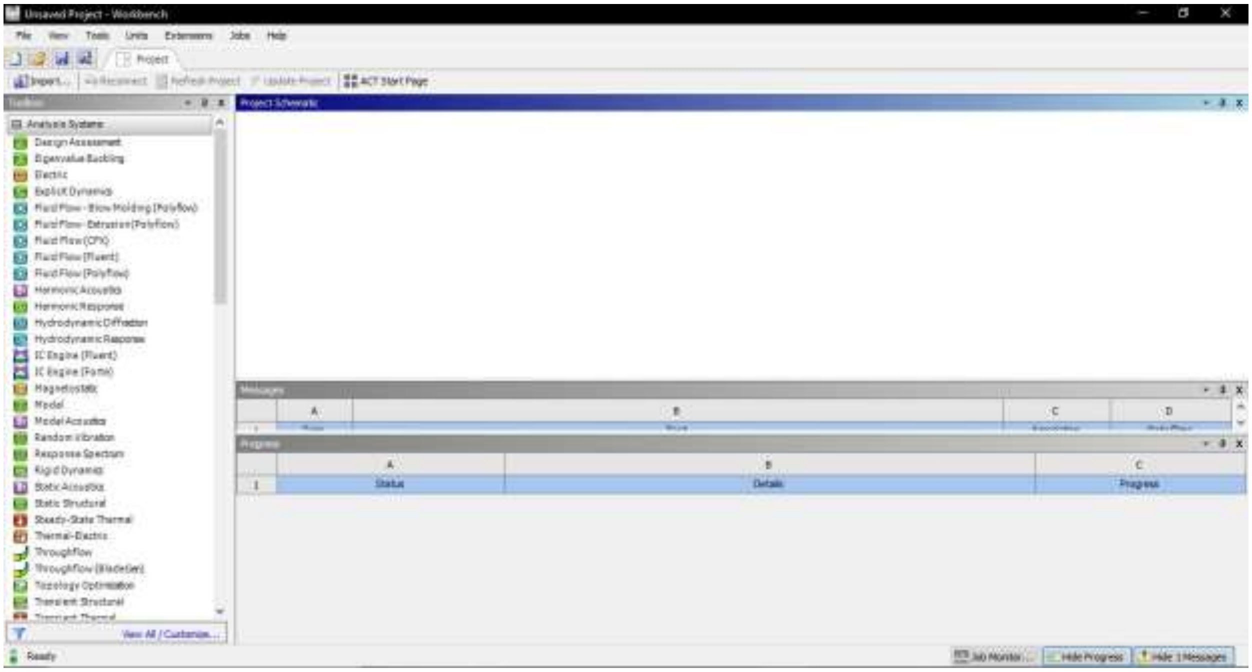
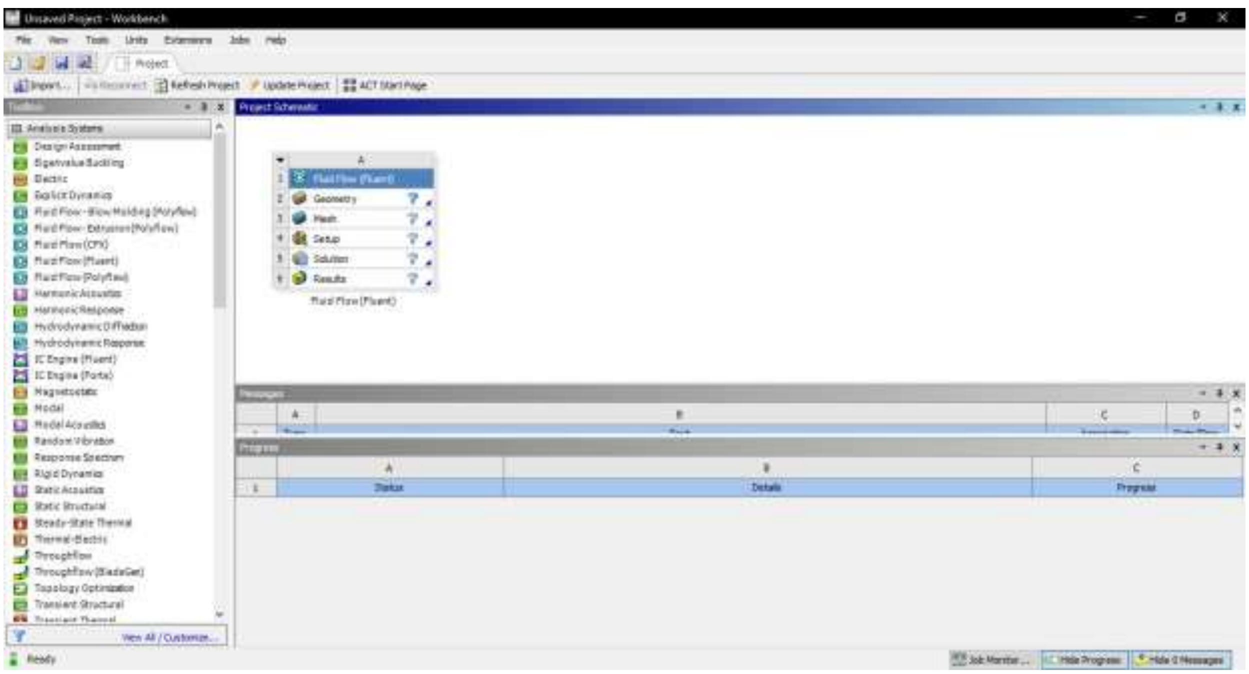
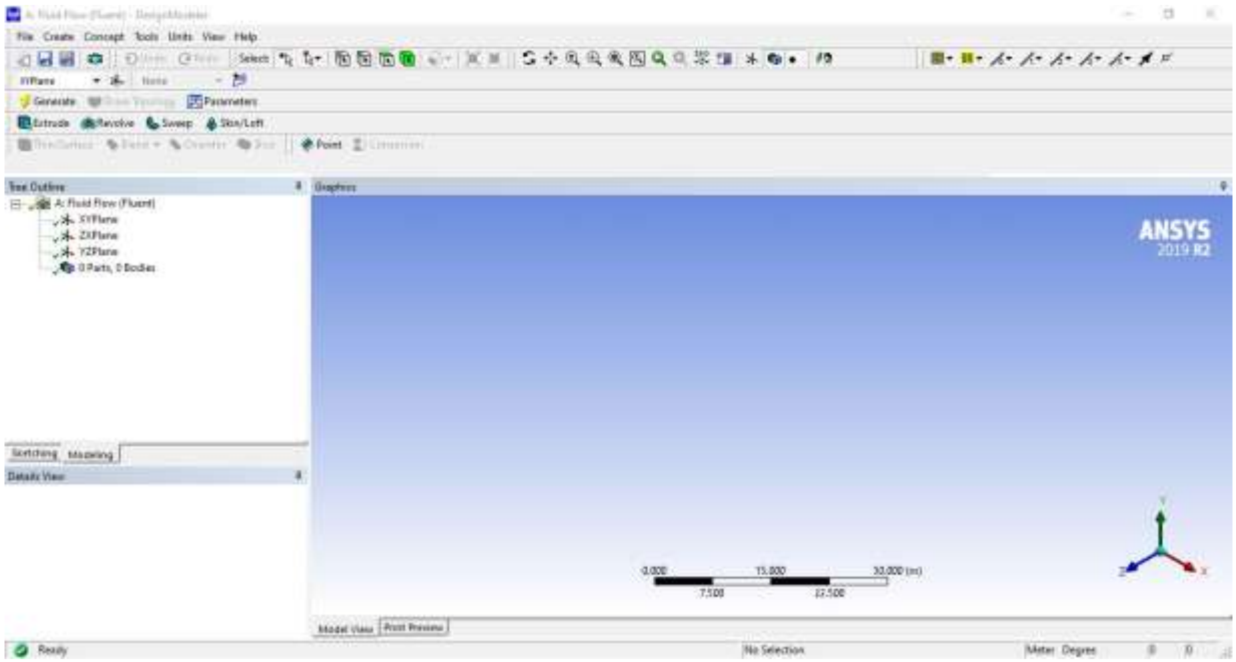
Step 1: Open Ansys Workbench



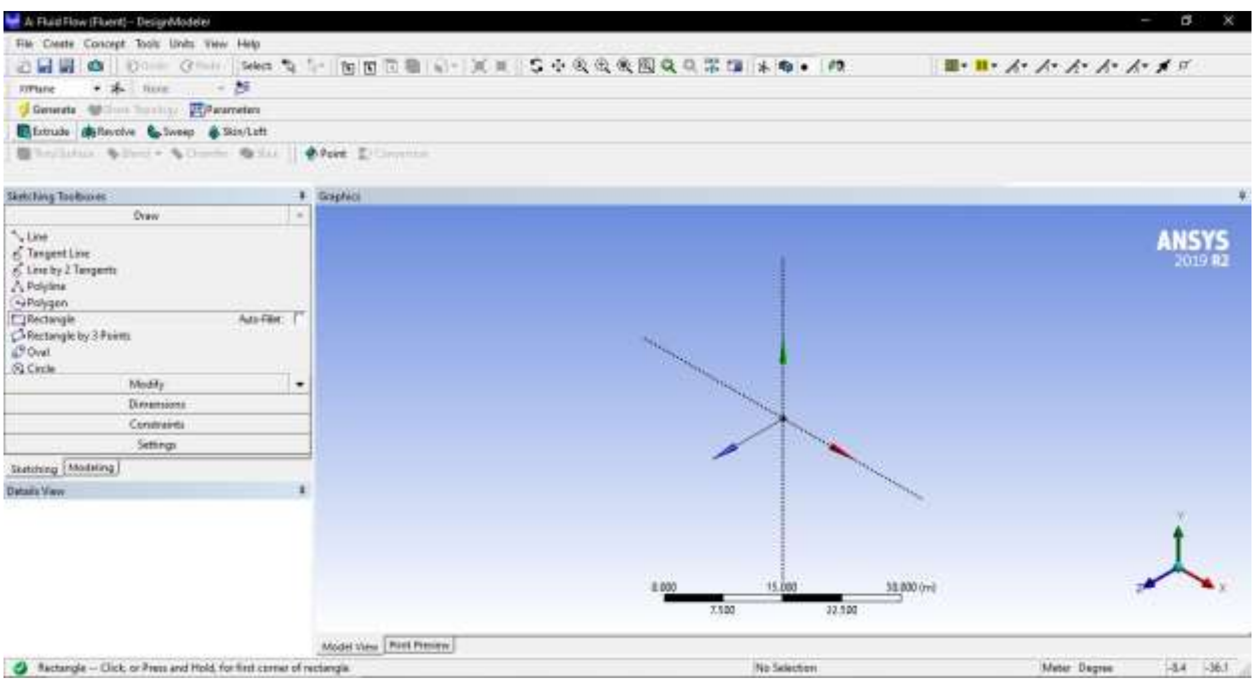
Step 2: Drag the Fluent Box into the Workbench Schematic Area



