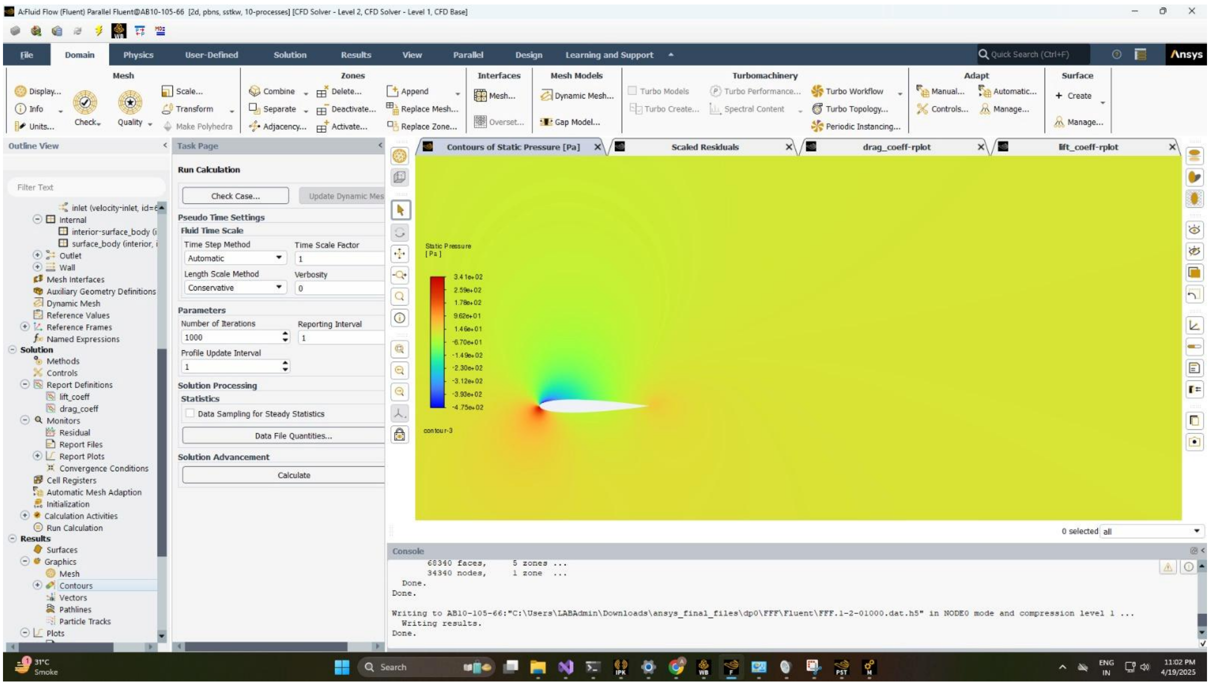

Lift Report Definition
Name
lift_coeff
Options
Per Zone
Average Over(Iterations)
1
Force Vector
X
Y
Z
-0.0871
0.9961
1
Report Files [0/0]
Report Plots [1/2]
drag_coeff-rplot
lift_coeff-rplot
Create
Report File
Report Plot
Frequency
1
Print to Console
Create Output Parameter
Report Output Type
Lift Coefficient
Lift Force
Zones
Filter Text
airfoil
OK
Compute
Cancel
Help


Drag Report Definition
Name
drag_coeff
Options
Per Zone
Average Over(Iterations)
1
Force Vector
X
Y
Z
0.9961
0.0871
1
Report Files [0/0]
Report Plots [1/2]
drag_coeff-rplot
lift_coeff-rplot
Create
Report File
Report Plot
Frequency 1
Print to Console
Create Output Parameter
Report Output Type
Drag Coefficient
Drag Force
Zones
Filter Text
airfoil
OK
Compute
Cancel
Help


