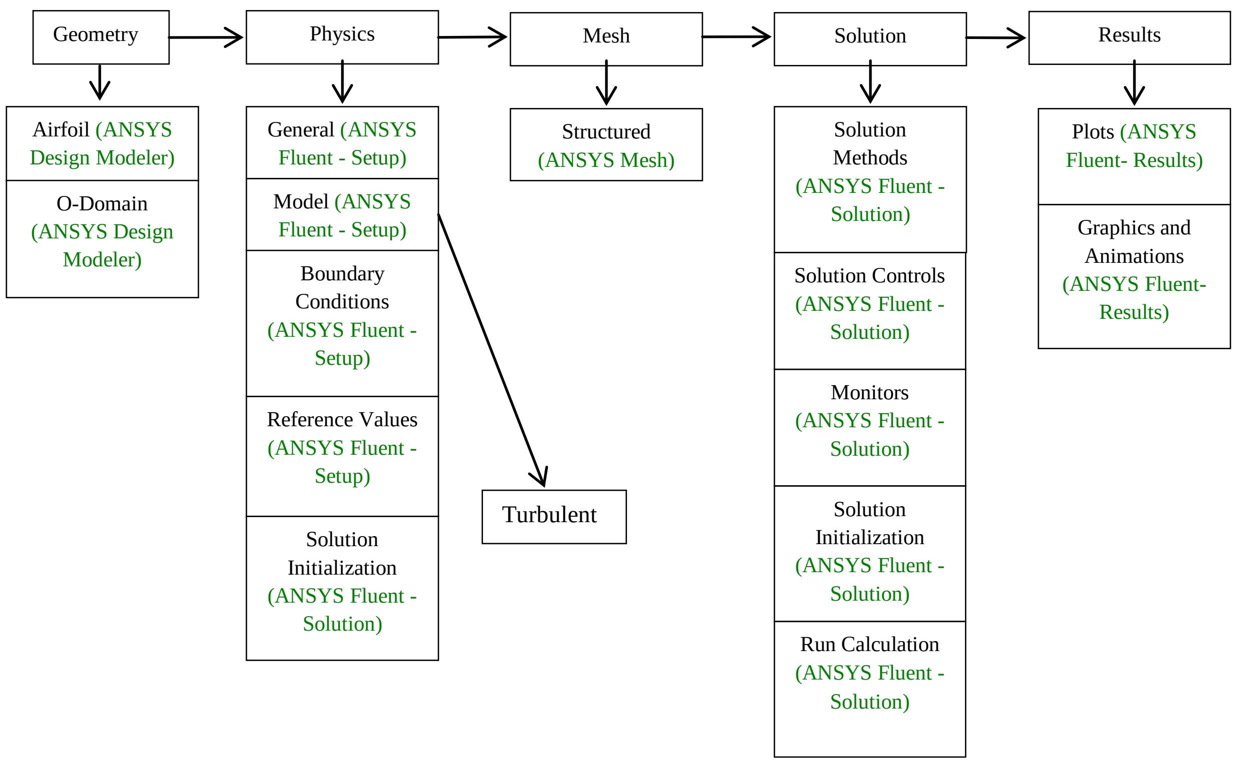
Simulation of Turbulent Flow around an Airfoil

57:020 Mechanics of Fluids and Transfer Processes
CFD Pre-Lab 2

By Timur Dogan, Michael Conger, Maysam Mousaviraad, Tao Xing and Fred Stern
IIHR-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 Clarky 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 ClarkY 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 ClarkY airfoil with angle of attack (α)

Table 1 - Geometry dimensions

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

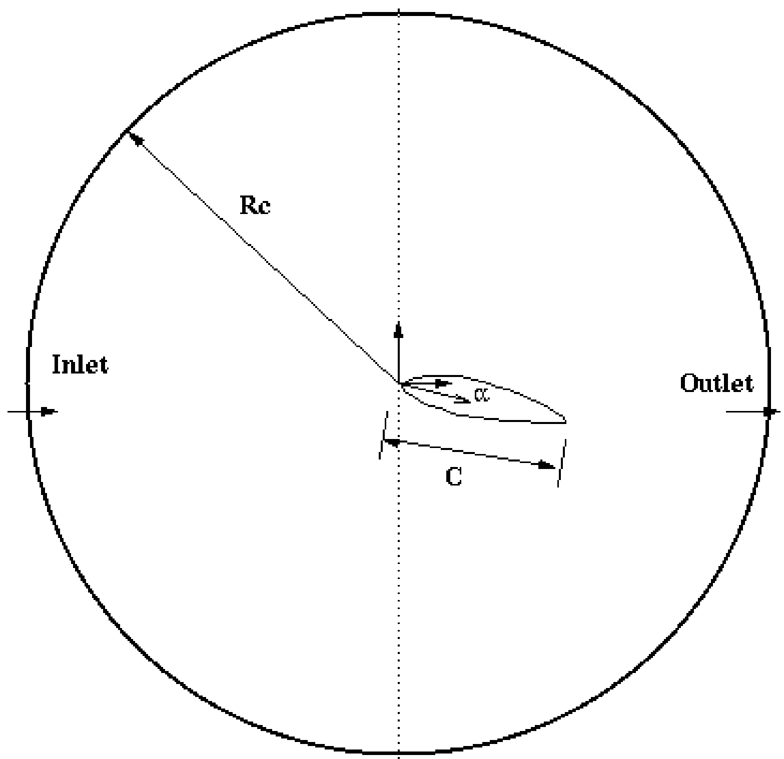
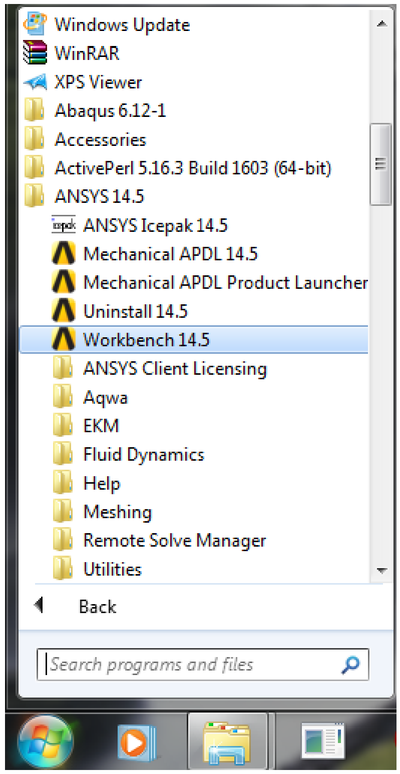


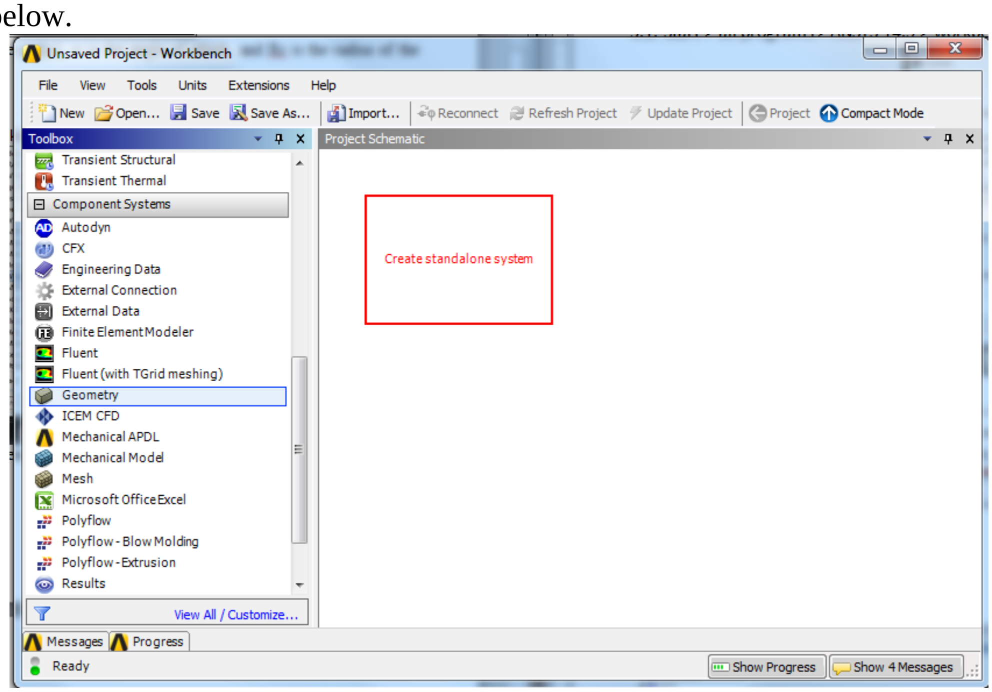
Figure 1 – Geometry

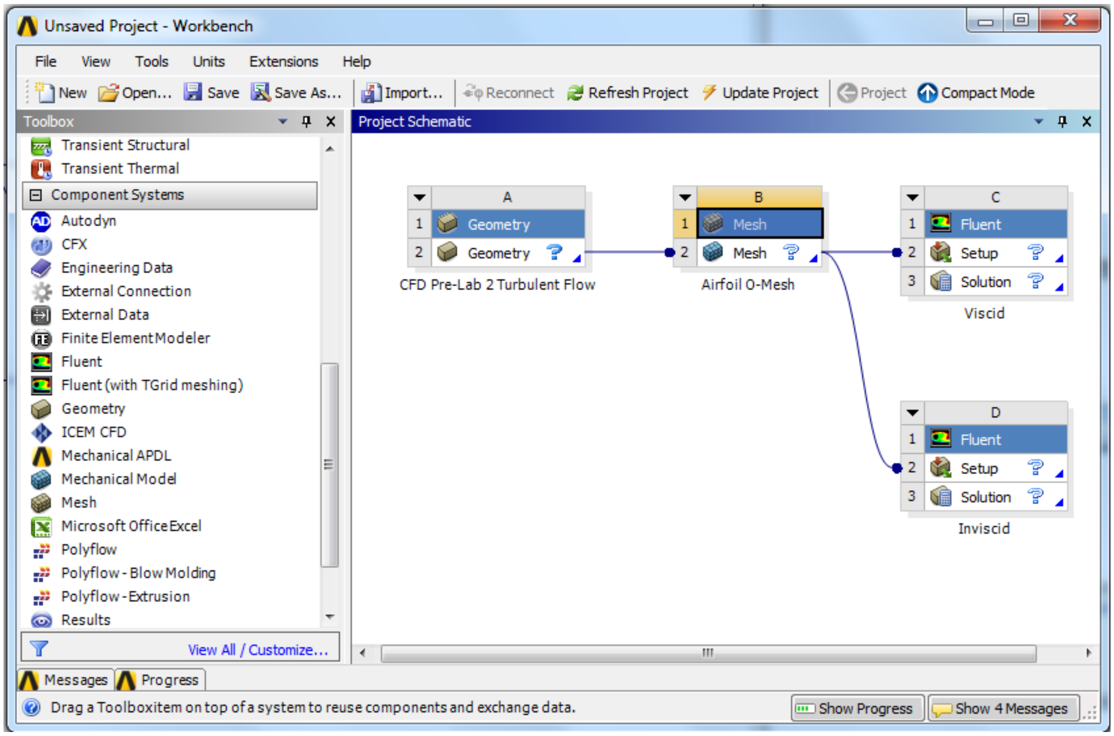
3. Open ANSYS Workbench

3.1. **Start** > **All Programs** > **ANSYS 14.5** > **Workbench 14.5**



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.

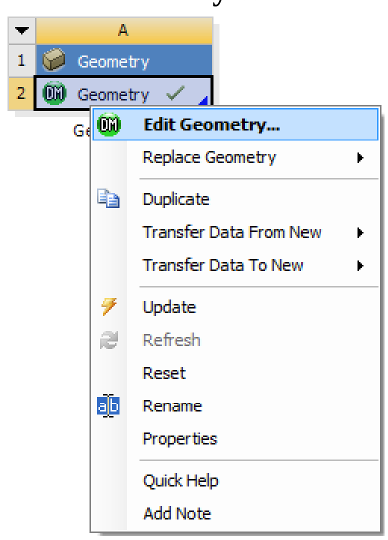




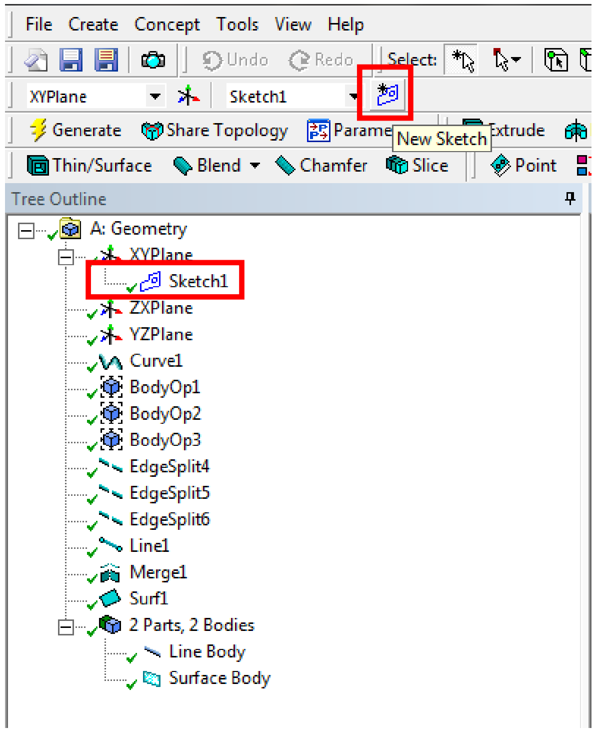
3.3. Create a Folder on the H: Drive called *CFD Pre-Lab 2*.
3.4. Save the project file by clicking **File** > **Save As…**
3.5. Save the project onto the H: Drive in the folder you just created and name it *CFD Pre-Lab 2 Turbulent Flow*.

4. Geometry

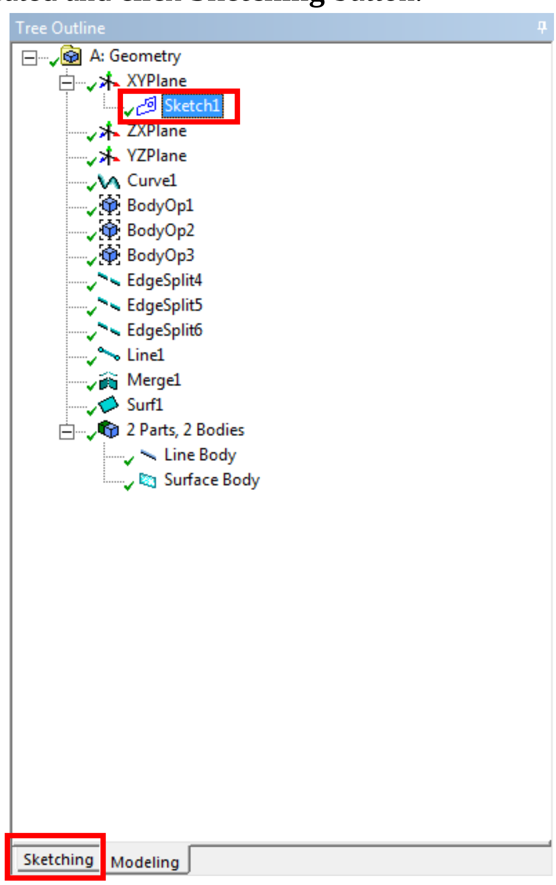
4.1. Right click **Geometry** then select **Import Geometry** > **Browse…** Select **intro_airfoil.igs** and click **OK**.
4.2. Right click **Geometry** and select **Edit Geometry…**



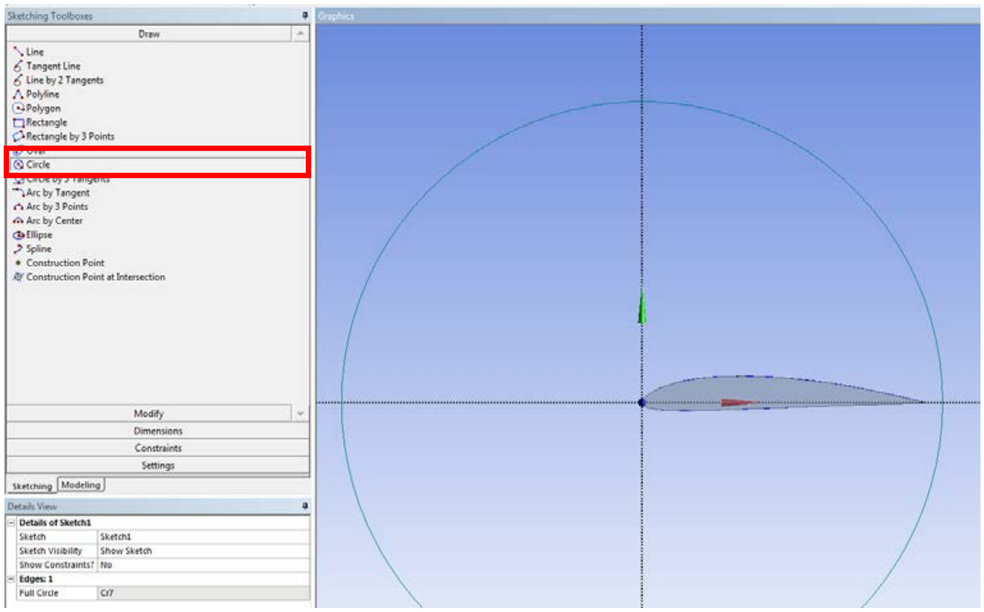
4.3. Select **XYPlane** and click **New Sketch** button.