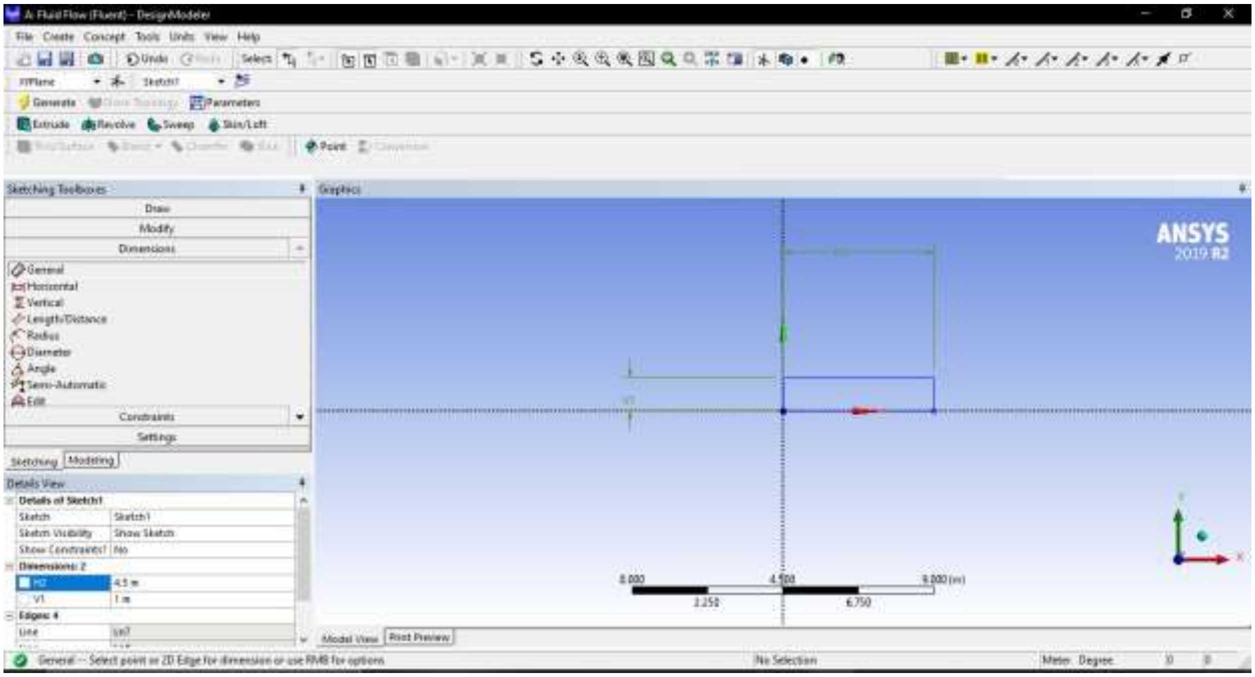
Step 3: RMB the Geometry and Edit Using Design Modeler



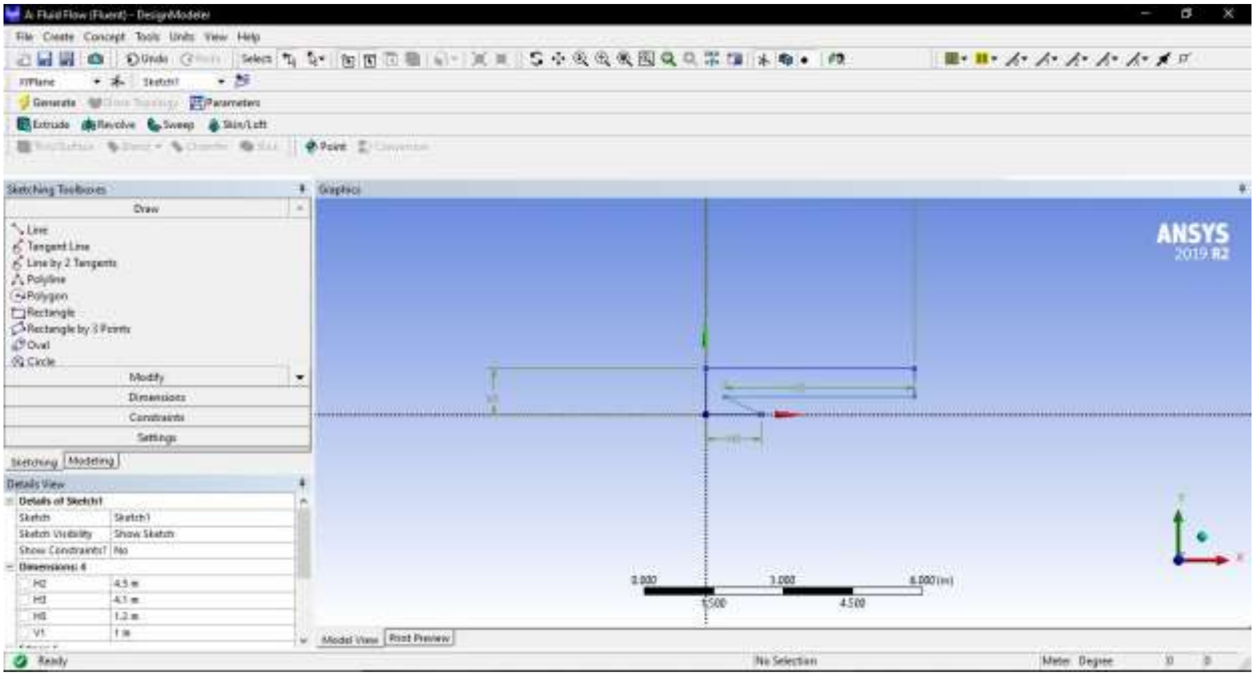
Step 4: Select the XY Plane and Select “Sketching”



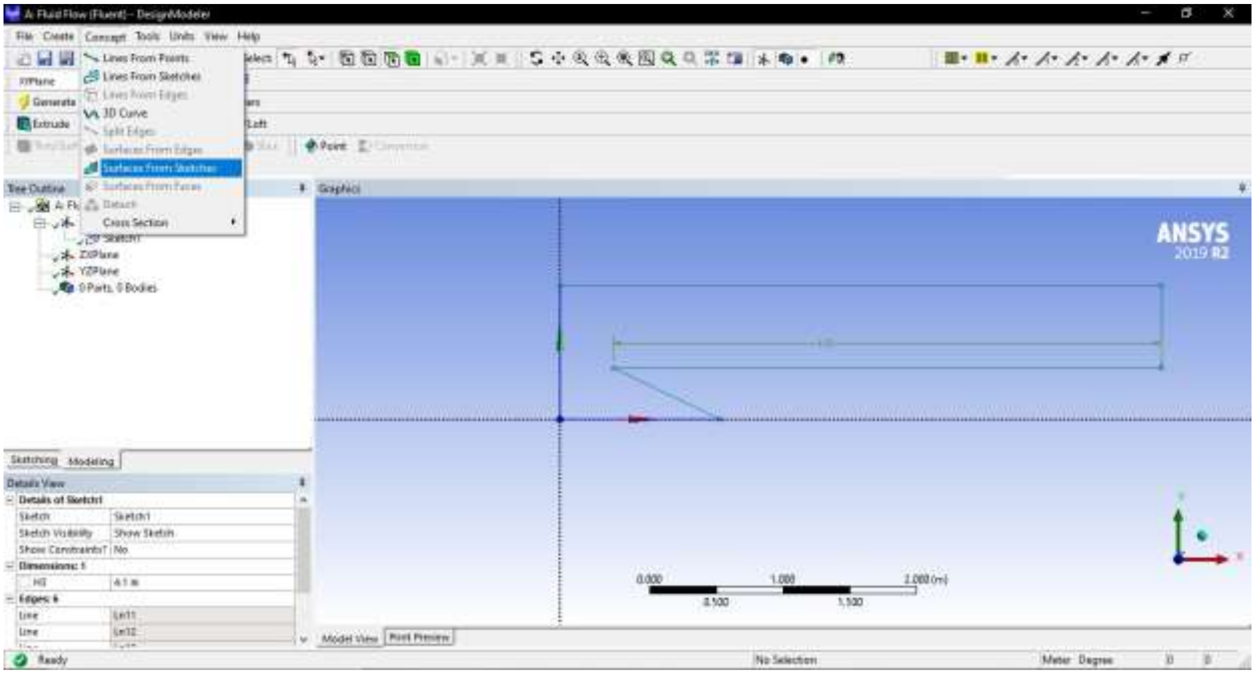
Step 5: Draw a Rectangle using the "Rectangle" Tool under the Draw Tab and Dimension this with the "General" Dimension Tool under the Dimensions Tab. Dimensions are shown in the Figure.



