

Simulation of Turbulent Flow in an Asymmetric Diffuser

1. Purpose

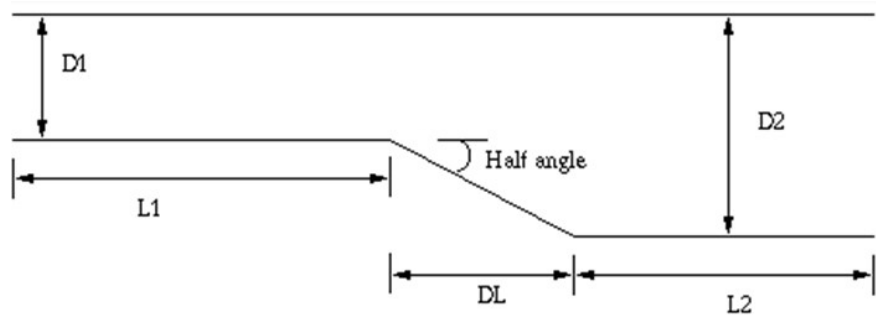
The Purpose of CFD Lab 2 is to simulate **turbulent** flows inside a diffuser following an interactive step-by-step approach and conduct verifications. Students will have “hands-on” experiences using ANSYS to conduct **validation of velocity, turbulent kinetic energy, and skin friction factor. Effect of turbulent models will be investigated, with/without separations.** Students will manually generate meshes, solve the problem and use post-processing tools (contours, velocity vectors, and streamlines) to visualize the flow field. Students will analyze the differences between CFD and EFD and present results in a CFD Lab report.

2. Simulation Design

The problem to be solved is that of turbulent flows inside an asymmetric diffuser (2D). Reynolds number is 17,000 based on inlet velocity and inlet dimension (D1). The following figure shows what the geometry looks like with definitions for all geometry parameters. Before the diffuser, a straight channel was used for generating fully developed channel flow at the diffuser inlet. You will conduct simulation for two different half angles of 4 and 10 with two different turbulence models of SST and k- ϵ .

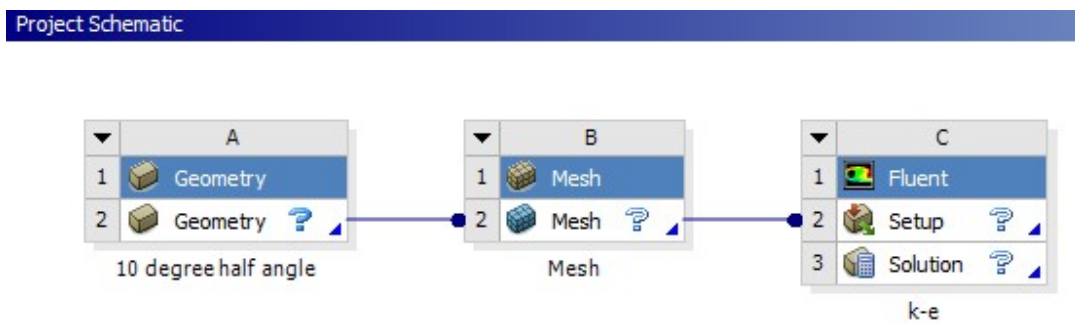
Table 1 – Main particulars

Parameter	Symbol	Unit	Value
Inlet dimension	D1	m	2
Inlet length	L1	m	60
Diffuser half angle	α	degree	4 or 10
Outlet dimension	D2	m	9.4
Outlet length	L2	m	70



3. Open ANSYS Workbench

3.1. Create the layout as per below.

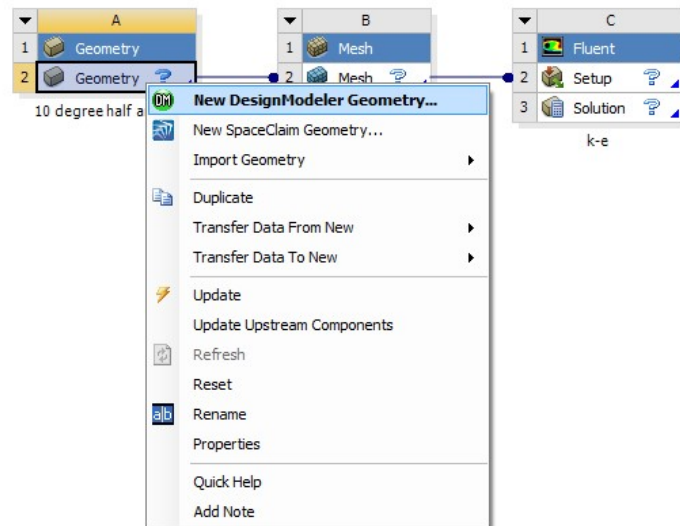


3.2 **File > Save.** Save the project on the D: drive.

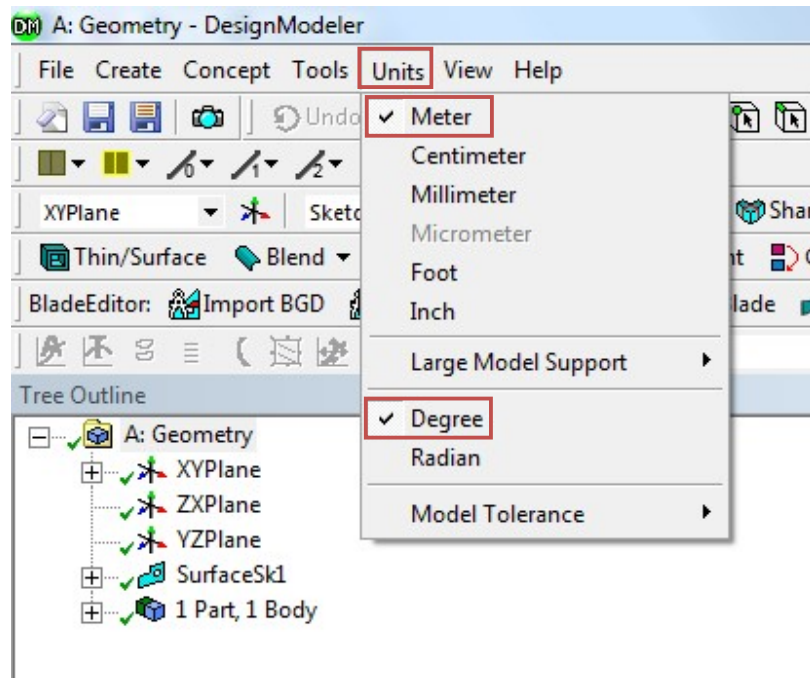
4. Geometry Creation

In this section we will create the geometry for the diffuser with 10 degree half angle then copy and modify the geometry for 4 degree half angle.

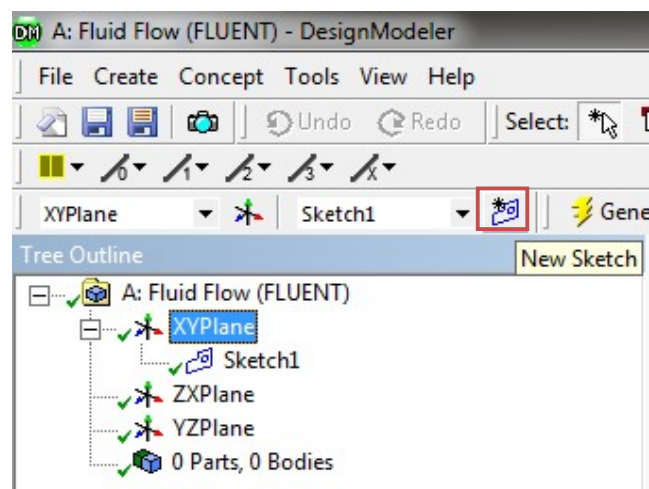
- 4.1 Right click **Geometry** and select **New DesignModeler Geometry...**



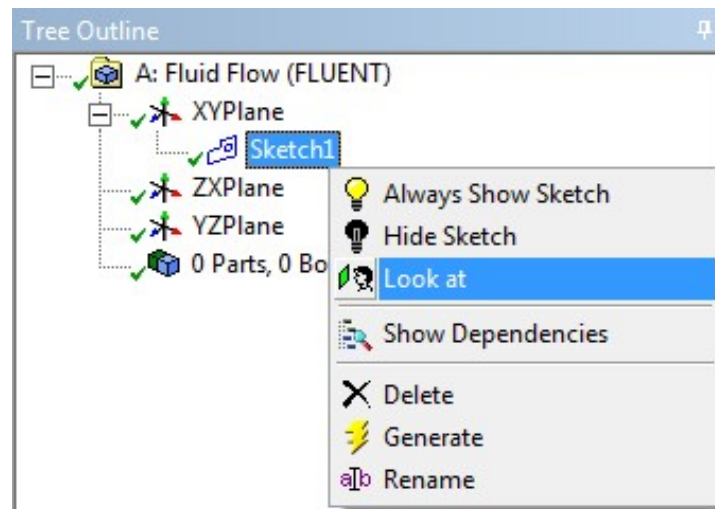
- 4.2 Make sure that Unit is set to **Meter** and **Degree** (default value).



