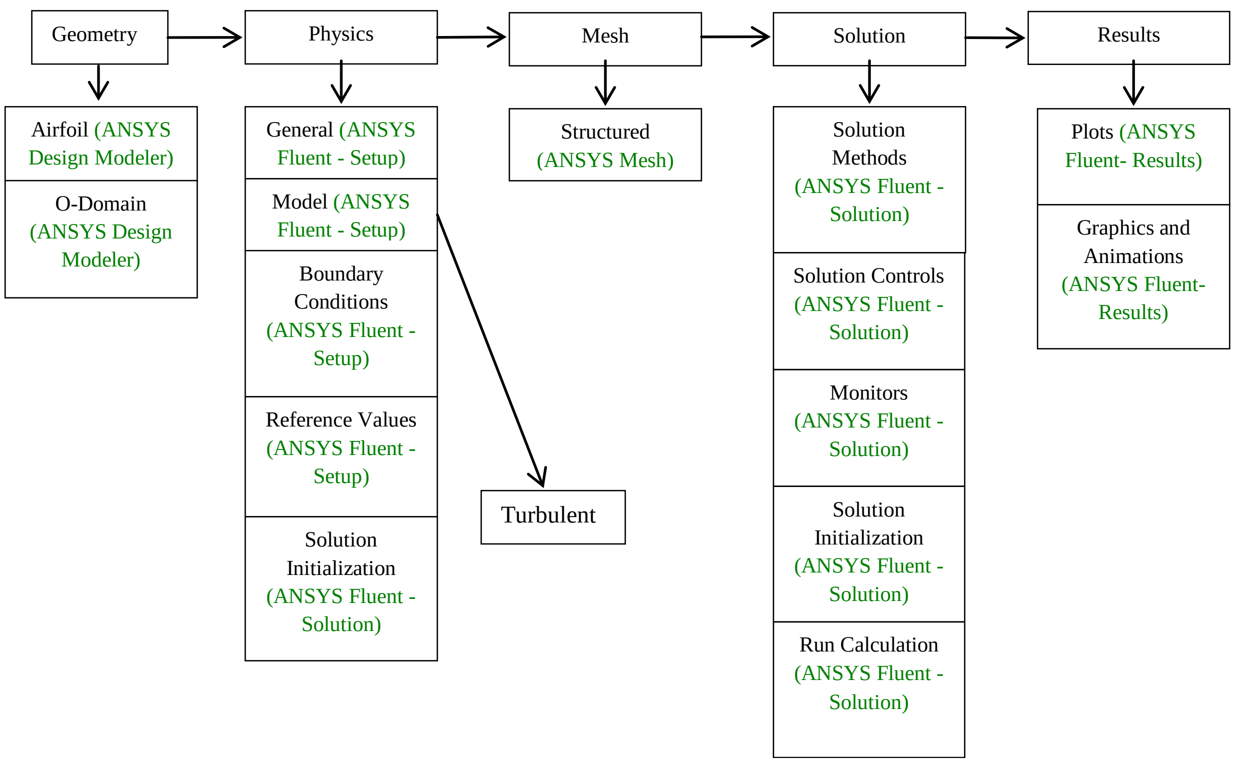
Simulation of Turbulent Flow around an Airfoil

57:020 Mechanics of Fluids and Transfer Processes
CFD 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 Lab 2 is to conduct **parametric studies** for **turbulent** flow around Clarky airfoil following the "CFD process" by an interactive step-by-step approach. Students will have "hands-on" experiences using ANSYS to investigate the **effect of angle of attack** and **effect of different turbulence models** on the simulations results. These effects will be studied by comparing simulation results with EFD data. Students will 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 airfoil flow around a ClakY airfoil (Re≈300,000). The data you have measured were used for CFD PreLab 2. In CFD Lab 2, simulation will be conducted under the same conditions of EFD Lab 3, except angle of attack and turbulent models that will be changed in this lab.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	12
Angle of attack	α	m	16

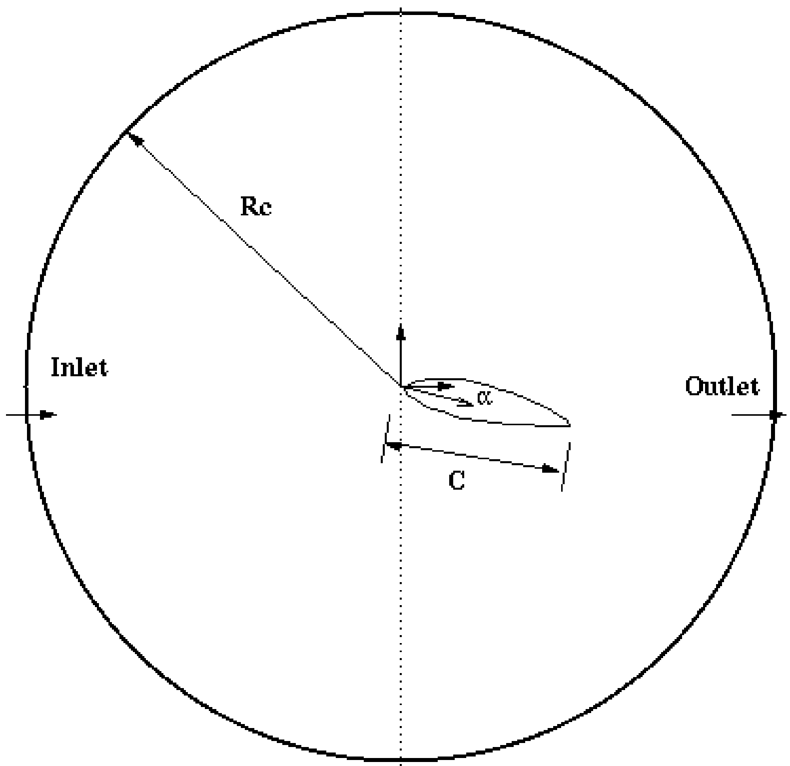
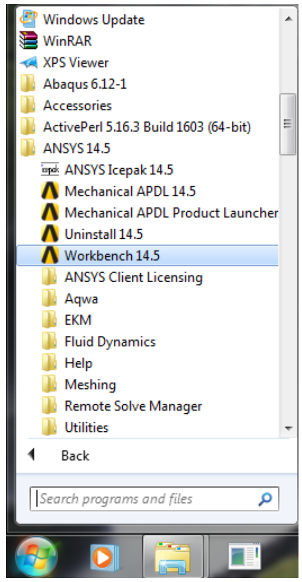


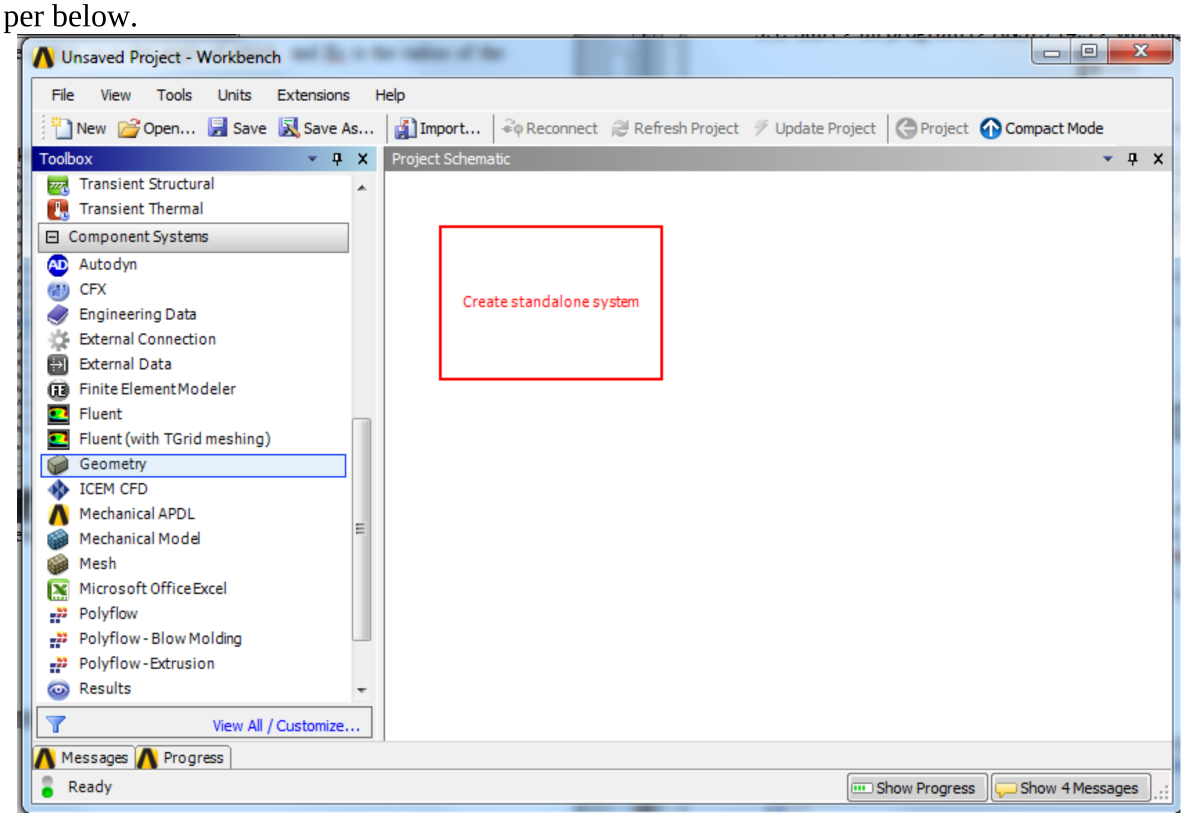
Figure 1 – Geometry

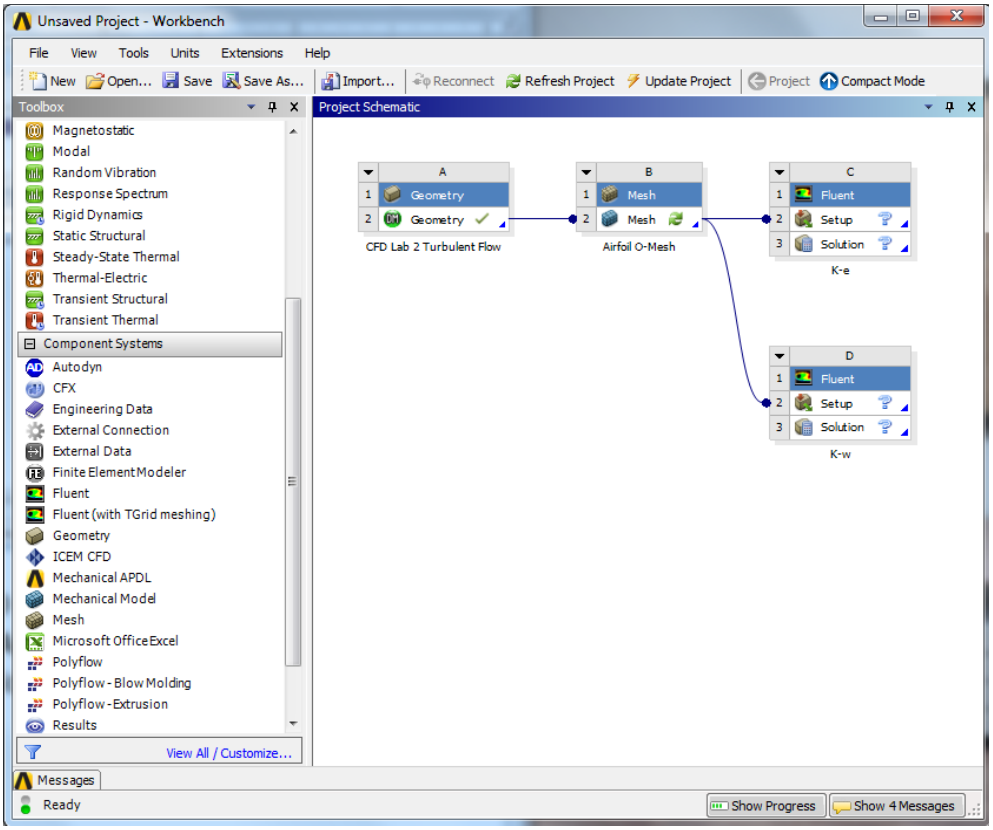
3. Open ANSYS Workbench

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 a **Fluent** component from the **Component Systems** drop down menu onto the **Project Schematic. Project Schematic** should resemble the schematic below. Rename the components as per below.

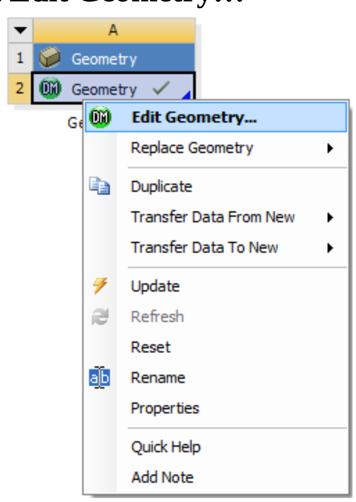




3.3. Create a Folder on the H: Drive called *CFD Lab 2*.
3.4. Save the project file by clicking **File** > **Save As…**
3.5. Save the project onto the H: Drive in the folder you just created and name it *CFD Lab 2 Turbulent Flow*.

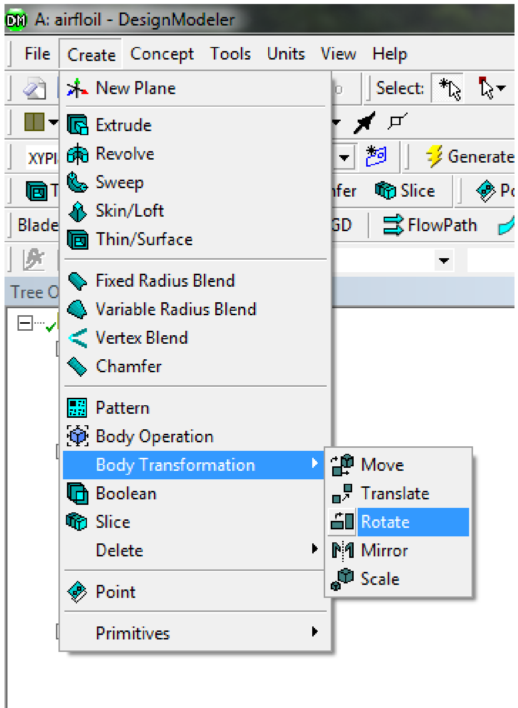
4. Geometry

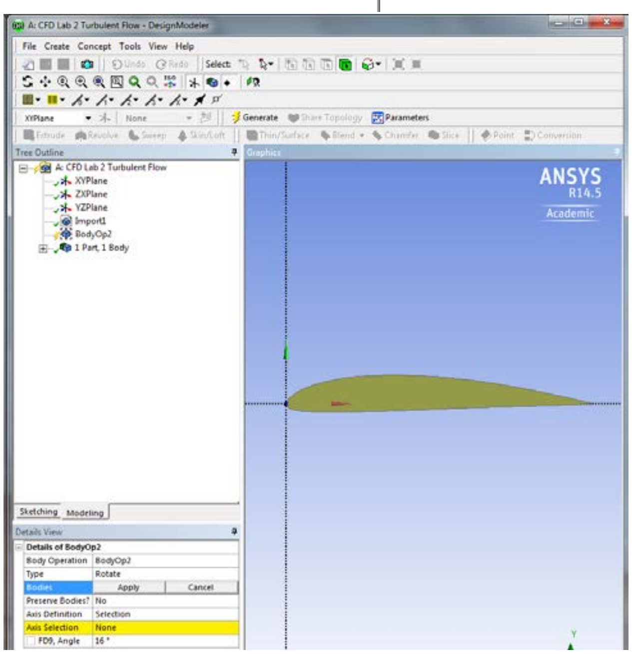
4.1. Right click **Geometry** then select **Import Geometry** > **Browse…** Select **airfoil.igs** and click **OK**.
4.2. Right click **Geometry** and select **Edit Geometry…**

