

# Simulation of Turbulent Flow around an Airfoil

ENGR:2510 Mechanics of Fluids and Transfer Processes

CFD Pre-Lab 2

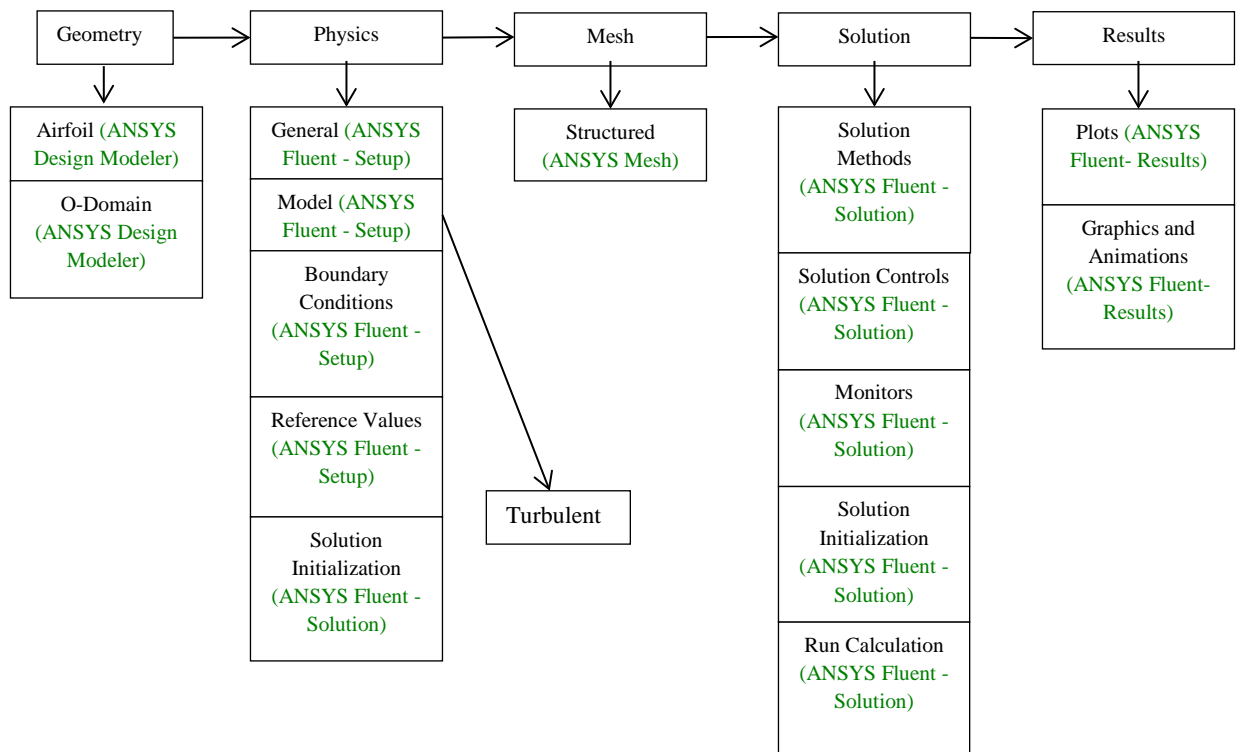
(ANSYS 17.1; Last Updated: Nov. 7, 2016)

By Timur Dogan, Michael Conger, Andrew Opyd, Dong-Hwan Kim  
Maysam Mousaviraad, Tao Xing and Fred Stern

IHR-Hydroscience & Engineering  
The University of Iowa  
C. Maxwell Stanley Hydraulics Laboratory  
Iowa City, IA 52242-1585

## 1. Purpose

The Purpose of CFD PreLab 2 is to simulate **turbulent** flow around a Clark-Y airfoil following the “CFD process” with an interactive step-by-step approach. Students will have “hands-on” experiences using ANSYS to compute pressure, lift and drag coefficients using both **viscous and inviscid** models. Students will **validate** simulation results with EFD data measured at EFD Lab 3, analyze the differences and possible numerical errors, and present results in Lab report.



Flow chart for “CFD Process” for airfoil flow

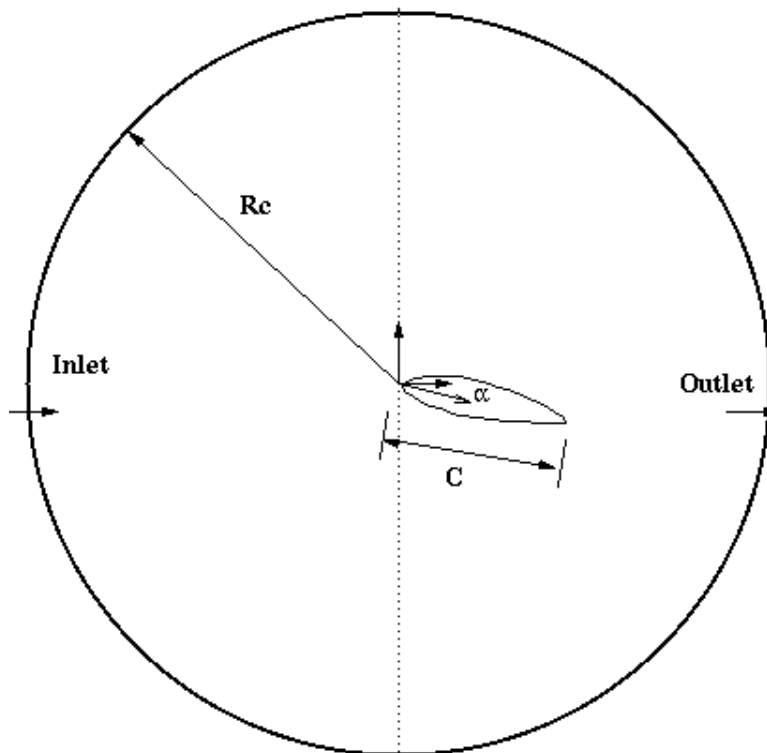
## 2. Simulation Design

In EFD Lab 3, you have conducted experimental study for turbulent flow around a Clark-Y airfoil ( $Re=300,000$ ) for two angles of attack 0 and 16 degrees. The pressure on the foil surface you have measured will be used for CFD PreLab 2. In CFD PreLab 2, simulation will be conducted under the same conditions of EFD Lab 3 (geometry, Reynolds number, fluid properties) at angle of attack 0 degree using both viscous and inviscid models. Simulation results will be validated by your own EFD data.

The problem to be solved is turbulent flow around the Clark-Y airfoil with angle of attack ( $\alpha$ )

**Table 1 - Geometry dimensions**

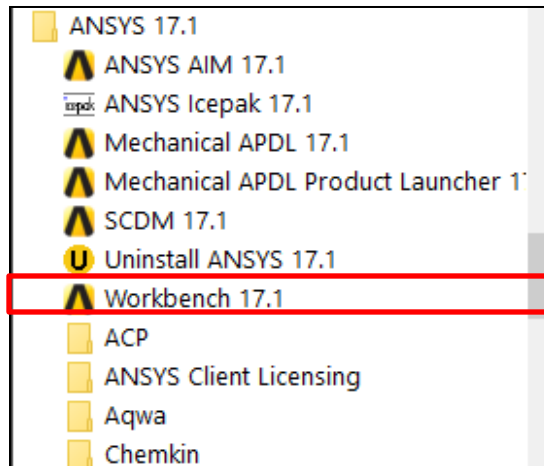
Parameters	Symbol	Unit	Value
Chord Length	C	m	0.3048
Domain radius	Rc	m	6
Angle of attack	$\alpha$	m	0



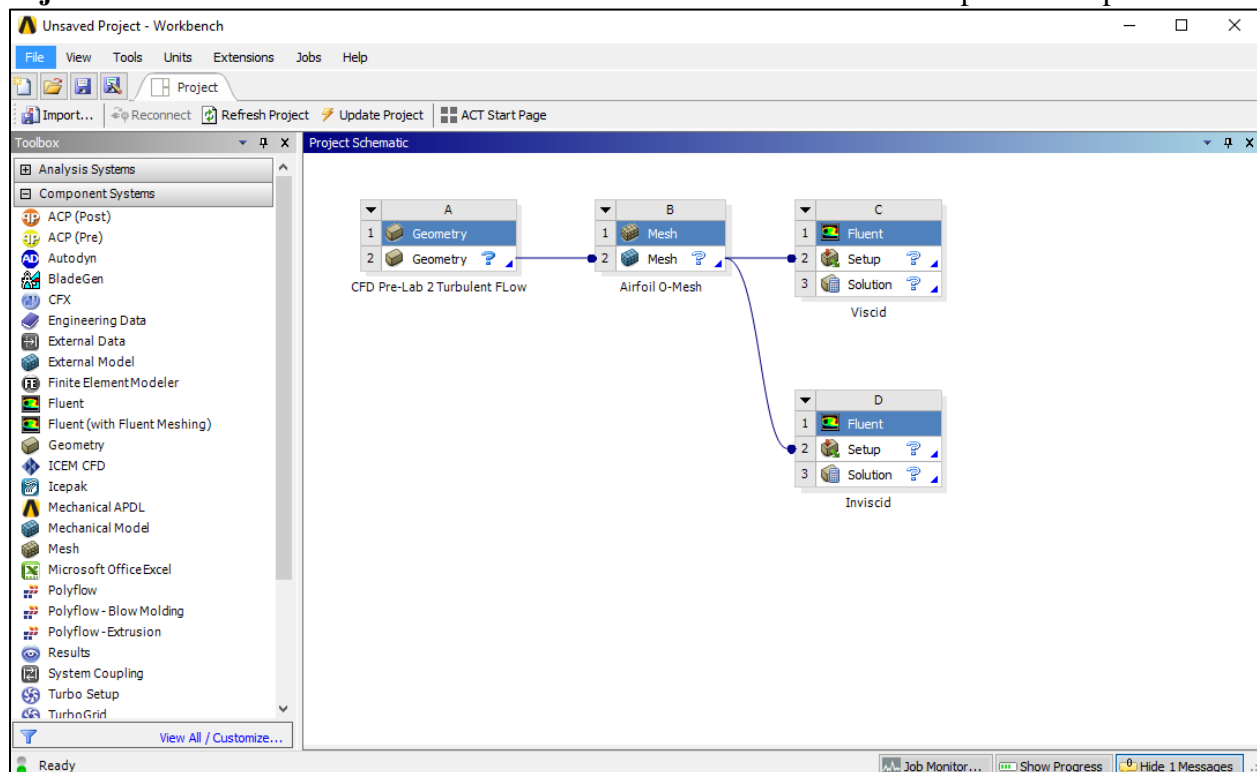
**Figure 1 – Geometry**

## 3. Open ANSYS Workbench

### 3.1. Start > All Programs > ANSYS 17.1 > Workbench 17.1



3.2. From the ANSYS Workbench home screen (**Project Schematic**), drag and drop a **Geometry**, **Mesh**, and **Fluent** component from the **Component Systems** drop down menu onto the **Project Schematic**. The **Project Schematic** should resemble the schematic below. Rename the components as per below.



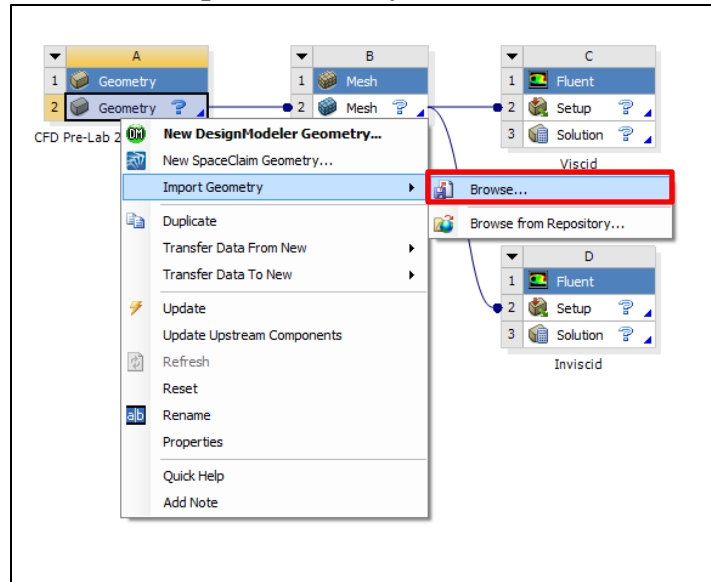
3.3. Create a Folder on the H: Drive called “CFD Pre-Lab 2”.

3.4. Save the project file by clicking **File > Save As...**

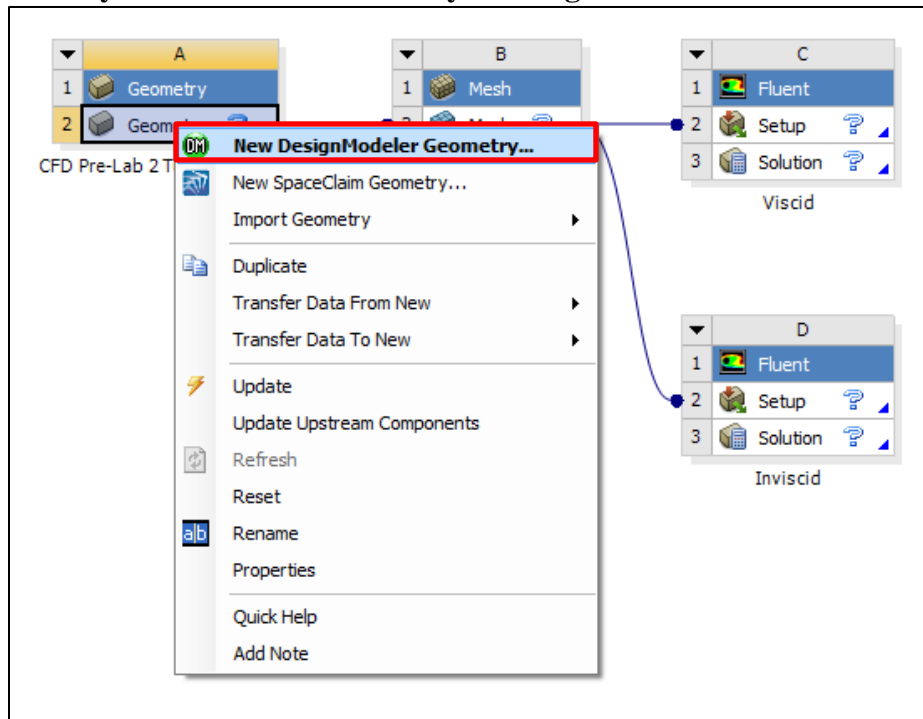
3.5. Save the project onto the H: Drive in the folder you just created and name it “CFD Pre-Lab 2 Turbulent Flow”.

## 4. Geometry

4.1. Right click **Geometry** then select **Import Geometry > Browse...** Select **intro\_airfoil.igs** and click **OK**.