Drag Report Definition
Viscous Model
Name
drag_c
Options
Per
Average
1
Force V
X
0.996
Report F
Report P
drag_c
lift_coef
Create
Rep
Rep
Frequer
Prin
Crea
Model
Inviscid
Laminar
Spalart-Allmaras (1 eqn)
k-epsilon (2 eqn)
k-omega (2 eqn)
Transition k-kl-omega (3 eqn)
Transition SST (4 eqn)
Reynolds Stress (5 eqn)
Scale-Adaptive Simulation (SAS)
Detached Eddy Simulation (DES)
k-omega Model
Standard
GEKO
BSL
SST
k-omega Options
Low-Re Corrections
Near-Wall Treatment
correlation
Options
Curvature Correction
Corner Flow Correction
Production Kato-Launder
Production Limiter
Transition Options
Transition Model
none
Model Constants
Alpha*_inf
1
Alpha_inf
0.52
Beta*_inf
0.09
a1
0.31
Beta_i (Inner)
0.075
Beta_i (Outer)
0.0828
TKE (Inner) Prandtl #
1.176
TKE (Outer) Prandtl #
1
SDR (Inner) Prandtl #
2
SDR (Outer) Prandtl #
1.168
Production Limiter Clip Factor
10
User-Defined Functions
Turbulent Viscosity
none
OK
Cancel
Help

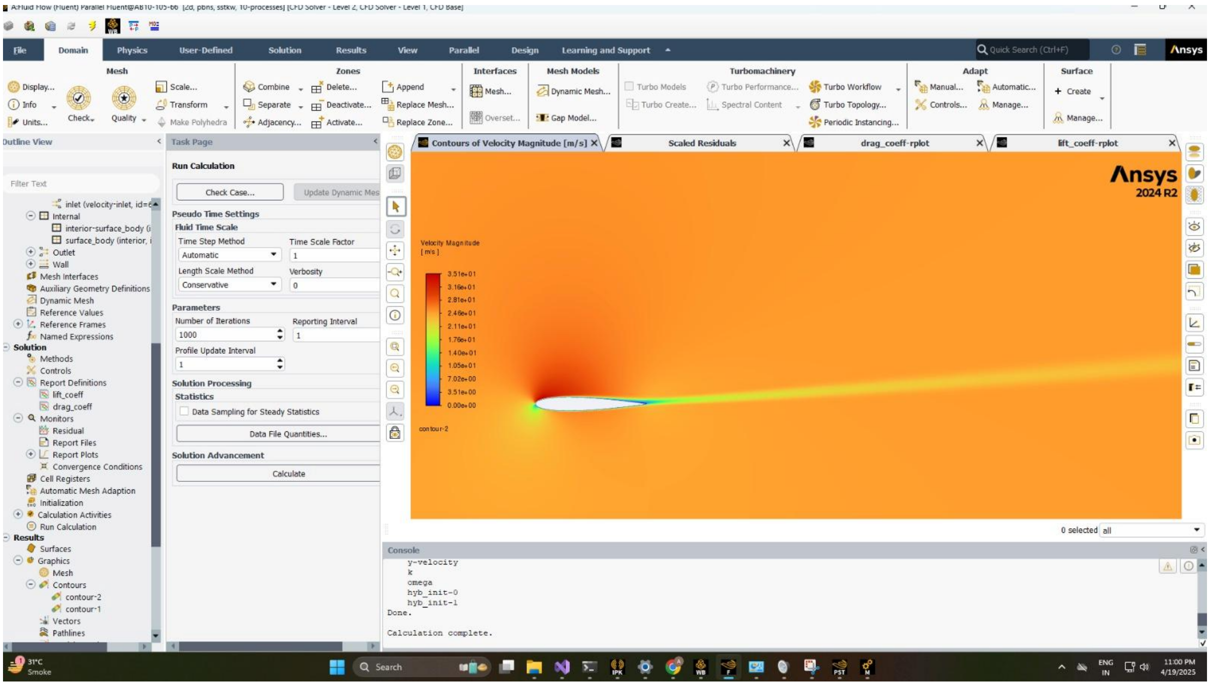


Pressure Contour:



