



2D Airfoil CFD Analysis



JUNE 2025

SIDHARTHA MAURYA
Aerodynamics Design Intern
Zmotion Autonomous Systems
Hobli, Bangalore

Abstract

This study presents a comprehensive 2D CFD analysis of the NACA 0012 airfoil at low Reynolds numbers (**$Re = 2 \times 10^5$ and 3.5×10^5**) to evaluate aerodynamic performance using two turbulence models: k- ω SST and **Transition SST ($\gamma-\theta$)**. A structured C-type mesh was generated with domain extents of 15 chord lengths downstream and 7.5 upstream/normal was employed with fine near-wall resolution ($Y^+ < 1$) to accurately capture boundary layer behavior. Mesh independence was verified through a detailed Grid Convergence Index (GCI) study, demonstrating monotonic convergence for both lift and drag coefficients, with **GCI errors below 0.037% for Cl and 0.059% for Cd at the finest mesh (800,000 cells)**.

Simulation results showed strong agreement with experimental data, with **mean errors of 3.52% in Cl and 2.76% in Cd for the Transition SST model**. The k- ω SST model, while robust, overpredicted lift at higher angles of attack (AoA) due to its assumption of fully turbulent flow. In contrast, the Transition SST model successfully captured **laminar separation bubbles**, transition onset, and flow reattachment—particularly critical in the AoA range of 8° – 12° . The moment coefficient (C_m), however, displayed unstable and divergent behavior, attributed to mesh sensitivity near the reference point and limitations of the steady-state solver.

A comparative analysis using the Clark Y airfoil revealed higher lift generation due to its camber but also earlier stall onset. Insights from relevant literature supported the observation that transition modeling is essential for accurately predicting flow behavior at low Re , and that Reynolds number has a stronger impact than turbulence intensity on aerodynamic characteristics.

The study concludes with recommendations for future work, including the **use of unsteady RANS (URANS)** approaches, higher-order discretization schemes, and further calibration of transition models to improve predictions at higher AoA. Overall, this work establishes a validated and mesh-independent CFD workflow for low- Re airfoil analysis, emphasizing the critical role of turbulence modeling, near-wall resolution, and grid quality in capturing complex flow phenomena with high accuracy.

Problem Statement

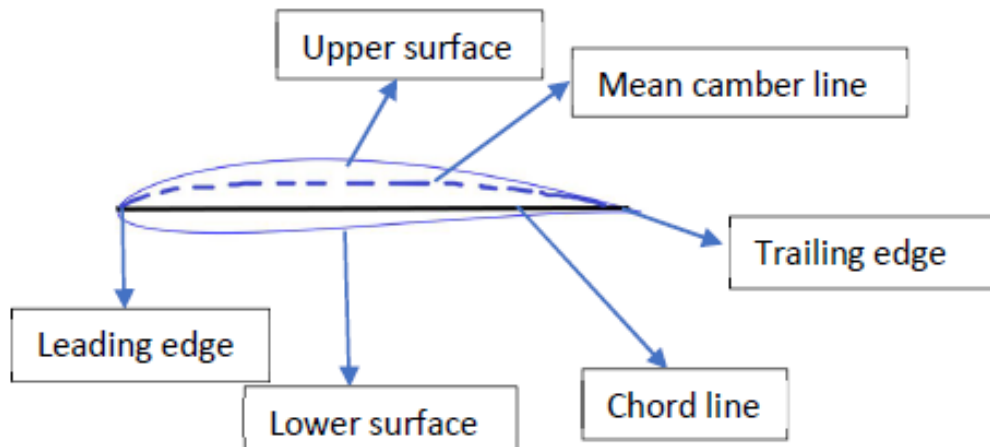
- **Simulate steady, incompressible external flow** over 2D airfoils (NACA 0012 & Clark Y).
- **Compare turbulence models** (k- ω SST, Transition SST) for predicting lift (Cl), drag (Cd), and boundary layer behaviour.
- Conduct **mesh refinement and GCI (Grid Convergence Index)** study.
- Evaluate aerodynamic performance at **$Re = 2 \times 10^5$ and 3.5×10^5** for various angles of attack.

Geometry Setup

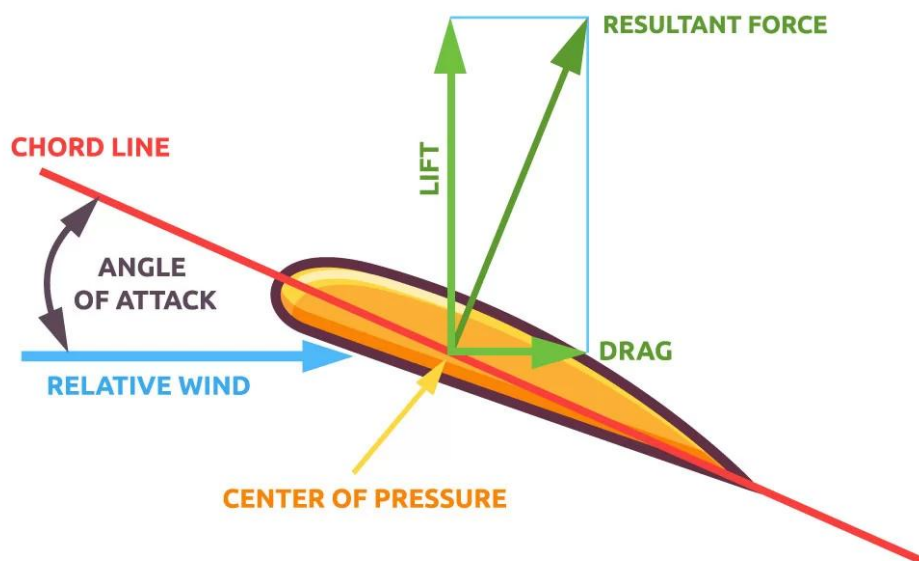
- **Chord length:** 180 mm
- **Freestream Velocity:** 25 m/s, 16.23 m/s
- **Angle of Attack Range:** 0° to 16°
- **Domain Type:** C-type, radius = $7.5c$, length = $15c$
- **Boundary Condition:** Inlet velocity components decomposed for different AOAs

Basic Terminology

Airfoil, a 2D cross-sectional area of a wing, when kept in a fluid experiences motion. The fluid can be air for flight purposes or water/liquid for experiment point of view. The air gets bifurcated into two streams, i.e., upper and lower. Pressure distribution arises on upper and lower surfaces of an airfoil. The difference in pressure is the main cause for the aerodynamic forces Lift and Drag.

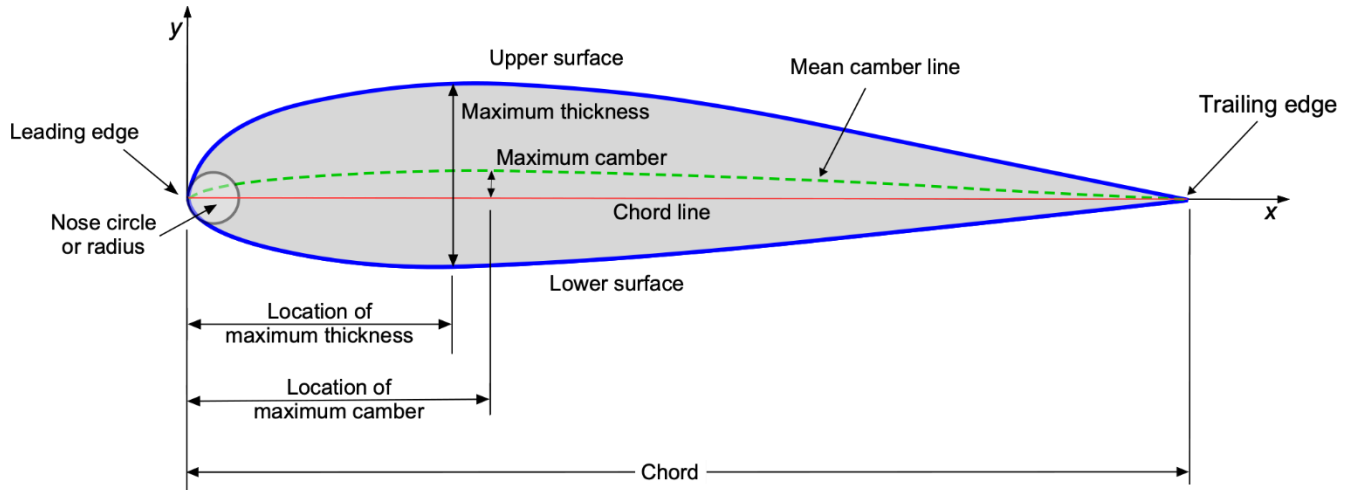


Lift is generated when a solid object moves through a fluid (like air or water), causing a difference in pressure and velocity between the top and bottom of the object. This pressure difference creates an upward force that opposes the object's weight, allowing it to stay aloft. The shape and angle of the object, as well as the speed and direction of the fluid flow, all play a role in how much lift is produced.



NACA 0012:

NACA 0012 has been chosen for the present study owing to its very basic geometry and since it does not have any camber, its L/D ratio is sometimes suitable for some portions of flight. Also, the movement of the centre of pressure of symmetrical aerofoil is less than with cambered aerofoils.



Four-digit Series - The NACA four-digit wing sections define the profile by:

1. The first digit describes maximum camber as percentage of the chord.
2. The second digit describes the distance of maximum camber from the airfoil leading edge in tens of percent of the chord.
3. The last two digits describe the maximum thickness of the airfoil as percentage of the chord.

NACA 0012 means that there is no camber i.e. zero camber is present, and it has the maximum thickness of the airfoil at 12% of the chord from the leading edge.

Visit these websites for Airfoil Generator:

1. [Airfoil Shapes](#)
2. [Airfoil tools \(NACA 4-digits Series\)](#)

Airfoil Coordinates:

The airfoil geometry used in this project is based on the NACA 0012 profile, with a chord length of 180 mm. The coordinate data was obtained from [Airfoil Tools](#) and requires specific formatting to be correctly imported into ANSYS and SolidWorks. Below are the necessary steps and considerations for processing the airfoil coordinate data:

#group	#point	#x	#y	#z	
1	1	180	0	0	Upper Surface
1	2	179.9555	0.00648	0	
1	3	179.8223	0.02574	0	
.	
.	
.	
1	99	0.17766	0.99378	0	
1	100	0.04446	0.50022	0	
1	101	0	0	0	
1	102	0.04446	-0.50022	0	Lower Surface
1	103	0.17766	-0.99378	0	
.	
.	
.	
.	
1	199	179.8223	-0.02574	0	
1	200	179.9555	-0.00648	0	
1	201	180	0	0	
1	0				

1. Coordinate Table Format for ANSYS

- Group Column (#group):** This identifies which surface the point belongs to. In this case, all points are assigned to group 1, combining both the upper and lower surfaces of the airfoil. Optionally, the upper surface may be assigned to group 1 and the lower surface to group 2 to help ANSYS interpret them as distinct geometries.
- Point Column (#Point):** This gives the index of the point in the sequence, beginning from the leading edge, moving along the upper surface to the trailing edge, and then back along the lower surface to the starting point to form a closed loop.
- Coordinate Columns (X, Y, Z):** These represent the spatial coordinates of each point. The airfoil contour starts from the leading edge at (180, 0, 0), proceeds counterclockwise over the upper surface to (0, 0, 0), and then continues along the lower surface back to the trailing edge, ensuring a continuous, closed geometry. **All #z values to 0 (2D simulation).**

Note: The final trailing edge point (i.e. 201st point in red line) must be modified slightly to avoid duplication. Instead of repeating the point (180, 0, 0) at the end, it is recommended to use the penultimate point (Last line in yellow) as the final point. This ensures a closed trailing edge, which is critical for ANSYS to recognize the geometry as a valid loop. **## Save file in Text (Tab Delimited).txt Format**

2. **Coordinate Format for SolidWorks:** When importing the airfoil into SolidWorks:

- a. Only the last three columns (X, Y, Z) are needed.
- b. The point at the leading edge (typically repeated at the end) must be removed to prevent geometry errors. (180, 0, 0)
- c. SolidWorks often encounters issues with overlapping points, especially at the trailing edge. Therefore, the model should be **manually adjusted** in Solidworks Design Modeler to ensure the trailing edge is closed without duplicate coordinates.

Note: The coordinate at the end of leading edge (e.g., point 101: 0, 0, 0) may result in an import error. If this happens, it can either be removed or slightly perturbed to maintain uniqueness in the coordinate list.

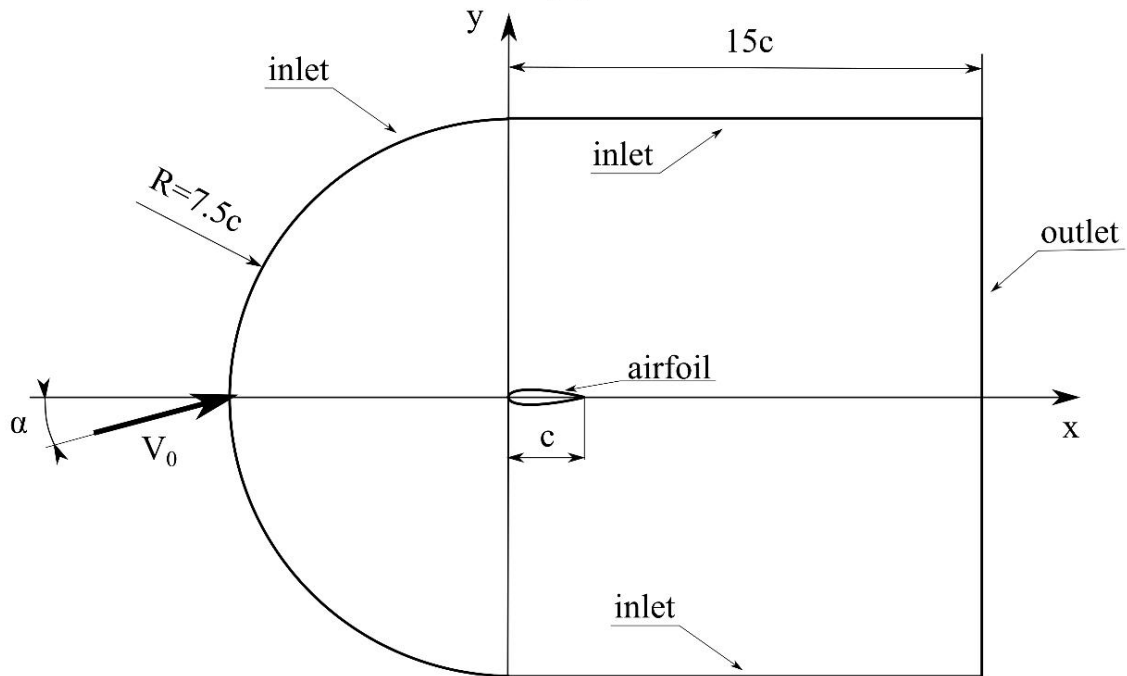
Ref Video:

1. [NACA 2412 2D CFD Analysis- Cillian Thomas](#)
2. [How to import airfoil coordinates | Modeling Airfoils In SOLIDWORKS](#)

Ansys Geometry Modeler:

A critical factor influencing CFD accuracy is the size of the computational domain. As discussed by [Rogowski et al. \(2021\)](#), the impact of domain size is highly dependent on mesh topology (**C-mesh, O-mesh, etc.**), mesh density, and flow characteristics. Although no universal consensus exists in the literature regarding optimal domain dimensions relative to airfoil chord length, practices vary widely—from a few chord lengths to several hundred chord lengths away from the geometry.

In their numerical study of the NACA 0018 airfoil at low Reynolds number ($Re = 150,000$) for Darrieus wind turbines using the Transition SST turbulence model, [Rogowski et al.](#) evaluated the effects of

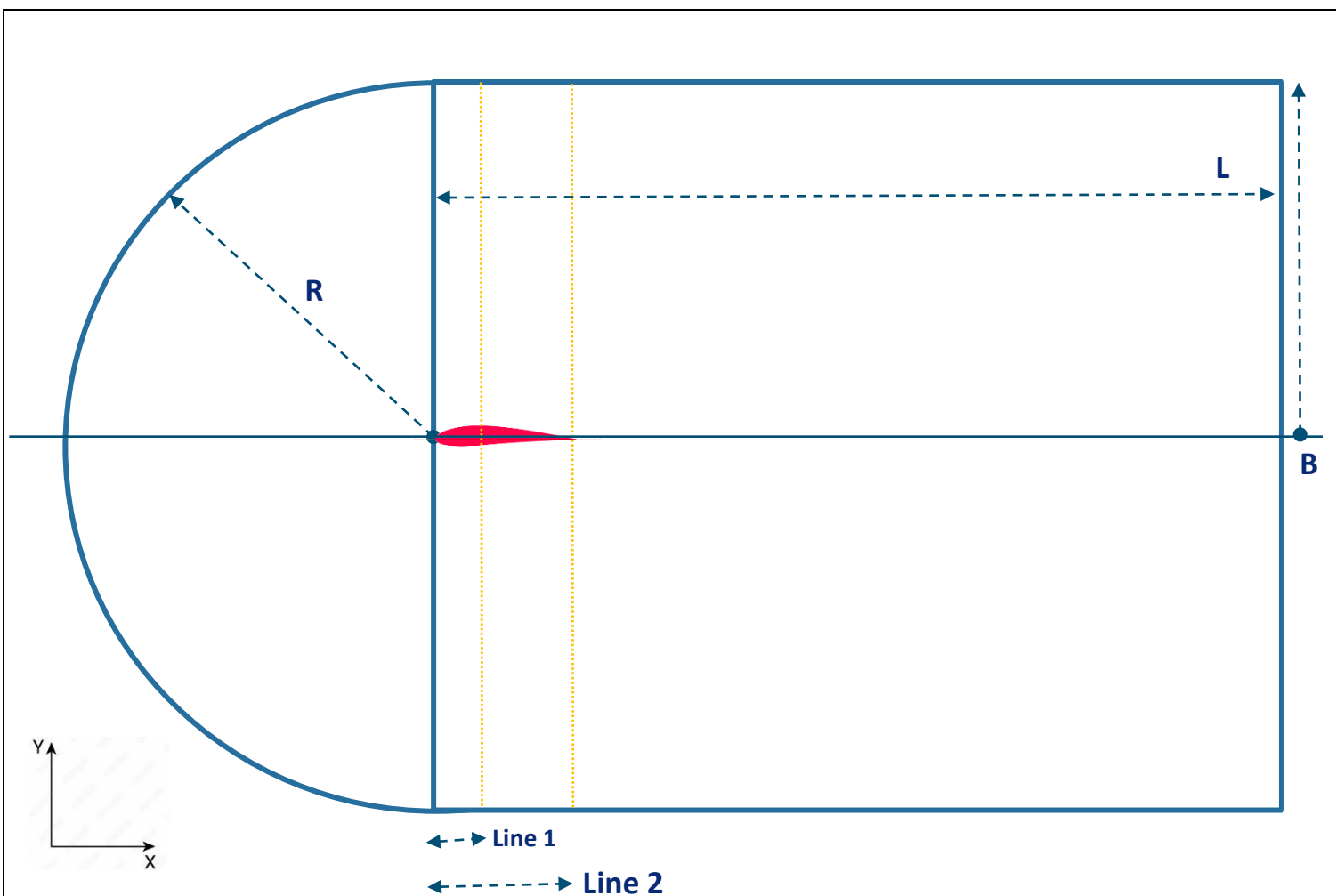


domain width and wake length. They tested three domain widths— $3.75c$, $7.5c$, and $15c$ —and found that while the lift coefficient remained largely unaffected, the drag coefficient showed moderate sensitivity to domain width. A domain width of **$15c$ was deemed sufficient for accurate aerodynamic predictions, balancing fidelity and computational cost** (resulting in roughly half the mesh elements compared to a $30c$ domain).

So, in this study we are going to use **domain size of $15 c$** .

Yellow Lines 1 and 2 are introduced to ensure **fine mesh resolution** near the airfoil's surface. Given the presence of a finite nose radius, accurately resolving this region is critical for capturing boundary layer development and separation.

These lines help define refined mesh zones that enable the placement of the first cell height within $Y^+ < 1$, ensuring accurate resolution of the near-wall region when using wall-resolved turbulence models like SST or Transition SST.