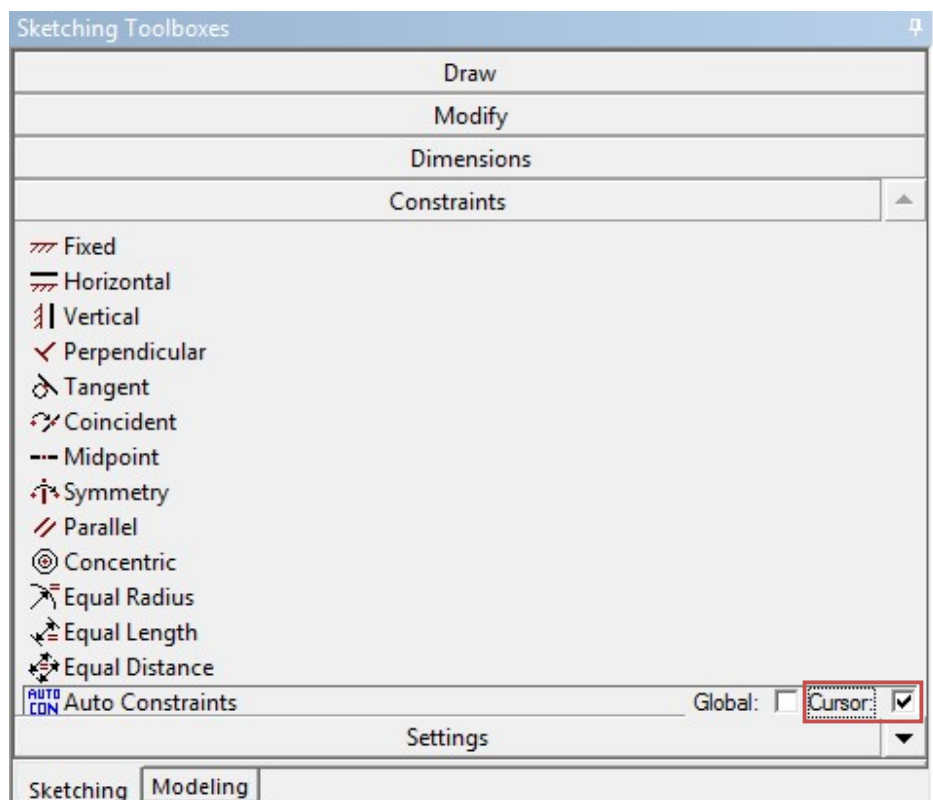
4.3 Select **XYplane** and click **New Sketch** button.



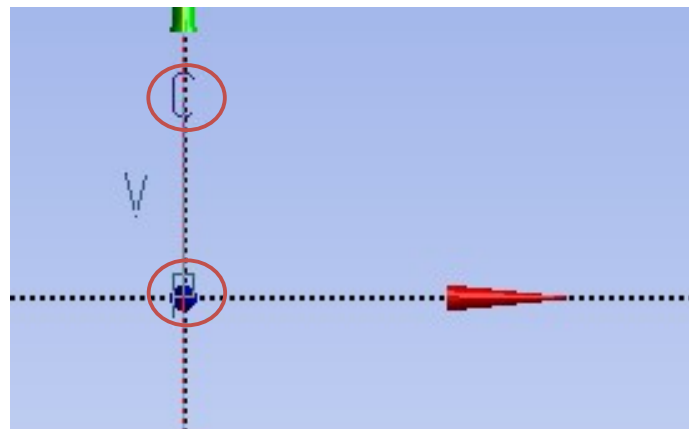
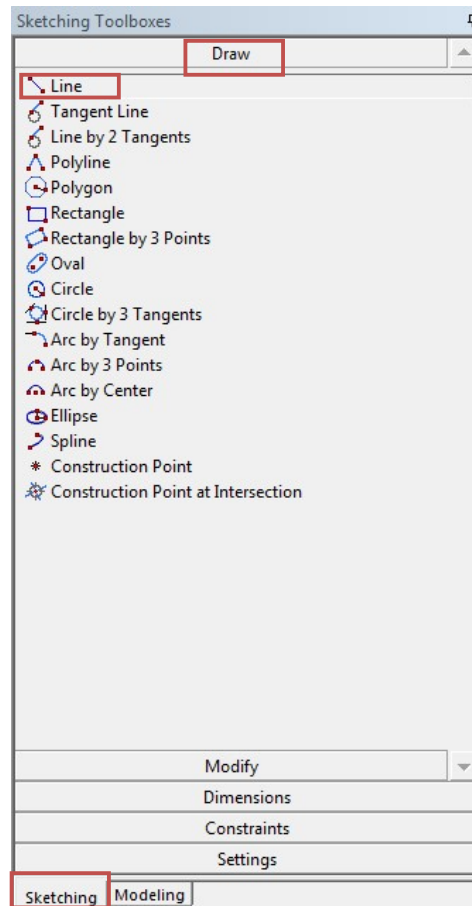
4.4 Right click **Sketch1** and select **Look at**.



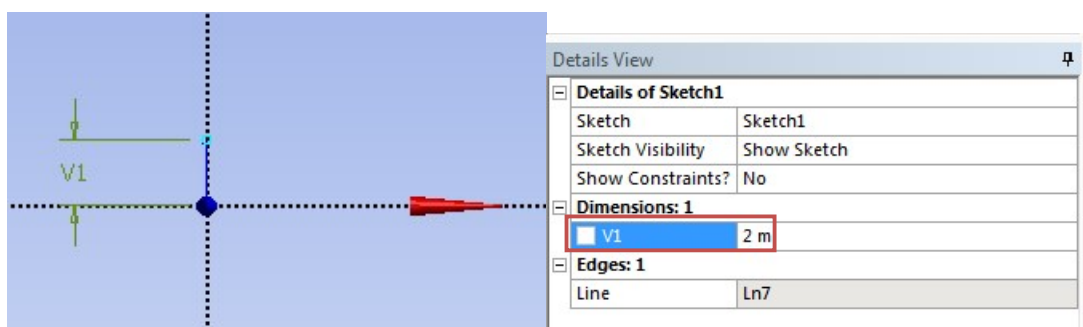
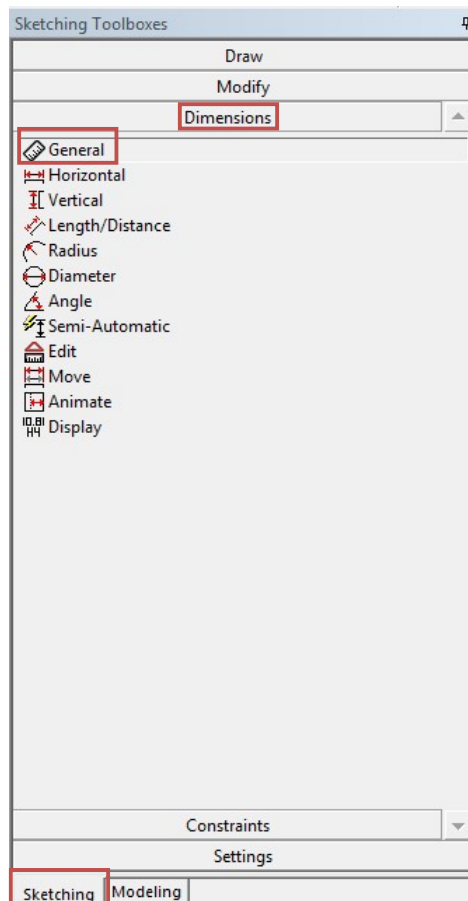
4.5 **Sketching > Constraints > Auto Constraints.** Enable the auto constraints option to pick the exact point as below.



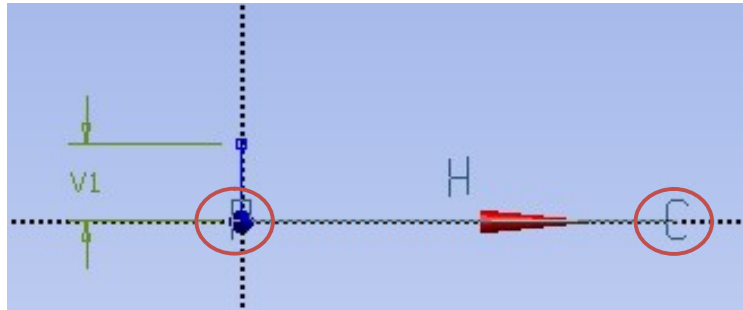
- 4.6 **Sketching > Draw > Line.** Draw a vertical line on the y-axis starting from the origin as shown below (**P** indicates that the origin point is selected and **V** indicates that the line is vertical).