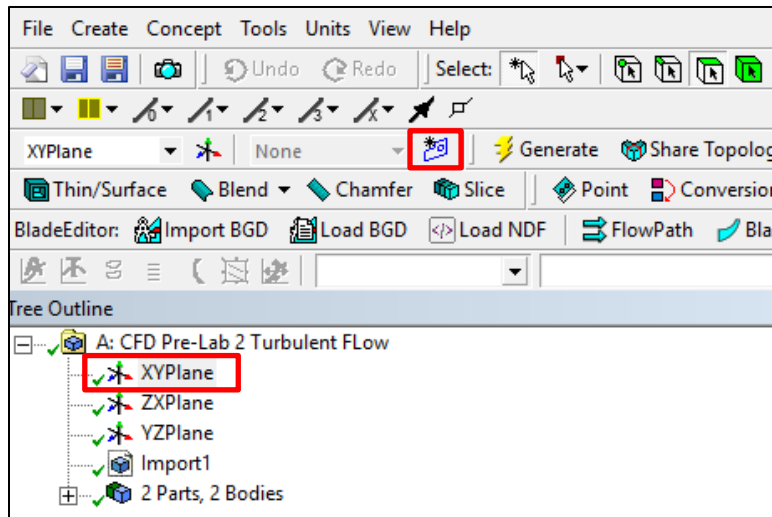


4.2. Right click **Geometry** and select **Edit Geometry in DesignModeler...**

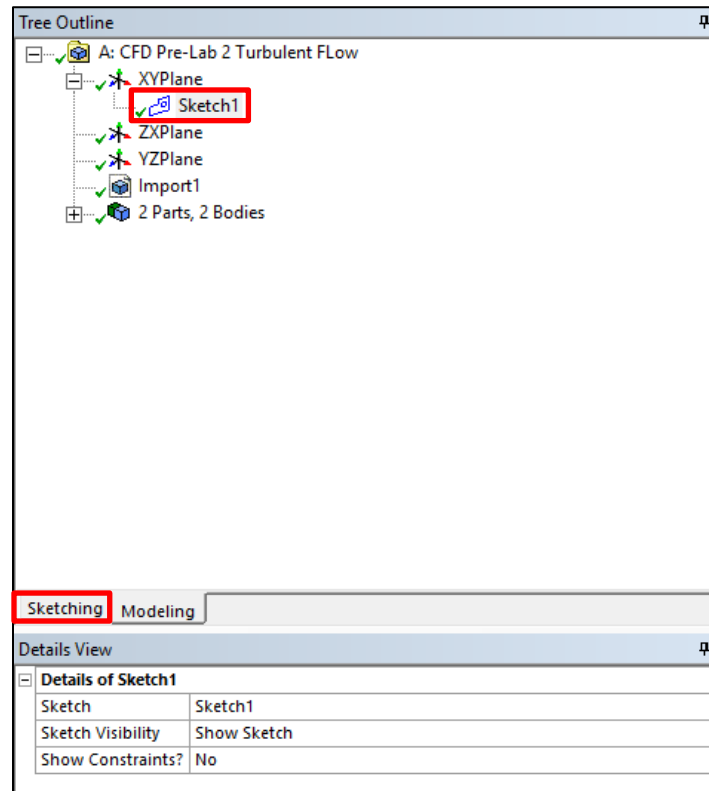


4.3. Click **Generate** to generate the airfoil geometry.

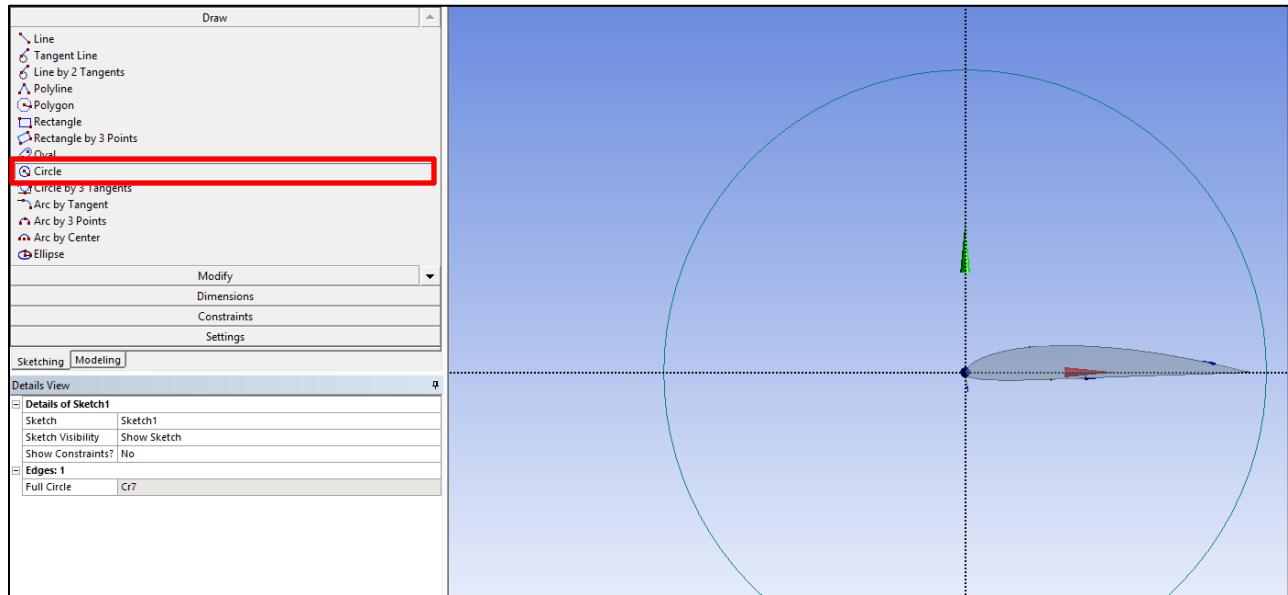
4.4. Select **XYPlane** and click **New Sketch** button.



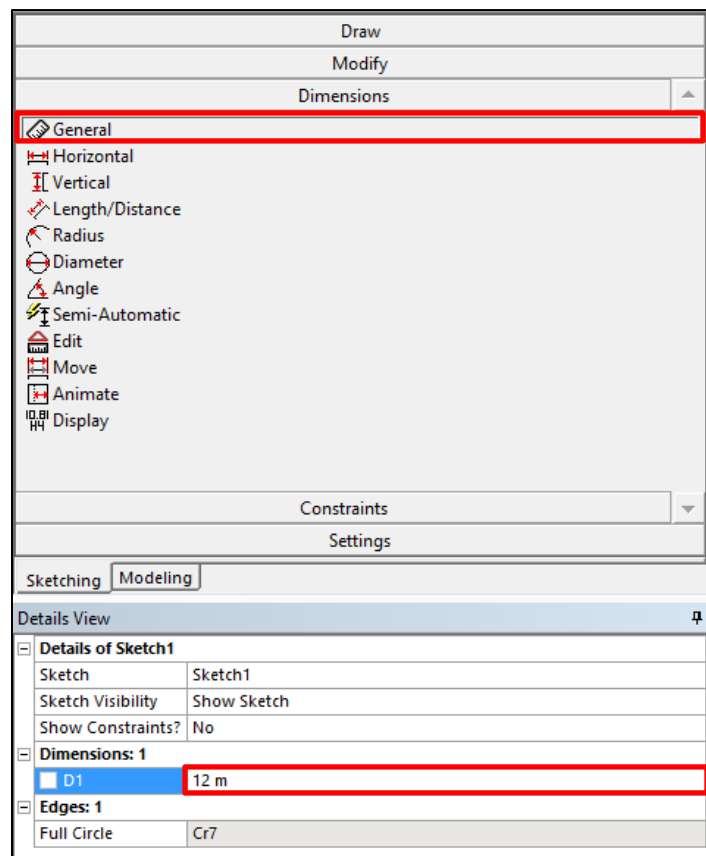
4.5. Select the sketch you created and click **Sketching** button.



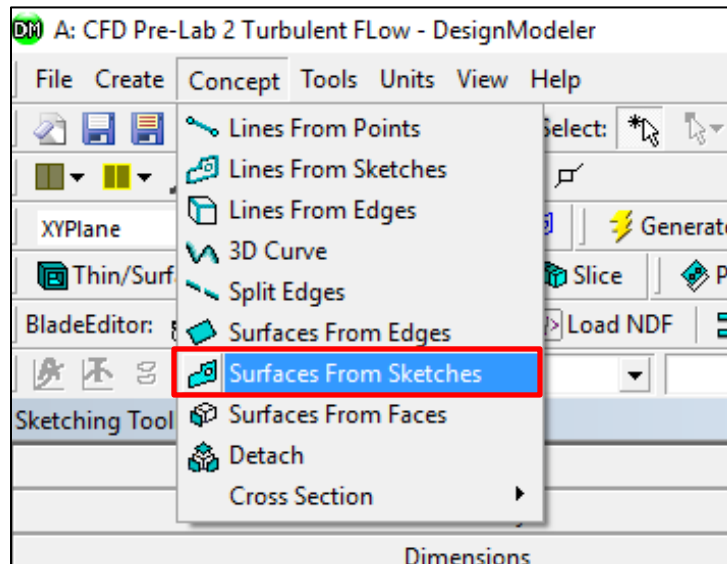
4.6. **Sketching Toolbox > Draw > Circle.** Click on the xy-plane origin, “P” should pop up when you are on the origin, and click off, encapsulating the airfoil geometry. (Click z-arrow at right bottom to change the view as perpendicular to xy plane)



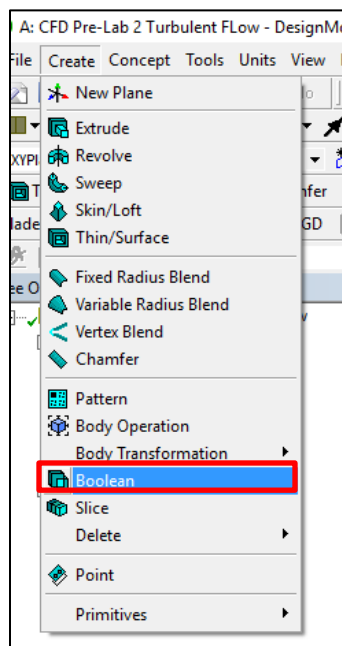
4.7. **Sketching Toolboxes > Dimensions > General.** Click on the circle and change the diameter to 12m.



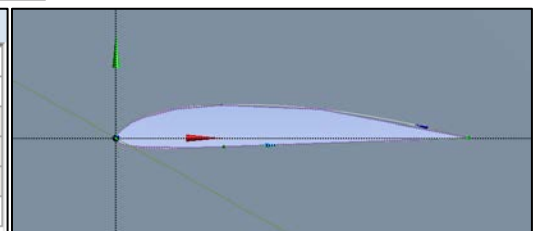
4.8. Select the **Modeling** tab. **Concept** > **Surfaces From Sketches**. Select your sketch, click apply in the **Details View** for **Base Objects**, and click the **Generate** button.



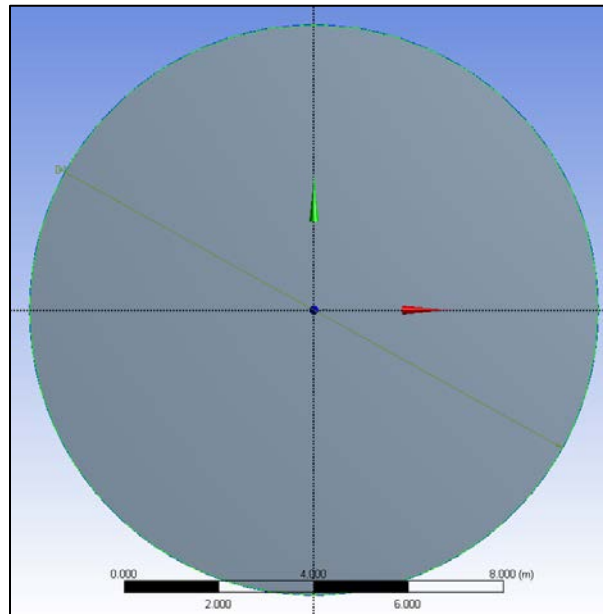
4.9. **Create** > **Boolean**. Change operation to **Subtract** then select the circle for **Target Bodies** then select the airfoil (click on the airfoil until only the airfoil's color turns yellow) for **Tool Bodies** and click **Generate**. This will subtract the airfoil surface from the circle.



Details View	
Details of Boolean1	
Boolean	Boolean1
Operation	Subtract
Target Bodies	1 Body
Tool Bodies	1 Body
Preserve Tool Bodies?	No

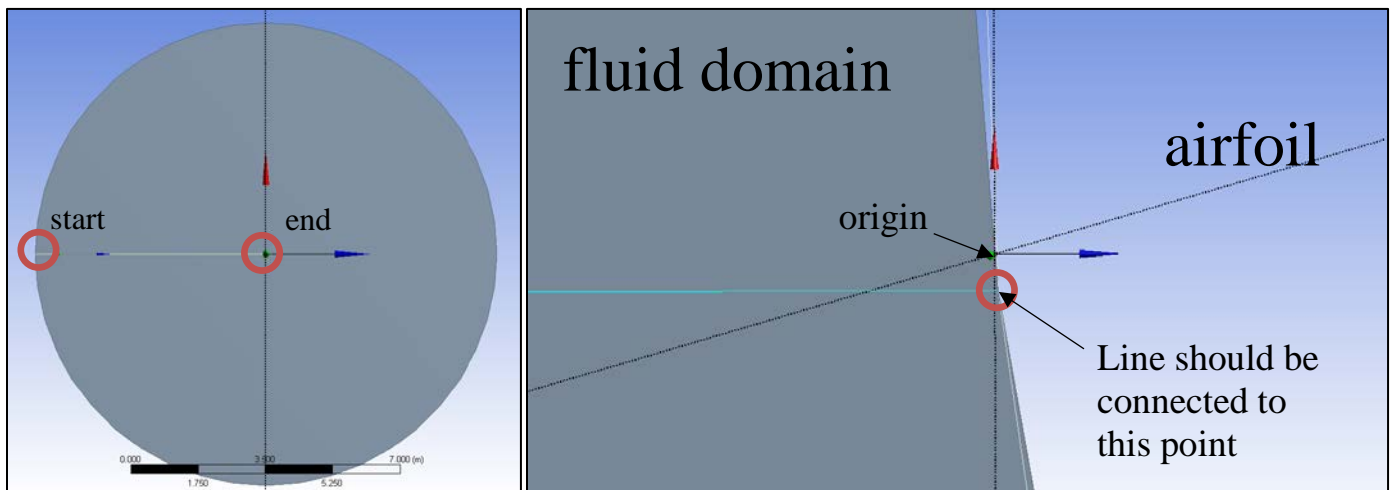


4.10. **Concept > Split Edges.** Select the perimeter of the circle, make sure the **Fraction** is **0.5**, and click **Apply**. Select **Generate**. This should split the circle into two semicircles. You can see the semicircles by selecting the perimeter above and below the x-axis.



4.11. Repeat the process from 4.10 on the two semicircles. This should yield four circular quadrants.

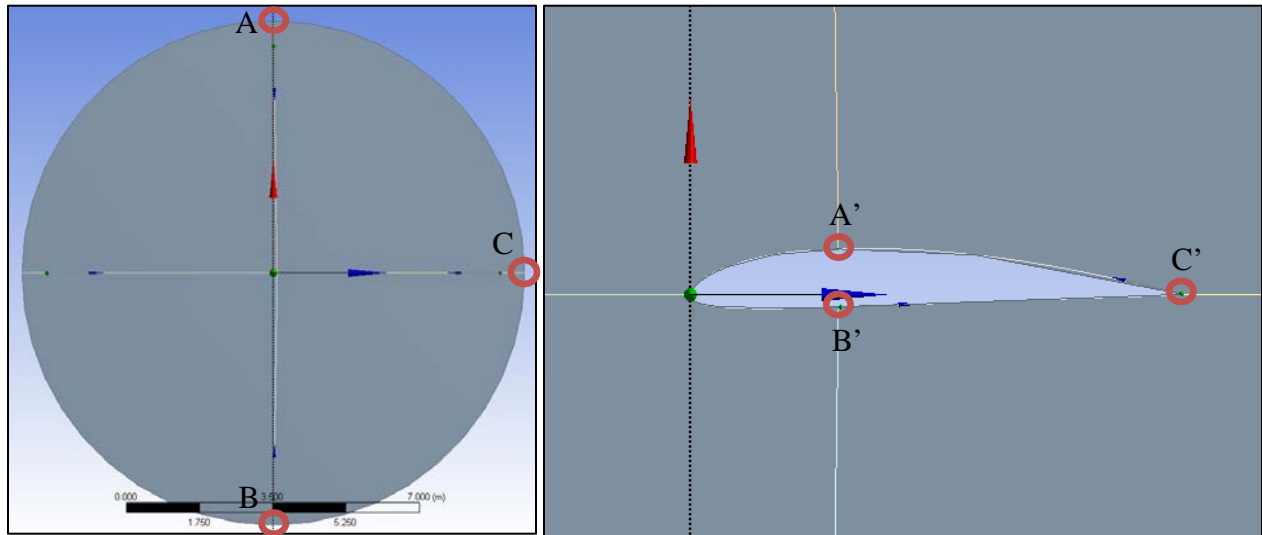
4.12. **Concept > Lines From Points.** Draw a line from the point on the circle to the point on the airfoil making sure to hold **Ctrl** while doing so. (Note: The point on the front part of the airfoil is not exactly on the origin, you need to make sure to select the point on the airfoil and not the origin. Zoom in and find the point just below the origin and select that point. The images below show the locations of the points circled. When selecting points to generate lines with, be sure to select the point on circle and then the point on the airfoil as to avoid complications when sizing mesh). Apply **View > Display Vertices** to alleviate this issue.