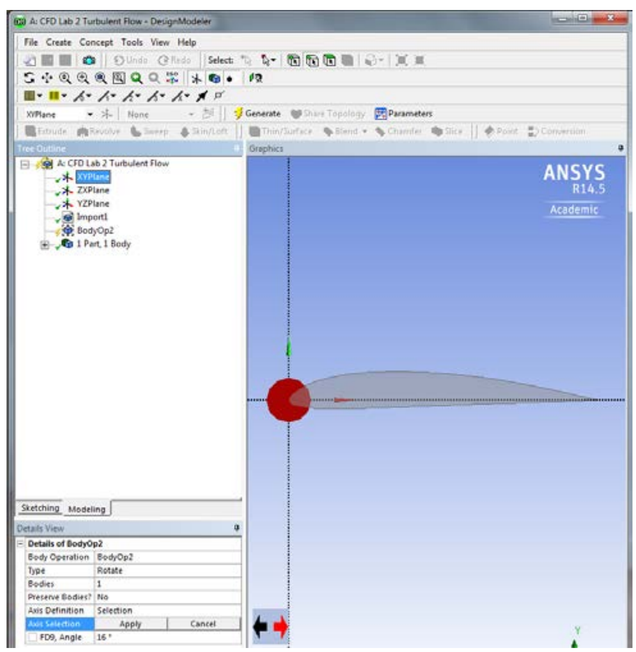


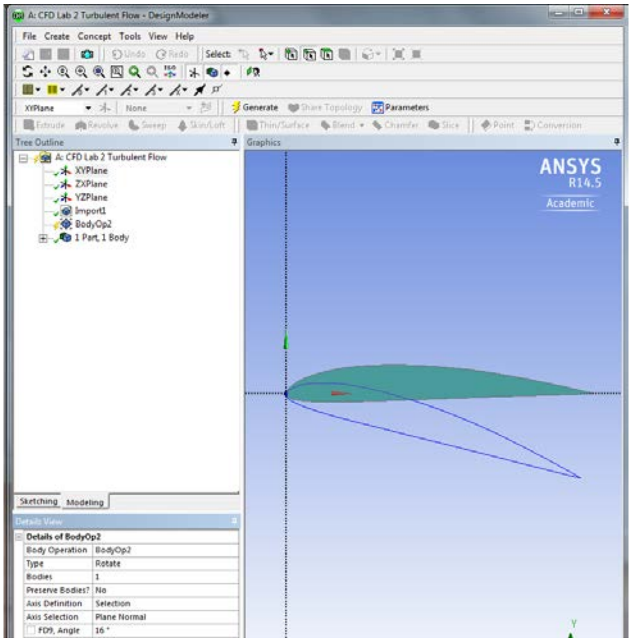
4.3. Click **Generate.**

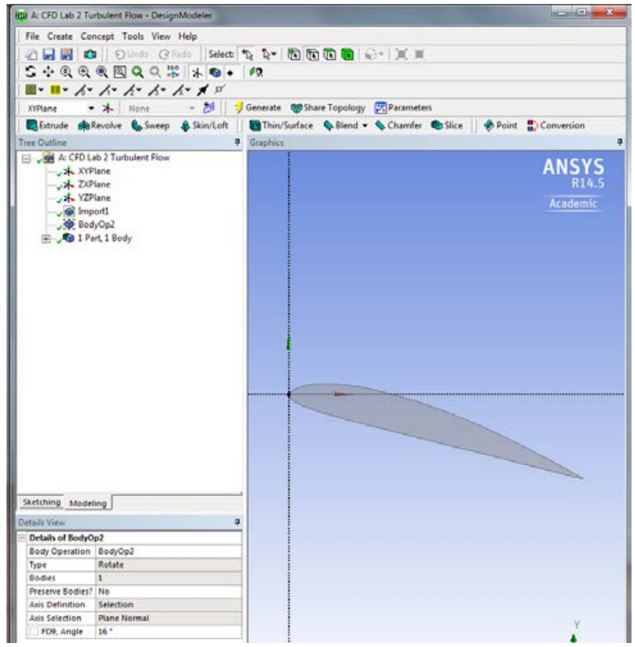
4.4. **Create > Body Transformation > Rotate.** Select the airfoil and click **Apply.** Click the yellow box labeled **Axis Selection** then click the **XYPlane** in the **Tree Outline,** then click **Apply.** Change the **Angle** to 16° and click **Generate.**



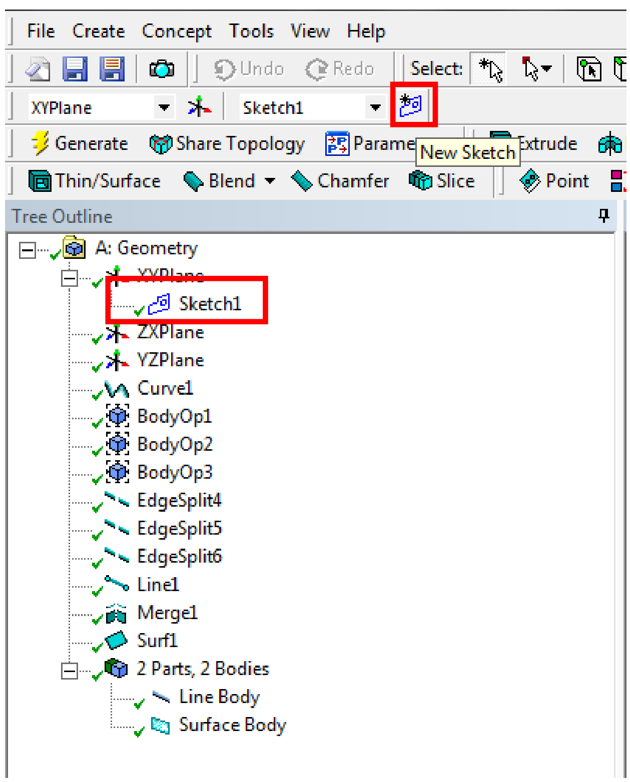




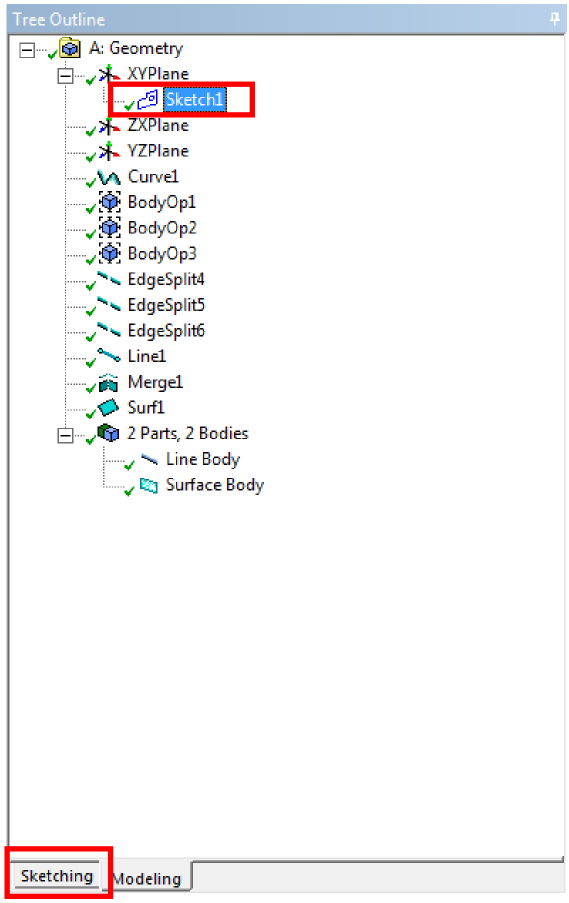




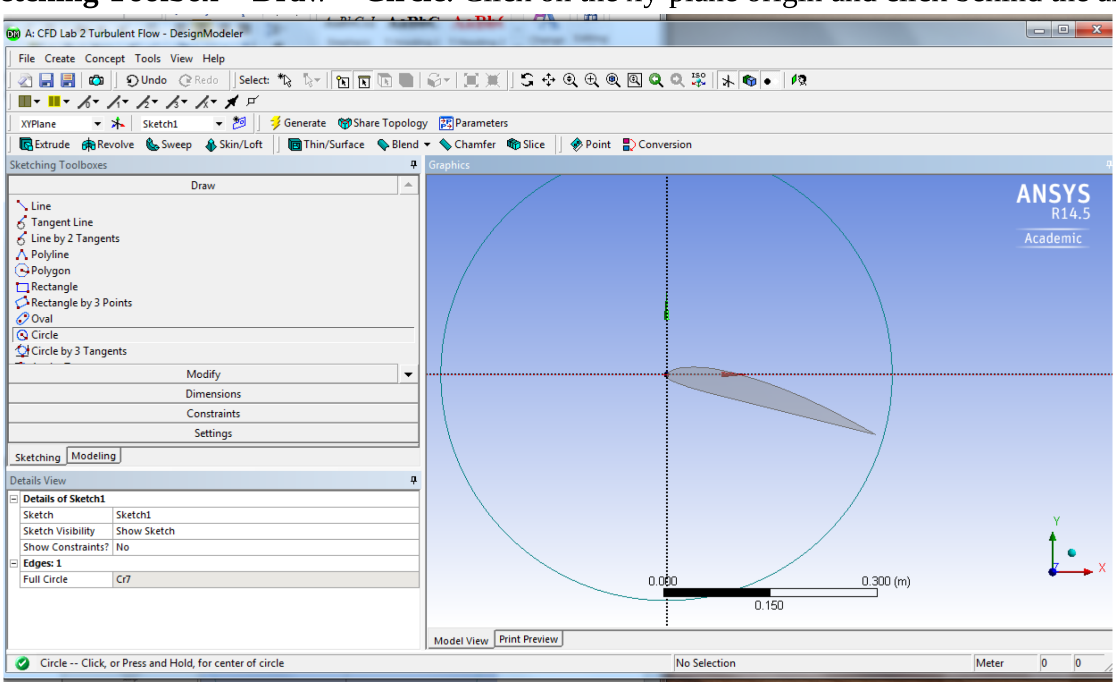
4.5. Select **XYPlane** and click New Sketch button.



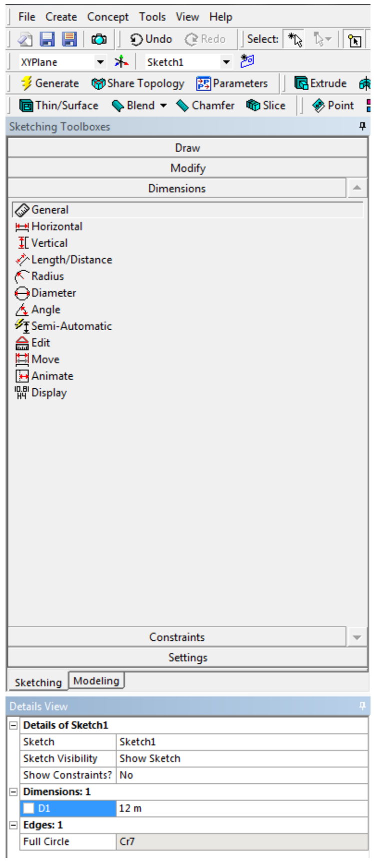
4.6. Select the sketch you created and click sketching button.



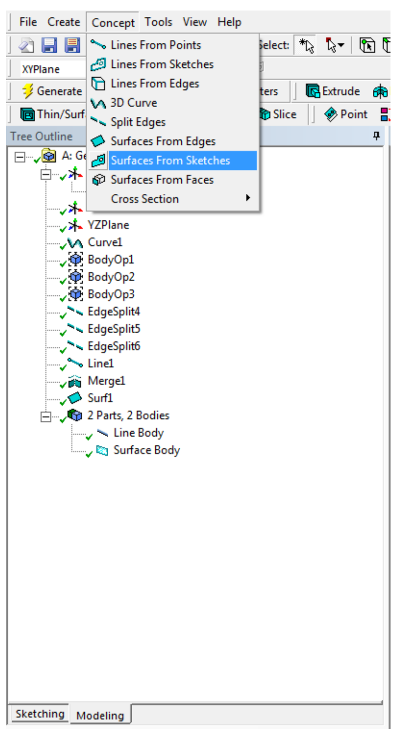
4.7. **Sketching Toolbox** > **Draw** > **Circle**. Click on the xy-plane origin and click behind the airfoil.



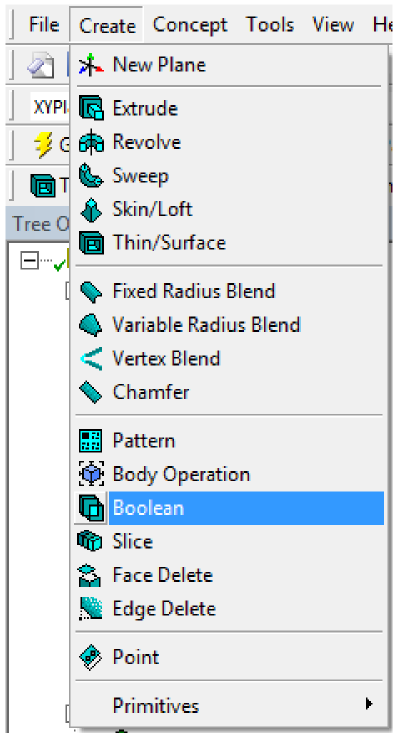
4.8. **Sketching Toolboxes** > **Dimensions** > **General**. Click on the circle and change the diameter to 12m.

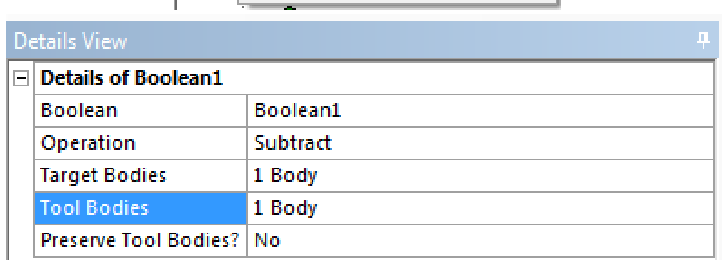


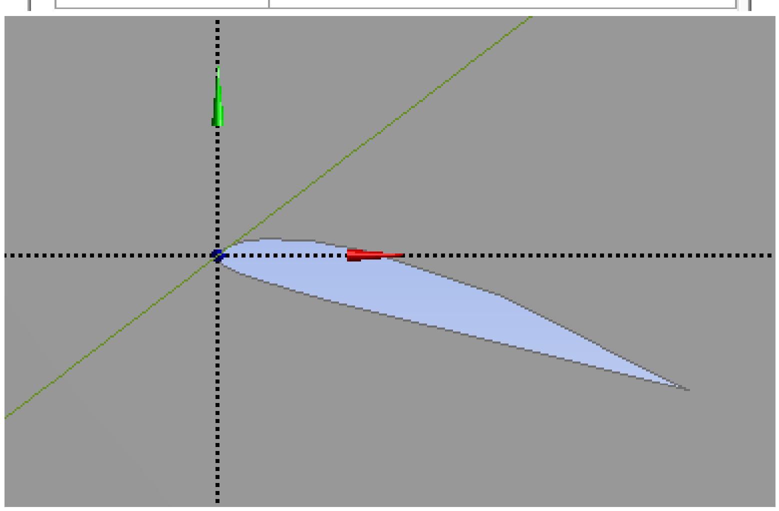
4.9. **Concept** > **Surface From Sketches**. Select your sketch, click **Apply**, then click the **Generate** button.



