

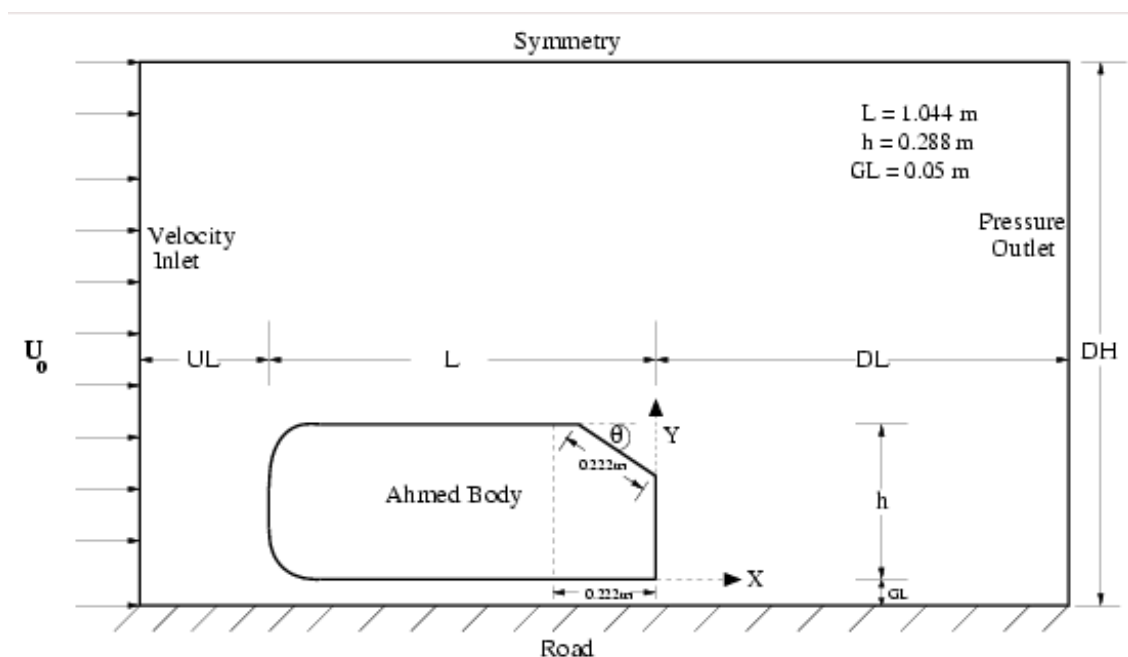
Simulation of Turbulent Flow over the Ahmed Body

1. Purpose

The Purpose of Task6 is to simulate **unsteady turbulent** flows over the Ahmed body following the “CFD process” by an interactive step-by-step approach and conduct verifications using ANSYS Fluent. Students will have “hands-on” experiences using ANSYS to **predict drag coefficients and axial velocity for slant angle 25 degrees and compare them with experimental data**. Students will use post-processing tools (streamlines, velocity vectors, contours, animations) to **visualize the mean and instantaneous flow fields and compute the non-dimensional shedding frequency (Strouhal number)**. Students will analyze the differences between CFD and experimental data and present results in a report.

2. Simulation Design

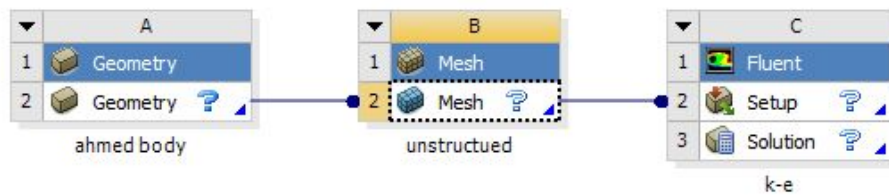
The problem to be solved is unsteady turbulent flows over the Ahmed body (2D). Reynolds number is around 768,000 based on inlet velocity and vehicle height (h). The following figure shows the sketch window you will see in ANSYS with definitions for all geometry parameters. The origin of the simulation is located at the rear of the body. θ is the slant angle. L is the length of the body and h is the height of the body. Uniform velocity specified at inlet and constant pressure specified at outlet. The top boundary of the simulation domain is regarded as “Symmetry” and there is a distance between the car body and road, GL.