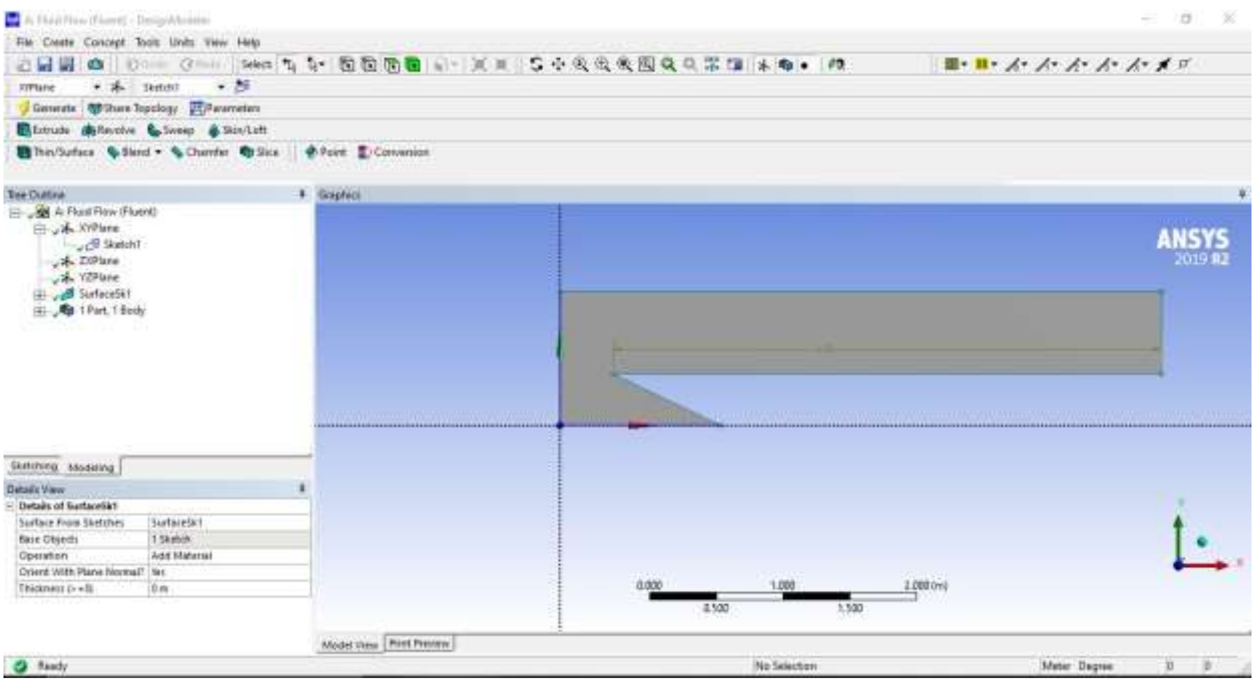
Step 6: Use the Line Tool to draw a horizontal line in the middle of the rectangle and a diagonal line to a form the figure shown below. Use the "Trim" Tool under the Modify tab to remove the excess lines and dimension as before using the dimensions shown in the figure.



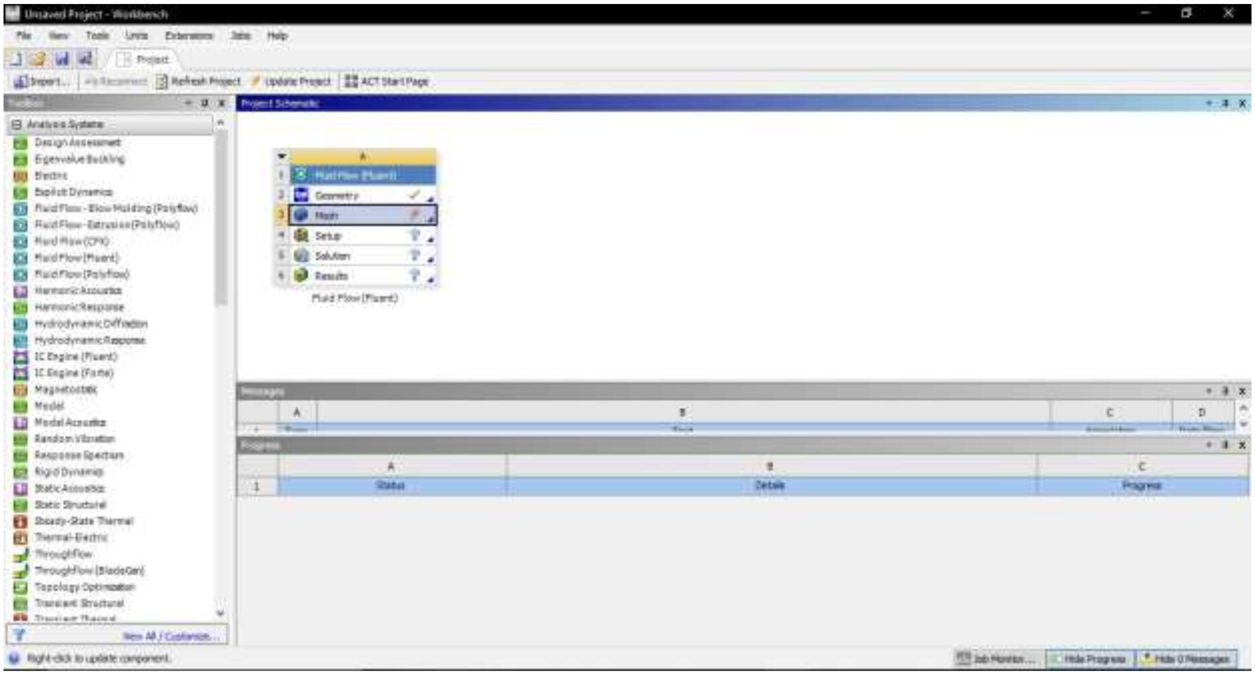
Step 7: Under the "Concept" Tab, select "Surfaces from Sketches". Select all edges of the drawing.



Step 8: Click "Generate" to generate the surface and close the Design Modeler.