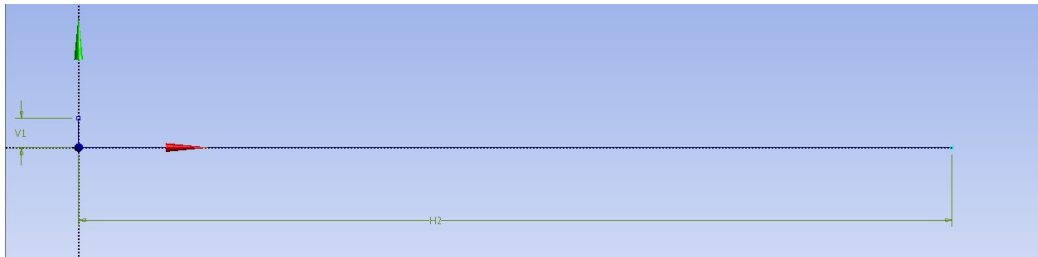
- 4.7 **Sketching > Dimensions > General**. Click on the vertical line then click on the left side of the line to place the dimension. Change the dimension in **Details View** to **2m**.



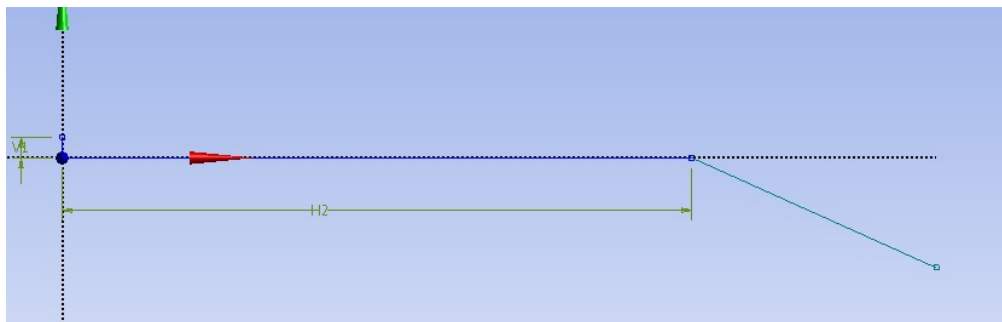
- 4.8 **Sketching > Draw > Line.** Create a horizontal line on the x-axis starting at the origin as per below (**H** indicates that line is horizontal).



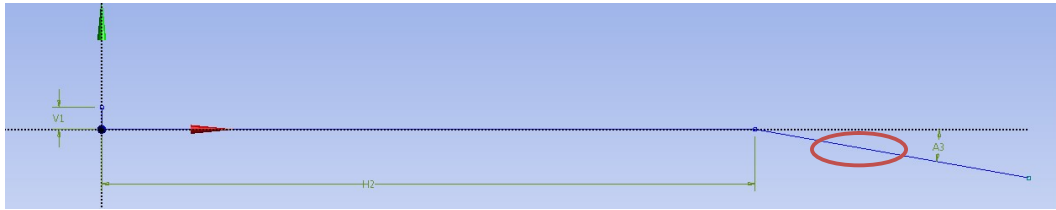
- 4.9 **Sketching > Dimensions > General.** Change the length of the horizontal line you created to **60m**.



- 4.10 **Sketching > Draw > Line.** Create line at an angle with respect to x-axis as shown below.

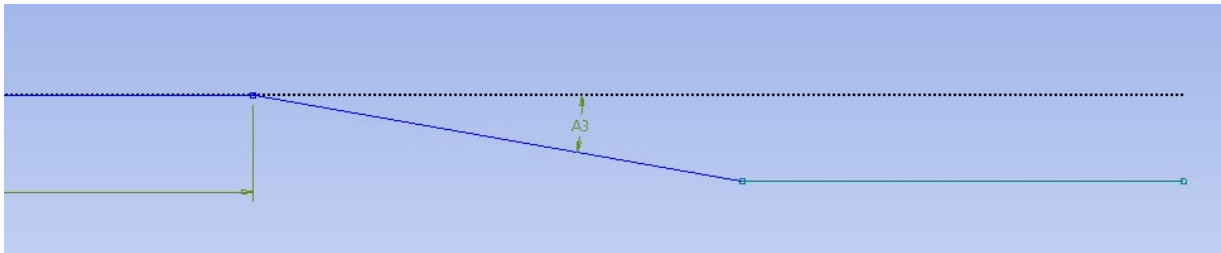


- 4.11 **Sketching > Dimensions > Angle.** Select the line circled in red below then select the x-axis then change the angle to **10°**. (Note: if ANSYS gives a default exterior angle instead of the interior angle, right click and select **Alternate Angle**.)

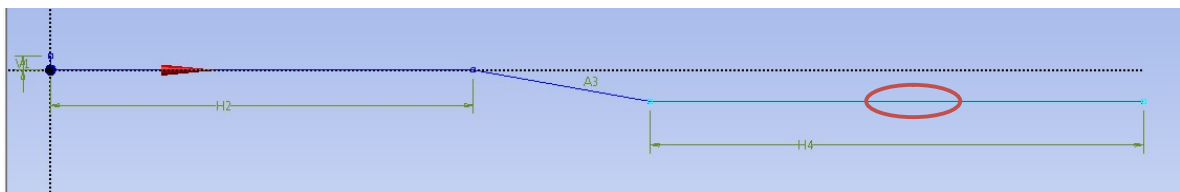


Details View	
Details of Sketch1	
Sketch	Sketch1
Sketch Visibility	Show Sketch
Show Constraints?	No
Dimensions: 3	
<input checked="" type="checkbox"/> A3	10 °
<input type="checkbox"/> H2	60 m
<input type="checkbox"/> V1	2 m
Edges: 3	
Line	Ln7
Line	Ln8
Line	Ln9

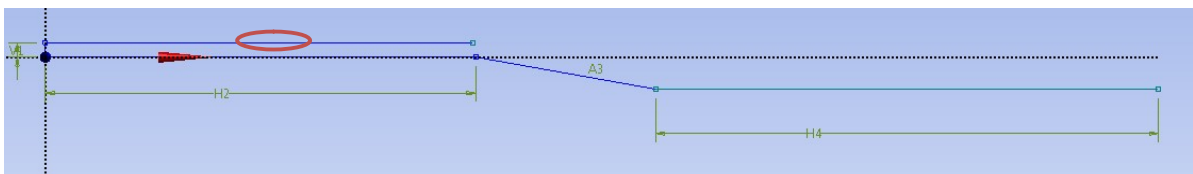
4.12 **Sketching > Draw > Line.** Create a horizontal line as per below.



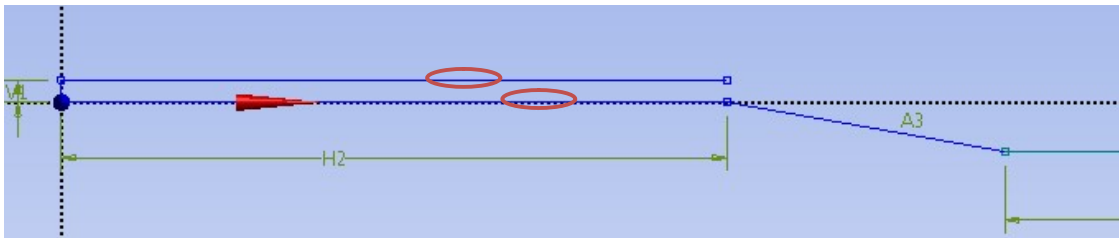
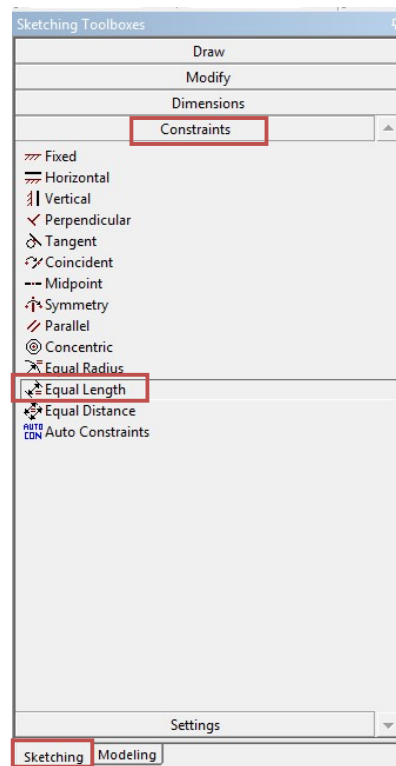
4.13 **Sketching > Dimensions > General.** Change the length of the line circled in red to **70m**.