Name	Dimension (in mm)		Formula
Chord Length, C	180		C
Arc Radius, R	1350		$7.5 \times C$
Length, L	2700		$15 \times C$
Height from x-axis, B	1350		$7.5 \times C$
Line 1	54		$0.3 \times C$
Line 2	180		$1.0 \times C$

Process:

1. Import into DesignModeler:

- Open ANSYS DesignModeler, set units to mm, and analysis type to 2D.
- Use Concept → 3D Curve → From File to import airfoil.
- Generate the airfoil curve

2. Create Fluid Domain (C-Type Domain)

- Sketch a semi-circle in front of airfoil (radius = 1350 mm = 7.5c).
- Sketch a rectangle extending 2700 mm behind trailing edge.
- Close the domain around airfoil (C-type).

3. Add Helper Lines for Structured Meshing

- Draw vertical lines: Near **trailing edge** (at 180 mm) and near **leading edge** (~54 mm offset).
These helps define **inflation layers** and **block structure**.

4. Create Surfaces

- Convert sketches to lines, then Concept → Surface from Edges.
- Use Boolean operation to subtract airfoil from domain.
- Use Tools → Face Split to divide domain for better meshing control.

5. Merge Small Edges

- Merge short edges near airfoil with neighbours to reduce meshing errors.

Meshing:

1. Meshing divides the continuous fluid domain into discrete cells, allowing the Navier-Stokes equations to be solved numerically in each control volume.
2. A high-quality mesh:
 - Ensures accurate gradients near boundaries (e.g., airfoil surface).
 - Prevents numerical errors (non-convergence, poor results).
 - Reduces computational cost by refining only where needed.
3. **Mesh Type Selection** for airfoil CFD:
 - C-type structured mesh is most preferred because of the smooth mesh around leading edge, good control of wake refinement, High-quality boundary layer resolution.
 - O-mesh: Fully wraps airfoil but less wake control.
 - Unstructured (tri/tet): Flexible but lower accuracy near walls.
4. **Inflation Layers:** Apply inflation near the airfoil to resolve the boundary layer.
 - First cell height: 0.005–0.01 mm (adjust based on Y^+)
 - No. of layers: 20–30
 - Growth rate: 1.2–1.25
 - Maximum thickness: ~20% of chord
 - Use smooth transition to outer mesh.

Ref:

1. [CFDFLOWENGINEERING : Mesh Quality Assestment](#)
2. [Anthony T – Crash Course on CFD Fundamental](#)
3. [Inflation Layers - Part 2 \(Corners, Orthogonality, Smoothing\)](#)

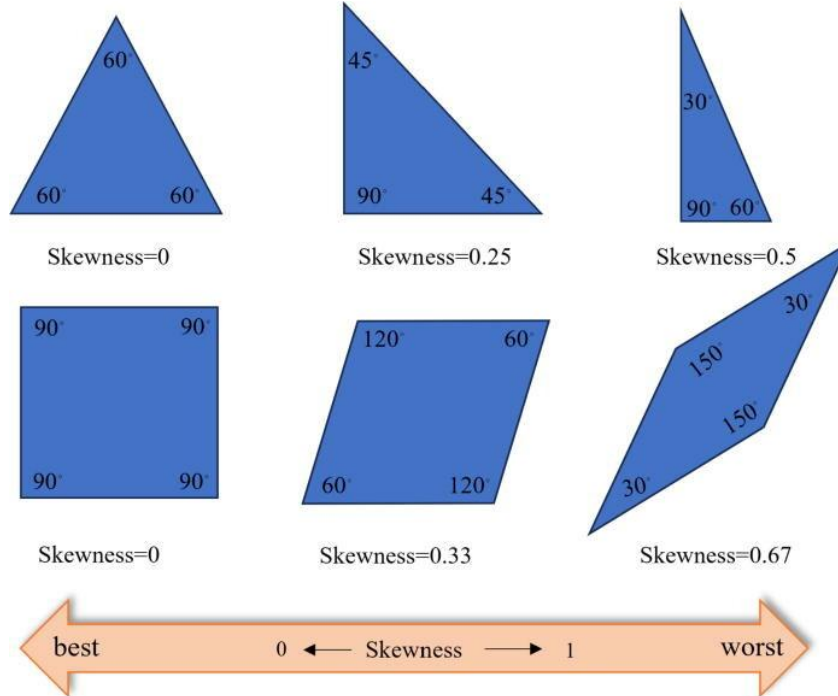
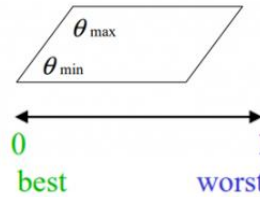
Assessment of Mesh Quality in CFD

1. **Skewness:** Skewness measures the deformation of an element from its ideal shape (usually an equilateral triangle or cube).

➤ Mesh quality: skewness

- Based on Equi-angle skewness
- Range of skewness: 0 -1

$$\max \left[\frac{\theta_{\max} - \theta_e}{180 - \theta_e}, \frac{\theta_e - \theta_{\min}}{\theta_e} \right]$$



Why It Matters:

- The skewness of the mesh should be below as much as possible (< 0.85) above is for unstructured mess.
- Higher skewness can slow down the convergence of simulation and incorrect computations fluxes near walls.
- Skewness and orthogonal both are opposite each other. Higher orthogonal quality means lower the skewness of mesh,
- For ANSYS FLUENT: **Orthogonal quality = 1 – Cell skewness**

Tips:

- Use structured meshing or smoothing to reduce skewness.
- Focus especially on near-wall regions, sharp corners, and wake zones.

2. Aspect Ratio (AR): The ratio of the longest edge to the shortest edge in a mesh element.

$$AR = \frac{\text{Longest Side}}{\text{Shortest Side}} = \frac{\text{Largest Bounding Box Area}}{\text{Smallest Bounding Box Area}}$$

In the direction of the high gradient region, the aspect ratio should be less (< 20) for laminar and turbulent flow (RANS Modeling).

Ideal Range: A well-shaped element should have an aspect ratio close to 1.

Simulation Type	Recommended AR	Notes
General CFD (RANS)	AR < 10–20	Maintain well-shaped elements
In Boundary Layer (aligned)	AR < 35 acceptable	Long cells allowed if aligned with flow direction
In Laminar	AR > 100	In laminar or highly stretched regions but must be flow-aligned.
Far-Field / No Gradient	AR up to 250	Only if no strong gradients exist
Multiphase / LES / DNS	AR < 2–3	Requires highly isotropic elements for fidelity

Why It Matters:

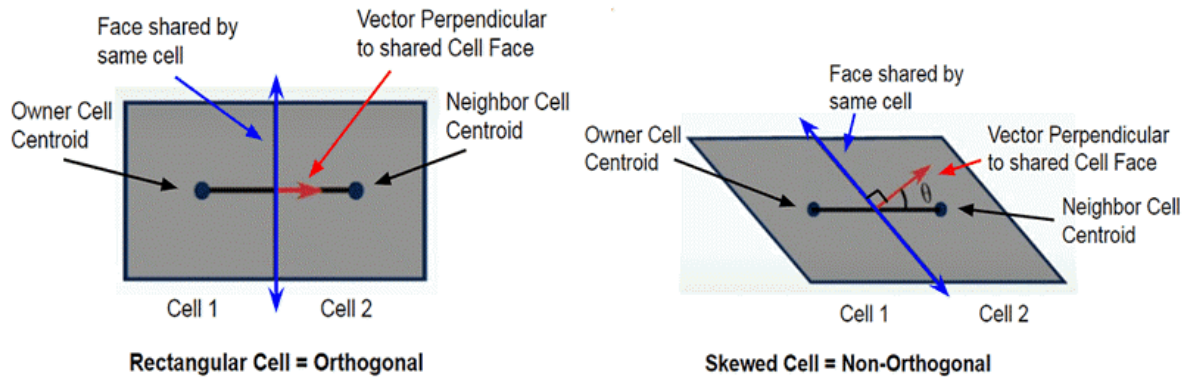
- Avoid high AR in recirculation, curved, or separated regions.
- High AR can distort the **solution gradients**, especially in directions **not aligned with the flow**, especially in boundary layers
- In boundary layers, **long thin cells are beneficial** if their long edge is tangential to the surface.

Tips:

- Use **anisotropic meshing** with **inflation layers** near walls.
- Keep AR reasonable unless physically justified (like for high-speed boundary layers).
- Apply inflation layers carefully — ensure the first layer height and growth rate do not create excessive AR.

3. **Orthogonality:** Orthogonality measures the angle between the face normal and the cell face.

- Orthogonality quantifies how well a mesh's face normals align with the vector connecting



adjacent cell centers.

- It is critical for accurate evaluation of diffusive and convective fluxes in finite-volume solvers.
- Low orthogonality can lead to interpolation errors and reduced solution accuracy, particularly in finite-volume methods. Higher orthogonality is desired.

Ideal Range: A well-shaped element should have an aspect ratio close to 1.

Recommended "O"	Remarks
1	Perfect alignment (ideal)
0	Vectors are perpendicular (poor).
<0.15	In Fluent considered poor and can lead to numerical issues.
0.01	Practical lower limit suggested in ANSYS forums

Why It Matters:

- Poor orthogonality causes errors in the calculation of **diffusive and convective fluxes**.
- May result in **non-physical results** or **non-convergence** in complex geometries.

Tips:

- Use **hex/quad elements** over **tet/prism** to improve orthogonality.
- Apply **face alignment** and **sweep methods** in structured mesh regions.

Orthogonality and non-orthogonality are two sides of the same mesh quality measure in CFD, but they describe opposite conditions

Aspect	Orthogonality	Non-Orthogonality
Definition	Measures alignment between the face-normal vector and the centroid-centroid vector in adjacent cells.	Angle between the centroid-centroid vector and the shared face normal.
Ideal Value	1.0 (perfectly aligned)	0° (perfectly orthogonal)
Poor Value	< 0.15 (in ANSYS Fluent flagged as poor quality)	> 70° (risky); > 80° (serious instability); > 85° likely to lead to divergence
Inverse Relationship	Orthogonality = 1 – (normalized non-orthogonality)	Non-orthogonality = arccos(face-normal · centroid-vector)
Why It Matters	Higher values ensure accurate evaluation of diffusion/ convection terms; low values cause errors.	Larger angles increase truncation errors in Laplacian terms; can lead to solver instability/ divergence
Solver Flags	Fluent marks cells with <0.15 as problematic	OpenFOAM and others treat angles >70° as critical; may require mesh correction or solver adjust
Improvement Tips	Use hex/quad structured meshes, face alignment, sweep methods, mesh smoothing	Similar actions: structured mesh in complex zones, add non-orthogonal correctors or remesh regions

Ref:

1. [What is Mesh Non-Orthogonality?](#)
2. [Simscale Mesh Quality](#)
3. ansyshelp.ansys.com,

4. Growth Ratio:

- The ratio by which the size of adjacent cells increases or decreases.
- Change in volume from one cell to another.

$$GR = \frac{Cell\ Size_{(i+1)}}{Cell\ Size_{(i)}}$$

Recommended Limit: < 1.2

- Gradual increase ensures **smooth flow resolution**.
- Sudden jumps can lead to **gradient discontinuities**.

Why It Matters:

1. Large jumps between cell sizes cause:
 - **Numerical diffusion**
 - **Loss of resolution**
 - **Spurious pressure waves**

Tips:

- When using **inflation layers**, maintain a **growth factor of ~1.1 to 1.2**.
- For wake regions or shear layers, avoid rapid size transitions.

5. Bias Factor:

- Often used in structured meshing, edge sizing, or line division.
- Higher BF value means smaller cells near the start and larger cells at the end.
- In ANSYS DesignModeler, bias is applied when setting edge divisions:
 - Use soft behaviour for smooth gradation.
 - Use hard behaviour if precise control is needed.
- The **growth factor and bias factor** are related but used in slightly different contexts in meshing tools. Both controls how element sizes change along a mesh edge, especially when you want cells to gradually increase or decrease in size (commonly used in boundary layers, wakes, etc.).

They are mathematically related through the number of divisions (N) along the edge.

$$\text{Bias Factor (BF)} = \frac{S_{max}}{S_{min}}$$

where,

- *S_{max}: maximum element size along the edge*
- *S_{min}: minimum element size*

Growth Factor (GR): If we divide a line into **N segments** using a geometric progression:

$$S_{min}, S_{min} \cdot r, S_{min} \cdot r^2, \dots, S_{min} \cdot r^{N-1}$$

Then,

$$B = r^{N-1} \quad \text{or} \quad r = B^{1/(N-1)}$$

Parameter	Definition	Where to Used
Growth Factor	The ratio of the size of one mesh cell to the previous one (e.g., 1.2 = 20% increase per cell)	ANSYS Meshing (Inflation layers), Fluent
Bias Factor	The ratio of the largest element size to the smallest along an edge	ANSYS DesignModeler, ICEM CFD, Pointwise

Example: We want to have a fine mesh near airfoil to capture boundary layer growth and separation

Dimension	No of Segments (N)	Bias factor (BF)	Growth Factor (GR)	Use Case
Radius	250	50000	1.045	Fine inflation layers near airfoil wall (boundary layer resolution)
Length	150	300	1.039	Graded mesh in wake region behind trailing edge

These values are conservative and effective for low-Re turbulence models like SST or Transition SST, especially when trying to achieve $Y^+ < 1$.

Ref:

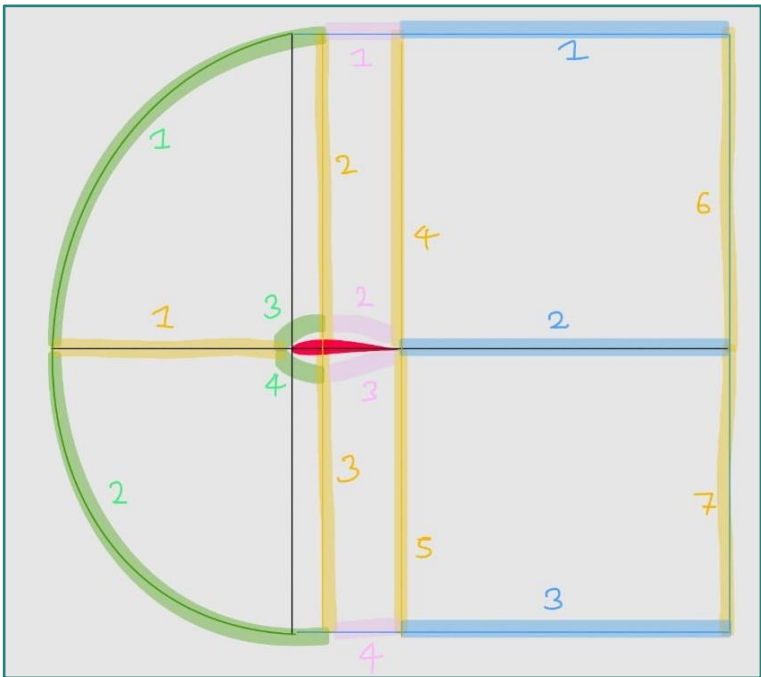
1. <https://cfd.ninja/ansys-meshing/ansys-meshing-bias-factor/>
2. [CFD Online](#)

Bias Factor Ranges and Applications in CFD Meshing

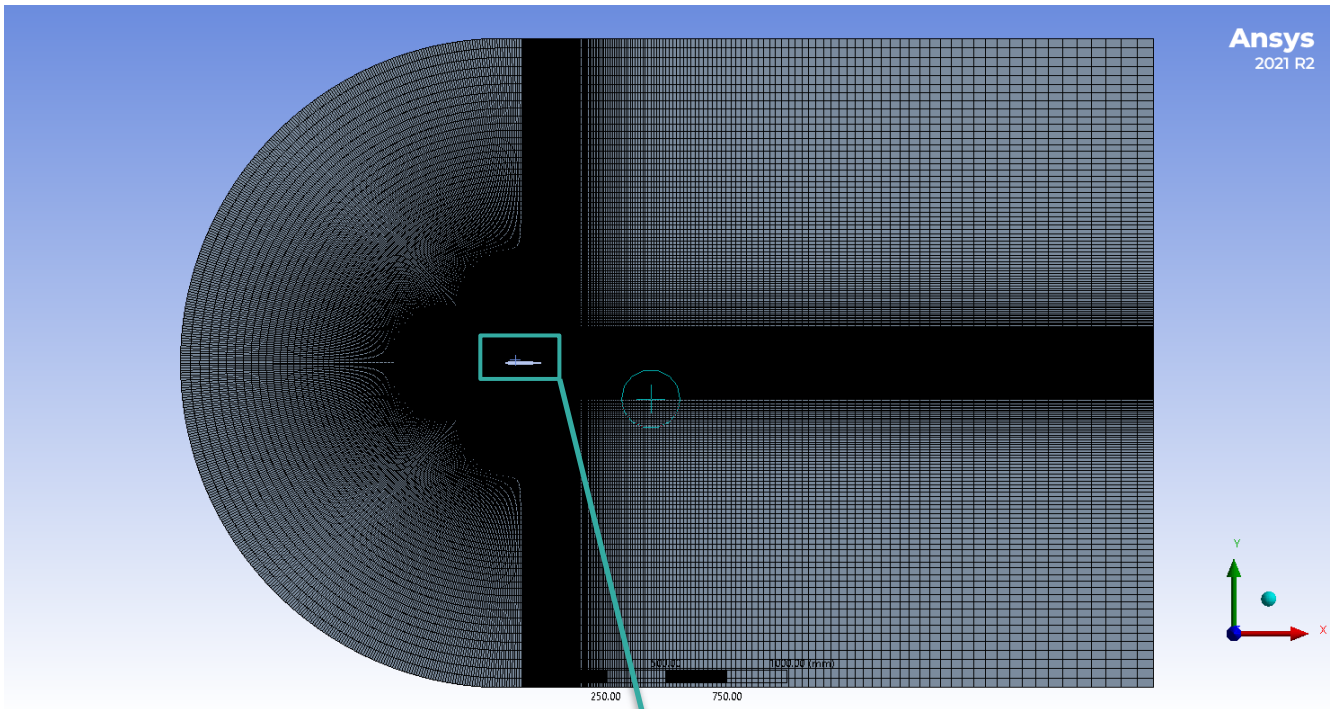
Bias Factor	Effect on Mesh	Typical Use Case	Remarks
1	Uniform mesh (equal cell sizes)	General domain meshing; far-field regions	No refinement; good for uniform flow zones
1.1 – 3	Very mild grading	Inlet/outlet regions with gradual flow variation	Smooth transition; conservative
3 – 10	Moderate refinement toward one end	Boundary layers; regions near walls or LE/TE	Balances resolution and cell count
10 – 30	Strong refinement at one end	Airfoil surfaces, heat transfer surfaces	Good for Y ⁺ control (with growth factor ~1.1–1.2)
30 – 100	Very strong grading	Sharp gradients; thin wakes; high aspect boundary layers	Risk of high aspect ratio or skewness if used improperly
> 100	Extreme refinement	Only in special cases (e.g., near shock, viscous sublayer in LES/DNS)	May cause poor mesh quality or solver instability if not aligned with flow

Behaviour	Definition	When to Use
Hard	Enforces the exact number of divisions and sizing regardless of global mesh controls	<ul style="list-style-type: none">- Boundary layers- Shock zones- High-gradient regions- Leading Edge- Stagnation Zone
Soft	Suggests the mesh size/division but lets the mesher adjust based on other settings	<ul style="list-style-type: none">- Far-field- Wake regions- Transition Region- Areas without strong gradients