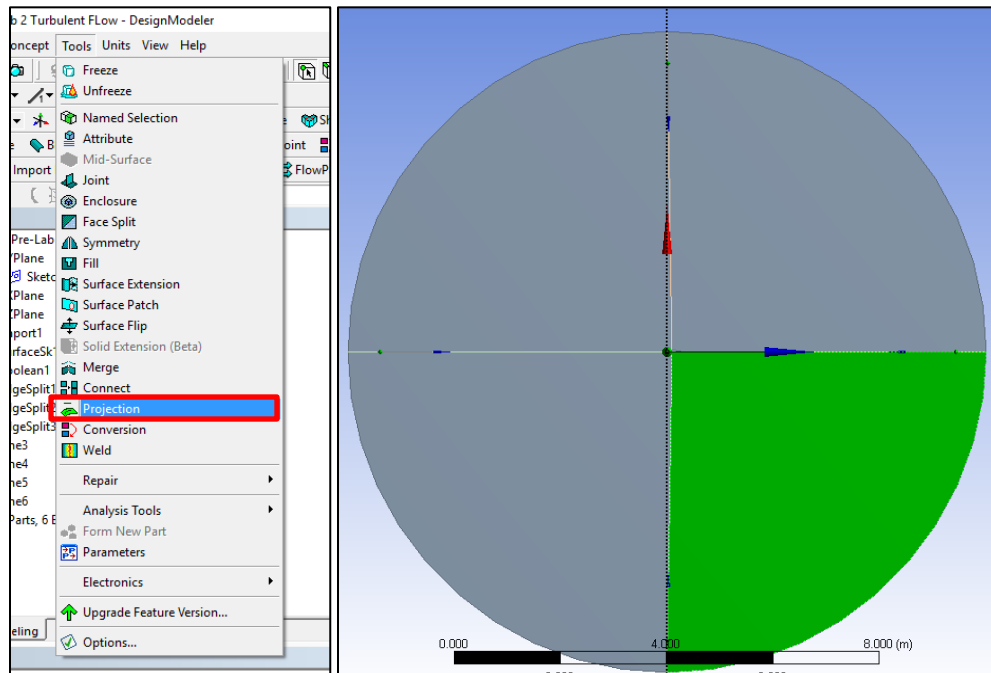


4.13. Once you select both points hit **Apply**. Then click **Generate**.

4.14. Repeat this process creating lines from the edge of the circle to the airfoil starting from the circle and ending at the airfoil (**A to A'**, **B to B'** and **C to C'**). The images below show the locations of the points on the airfoil and the points on the circle.



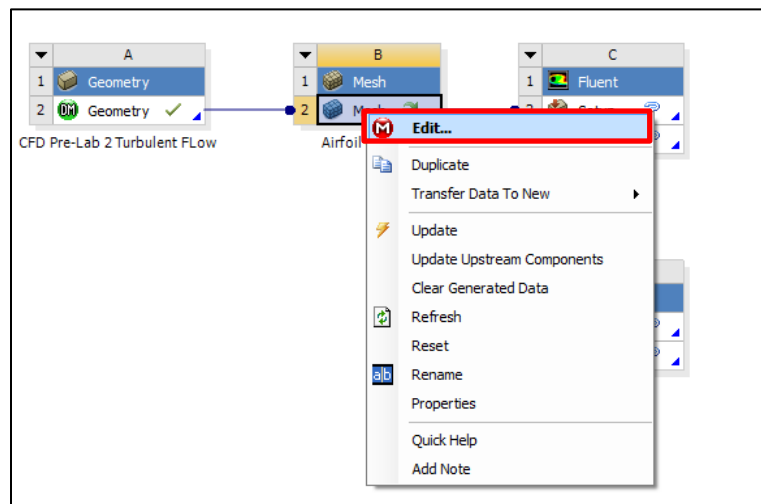
4.15. **Tools > Projection**. Select the four edges you created for **Edges** and click **Apply** then select the domain circle for **Target** and click **Apply**, then click **Generate**. This will split your geometry into four sections.



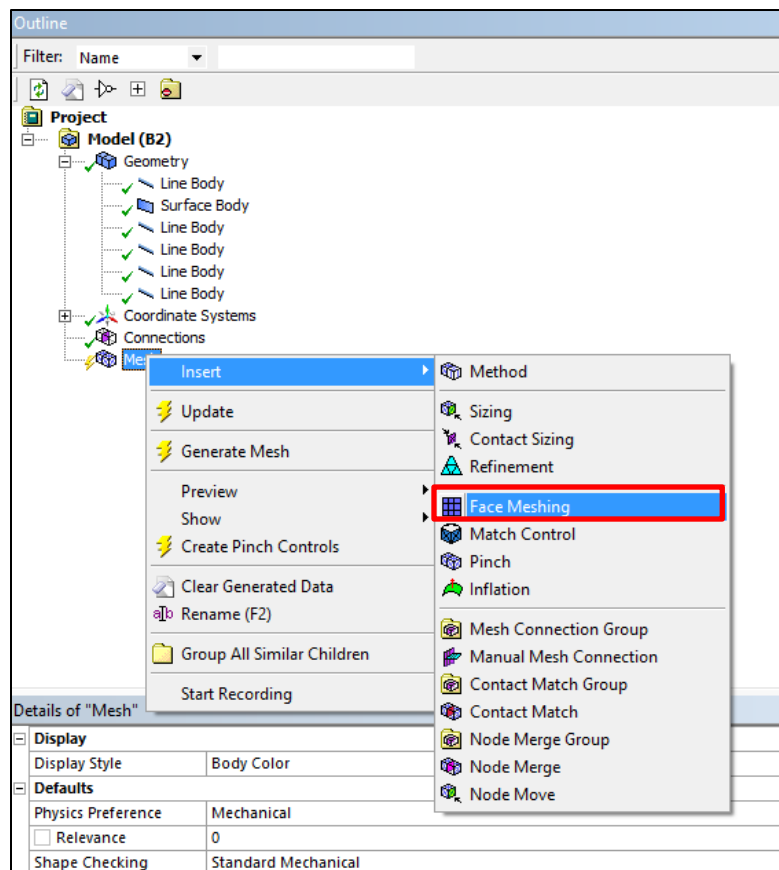
4.16. **File > Save project** and exit.

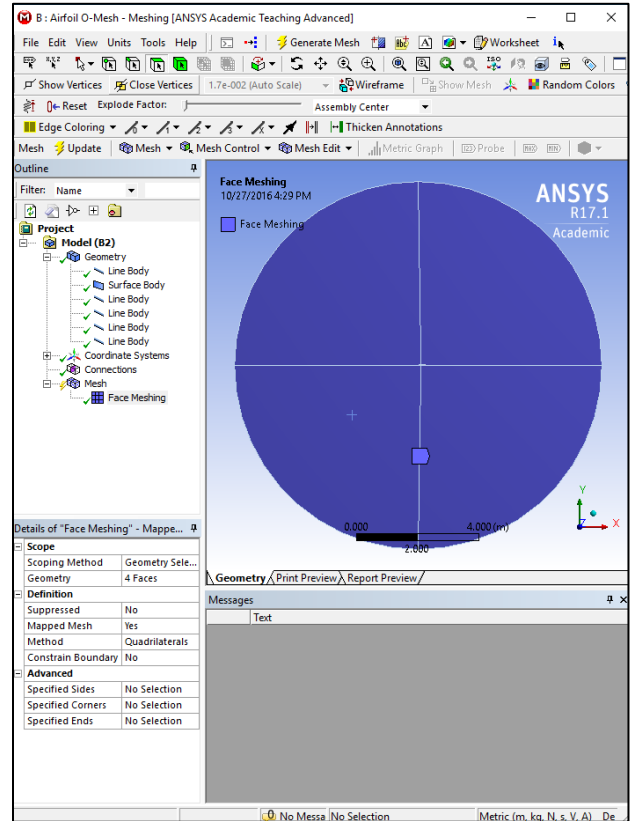
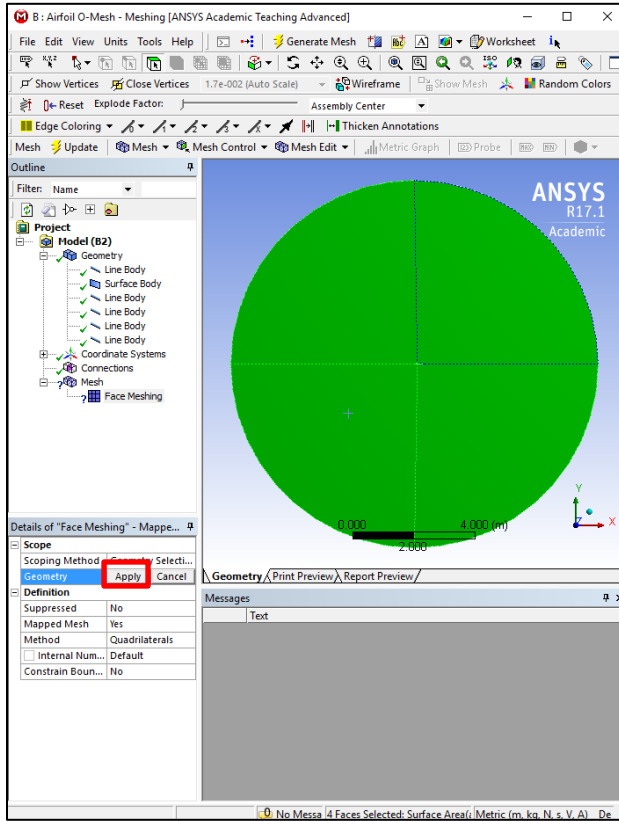
## 5. Mesh Generation

5.1. From Workbench home screen right click on **Mesh** and select **Edit...** from the dropdown menu.

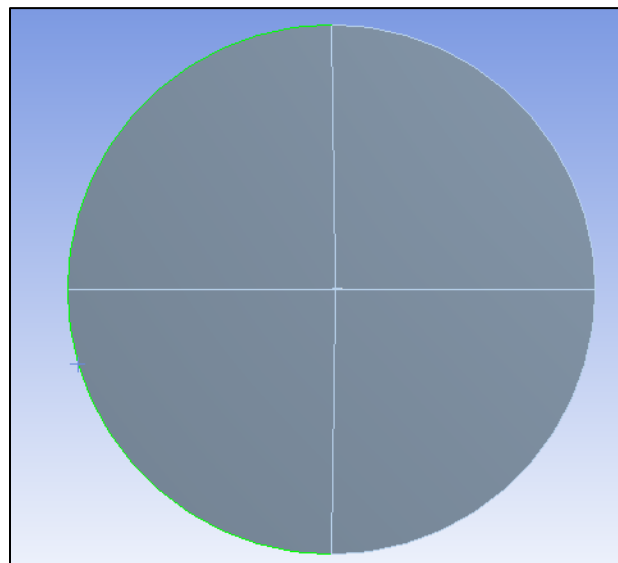
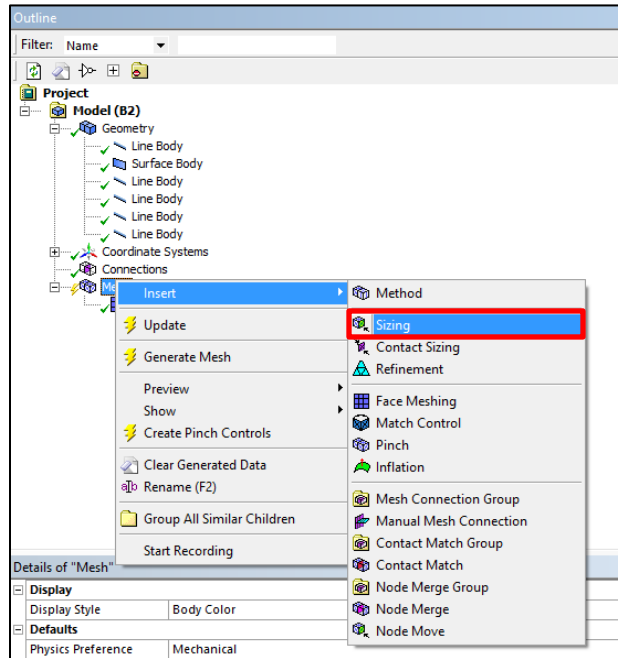


5.2. Right click **Mesh** then **Insert > Face Meshing**. Select the four surfaces while holding Ctrl then click **Apply**.



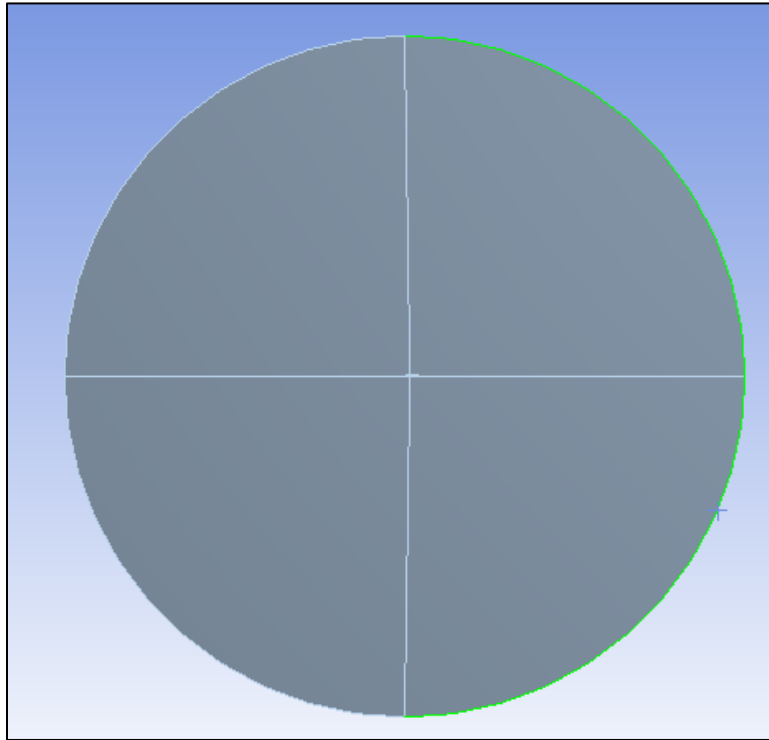


5.3. Right click **Mesh** and **Insert > Sizing**. Make sure to change the selection tool to edge and select two edges as per below and change the parameters as per below (Click 'Edge selector' from the top).



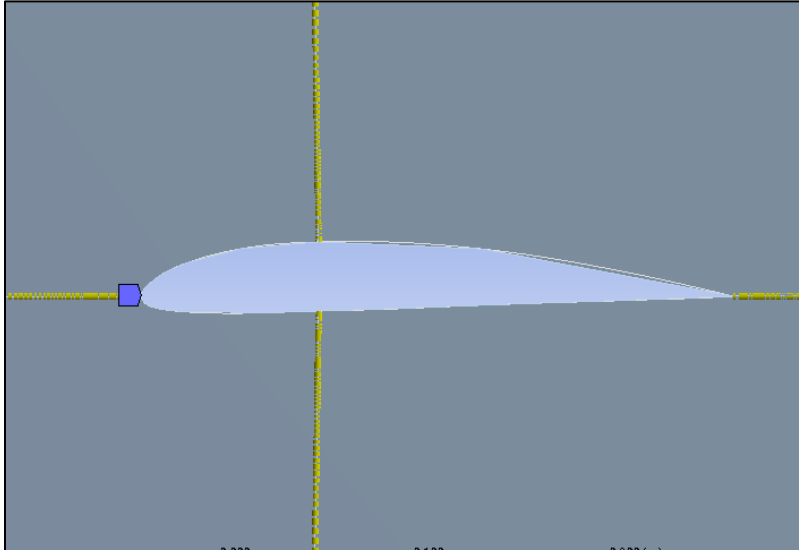
Details of "Edge Sizing" - Sizing	
<b>Scope</b>	
Scoping Method	Geometry Selection
Geometry	2 Edges
<b>Definition</b>	
Suppressed	No
Type	Number of Divisions
<input type="checkbox"/> Number of Divisions	50
Behavior	Hard
Bias Type	No Bias

5.4. Right click **Mesh** and **Insert** > **Sizing**. Select two edges as per below and change the parameters as per below.