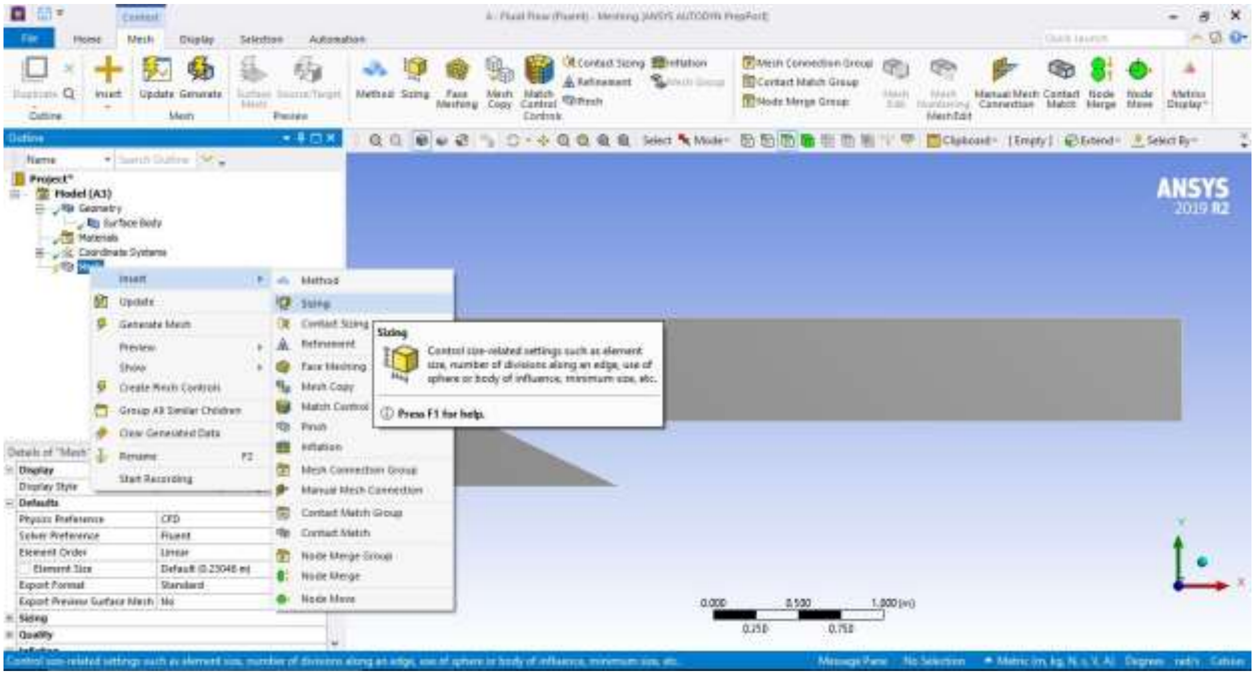


Step 9: Edit the Mesh by selecting RMB -> Edit

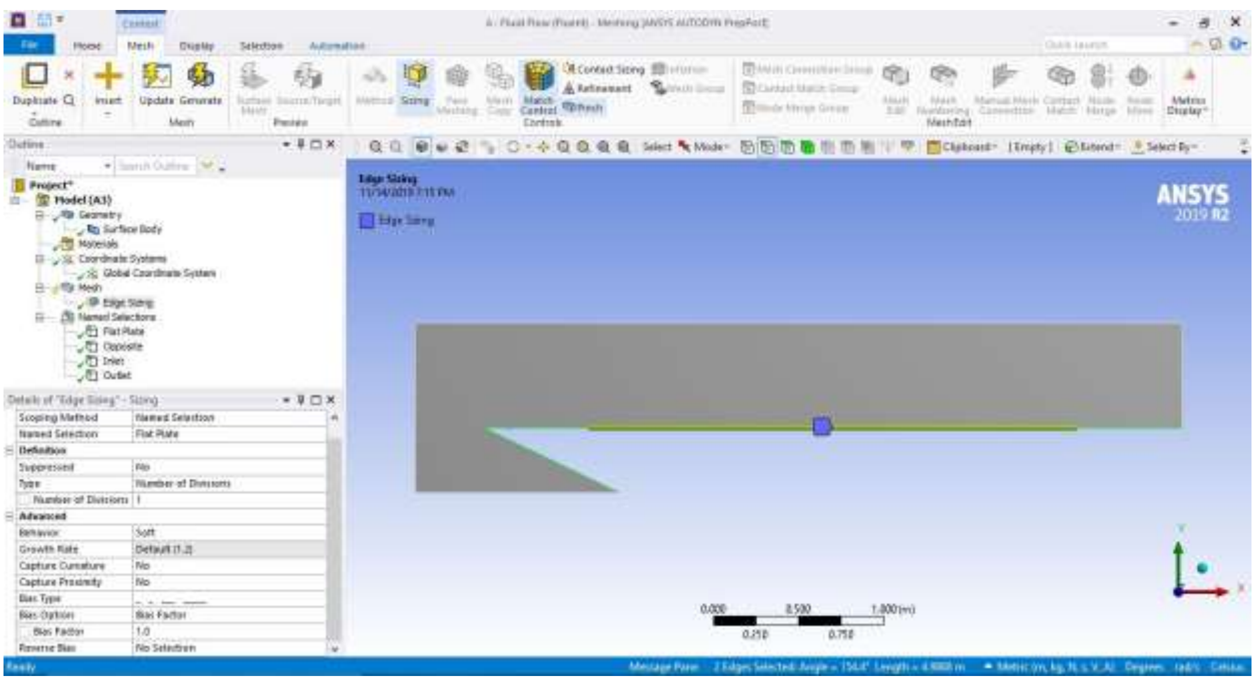


Step 10: Select the flat plate and RMB -> Create Named Selection; call this “Flat Plate”. Repeat this process for the other edges, such as the Inlet, Outlet, and Opposite (Left, Right, and Top, respectively)

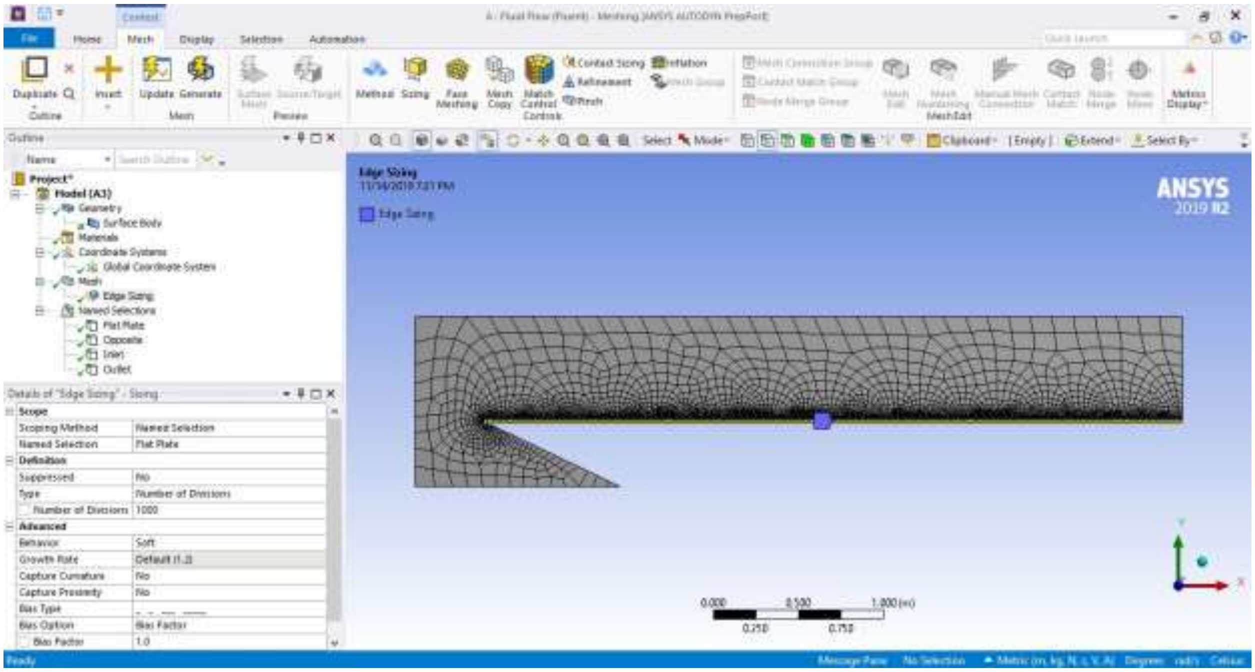
Step 11: RMB Mesh -> Insert -> Sizing, to define a mesh size.



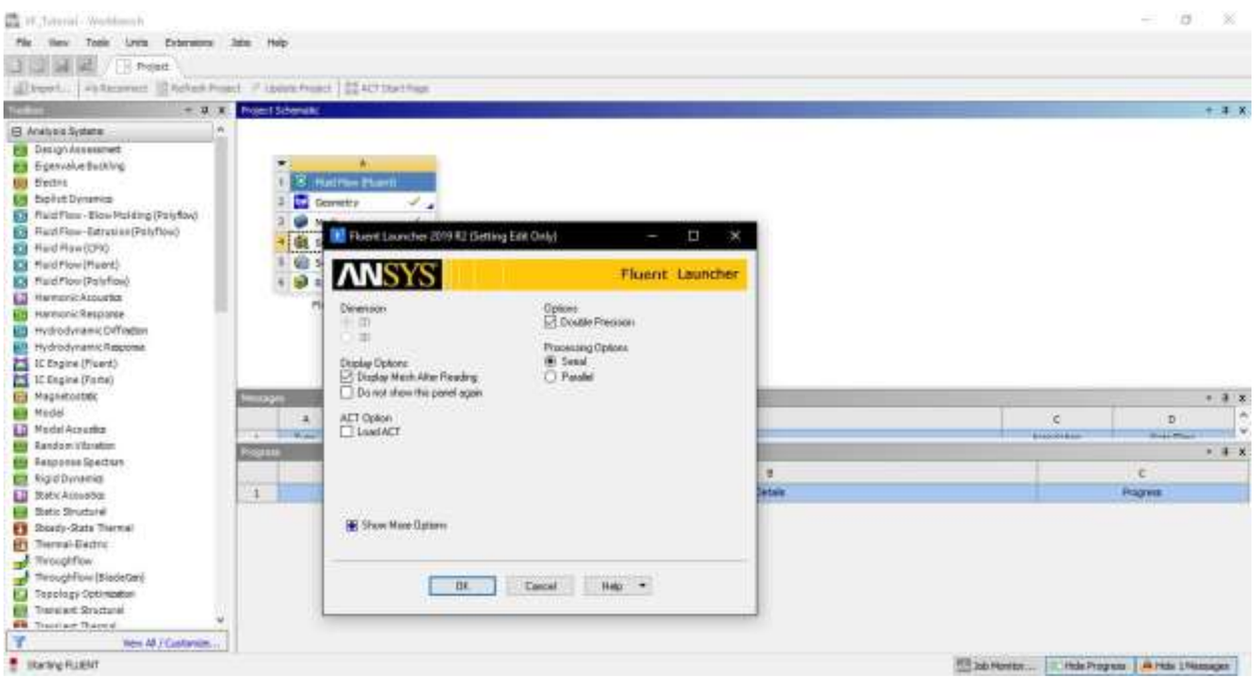
Step 12: Create a Mesh based on a Named Selection and select the "Flat Plate" as shown before. Input the Number of Divisions (Select 1000 for a fine mesh, or 200 if you have the Academic version). Set the Bias Type and Bias Factor as shown.



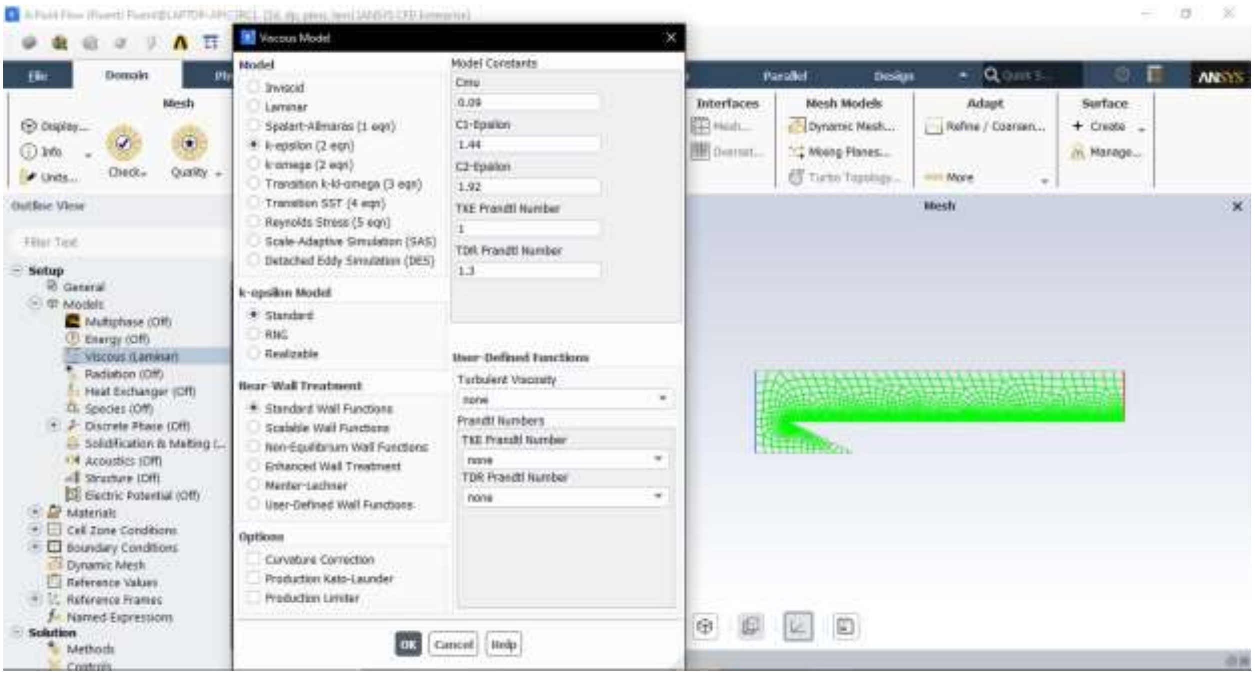
Step 13: Generate the Mesh, it should look similar to what is shown below, then close the Mesh Editor.