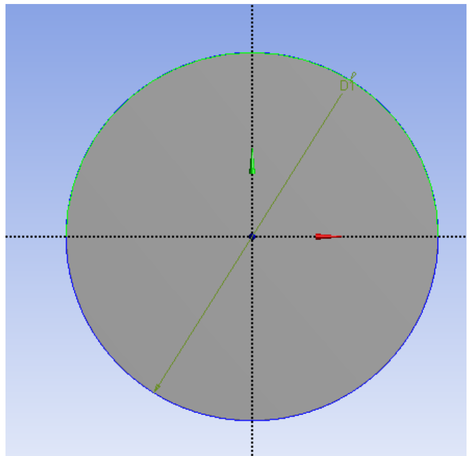
4.10. **Create** > **Boolean**. Change operation to **Subtract** then select the circle for **Target Bodies** then select airfoil for **Tool Bodies** and click **Generate**. This will subtract the airfoil surface from the circle.







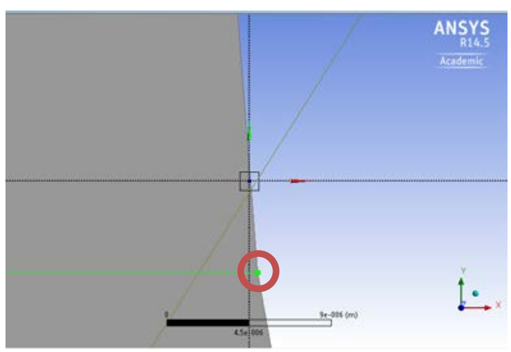
4.11. **Concept** > **Split Edges.** Select the perimeter of the circle and click **Apply.** Select **Generate.** This should split the circle into two semicircles. You can see the semicircles by selecting the perimeter above and below the x-axis.

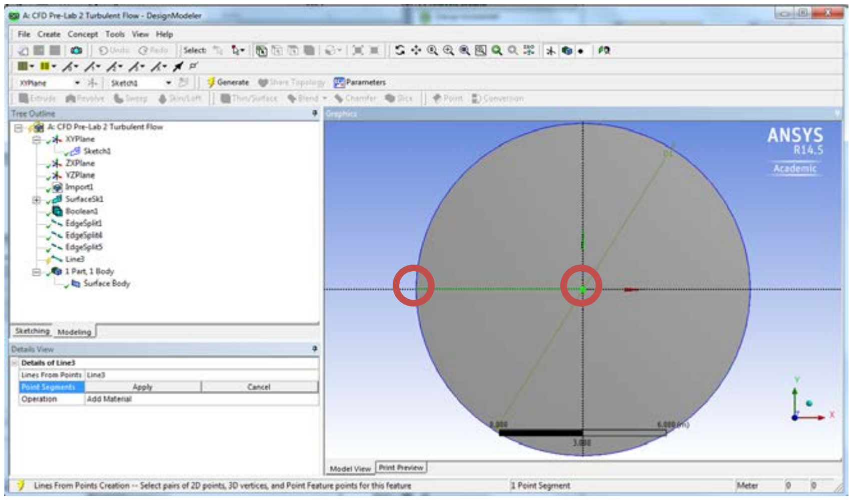


4.12. Repeat the process from 4.11 on the two semicircles. This should yield four circular quadrants.

4.13. Repeat this process for the arc in quadrant IV. Change the Fraction to 0.822222. This splits the arc into a 16° and a 74° arc.

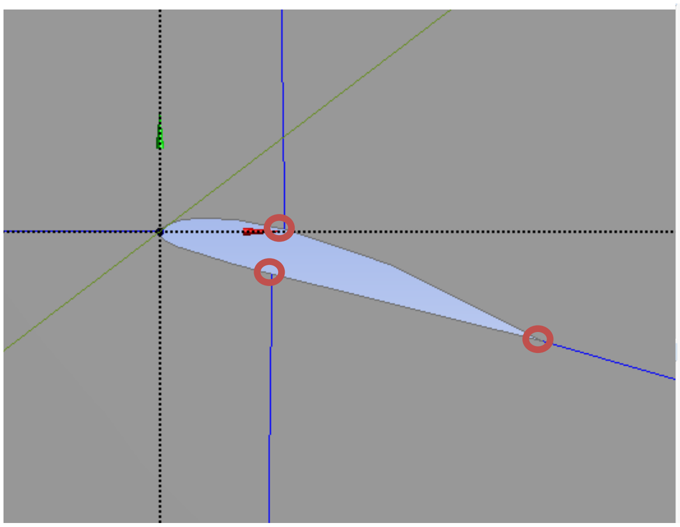
4.14. **Concept** > **Lines From Points**. Draw a line from the point on the circle to the point on the airfoil making sure to hold Ctrl while doing so. Note: The point on the airfoil is not exactly on the origin. Zoom in and find the point just below the origin and use that point. The images below show the locations of the points circled.

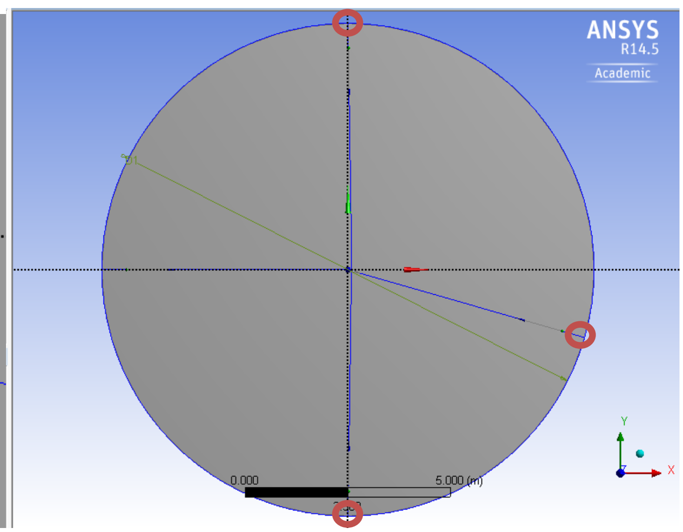




4.15. Once you select both points hit **Apply.** Then click **Generate.**

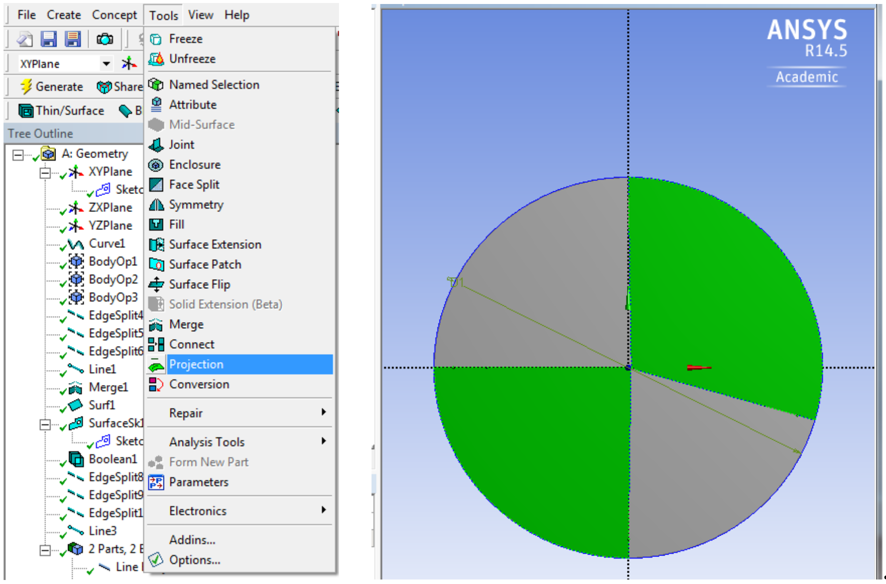
4.16. Repeat this process creating lines from the edge of the circle to the airfoil starting from the circle and ending at the airfoil. The images below show the locations of the points on the airfoil and the points on the circle.





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

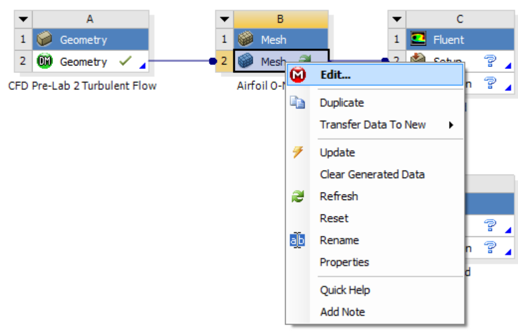
4.18. **Tools** > **Merge.** Select the 16° arc and the arc in quadrant I and select **Apply.** Click **Generate.** This merges the lines into one line which can be sized for meshing easier.



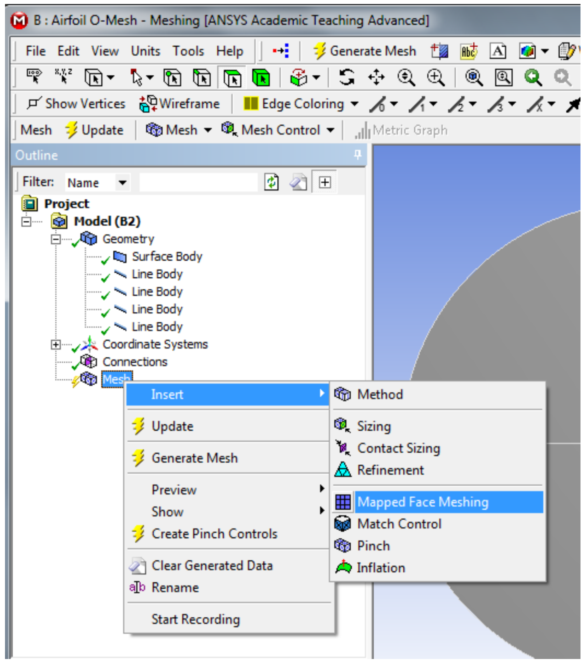
4.19. **File** > **Save project** and exit.

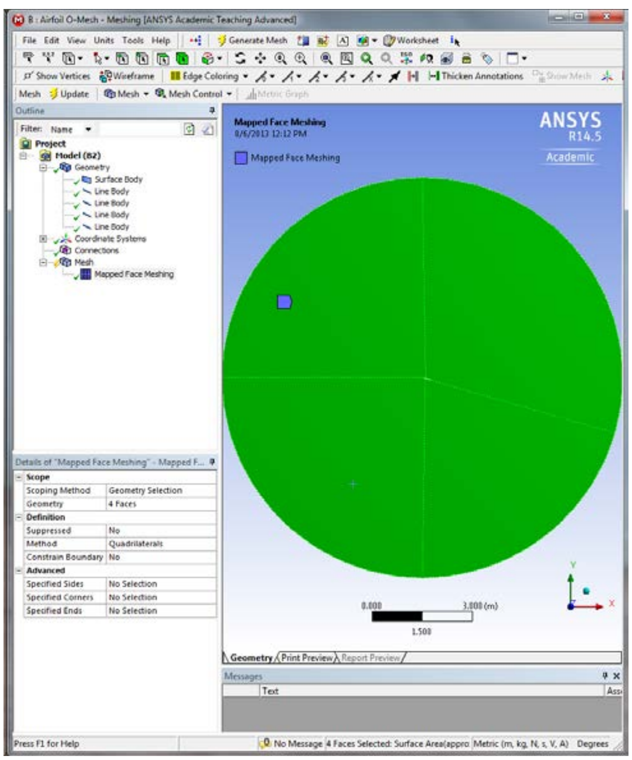
5. Mesh Generation

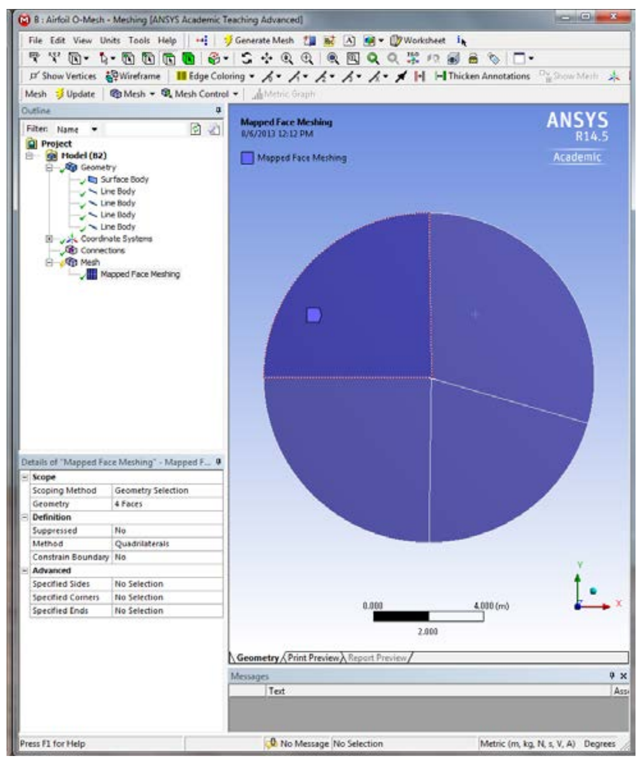
5.1. From the Project Schematic right click on **Mesh** and select **Edit…** from the dropdown menu.



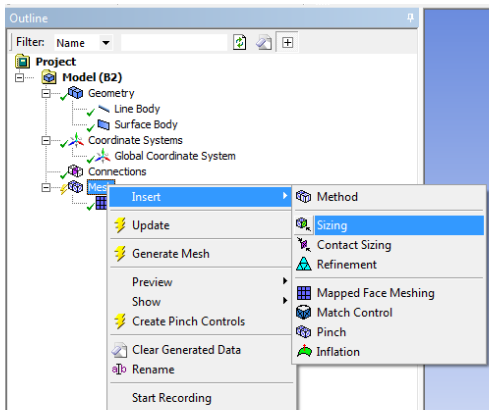
5.2. Right click **Mesh** then **Insert** > **Mapped Face Meshing**. Select the four surfaces then click **Apply**.

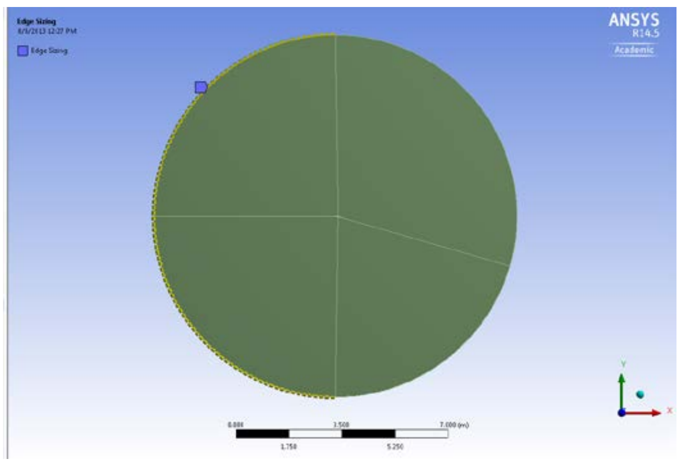






5.3. Right click **Mesh** and **Insert** > **Sizing**. Select two edges as per below and change the parameters as per below.