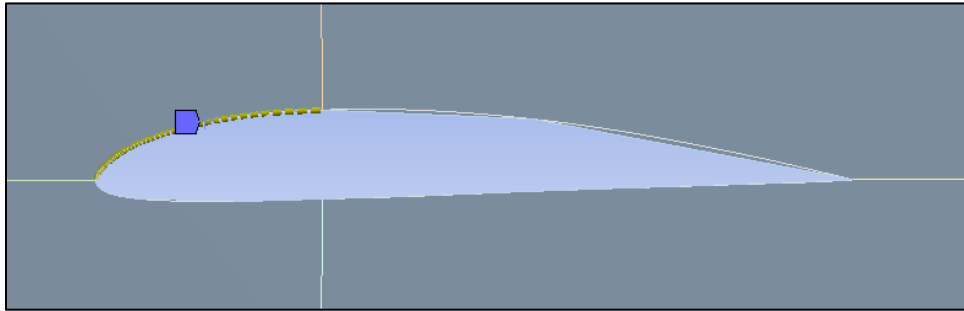
Details of "Edge Sizing 2" - Sizing	
<b>Scope</b>	
Scoping Method	Geometry Selection
Geometry	2 Edges
<b>Definition</b>	
Suppressed	No
Type	Number of Divisions
<input type="checkbox"/> Number of Divisions	75
<b>Behavior</b>	Hard
Bias Type	No Bias

5.5. Right click **Mesh** and **Insert** > **Sizing**. Select all for lines leading from the circle to the airfoil surface, and click **Apply**. Change parameters as per below. (Note: If you did not create the lines starting from the outer circle and ending on the airfoil surface, you may have issues with biasing. If this is your case, size the lines individually making sure that the sizing is finest at the surface of the airfoil.)



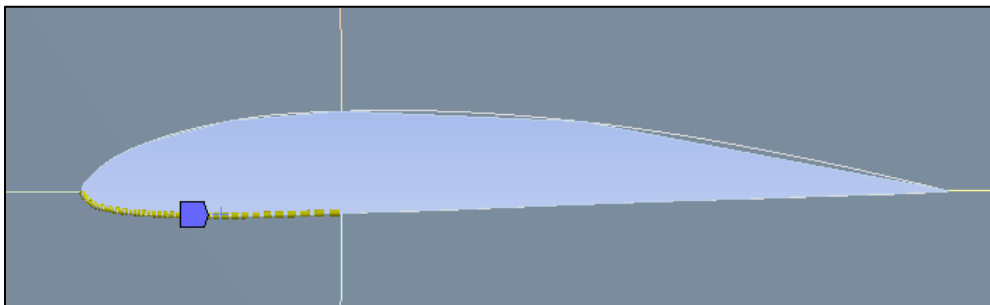
Details of "Edge Sizing 3" - Sizing	
[-] <b>Scope</b>	
Scoping Method	Geometry Selection
Geometry	4 Edges
[-] <b>Definition</b>	
Suppressed	No
Type	Number of Divisions
<input type="checkbox"/> Number of Divisions	200
Behavior	Hard
Bias Type	----- - - - - -
Bias Option	Bias Factor
<input checked="" type="checkbox"/> Bias Factor	5000.

5.6. Right click **Mesh** and **Insert** > **Sizing**. Select the surface at the top of leading edge of the airfoil and click **Apply**. Change Parameters as per below.



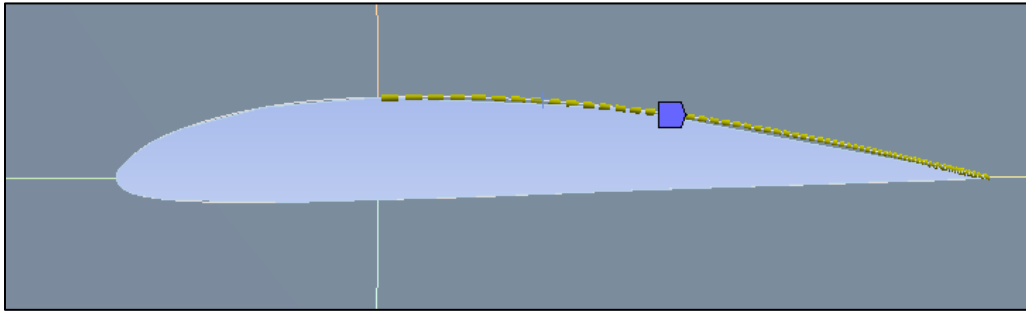
[-] <b>Scope</b>	
Scoping Method	Geometry Selection
Geometry	1 Edge
[-] <b>Definition</b>	
Suppressed	No
Type	Number of Divisions
<input type="checkbox"/> Number of Divisions	50
Behavior	Hard
Bias Type	-----
Bias Option	Bias Factor
<input checked="" type="checkbox"/> Bias Factor	15.
Reverse Bias	No Selection

5.7. Right click **Mesh** and **Insert** > **Sizing**. Select the surface at the bottom of leading edge of the airfoil and click **Apply**. Change Parameters as per below.



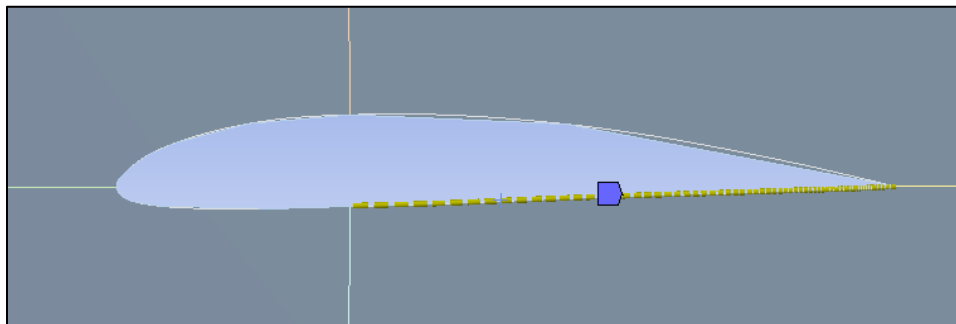
[-] <b>Scope</b>	
Scoping Method	Geometry Selection
Geometry	1 Edge
[-] <b>Definition</b>	
Suppressed	No
Type	Number of Divisions
<input type="checkbox"/> Number of Divisions	50
Behavior	Hard
Bias Type	- - - - -
Bias Option	Bias Factor
<input checked="" type="checkbox"/> Bias Factor	15.
Reverse Bias	No Selection

5.8. Right click **Mesh** and **Insert** > **Sizing**. Select the surface at the top of trailing edge of the airfoil and click **Apply**. Change Parameters as per below.



Scope	
Scoping Method	Geometry Selection
Geometry	1 Edge
Definition	
Suppressed	No
Type	Number of Divisions
<input type="checkbox"/> Number of Divisions	75
Behavior	Hard
Bias Type	- - - - -
Bias Option	Bias Factor
<input checked="" type="checkbox"/> Bias Factor	15.
Reverse Bias	No Selection

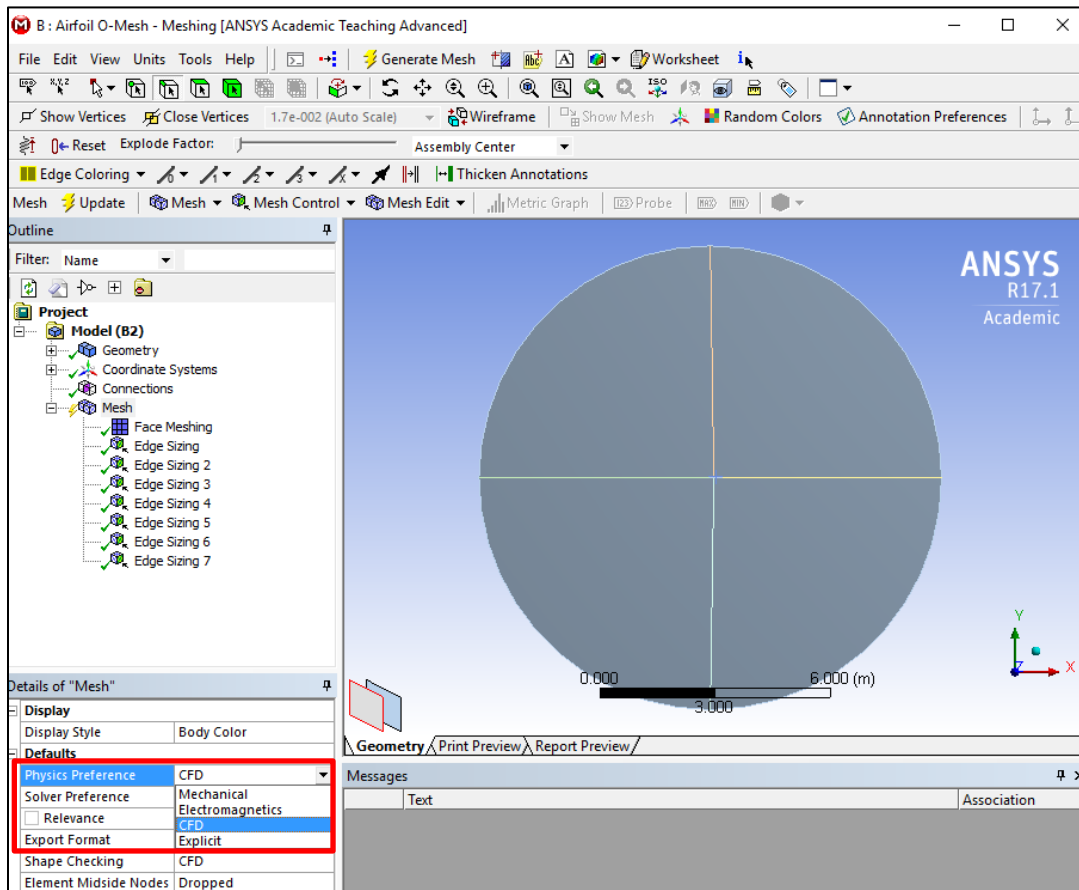
5.9. Right click **Mesh** and **Insert** > **Sizing**. Select the surface at the bottom of trailing edge of the airfoil and click **Apply**. Change Parameters as per below.



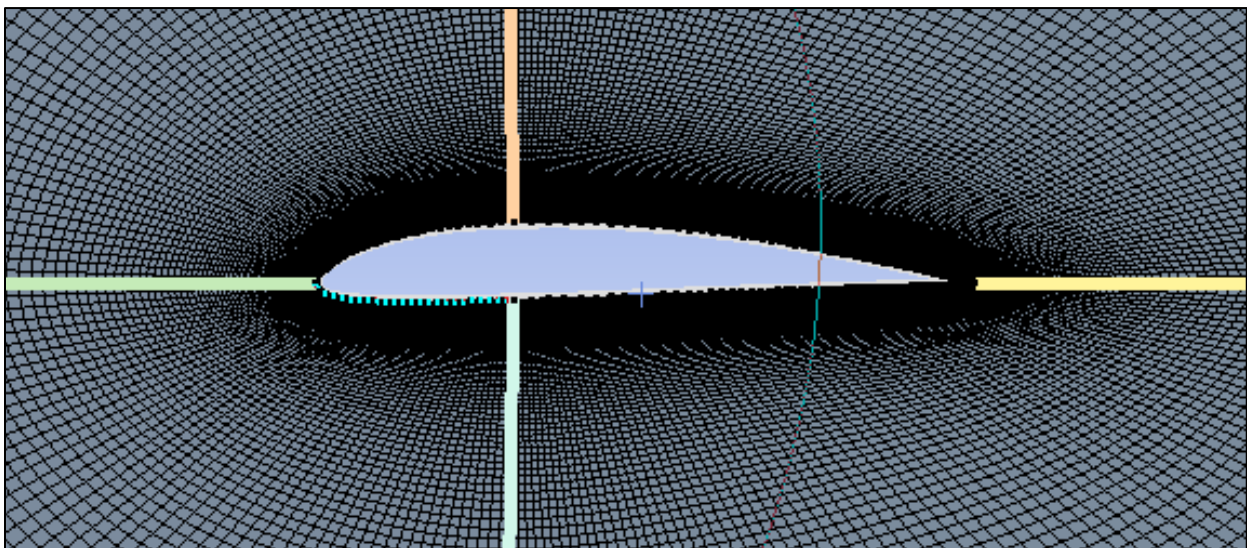
Scope	
Scoping Method	Geometry Selection
Geometry	1 Edge
Definition	
Suppressed	No
Type	Number of Divisions
<input type="checkbox"/> Number of Divisions	75
Behavior	Hard
Bias Type	-----
Bias Option	Bias Factor
<input checked="" type="checkbox"/> Bias Factor	15.
Reverse Bias	No Selection



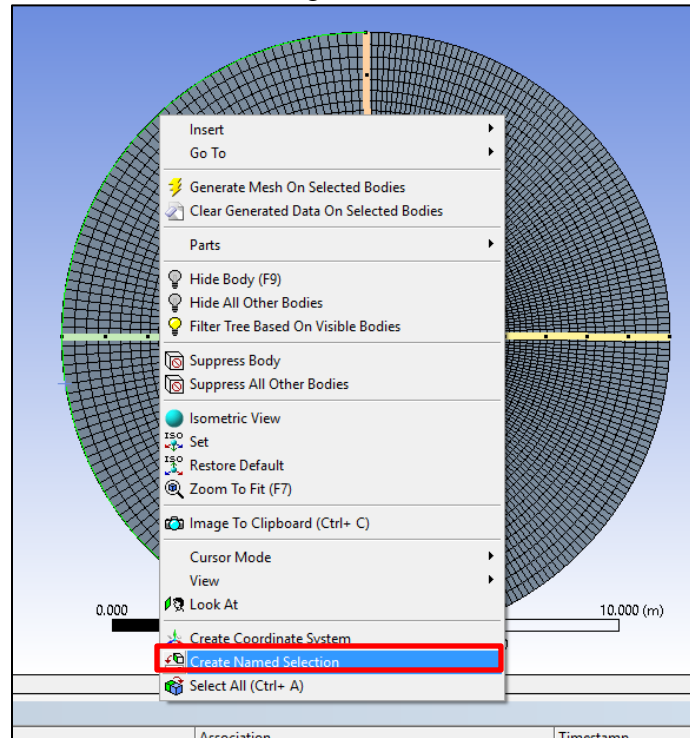
5.10. Click on Mesh in the **Outline**, this will bring up **Details of “Mesh”**, under **Physics Preference**, change the mesh type to **CFD** from **Mechanical**.



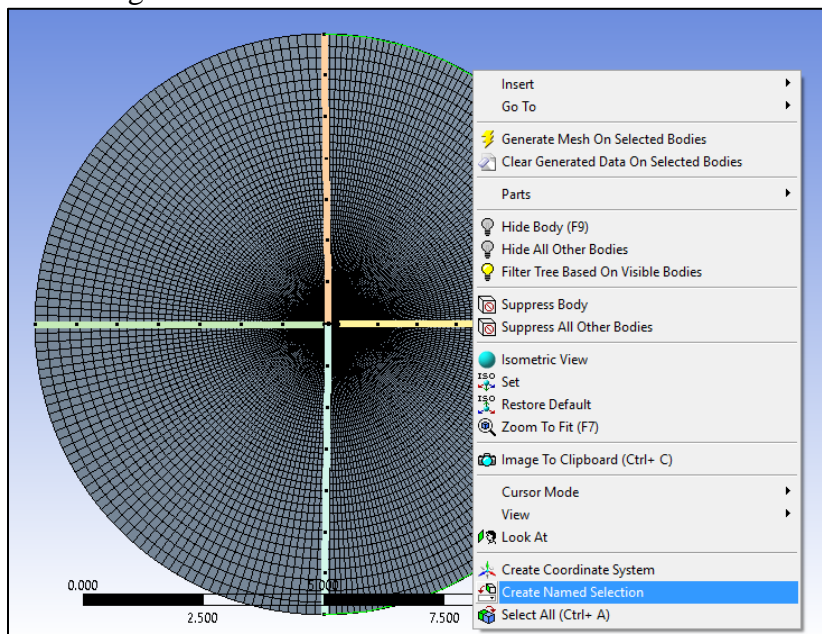
5.11. Click **Generate Mesh**. Click on the **Mesh** button under the **Outline** and make sure it resembles the mesh below.