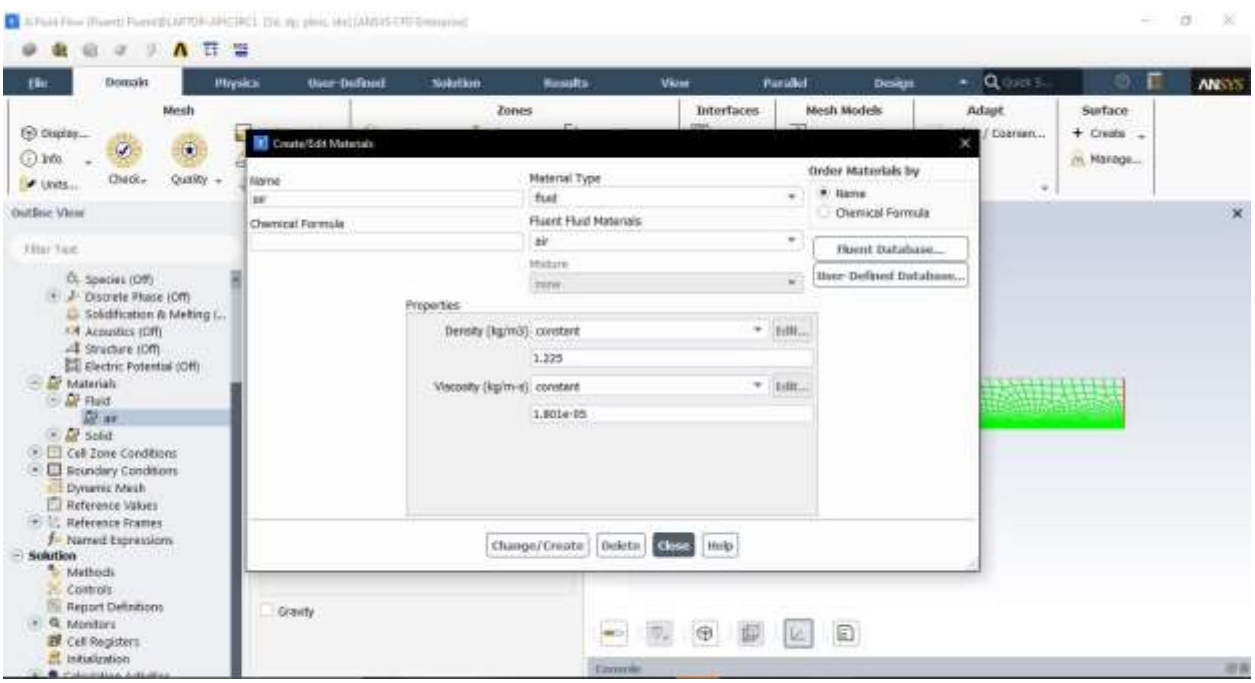
Step 14: Open Ansys Fluent by RMB -> Setup, select Double Precision and then OK.



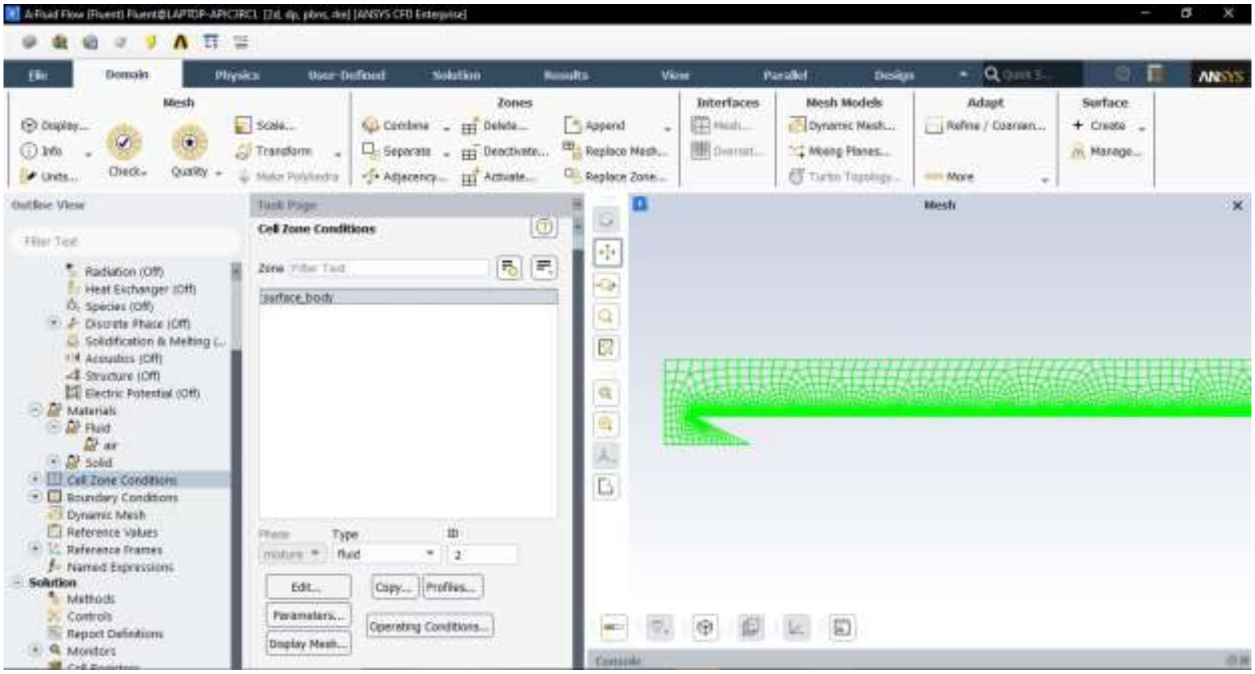
Step 15: Perform a Mesh Check, then set the Model to "Viscous – K-Epsilon (2 eqn.)" as shown.



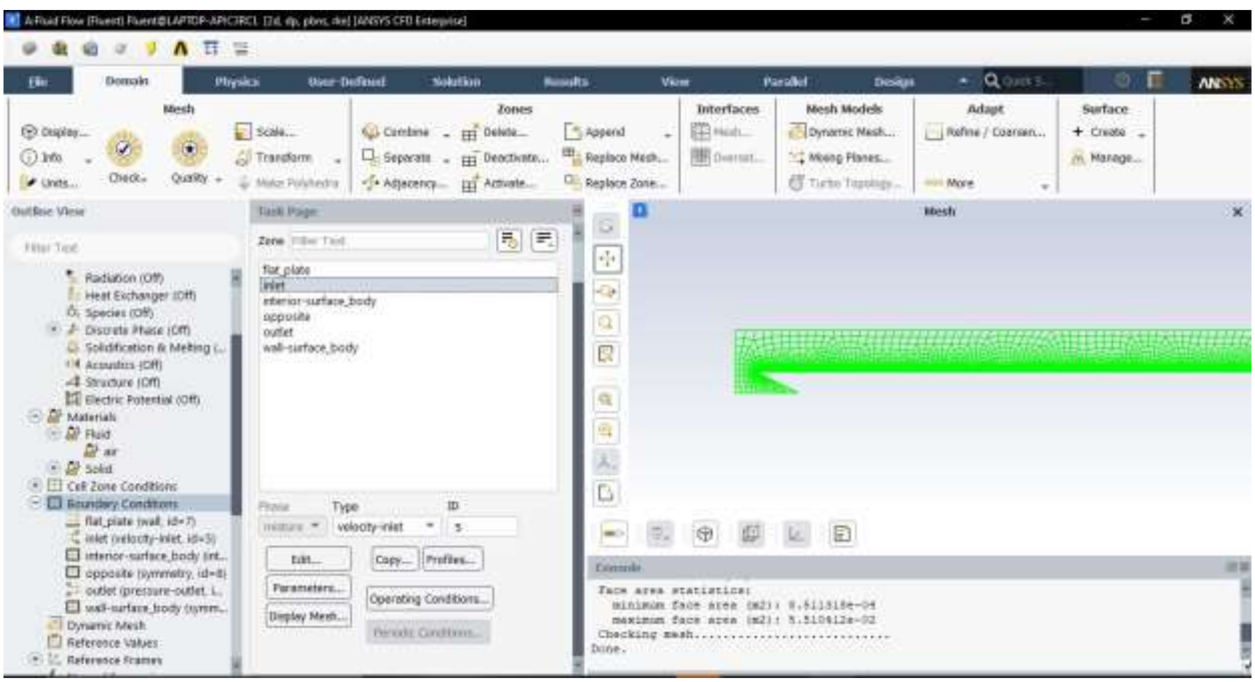
Step 16: Set the Material to a Fluid and Air, with the density and viscosity values shown in the figure.