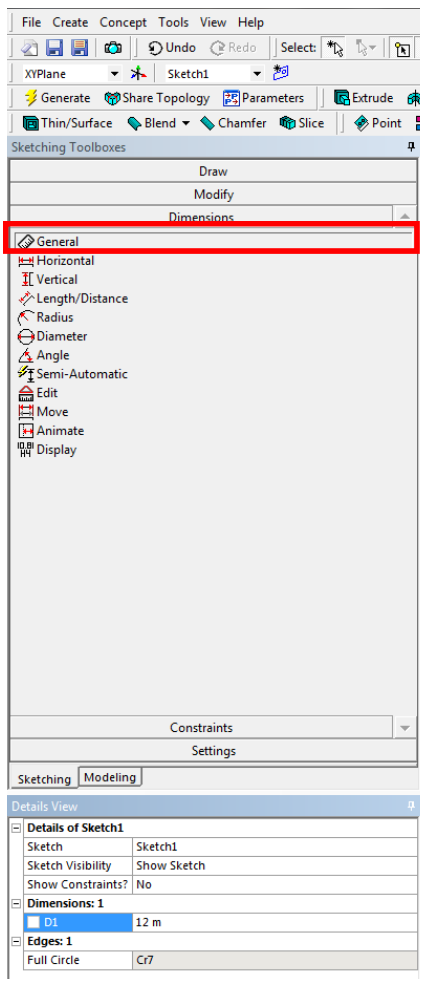
4.4. Select the sketch you created and click **Sketching** button.



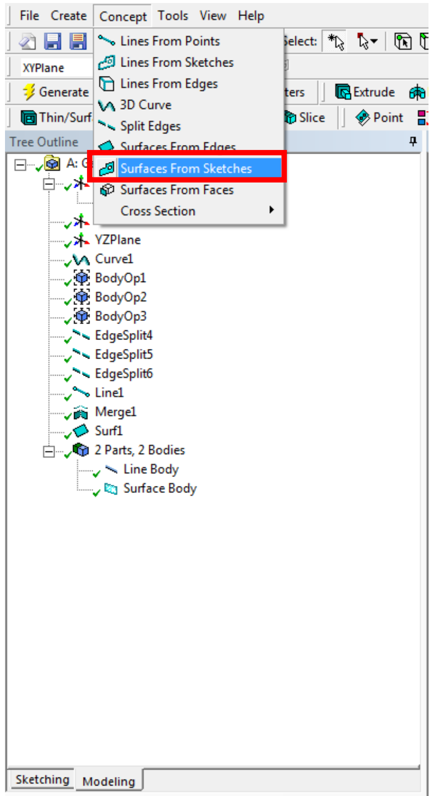
4.5. **Sketching Toolbox** > **Draw** > **Circle**. Click on the xy-plane origin and click behind the airfoil.



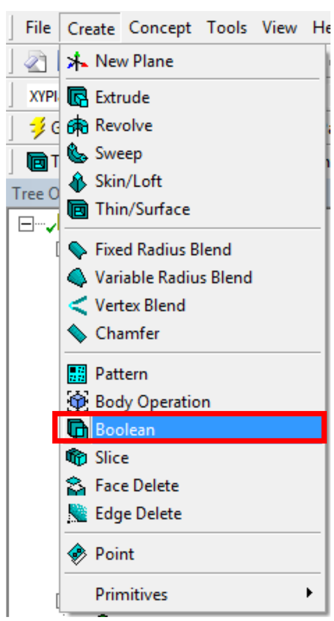
4.6. **Sketching Toolboxes** > **Dimensions** > **General**. Click on the circle and change the diameter to 12m.

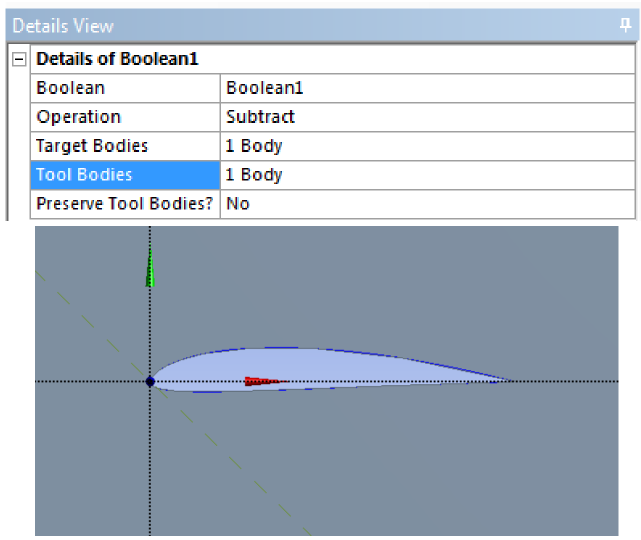


4.7. **Concept** > **Surface From Sketches**. Select your sketch and click the **Generate** button.

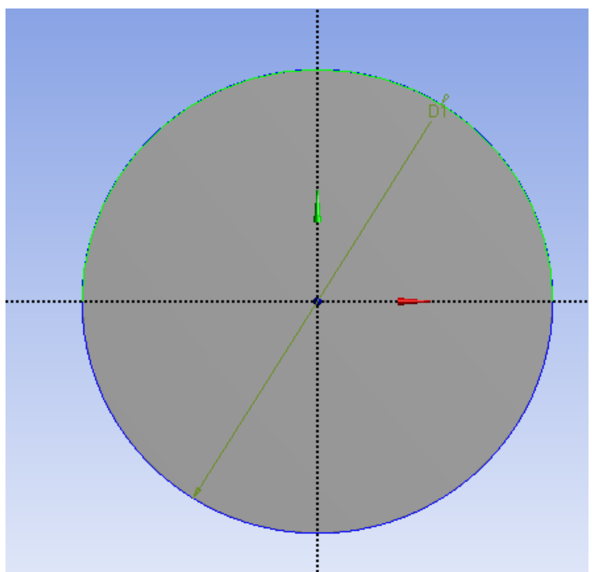


4.8. **Create** > **Boolean**. Change operation to **Subtract** then select the circle for **Target Bodies** then select the airfoil for **Tool Bodies** and click **Generate**. This will subtract the airfoil surface from the circle.



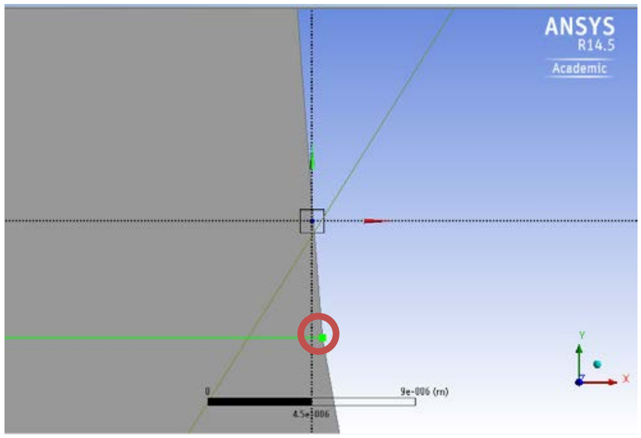


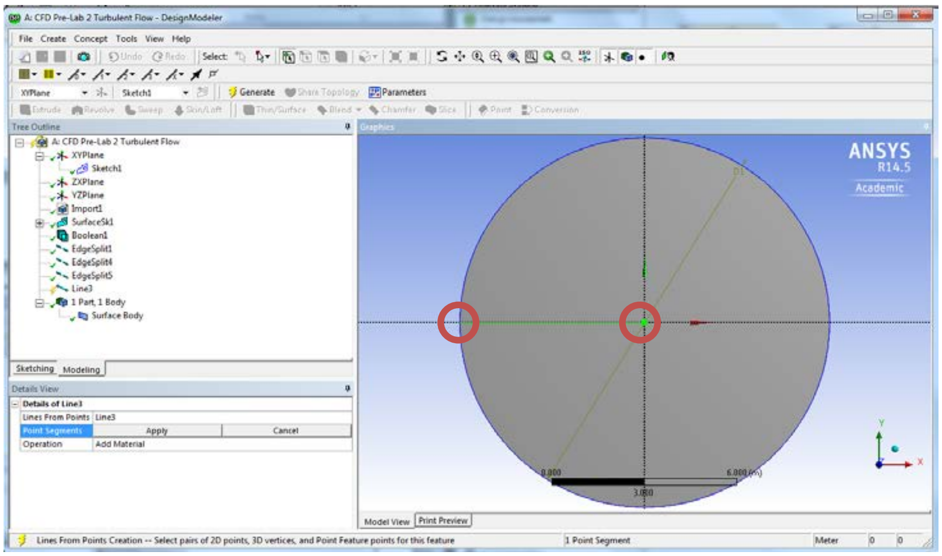
4.9. **Concept** > **Split Edges.** Select the perimeter of the circle and click **Apply.** Select **Generate.** This should split the circle into two semicircles. You can see the semicircles by selecting the perimeter above and below the x-axis.



4.10. Repeat the process from 4.9 on the two semicircles. This should yield four circular quadrants.

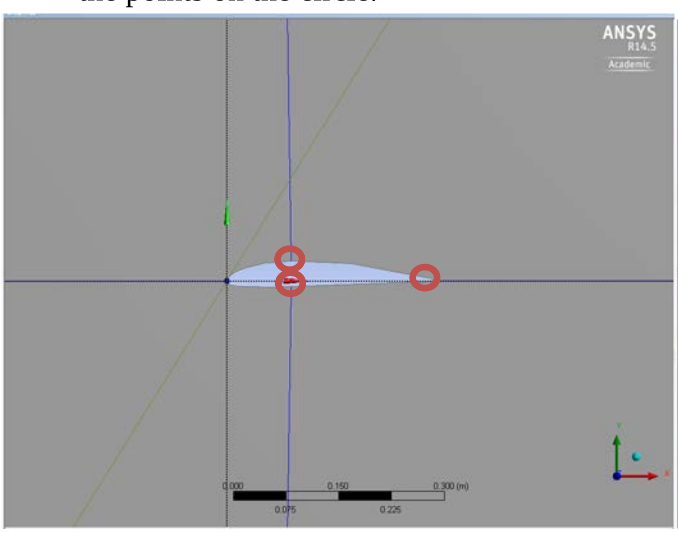
4.11. **Concept** > **Lines From Points**. Draw a line from the point on the circle to the point on the airfoil making sure to hold **Ctrl** while doing so. (Note: The point on the airfoil is not exactly on the origin. Zoom in and find the point just below the origin and select that point. The images 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).

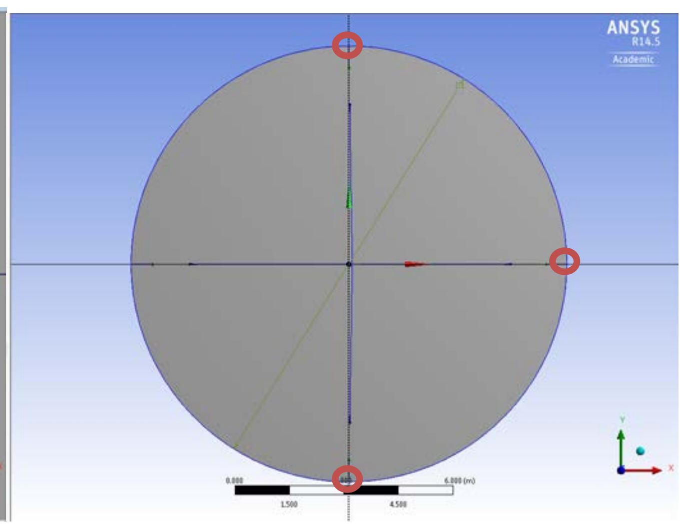




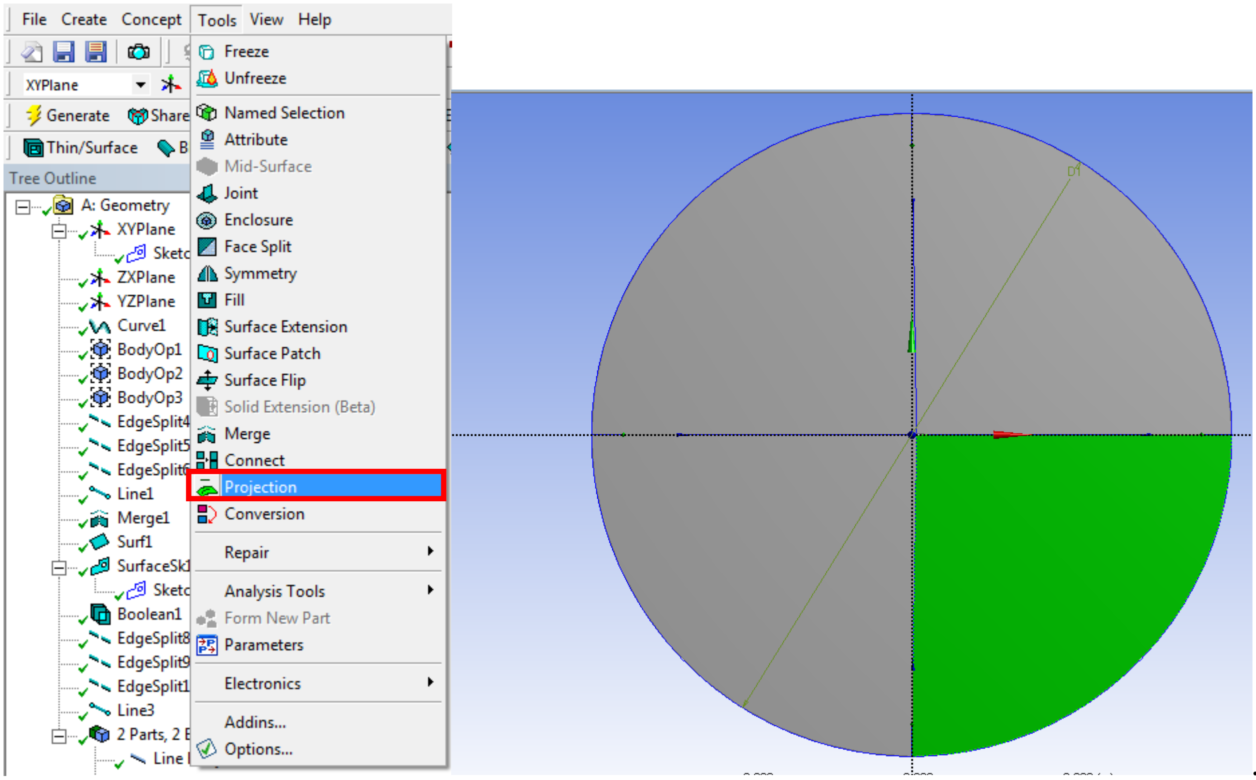
4.12. Once you select both points hit **Apply.** Then click **Generate.**

4.13. Repeat this process creating lines from the edge of the circle to the airfoil starting from the circle and ending at the airfoil. The images below show the locations of the points on the airfoil and the points on the circle.