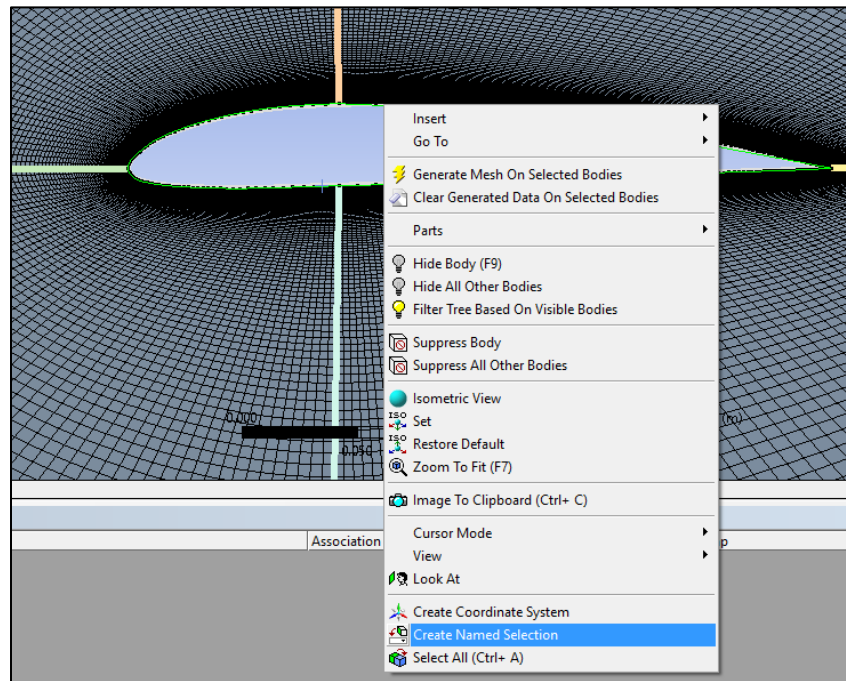
5.12. Hold **Ctrl** and select the two left most semicircle arcs, right click on them and select **Create Named Selection**, name the selection *inlet*. Use the edge select button from the toolbar.



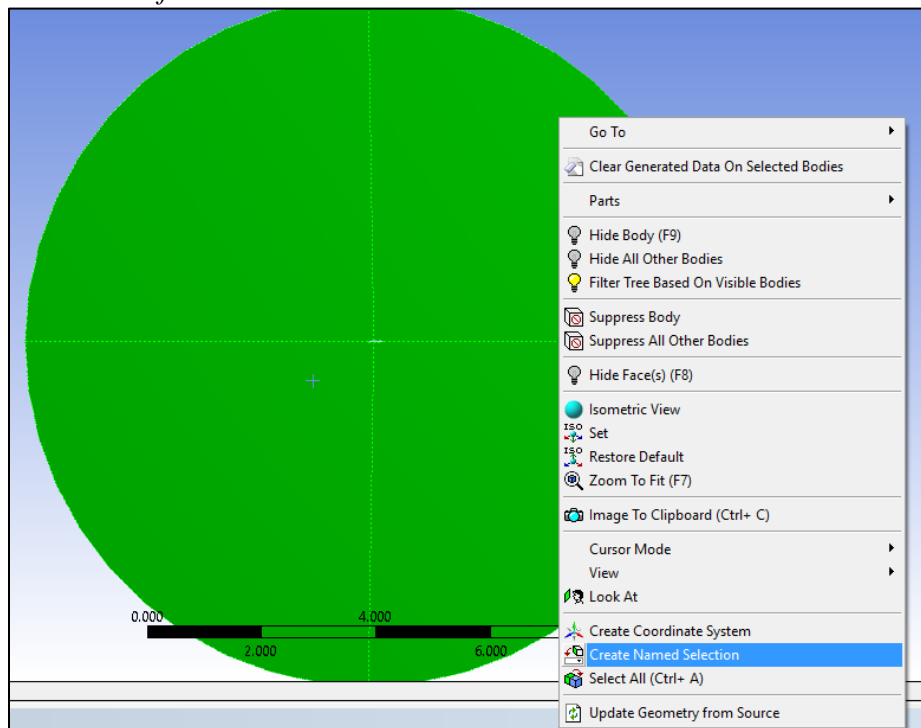
5.13. Do the same for the two right most semicircle arcs and name them *outlet*.



5.14. Select the four regions that make the airfoil, right click and **Create Named Selection** and name it *airfoil*.



5.15. Now use the 'Face selector' button to select the four semicircle quadrants and **Create Named Selection** and name them *fluid*.

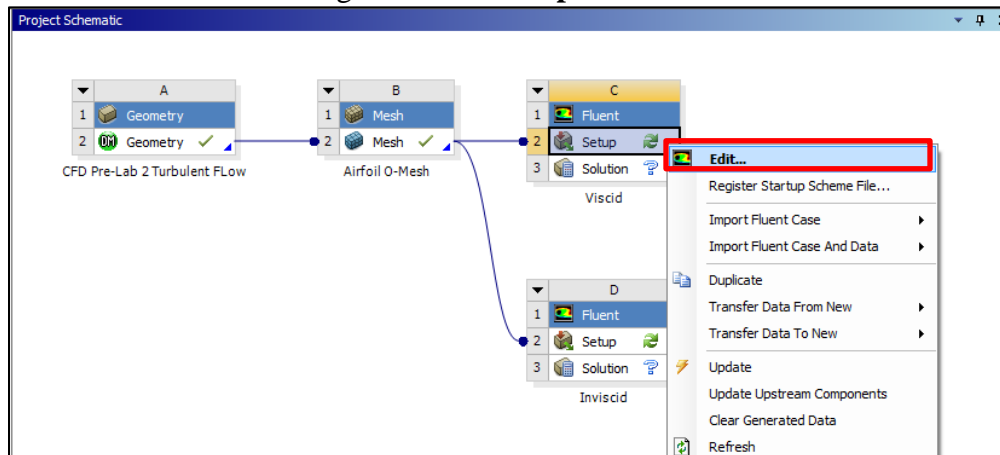


5.16. **File > Save Project.** Exit the window

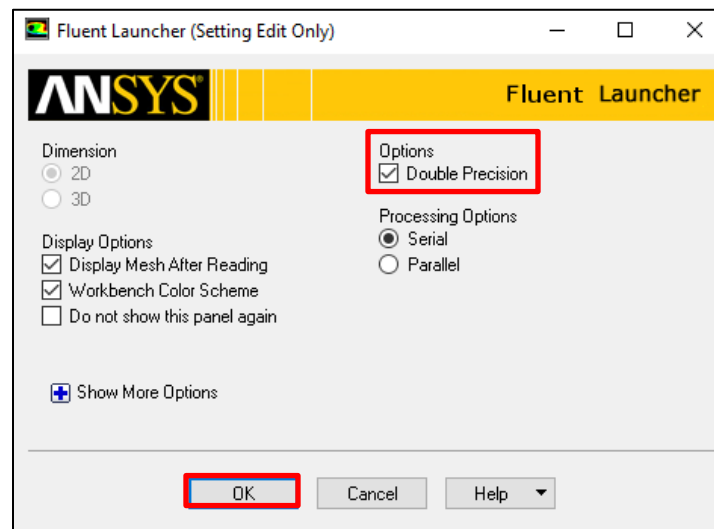
## 6. Setup

6.1. If there is not a checkmark next to **Mesh**, right click **Mesh**, and click the **Update**.

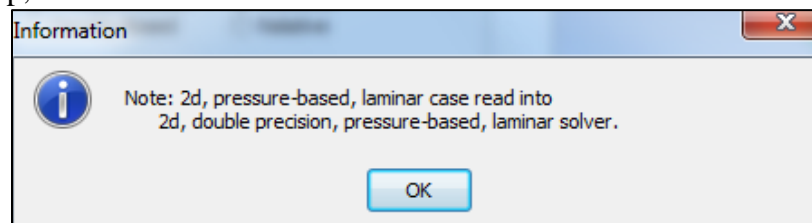
6.2. From the Workbench home screen right click on **Setup** and select **Edit...**



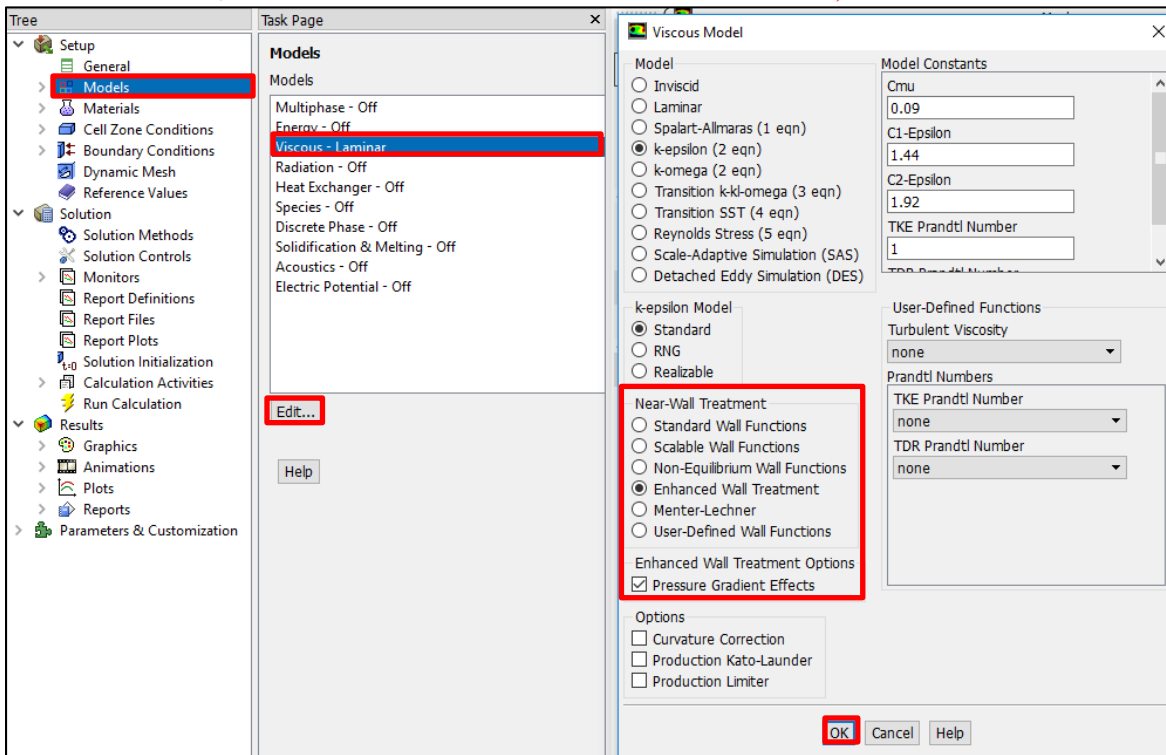
6.3. Select **Double Precision** and click **OK**.



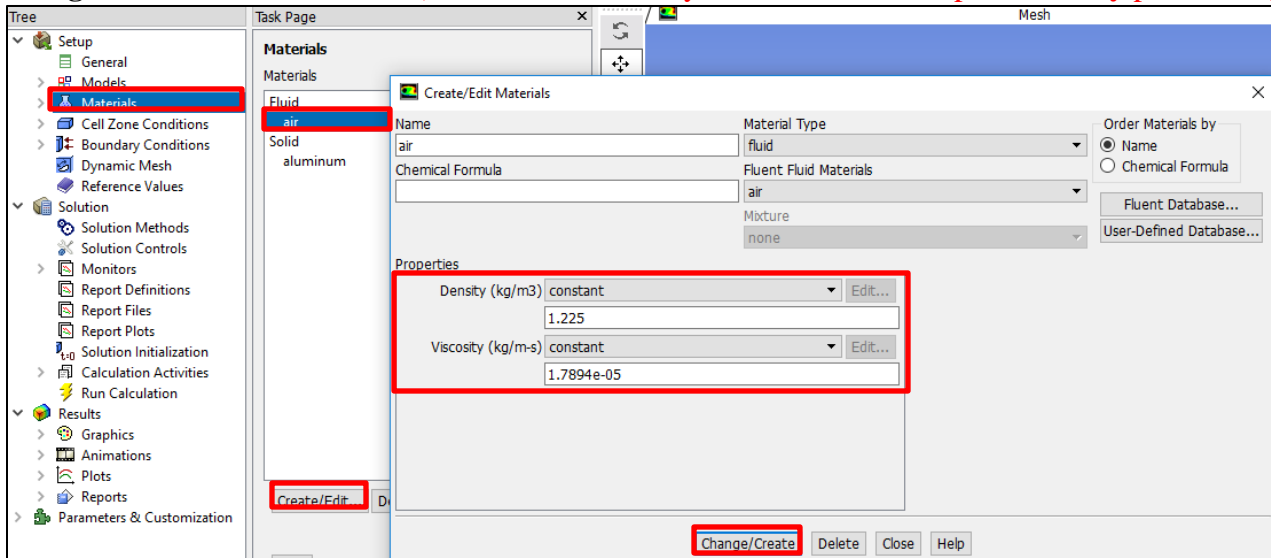
6.4. If this note pops up, click OK



6.5. **Setup > Models > Viscous –Laminar > Edit...** Change the parameters as per below and click **OK**.  
 (For the inviscid case, select **Inviscid** from the **Viscous Model** menu.)



6.6. **Setup > Materials > air > Create/Edit...** Change parameters as per experimental data and click **Change/Create** and then **Close**. (For inviscid model, you do not need to input **Viscosity** parameter.)



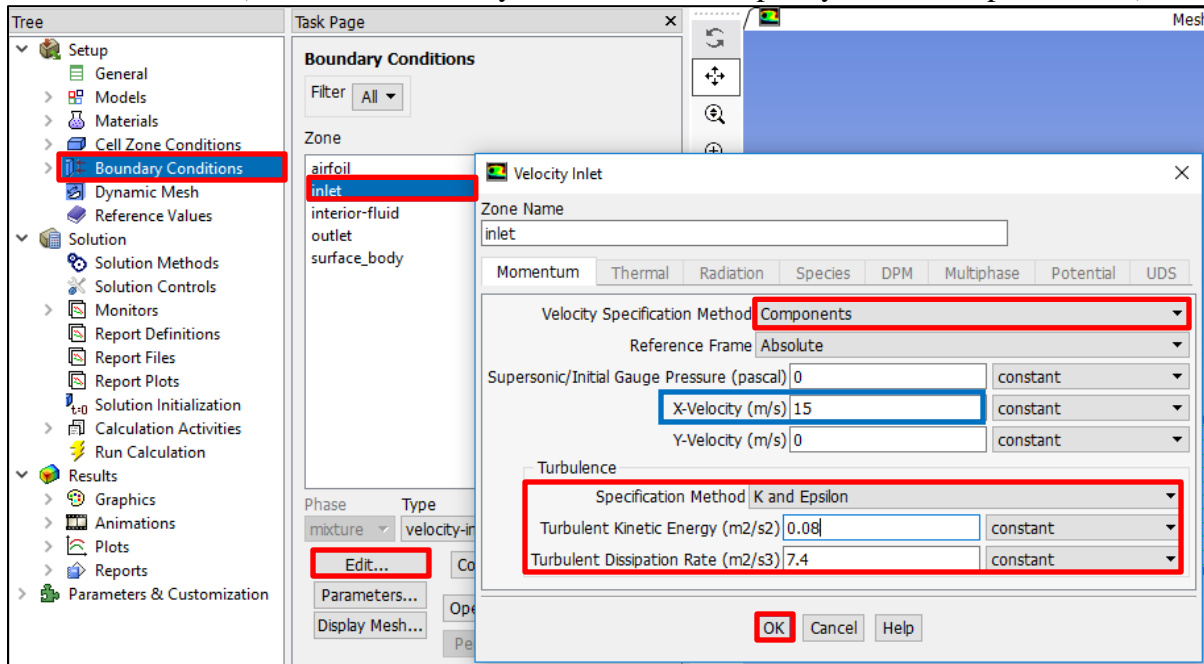
Use the air properties at the **room temperature** when you conducted EFD Lab3. You can use the following website to calculate air properties from the temperature:

<http://www.mhfl.uwaterloo.ca/old/onlinetools/airprop/airprop.html>

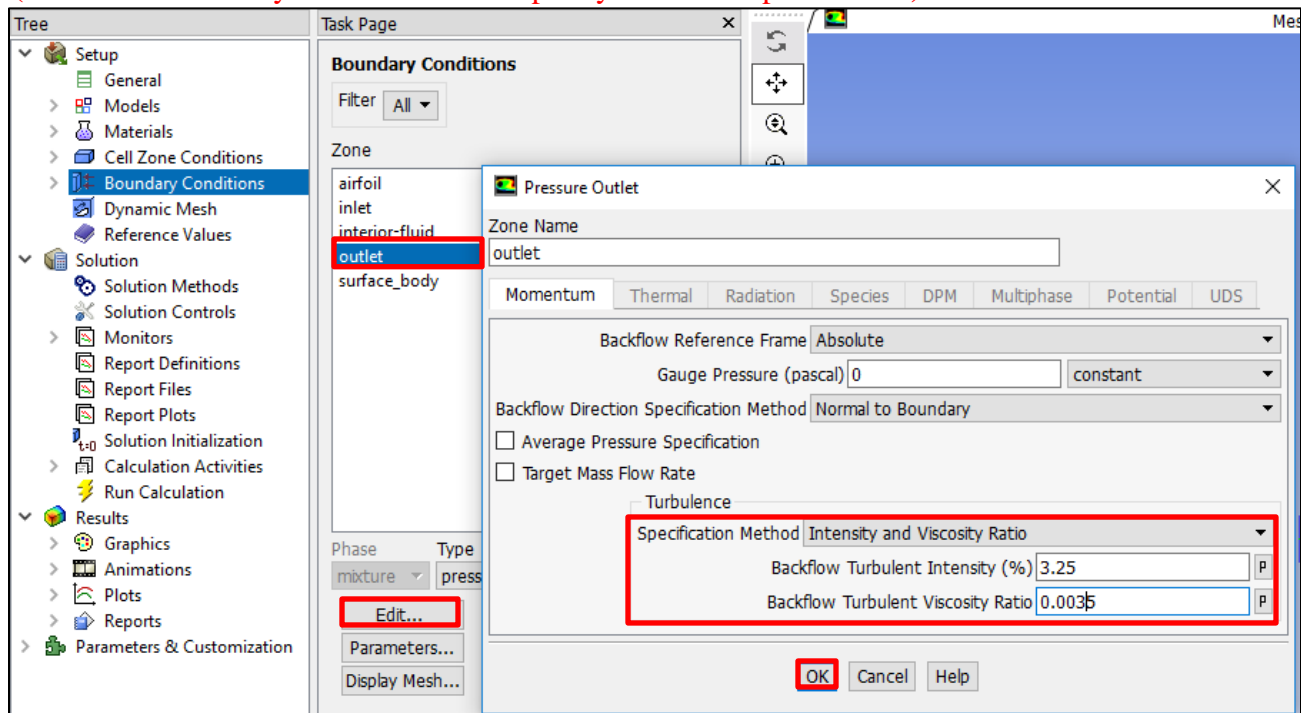
The values in the figure above are for 24° temperature.

**NOTE:** viscosity used in ANSYS is the **dynamic viscosity** ( $kg/m \cdot s$ ), **NOT** kinematic viscosity ( $m^2/s$ )

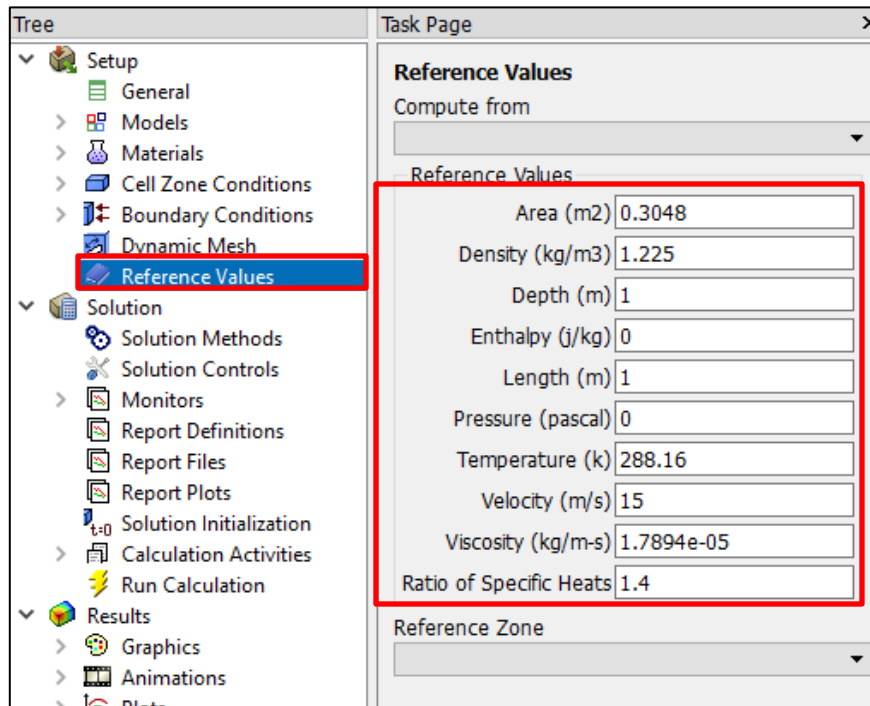
6.7. **Setup** > **Boundary Conditions** > **inlet** > **Edit...** Change parameters as per experimental data (blue box) and default values (red boxes) and click **OK**. The experimental value can be found from the EFD Lab 3 data reduction sheet. (For inviscid model you do not need to specify turbulence parameters.)



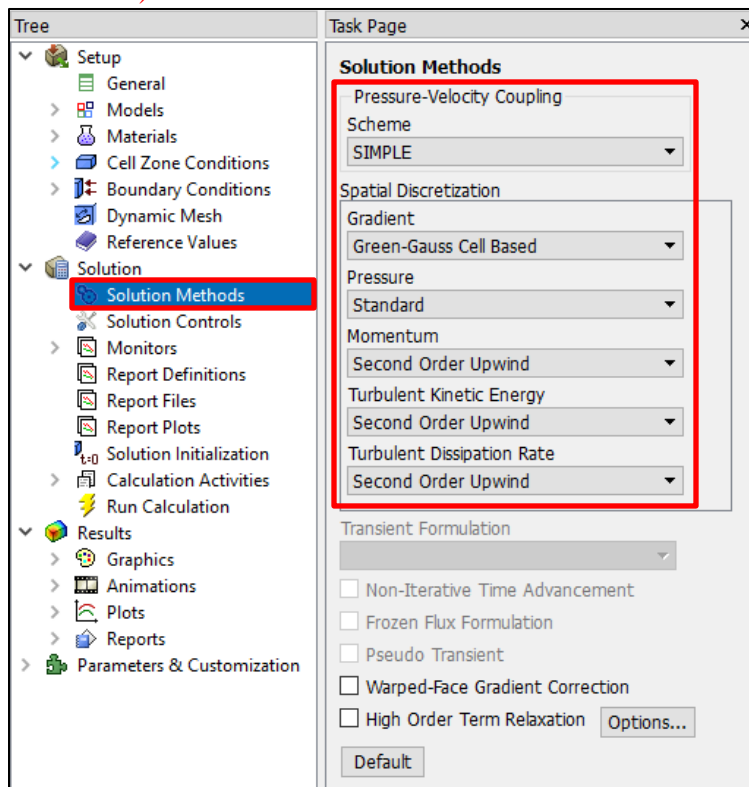
6.8. **Setup** > **Boundary Conditions** > **Outlet** > **Edit...** Change the parameters as per below and click **OK**. (For inviscid model you do not need to specify turbulence parameters.)



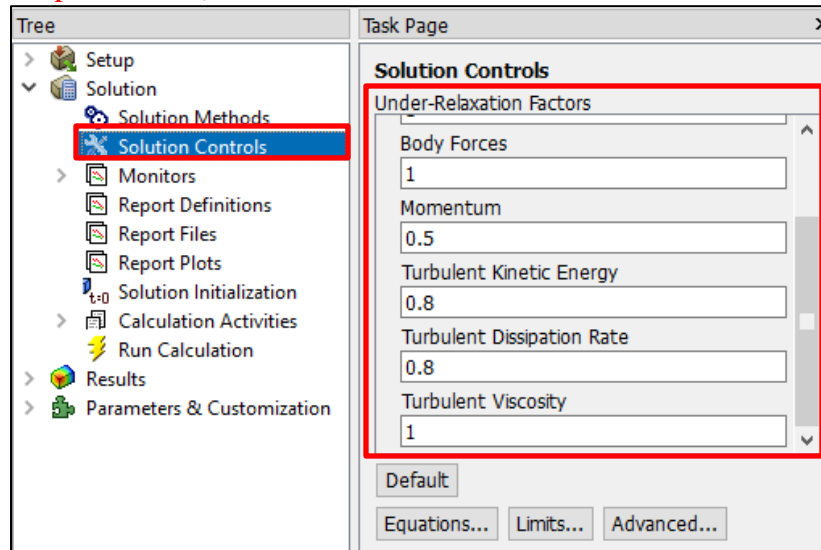
6.9. **Setup > Reference Values.** Change parameters as per below. The velocity, temperature, density, and viscosity should be entered from EFD data.



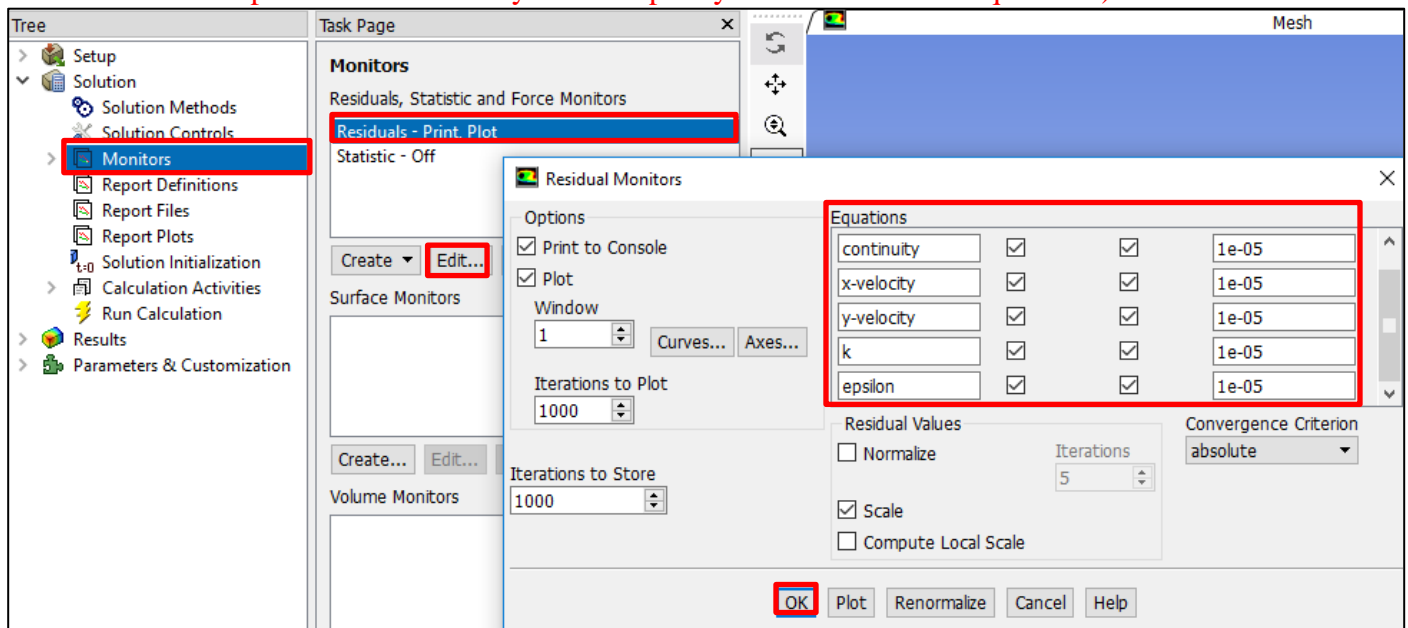
6.10. **Solution > Solution Methods.** Change parameters as per below. (For inviscid case you do not need to input turbulence parameters.)



6.11. **Solution > Solution Controls.** Change Parameters as per below. (If you have problems with the solution converging, you may decrease the Under-Relaxation Factors.) (For inviscid case you do not need to specify turbulence parameters.)

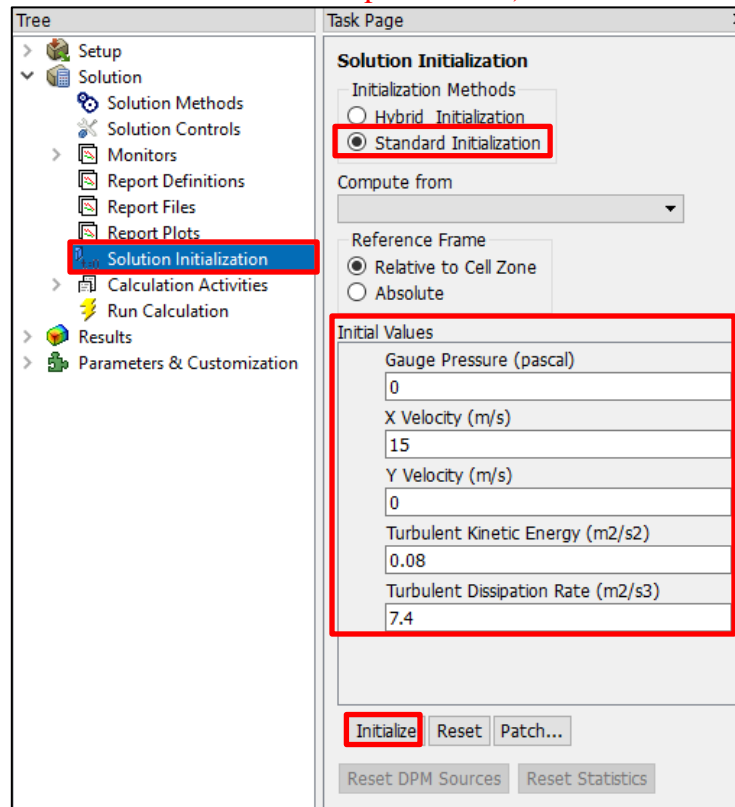


6.12. **Solution > Monitors > Residuals –Print, Plot > Edit....** Change the convergence limit to  $1e-05$  for all five equations and click **OK**. (For inviscid case you do not need to specify convergence criterion for turbulence parameters. You only need to specify criterion for three equations.)