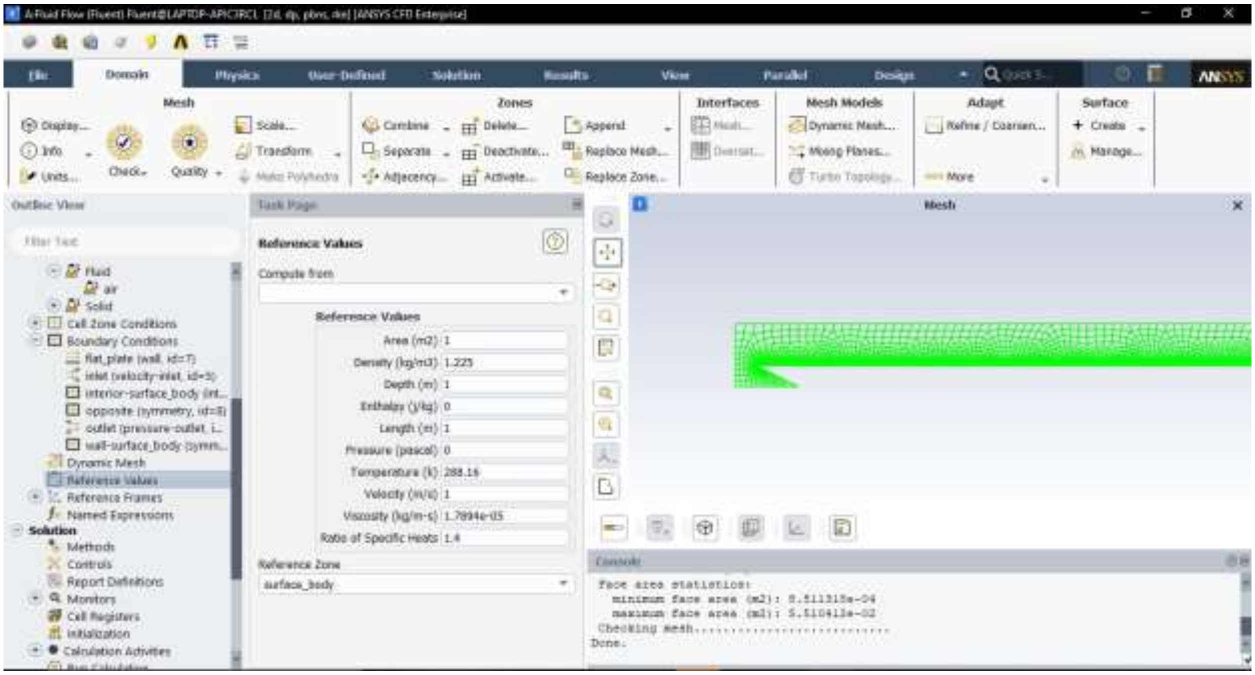
Step 17: Set Cell Zone Conditions to “Fluid” for the surface_body.



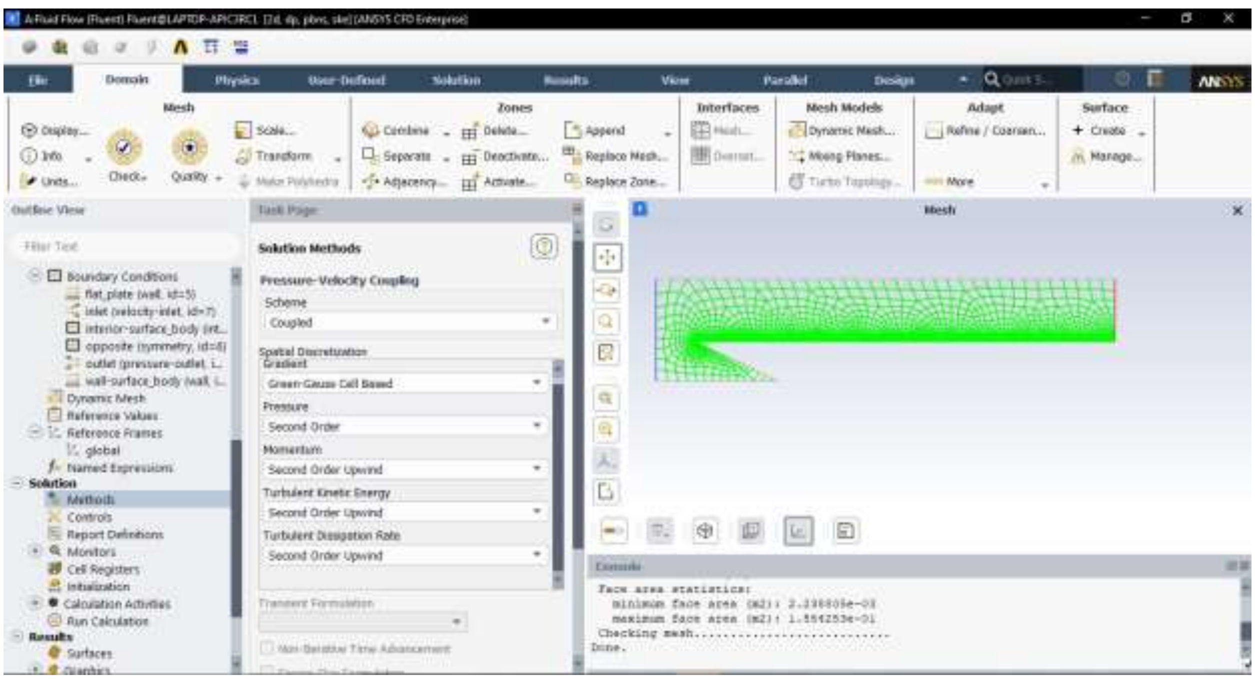
Step 18: Set the Boundary Conditions. The “flat_plate” and “Wall-surface_body” should be set to wall, all others should be set to their respective types.



Step 19: Set the reference values as shown in the figure below.

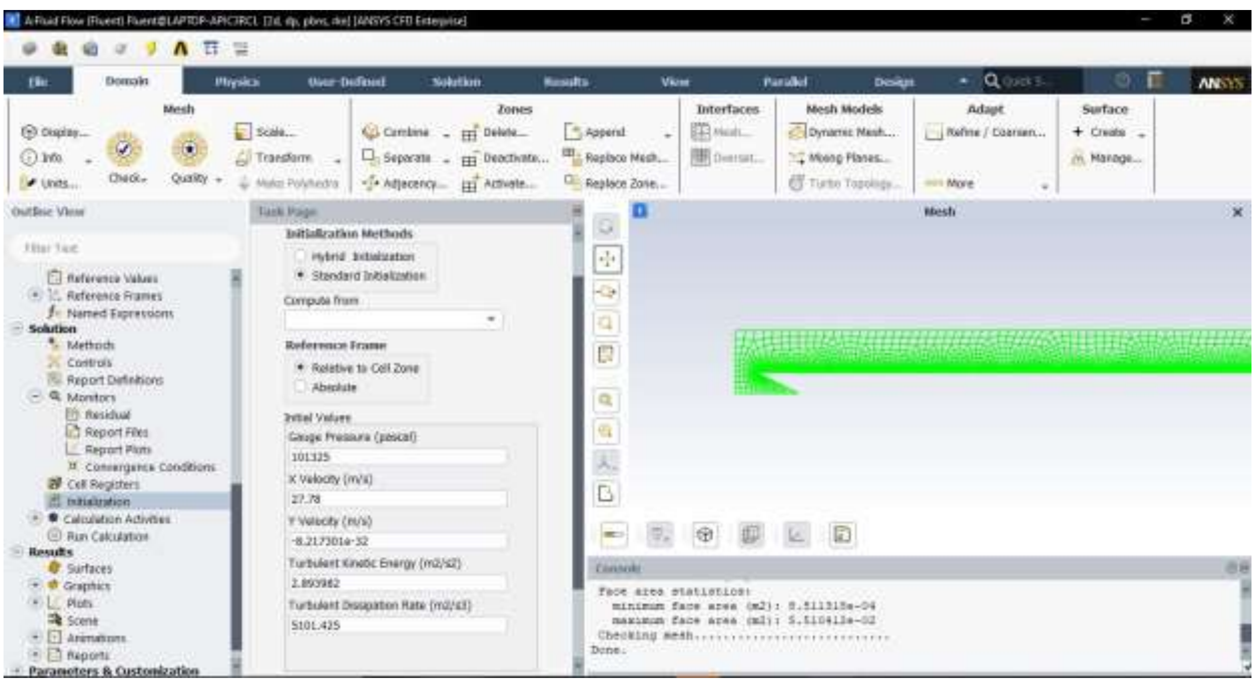


Step 20: Set the Methods to the values shown in the figure.