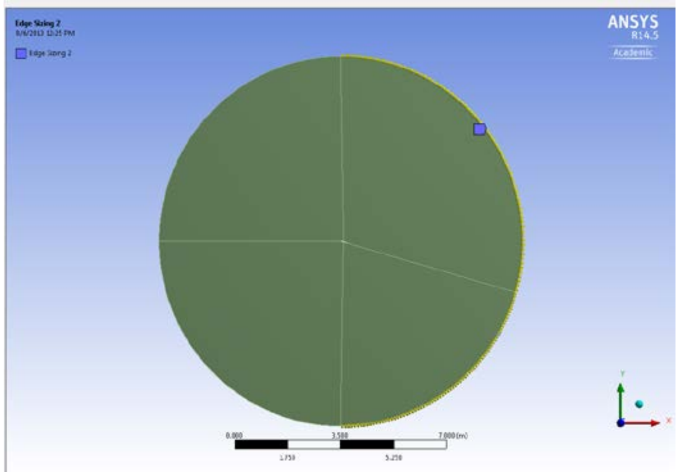




Details of "Edge Sizing" - Sizing

Scope	
Scoping Method	Geometry Selection
Geometry	2 Edges
Definition	
Suppressed	No
Type	Number of Divisions
Number of Divisions	45
Behavior	Hard
Bias Type	No Bias

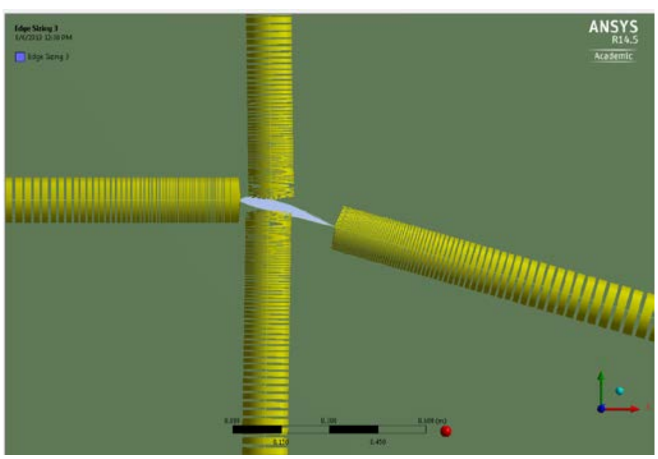
5.4. Right click **Mesh** and **Insert** > **Sizing**. Select two edges as per below and change the parameters as per below.



Details of "Edge Sizing 2" - Sizing

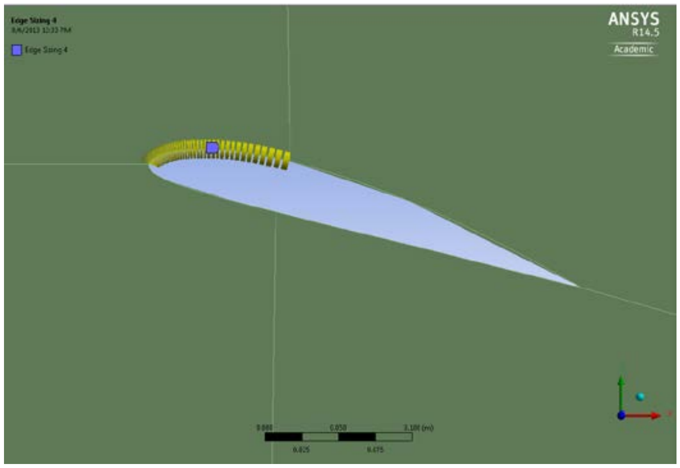
Scope	
Scoping Method	Geometry Selection
Geometry	2 Edges
Definition	
Suppressed	No
Type	Number of Divisions
Number of Divisions	60
Behavior	Hard
Bias Type	No Bias

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.



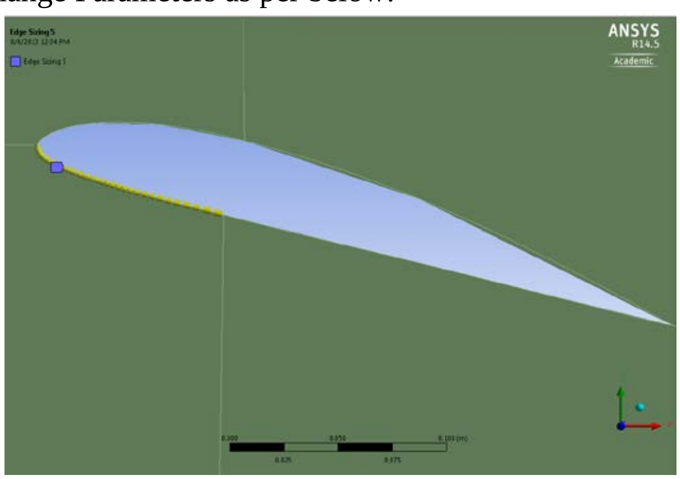
Details of "Edge Sizing 3" - Sizing	
Scope	
Scoping Method	Geometry Selection
Geometry	4 Edges
Definition	
Suppressed	No
Type	Number of Divisions
Number of Divisions	115
Behavior	Hard
Bias Type	----- --- - -
Bias Option	Bias Factor
Bias Factor	5000.

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 4" - Sizing	
Scope	
Scoping Method	Geometry Selection
Geometry	1 Edge
Definition	
Suppressed	No
Type	Number of Divisions
Number of Divisions	45
Behavior	Hard
Bias Type	----- --- - -
Bias Option	Bias Factor
Bias Factor	15.

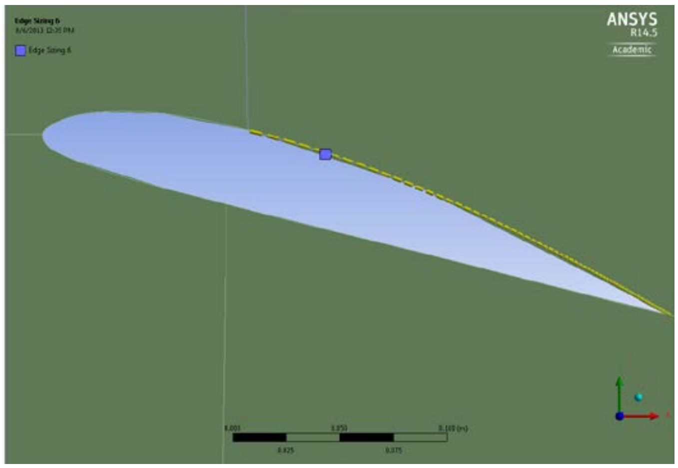
5.7. Right click **Mesh** and **Insert** > **Sizing.** Select the surface at the bottom of leading edge of the airfoil and click **Apply**. Change Parameters as per below.



Details of "Edge Sizing 5" - Sizing

Scope	
Scoping Method	Geometry Selection
Geometry	1 Edge
Definition	
Suppressed	No
Type	Number of Divisions
Number of Divisions	45
Behavior	Hard
Bias Type	_ _ ___ _____
Bias Option	Bias Factor
Bias Factor	15.

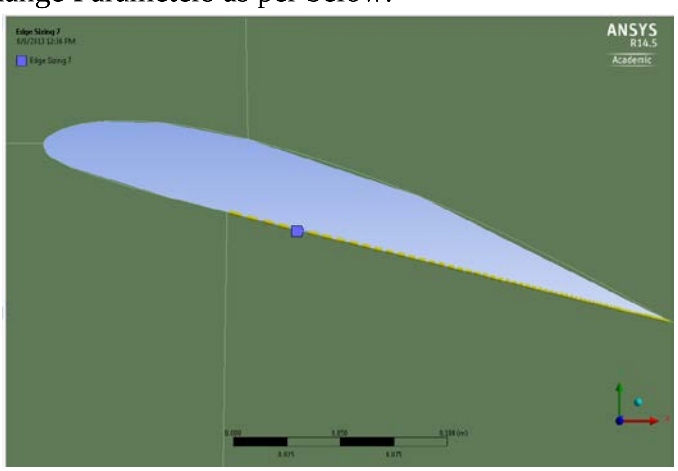
5.8. Right click **Mesh** and **Insert** > **Sizing.** Select the surface at the top of trailing edge of the airfoil and click **Apply**. Change Parameters as per below.

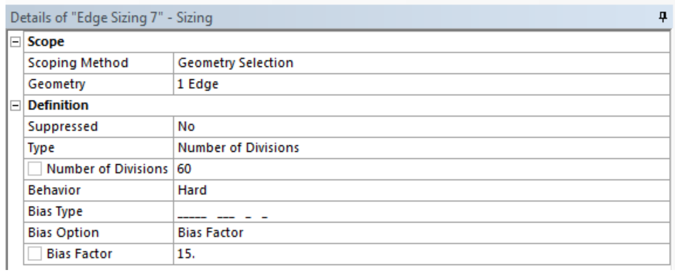


Details of "Edge Sizing 6" - Sizing

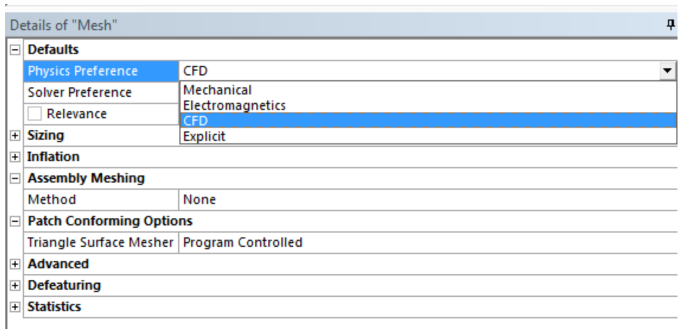
Scope	
Scoping Method	Geometry Selection
Geometry	1 Edge
Definition	
Suppressed	No
Type	Number of Divisions
Number of Divisions	60
Behavior	Hard
Bias Type	_ _ ___ _____
Bias Option	Bias Factor
Bias Factor	15.

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.

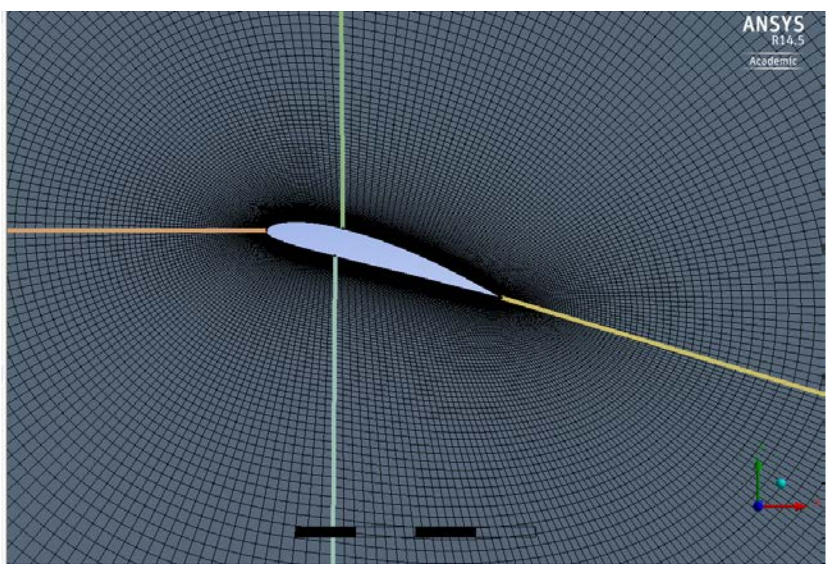




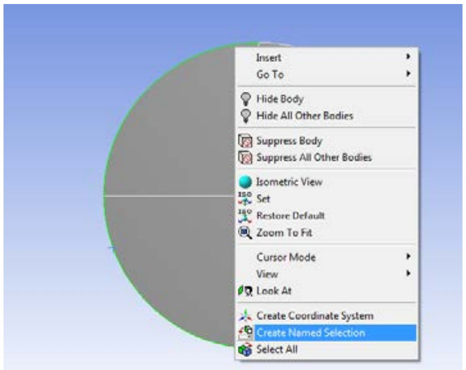
5.10. Click on **Mesh** under the **Outline**, under the **Details of "Mesh"**, change the **Physics Preference** from **Mechanical** to **CFD.** This changes the grid solver to a fluids style solver rather than a FEA style solver.



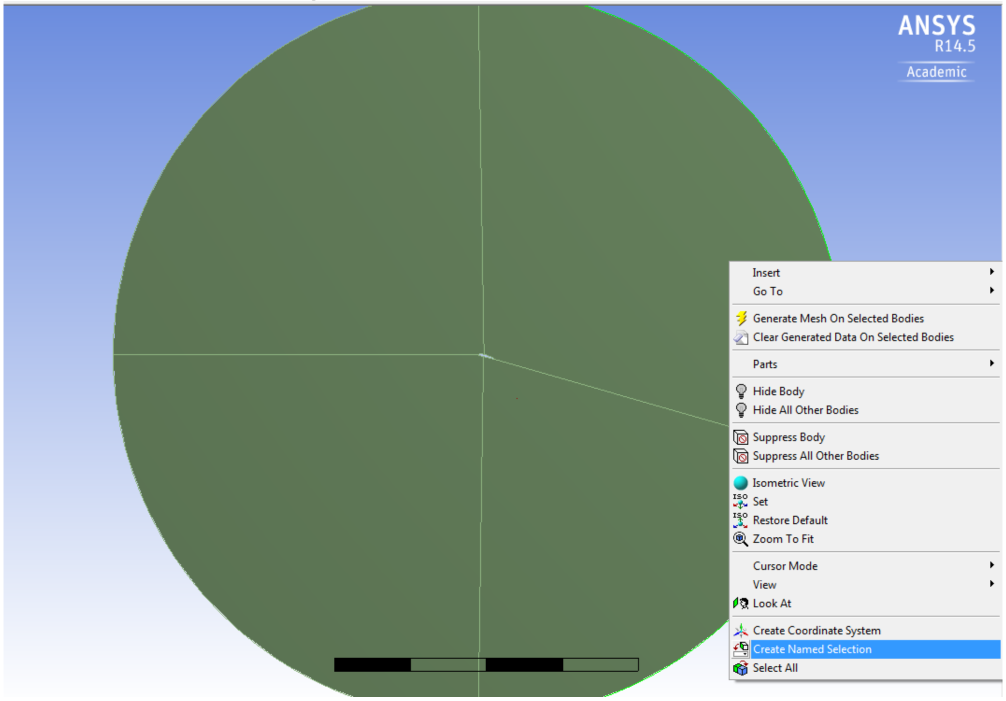
5.11. Click **Generate Mesh**. Click on the **Mesh** button under the **Outline** and make sure it resembles the mesh below.



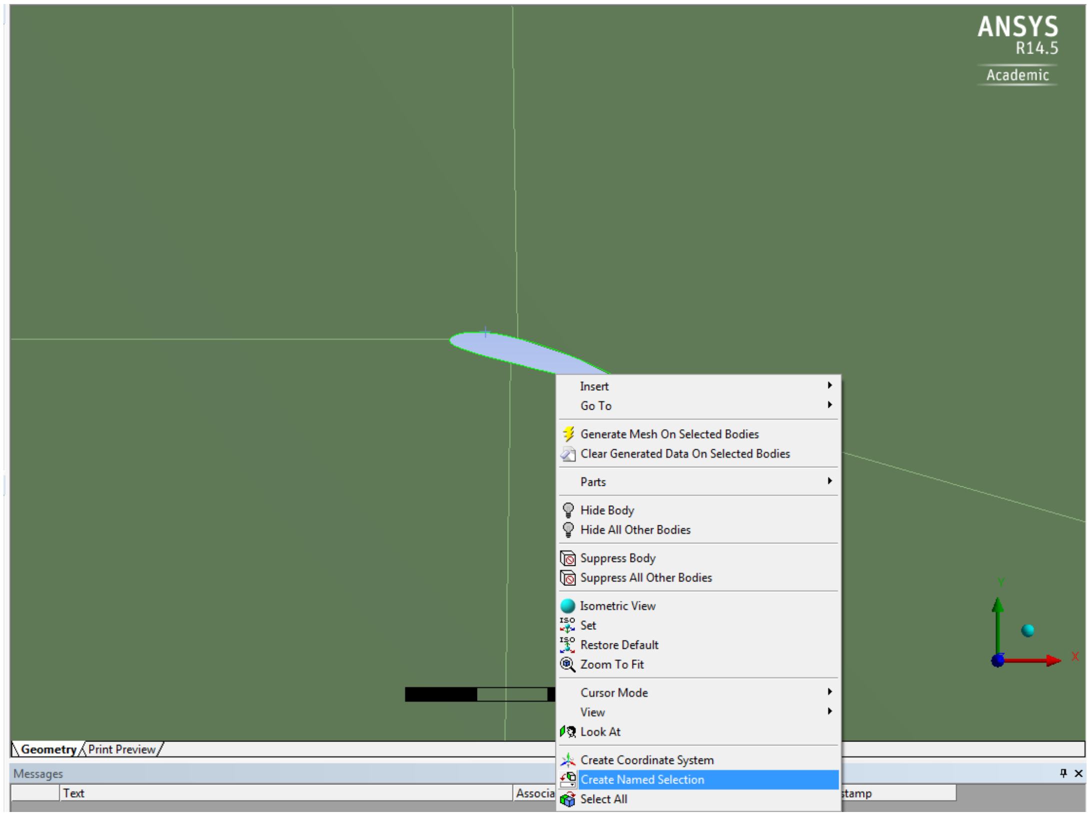
5.12. Hold Ctrl and select the two left most semicircle arcs, right click on them and select **Create Named Selection**, name the selection *inlet*. Use the edge select button from the toolbar.