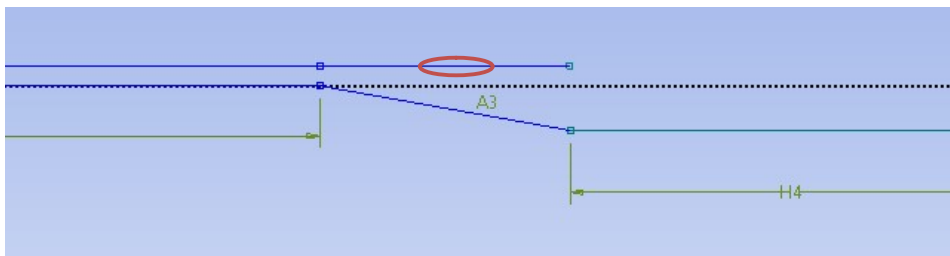
4.14 **Sketching > Draw > Line.** Draw the horizontal line circled in red line as per below.



4.15 **Sketching > Constraints > Equal Length.** Select the two lines circled in red as shown below.



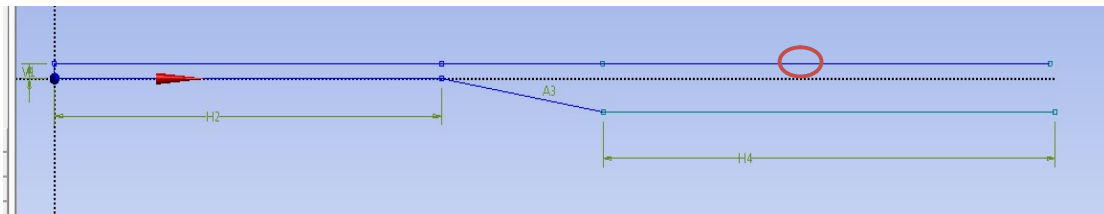
4.16 **Sketching > Draw > Line.** Draw the horizontal line circled in red as per below.



- 4.17 **Sketching > Constraints > Equal Distance.** Click on point 1 and then click on the point 2. Click point 3 and then click on line 4. This makes points 1 and 3 the same distance from the y-axis in the horizontal direction.



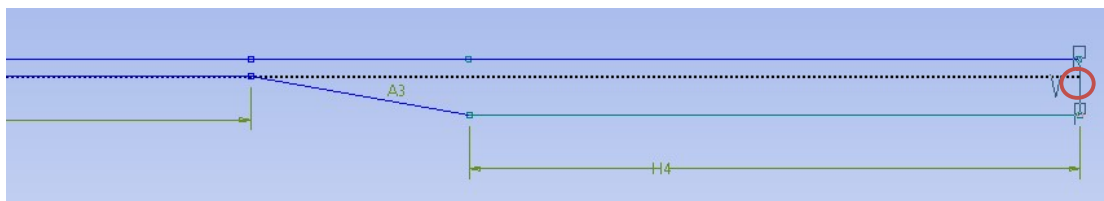
- 4.18 **Sketching > Draw > Line.** Draw the horizontal line circled in red as shown below.



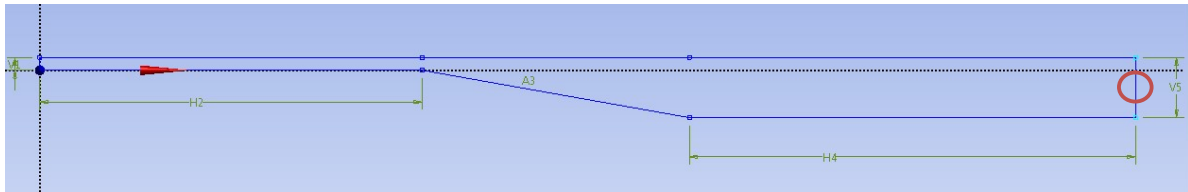
- 4.19 **Sketching > Constraints > Equal Length.** Click on the two lines circled in red as shown below.



- 4.20 **Sketching > Draw > Line.** Draw the final line circled in red as shown below. When you draw this line, if all previous dimensions and constraints are correct, the line should have two **P**'s at the ends with a **V** in the center. This indicates that the line starts and ends on the two points and is perfectly vertical. If you do not get the **V** recheck all dimensions and constraints.

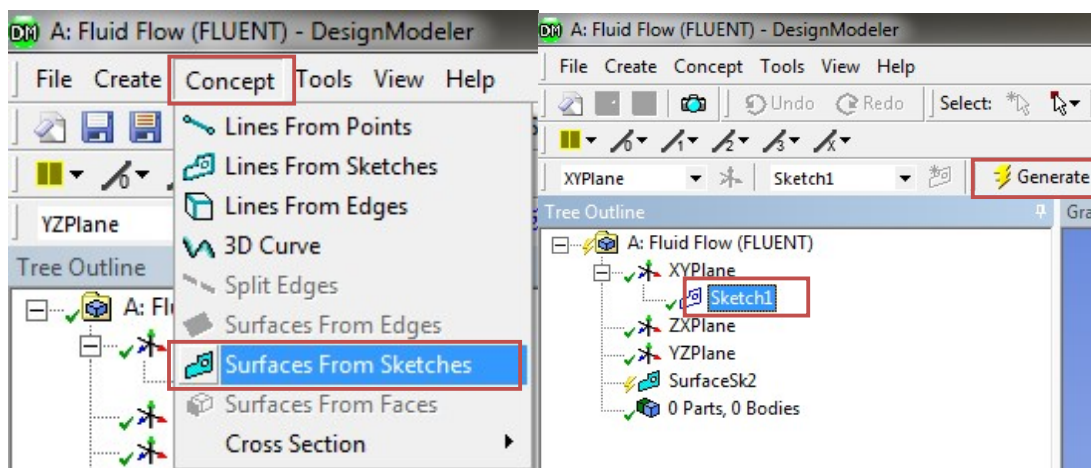


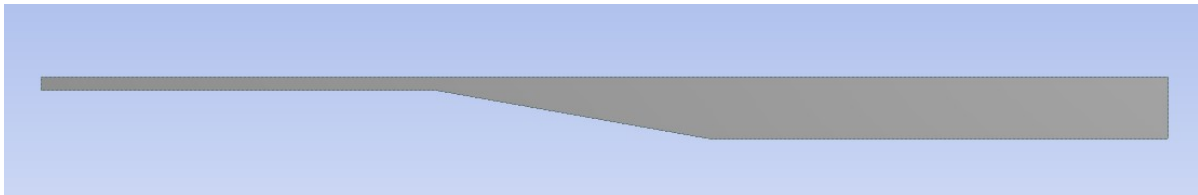
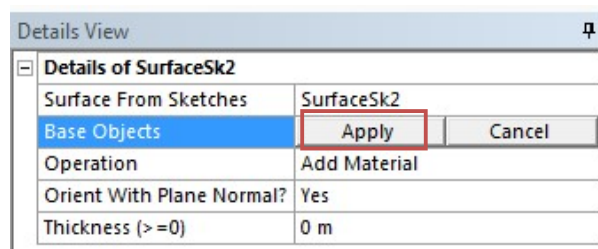
- 4.21 **Sketching > Dimensions > General.** Change the length of the line circled in red to **9.4m**, this will automatically adjust the length of the expansion region because of the applied constraints.



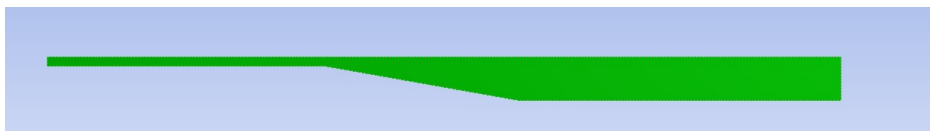
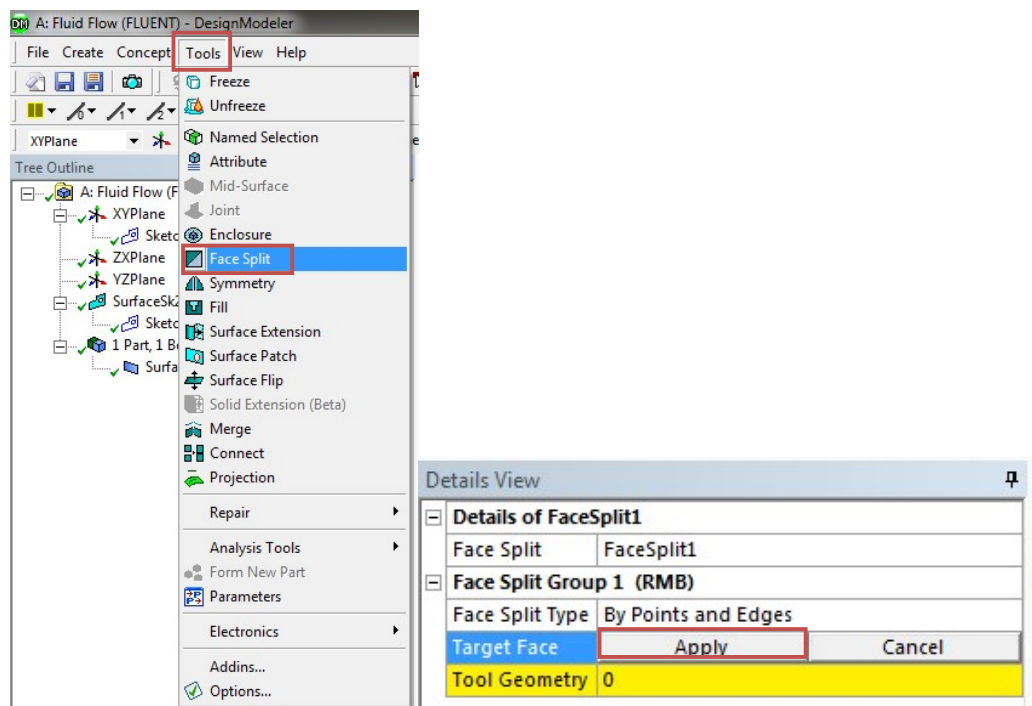
Details View	
Details of Sketch1	
Sketch	Sketch1
Sketch Visibility	Show Sketch
Show Constraints?	No
Dimensions: 5	
<input type="checkbox"/> A3	10 °
<input type="checkbox"/> H2	60 m
<input type="checkbox"/> H4	70 m
<input type="checkbox"/> V1	2 m
<input checked="" type="checkbox"/> V5	9.4 m
Edges: 8	
Line	Ln7
Line	Ln8
Line	Ln9
Line	Ln10
Line	Ln11
Line	Ln12
Line	Ln13
Line	Ln14