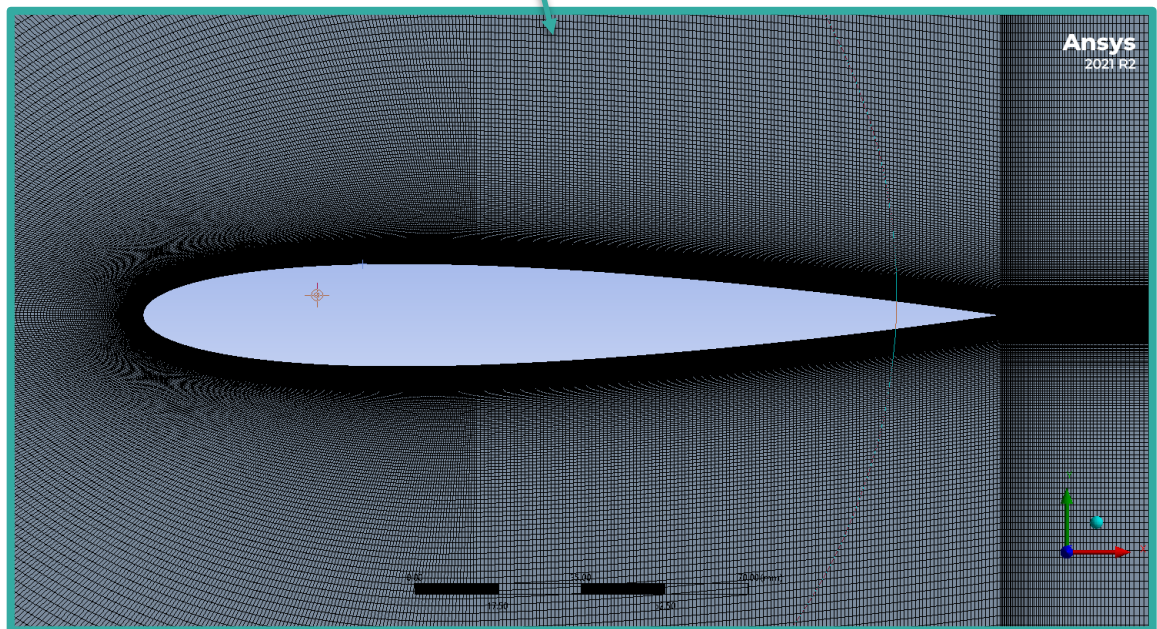
Meshing Details:

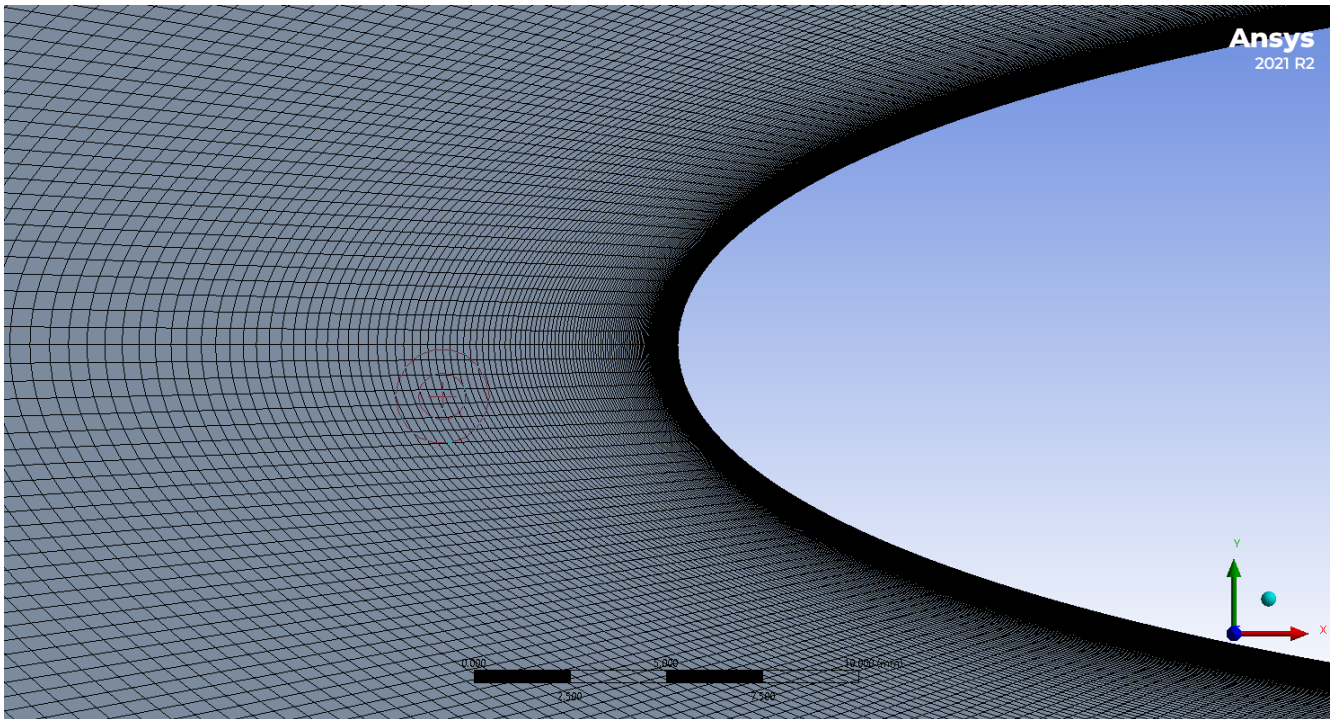


Mesh Type	Bias Type	Bias Factor	Coarse	Medium	Fine	Very Fine
ARC (4)	Hard	-	106	150	212	300
Radius (7)	Soft	50000	177	250	354	500
Length (3)	Hard	300	106	150	212	300
Airfoil back (4)	Hard	-	71	100	141	200
# Elements			100,182	200,000	400,020	800,000
# Nodes			100,819	200,900	401,292	801,800
Y+ (mm)			0.00162580	0.00115850	0.00081497	0.00057718

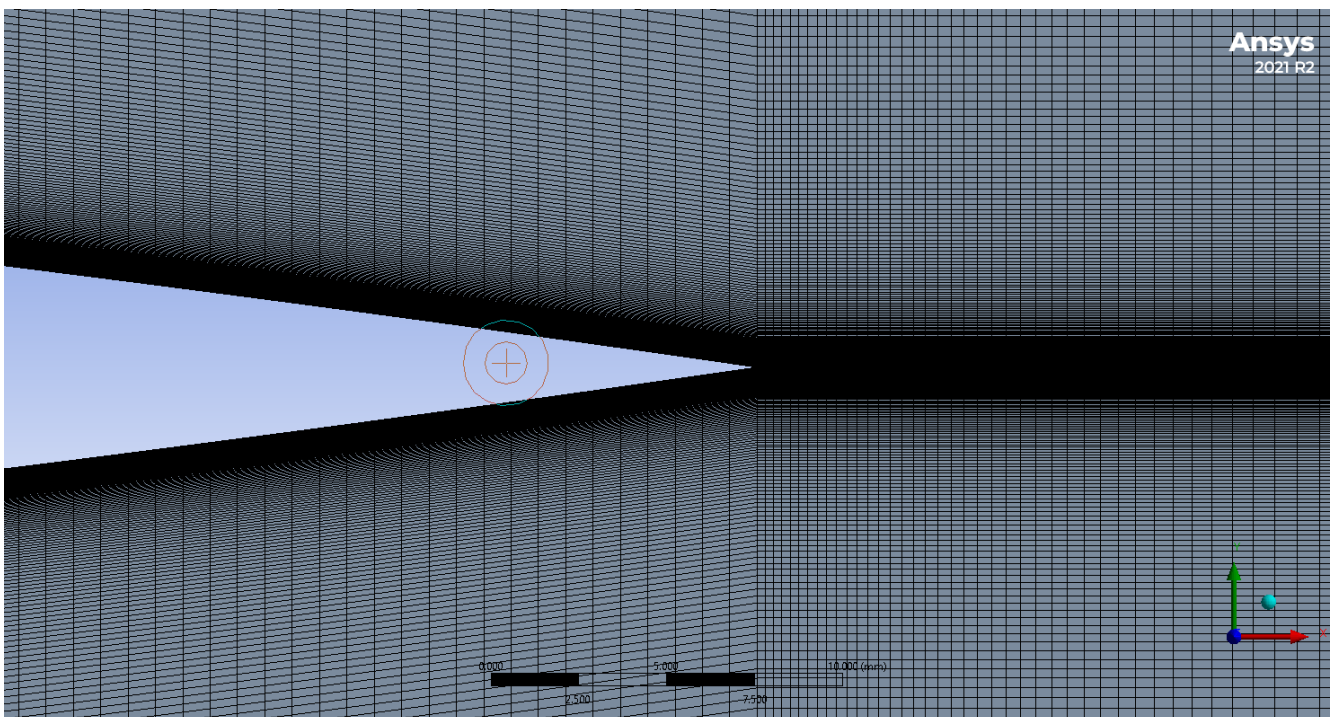


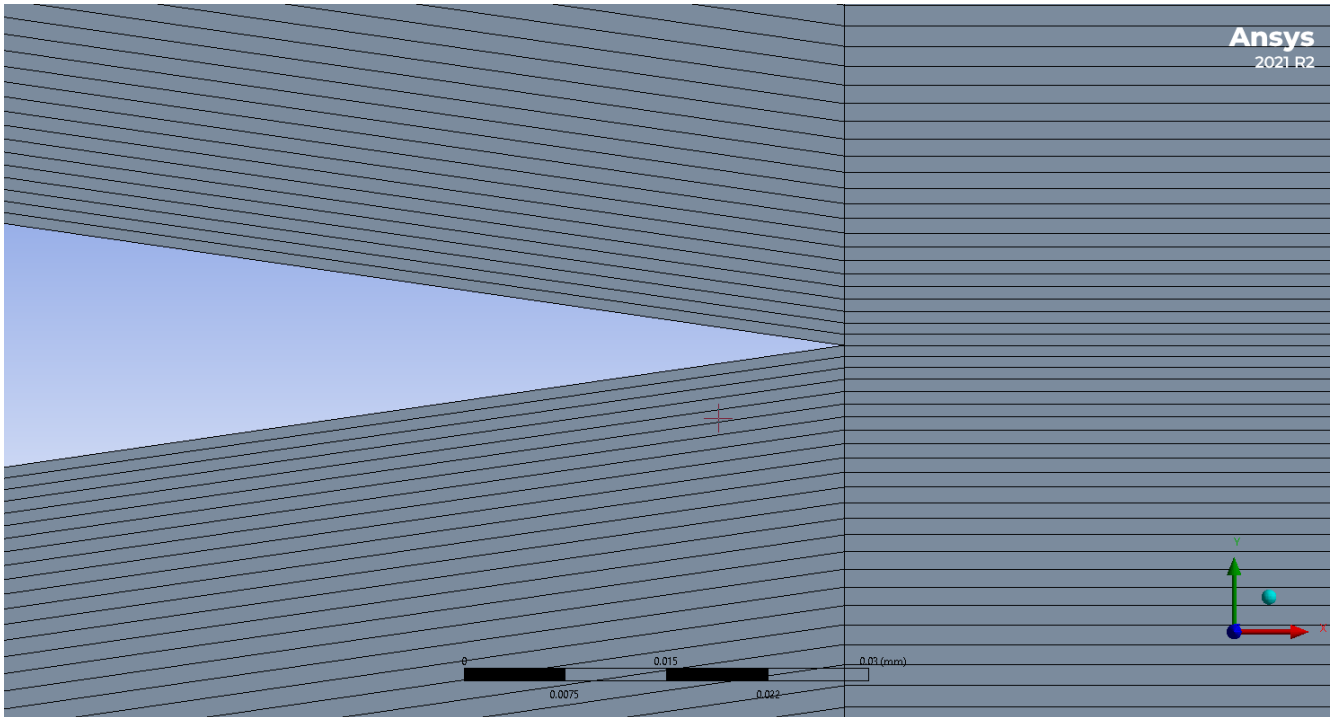
Structured Fine Mesh Image of NACA 0012



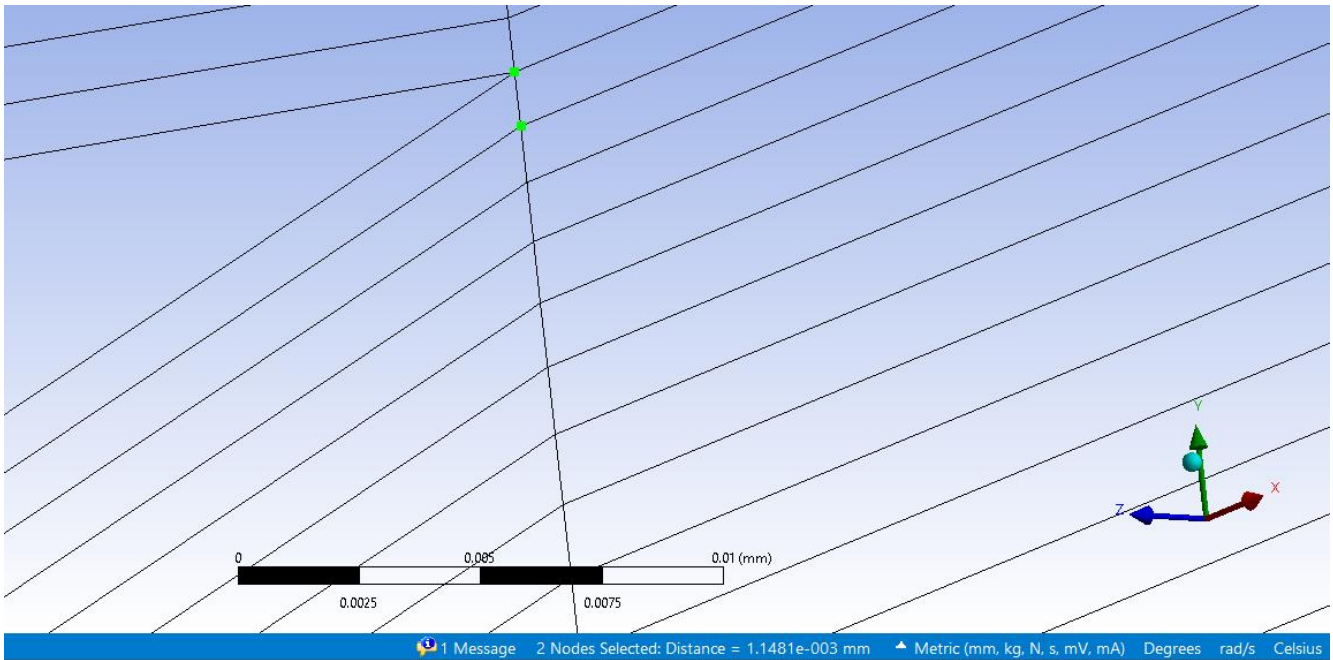


Magnified view of mesh at leading and trailing edges

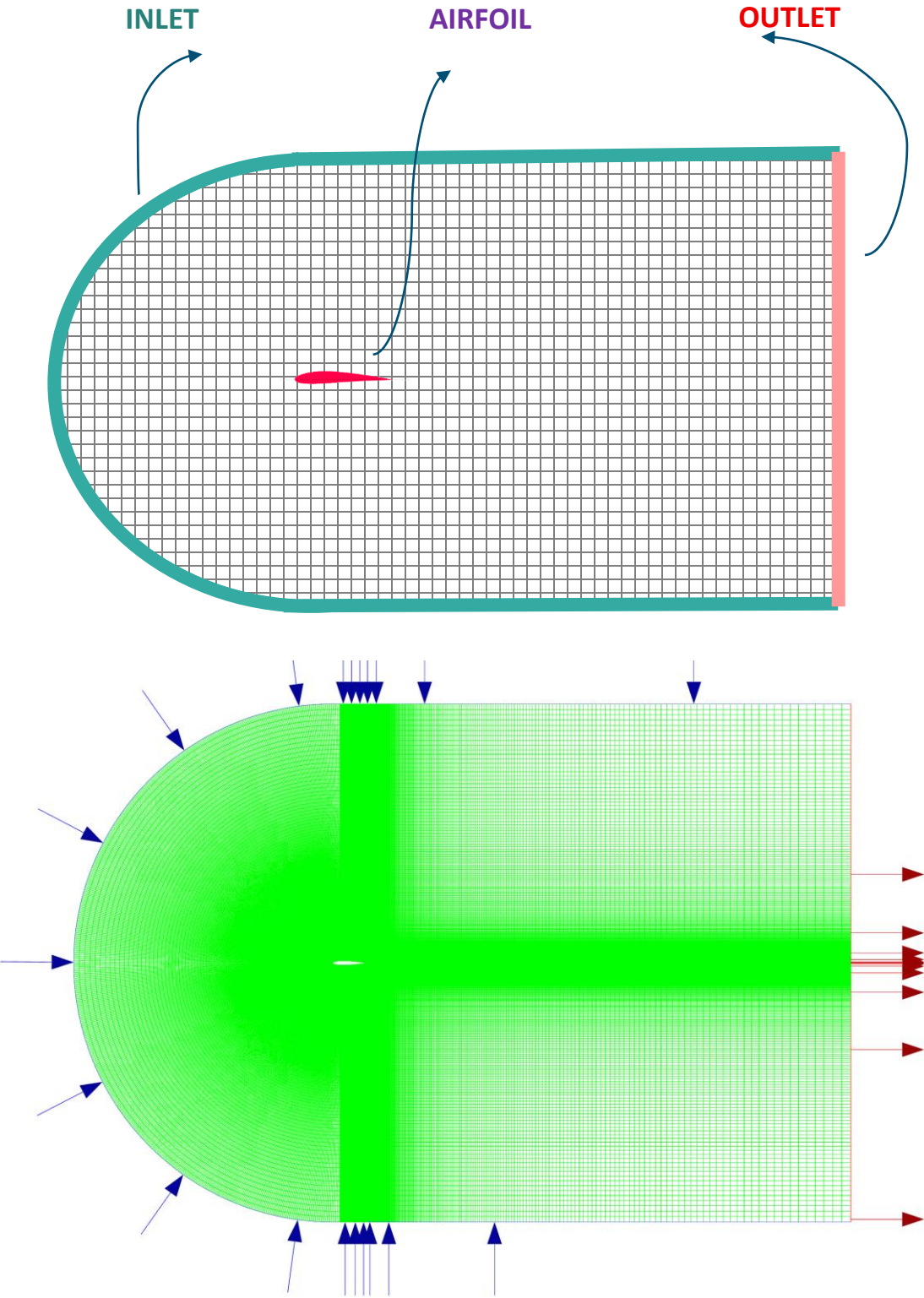




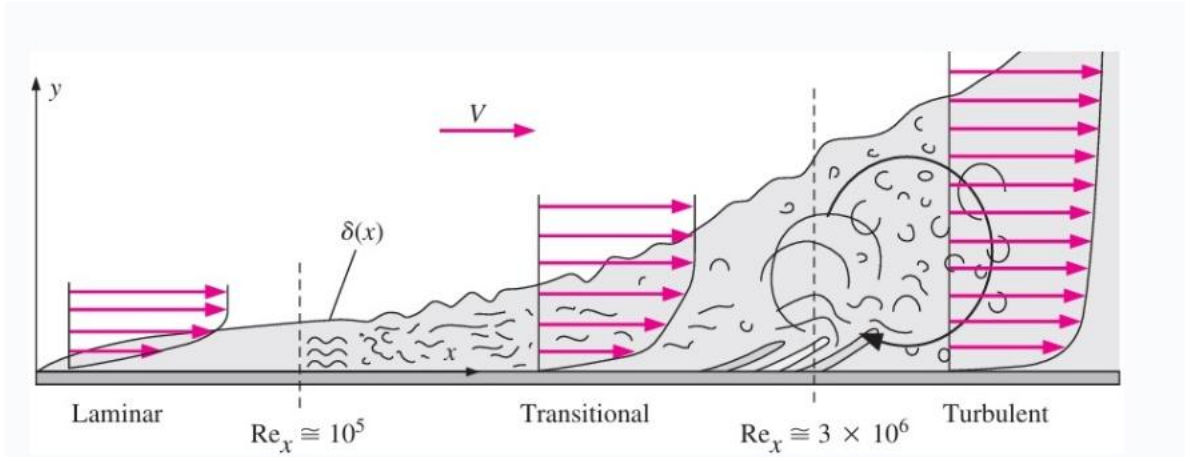
First layer growth checking near trailing edge



Named Selection:



Y+ and First layer height



y^+ is a *dimensionless wall distance* representing how close the first cell center lies to the wall, in wall units:

$$y^+ = \frac{u_\tau \cdot y_p}{\nu}$$

Where

- $u_\tau = \sqrt{\frac{\tau_w}{\rho}}$ = friction velocity
- y_p = **Absolute Distance from the centroid of the wall**
- $\nu = \frac{\mu}{\rho}$ = Kinematic viscosity

One can interpret y^+ as a local Reynolds number, which means that its magnitude can be expected to determine the relative importance of viscous and turbulent processes

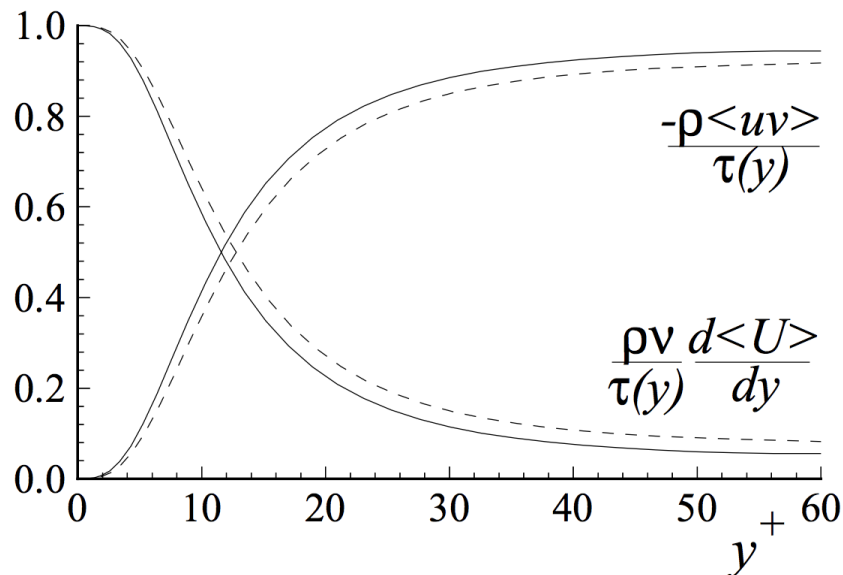
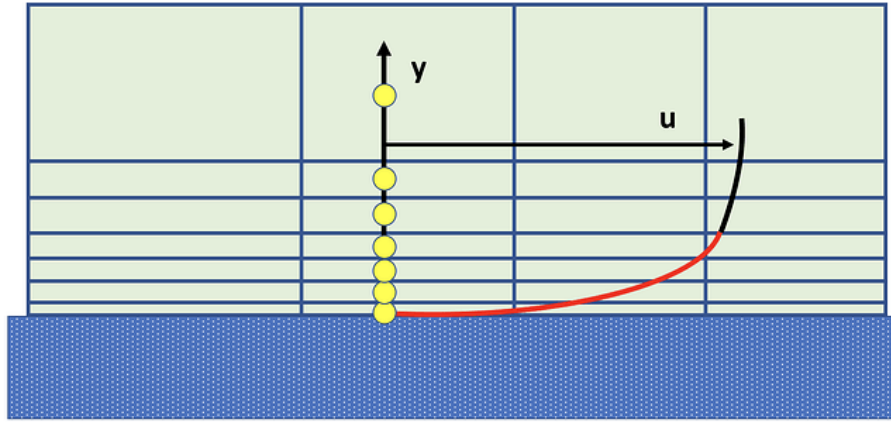


Figure: contributions to the total stress from the viscous and Reynolds stresses in the near wall region of channel flow

One can easily see that if we are in the viscous wall region with $y^+ < 50$, there is a direct effect of the viscosity on the shear stress. Conversely in the outer layer with $y^+ > 50$, the effect of viscosity is negligible.