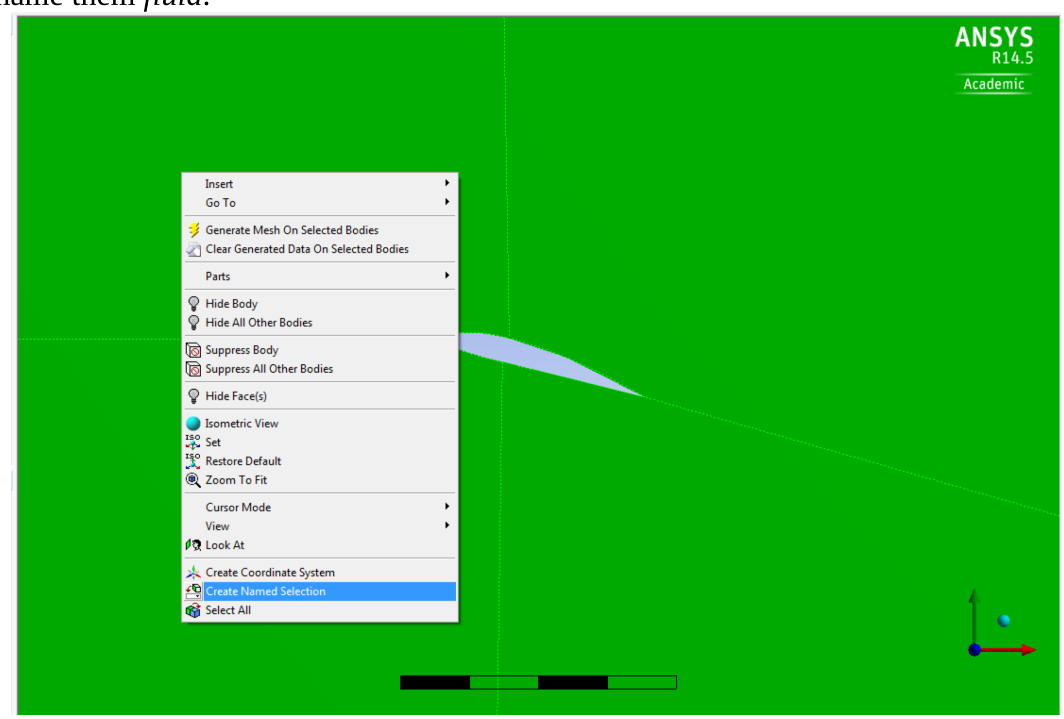
5.13. Do the same for the two right most semicircle arcs and name them *outlet*.



5.14. Select the four arcs 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*.

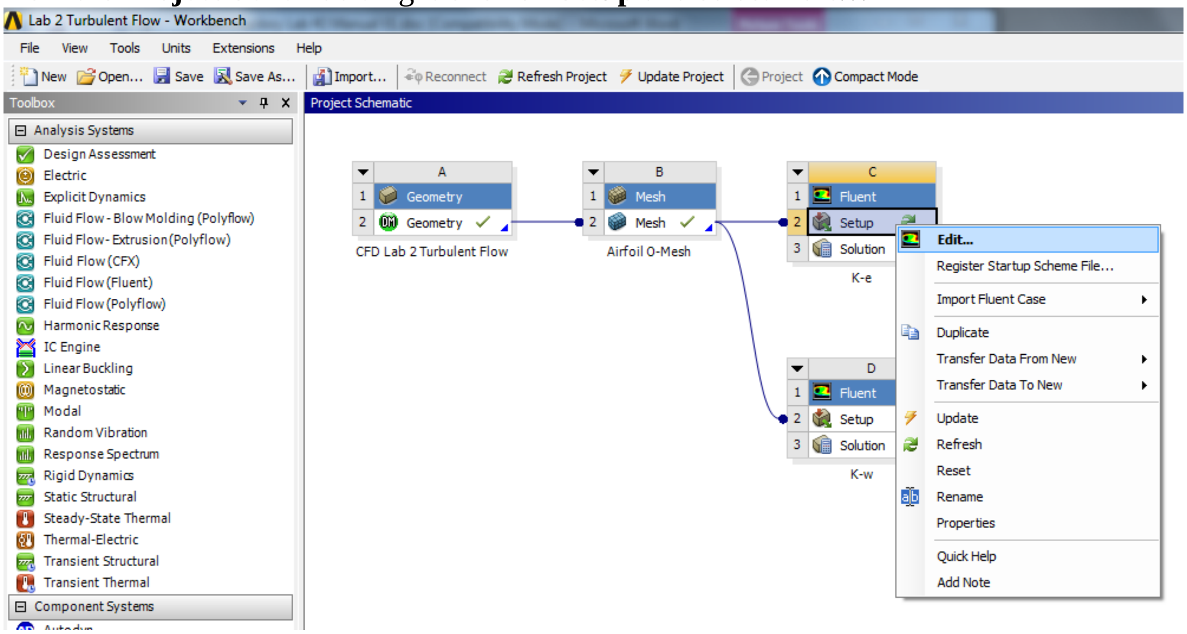


5.16. **File > Save Project.** Exit the window.

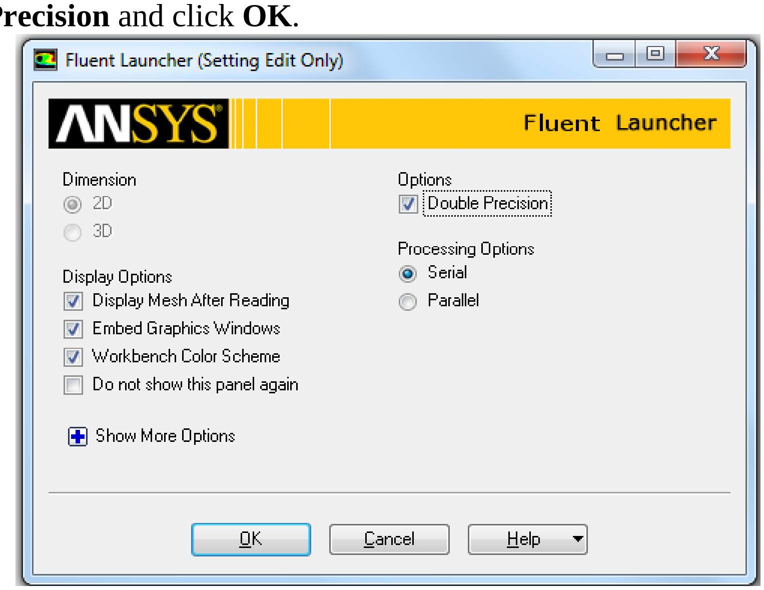
5.17. Right click **Mesh** and select **Update** from the dropdown menu.

6. Physics

6.1. From the **Project Schematic** right click on **Setup** and select **Edit…**

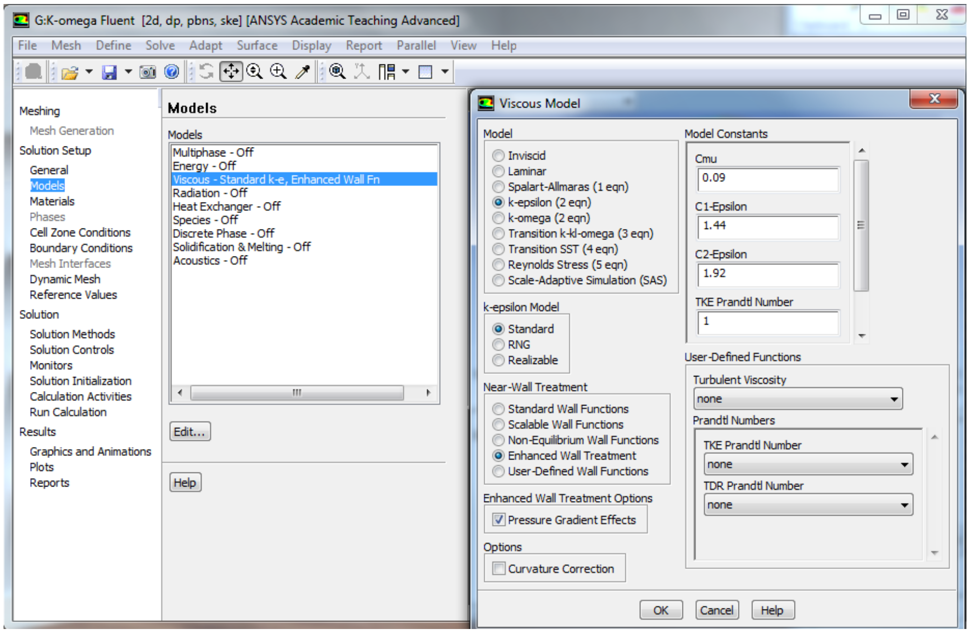


6.2. Select **Double Precision** and click **OK**.

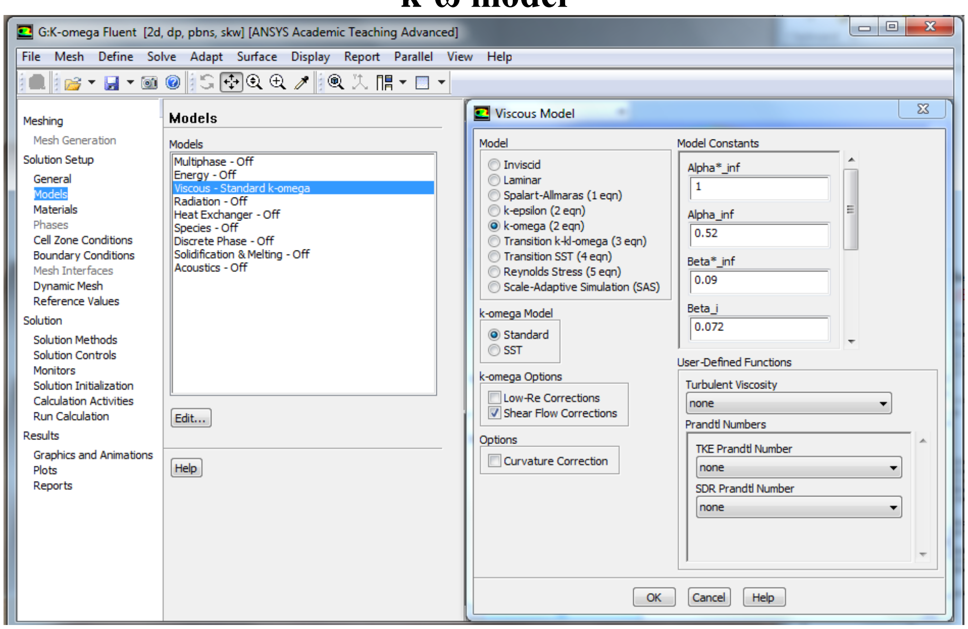


6.3. **Solution Setup** > **Models** > **Viscous –Laminar** > **Edit…** Change the parameters as per below and click **OK.** (For the k-ω case, you will select **k-omega (2 eqn)**.

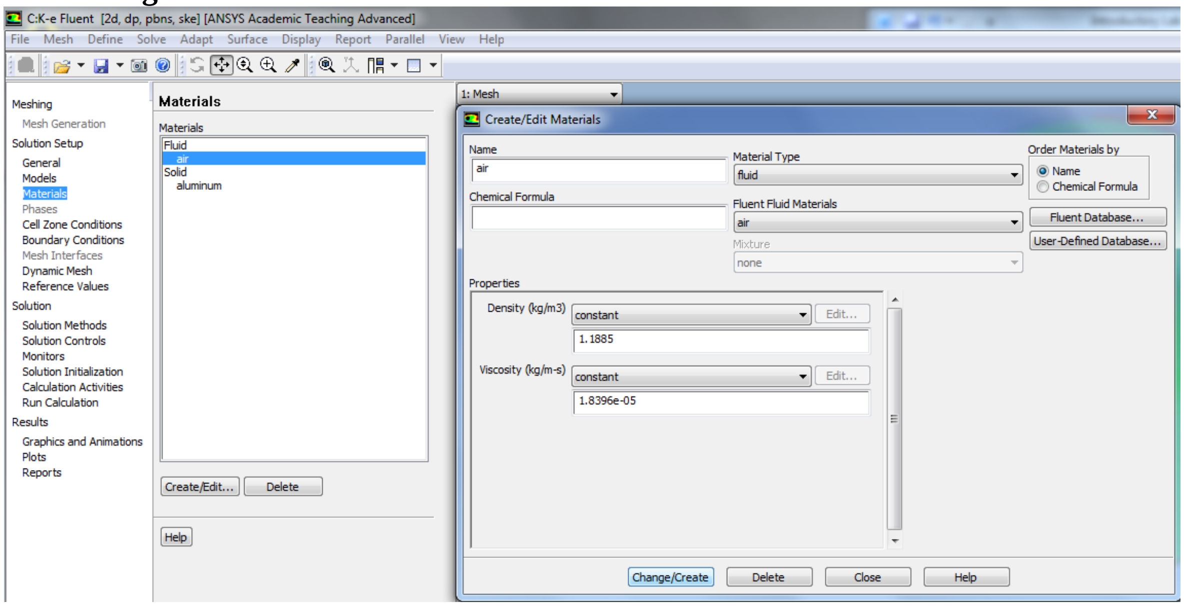
k-ε model



k-ω model



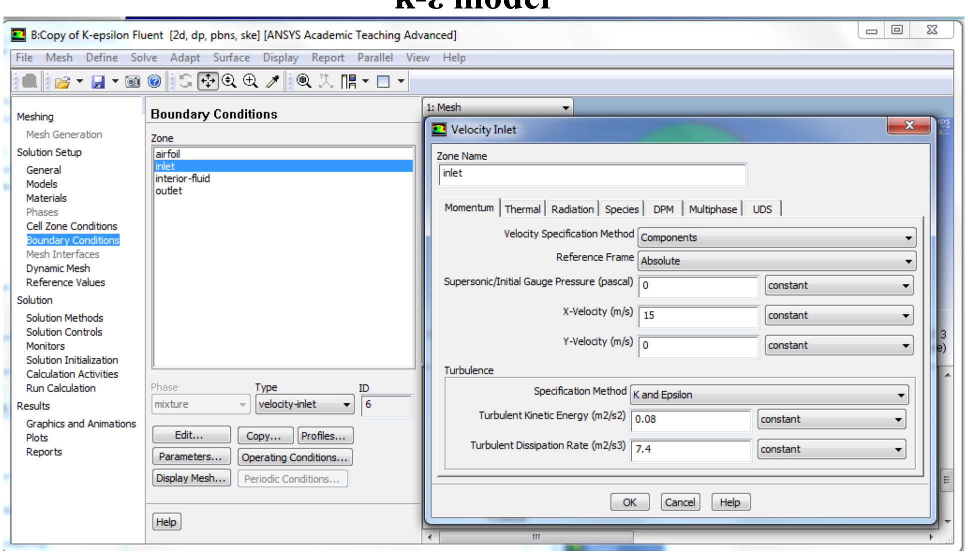
6.4. **Solution Setup** > **Materials** > **air** > **Create/Edit…** Change parameters as per experimental data and click **Change/Create**.