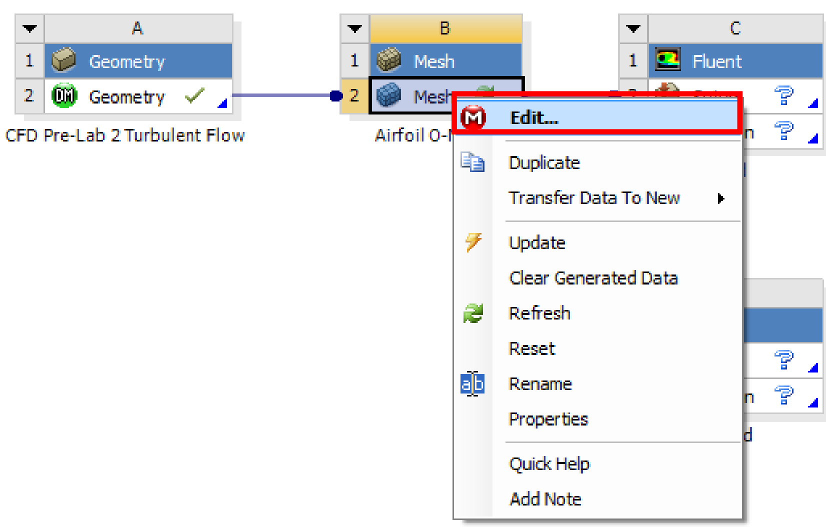
4.14. **Tools** > **Projections**. Select the four edges you created for **Edges** and click **Apply** and select the circle for **Target** and click **Apply**, then click **Generate**. This will split your geometry into four sections.



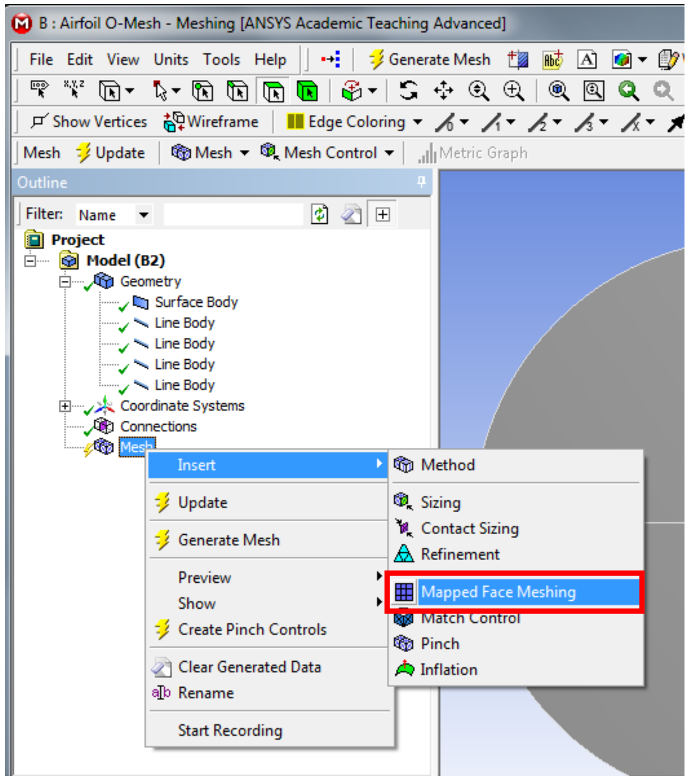
4.15. **File** > **Save project** and exit.

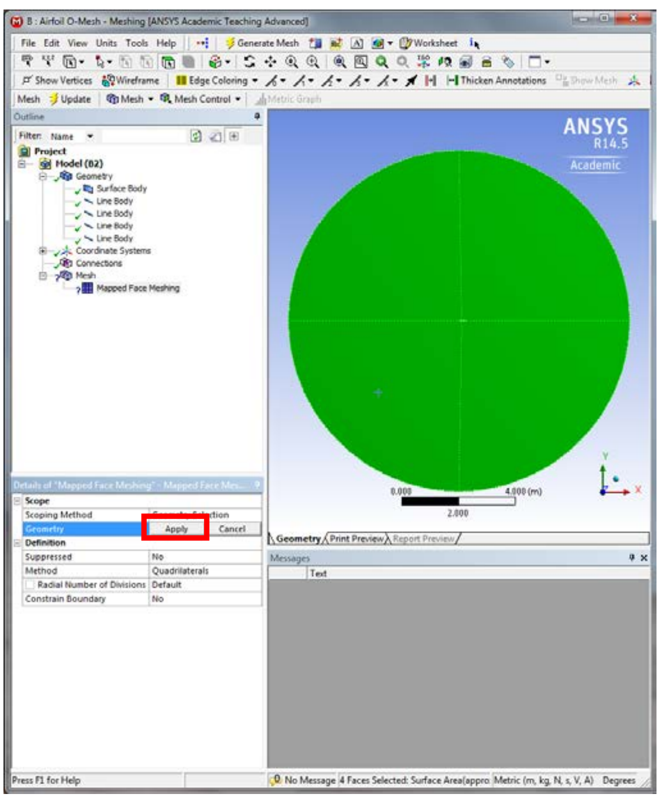
5. Mesh Generation

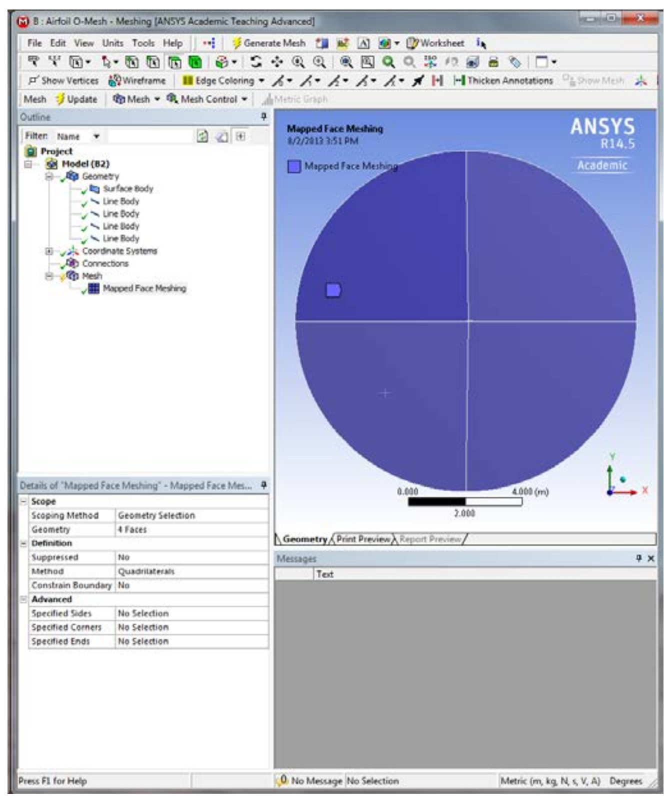
5.1. From Workbench home screen right click on **Mesh** and select **Edit…** from the dropdown menu.



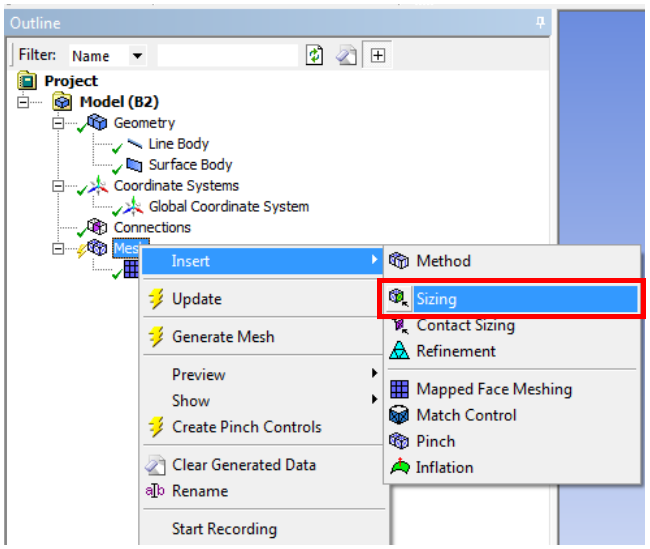
5.2. Right click **Mesh** then **Insert** > **Mapped Face Meshing**. Select the four surfaces then click **Apply**.

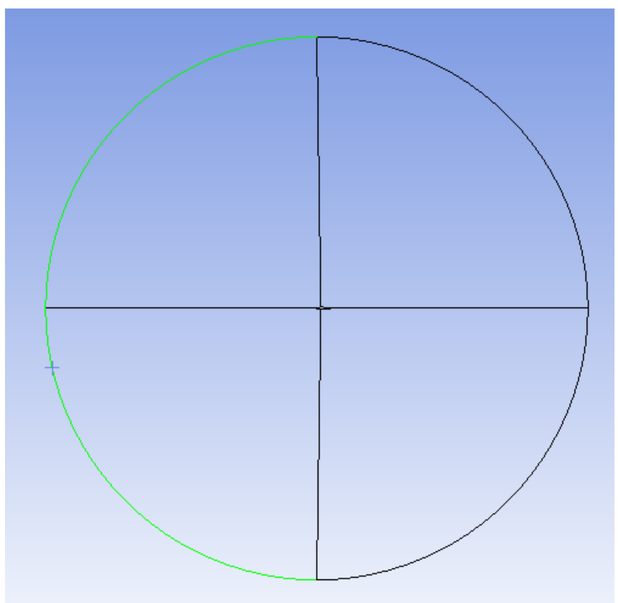


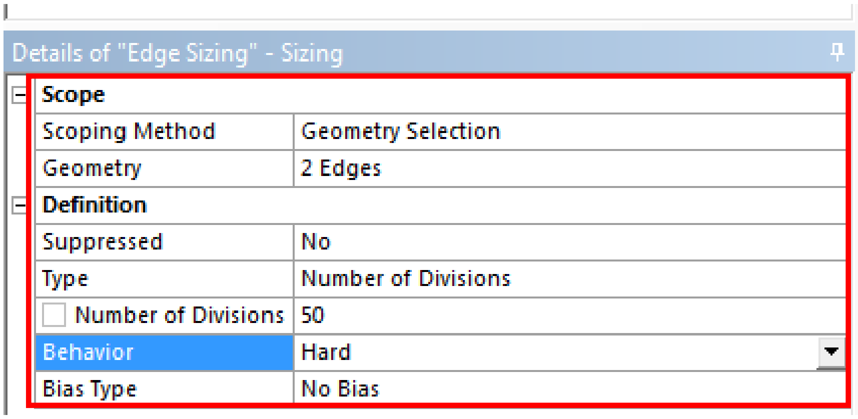




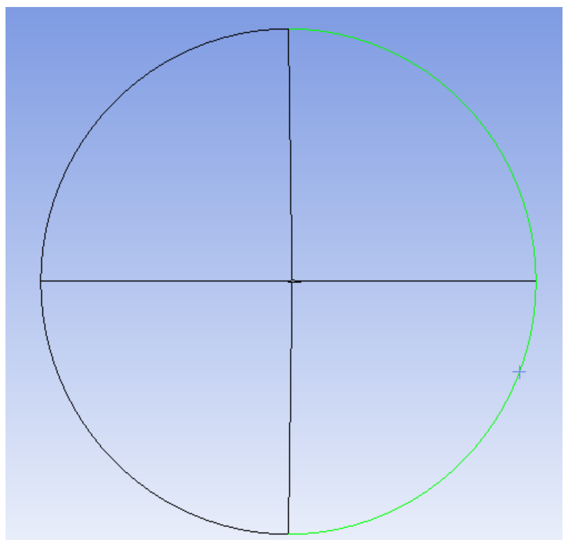
5.3. Right click **Mesh** and **Insert** > **Sizing**. Select two edges as per below and change the parameters as per below.

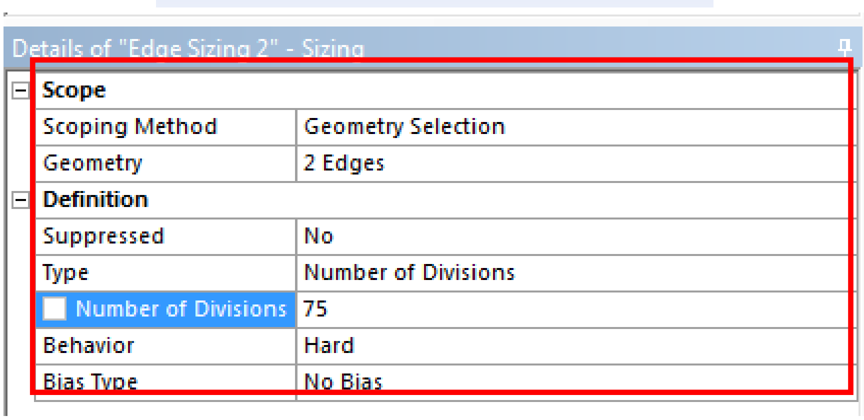




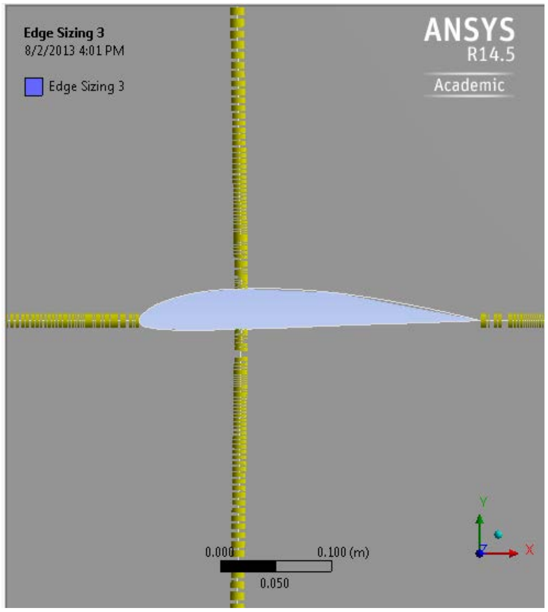


5.4. Right click **Mesh** and **Insert** > **Sizing**. Select two edges as per below and change the parameters as per below.