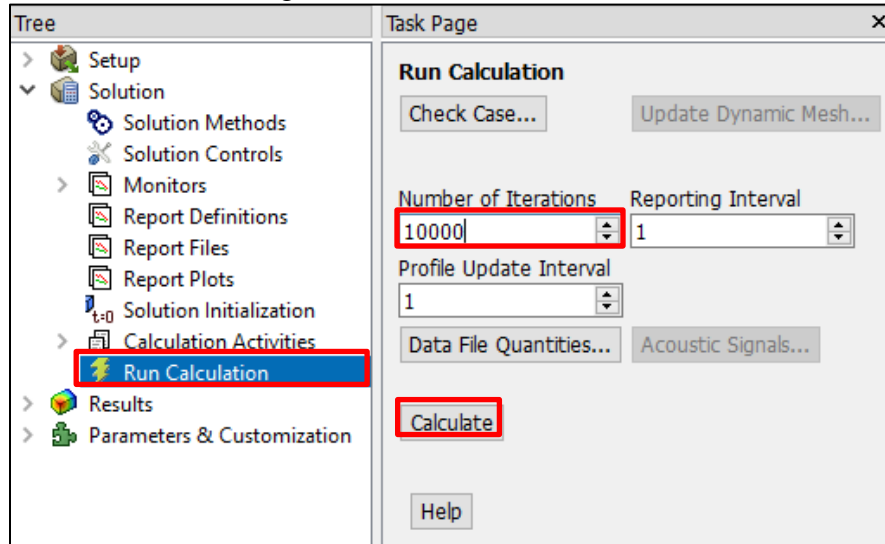




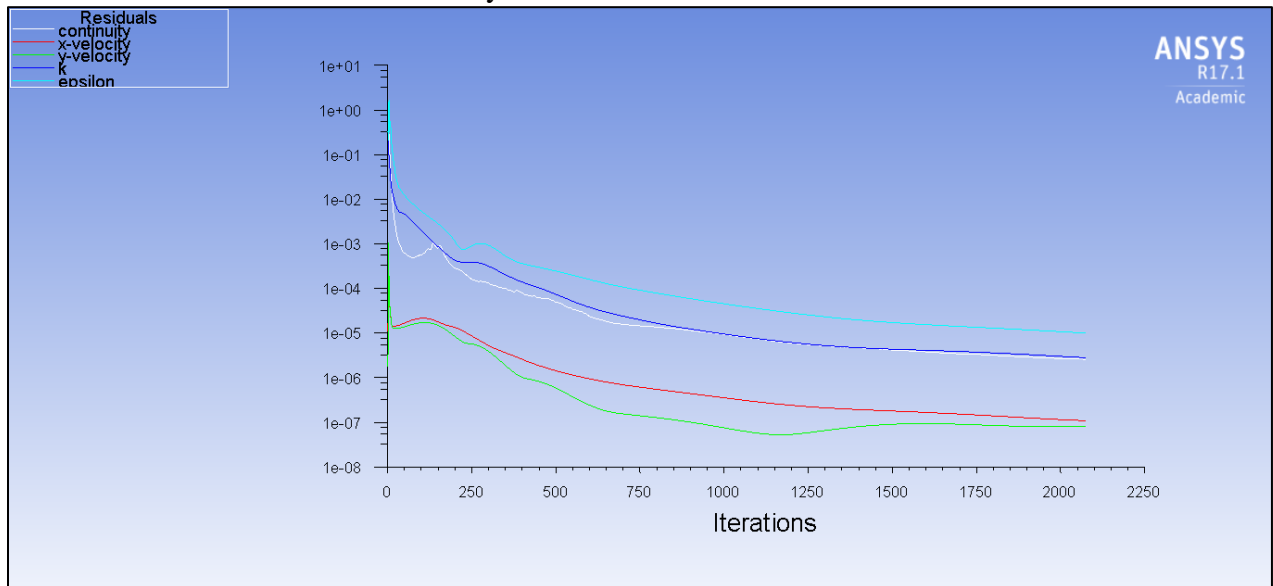
6.13. **Solution > Solution Initialization.** Change the parameters as per below and click **Initialize**. (For inviscid case you do not need to initialize turbulence parameters.)



6.14. **Solution > Run Calculation.** Change the **Number of Iterations** to **10000** and click **Calculate**.



Residual history should look similar to the one below.

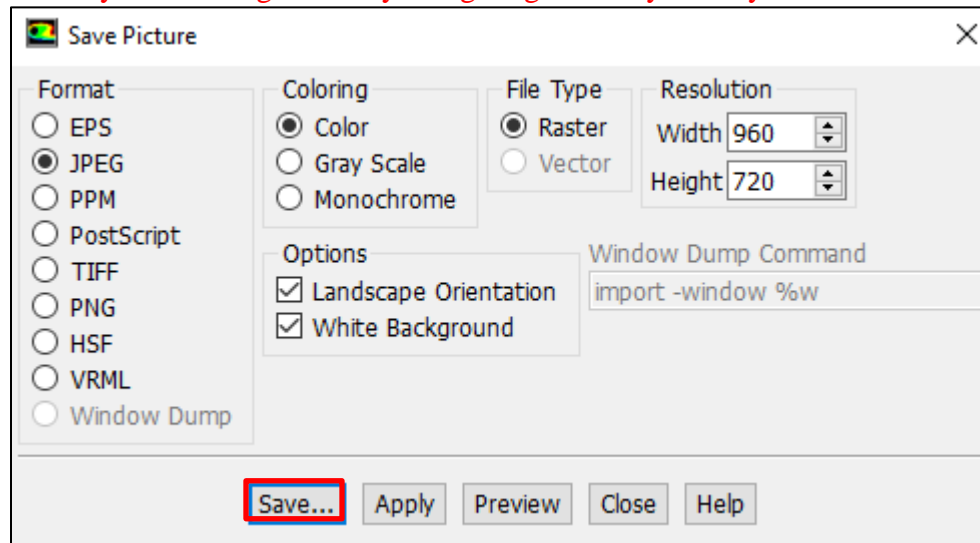


6.15. **File > Save Project.**

## 7. Results

### 7.1. Saving Pictures

**File > Save Picture... > Save...** You can take a snapshot of a current display by using this tool. **Please make sure to save pictures in your working folder by navigating correctly when you enter the browser.**



## 7.2. Plotting Residual History

**Solution > Monitors > Residuals –Print, Plot > Edit... > Plot > OK.** Make sure the parameters are as per below. Name the file “*CFD Pre-Lab 2 Residual History*” when you save the picture.

The screenshot displays the ANSYS Fluent interface. On the left is the 'Tree' panel with 'Monitors' selected. The 'Task Page' shows 'Residuals - Print, Plot' selected. The 'Residual Monitors' dialog box is open, showing the following settings:

- Options:**  Print to Console,  Plot, Window: 1, Iterations to Plot: 1000, Iterations to Store: 1000.
- Equations:**

Residual	Monitor	Check	Convergence	Absolute Criteria
continuity	<input checked="" type="checkbox"/>	<input checked="" type="checkbox"/>	<input checked="" type="checkbox"/>	1e-05
x-velocity	<input checked="" type="checkbox"/>	<input checked="" type="checkbox"/>	<input checked="" type="checkbox"/>	1e-05
y-velocity	<input checked="" type="checkbox"/>	<input checked="" type="checkbox"/>	<input checked="" type="checkbox"/>	1e-05
k	<input checked="" type="checkbox"/>	<input checked="" type="checkbox"/>	<input checked="" type="checkbox"/>	1e-05

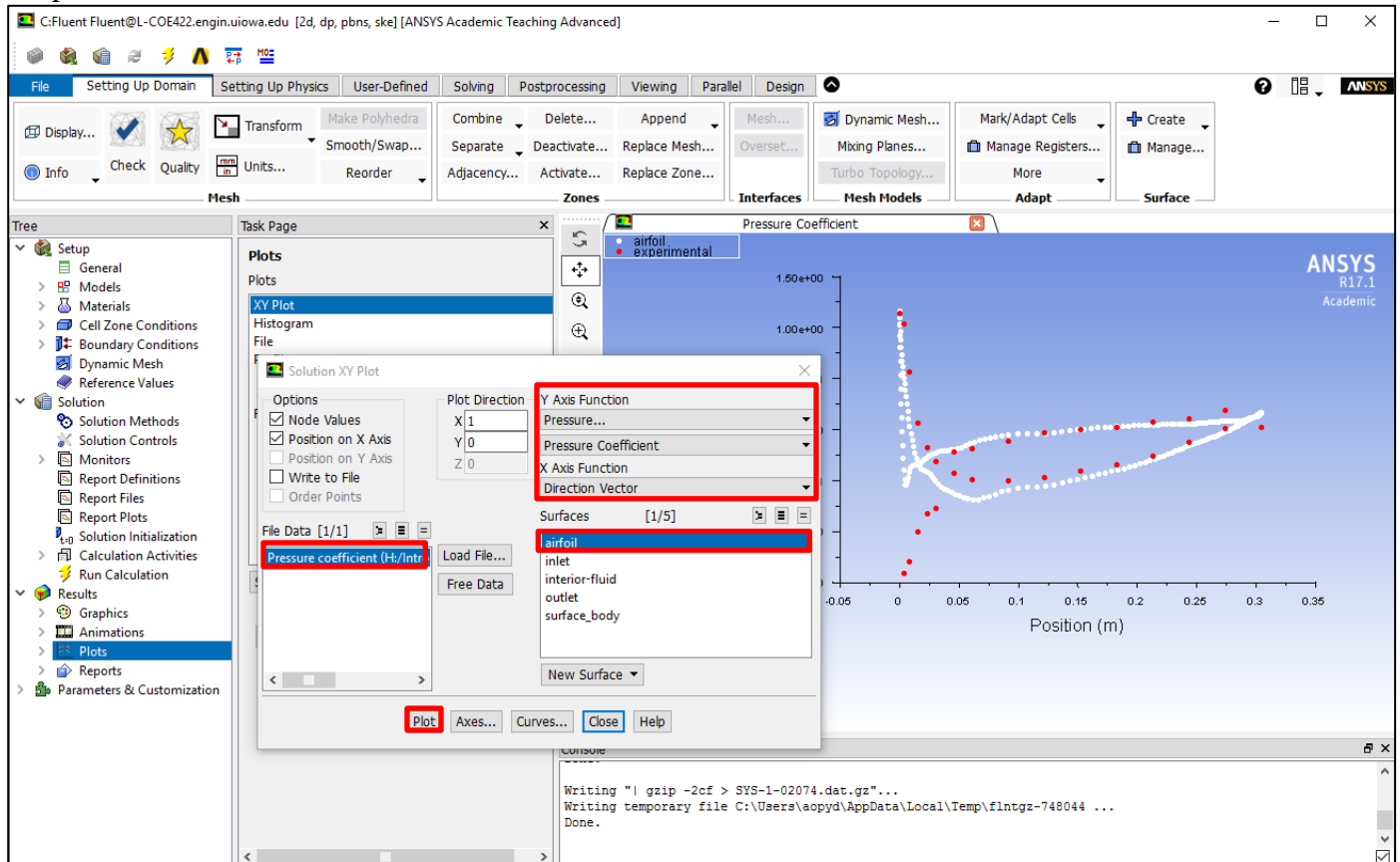
- Residual Values:**  Normalize,  Scale,  Compute Local Scale, Iterations: 5.
- Convergence Criterion:** absolute

Buttons at the bottom of the dialog: **OK**, **Plot**, Renormalize, Cancel, Help.

In the background, the 'Residuals' plot window shows a legend with the following items: continuity (red line), x-velocity (green line), y-velocity (blue line), k (cyan line), and epsilon (magenta line).

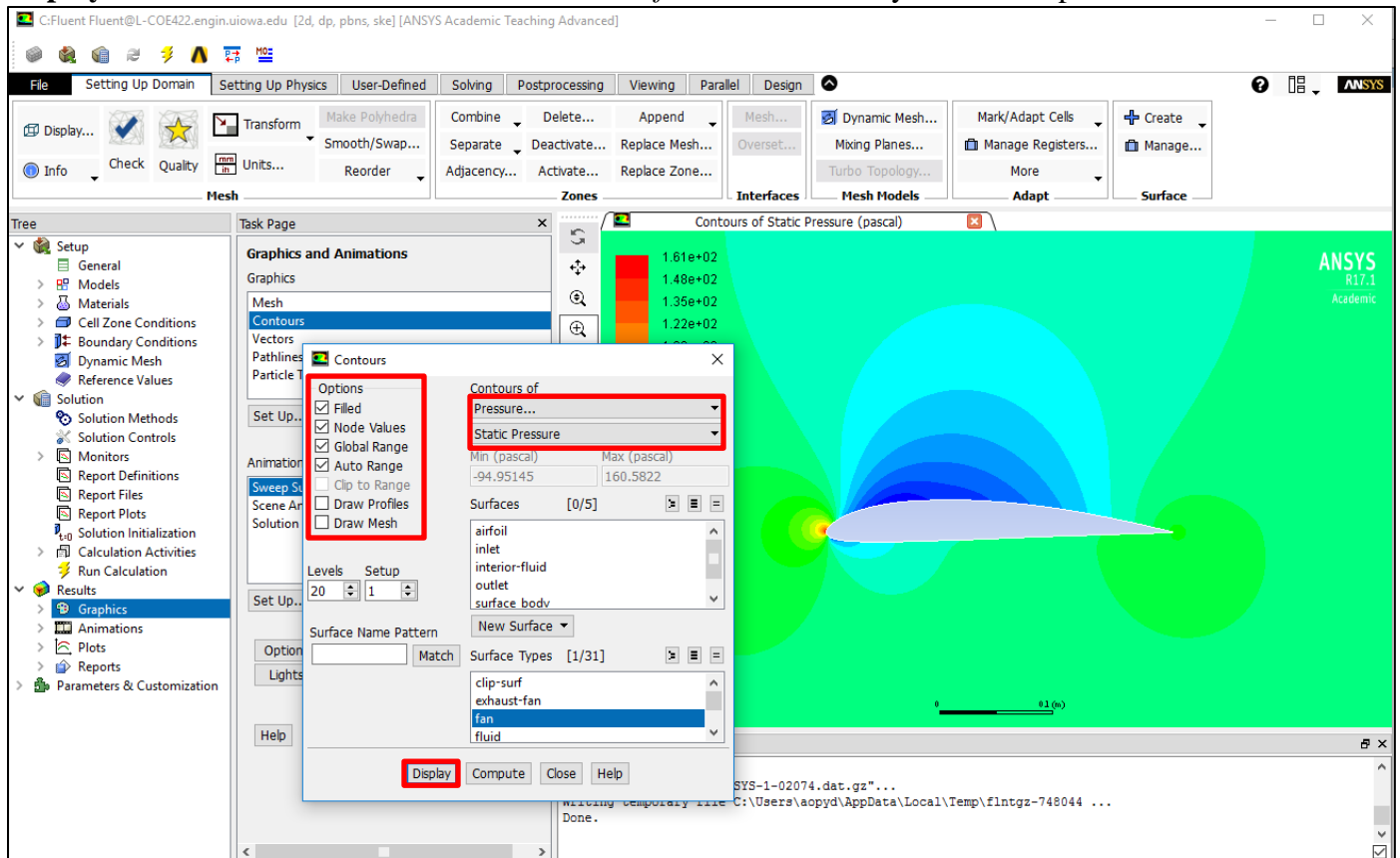
### 7.3. Plotting Pressure Coefficient Distribution with CFD and EFD Data

**Results > Plots > XY Plot > Set Up... > Load File...** Select Pressure-coef-attack0.xy. Change the parameters as per below and click **Plot**. Name the file “*CFD Pre-Lab 2 Pressure Coefficient Distribution*” when you save the picture.



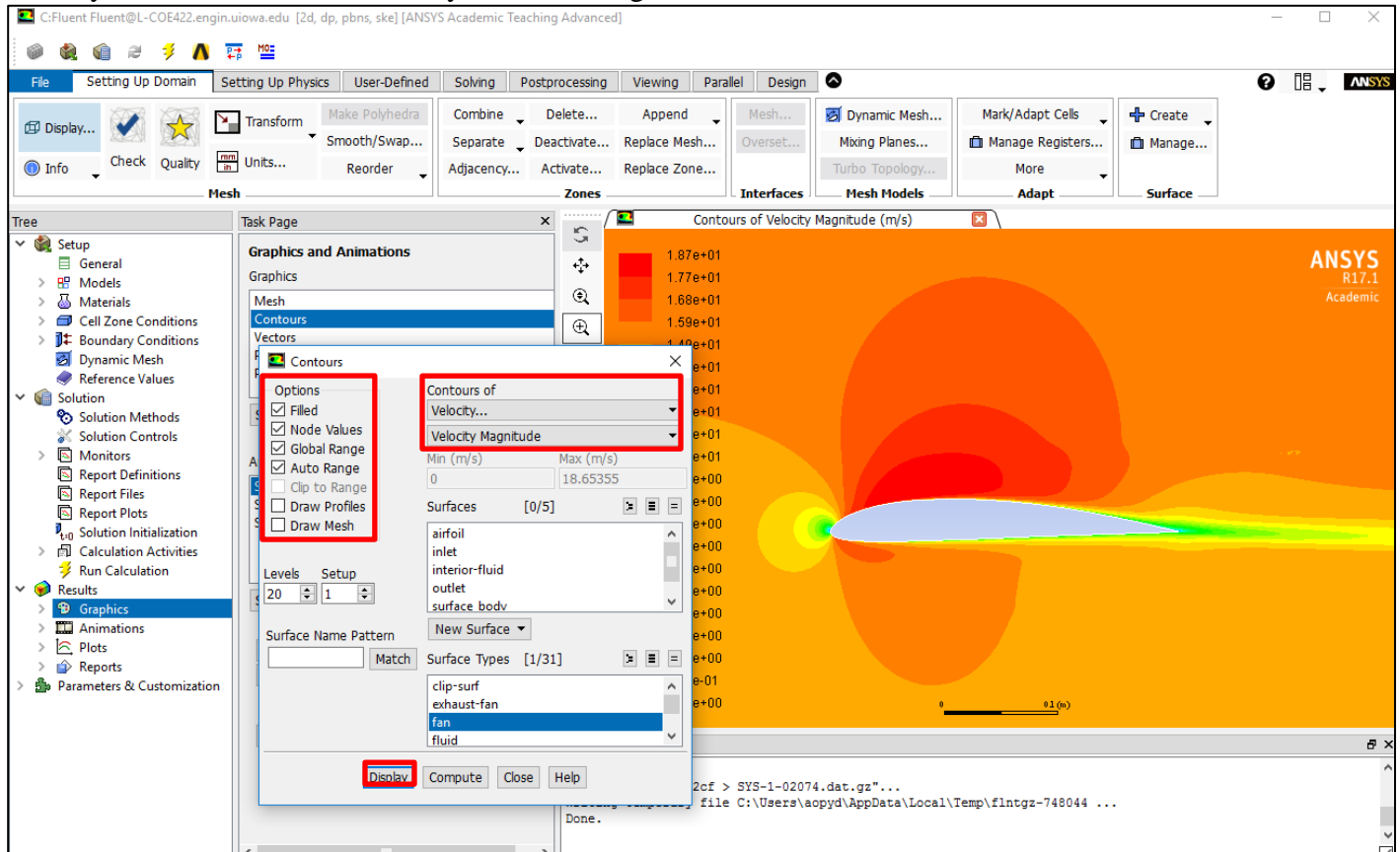
## 7.4. Plotting Contour of Pressure

**Results > Graphics and Animations > Contours > Set Up...** Change the parameters as per below and click **Display**. Name the file “*CFD Pre-Lab 2 Contour of Pressure*” when you save the picture.