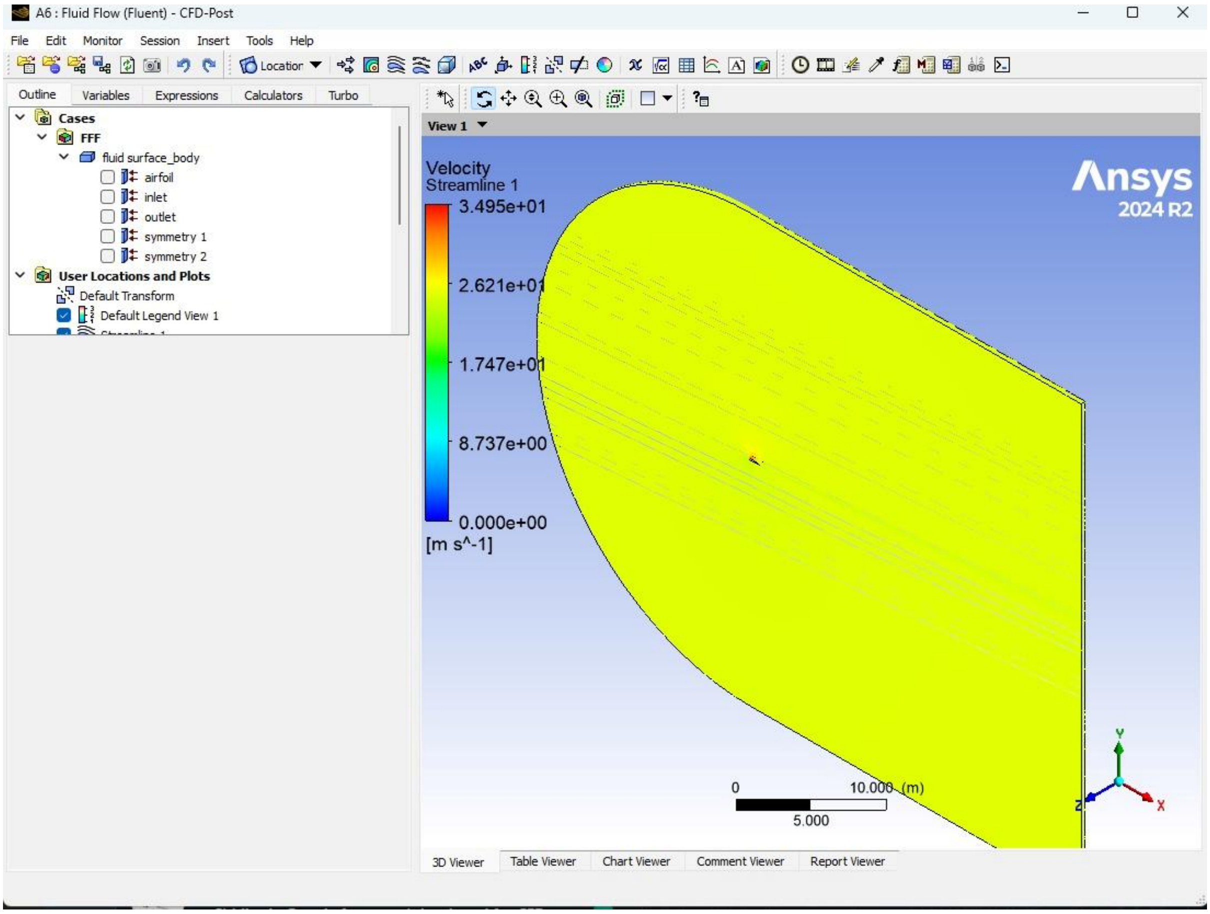
At 5° AOA, the static pressure contour reveals a clear asymmetry. A low-pressure zone is intensified on the upper surface, while a high-pressure zone develops below, indicating lift generation. The upper surface shows strong pressure gradients near the leading edge due to flow acceleration. This pressure difference across the airfoil surfaces is the primary source of lift. The boundary layer remains mostly attached but is under higher stress on the suction side, especially near the trailing edge, increasing the risk of early separation at higher AOAs.