3. Project Schematic in ANSYS Workbench

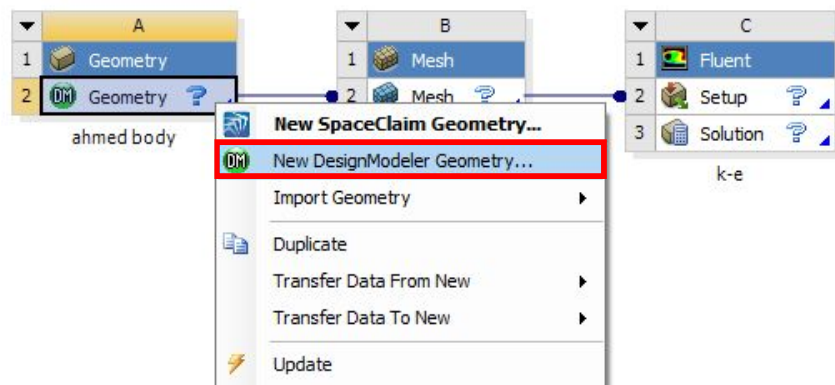
3.1. Start ANSYS Workbench.

3.2. Drag and drop three component into the **Project Schematic**, name the components and create connections between components as per below.

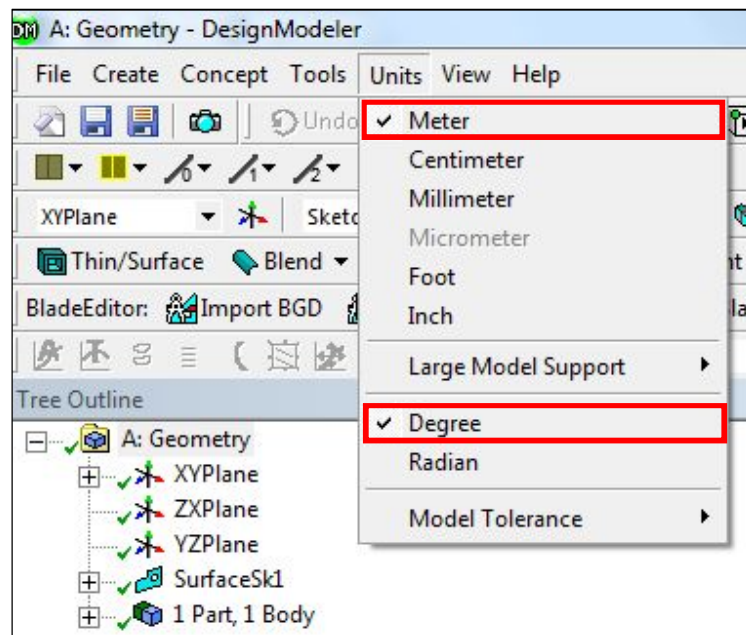


4. Geometry Creation

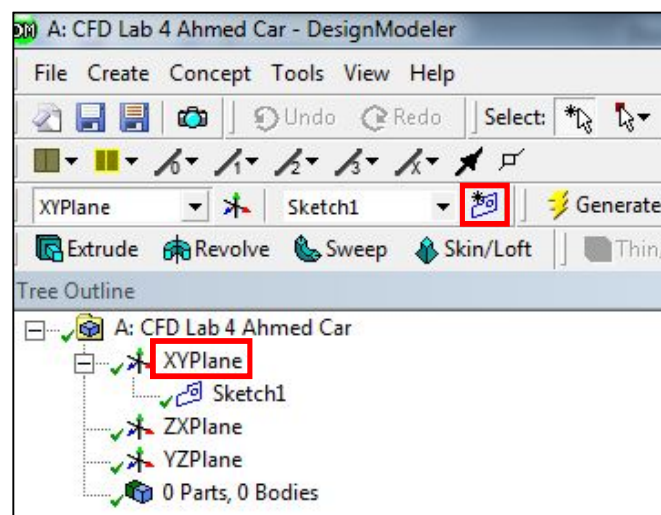
4.1. From the **Project Schematic** right click **Geometry** and select **New DesignModeler Geometry....**