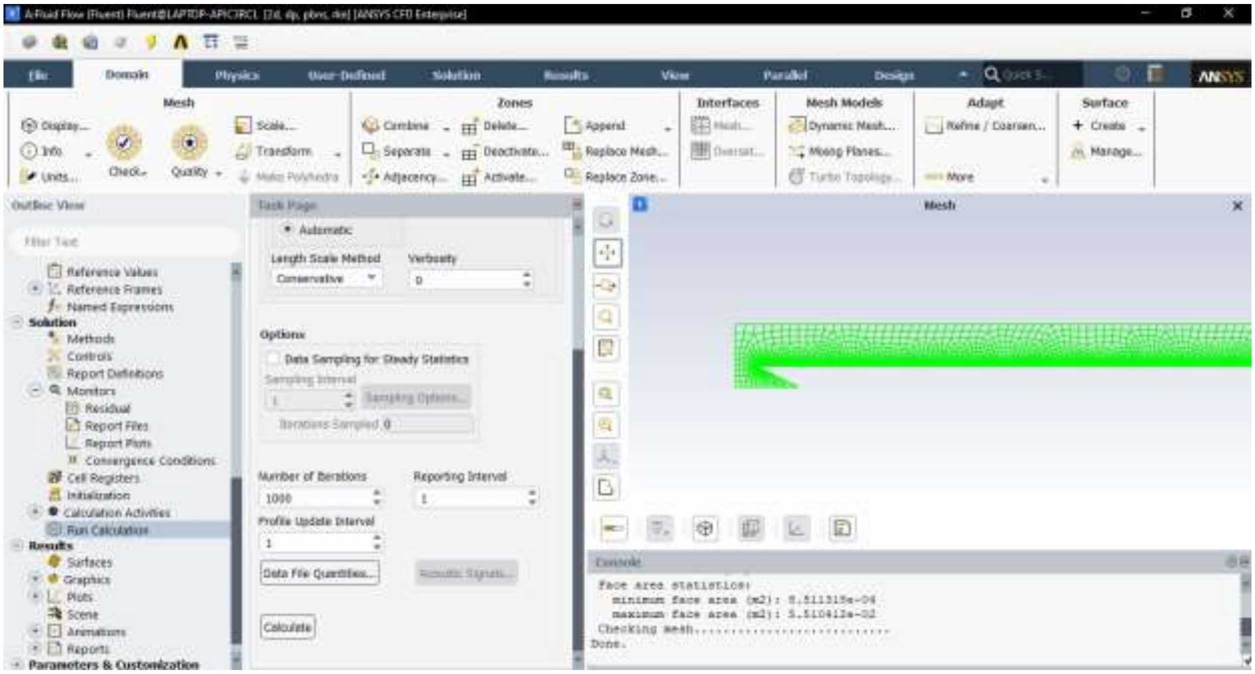


Step 21: Set the residuals to 1e-06 as shown.

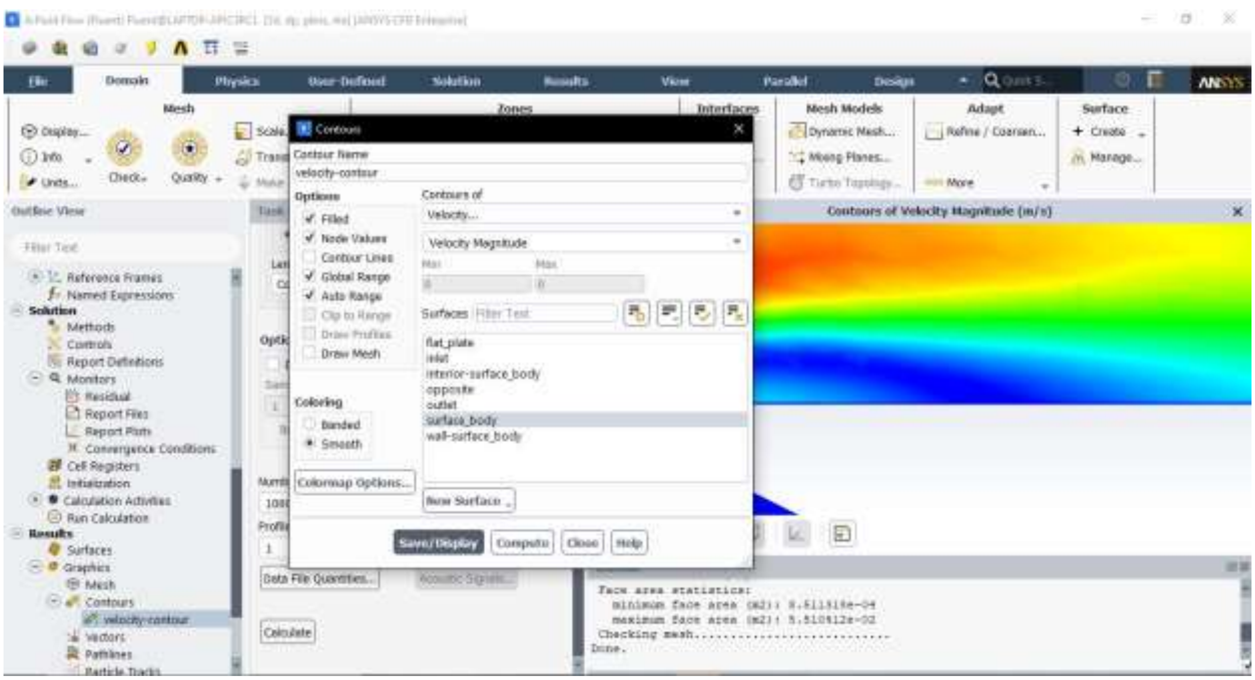
Step 22: Change the Initialization values to those shown below and then click, “Initialize”.



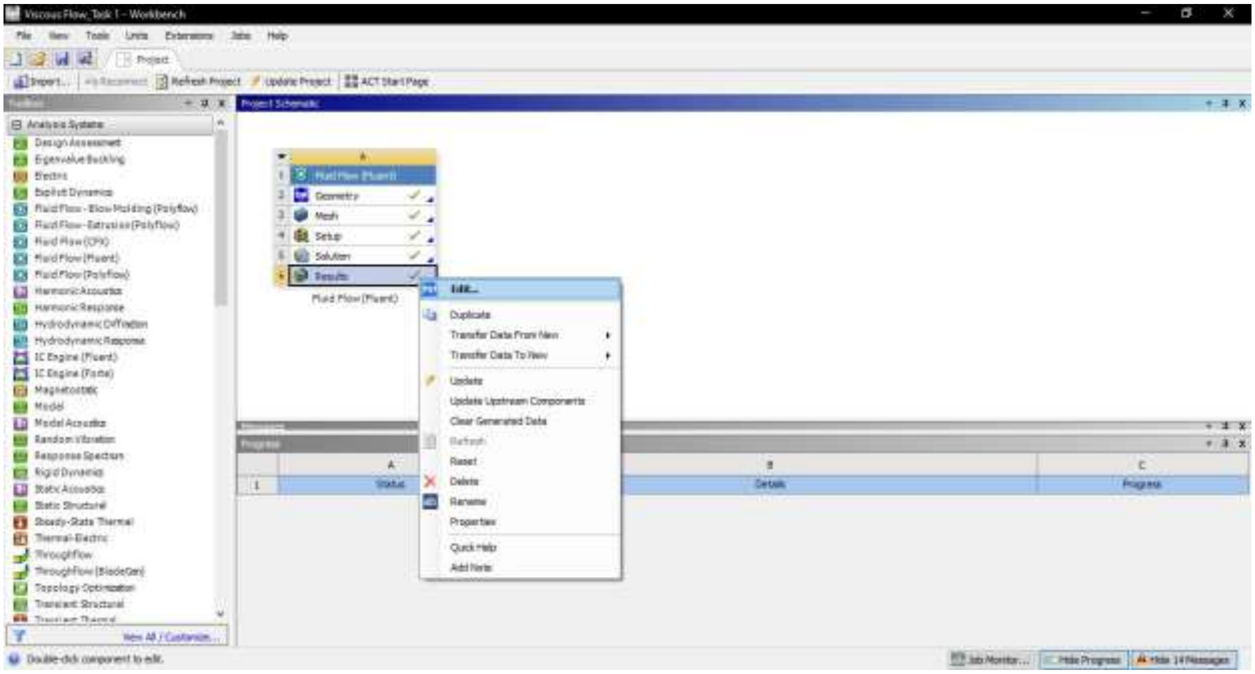
Step 23: Go to “Run Calculation” set the Number of Iterations to 1000 and click “Calculate”



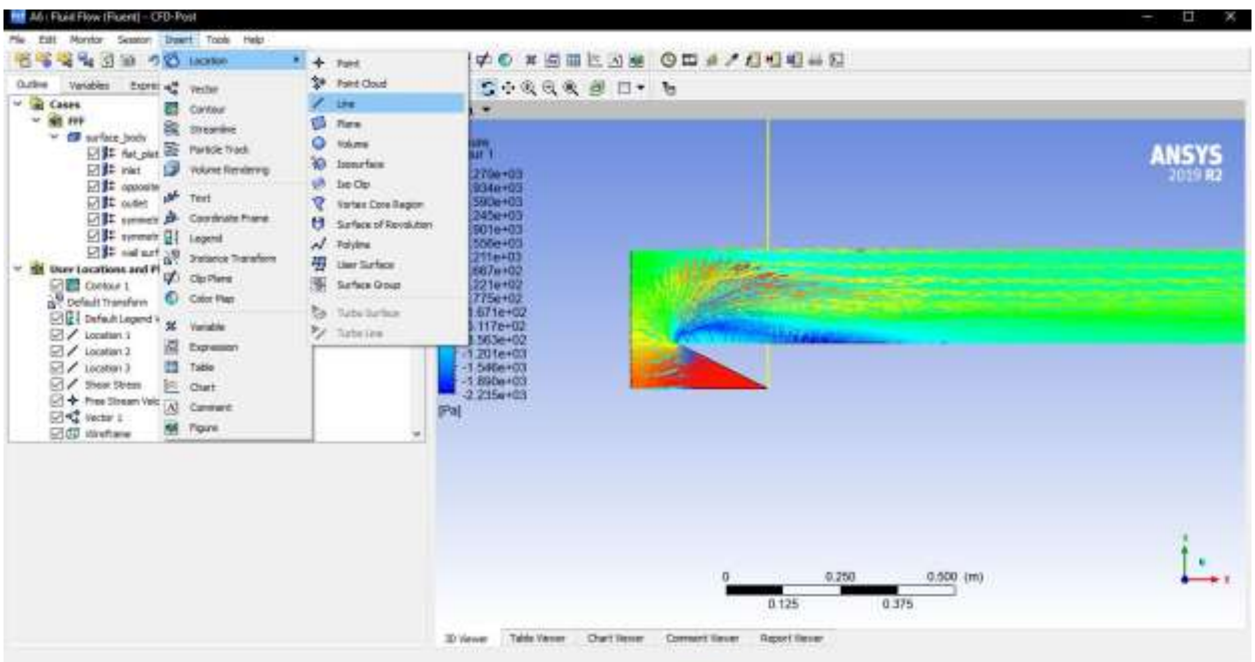
Step 24: After the calculations have been run, under “Results”, double-click Contours and select “Velocity” of the surface_body, then press Save/Display.