- 4.22 **Concept > Surfaces From Sketches.** Select the sketch you created and click **Apply** then click **Generate**. This will create a surface as shown below.

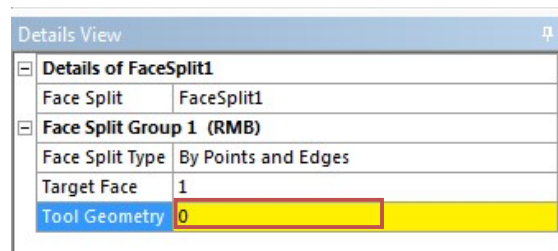




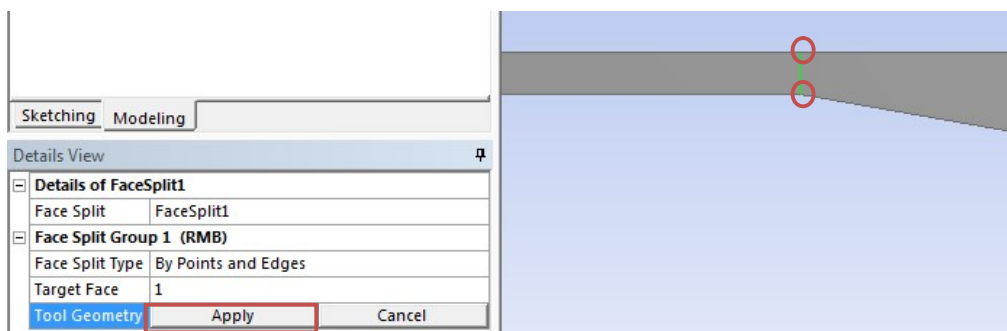
4.23 **Tools > Face Split.** Select the surface you created (it will be highlighted in green when you select it as shown below) then click **Apply** for **Target Face**.



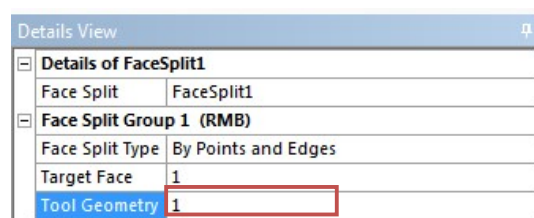
4.24 Click on the yellow region shown below.



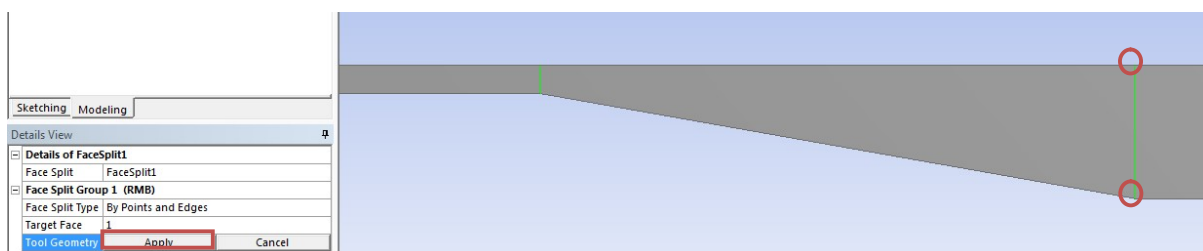
4.25 While holding **Ctrl** button click on the two points circled in red then click **Apply** button.



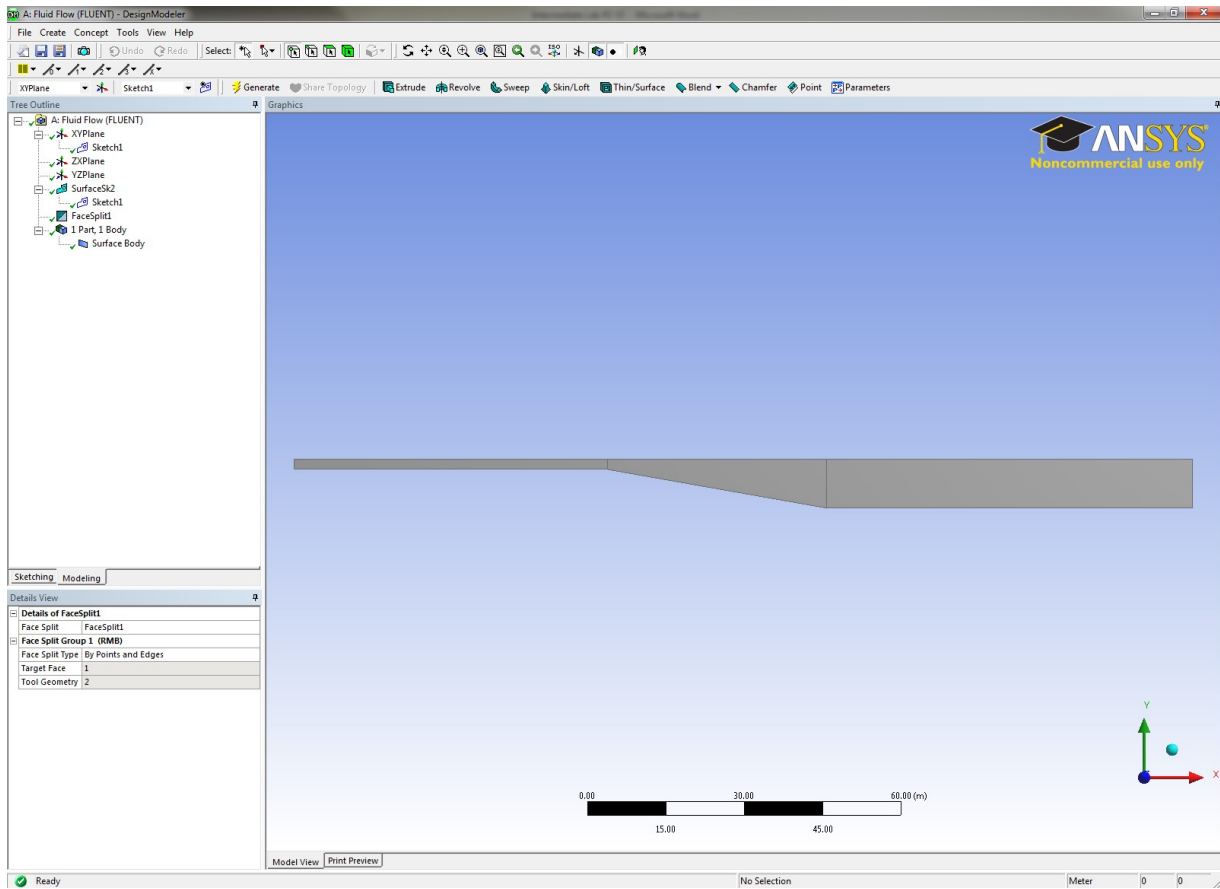
4.26 Click on the region marked with red rectangle below.