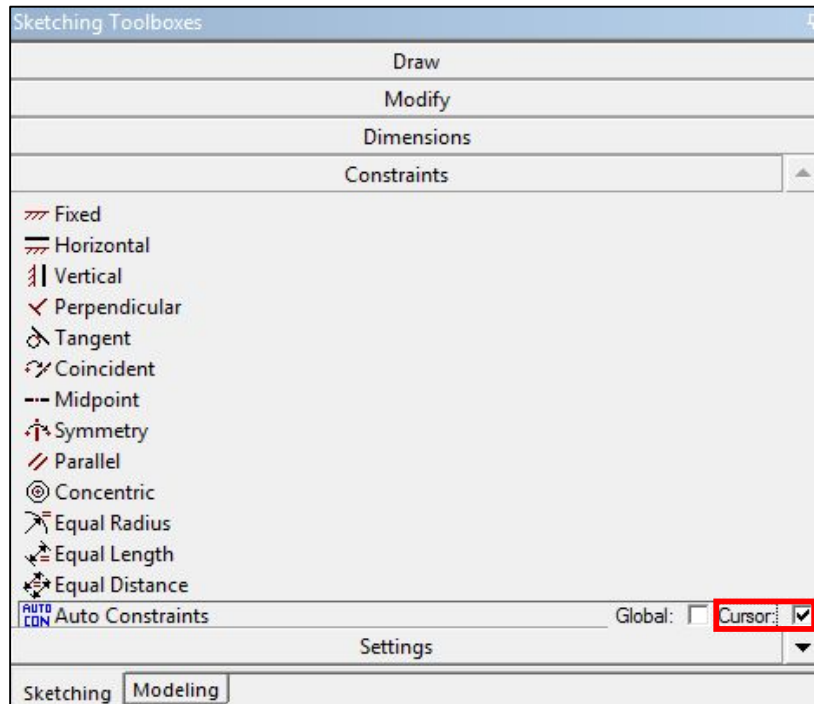
4.2. Make sure that Unit is set to **Meter** and **Degree** (default settings).



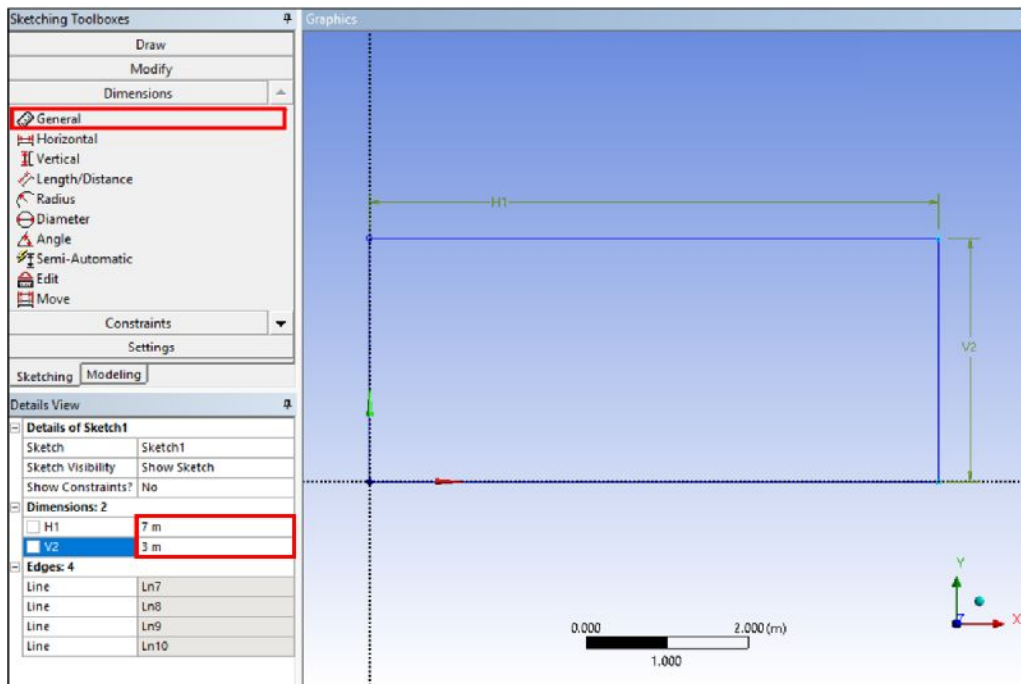
4.3. Select the **XYPlane** then click the **New Sketch** button.