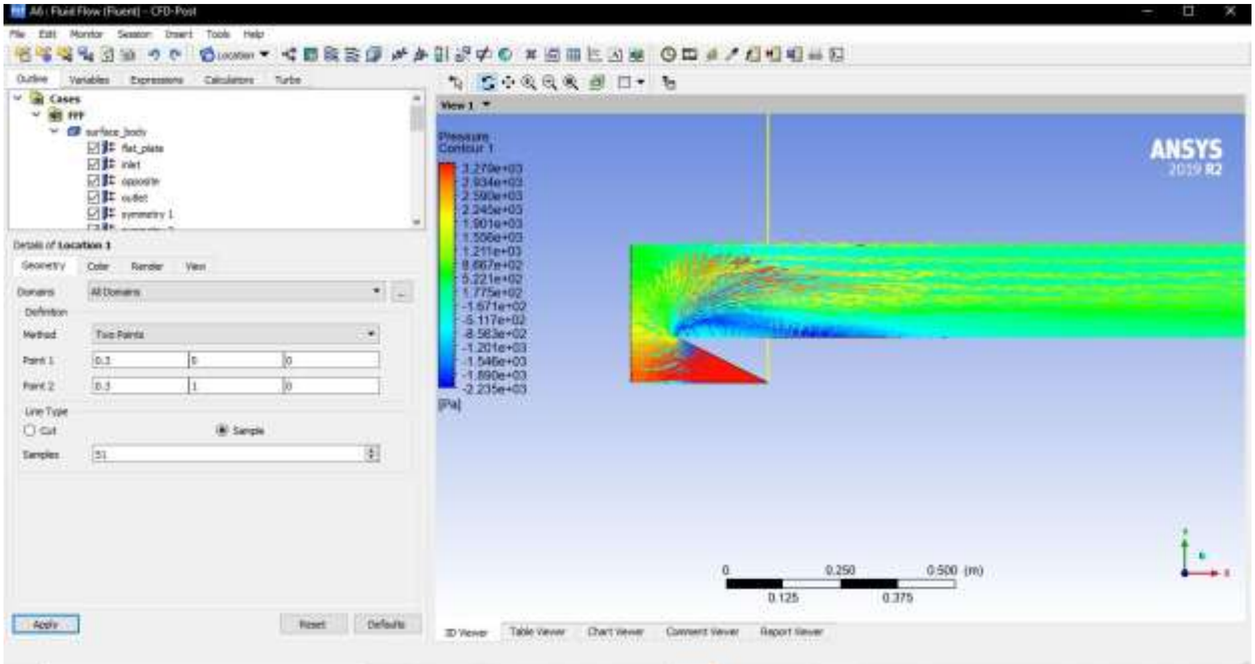
Step 25: Close Fluent, then in the Workbench, go to Results, then RMB -> Edit.



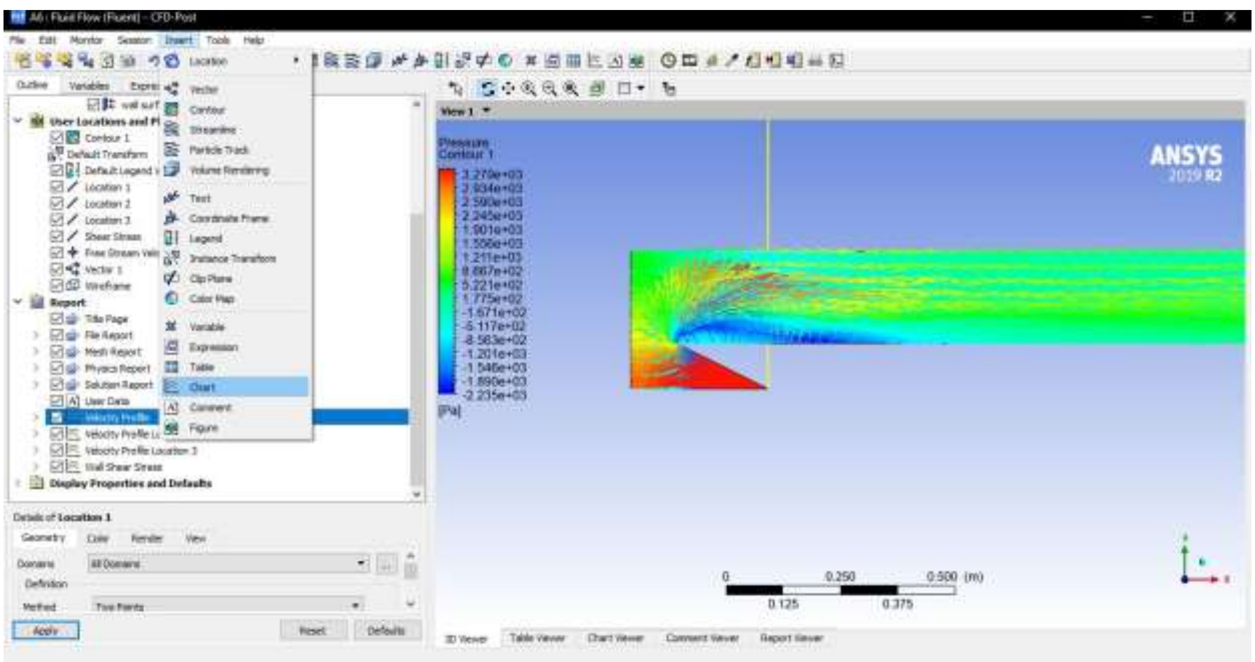
Step 26: Go to Insert, Location, Line as shown below.



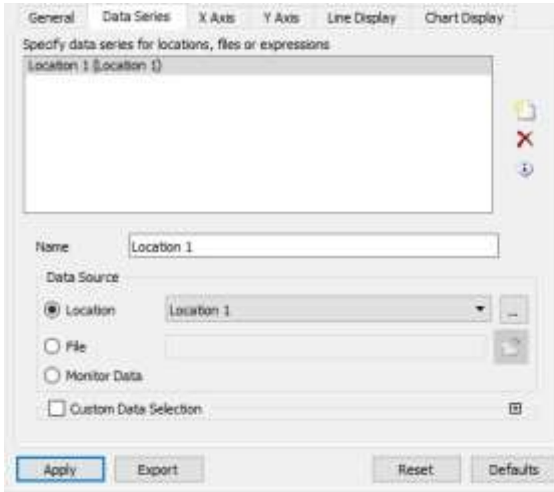
Step 27: Set the Values to the location desired, the first one should be the values shown below. Repeat this process for the other 2 locations.

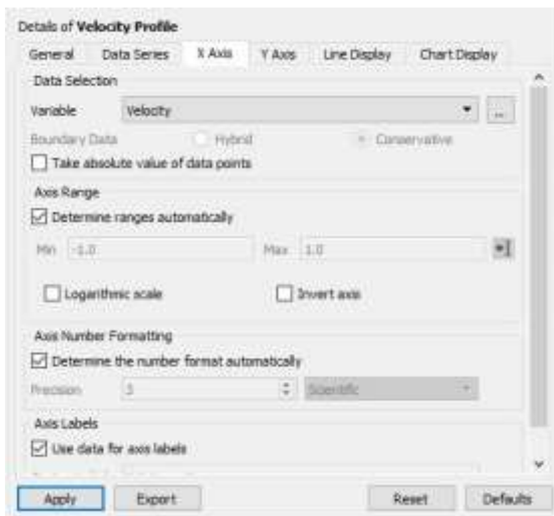


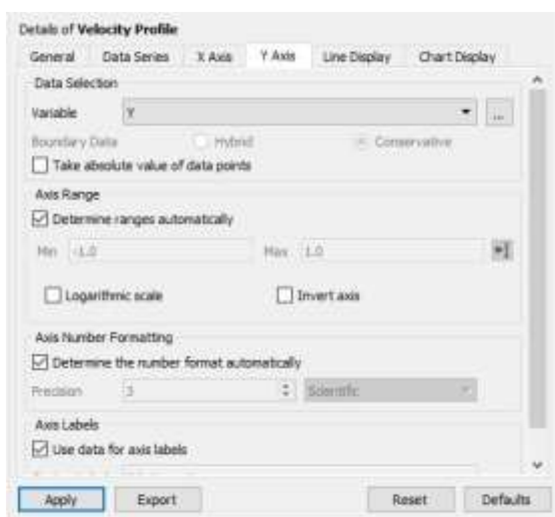
Step 28: Go to Insert, Chart.



Step 29: Input the information shown in the following 3 figures to generate your Chart. Repeat this process for the other 2 locations.







Step 30: The chart should look like the figure below. To generate Contours and Vectors, click the corresponding Contour and Vector buttons.