4.27 While holding **Ctrl** button click on the two points circled in red then click **Apply** button.

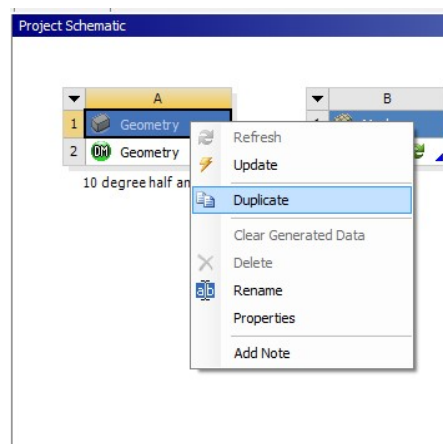


4.28 Click the **Generate** button then close the window and update geometry.



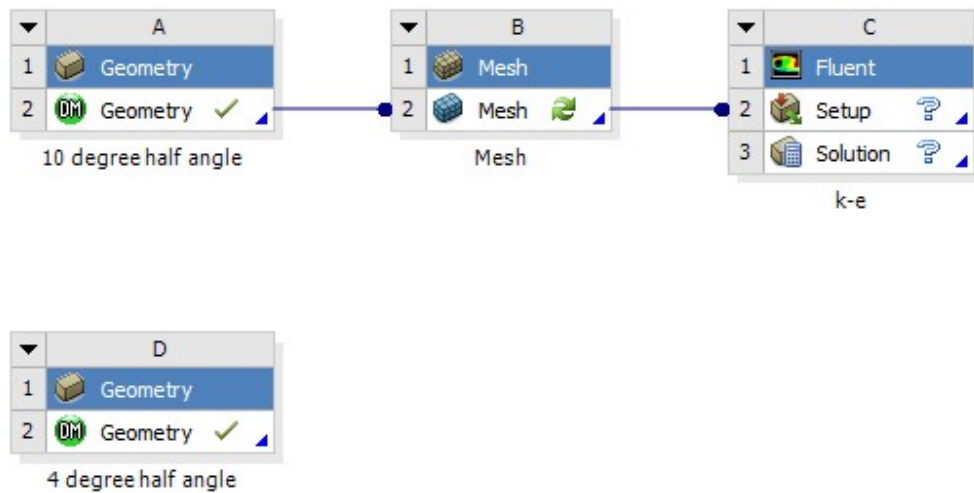
4.29 Save your progress and close ANSYS Design Modeler

4.30 Right click on geometry and select Duplicate.

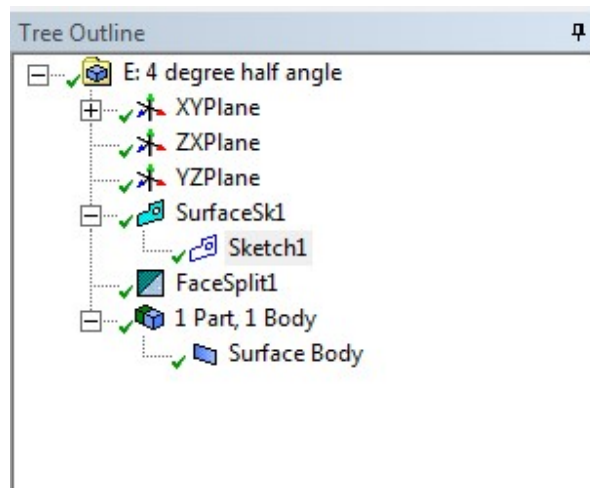


4.31 Rename the new geometry file as per below.

Project Schematic



4.32 Open the new geometry file you created and select Sketch1 under the tree outline as per below. Change the half angle to 4 degrees under details view as per below then click the generate button.



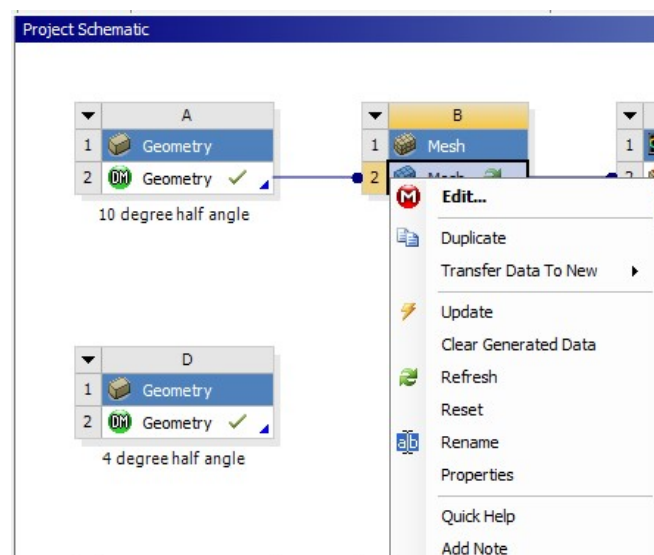
Details View	
Details of Sketch1	
Sketch	Sketch1
Sketch Visibility	Show Sketch
Show Constraints?	No
Dimensions: 5	
<input type="checkbox"/> A3	4 °
<input type="checkbox"/> H2	60 m
<input type="checkbox"/> H4	70 m
<input type="checkbox"/> V1	2 m
<input type="checkbox"/> V5	9.4 m
Edges: 8	
Line	Ln7
Line	Ln8
Line	Ln9
Line	Ln10
Line	Ln11
Line	Ln15
Line	Ln16
Line	Ln17

4.33 Save your file and quit ANSYS Design Modeler

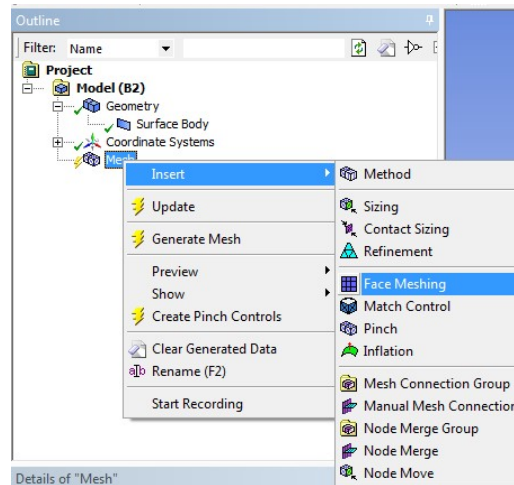
5. Mesh Generation

This section shows how to generate the mesh for both 4 degree and 10 degree half angle cases.

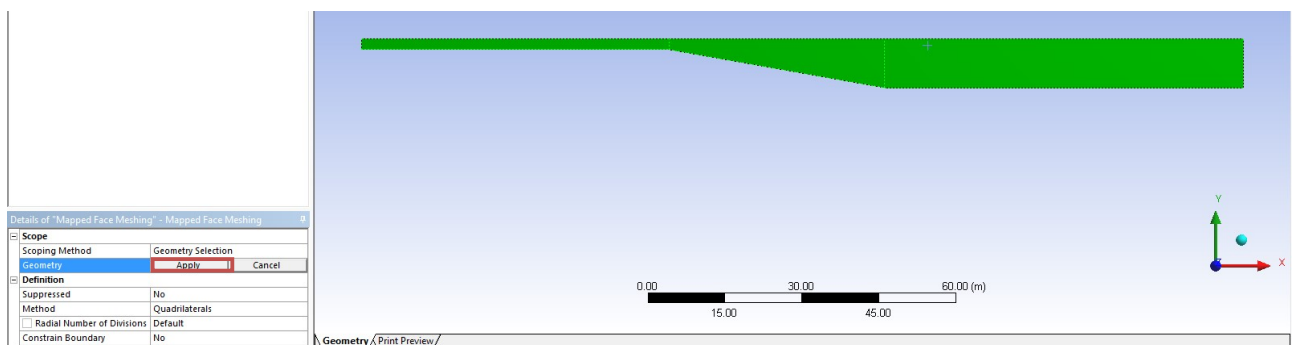
5.1 Right click on **Mesh** and click **Edit...**



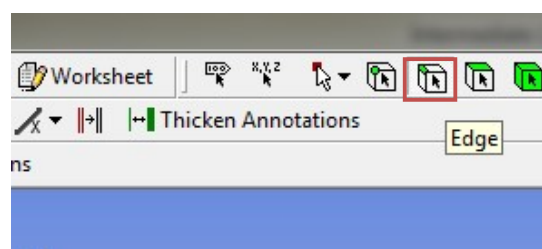
5.2 Right click on **Mesh** then select **Insert > Face Meshing**.



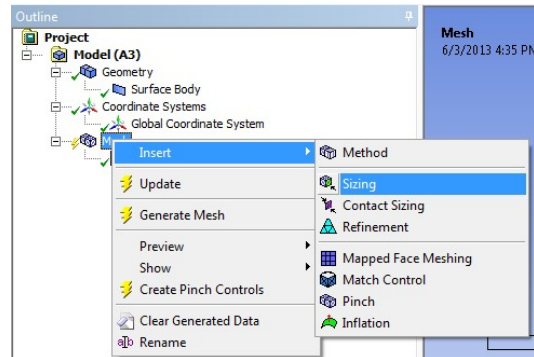
5.3 Select all three surface while holding ctrl button and click **Apply**.