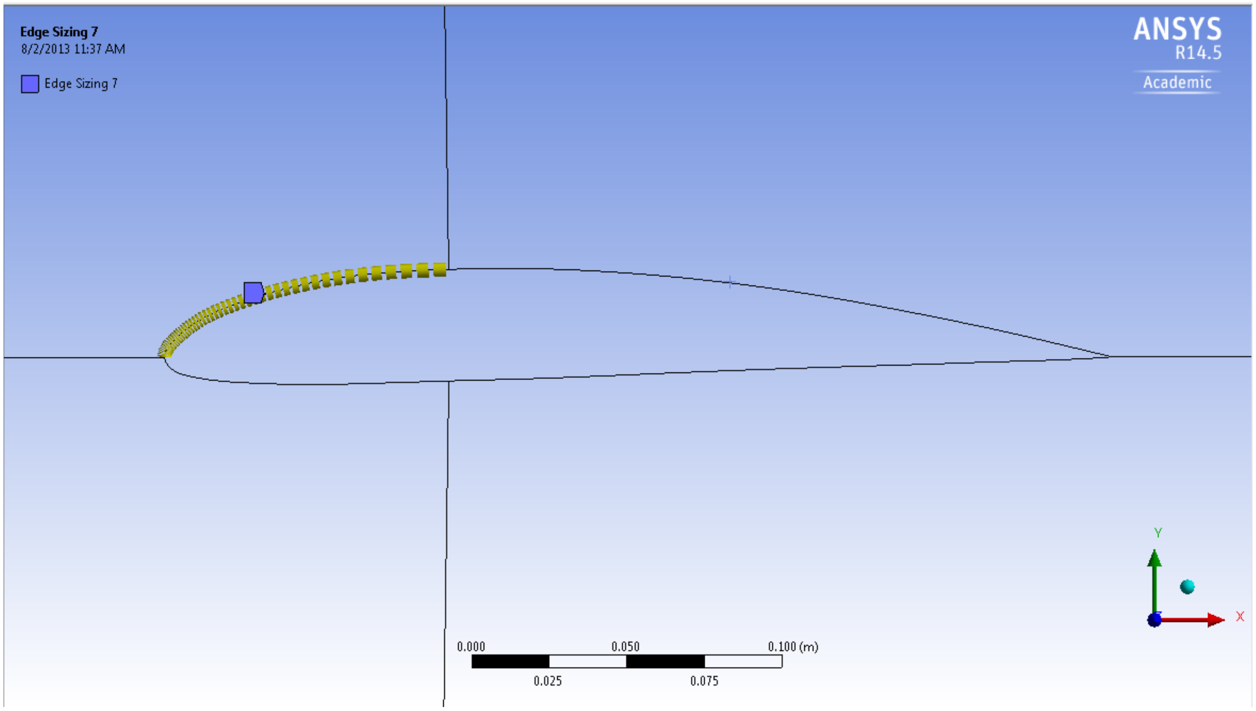
5.5. Right click **Mesh** and **Insert** > **Sizing**. Select all for lines leading from the circle to the airfoil surface, and click **Apply**. Change parameters as per below. (Note: If you did not create the lines starting from the outer circle and ending on the airfoil surface, you may have issues 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

Scope	
Scoping Method	Geometry Selection
Geometry	4 Edges
Definition	
Suppressed	No
Type	Number of Divisions
Number of Divisions	200
Behavior	Hard
Bias Type	----- --- - -
Bias Option	Bias Factor
Bias Factor	5000.

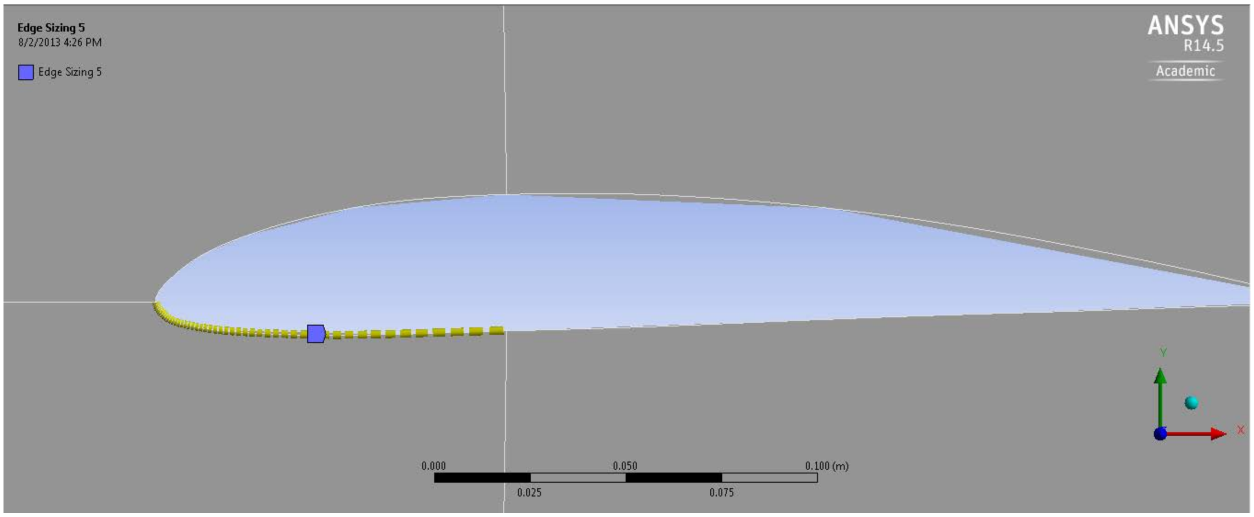
5.6. Right click **Mesh** and **Insert** > **Sizing.** Select the surface at the top of leading edge of the airfoil and click **Apply**. Change Parameters as per below.



Details of "Edge Sizing 7" - Sizing

Scope	
Scoping Method	Geometry Selection
Geometry	1 Edge
Definition	
Suppressed	No
Type	Number of Divisions
Number of Divisions	50
Behavior	Hard
Bias Type	----- --- - -
Bias Option	Bias Factor
Bias Factor	15.

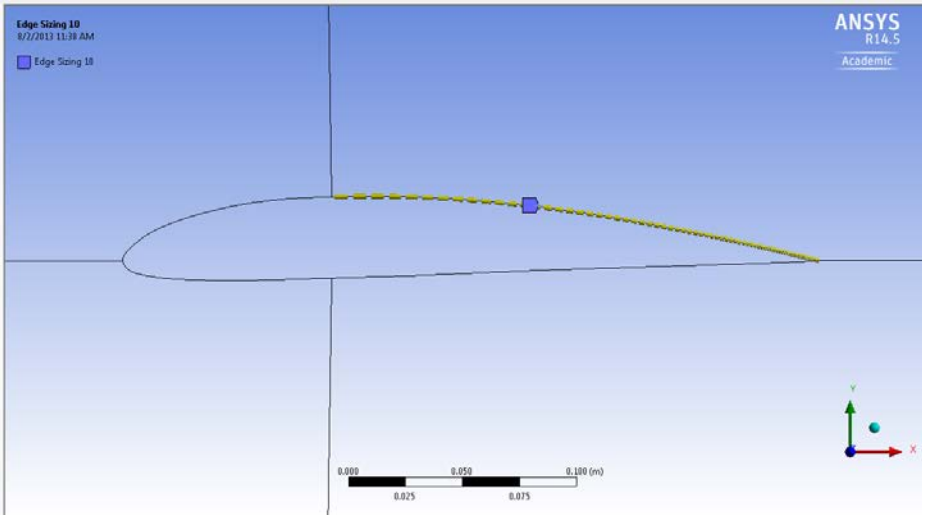
5.7. Right click **Mesh** and **Insert** > **Sizing.** Select the surface at the bottom of leading edge of the airfoil and click **Apply**. Change Parameters as per below.



Details of "Edge Sizing 5" - Sizing

Scope	
Scoping Method	Geometry Selection
Geometry	1 Edge
Definition	
Suppressed	No
Type	Number of Divisions
Number of Divisions	50
Behavior	Hard
Bias Type	_ _ ___ _____
Bias Option	Bias Factor
Bias Factor	15.

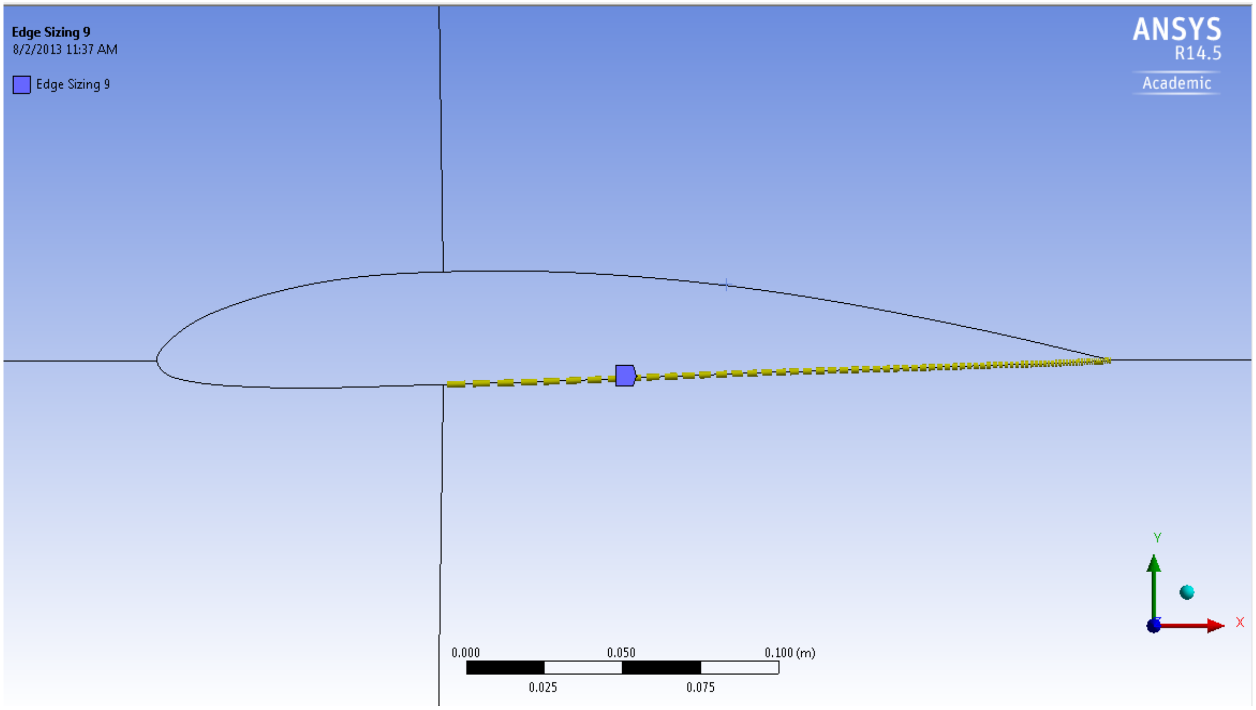
5.8. Right click **Mesh** and **Insert** > **Sizing.** Select the surface at the top of trailing edge of the airfoil and click **Apply**. Change Parameters as per below.



Details of "Edge Sizing 10" - Sizing

Scope	
Scoping Method	Geometry Selection
Geometry	1 Edge
Definition	
Suppressed	No
Type	Number of Divisions
Number of Divisions	75
Behavior	Hard
Bias Type	_ _ ___ _____
Bias Option	Bias Factor
Bias Factor	15.

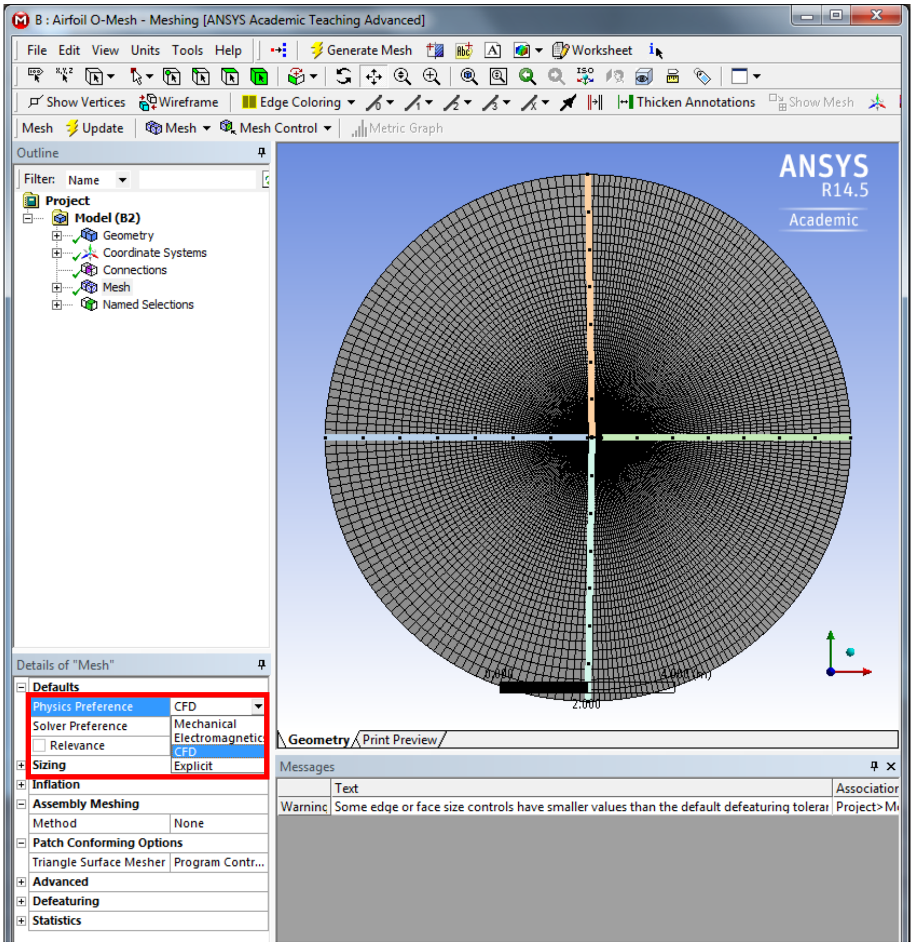
5.9. Right click **Mesh** and **Insert** > **Sizing.** Select the surface at the bottom of trailing edge of the airfoil and click **Apply**. Change Parameters as per below.



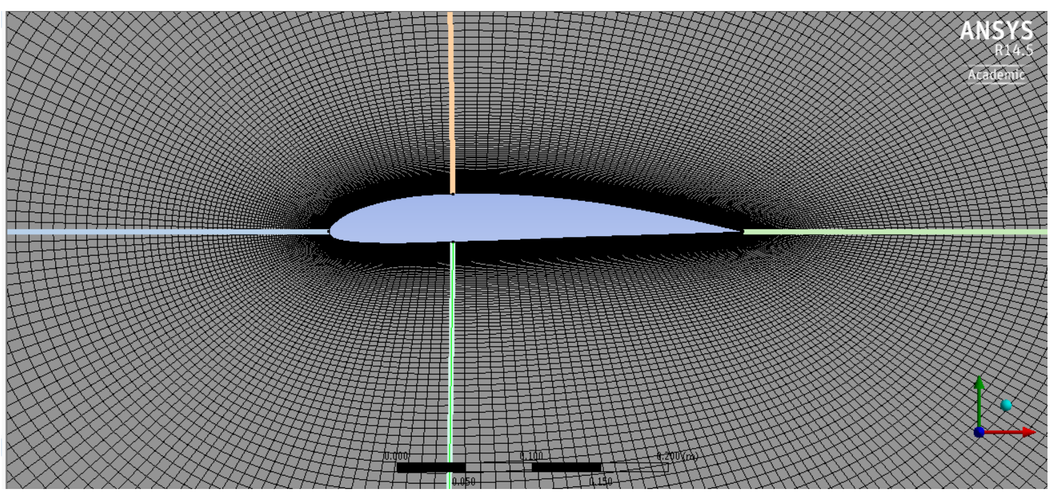
Details of "Edge Sizing 9" - Sizing

Scope	
Scoping Method	Geometry Selection
Geometry	1 Edge
Definition	
Suppressed	No
Type	Number of Divisions
Number of Divisions	75
Behavior	Hard
Bias Type	----- --- - -
Bias Option	Bias Factor
Bias Factor	15.

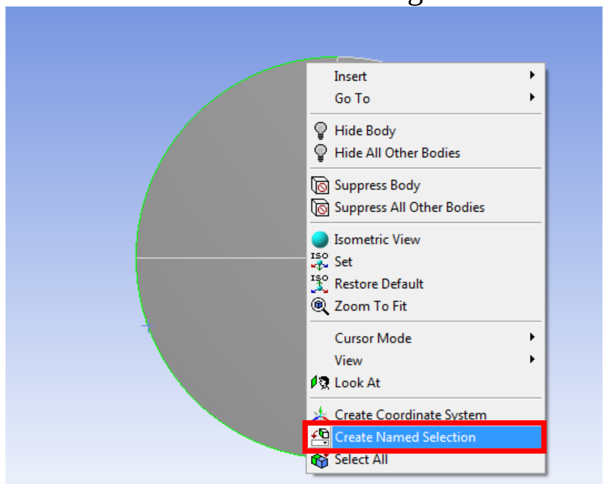
5.10. Click on **Physics Preference** under the **Details of "Mesh"**. Change the mesh type to **CFD** from **Mechanical**.