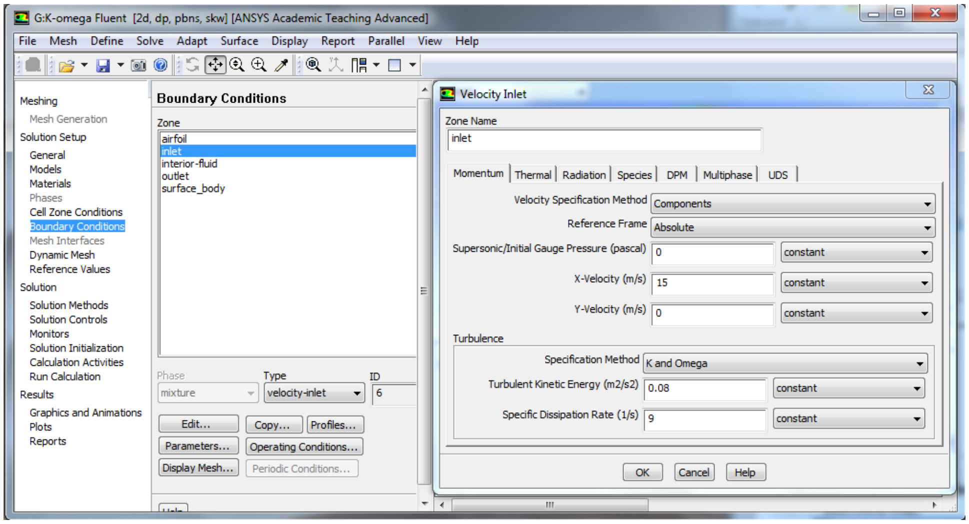
Use the air properties at the **room temperature** when you conducted EFD Lab3. **You can use the following website to calculate air properties from the temperature:**
http://www.mhtl.uwaterloo.ca/old/onlinetools/airprop/airprop.html
The values in the figure above are for 24° temperature.
NOTE: viscosity 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.

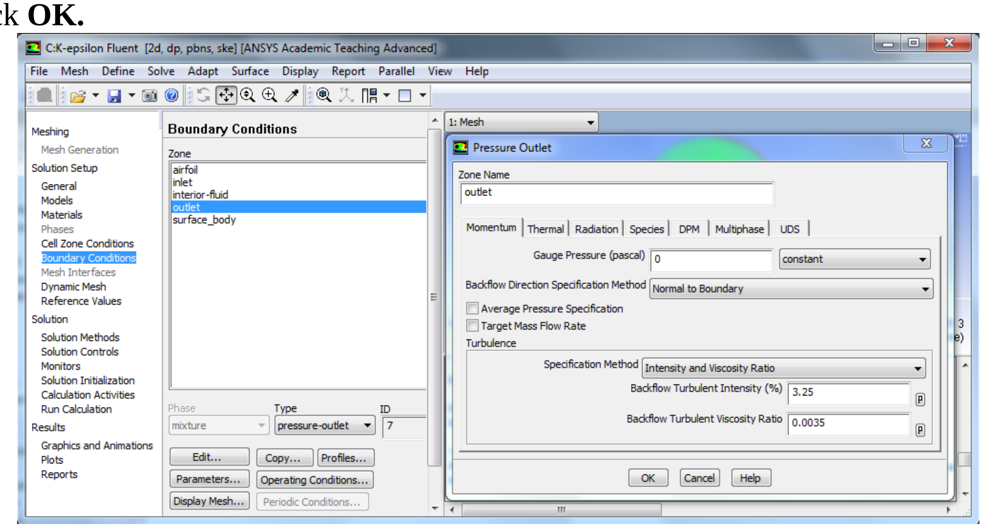
k-ε model



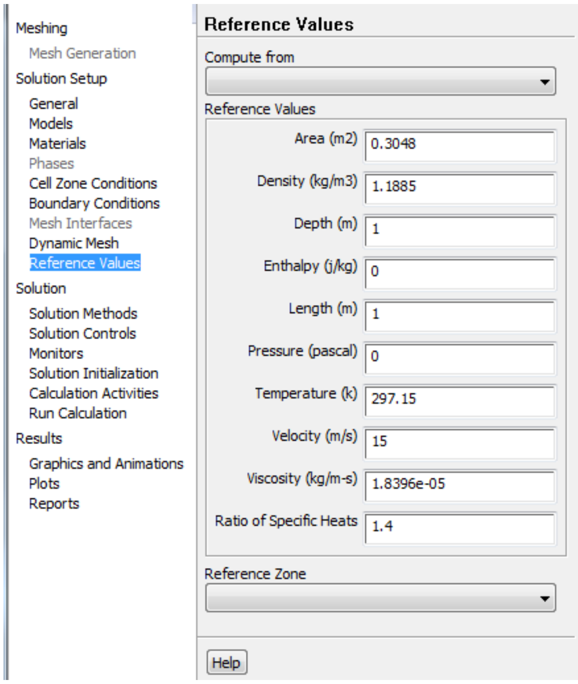
k-ω model



6.6. **Solution Setup** > **Boundary Conditions** > **Outlet** > **Edit…** Change the parameters as per below and click **OK.**

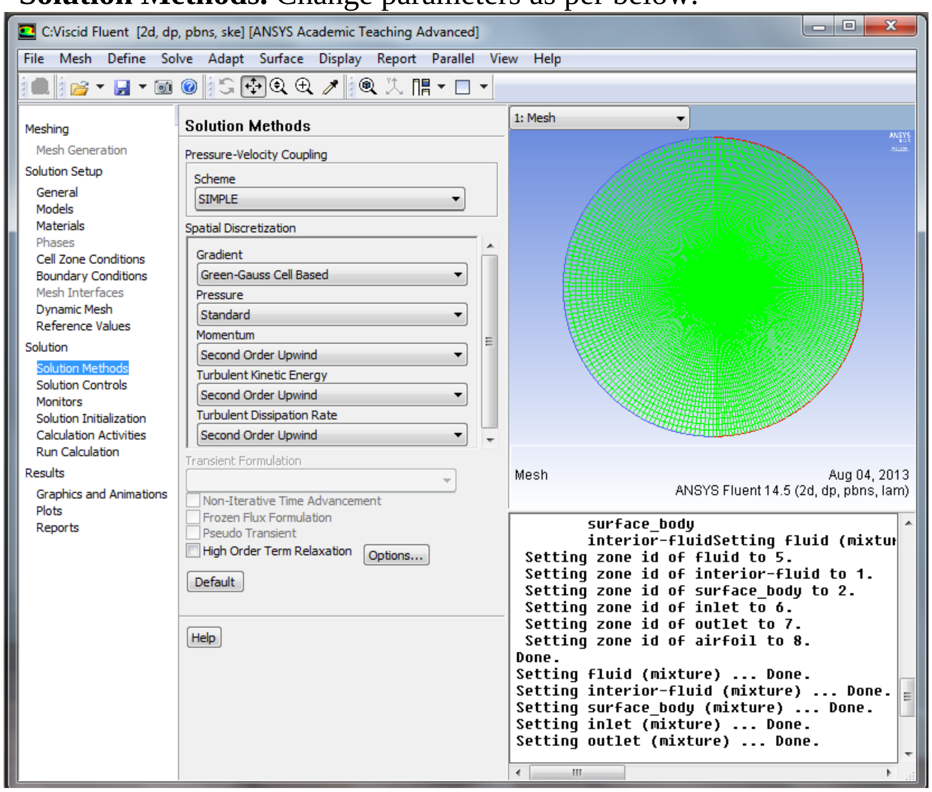


6.7. **Solution Setup** > **Reference Values**. Change parameters as per below. The **Velocity**, **Temperature**, **Density**, and **Viscosity** should be entered from EFD data.

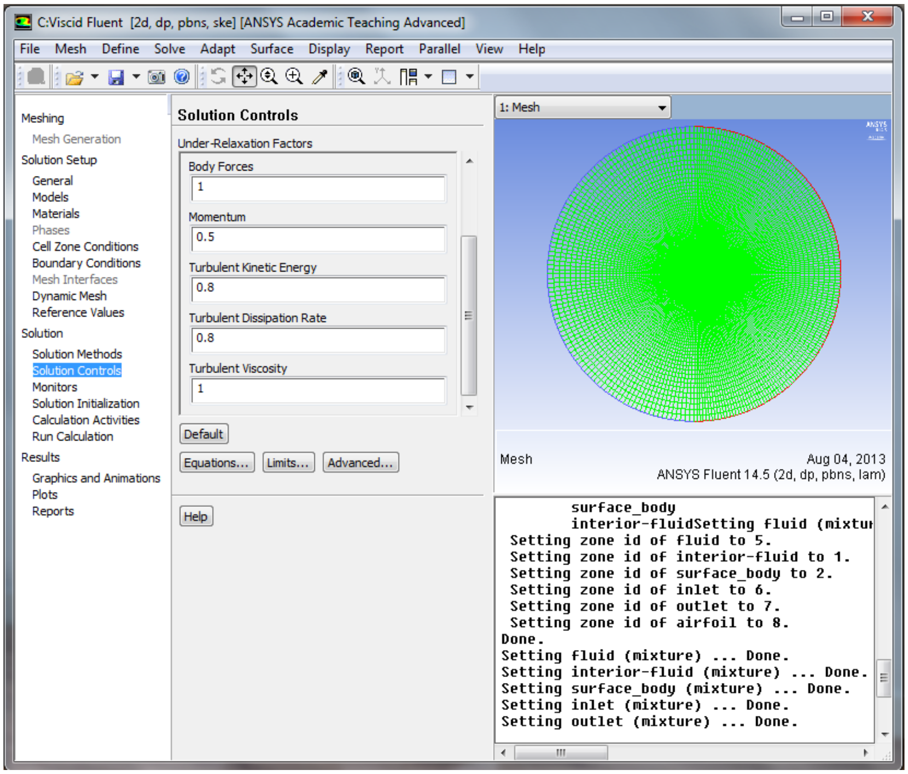


7. Solution

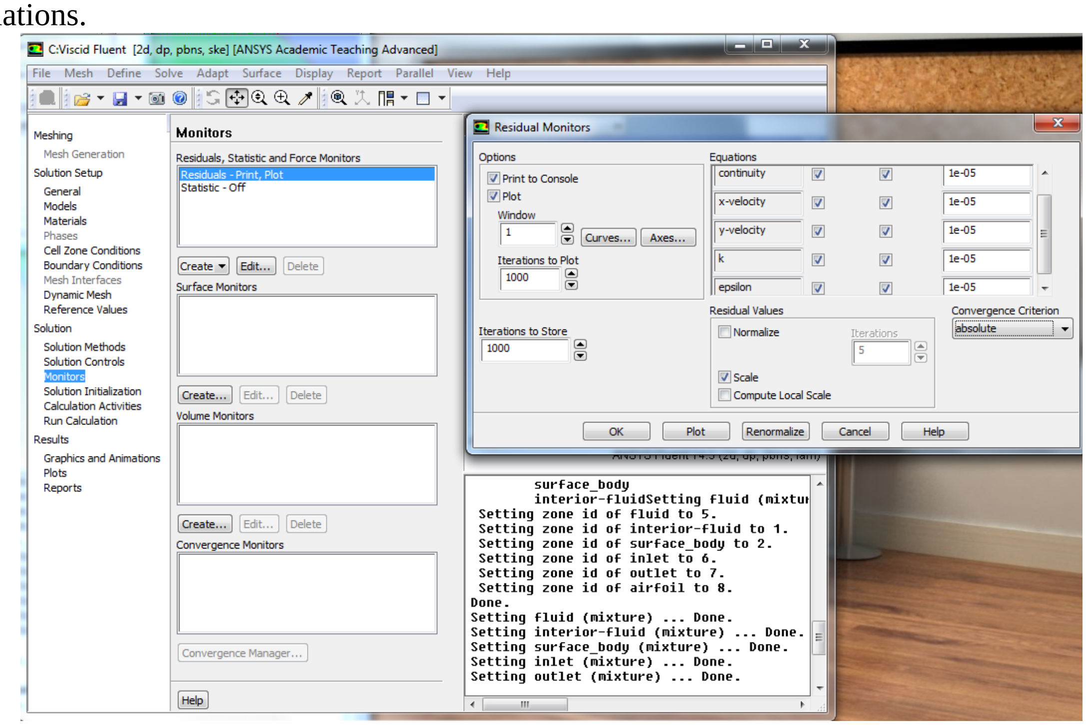
7.1. **Solution** > **Solution Methods.** Change parameters as per below.



7.2. **Solution > Solution Controls**. Change Parameters as per below. (If you have problems with the solution converging, you can decrease the **Under –Relation Factors**.)

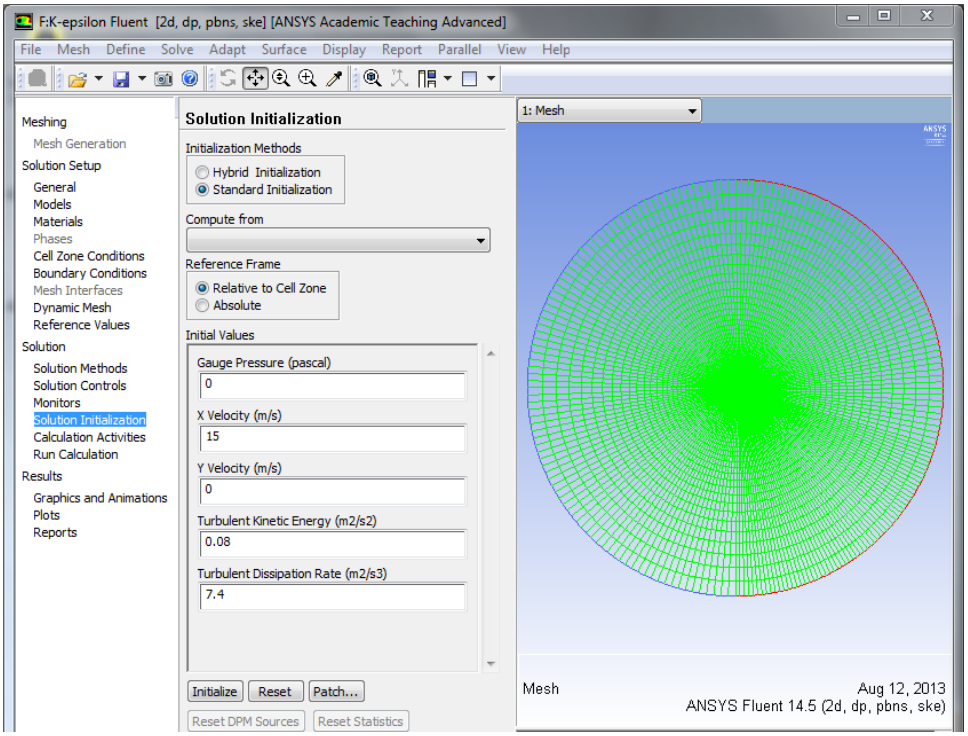


7.3. **Solution > Monitors > Residuals –Print, Plot.** Change the convergence limit to 1e-05 for all five equations.