For the sake of completeness, let's briefly introduce the friction velocity u_τ . It should be evident that the viscosity ν and wall shear stress τ_w are important parameters. From these quantities and ρ we define the viscous scales being appropriate viscous scales and length scales in the near-wall region.

$$u_\tau = \sqrt{\frac{\tau_w}{\rho}},$$

$$\tau_w = \rho \nu \left(\frac{d \langle U \rangle}{dy} \right)_{y=0}$$

The dimensionless velocity is given by:

$$u^+ = \frac{u}{u_\tau}$$

The basic idea is that additional boundary conditions are applied at some distance to the wall to fulfill the **log-law**. Hence the additional equations introduced by the turbulence model are not solved close to the wall. Depending on the turbulence model used, different wall functions must be applied to respective fields of the turbulence model, which means that k- ϵ has different wall functions than the k- ω model.

Due to this fact, the different turbulence models and the associated wall functions require different values of y^+ as well as different spatial resolutions of the near wall area. Please note that if the **log-law** region is **resolved geometrically by the mesh, no wall functions must be applied!** The downside of this approach is that depending on the simulation, such low y^+ values are **hard to create** during the pre-processing step (meshing) or even **undesirable**, as this will **significantly decrease the time step!**

Best Practices

- In the pre-processing stage of the simulation we need to make sure that y^+ is in the desired range. For that, we have to calculate the size of the first layer of our mesh.
- Check the y^+ value after the computation again as the real flow field will develop during the simulation and a remesh might be necessary.
- To avoid a remesh (and the use of too many core hours), it makes sense to really calculate the value beforehand.

Wall Regions and Layers as well as their properties:

1. The viscous sublayer ($y^+ < 5$)

In the viscous layer, the fluid is dominated by the viscous effect, so it can be assumed that the Reynolds shear stress is negligible. The “linear velocity law” is given by:

$$u^+ = y^+$$

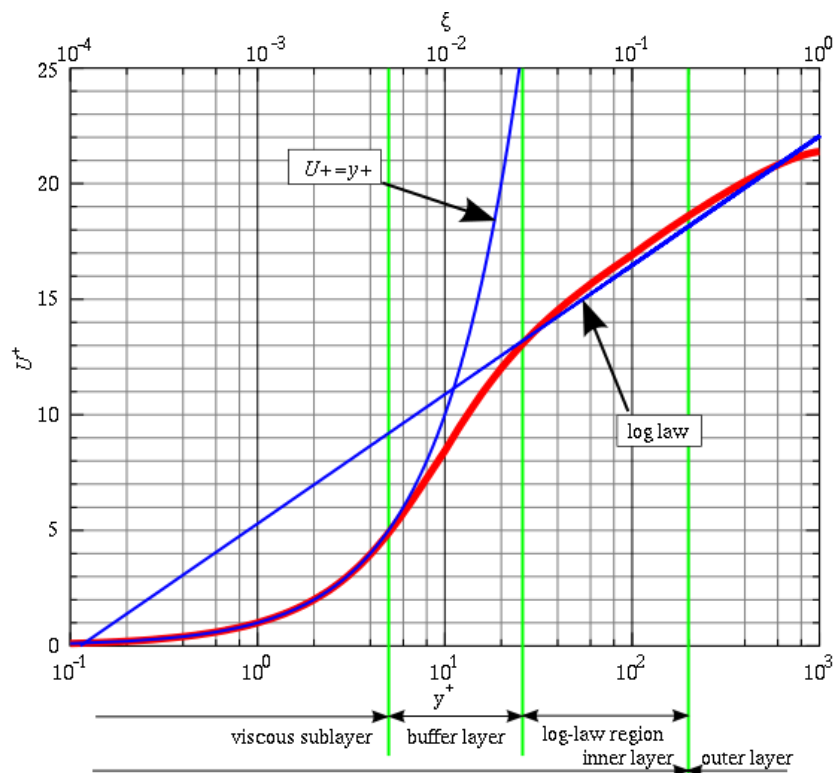
2. The logarithmic area ($y^+ > 30$)

In the logarithmic layer, turbulence stress dominates the flow and velocity profile varies very slowly with a logarithmic function along the distance y . Below Eqn. describe this region with the Karman constant κ of 0.41 and the constant $B=5.2$.

$$u^+ = \frac{1}{\kappa} \ln(y^+) + B$$

3. The Buffer layer ($5 < y^+ < 30$)

The buffer layer is the transition region between the viscosity-dominated region and turbulence-dominated part of the flow. Viscous and turbulent stresses are of similar magnitude and since it is complex, the velocity profile is not well defined and the original wall functions avoid the first cell center located in this region



Estimating the First Cell Height for given Y^+

1. Calculate Reynolds Number (Re)

$$Re = \frac{\rho \cdot U_{\infty} \cdot L}{\mu}$$

- ρ = fluid density
- U_{∞} = freestream velocity
- L = characteristic length (e.g., Chord length, pipe diameter, plate length)
- μ = dynamic viscosity

2. Determine the skin friction coefficient

Flow Type	Correlation for C_f
Internal Flow	$C_f = 0.079 \cdot Re^{-0.25}$
External Flow	$C_f = 0.058 \cdot Re^{-0.2}$
Flat Plate (laminar)	$C_f = 0.0576 \cdot Re^{-0.2}$
Flat Plate (turbulent)	$C_f = 0.026 \cdot Re^{-1/7}$

3. The wall shear stress τ_w can be calculated from the skin friction coefficient C_f

$$\tau_w = \frac{1}{2} \cdot C_f \cdot \rho \cdot U_{\infty}^2$$

4. Next, we calculate:

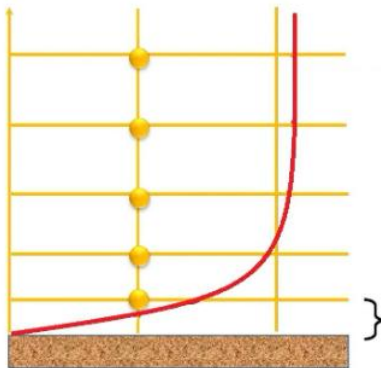
$$u_{\tau} = \sqrt{\frac{\tau_w}{\rho}}$$

5. Calculate Kinematic Viscosity:

$$\nu = \frac{\mu}{\rho}$$

6. First Cell Height (y_p):

$$y^+ = \frac{u_{\tau} \cdot y_p}{\nu}$$



$$y_p = \frac{y^+ \cdot \nu}{u_{\tau}}$$

First Cell Height Results:

Parameter	Value	Formula / Description
Chord Length, L (m)	0.180	Input
Freestream Velocity, U (m/s)	25.00	Input
Fluid Density, ρ (kg/m ³)	1.2250	Standard atmospheric air
Dynamic Viscosity, μ (kg/m·s)	1.812053×10^{-5}	From Sutherland's formula or viscosity calculator
Reynolds Number, Re_x	304,212.99	$Re = \frac{\rho \cdot U_{\infty} \cdot L}{\mu}$
Skin Friction Coefficient, C_f	0.004282154	$C_f = 0.026 \cdot Re^{-1/7}$
Wall Shear Stress, τ_w (Pa)	1.77737	$\tau_w = \frac{1}{2} \cdot C_f \cdot \rho \cdot U_{\infty}^2$
Friction Velocity, u_{τ} (m/s)	1.20454	$u_{\tau} = \sqrt{\frac{\tau_w}{\rho}}$
Target y^+	1.0	Desired value
First Cell Height, Δy (m)	1.228043×10^{-5}	$y_p = \frac{y^+ \cdot \nu}{u_{\tau}}$
First Cell Height, Δy (mm)	0.01228 mm	Converted to mm

- **Pre-Simulation Insight:** *With $\Delta s \approx 0.0123$ mm, $y^+ \approx 1$* , so viscous sublayer will be fully resolved — ideal for low-Re turbulence models like k- ω SST.
- **ANSYS Value for First Layer:** This ANSYS mesh is finer than required ($y^+ < 1$), so it will resolve boundary effects very well but increase computational cost.

In ANSYS Settings	Value
First Cell Height (m)	1.1585×10^{-6} m
First Cell Height (mm)	0.00116 mm

Ref:

1. https://imechanica.org/files/fluent_13.0_lecture06-turbulence.pdf
2. <https://www.flowthermolab.com/y-plus-calculator-for-internal-and-external-flows/>
3. [Simscale](#)
4. [Cadence](#)

Dynamic Viscosity (μ) Calculation

Sutherland’s Law (General Formula)

$$\mu = \mu_0 \cdot \left(\frac{T}{T_0}\right)^{3/2} \cdot \left(\frac{T_0 + C}{T + C}\right)$$

Where:

- μ_0 : Reference dynamic viscosity (kg/m·s)
- T_0 : Reference temperature (K)
- C: Sutherland’s constant (K)
- T : Operating temperature (K)

Parameter	LMNO / Crane (R-based)	CRC / Sutherland	NASA / COMSOL
T_0 (K)	291.15	524.07 R	273.15
C (K)	120	110.4	110.4
μ_0 (kg/m·s)	1.827×10^{-5}	1.827×10^{-5}	1.716×10^{-5}
T (K)	293.00	518.67 R	293.00
$a = T_0 + C$	411.15	634.47	383.55
$b = T + C$	413.00	629.07	403.40
μ (kg/m·s)	1.81205×10^{-5}	1.81238×10^{-5}	1.7893×10^{-5}
Notes	Common in HVAC, engineering fluids (CFD)	Slightly varies due to unit base	Used in aerospace & High- fidelity / Scientific code

- The small variation (~1–1.3%) between models affects high-fidelity wall treatment like y^+ -based mesh design.

Solver Setup

1. General Setup

Setting	Selected Option	Justification
1. Type	Pressure-Based	<ul style="list-style-type: none">• Best for incompressible or mildly compressible flows ($Ma < 0.3$).• Suitable for external airfoil flow at 25 m/s.
2. Velocity Formulation	Absolute	The airfoil is stationary, and air is moving — absolute velocity is appropriate.
3. Time	Steady	Appropriate for steady aerodynamic analysis (C_l , C_d). Unsteady effects like vortex shedding can be explored later using transient setup.
4. 2D Space	Planar	A 2D simulation of the airfoil cross-section is accurate and computationally efficient.
5. Gravity	Unchecked	Gravity has no significant influence in aerodynamic force calculation; not relevant in this case.

General

Mesh

Scale...

Check

Report Quality

Display...

Units...

Solver

Type

☒ Pressure-Based

☐ Density-Based

Velocity Formulation

☒ Absolute

☐ Relative

Time

☐ Steady

☒ Transient

2D Space

☒ Planar

☐ Axisymmetric

☐ Axisymmetric Swirl

☐ Gravity

2. Viscous Model

Viscous Model

Model

☐ Inviscid

☐ Laminar

☐ Spalart-Allmaras (1 eqn)

☐ k-epsilon (2 eqn)

☒ k-omega (2 eqn)

☐ Transition k-k-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

Options

☐ Curvature Correction

☐ Corner Flow Correction

☒ Production Kato-Launder

☐ Production Limiter

Transition Options

Transition Model

gamma-transport-eqn

☐ Include Crossflow Transition

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

CTU1

User-Defined Functions

Turbulent Viscosity

none

OK

Cancel

Help

Constant	Value	Description
Alpha*_inf	1	Diffusion term in turbulence transport equations
Alpha_inf	0.52	Turbulence viscosity blending factor
Alpha_0	0.11111	SST-specific parameter
Beta*_inf	0.09	Dissipation term constant
R_beta, R_k, R_w	8, 6, 2.95	Reynolds stress and blending control parameters
a1	0.31	SST model constant for blending functions
Beta_i (Inner/Outer)	0.075 / 0.0828	Adjusts ω equation in boundary layer
TKE Prandtl # (Inner)	1.176	Turbulent kinetic energy transport

Setting	Value / Status	Rationale
Turbulence Model	k-omega SST (2 eqn)	Handles adverse pressure gradients and near-wall effects effectively
k-omega Model	SST	Blends k- ϵ (outer) and k- ω (inner) for robustness and accuracy <ul style="list-style-type: none">k-ϵ model in free stream (good for outer region)k-ω model near the wall (accurate for boundary layer)
Low-Re Corrections	Enabled	Necessary for low Reynolds number regime and accurate near-wall resolution
Transition Model	gamma-transport-eqn	<ul style="list-style-type: none">Enables modeling of laminar-to-turbulent transition, crucial at low Re no.Helps capture laminar separation bubble
Include Crossflow Transition	Not Enabled	Not needed for 2D symmetric airfoil (no 3D crossflow effects)
Production Kato-Launder	Enabled	Controls turbulence production near stagnation zones (e.g., airfoil leading edge)
Curvature Correction	Disabled (Not relevant in current 2D setup)	Improves accuracy for rotating or curved surfaces.
Corner Correction		For 3D corner eddies

3. Fluid Model:

selected is Air with:

- Density: 1.225 kg/m³ (Sea Level)
- Dynamic Viscosity: 1.789e-05 kg/m-s

4. Inlet Conditions:

AOA Sweep with Velocity Decomposition

At a given angle of attack (α), the total velocity V is decomposed into x and y components:

$$V_x = V \cdot \cos(\alpha)$$

$$V_y = V \cdot \sin(\alpha)$$

Lift and drag are the aerodynamic forces and to use them in simulations (e.g., for structural input or stability), we resolve them into Cartesian x and y components.:

- Drag (D): along the freestream direction (opposes motion)

$$D_x = D \cdot \cos(\alpha)$$

$$D_y = D \cdot \sin(\alpha)$$

- Lift (L): perpendicular to the freestream direction

$$L_x = -L \cdot \sin(\alpha)$$

$$L_y = L \cdot \cos(\alpha)$$

Negative sign in L_x because lift is perpendicular to freestream and we're rotating from vertical to horizontal axis.

AOA	V (m/s)	V _x	V _y	D _x	D _y	L _x	L _y
0	25	25	0	1	0	0	1
2	25	24.9848	0.872487	0.999391	0.034899	-0.0349	0.999391
4	25	24.9391	1.743912	0.997564	0.069756	-0.06976	0.997564
6	25	24.863	2.613212	0.994522	0.104528	-0.10453	0.994522
8	25	24.7567	3.479328	0.990268	0.139173	-0.13917	0.990268
10	25	24.6202	4.341204	0.984808	0.173648	-0.17365	0.984808
12	25	24.4537	5.197792	0.978148	0.207912	-0.20791	0.978148
14	25	24.2574	6.048047	0.970296	0.241922	-0.24192	0.970296
16	25	24.0315	6.890934	0.961262	0.275637	-0.27564	0.961262

Velocity Inlet

Zone Name

Momentum Thermal Radiation Species DPM Multiphase Potential Structure UDS

Velocity Specification Method

Reference Frame

Supersonic/Initial Gauge Pressure [Pa]

X-Velocity [m/s]

Y-Velocity [m/s]

Turbulence

Specification Method

Intermittency

Turbulent Intensity [%]

Turbulent Length Scale [m]

Apply Close Help

5. Reference Values

Computed from the inlet with Hybrid Initialization

Reference Values

Compute from

Reference Values

Area [m²]
 Density [kg/m³]
 Depth [m]
 Enthalpy [J/kg]
 Length [m]
 Pressure [Pa]
 Temperature [K]
 Velocity [m/s]
 Viscosity [kg/(m s)]
 Ratio of Specific Heats
 Yplus for Heat Tran. Coef.

Reference Zone

Solution Initialization

Initialization Methods

☒ Hybrid Initialization
☐ Standard Initialization

More Settings... Initialize

Patch...

Reset DPM Sources Reset LWF Reset Statistics

6. Solution Setup

Solution Methods?

Pressure-Velocity Coupling

Scheme

Coupled

Flux Type

Rhie-Chow: distance based

☐ Auto Select

Spatial Discretization

Gradient

Green-Gauss Node Based

Pressure

Second Order

Momentum

Second Order Upwind

Intermittency

Second Order Upwind

Turbulent Kinetic Energy

Second Order Upwind

Specific Dissipation Rate

Second Order Upwind

Transient Formulation

☐ Non-Iterative Time Advancement

☐ Frozen Flux Formulation

☒ Pseudo Transient

☐ Warped-Face Gradient Correction

☐ High Order Term Relaxation

Structure Transient Formulation

Default

Run Calculation?

Check Case...

Update Dynamic Mesh...

Pseudo Transient Settings

Fluid Time Scale

Time Step Method

Automatic

Time Scale Factor

1

Length Scale Method

Conservative

Verbosity

0

Parameters

Number of Iterations

1600

Reporting Interval

1000

Profile Update Interval

1

Solution Processing

Statistics

☐ Data Sampling for Steady Statistics

Data File Quantities...

Solution Advancement

Calculate

Setting	Selected Option	Explanation
Scheme	Coupled	Pressure and momentum equations solved simultaneously for faster convergence in steady simulations.
Flux Type	Rhie-Chow: distance based	Prevents pressure-velocity decoupling on co-located grids. Recommended in Fluent for Coupled solver.
Gradient	Green-Gauss Node Based	Accurate for boundary layers, uses node-based gradient reconstruction.
Pressure	Second Order	Improves accuracy in capturing pressure variations around the airfoil (e.g. in the stagnation region).
Momentum	Second Order Upwind	Better accuracy in predicting flow separation and wake characteristics.
Intermittency	Second Order Upwind	Used in transition-sensitive turbulence models (e.g., k- κ - ω or γ - θ).
Turbulent Kinetic Energy	Second Order Upwind	Captures energy in turbulence more precisely.
Specific Dissipation Rate (ω)	Second Order Upwind	Key variable in SST models, accuracy here affects boundary layer modeling.
Pseudo Transient	Enabled	Helps steady-state solver converge better by introducing pseudo-time stepping.
Warped-Face Gradient Correction	Not enabled	Can be used for poor quality meshes; not necessary if mesh quality is good.