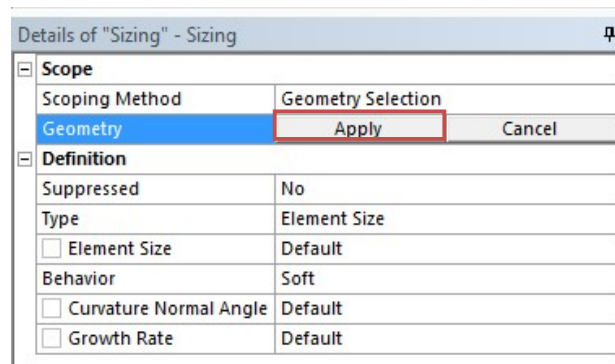
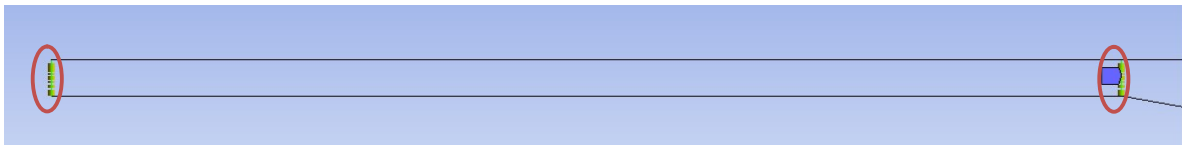
5.4 Select the **Edge** button. This will allow you to select edges of your geometry.



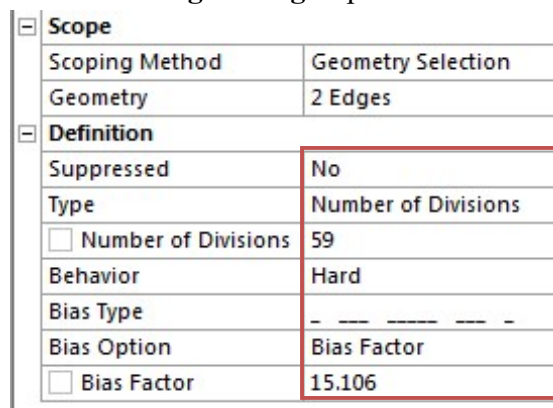
5.5 Right click on **Mesh** and **Insert > Sizing**.

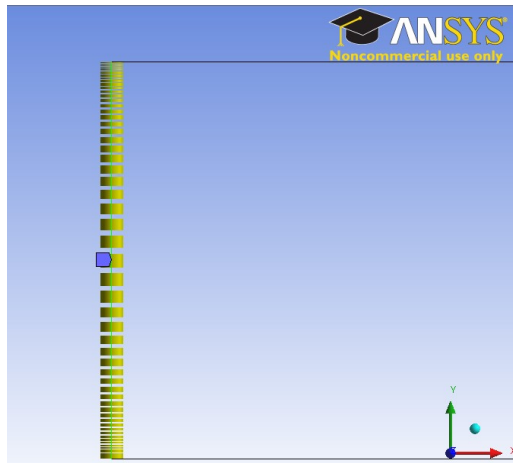


5.6 While holding **Ctrl** click on the edge shown below and click **Apply**.



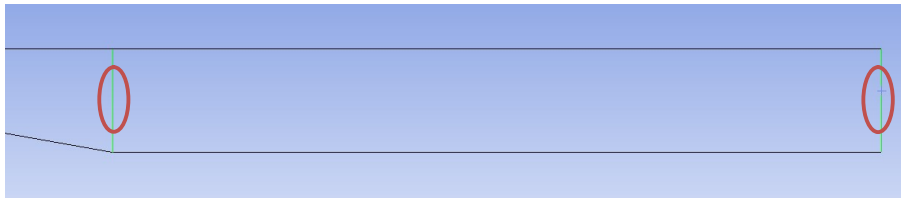
5.7 Change parameter for **Edge Sizing** as per below.



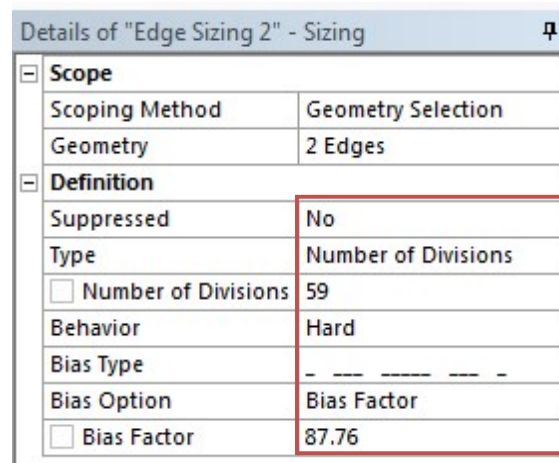


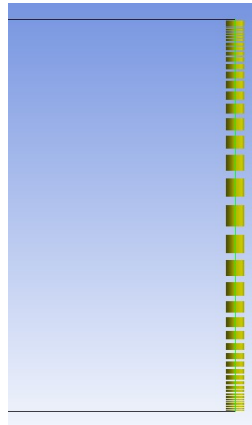
5.8 Right click on **Mesh** and **Insert > Sizing**.

5.9 While holding **Ctrl** click on the edge shown below and click **Apply**.



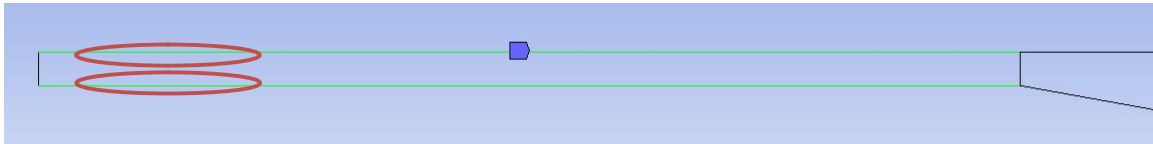
5.10 Change parameter for **Edge Sizing** as per below and click **Apply**.





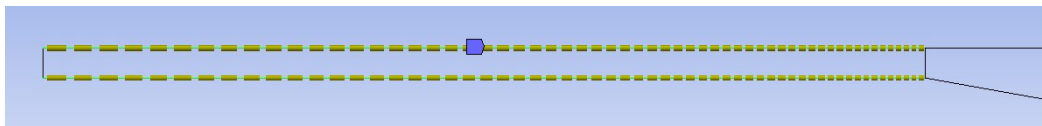
5.11 Right click on **Mesh** and **Insert > Sizing**.

5.12 While holding **Ctrl** click on the edge shown below and click **Apply**.



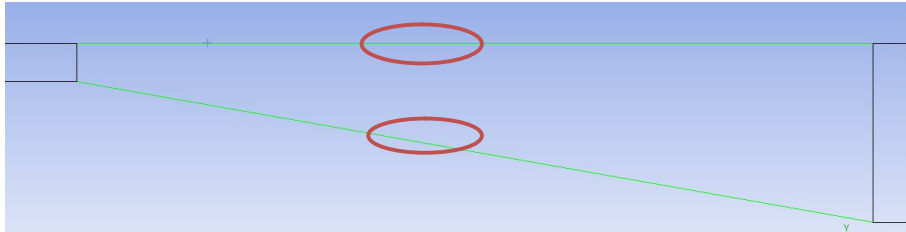
5.13 Change parameter for **Edge Sizing** as per below and click **Apply**.

Details of "Edge Sizing 3" - Sizing	
Scope	
Scoping Method	Geometry Selection
Geometry	2 Edges
Definition	
Suppressed	No
Type	Number of Divisions
<input type="checkbox"/> Number of Divisions	59
Behavior	Hard
Bias Type	-----
Bias Option	Bias Factor
<input type="checkbox"/> Bias Factor	3.6776

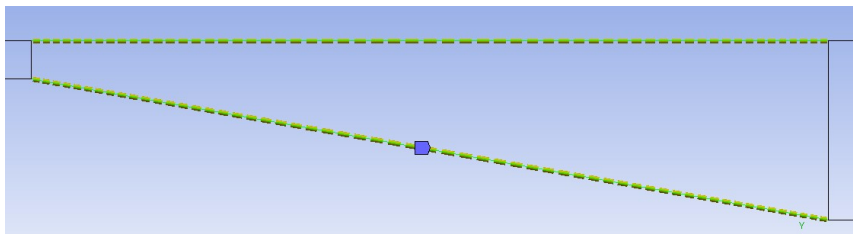
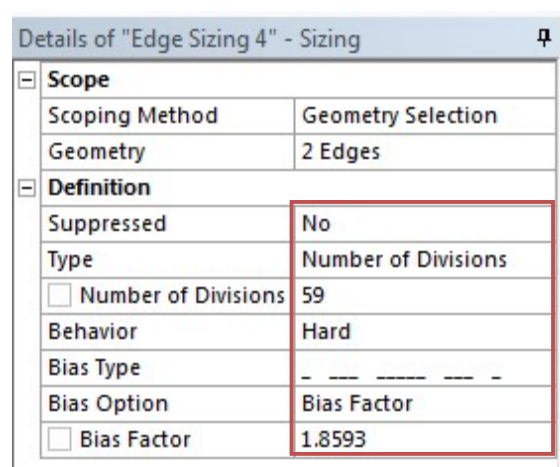