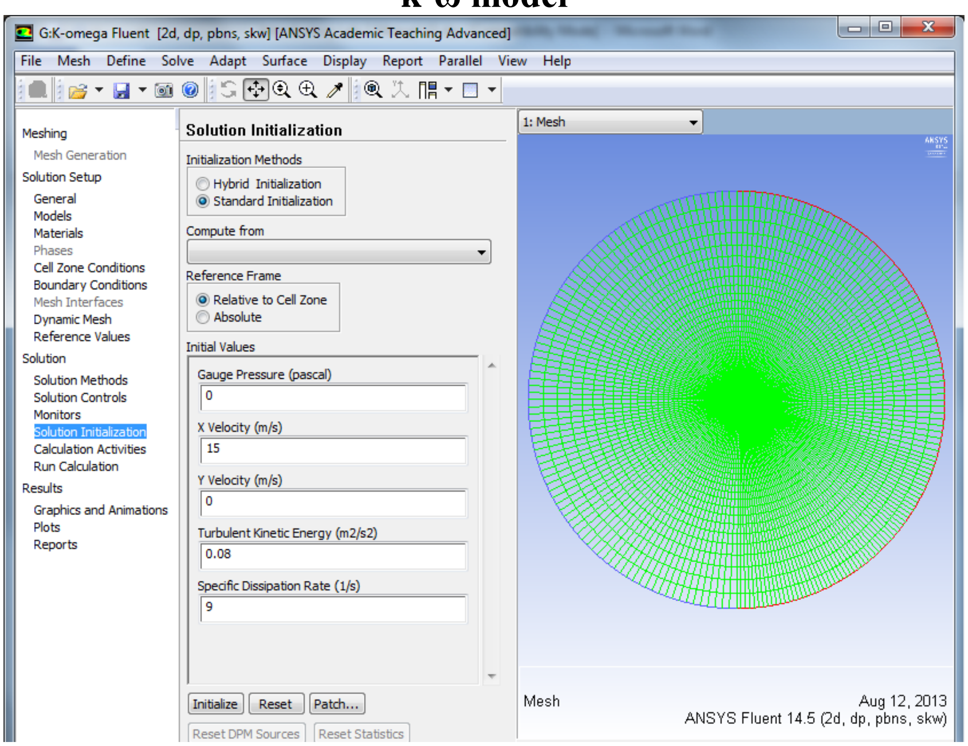


7.4. **Solution > Solution Initialization.** Change the parameters as per below and click **Initialize.**

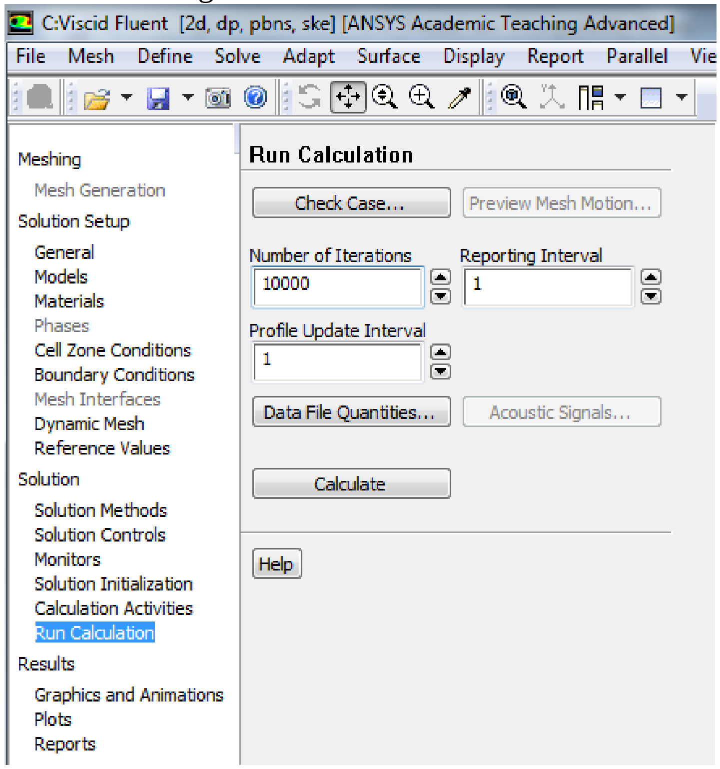
k-ε model



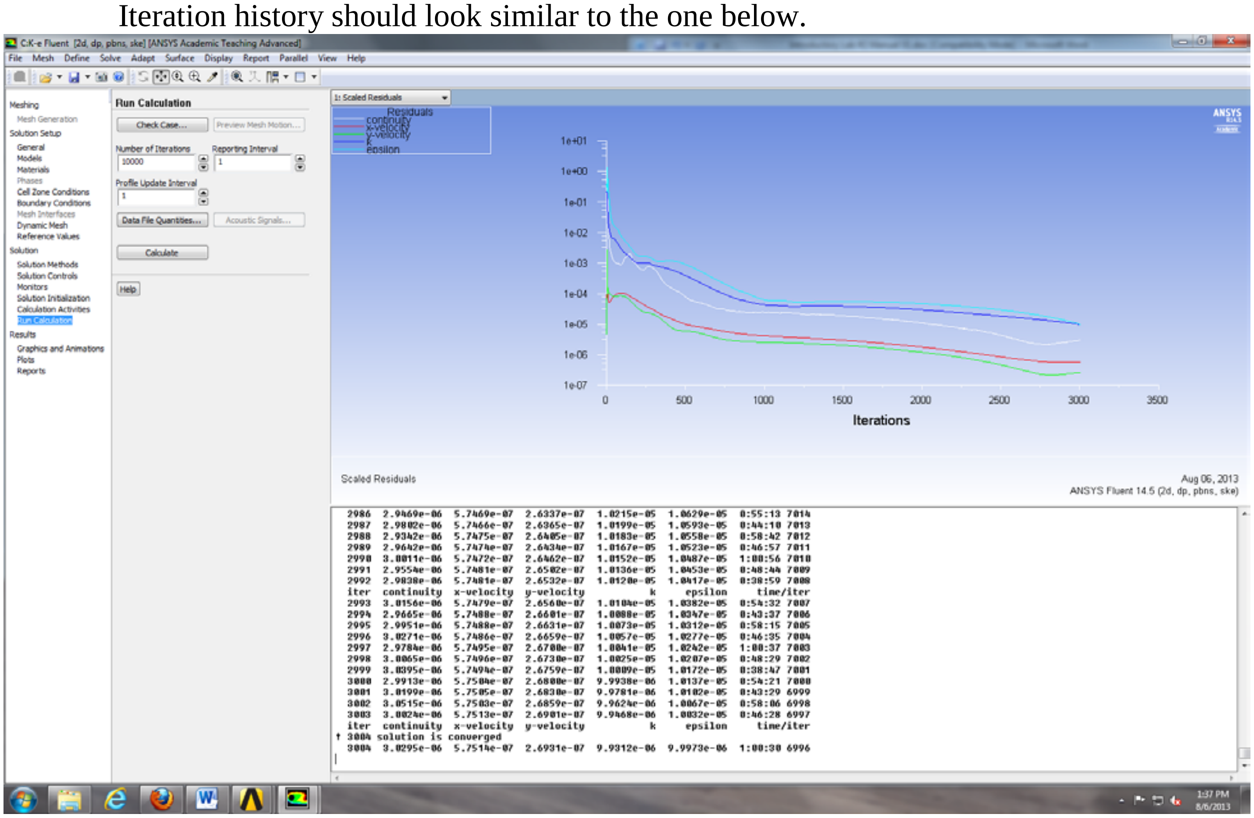
k-ω model



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



Iteration history should look similar to the one below.

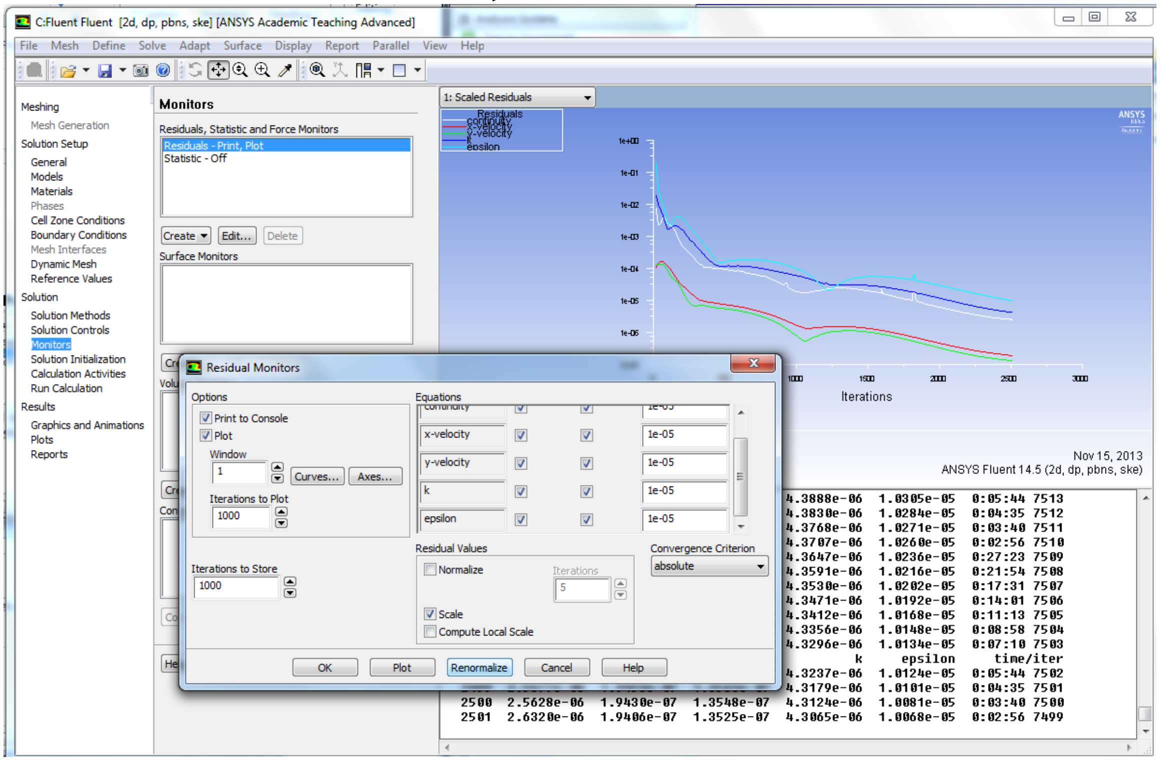


7.6. **File** > **Save Project.**

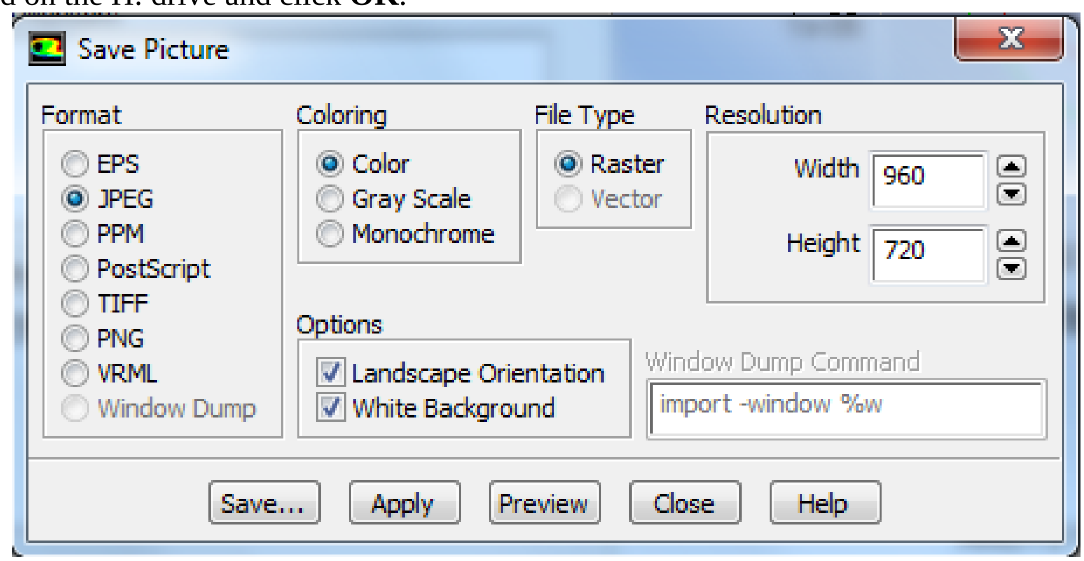
8. Results

8.1. Plotting and Saving Residual History

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

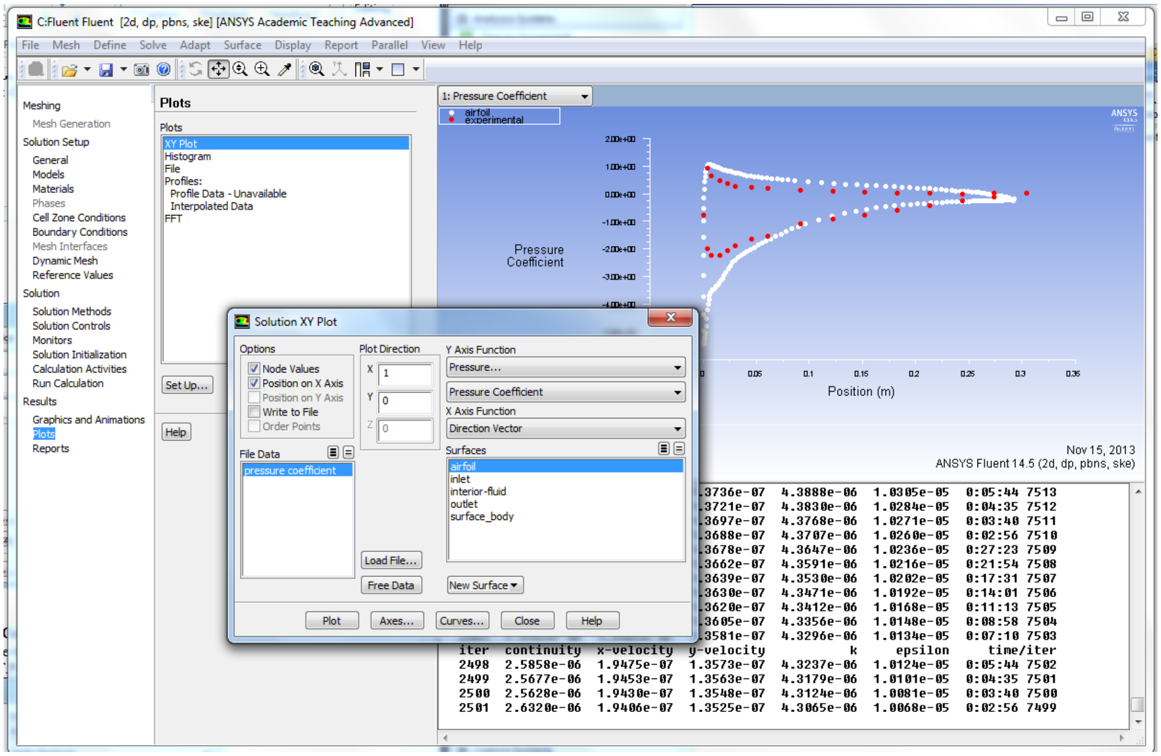


File > **Save Picture…** > **Save…** Make sure the parameters are as per below and click **Save…** Name the file *CFD Lab 2 Residual History* change the file directory to the CFD Lab 2 file you created on the H: drive and click **OK**.