Mesh Quality check

After Performing the Mesh Quality check in Ansys it was found that:

(for very fine mesh 9e+05 elements)

Metric	Value	Acceptable Range	Remarks
Min Orthogonal Quality	0.558	> 0.1 (acceptable), > 0.7 (good)	Acceptable ; however, improving toward 0.7+ ensures better stability.
Max Aspect Ratio	82,487.7	< 100 (preferred), < 1000 (okay)	Too high – risk of numerical diffusion, poor accuracy near wall.
Min Cell Volume (m³)	4.97×10^{-11}	> 1e-12 (typical lower bound)	Very fine ; helps with boundary layer resolution if properly placed.
Max Cell Volume (m³)	1.383×10^{-3}	N/A	Depends on far-field size – okay if in outer domain.
Min Face Area (m²)	5.75×10^{-7}	Small for near-wall expected	Likely from refined region near the airfoil.
Max Face Area (m²)	4.78×10^{-2}	Large for far-field okay	Acceptable for outer domain boundaries.
Domain x Extents (m)	-1.35 to 2.70	> 15× chord upstream/downstream	Good : Chord = 0.18 m → extents are large enough for external flow.
Domain y Extents (m)	-1.35 to 1.35	> 15× chord vertically	Matches best practices for external simulations.

Mesh Refinement in CFD

- **CFD Verification and Validation** are fundamental processes for ensuring simulation accuracy
- **Grid Convergence Index (GCI) Method** provides systematic approach to mesh independence studies.

1. Verification vs. Validation

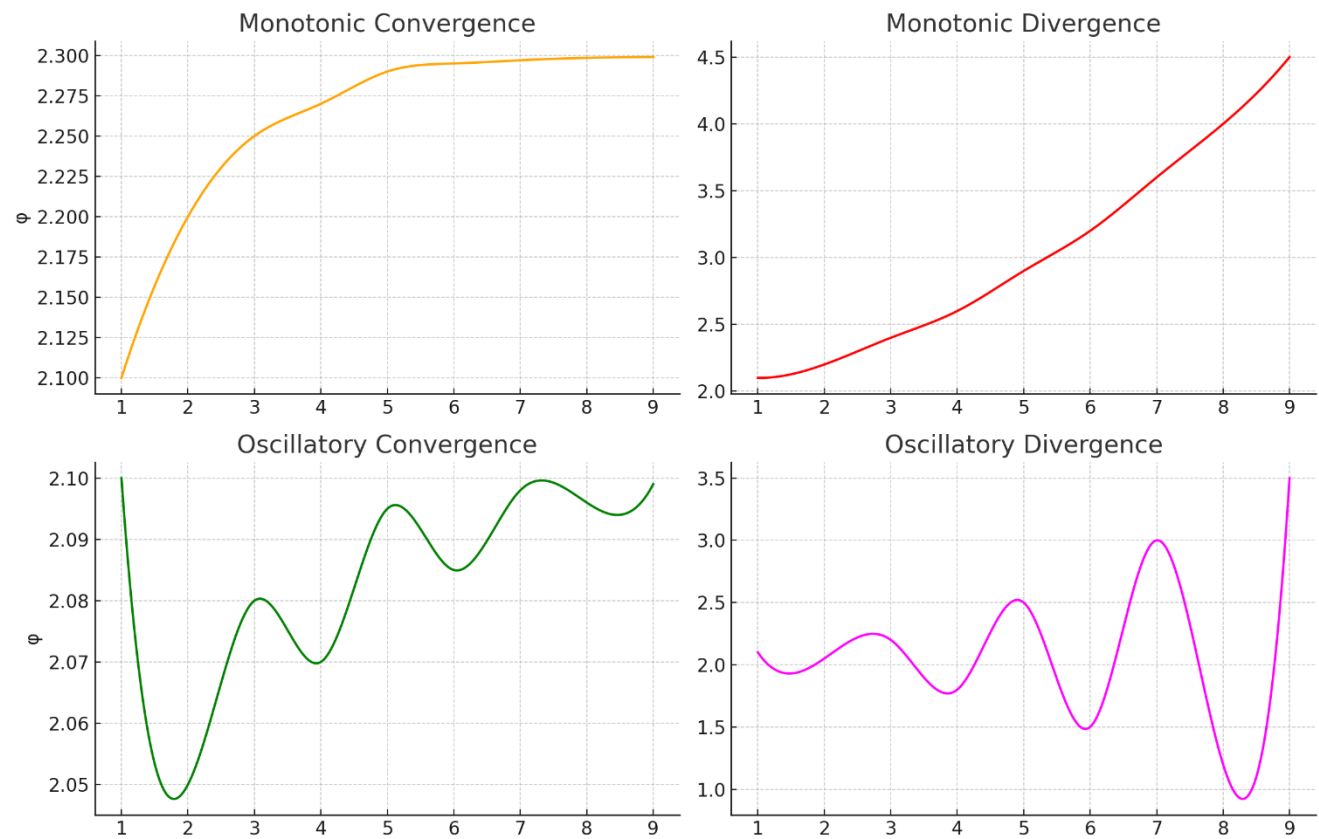
Aspect	Verification	Validation
Definition	Confirms that the numerical solution approaches the exact mathematical solution	Compares CFD results to experimental or real-world data
Purpose	Checks if we have solved the equations correctly	Checks if we have solved the correct physical problem
Focus	Mathematical accuracy of the numerical method	Physical accuracy of the mathematical model

2. Types of CFD Errors

Error Type	Definition / Cause	Reduction Method	Impact / Notes
Iterative Error	Difference between a CFD solution and the solution converged to machine accuracy	Ensure convergence of normalized residuals	Residuals should decrease by at least 3 orders of magnitude
Round-off Error	Caused by limited number of digits available on the machine or numerical schemes	Use double precision calculations	Generally minimal when double precision is enabled in the solver
Discretization Error	Difference between exact mathematical solution and the machine-accurate solution	Perform mesh refinement and GCI study	Main focus of mesh independence; assumes iterative and round-off errors are negligible

3. Variable Behavior Types in Mesh Refinement

Behavior Type	Example Values	Characteristic	GCI Applicability
Monotonic Convergence	$\phi = 2.1, 2.2, 2.25, 2.27\dots$	Steady approach to final value	Works well
Monotonic Divergence	$\phi = 2.1, 2.2, 2.4, 2.6\dots$	Continuous increase/decrease	Not suitable
Oscillatory Convergence	$\phi = 2.1, 2.05, 2.08, 2.07\dots$	Oscillates around final value	Works but may be inaccurate if oscillations don't occur at min/max
Oscillatory Divergence	$\phi = 2.1, 2.05, 2.2, 1.8, 3.4\dots$	Increasing oscillation amplitude	Not suitable



Grid Convergence Index (GCI) Method

Theoretical Foundation

- Based on Richardson Extrapolation
- Only works for monotonic convergence and oscillatory convergence
- Assumes monotonic convergence for most accurate results

Mesh Refinement Strategy

- Cell Size Reduction
- Goal: Reduce cell size in even, structured manner
- 2D Case: Halving cell size $\rightarrow 4\times$ mesh elements ($2^2 = 4$)
- 3D Case: Halving cell size $\rightarrow 8\times$ mesh elements ($2^3 = 8$)

Practical Refinement

- To double mesh size in 2D: Reduce cell size by factor of $\sqrt{2}$
- To double mesh size in 3D: Reduce cell size by factor of $\sqrt[3]{2}$

GCI Method Step-by-Step Procedure

Step 1: Define Characteristic Cell Size

2D Meshes:

$$h = \left[\frac{1}{N} \sum_{i=1}^N A_i \right]^{1/2}$$

3D Meshes:

$$h = \left[\frac{1}{N} \sum_{i=1}^N V_i \right]^{1/3}$$

Where:

- A_i, V_i = area, volume of *ith* cell
- N = total number of cells

Step 2: Create Three Meshes

- Mesh sizes: $h_1 < h_2 < h_3$ (fine < medium < coarse)
- Refinement factor: $r > 1.3$
- Refinement ratios:
 - $r_{21} = h_2/h_1$

$$\circ \quad r_{32} = h_3/h_2$$

Example Calculations:

- **2D case:** $r = 2\text{mm}/(2\text{mm}/\sqrt{2}) = \sqrt{2} = 1.414 > 1.3 \checkmark$
- **3D case:** $r = 2\text{mm}/(2\text{mm}/\sqrt[3]{2}) = \sqrt[3]{2} = 1.26 \approx 1.3 \checkmark$
- We can go for 1.5 or 1.6 then fine mesh would takes too long for convergence.

Step 3: Calculate Apparent Order of Accuracy

$$p = \frac{1}{\ln(r)} \times \left| \ln\left(\frac{\varepsilon_{32}}{\varepsilon_{21}}\right) \right|$$

Where:

- $\varepsilon_{32} = \varphi_3 - \varphi_2$
- $\varepsilon_{21} = \varphi_2 - \varphi_1$

Convergence Criteria:

- $0 < \varepsilon_{21}/\varepsilon_{32} < 1$: Monotonic convergence
- $-1 < \varepsilon_{21}/\varepsilon_{32} < 0$: Oscillatory convergence
- $\varepsilon_{21}/\varepsilon_{32} < -1$: Oscillatory divergence
- $\varepsilon_{21}/\varepsilon_{32} > 1$: Monotonic divergence

Step 4: Calculate Extrapolated Values

$$\varphi_{ext}^{21} = \frac{(r^p \varphi_1 - \varphi_2)}{(r^p - 1)}$$

This represents the theoretical value as cell size approaches zero.

Similarly, for φ_{ext}^{32}

Step 5: Calculate Error Estimates

Approximate Relative Error

$$e_a^{21} = \frac{(\varphi_1 - \varphi_2)}{\varphi_1}$$

Extrapolated Relative Error

$$e_{ext}^{21} = \frac{(\varphi_{ext}^{21} - \varphi_1)}{\varphi_{ext}^{21}}$$

Fine Grid Convergence Index

$$GCI_{fine}^{21} = \frac{(1.25 \times e_a^{21})}{(r^p - 1)}$$

Key Points:

- **1.25:** Safety factor
- GCI_{fine}^{21} : Percentage numerical error (excluding iterative and rounding errors)
- Lower GCI values indicate better mesh convergence

Practical Application

NACA 0012 Example Study

- **Mesh Strategy:** 3 grids with refinement factor $r = 2$
- **Mesh Relationship**

$$N_{fine} = 2 \times N_{medium} ,$$

$$N_{coarse} = 0.5 \times N_{medium}$$

- **Variables Examined:**
 - Pressure coefficient
 - Drag coefficient
 - Lift coefficient
- **Validation:** Comparison with experimental data at different angle of attack

Best Practices

Mesh Independence Study Protocol

1. Conduct simulations with multiple meshes
2. Plot results (Number of cells vs. variable of interest ϕ)
3. Follow Journal of Fluid Engineering procedures for publication-level work
4. Ensure proper convergence criteria are met
5. Document all error estimates and safety factors

Quality Assurance

- Always verify iterative convergence before mesh studies
- Use double precision for calculations

- Maintain consistent boundary conditions across all meshes
- Document refinement ratios and characteristic cell sizes
- Compare final results with experimental data when available

Key References

1. [Celik et al. \(2008\) - Standard procedure for CFD uncertainty estimation](#)
2. [Roache \(1997\) - Quantification of uncertainty in CFD](#)
3. [Eça & Hoekstra \(2014\) - Grid refinement study procedures](#)

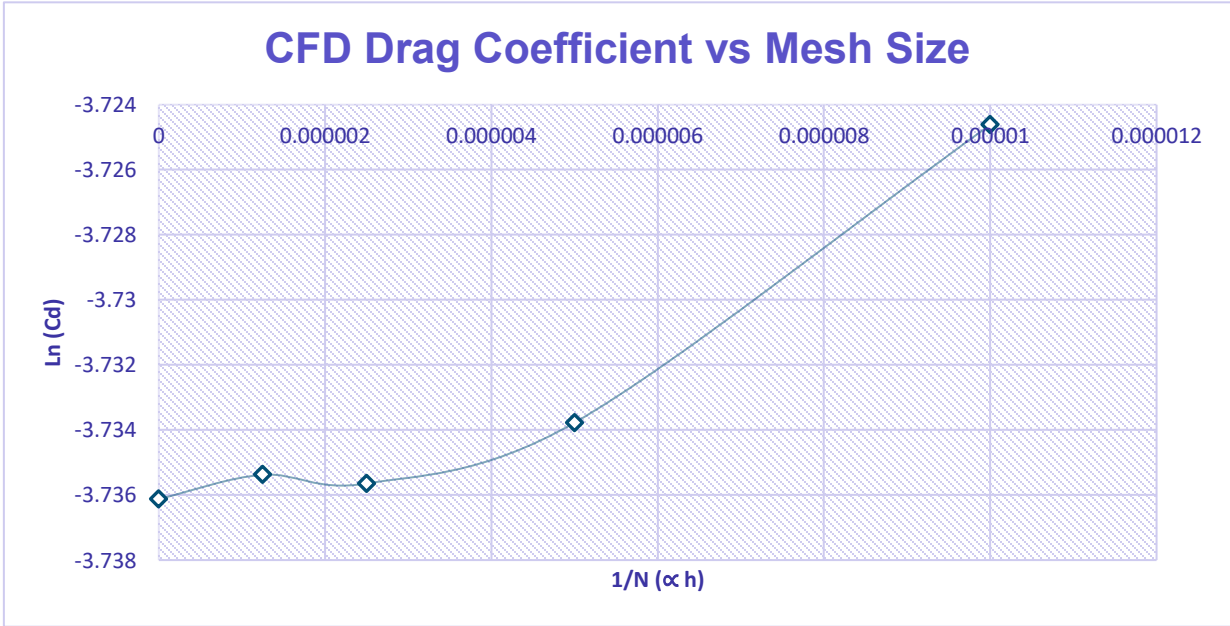
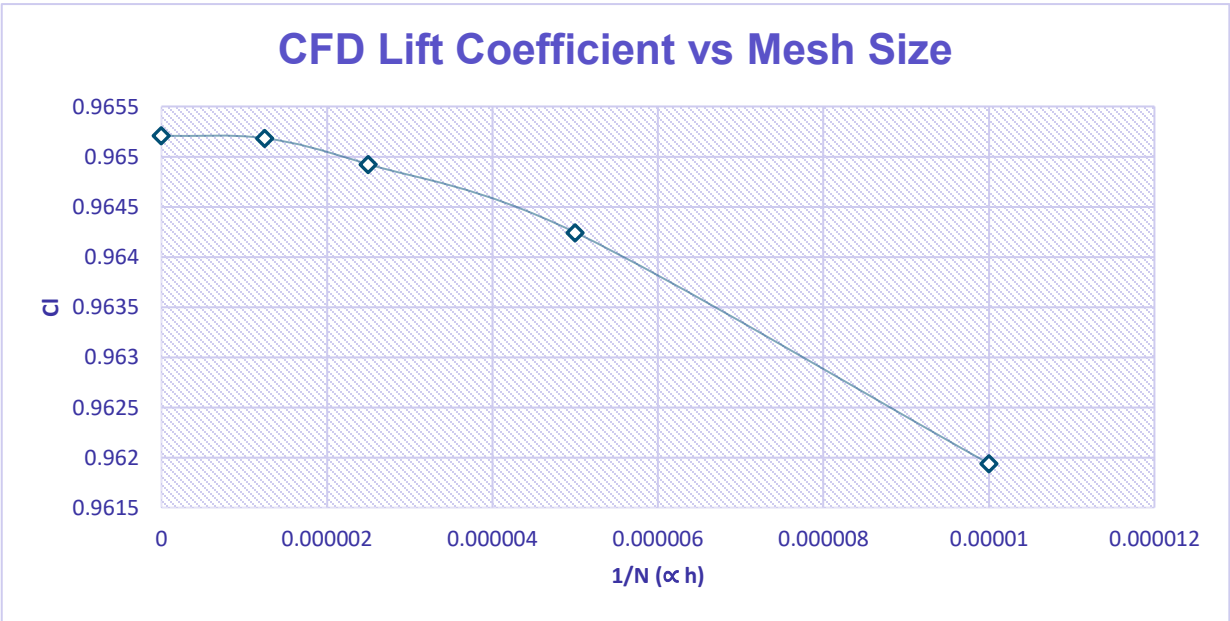
Mesh Refinement Results - NACA 0012 Airfoil at 10° AOA

Refinement Factor (r): 1.414213562

Metric	CL	CD	Cm
Experimental Data (Airfoil Tools)	1.006900	0.024810	-0.015700
ANSYS Coarse (ϕ_3)	0.961938	0.024122	-0.017405
ANSYS Medium (ϕ_2)	0.964244	0.023902	-0.017396
ANSYS Fine (ϕ_1)	0.964924	0.023858	-0.172146
ϵ_{32}	-0.002306	0.000220	-0.000009
ϵ_{21}	-0.000680	0.000045	0.154750
$\epsilon_{21}/\epsilon_{32}$	0.294732	0.202914	-17194.394330
Convergence Behaviour	Monotonic Convergence	Monotonic Convergence	Oscillatory Divergence
Order p	3.525054	4.602124	—
$\phi_{\text{ext } 21}$	0.965208	0.023846	-0.211540
$\phi_{\text{ext } 32}$	0.965208	0.023846	-0.017394
Approx. Rel. Error (e_{a21})	0.000704	0.001870	-0.898946
Extrapolated Rel. Error ($e_{\text{ext}21}$)	0.000294	0.000476	-0.186227
GCI fine 21	0.000368	0.000595	—
GCI fine 21 (%)	0.036787	0.059504	—
% Error to Experiment	-4.140679	-3.883660	/

Mesh Refinement and Convergence Study

Mesh Level	Number of Cells (N)	1/N ($\propto h$)	CL	CD	Cm
Coarse	100,000	0.00001	0.9619	0.02412	-0.01741
Medium	200,000	0.000005	0.9642	0.02390	-0.01740
Fine	400,000	0.0000025	0.9649	0.02386	-0.17215
Very Fine	800,000	0.00000125	0.9652	0.02386	—
Extrapolated (∞)	∞	0	0.9652	0.02385	-0.2115



Key Observations

Successful Convergence

- **Lift Coefficient (CL):** Shows monotonic convergence with $GCI_{fine} = 0.037\%$
- **Drag Coefficient (CD):** Shows monotonic convergence with $GCI_{fine} = 0.060\%$
- Both CL and CD demonstrate excellent mesh independence with very low numerical uncertainty

Problematic Results

- **Moment Coefficient (Cm):** Shows oscillatory divergence behavior
- The moment coefficient results are unreliable and require investigation
- Possible causes: numerical instability, insufficient mesh quality near moment reference point, or solver settings

Validation Against Experimental Data

- **CL Error:** -4.14% (reasonable agreement)
- **CD Error:** -3.88% (good agreement)
- **Cm Error:** 1247% (significant discrepancy - requires investigation)