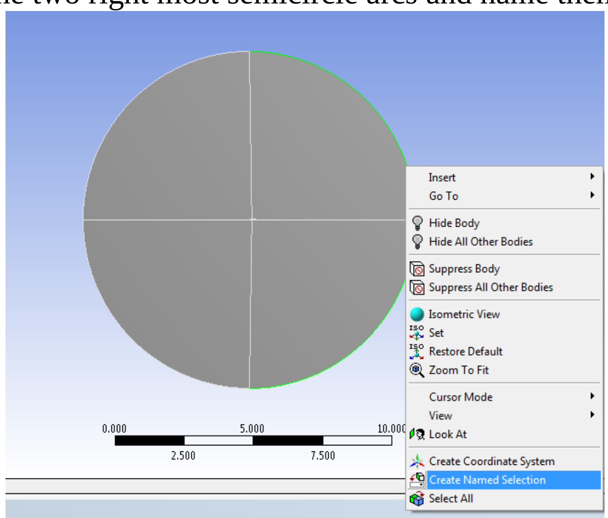
5.11. Click **Generate Mesh**. Click on the **Mesh** button under the **Outline** and make sure it resembles the mesh below.



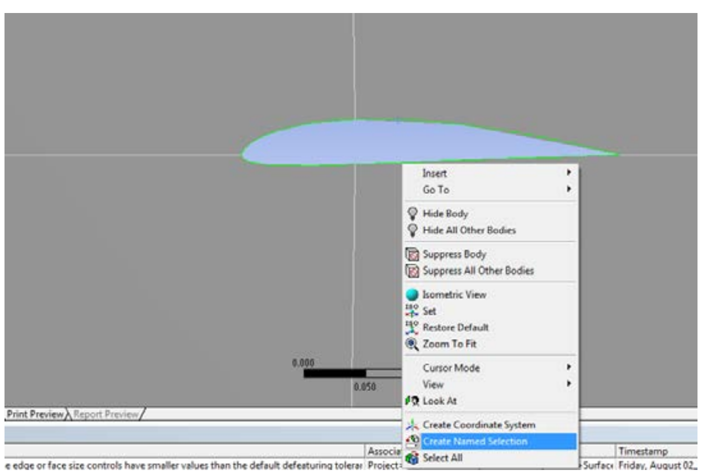
5.12. Hold **Ctrl** and select the two left most semicircle arcs, right click on them and select **Create Named Selection**, name the selection *inlet*. Use the edge select button from the toolbar.



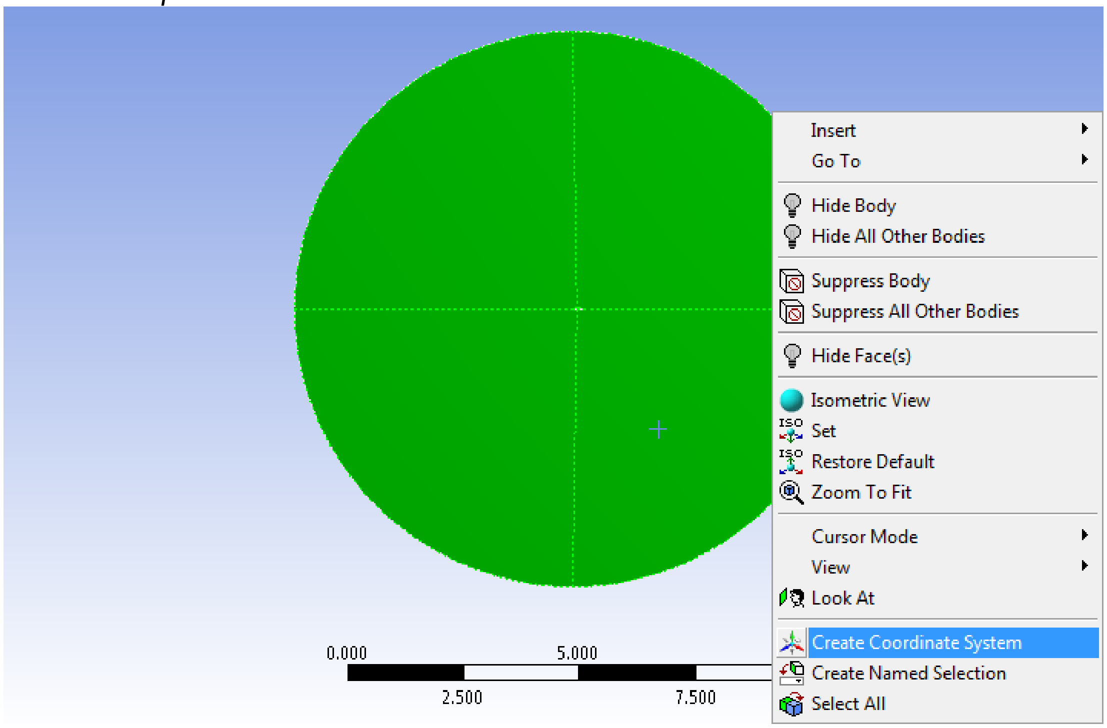
5.13. Do the same for the two right most semicircle arcs and name them *outlet*.



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



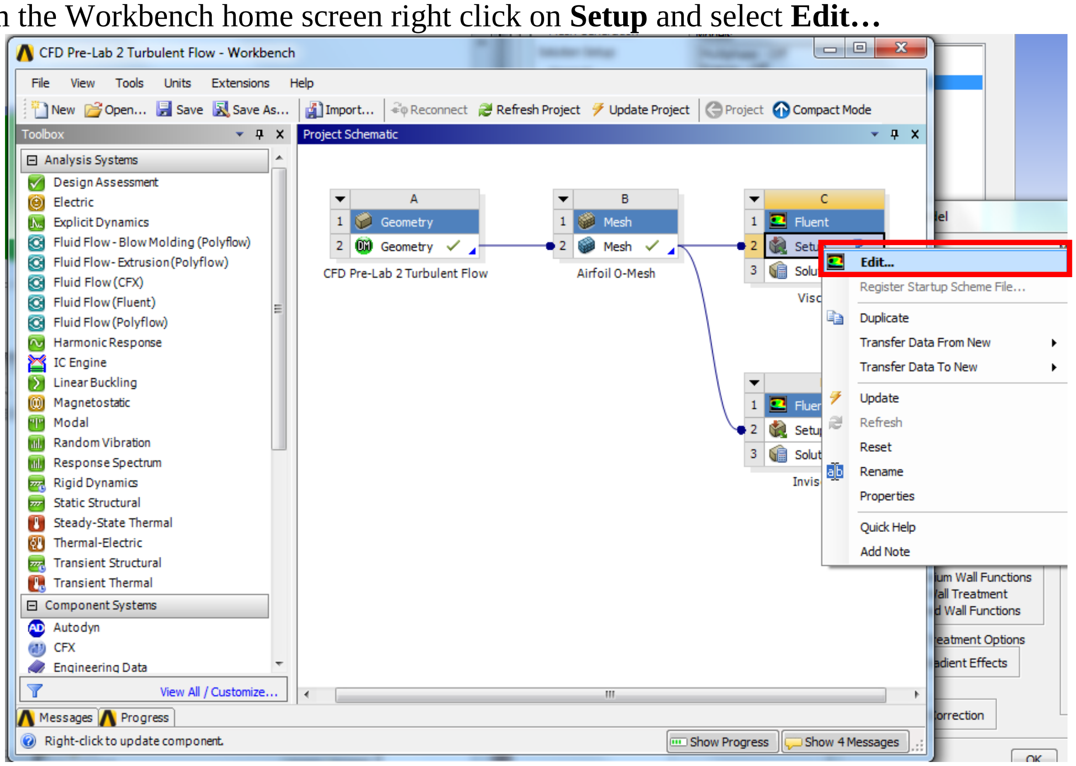
5.15. Now use the face button to select the four semicircle quadrants and **Create Named Selection** and name them *fluid*.



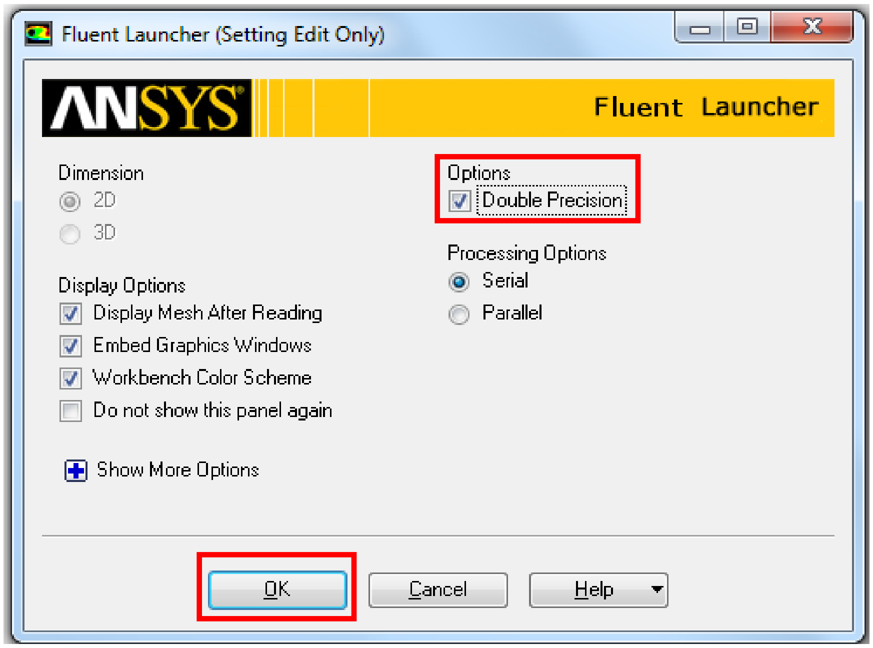
5.16. **File > Save Project.** Exit the window

6. Setup

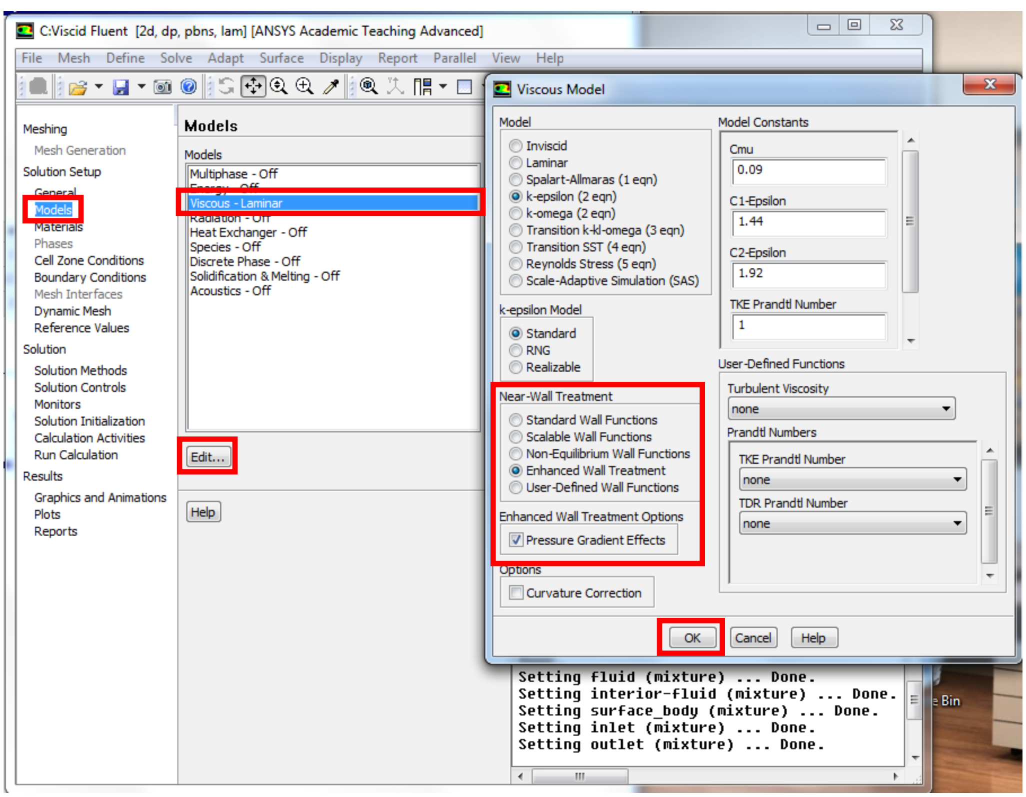
6.1. From the Workbench home screen right click on **Setup** and select **Edit…**



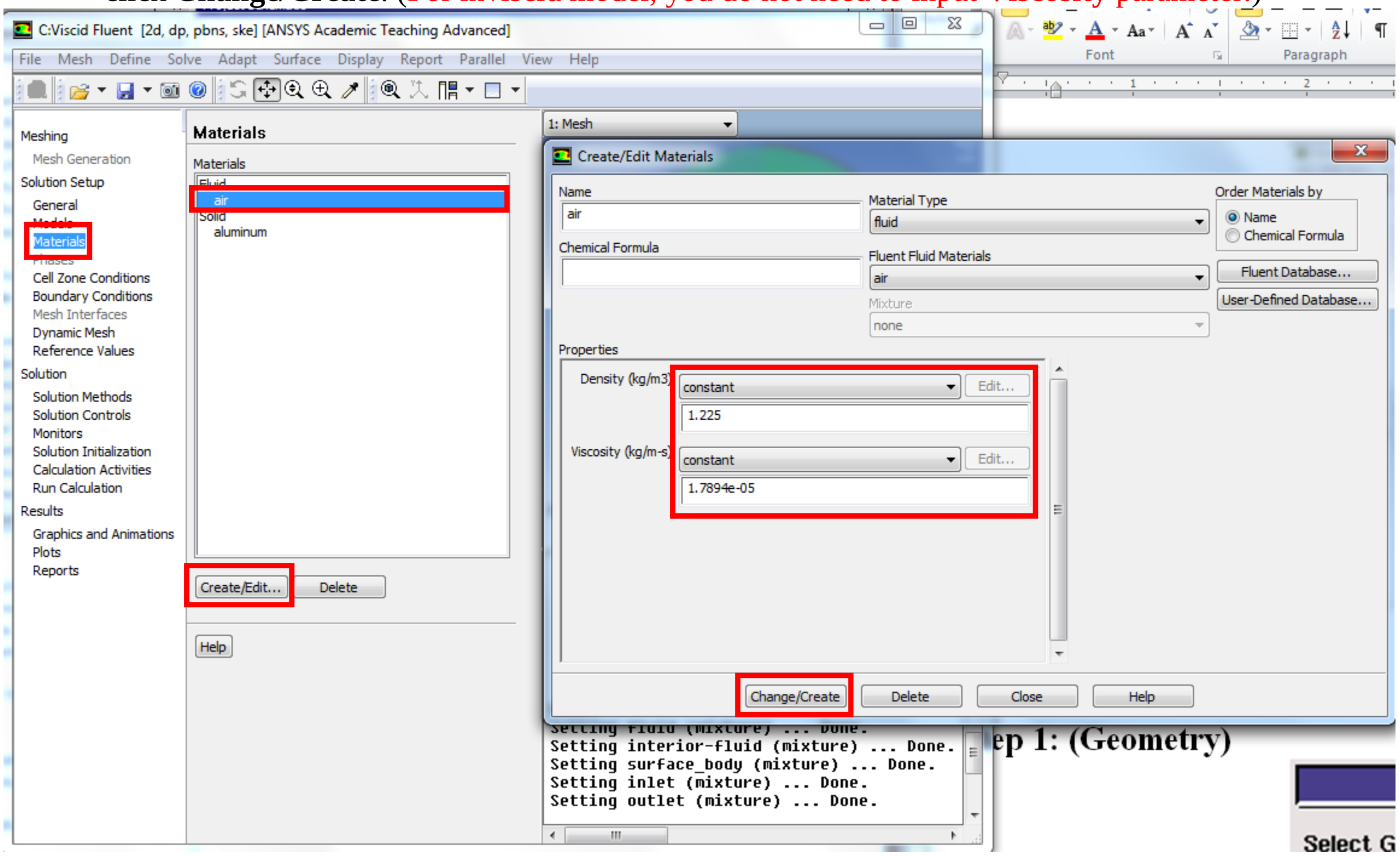
6.2. Select **Double Precision** and click **OK**.