5.14 Right click on **Mesh** and **Insert > Sizing**.

5.15 While holding **Ctrl** click on the edge shown below and click **Apply**.

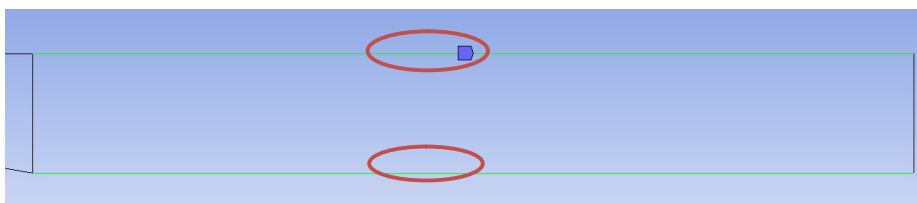


5.16 Change parameter for **Edge Sizing** as per below and click **Apply**.

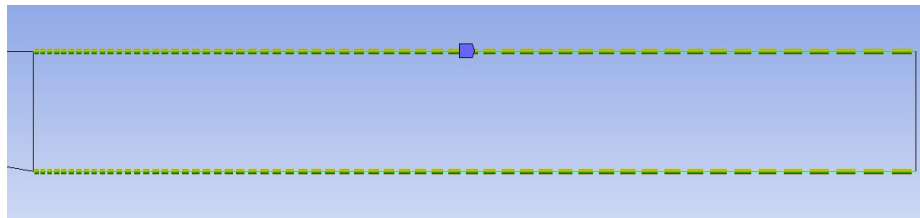
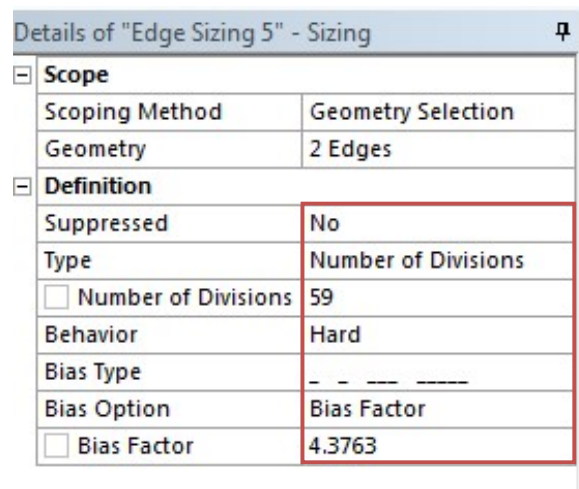


5.17 Right click on **Mesh** and **Insert > Sizing**.

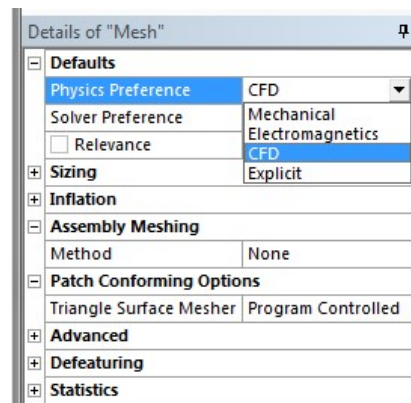
5.18 While holding **Ctrl** click on the edge shown below and click **Apply**.



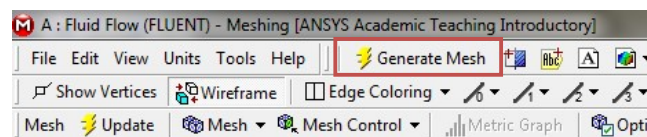
5.19 Change parameter for **Edge Sizing** as per below and click **Apply**.

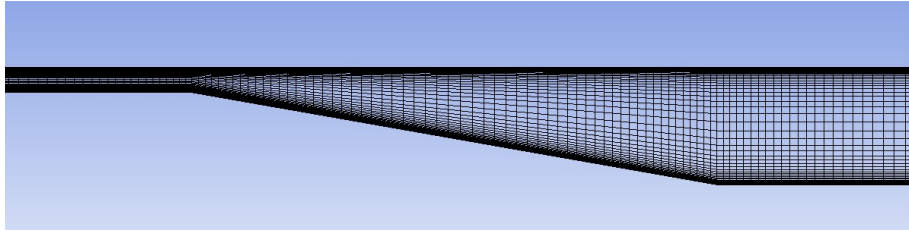


5.20 **Mesh > Physics Preference.** Change from **Mechanical** to **CFD**.

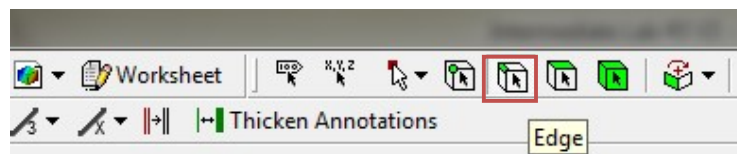
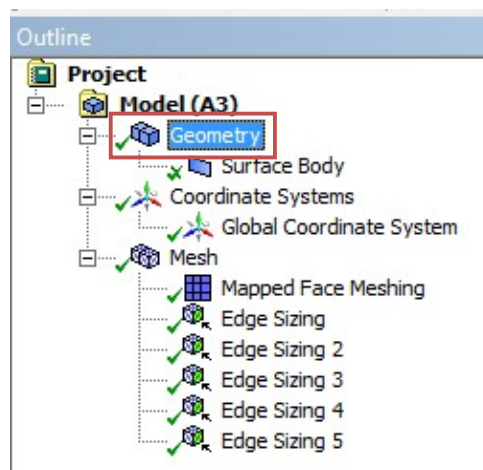


5.21 Click the **Generate Mesh** button.

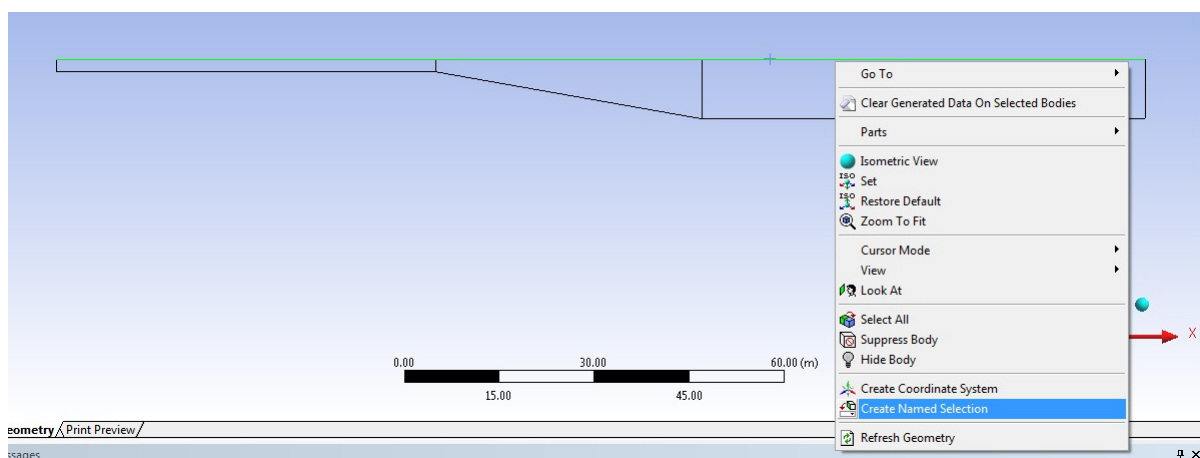


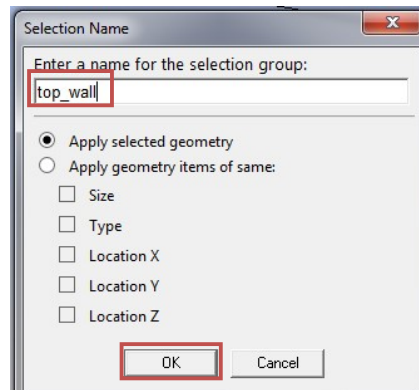


5.22 Select **Geometry** and click the **Edge** button.



5.23 While holding the **Ctrl** button select the three top edges and right click on them, then select **Create Named Selection**. Change the name to *top_wall* and click **OK**. Similarly name the *bottom_wall*, *inlet (left)* and *outlet(right)*.





- 5.24 **File > Save Project** and quit ANSYS Mesh.
- 5.25 Repeat this process for 4 degree half angle cases.
- 5.26 You should have the project schematic below.

Project Schematic