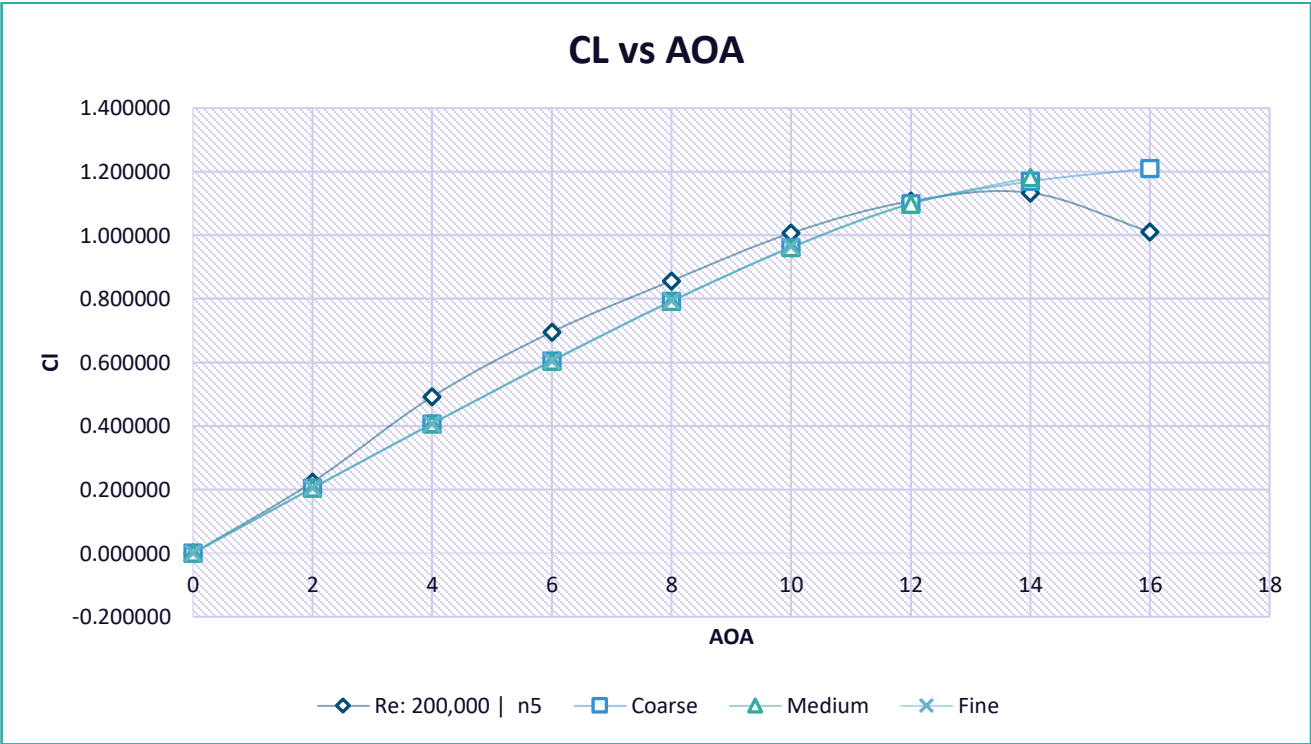
Recommendations

1. **For CL and CD:** Mesh convergence achieved - fine mesh (800,000 cells) is adequate
2. **For Cm:**
 - Investigate mesh quality near airfoil surface
 - Check moment reference point location
 - Consider higher-order discretization schemes
 - Verify solver convergence criteria
3. **Overall:** Results show good mesh independence for primary aerodynamic coefficients

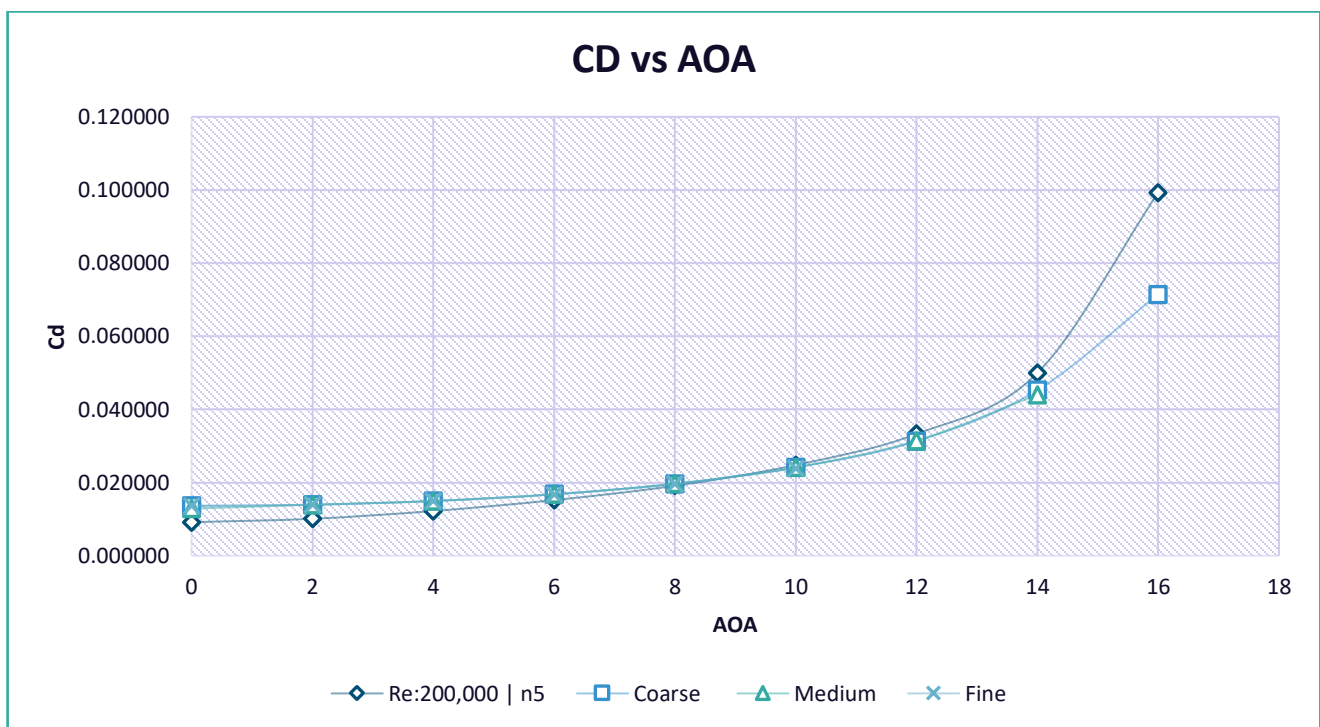
CFD Results

Case 1: NACA 0012, 25 m/s, k-w SST Model

Re: 200,000 μ : 1.7893e-05 kg/m-s			Re: 304,212 μ : 1.812e-05 kg/m-s		
AOA	CL n5	CL n9	CL (Coarse)	CL (Medium)	CL (Fine)
0	0.000000	0.000000	-0.000005	0.000000	0.000037
2	0.224000	0.308500	0.204617	0.205280	0.204503
4	0.492200	0.535700	0.407216	0.407960	0.407192
6	0.695600	0.697100	0.604631	0.605120	0.605022
8	0.856700	0.848100	0.792273	0.792260	0.793403
10	1.006900	1.006700	0.961938	0.961550	0.964924
12	1.107200	1.106800	1.099477	1.098500	--
14	1.133600	0.963200	1.169467	1.181400	--
16	1.010700	--	1.209848	--	--



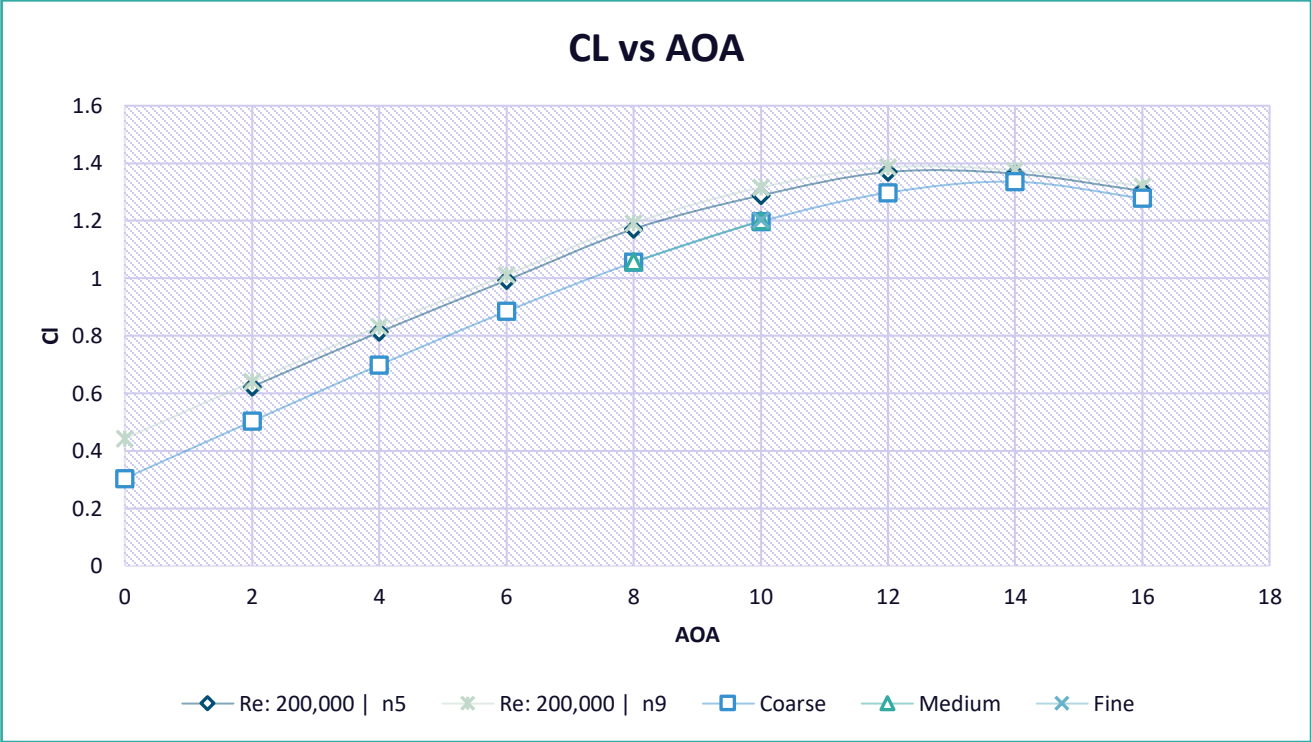
AOA	CD n5	CD n9	CD (Coarse)	CD (Medium)	CD (Fine)
0	0.009180	0.010200	0.013607	0.012870	0.013693
2	0.010110	0.010660	0.013946	0.013950	0.013960
4	0.012230	0.011760	0.014962	0.014978	0.014976
6	0.015220	0.015160	0.016802	0.016818	0.016784
8	0.019130	0.021120	0.019684	0.019714	0.019588
10	0.024810	0.029660	0.024122	0.024192	0.023858
12	0.033290	0.043660	0.031306	0.031458	--
14	0.049950	0.079760	0.045265	0.044055	--
16	0.099290	--	0.07145436	--	--



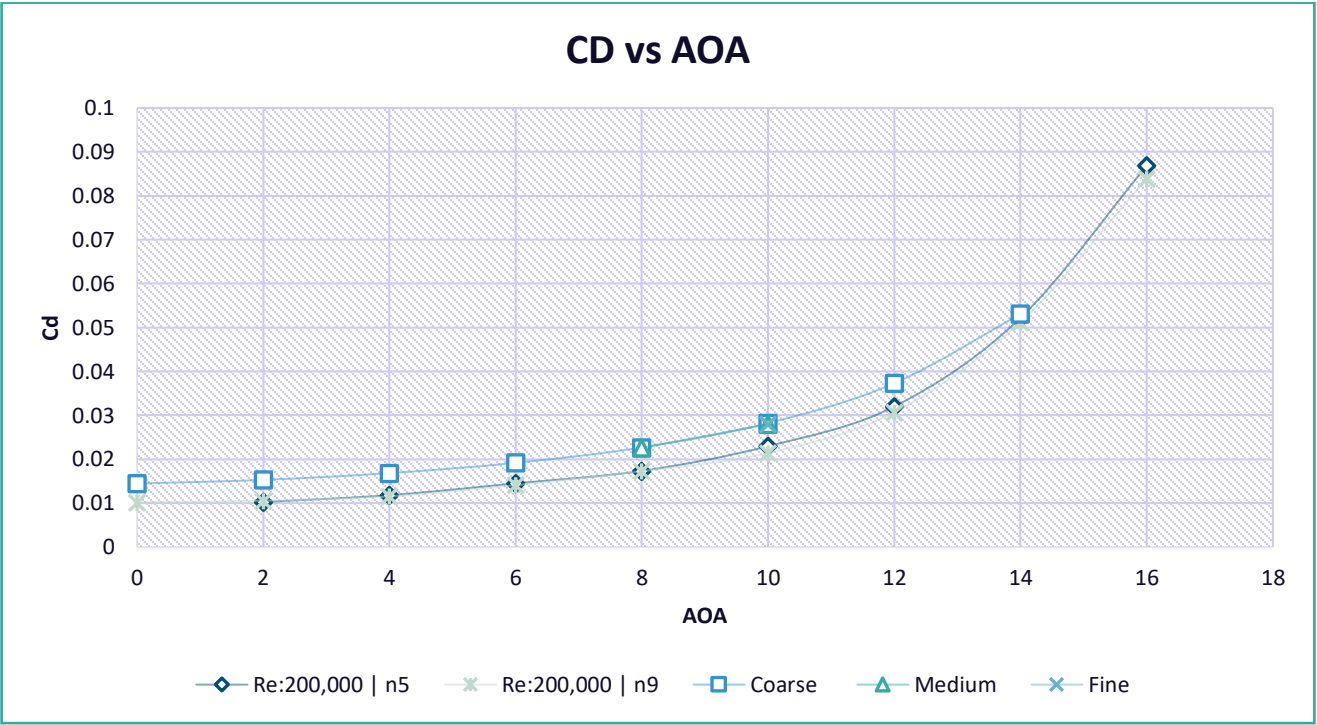
- Note: Check case for clark y airfoil for 25 m/s to verify GCI

Case 2: Clark Y, 25 m/s . k-w SST Model

Re: 200,000 μ : 1.7893e-05 kg/m-s			Re: 304,212 μ : 1.812e-05 kg/m-s		
AOA	CL n5	CL n9	CL (Coarse)	CL (Medium)	CL (Fine)
0	--	0.4424	0.30412		
2	0.623	0.6402	0.50281		
4	0.8129	0.8324	0.69816		
6	0.9928	1.0116	0.88502		
8	1.1714	1.1918	1.05546	1.0571509	1.05809
10	1.2881	1.3158	1.19728	1.2010779	1.2022572
12	1.3695	1.3859	1.29780		
14	1.364	1.3753	1.33578		
16	1.3038	1.3199	1.27800		



AOA	CD n5	CD n9	CD (Coarse)	CD (Medium)	CD (Fine)
0		0.01016	0.01444		
2	0.01026	0.01053	0.01528		
4	0.0118	0.01152	0.01682		
6	0.01448	0.01408	0.01917		
8	0.01733	0.01726	0.02268	0.02261252	0.02189
10	0.02296	0.02138	0.02820	0.027942752	0.027929596
12	0.03199	0.03069	0.03728		
14	0.05219	0.05122	0.05312		
16	0.08685	0.08387			



Observation & Conclusion:

- There were errors in C_l vs α and C_d vs α plot when compared with Experimental data for N_{crit} 5 and N_{crit} 9 (from Airfoil Tools)
- Possible reason could be because of different Reynolds no and dynamic viscosity
- GCI conclude that after 800k elements there is very less change in values

Next Step-

- To run the simulation with same Reynolds no and velocity and dynamic viscosity
- Run different turbulent model to capture data on 16 deg AOA

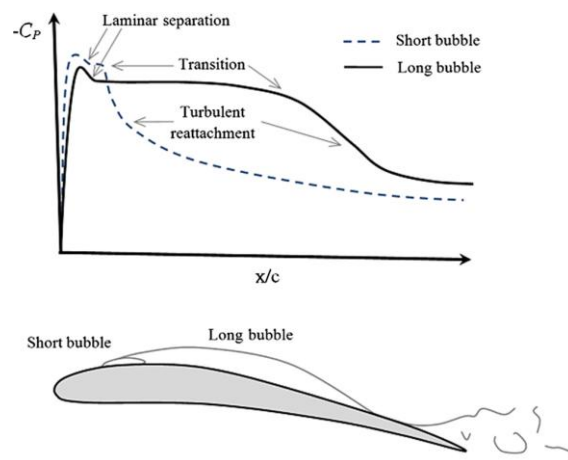
After reviewing the following research papers:

1. *Numerical Study of the Effect of the Reynolds Number and the Turbulence Intensity on the Performance of the NACA 0018 Airfoil at the Low Reynolds Number Regime* – Michna & Rogowski (2022)
2. *A Study of Long Separation Bubble on Thick Airfoils and Its Consequent Effects* – Choudhry et al. (2015)

I have drawn the following insights relevant to CFD simulations of airfoils like NACA 0012 and NACA 0018 operating in the low Reynolds number regime ($Re < 500,000$):

1. Presence and Nature of Laminar Separation Bubbles (LSBs)

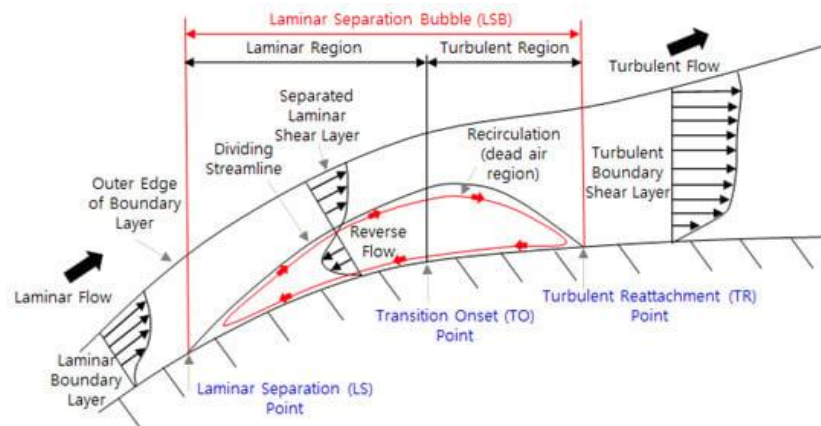
- Both studies highlight laminar separation bubbles (LSBs) as a dominant flow feature at low Re , especially on thick or symmetric airfoils.
- At Re below $\sim 500,000$, the NACA 0018 and 0021 profiles experience early laminar separation near the leading edge, followed by reattachment after transition—creating a closed separation bubble.
- The bubble's location and size are sensitive to both Reynolds number and angle of attack.



2. Effects of Laminar Separation Bubbles on Performance

- As angle of attack (AoA) increases, the LSB moves upstream and grows in size, eventually leading to abrupt stall if transition or reattachment is not stabilized.
- LSBs result in:
 - Increased drag,

- **Distorted pressure distribution,**
- **Reduction in C_l/C_d (lift-to-drag ratio),**
- LSBs introduce an apparent camber on symmetric airfoils, causing a nonlinear increase in lift with angle of attack. (A pseudo “camber” effect on symmetric airfoils due to pressure imbalance across the chord.)



3. Importance of Transition Modeling

- Standard RANS turbulence models (e.g., $k-\omega$ SST) fail to capture LSB dynamics or predict the transition onset accurately at low Re. They are only valid for fully turbulent flows or post-transition conditions.
- Both studies emphasize that transition-capable turbulence models are crucial for accurate low-Re simulations:
 - **$\gamma-Re\theta$ (Transition SST) model** showed good agreement with experimental lift/drag trends.
 - $k-kL-\omega$ model gave better prediction of bubble reattachment location and more accurate lift values.
- Transition models are especially essential in capturing separation-induced transition, the dominant mechanism at low Re.

4. Grid Independence and Mesh Size

- Choudhry et al. demonstrate that a mesh with $\sim 900,000$ cells (9 lakh) ensure mesh-independent results for thick airfoils at low Re.
- This resolution:
 - Accurately captures the near-wall boundary layer,
 - Resolves the shear layer within the bubble,
 - Allows reliable prediction of C_l , C_d , C_p , and reattachment.
- Boundary size of $15c-20c$ from the airfoil is adequate; farfield location beyond that has minimal additional effect.

5. Reynolds Number Impact

- The aerodynamic performance of airfoils like NACA 0018 is highly sensitive to Reynolds number, especially in the low Re regime.

- As shown in Michna & Rogowski (2022), airfoil characteristics change drastically between $Re = 50,000$ and $200,000$:
 - At $Re = 50,000$, both lift and drag curves exhibit non-linear behavior and cannot be modeled using traditional aerodynamic derivatives.
 - At higher Reynolds numbers, the flow separates later and reattaches sooner, leading to:
 - A smaller or shorter separation bubble,
 - Stabilized flow over the suction surface,
 - Improved aerodynamic metrics like lift coefficient (C_l) and C_l/C_d ratio.