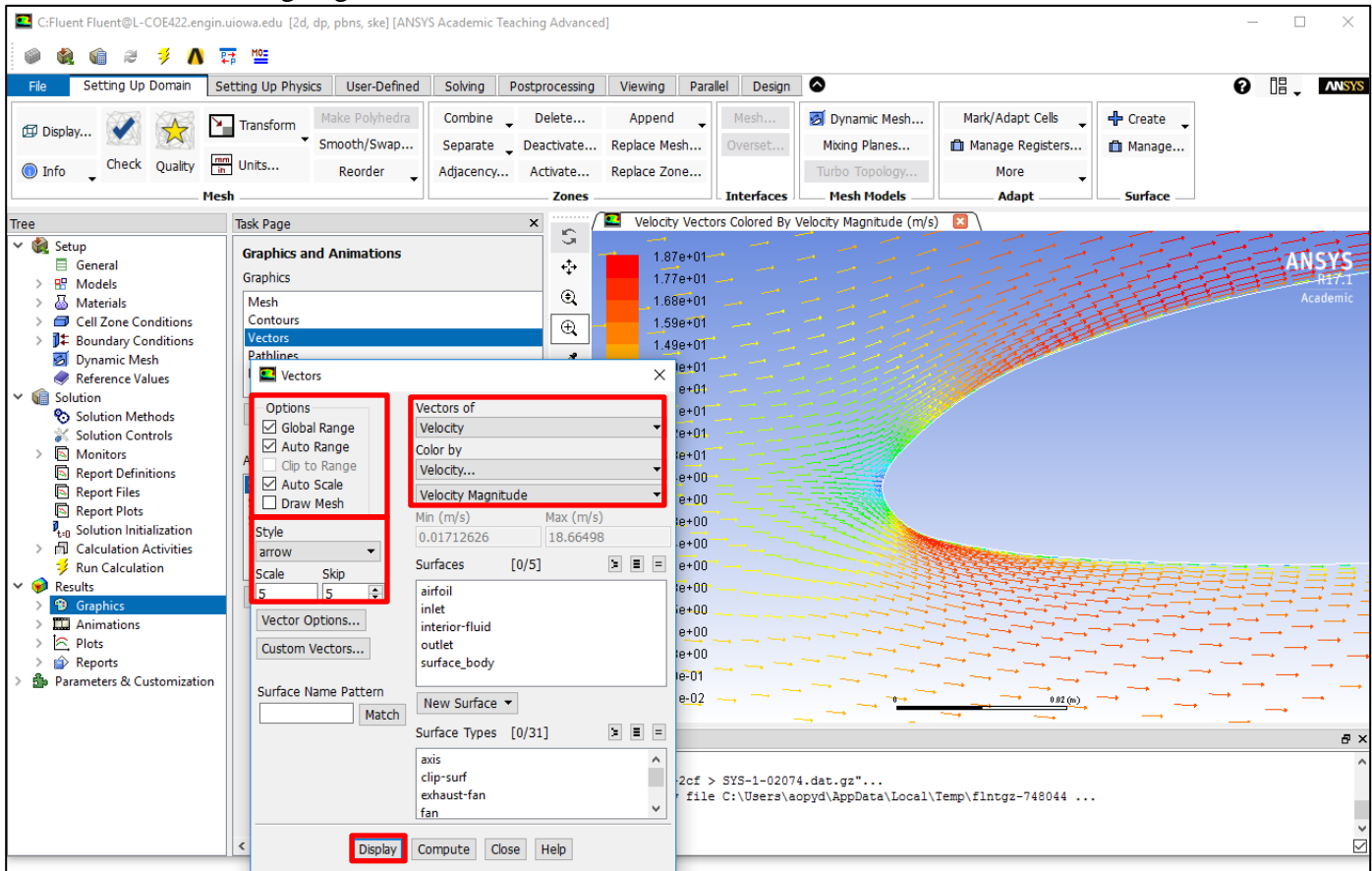
## 7.5. Plotting Contour of Velocity Magnitude

**Results > Graphics and Animations > Contours > Set Up...** Change the parameters as per below and click **Display**. Name the file “*CFD Pre-Lab 2 Contour of Velocity Magnitude*” when you save the picture. Zoom in where you can see the airfoil clearly and the change in contour levels around the airfoil.



## 7.6. Plotting Velocity Vectors at Leading Edge

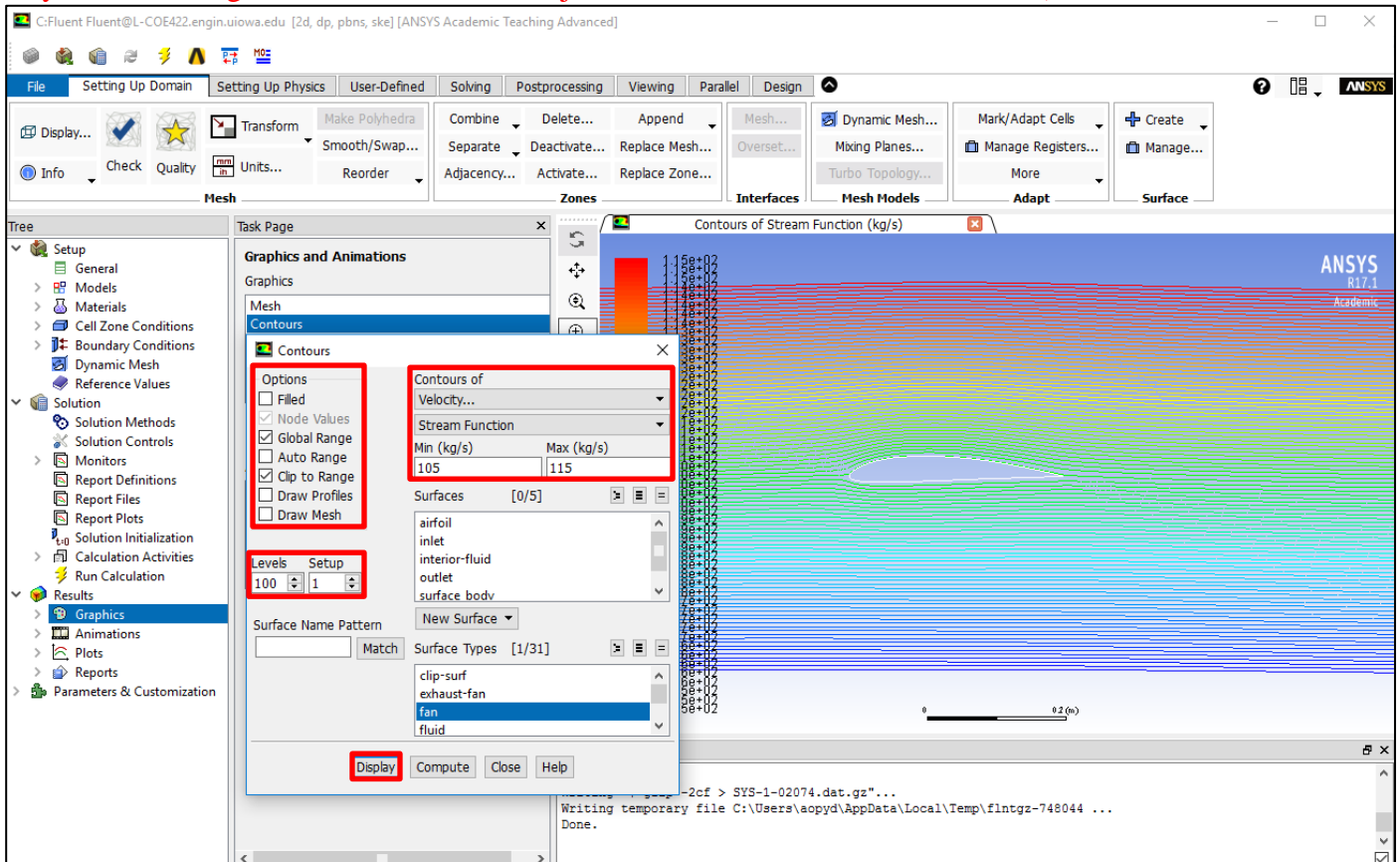
**Results > Graphics and Animations > Vectors > Set Up...** Change parameters as per below and click **Display**. Name the file “*CFD Pre-Lab 2 Vectors of Velocity at Leading Edge*” when you save the picture. Zoom in on the leading edge.





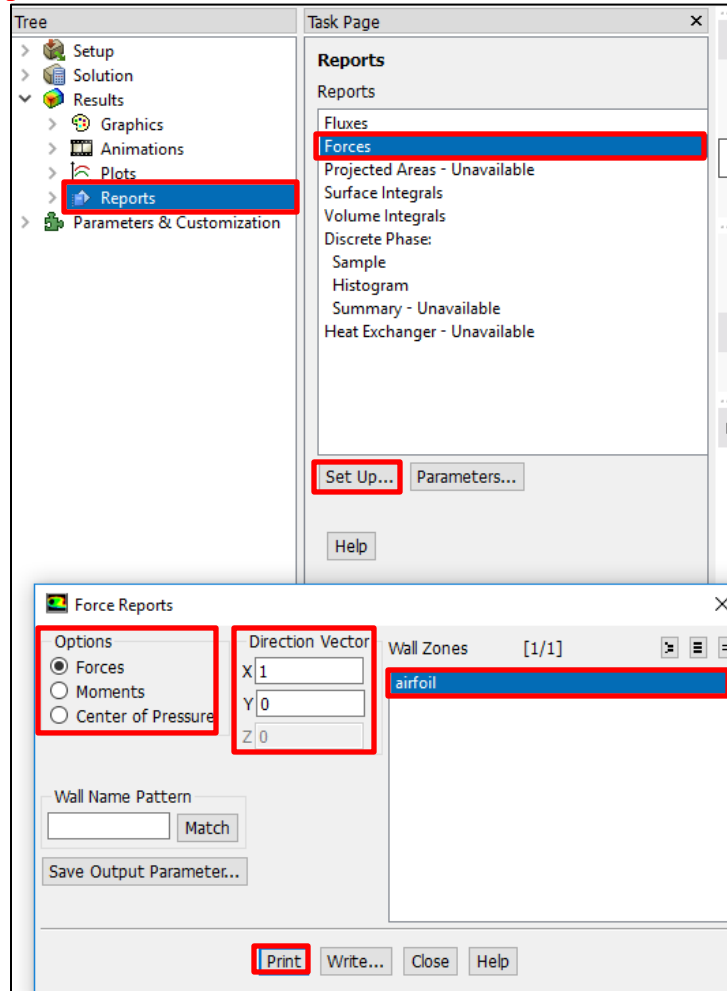
## 7.7. Plotting Streamlines Close to Airfoil Surface

**Results > Graphics and Animations > Contours > Set Up...** Change parameters as per below and click display. Name the file “*CFD Pre-Lab 2 Streamlines Close to Surface*” when you save the picture. (Note: You may need to change the **Min** and **Max** to adjust the streamlines around the airfoil.)



## 7.8. Printing Lift and Drag Coefficients

**Results > Reports > Forces > Set Up...** Change parameters as per below and click **Print**. This prints out the drag force. If you change the X parameter to 0 and the Y parameter to 1, this prints out the lift force. If you change the X parameter to 1 and the Y parameter to 0, this prints out the drag force. **Save these values on to the CFD Lab 2 report template.**



Forces			Coefficients			
Zone	Forces (n) Pressure	Viscous	Total	Pressure	Viscous	Total
airfoil	(0.40419517 13.379949 0)	(0.52802802 0.016180924 0)	(0.93222319 13.39613 0)	(0.0096224917 0.3185304 0)		
Net	(0.40419517 13.379949 0)	(0.52802802 0.016180924 0)	(0.93222319 13.39613 0)	(0.0096224917 0.3185304 0)		
<b>Forces - Direction Vector (1 0 0)</b>						
Zone	Forces (n) Pressure	Viscous	Total	Pressure	Viscous	Total
airfoil	0.40419517	0.52802802	0.93222319	0.0096224917	0.012570524	0.022193016
Net	0.40419517	0.52802802	0.93222319	0.0096224917	0.012570524	0.022193016

## 8. Exercises

### Simulation of Turbulent Flow around an Airfoil

You must complete all the following assignments and present results in your CFD Lab 2 reports following the CFD Lab Report Instructions.

#### 8.1. Validation using EFD Lab 3 data

(1). Use the same flow conditions as those in your EFD Lab 3, including **geometry** (chord length, angle of attack 0) and **Setup** (Flow properties, inlet velocity). Use **k-e model, 2<sup>nd</sup> order scheme, double precision** with **iteration number (10000)** and **convergent limit (1e-05)**. Run the simulation.

(2). Modify your EFD data of pressure coefficient in ANSYS format (sample EFD data format has been provided in CFD Lab 1) and import it into ANSYS Fluent for pressure coefficient distribution and conduct validation. Also compare the CFD lift coefficient value with EFD data.

- **Figures need to be saved:** (1) Time history of residuals (residual vs. iteration number); (2) Pressure coefficient distribution (CFD and EFD); (3) Contour of pressure; (4) Contour of velocity magnitude; (5) Velocity vectors (show the region that is interesting, such as the separation region when angle of attack is large or close to the leading edge of the airfoil) and (6) Streamlines close to airfoil surface.
- **Data need to be saved:** lift and drag coefficients

#### 8.2. Inviscid flow simulation

Use the same conditions as those in exercise 1, except choose “**inviscid**” for “viscous model”, set the iteration number to be (10000), and convergent limit to be  $10^{-5}$ . Conduct the simulation and compare solutions with viscous flow results in exercise 8.1.

- **Figures need to be saved:** (1) Time history of residuals; (2) Pressure coefficient distribution (CFD only); (3) Contour of pressure; (4) Contour of velocity magnitude; (5) Velocity vectors (show the same region that your picked in exercise 8.1) and (6) Streamlines close to airfoil surface.
- **Data need to be saved:** lift and drag coefficients

#### 8.3. Questions need to be answered in CFD Lab 2 report:

- 8.3.1 Does inviscid flow has boundary layer near the wall? Zoom in the near wall region and describe the differences of velocity vectors near the airfoil surface for inviscid and viscous flows.
- 8.3.2 What are the correct boundary conditions for velocity and pressure at “inlet” and “outlet”.
- 8.3.3 What are the values for lift and drag coefficients for inviscid flow around the airfoil? Are they both zero?
- 8.3.4 Where are the highest and lowest locations for pressure and velocity magnitude? Why? Is pressure constant for inviscid flows around airfoil?
- 8.3.5 For turbulent flow around airfoil, try to qualitatively explain why there is a lift force (vertical up) on an airfoil using the contour plot of pressure or the XY plot of pressure coefficient distribution.