6.3. **Solution 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.)

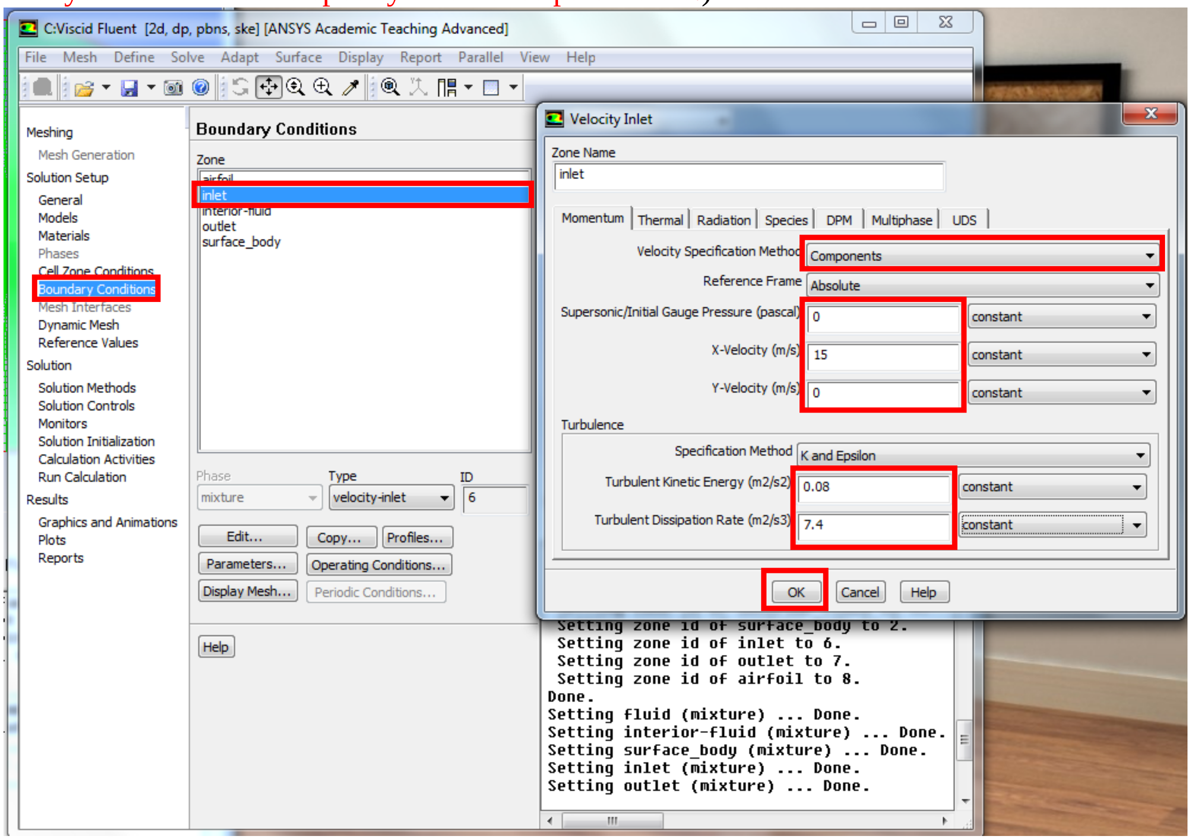


6.4. **Solution Setup** > **Materials** > **air** > **Create/Edit…** Change parameters as per experimental data and click **Change/Create**. (For inviscid model, you do not need to input **Viscosity** parameter.)

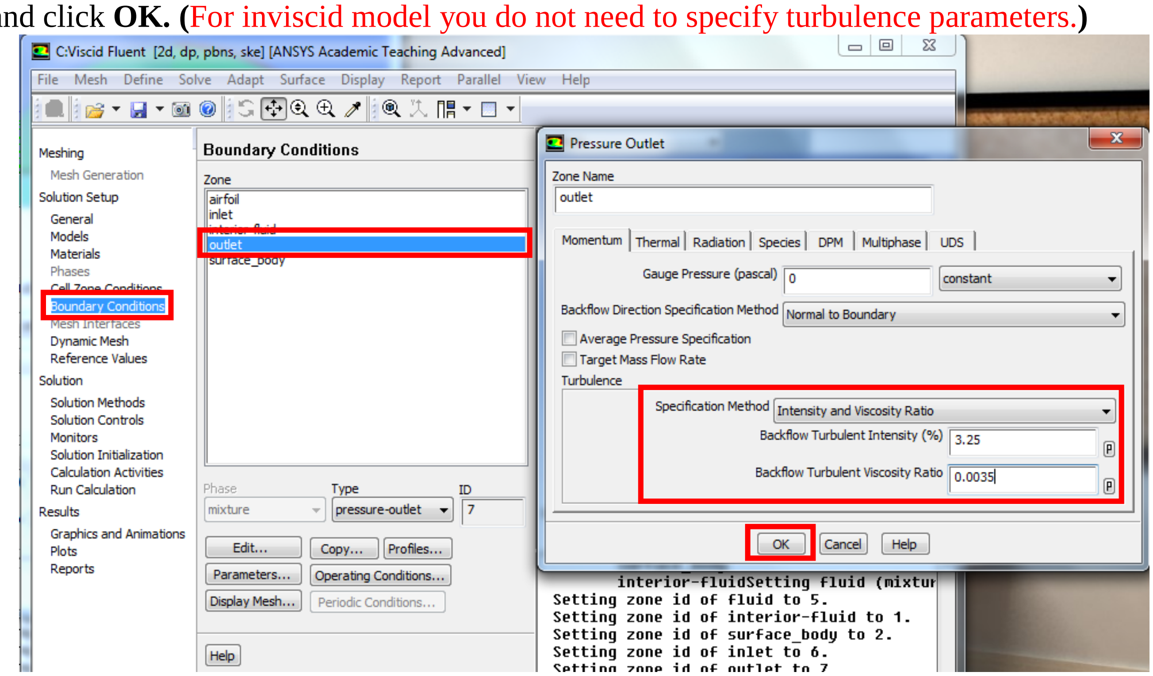


Use the air properties at the **room temperature** when you conducted EFD Lab3. **You can use the following website to calculate air properties from the temperature:**
http://www.mhtl.uwaterloo.ca/old/onlinetools/airprop/airprop.html
The values in the figure above are for 24° temperature.
NOTE: viscosity used in ANSYS is the **dynamic viscosity** ($kg/m \cdot s$), **NOT kinematic viscosity** (m^2/s)

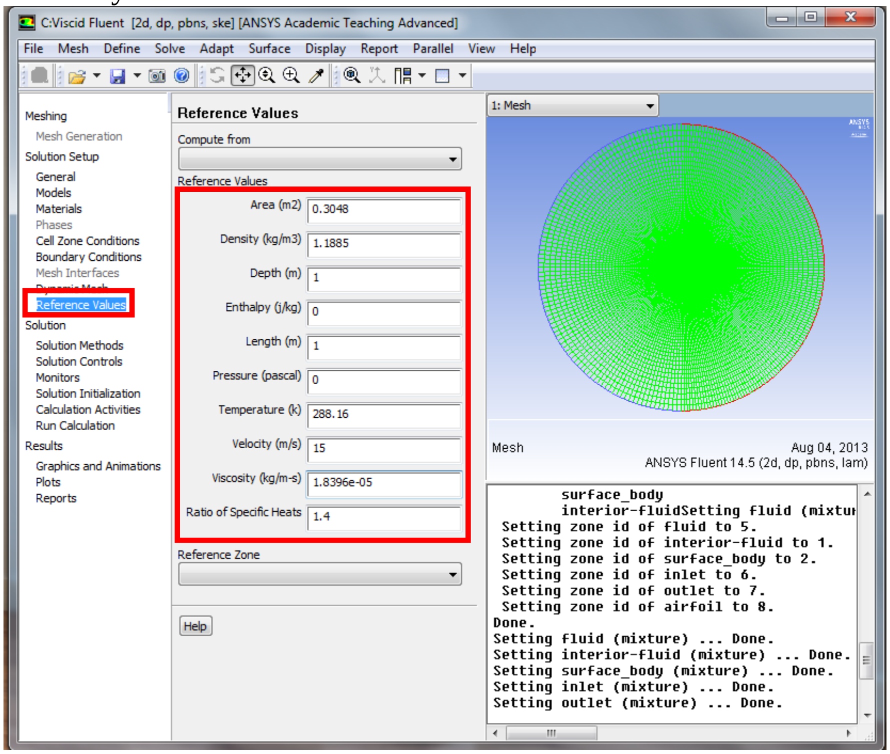
6.5. **Solution Setup** > **Boundary Conditions** > **inlet** > **Edit…** Change parameters as per experimental data and click **OK.** The value can be found from the EFD Lab 3 data reduction sheet. (For inviscid model you do not need to specify turbulence parameters.)



6.6. **Solution 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.)

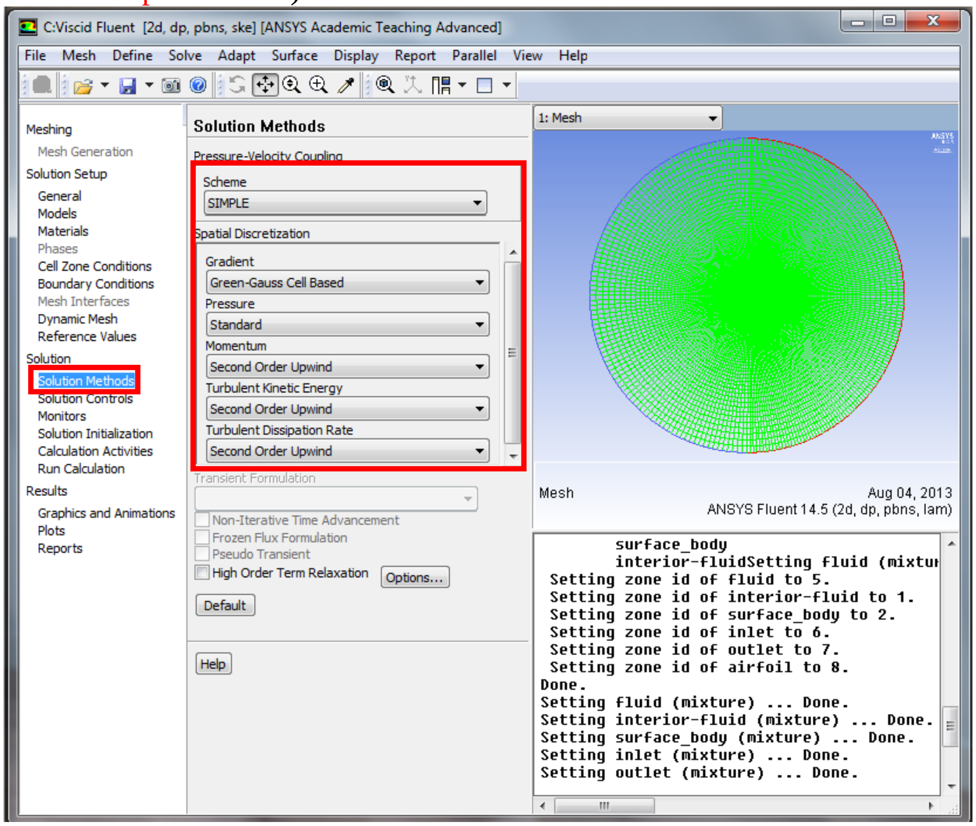


6.7. **Solution Setup** > **Reference Values**. Change parameters as per below. The velocity, temperature, density, and viscosity should be entered from EFD data.

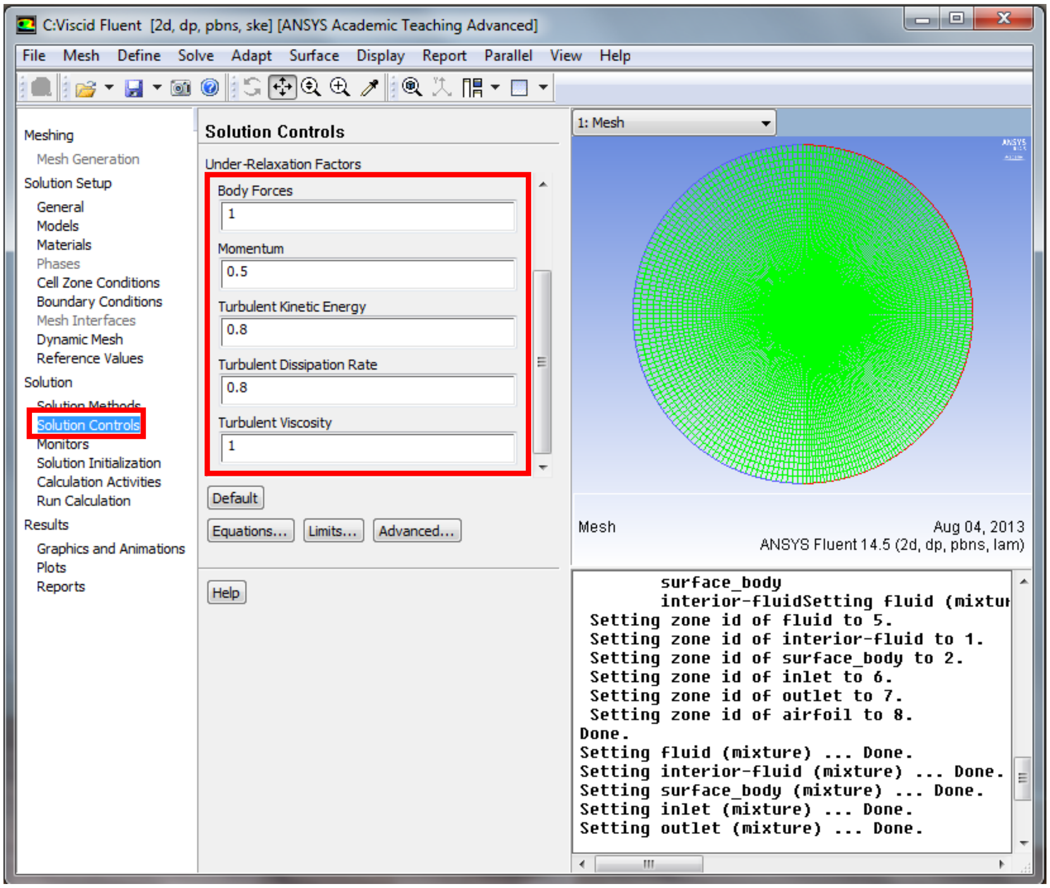


7. Setup

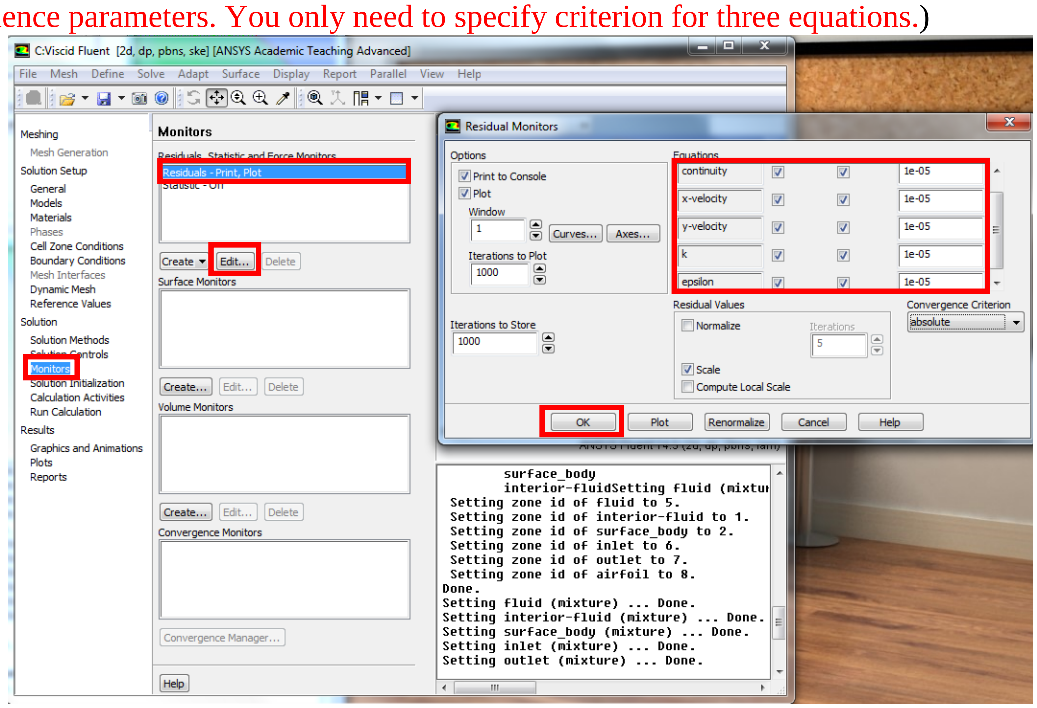
7.1. **Solution** > **Solution Methods.** Change parameters as per below. (For inviscid case you do not need to input turbulence parameters.)