6. *Turbulence Intensity (TI) Effects*

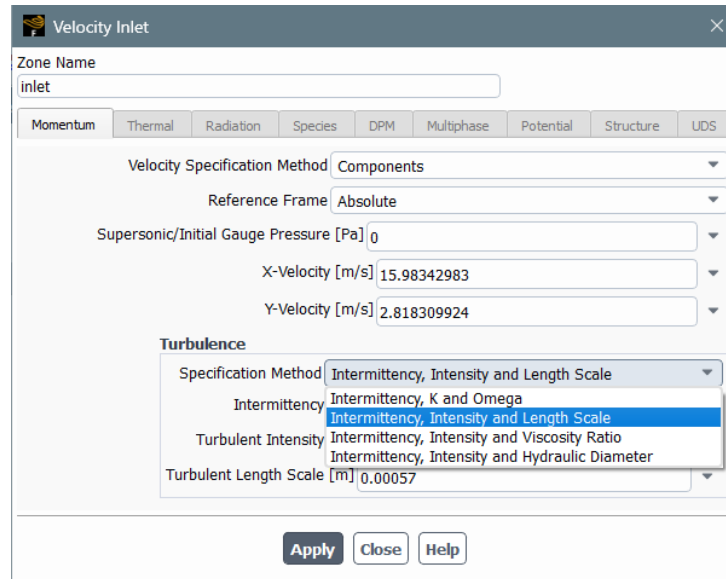
- Turbulence intensity influences the onset of transition and reattachment point, thereby affecting bubble behavior:
- **Higher TI (e.g., 0.5%)** promotes earlier transition and faster reattachment, resulting in a shorter separation bubble.
- However, the location of laminar separation is not significantly altered by TI at high angles of attack.
- For NACA 0018, simulations across a **TI range of 0.01% to 0.5%** showed:
 - Only mild variations in lift and drag coefficients,
 - Reynolds number remains the dominant factor in determining flow characteristics and performance.

7. *Influence of Angle of Attack on Bubble Behavior*

- The onset, growth, and disappearance of the LSB are tightly coupled with AoA:
 - At low AoA, a small bubble may exist or be absent.
 - As **AoA increases**, the bubble moves upstream, grows longer, and may burst, triggering stall.
 - Beyond transition AoA, the bubble may vanish due to full boundary layer transition.

Turbulence Property Calculations

(Using Mixing Length Model — ANSYS Fluent):



Turbulence length scale estimation

For external aerodynamics around airfoils, especially when using **Transition SST**, a common practice is to set the turbulence length scale (l) as a small fraction of a characteristic length — typically the chord length.

According to the reference guide:

- For external wall-bounded flows, the length scale should be approximately:
$$l = 0.01 \times \text{chord length (typical for low – turbulence cases)}$$

Some sources also suggest:

- 4% of boundary layer thickness, if known
- 1% to 5% of chord length, if boundary layer thickness isn't available

In our case:

$$l = 0.01 \times 0.18 = 0.0018 \text{ m} = 1.8 \text{ mm}$$

This is a conservative, physically reasonable value for capturing transition behavior with a low turbulence intensity inlet.

→ Turbulence Properties

① Turbulent Intensity: Also be known as turbulence level

$$\boxed{I = \frac{u'}{U}} = \frac{\text{root mean sq. of the Turbulent velocity fluctuations}}{\text{Mean Velocity (Reynolds Averaged)}}$$

If turbulent energy, k is known,

u' can be computed as →

$$u' = \sqrt{\frac{1}{3} (u_x'^2 + u_y'^2 + u_z'^2)}$$

$$\boxed{u' = \sqrt{\frac{2}{3} k}}$$

$$\boxed{U = \sqrt{u_x^2 + u_y^2 + u_z^2}}$$

Averaging a variable or an eqn in time.

$$\bar{\phi} = \frac{1}{T} \int_T \phi(t) dt$$

$$\phi' = \phi - \bar{\phi}$$

↓ ↓
Fluctuations Mean/Avg.

Turbulence Level

High → 5% < T.I. < 20%

Med → 1% < T.I. < 5%

Low → T.I. < 1%

- Fully developed Pipe flow → $I = 0.16 Re_{dn}^{-1/8}$
- Smooth pipe axis → $0.055 Re^{-0.0407}$

→ Turbulent Kinetic Energy: Avg. K.E per unit mass associated with eddies in a turbulent flows

$$k = \frac{1}{2} (\underbrace{\bar{u}'^2 + \bar{v}'^2 + \bar{w}'^2}_{\text{Turbulent vel. fluctuations}})$$

- In CFD simulations → $k = \frac{3}{2} (IU)^2$ $U \rightarrow$ Initial Velocity
 $I \rightarrow$ Turbulence Intensity

- Using Mixing Length Model → $k = \left[\frac{M_t}{M} \times \frac{\mu}{P \mu^{1/4} Tu-L} \right]^2$
(Length scale & eddy viscosity ratio)

$C_u = 0.09$, $\rho = \text{density}$, $\mu = \text{molecular dyn. viscosity}$
 $\frac{\mu_t}{\mu} = \text{viscosity ratio}$, $Tu-L \rightarrow \text{Tur. length scale}$

→ Turbulent Dissipation Rate (ϵ)

↳ rate at which Turbulent K.E is converted into thermal energy due to viscous effects.

Classical Def $\rightarrow \epsilon = C_u \frac{K^{3/2}}{T_L}$ } Using length scale

Mixing Model $\rightarrow \epsilon = C_u^{3/4} \frac{K^{3/2}}{T_L}$

$\epsilon = C_u \frac{\rho K^2}{\mu} \left[\frac{\mu_t}{\mu} \right]^{-1}$ } Using eddy vis. ratio

→ specific Turbulence Dissipation rate (ω)

Turbulence frequency (ω):

↳ rate at which Turbulent K.E is dissipated into thermal internal energy per unit vol. & time.

$$\omega = \frac{\epsilon}{C_u \cdot K}$$

Classical $\rightarrow \omega = \frac{\sqrt{K}}{T_L}$ } Using $Tu-L$

Mixing Model $\rightarrow \omega = C_u^{-1/4} \frac{\sqrt{K}}{T_L}$

$\omega = \frac{\rho K}{\mu} \left[\frac{\mu_t}{\mu} \right]^{-1}$ } using ϵ

→ Eddy Viscosity Ratio (or) Viscosity Ratio

$\frac{\mu_t}{\mu} = \frac{\text{Turbulent viscosity}}{\text{mol. dyn. viscosity}}$ } Indicates the dominance of turbulent effect

$$\mu_t = \rho C_u \frac{K^2}{\epsilon} = \rho \frac{K}{\omega} = \frac{\mu_t}{\mu} \times \frac{\mu}{\rho}$$

↳
$$\frac{\mu_t}{\mu} = \rho C_u \frac{K^2}{\mu_t} = \rho \frac{K}{\mu \omega}$$

Turbulent Length Scale :

$Tu-L \rightarrow$ A physical quantity describing the size of the large energy-containing eddies in a turbulent flow.

- Often used to estimate turbulent properties of the inlets of a CFD simulation.
- The turbulent Length Scale should not be larger than the dimension of the problem
- Low $Tu-L \rightarrow$ quickly dissipate all turbulent energy and gives a reduced turbulence intensity.

1) Classical definition (k-ε model)

\rightarrow Book on "Turbulence Modelling" by Wilcox

$$TuL_{\text{Classical}} = C_{\mu} \frac{K^{3/2}}{\epsilon}$$

2) Mixing - Length Based (Used in Ansys / OpenFoam)

$$TuL_{\text{mixing}} = C_{\mu}^{3/4} \frac{K^{3/2}}{\epsilon}$$

3) Purely based on dimensional argument. (Used in CFX)

$$TuL_{\text{dim}} = \frac{K^{3/2}}{\epsilon}$$

$$L_{\text{dim}} \approx 6 \times L_{\text{mix}} \approx L_{\text{classical}}$$

$C_{\mu} \rightarrow$ Modal Constant (0.09)

Estimation \rightarrow

It is common to set the $Tu-L$ scale to a certain percentage of a typical dimension of the problem.

• Fully developed pipe flow :

$$l = 0.038 d_h$$

hydraulic diameter

• Wall-bounded Inlet flows :

$$l = 0.22 \times \delta_{a-l}$$

Inlet Boundary Layer Thickness

\downarrow In Mixing - Length (Fluent)

When Inlet flow is bounded by walls with turbulent boundary layer

$$l = 0.4 \times \delta_{B-L}$$

For turbulence model based on the mixing length approach

$$Tu-L = 0.07 \times D_h$$

From Research Paper
(They used Ansys fluent documentation)

$$Tu-L = 0.4 \times \delta_{99}$$

$$\delta_{99} \approx \frac{5.0 x}{\sqrt{Re_x}} \quad , \quad \delta_{99} \approx \frac{0.37 x}{Re^{1/5}}$$

$x \rightarrow$ distance from leading edge ($x = 0.3c$)

$$Re_x \rightarrow \frac{U_{\infty} x}{\nu} = \frac{\rho U_{\infty} L}{\mu}$$

$$\begin{aligned} x &= 0.3 \times 180 \text{ mm} \\ &= 54 \text{ mm} \\ &\approx 0.054 \text{ m} \end{aligned}$$

$$Re = \frac{16.23 \times 0.054}{\left(\frac{1.789 \times 10^{-5}}{1.225} \right)}$$

$$Re = 0.6 \times 10^5$$

$$\begin{aligned} \delta_{99} \text{ Laminar} &= \frac{5.0 \times 0.054}{\sqrt{0.6 \times 10^5}} \\ &= 0.00110227 \text{ m} \end{aligned}$$

$$\begin{aligned} \delta_{99} &\approx 0.001102 \text{ m} \\ \text{Laminar} &= 1.102 \text{ mm} \end{aligned}$$

$$\begin{aligned} \delta_{99} \text{ Turbulent} &= \frac{0.37 \times 0.054}{(0.6 \times 10^5)^{1/5}} \\ &= 0.0022129 \text{ m} \end{aligned}$$

$$\begin{aligned} \delta_{99} &\approx 0.0022 \text{ m} \\ \text{Turbulent} &= 2.21 \text{ mm} \end{aligned}$$

$$\begin{aligned} Tu-L \text{ Laminar} &= 0.4 \times \delta_{99} \text{ Laminar} \\ &= 0.0004409 \text{ m} \\ &\Rightarrow 0.4409 \text{ mm} \end{aligned}$$

$$\begin{aligned} Tu-L \text{ Turbulent} &= 0.4 \times 0.0022 \\ &= 0.0008852 \text{ m} \\ &\Rightarrow 0.885 \text{ mm} \end{aligned}$$

If 0.377 is taken

$$Tu-L_{Tur} = 0.829 \text{ mm}$$



$$U \uparrow \rightarrow Re \uparrow$$

Flow Type : External Airfoil Flow
(Low turbulence Inflow / clean freestream)

Turbulence Intensity (I) : 0.1 - 1%
(Based on wind tunnel / external flow)

Turbulent Length Scale (Tu_L) : $1 \rangle = 3.8 - 7\% \text{ of } d_h \Rightarrow \frac{7 \times 180}{100} = 12.6 \text{ mm}$

$2 \rangle = 1\% \text{ to } 2\% \text{ of chord} \Rightarrow 1.8 \leftrightarrow 3.6 \text{ mm}$

External airflow
Wall-bounded flow



$3 \rangle = 0.4 \times \delta_{99}$ 200 mm

Laminar = 1.102 mm
Turbulent = 2.21 mm

$= 0.4409 \text{ mm}$ $= 0.885 \text{ mm}$

$= 0.00044$ $= 0.000885$

Conclusion \rightarrow

1) We are modelling Transition, need small Tu_L to allow LSB to naturally occur.

2) Too large ($> 5\% \text{ c}$) \rightarrow suppress transition

3) Too small ($< 0.5 \text{ mm}$) \rightarrow cause premature decay of turbulence or numerical dissipation

N_{crit} \rightarrow

1) Critical amplification factor used in XFOIL's

e^N transition Model (Research Paper)

2) It represents the Log. amp. Level (N) of
Tollmien - Schlichting waves at which Laminar to
turbulent transition occurs.

3) High N_{crit} \Rightarrow transition occurs later \rightarrow Clean & quiet flow
4) Low N_{crit} \Rightarrow " " earlier \rightarrow Higher Turbulent

Mark's empirical relation \rightarrow (NASA)

$$N_{crit} = -8.43 - 2.4 \ln \left(\frac{T_u(\%)}{100} \right)$$

$$\Rightarrow T.I. \% = \left[\exp \left(\frac{N_{crit} + 8.43}{-2.4} \right) \right] \times 100$$

$$\frac{N_{crit}}{5} = \frac{T.I.}{0.371 \%}$$

$$\boxed{9 = 0.07 \%}$$

$$< 1 \%$$



Also in one paper

XFOIL \rightarrow

$$T.I. = 0.03 \times 10^{\frac{-N_{crit.}}{20}}$$

Property	Formula (Mixing Length Model)	Value	Units
Turbulence Intensity I	<i>Given</i>	1	%
Turbulence Length Scale L	<i>Calculated</i>	0.0018	m
Kinematic Viscosity ν	$\nu = \mu / \rho$	1.51×10^{-5}	m^2/s
Turbulent Kinetic Energy k	$k = 32(IU)^2$	0.0395	m^2/s^2
Dissipation Rate ϵ	$\epsilon = C_\mu^{3/4} \cdot k^{3/2} / L$	0.716	m^2/s^3
Specific Diss. Rate ω	$\omega = C_\mu^{-1/4} \cdot k / L$	182.0	s^{-1}
Eddy Viscosity Ratio μ_t	$k \omega \cdot \rho \mu$	14.4	—

Ref:

1. [turbulence-properties-calculator](#)
2. [CFD Online Turbulent length Scale](#)

Case 3: NACA 0012 with same Reynolds No., dynamic viscosity, Turbulent Intensity

1. Model Characteristics and Limitations

- Model Used: *Steady-state, uncalibrated* γ -Re_θ (Transition SST) model in ANSYS Fluent.
- Limitation: No specific calibration was applied for low Reynolds numbers. As a result, accuracy is **acceptable but not optimal**.

2. Relative Error with Experimental Data (Airfoil Data Ncrit =9):

- Mean Lift Coefficient (CL) Error: **3.52%** (mean over AoA 0°–12°)
- Drag Coefficient (CD) Error: **2.76%** (mean over AoA 0°–12°)

3. Improvement Potential:

- **Fine-tuning transition onset criteria** (e.g., intermittency, turbulence intensity, length scale).
- Performing **unsteady simulations (URANS)** instead of steady-state for better capture of separation-induced transition dynamics.
- The 3D effects, which were not considered in the CFD analysis, are probably responsible for the underestimation of the results.

4. Grid Independence Study:

- Beyond 800,000 elements, only 0.01% variation observed in results.
- Final simulations performed on 900,000 cells to ensure accuracy.

5. Inlet Turbulence Intensity (TI) and Turbulence Length Scale (TuL):

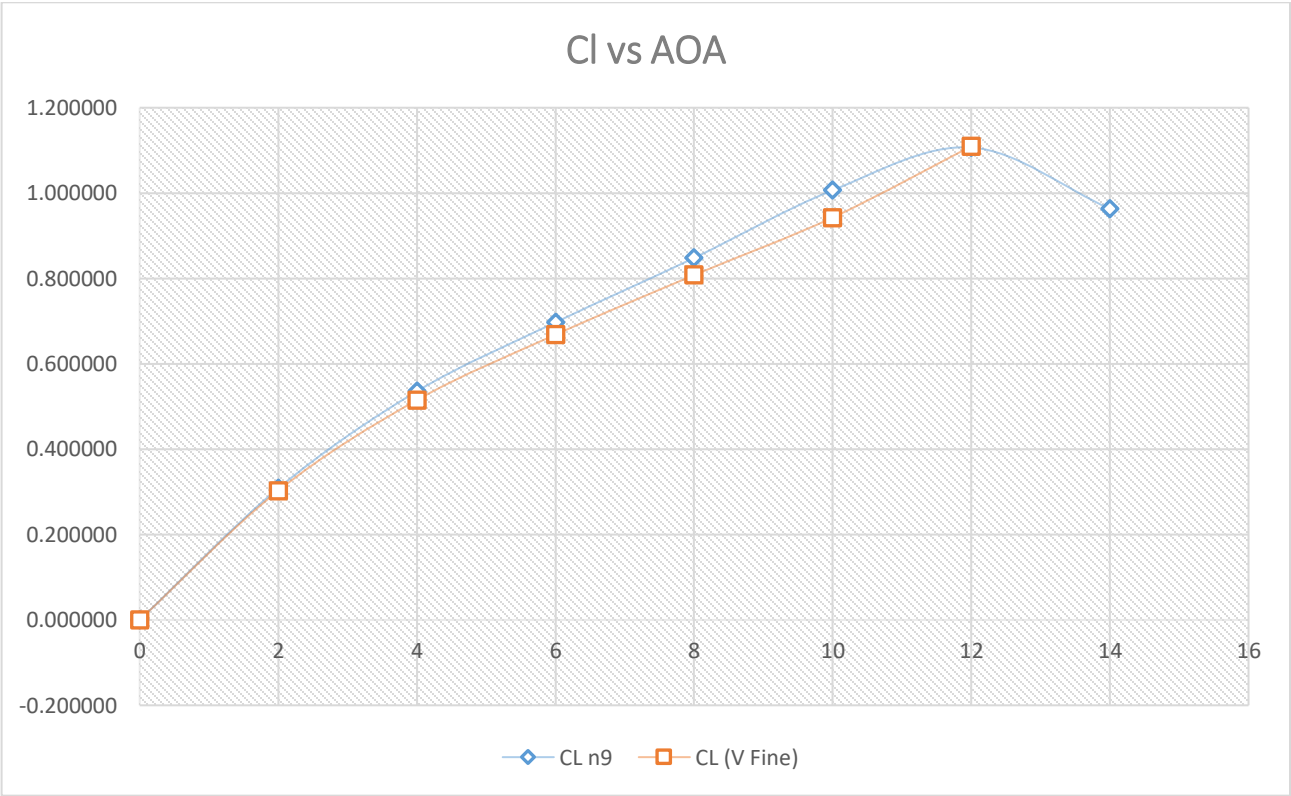
- TI Variations had negligible influence on results within the tested range (0.01%–0.1%).
- Selected based on wall-bounded flow recommendation from the ANSYS Manual.
- Turbulence Length Scale (TuL) variation was negligible if <2% of chord length — used value: 0.0018 m and 0.000885 m.