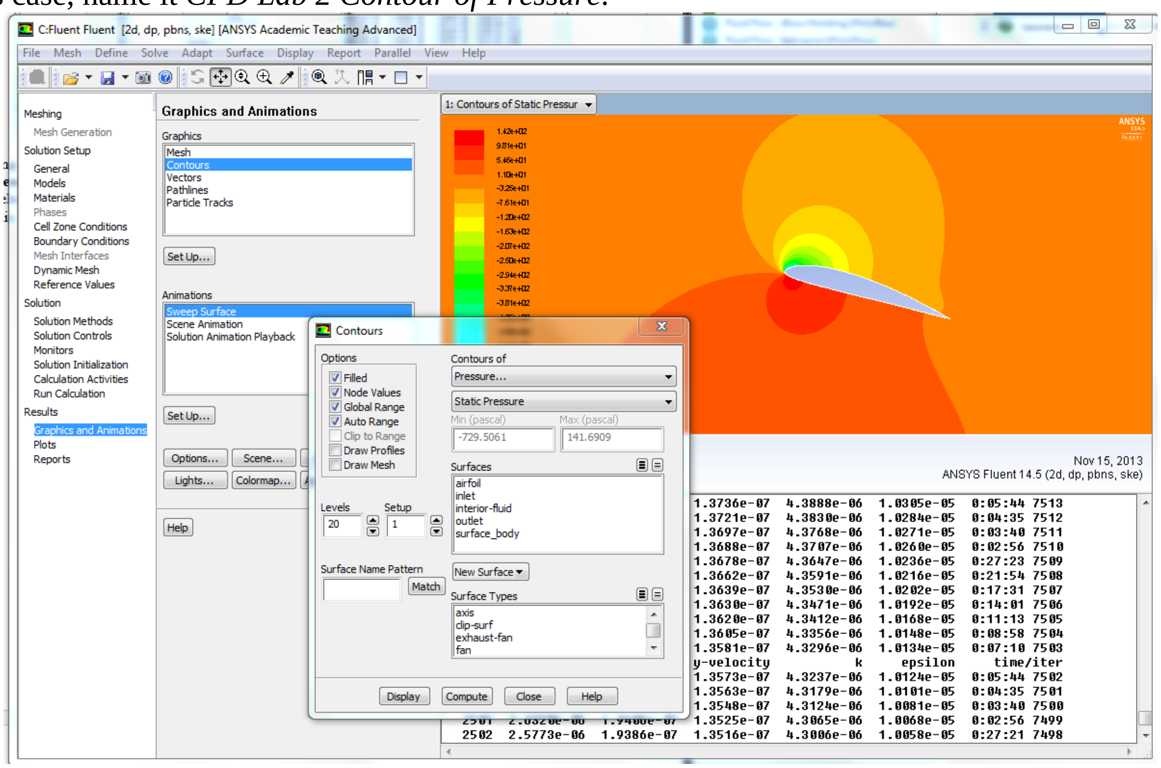
8.2. Plotting Pressure Coefficient Distribution with CFD and EFD Data

Results > **Plots** > **XY Plot** > **Set Up…** > **Load File…** Select Pressure-coef-attack16.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 Lab 2 Pressure Coefficient Distribution*.



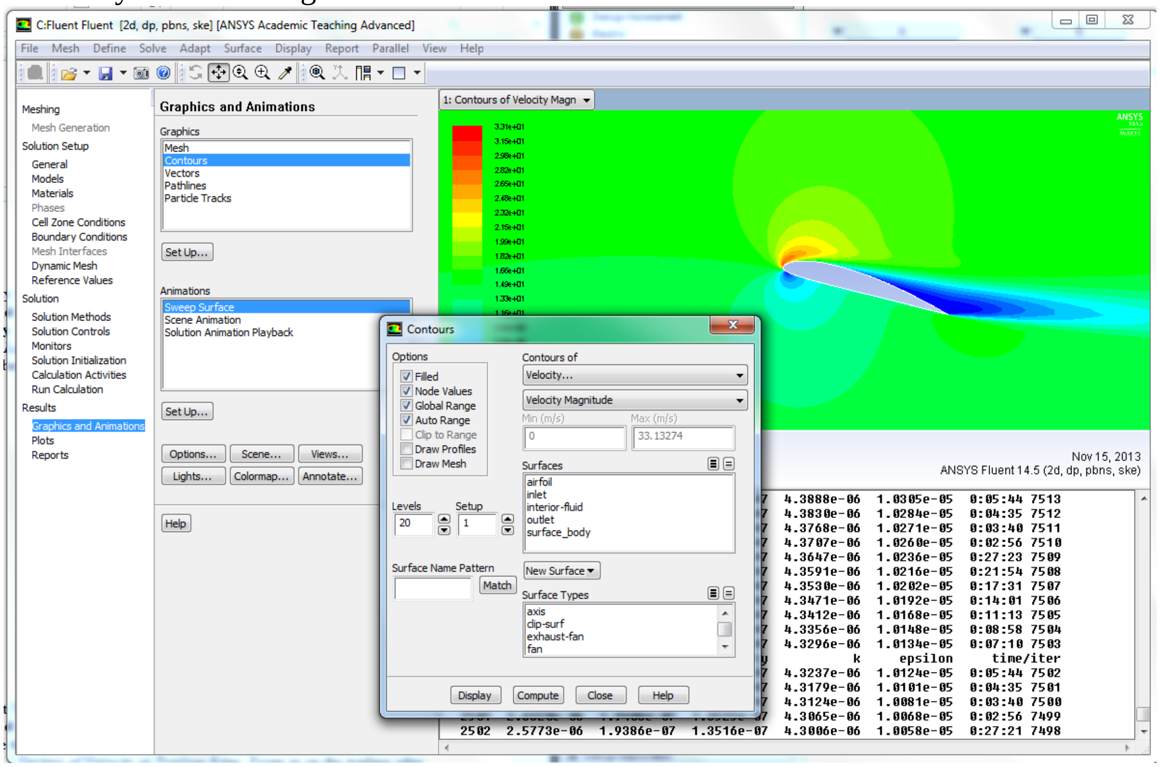
8.3. Plotting Contour of Pressure

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 Lab 2 Contour of Pressure*.



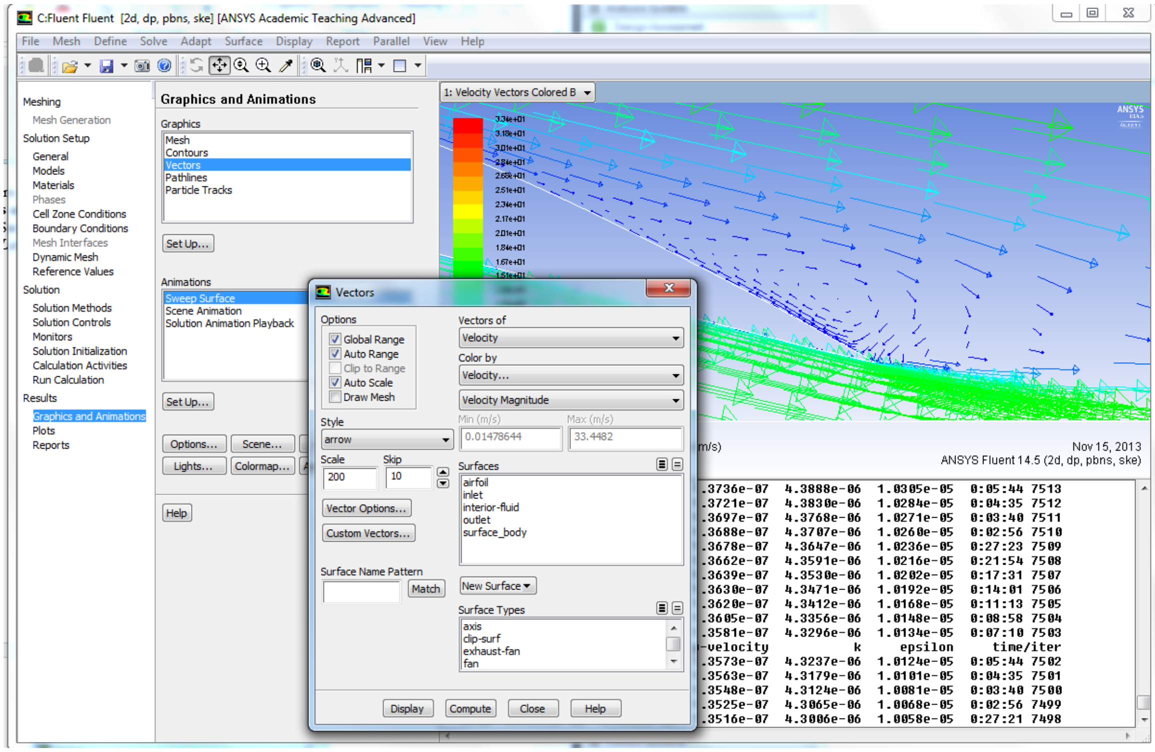
8.4. Plotting Contour of Velocity Magnitude

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 Lab 2 Contour of Velocity Magnitude*. Zoom in where you can see the airfoil clearly and the change in contour levels around the airfoil.



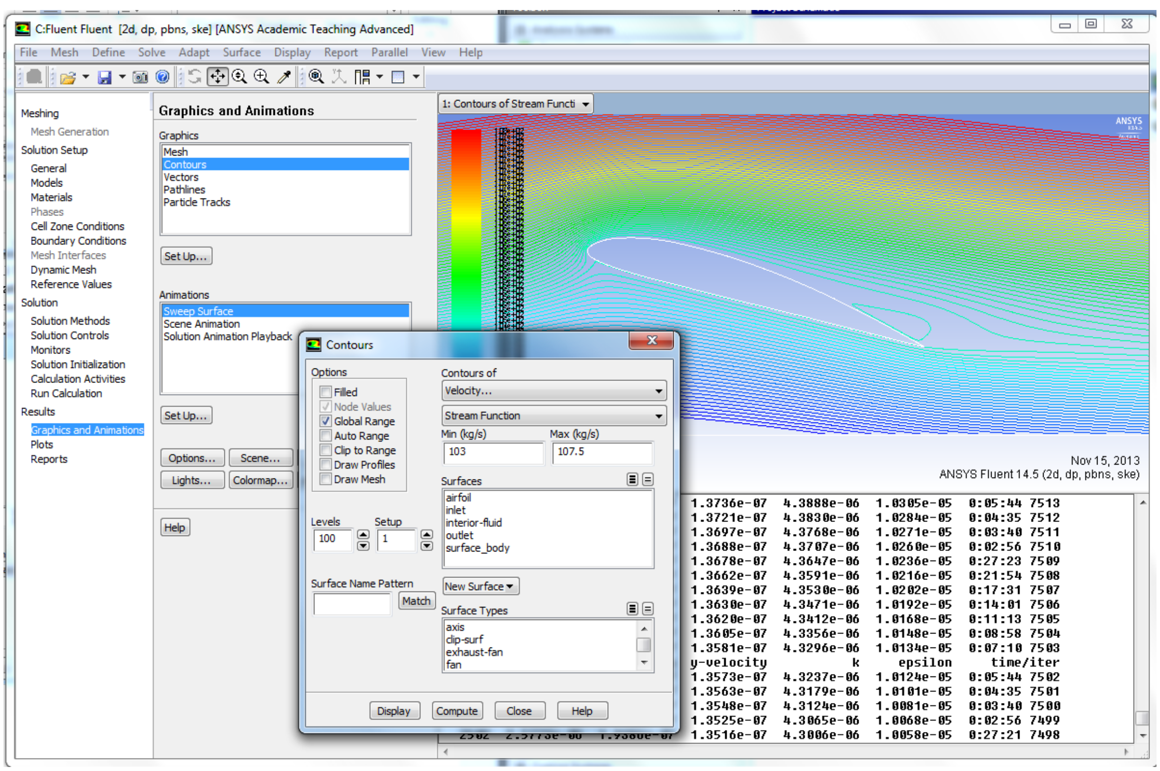
8.5. Plotting Velocity Vectors at Trailing Edge

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 Lab 2 Vectors of Velocity at Trailing Edge*. Zoom in on the trailing edge.



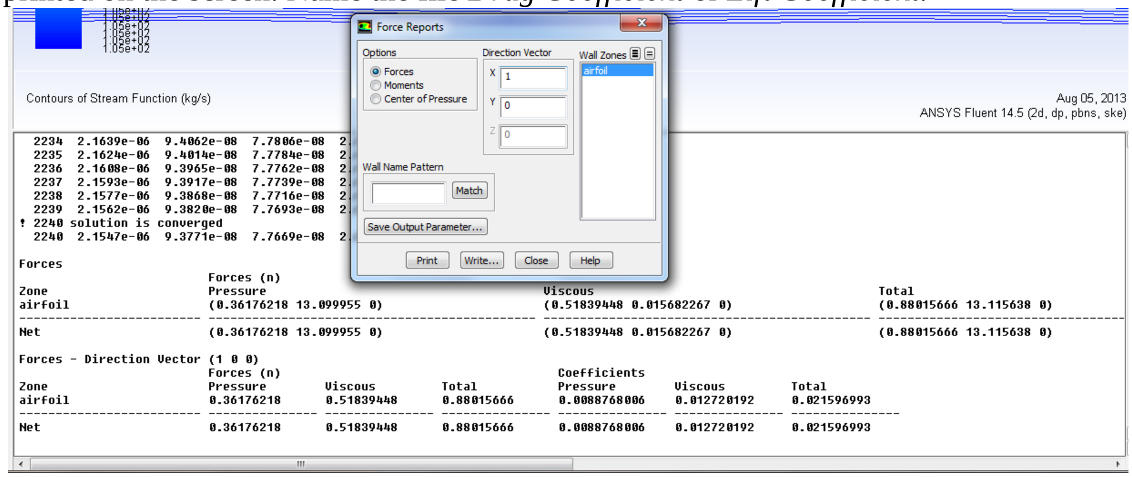
8.6. Plotting Streamlines Close to Airfoil Surface

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 Lab 2 Streamlines Close to Surface*.



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 zero and the Y parameter to 1, this prints out the lift force. Save the coefficients by clicking **Write**. This creates a text file of what was printed on the screen. Name the file *Drag Coefficient* or *Lift Coefficient.*



9. Exercises

Parametric Studies 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.
- Use “CFD Lab2 Report Template.doc” to save the figures and data for each exercise below.
- **Use the benchmark EFD data from the class website if your EFD Lab 3 data are with a different angle of attack than 16 degrees.**

1. **Effect of angle of attack**.

   1.1. Use the same flow conditions as those in your EFD Lab 3, including **geometry** (chord length) and **setup** (Flow properties, Reynolds number, inlet velocity), **EXCEPT use angle of attack 16 Degrees regardless of AOA in you EFD Lab 3**. Use **k-ε model**, **2nd order upwind scheme, double** precision with **iteration number (2000)** and **convergent limit (10^{-5})**.

   - **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.Velocity vectors, and 5. Streamlines
   - **Data need to be saved**: lift and drag coefficients

2. **Effect of turbulence models**

   2.1. Use the same conditions as those in exercise 1, **EXCEPT** using the “k-ω” for “viscous models”. Set up the boundary conditions following instructions part, set the iteration number to be (2000), and convergent limit to be 10^{-5}. Perform the simulation and compare solutions with the simulation results using "k-ε" model (you have finished in CFD PreLab2)

- **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. Velocity vectors, and 5. Streamlines
- **Data need to be saved**: lift and drag coefficients

3. Questions need to be answered in CFD Lab2 report:

Using the figures obtained in exercises 1 and 2 in this Lab and those figures you created in CFD PreLab2 to answer the following questions and present your answers in your CFD Lab 2 report.

Note: These questions are also available in the "CFD Lab2 Report Template" where you will type answers.

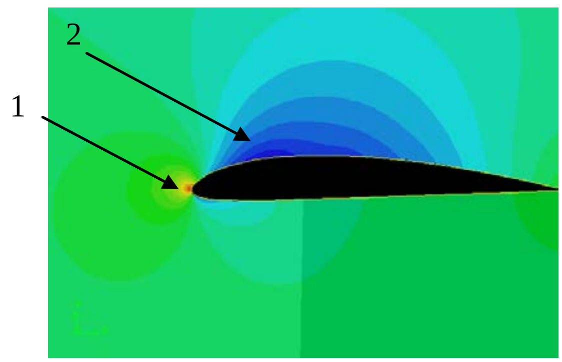
3.1. For Exercise 1 (effect of angle of attack):

(1). Which angle of attack simulation requires more iterations to converge?
(2). Which angle of attack produces higher lift/drag coefficients? Why?
(3). What is the effect of angle of attack on lift and drag coefficients?
(4). Describe the differences of streamline distributions near the trailing edge of airfoil surface for these two different angles of attack. Do you observe separations for both? If so, does the separation occur at the same location?

3.2. For Exercise 2 (different turbulence models):

(1). Do the two different turbulence models have the same convergence path? If not, which one requires more iterations to converge.
(2). Do the two different turbulence models predict the same results? If not, which model predicts more accurately by comparing with EFD data?

3.3. For the following contour plot, qualitatively compare the values of pressure and velocity magnitude at point 1 and 2, if the flow is from left to right. Which location has higher pressure and which location has higher velocity magnitude? Why?



3.4. Questions in CFD PreLab2.