Velocity Contour:

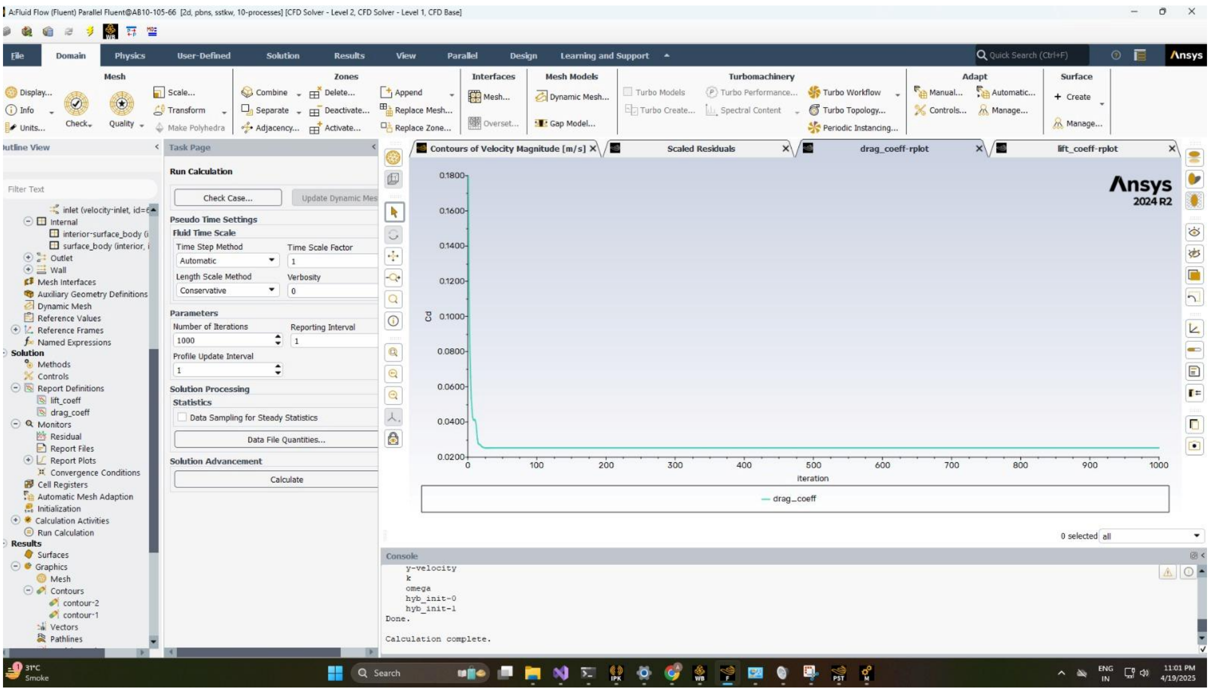


At 5° AOA, the velocity contour shows clear acceleration over the upper surface, with high-speed flow forming a narrow band due to airfoil camber and angle. The lower surface exhibits relatively slower flow, leading to a favourable pressure differential. Boundary layer on the upper surface is thinner due to increased shear but remains attached, indicating efficient lift generation. Flow separation is not visible, suggesting the AOA is still within the linear lift range.

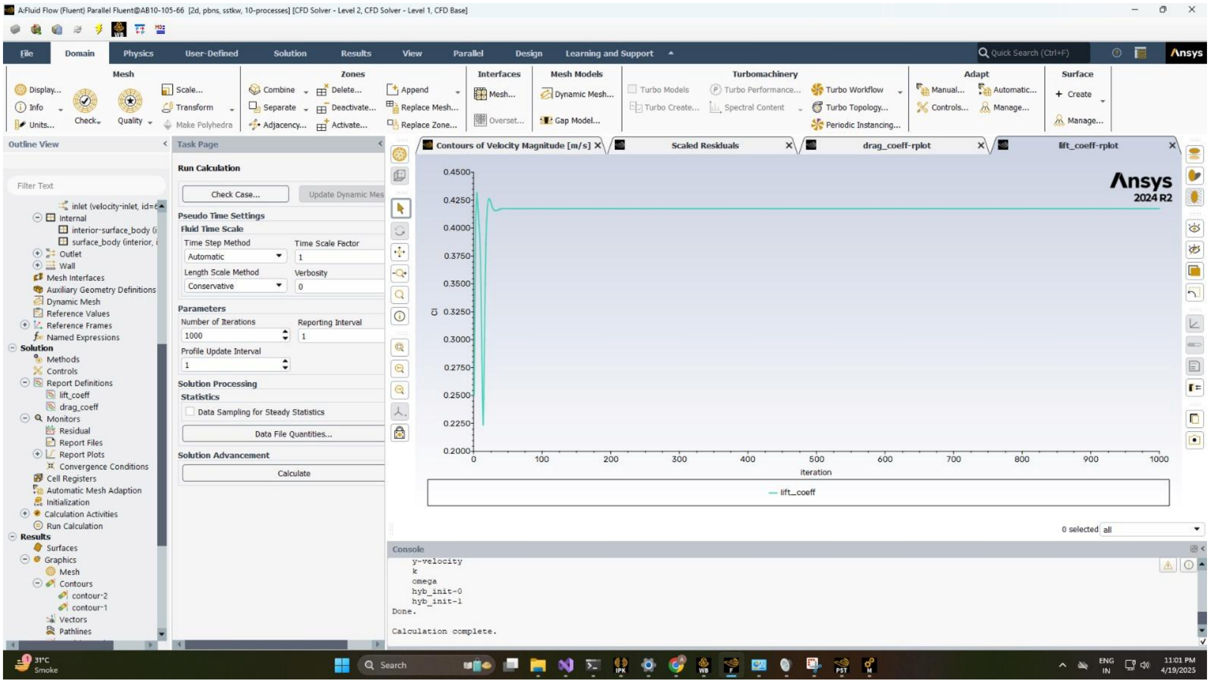
Streamline Contour:



Drag Coefficient vs Number of iterations:



Lift Coefficient vs Number of iterations:



Final Drag Coefficient: 0.0250

Final Lift Coefficient: Approximately 0.4125

Aerodynamic Coefficients Table

AOA	Parameters	Ansys – CFD results	Literature
0^0	C_D	2.603E-5	0
	C_L	0.00724	0.0065
5^0	C_D	0.0250	0.006
	C_L	0.4125	0.5

The angle of attack (AOA) significantly influences the aerodynamic forces acting on a body, particularly lift and drag. From the CFD results and literature, the lift coefficient (C_l) and drag coefficient (C_D) vary with AOA due to changes in pressure distribution and flow separation characteristics around the body.

At AOA = 0°

Parameter CFD Result Literature

C_D 2.603E-5 0.0

C_L 0.00724 0.0065

Lift (C_L)

- At zero AOA, the body is aligned with the flow direction, resulting in symmetric pressure distribution on both sides.
- However, a small positive lift is observed in CFD (C_L ≈ 0.00724), possibly due to:
  - Slight asymmetries in geometry or mesh.
  - Minor numerical perturbations inherent in simulation.
- This result is consistent with literature, which also reports a small C_L (~0.0065).

Drag (C_D)

- CFD predicts a near-zero drag coefficient (2.603E-5), which is expected because at zero AOA, pressure drag is minimal and viscous effects dominate.
- Literature reports zero drag, likely assuming ideal flow or neglecting viscous effects.

At AOA = 5°

Parameter	CFD Result	Literature
C_D	0.0250	0.006
C_L	0.4125	0.5

Lift (C_L)

- The increase in AOA to 5° causes a larger pressure difference between the upper and lower surfaces.
- As a result, lift increases substantially (C_L = 0.4125).
- CFD slightly underpredicts lift compared to literature (C_L = 0.5), possibly due to:
  - Mesh resolution at the leading edge.
  - Slight differences in flow assumptions.
  - Non-Ideal boundary layer resolution.

Drag (C_D)

- C_D increases to 0.0250 in CFD, compared to 0.006 in literature.
- The rise in drag is due to:

- Increase in pressure drag caused by flow separation beginning at higher AOAs.
- Viscous drag also contributes as boundary layers grow thicker.

- The CFD result shows a relatively higher drag possibly due to more realistic viscous effects being captured, unlike in literature models.

Conclusions

This study demonstrated the effective application of CFD techniques to analyze laminar flow over a NACA 0012 airfoil at differing angles of attack. By employing a carefully refined mesh and robust discretisation schemes, the simulations captured key flow phenomena—such as boundary-layer development, pressure distribution, and early separation—across both low and moderate angles. The observed trends in lift enhancement and drag increase with angle of attack align with aerodynamic theory, confirming the validity of the modelling approach. Overall, the work highlights how systematic CFD analysis can yield deep insights into airfoil performance and guide future investigations into more complex flow regimes.

References:

https://www.youtube.com/watch?v=nzvEvLCxOss&t=836s

https://turbmodels.larc.nasa.gov/naca0012_val.html