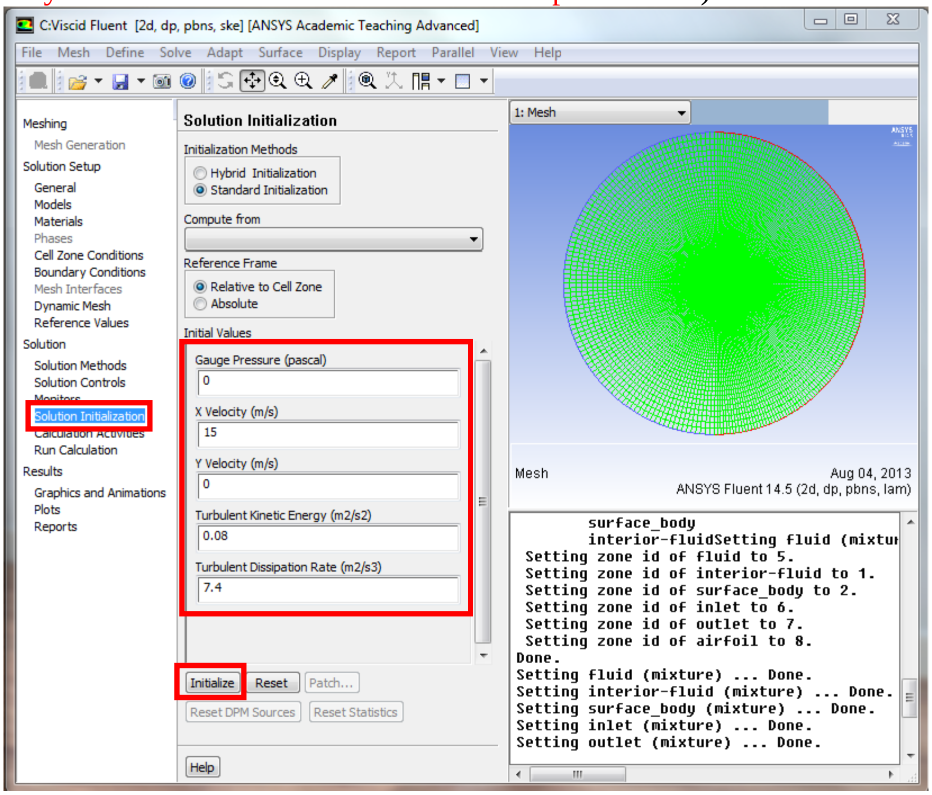
7.2. **Solution** > **Solution Controls**. Change Parameters as per below. (If you have problems with the solution converging, you may decrease the Under –Relation Factors.) (For inviscid case you do not need to specify turbulence parameters.)



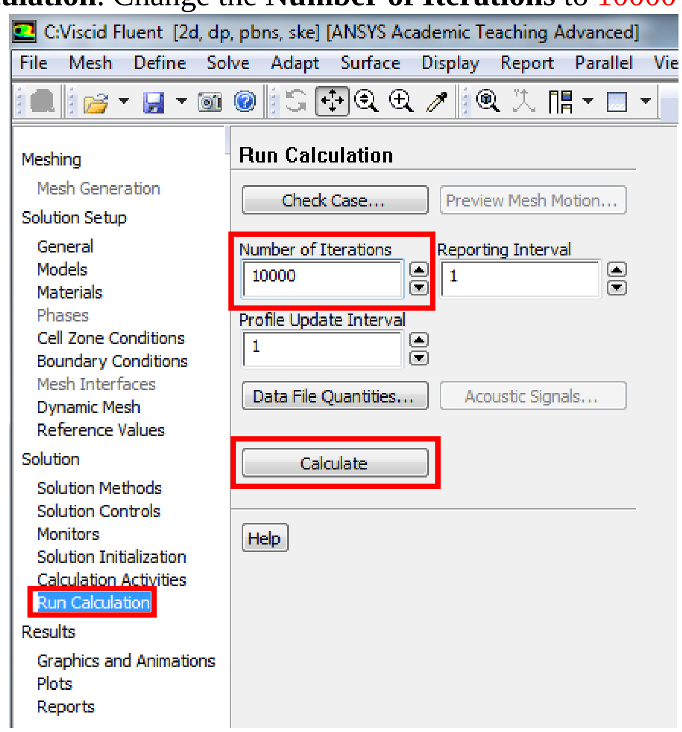
7.3. **Solution** > **Monitors** > **Residuals –Print, Plot.** Change the convergence limit to 1e-05 for all five equations and click **OK**. (For inviscid case you do not need to specify convergence criterion for turbulence parameters. You only need to specify criterion for three equations.)



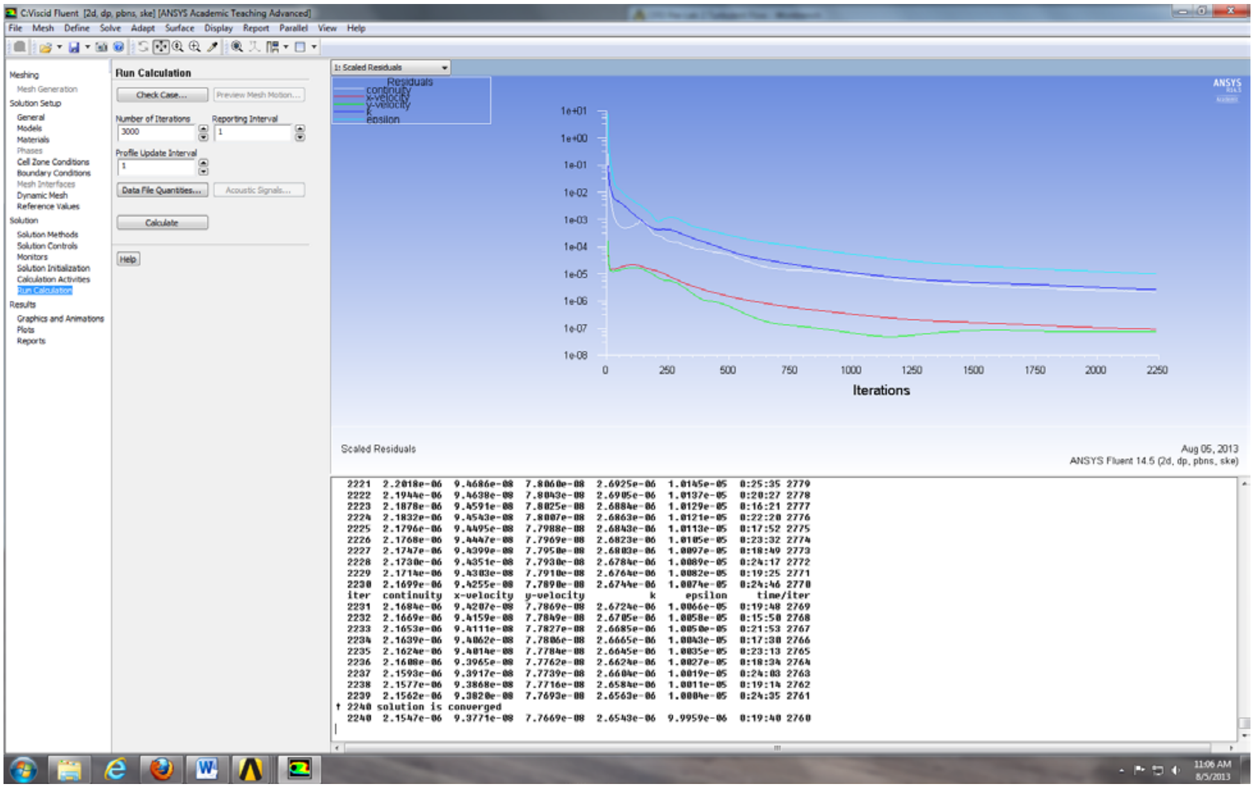
7.4. **Solution > Solution Initialization.** Change the parameters as per below and click **Initialize.** (For inviscid case you do not need to initialize turbulence parameters.)



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



Iteration history should look similar to the one below.

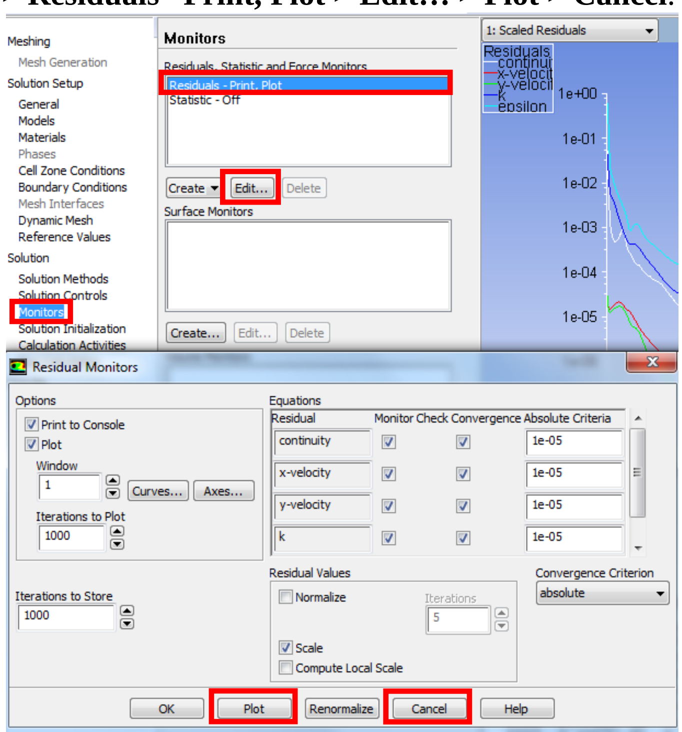


7.6. **File > Save Project.**

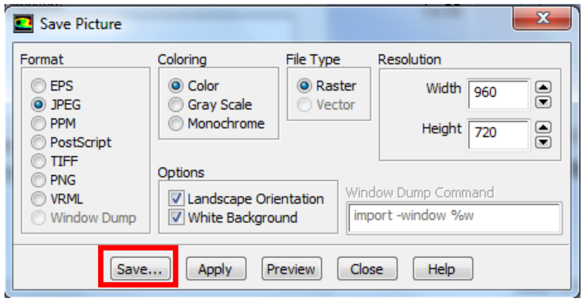
8. Results

8.1. Plotting and Saving Residual History

Solution > Monitors > Residuals –Print, Plot > Edit… > Plot > Cancel.

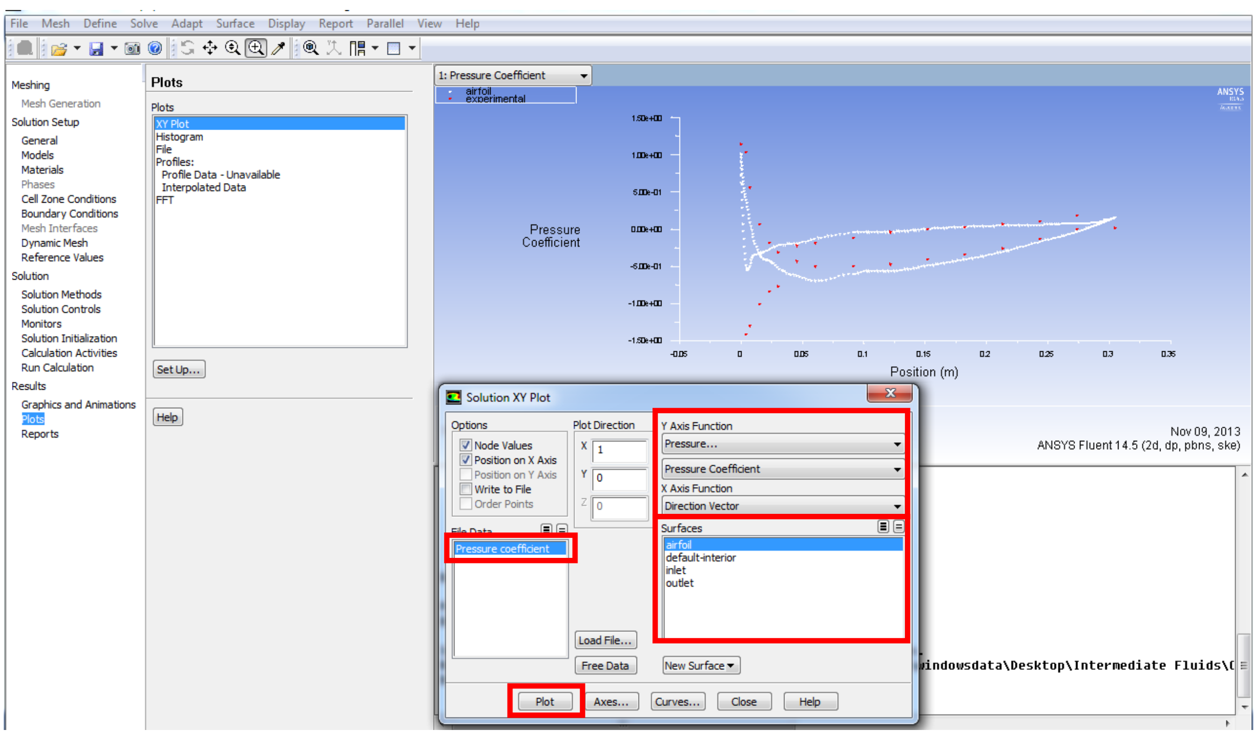


File > **Save Picture…** > **Save…** Make sure the parameters are as per below and click **Save…** Name the file *CFD Pre-Lab 2 Residual History* change the file directory to the CFD Pre-Lab 2 file you created on the H: drive and click **OK**.