Future Work

- The next phase of the investigation will extend the study to **higher Reynolds numbers** in the range of 3×10^5 to 5×10^5 , corresponding to **freestream velocities of 20–25 m/s**. Emphasis will be placed on:
 - Validation of aerodynamics coefficients at higher Re no.
 - Detailed study of the **location and behaviour of the laminar separation bubble**.
 - Assessing the effect of unsteady flow phenomena on airfoil performance

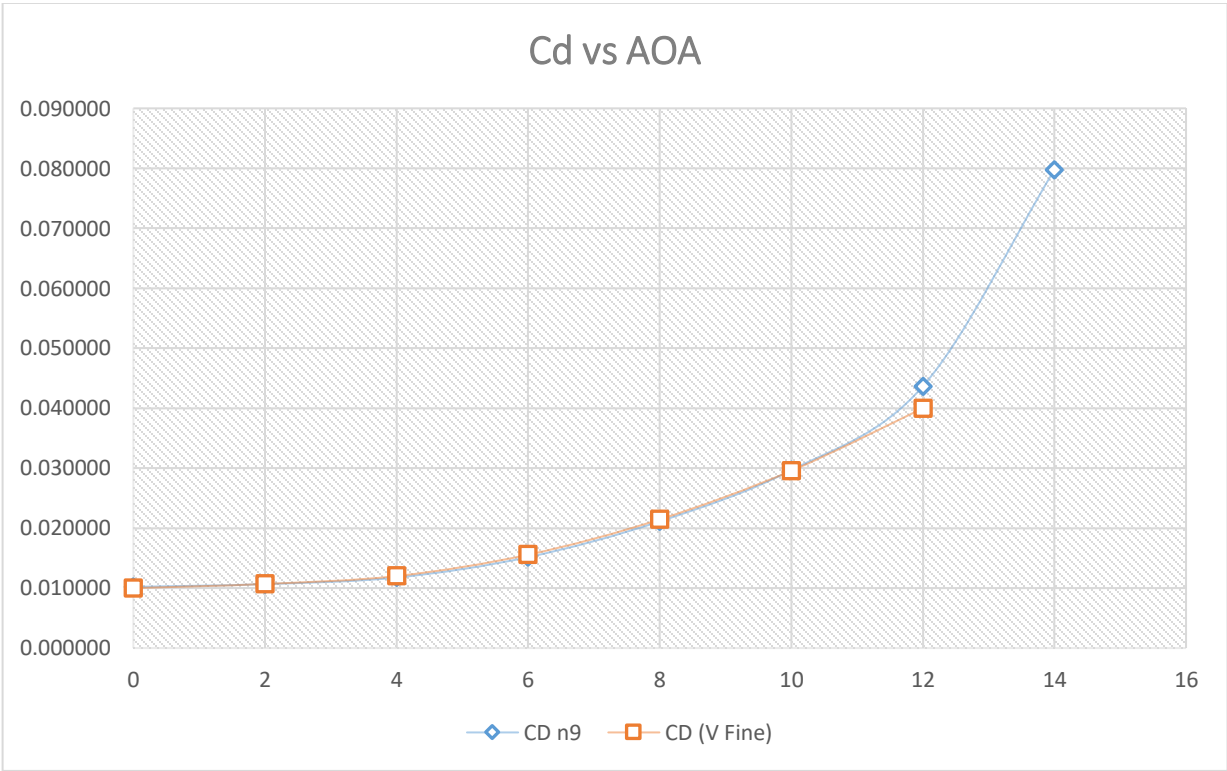
CL variation with angle of attack:

AOA	CL n9	CL (V Fine)	no of iterations	Relative error
0	0.000000	0.0000000	214	-
2	0.308500	0.30297371	317	1.79%
4	0.535700	0.51520034	381	3.83%
6	0.697100	0.66833238	1101	4.13%
8	0.848100	0.80789904	813	4.74%
10	1.006700	0.94274497	887	6.35%
12	1.106800	1.110000	1833	-0.29%
14	0.963200	--	--	--

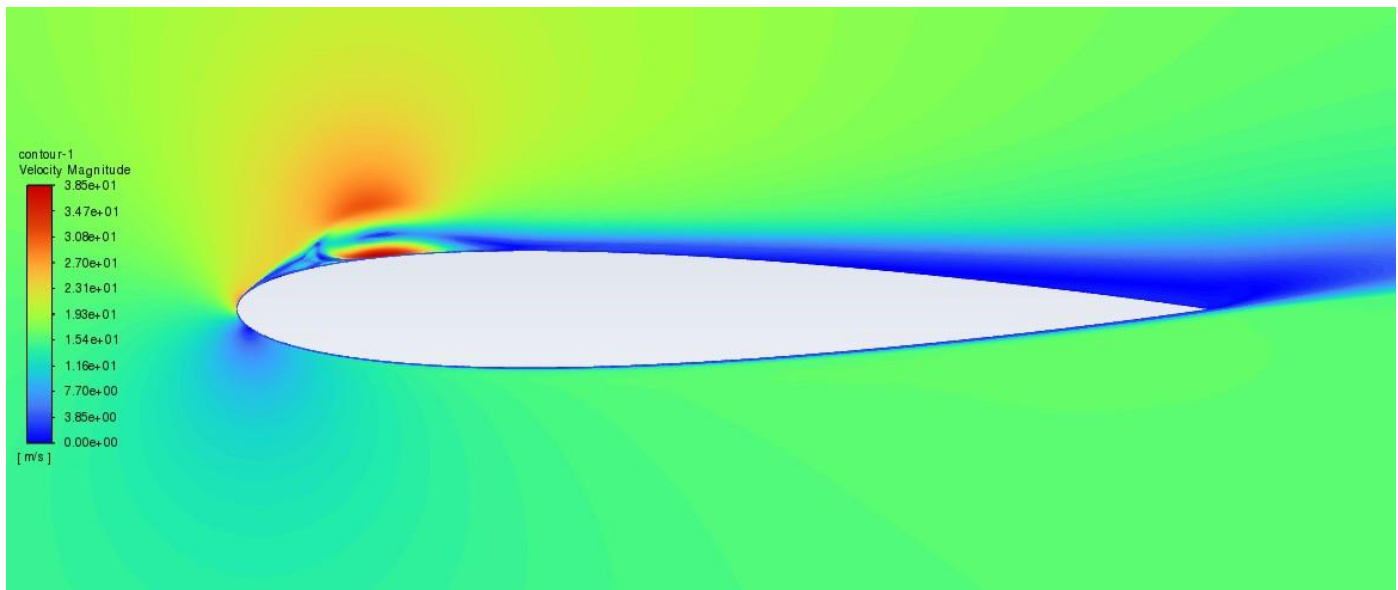
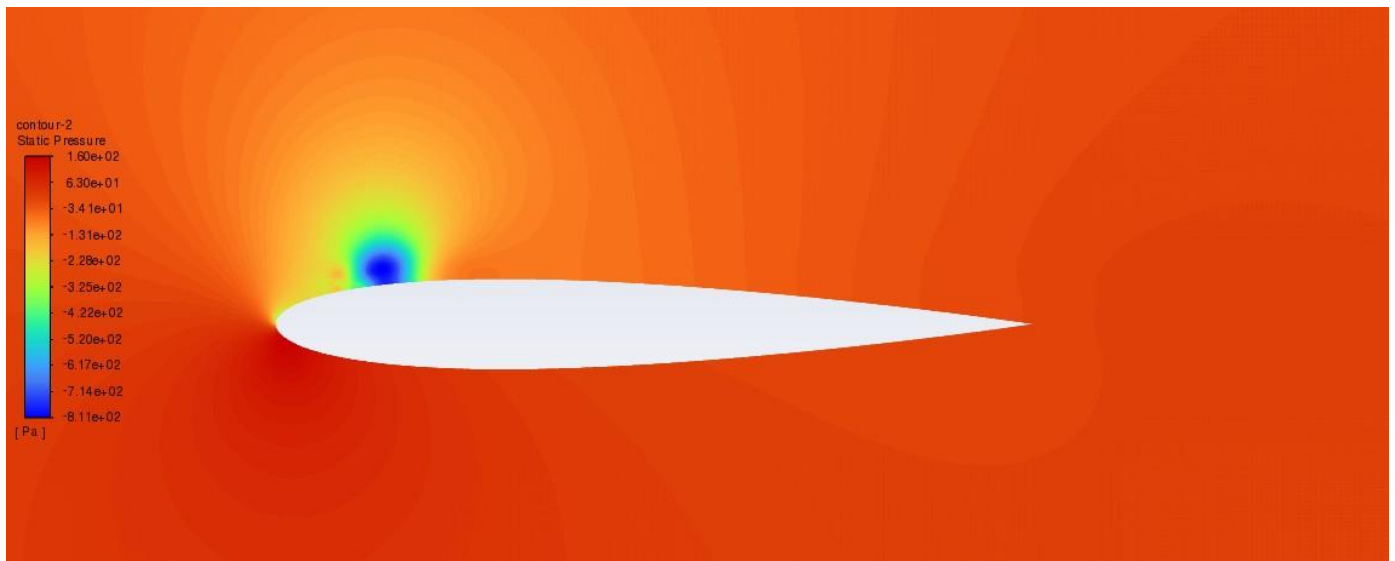


CD variation with angle of attack:

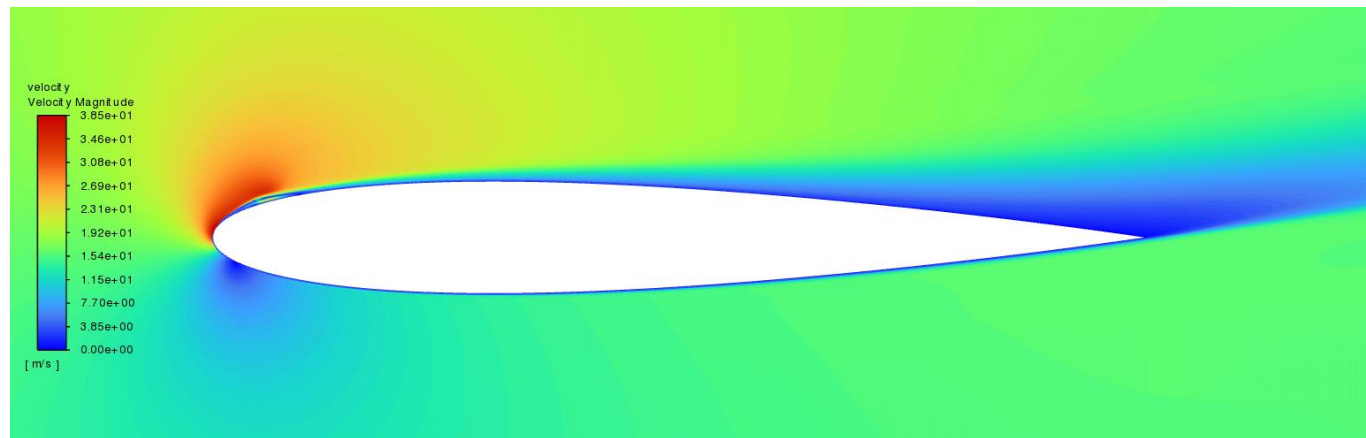
AOA	CD n9	CD (V Fine)	no of iterations	Relative error
0	0.010200	0.01000000	214	1.96%
2	0.010660	0.010690108	317	-0.28%
4	0.011760	0.012007455	381	-2.10%
6	0.015160	0.015569951	1101	-2.70%
8	0.021120	0.0214501	813	-1.56%
10	0.029660	0.029563079	887	0.33%
12	0.043660	0.040000	1833	8.38%
14	0.079760	--	--	--



Laminar Separation Bubble at 2 deg of Angle of attack, 16.23 m/s velocity



Laminar Separation Bubble at 12 ` of Angle of attack, 16.23 m/s velocity



→ FUTURE WORK

1) $C = 225 \text{ mm}$

$V_{\infty} = 20 \longleftrightarrow 25 \text{ m/s}$

$\text{A.O.A} = 0 - 10 \text{ deg}$

for same P, μ, T and height

check effect of Re variation on

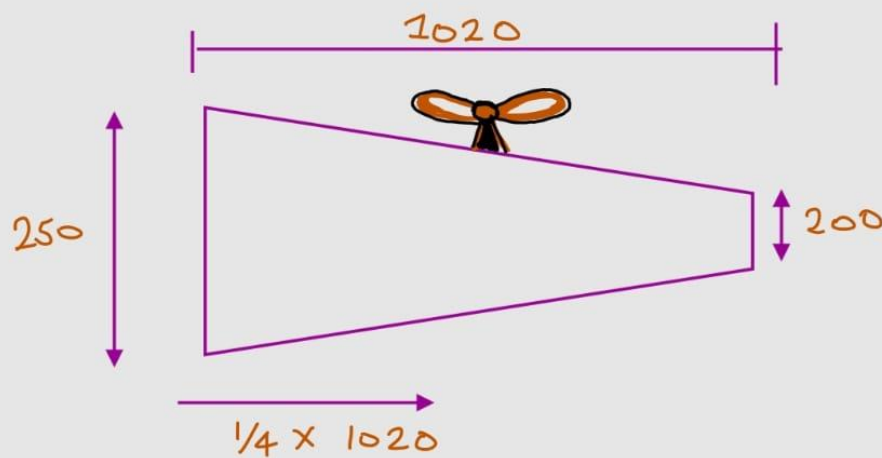
C_L & C_D .

2) Check if the effect of propeller is increasing or decreasing our lift on the airfoil

$V_{\infty} = 14 \text{ m/s}$

↓

16 m/s



Appendix

Appendix A: Model Constants

A1: Model Constants Table

Parameter	Value	Description / Purpose
Alpha*_inf	1	Controls blending in specific dissipation rate (ω) equation in the SST model.
Alpha_inf	0.52	Controls the turbulent viscosity formulation (SST model).
Alpha_0	0.11111	Near-wall value for turbulent viscosity function (low-Re correction).
Beta*_inf	0.09	Controls the dissipation of turbulent kinetic energy (k) in the SST model.
R_beta	8	Stabilization constant for blending functions.
R_k	6	Stabilization constant in the k-equation.
R_w	2.95	Stabilization constant in the ω -equation.
a1	0.31	Coefficient used in turbulent eddy viscosity limit formulation.

A2: Transition Model Constants (Gamma-Theta model)

Parameter	Value	Description / Purpose
CTU1	100	Constant for intermittency production term based on local turbulence.
CTU2	1000	Scales the onset criteria for transition; used in the γ -equation source term.
CTU3	1	Weighting factor in intermittency transport equation.
CPG1	1	Coefficient related to pressure gradient influence on transition.
CPG2	-0.5	Negative value to account for adverse pressure gradient effects.
CPG3	0	Usually related to crossflow or bypass transition effects.
CSB	1	Blending coefficient for separation bubble transition modeling.

A3: Turbulent Prandtl Numbers

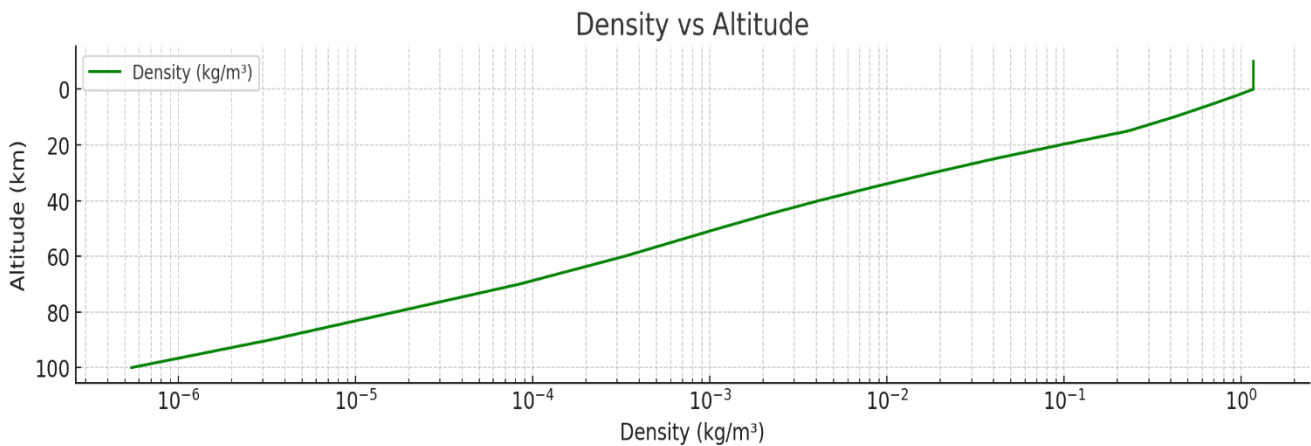
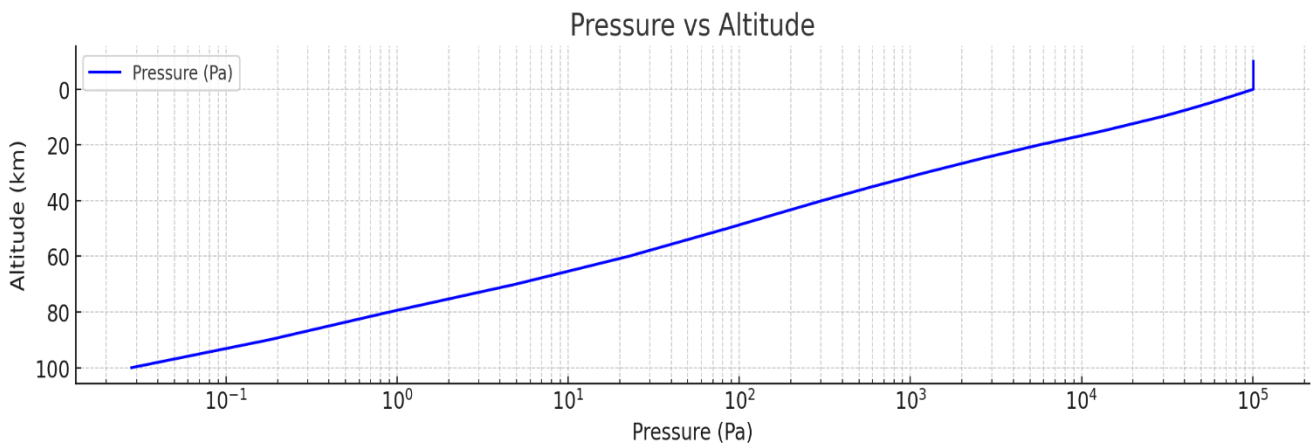
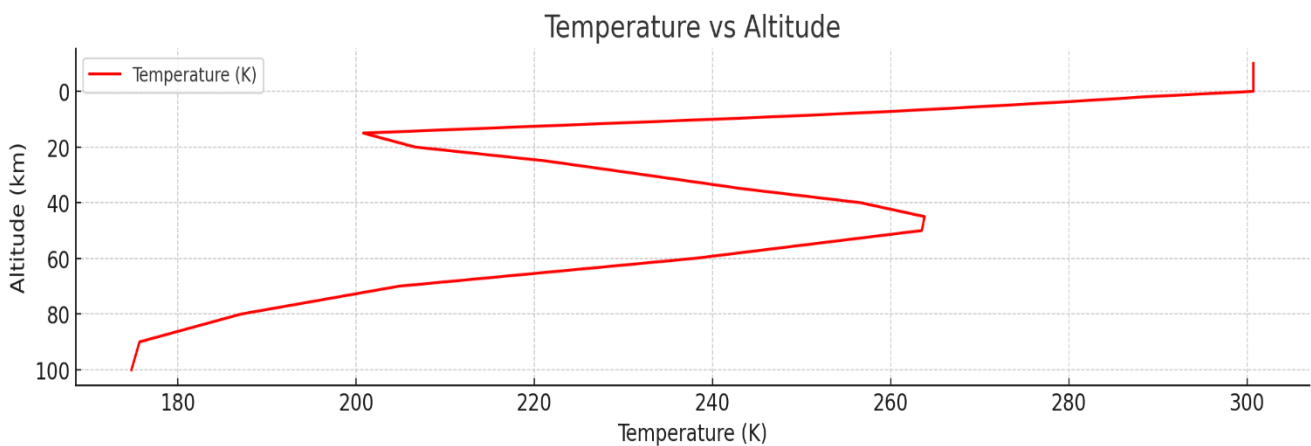
Parameter	Value	Description / Purpose
-----------	-------	-----------------------

TKE Prandtl # (Inner)	1.176	Effective Prandtl number for turbulent kinetic energy in the near-wall region.
TKE Prandtl # (Outer)	1	Effective Prandtl number for turbulent kinetic energy in the far-field.
SDR Prandtl # (Inner)	2	Specific Dissipation Rate Prandtl number (near wall).
SDR Prandtl # (Outer)	1.168	Specific Dissipation Rate Prandtl number (outer region).

A4: Other Coefficients

Parameter	Value	Description / Purpose
Beta_i (Inner)	0.075	Controls production/destruction of ω in near-wall region.
Beta_i (Outer)	0.0828	Controls production/destruction of ω in free-stream region.

Appendix B: Indian Standard Atmospheric Data



Atmospheric Properties (0–30 km Altitude)

Altitude (km)	Temperature (K)	Pressure (Pa)	Density (kg/m³)
0	300.7	100900	1.169
1	294.6	89980	1.064
2	288.4	80060	0.9672
3	283.6	71070	0.873
4	278.4	62970	0.7881
5	272.9	55650	0.7106
6	267.2	49060	0.6397
7	261.4	43140	0.5749
8	254.9	37820	0.5169
9	247.9	33030	0.4643
10	240.4	28740	0.4166
11	232.4	24890	0.3732
12	224.2	21450	0.3334
13	215.9	18390	0.2967
14	207.7	15660	0.2627
15	200.8	13270	0.2302
16	195.6	11180	0.1991
17	194.2	9395	0.1686
18	197.3	7901	0.1395
19	201.7	6667	0.1152
20	206.6	5647	0.09526
21	209.1	4798	0.07994
22	212.2	4086	0.06709
23	215.5	3488	0.0564
24	218.8	2985	0.04753
25	221.3	2560	0.0403
26	223.8	2199	0.03423
27	226.0	1892	0.02915
28	228.0	1631	0.02491
29	230.3	1407	0.02129
30	232.4	1216	0.01824