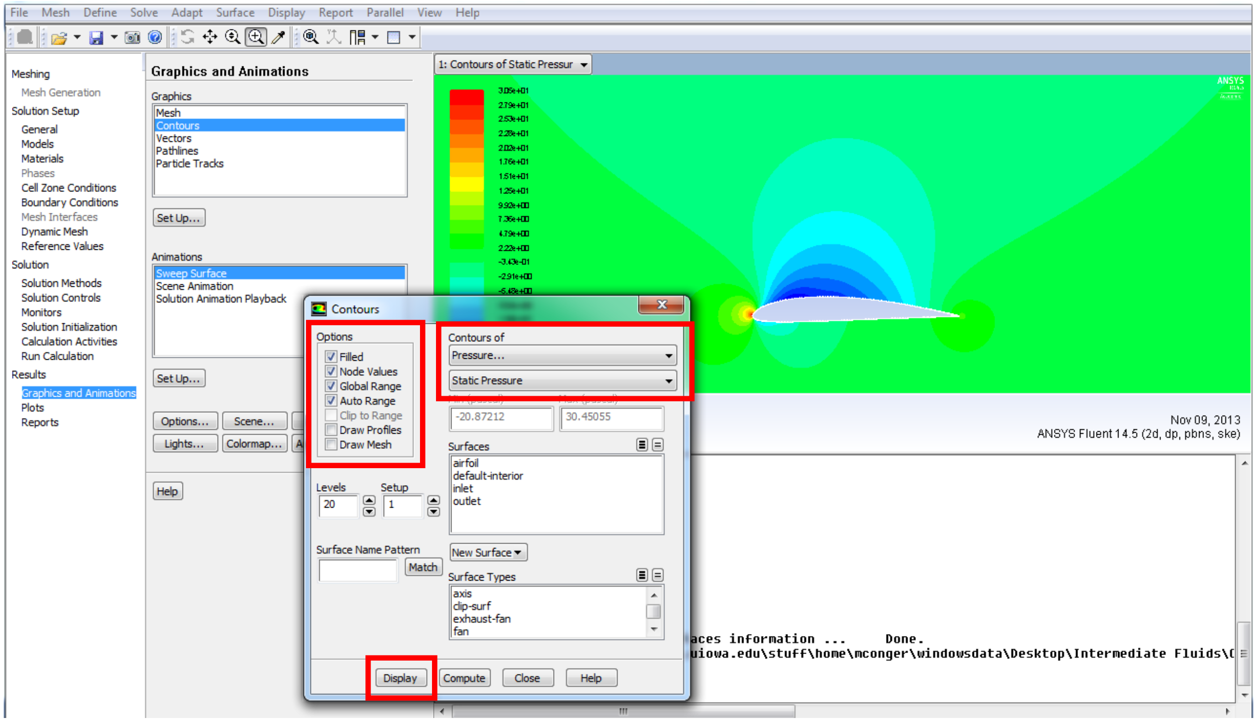
8.2. Plotting Pressure Coefficient Distribution with CFD and EFD Data

Results > **Plots** > **XY Plot** > **Set Up…** > **Load File…** Select Pressure-coef-attack0.xy. Change the parameters as per below and click **Plot**. Save the picture the same way as you did for Residual History but in this case, name it *CFD Pre-Lab 2 Pressure Coefficient Distribution*.



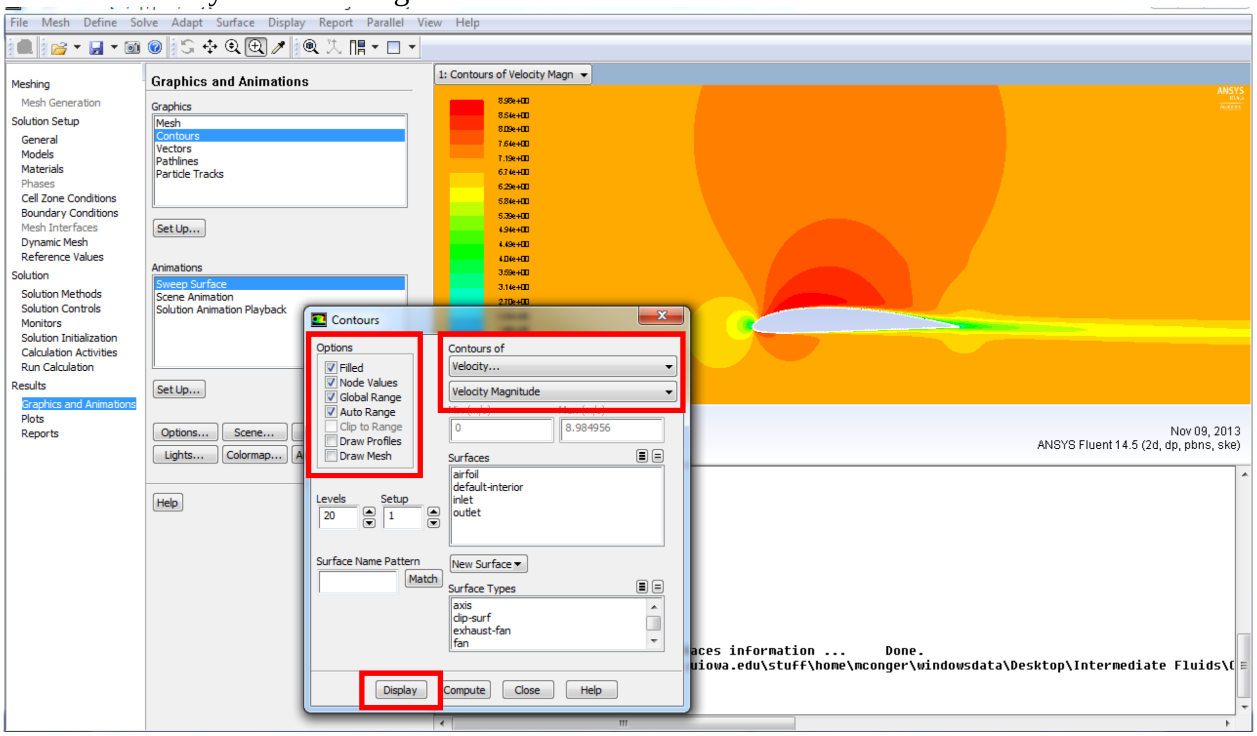
8.3. Plotting Contour of Pressure

Results > Graphics and Animations > **Contours** > **Set Up…** Change the parameters as per below and click **Display**. Save the picture the same way as you did for Residual History but in this case, name it *CFD Pre-Lab 2 Contour of Pressure*.



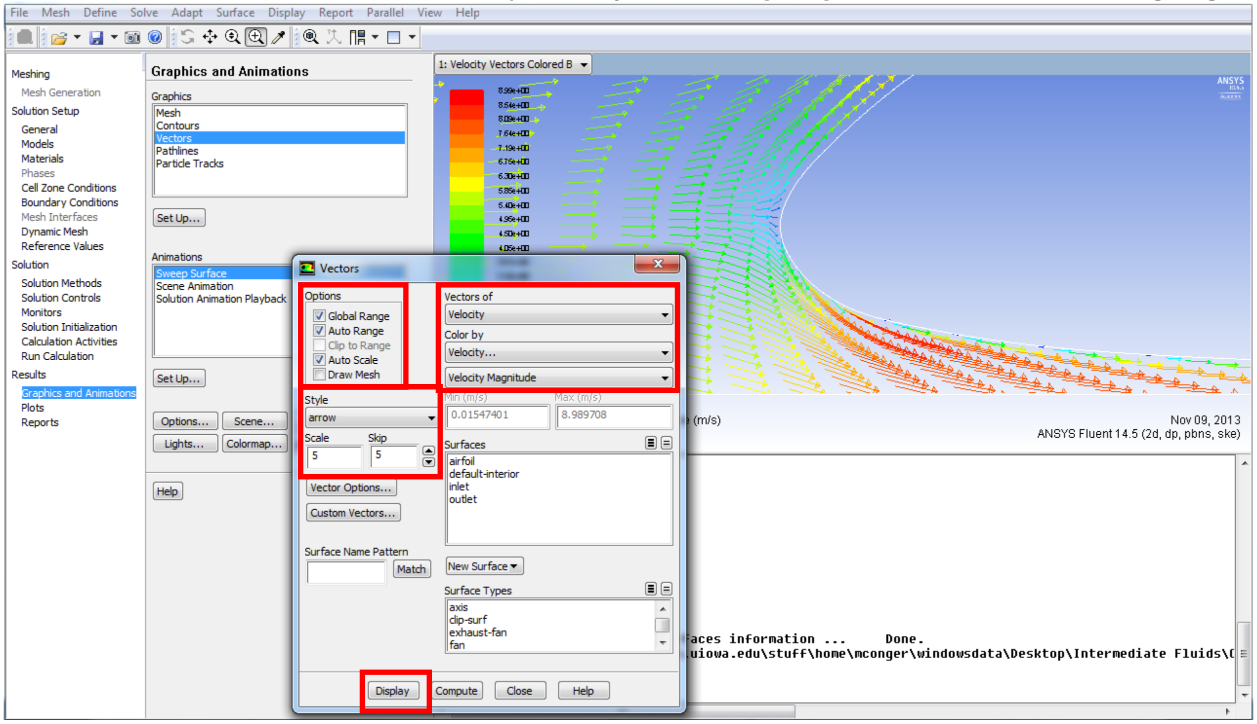
8.4. Plotting Contour of Velocity Magnitude

Results > **Graphics and Animations** > **Contours** > **Set Up…** Change the parameters as per below and click **Display**. Save the picture the same way as you did for Residual History but in this case, name it *CFD Pre-Lab 2 Contour of Velocity Magnitude*. Zoom in where you can see the airfoil clearly and the change in contour levels around the airfoil.



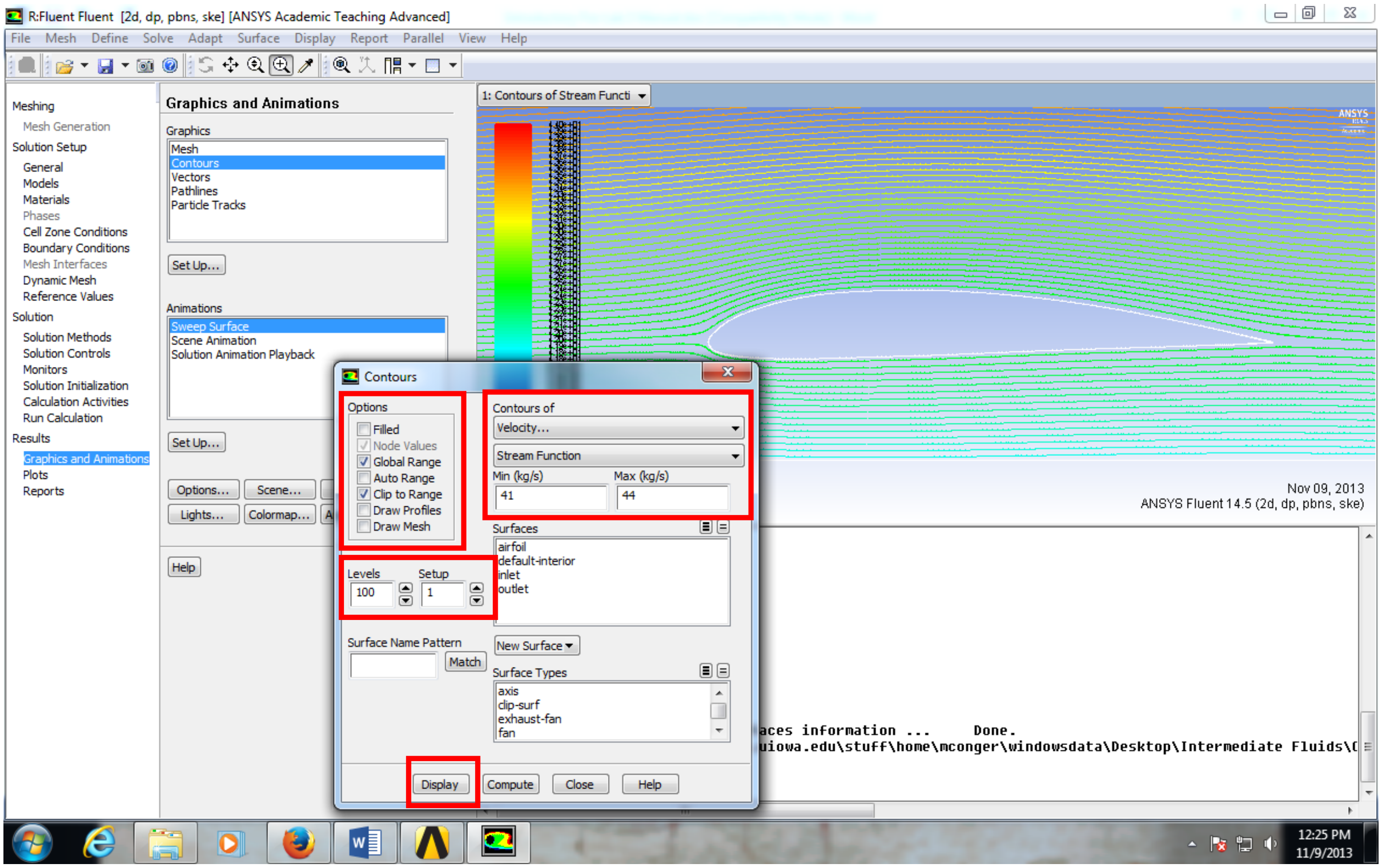
8.5. Plotting Velocity Vectors at Leading Edge

Results > **Graphics and Animations** > **Vectors** > **Set Up…** Change parameters as per below and click **Display**. Save the picture the same way as you did for Residual History but in this case, name it *CFD Pre-Lab 2 Vectors of Velocity at Leading Edge*. Zoom in on the leading edge.



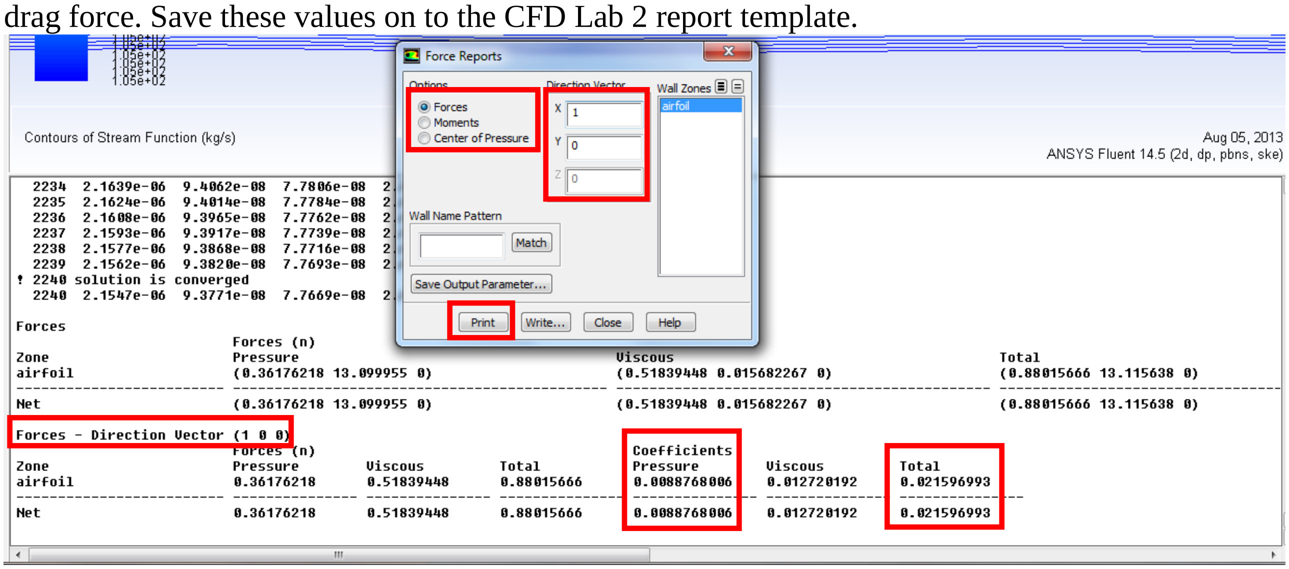
8.6. Plotting Streamlines Close to Airfoil Surface

Results > **Graphics and Animations** > **Contours** > **Set Up…** Change parameters as per below and click display. Save the picture the same way as you did for Residual History but in this case, name it *CFD Pre-Lab 2 Streamlines Close to Surface*. You may need to adjust the **Min** and **Max** to adjust the flow over the airfoil.



8.7. 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.



9. 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.

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**, **2nd 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

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 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 1), and 6. Streamlines close to airfoil surface.
   - **Data need to be saved**: lift and drag coefficients

3. **Questions need to be answered in CFD Lab 2 report:**

   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.
   3.2. What are the correct boundary conditions for velocity and pressure at "inlet" and "outlet".
   3.3. What are the values for lift and drag coefficients for inviscid flow around the airfoil? Are they both zero?
   3.4. Where are the highest and lowest locations for pressure and velocity magnitude? Why? Is pressure constant for inviscid flows around airfoil?
   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.