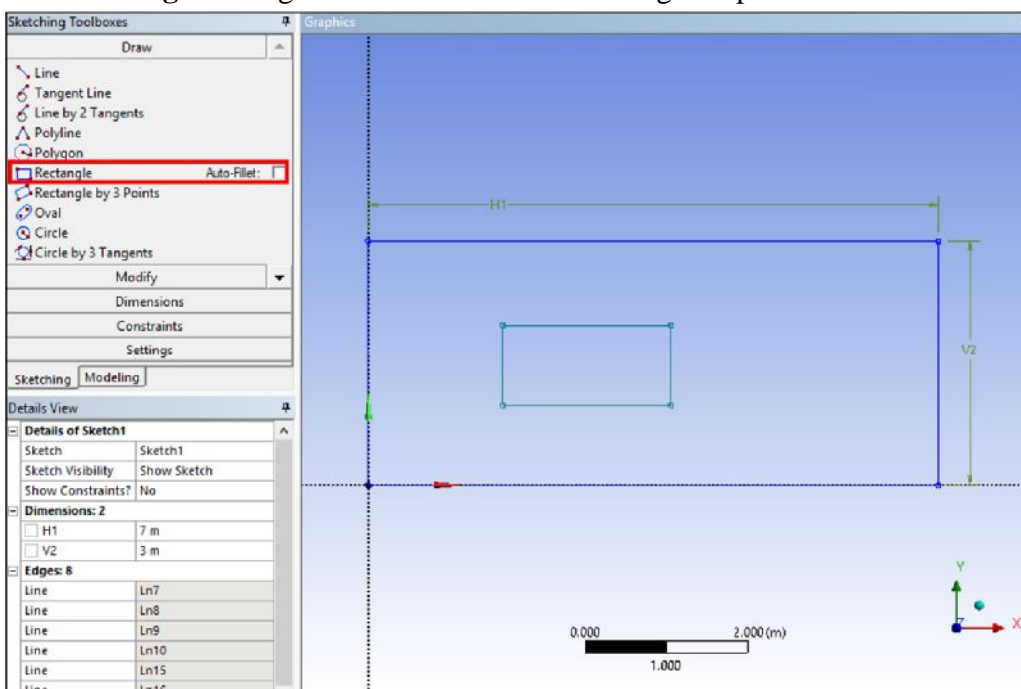
4.4. Enable the auto constraints option to pick the exact point as below



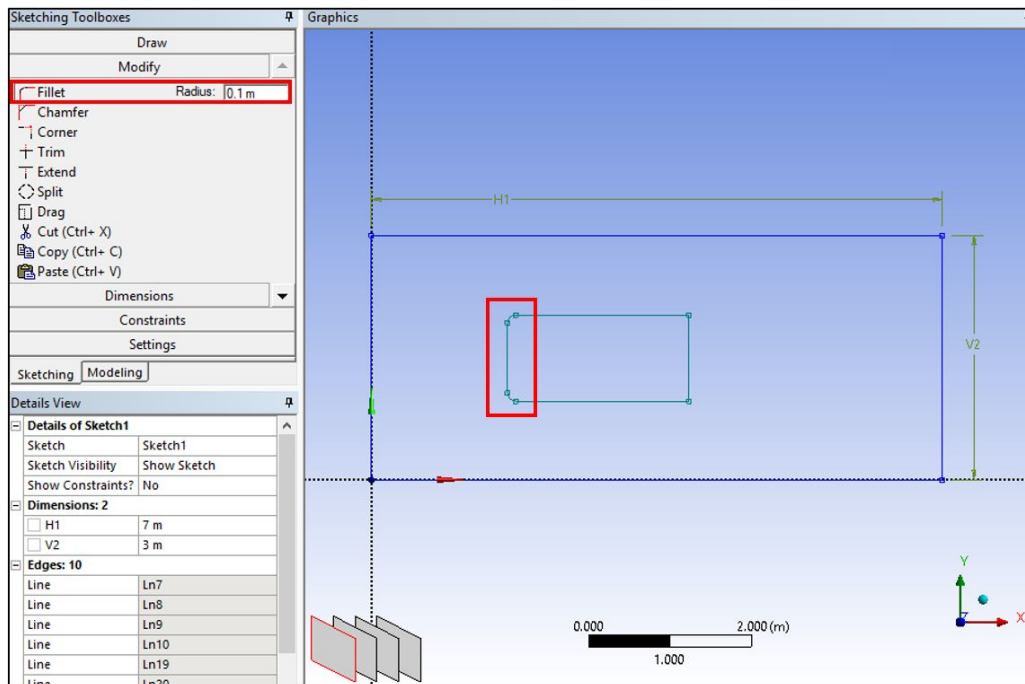
- 4.5. Use the **Rectangle** tool under **Draw** to make a rectangle starting from the origin and ending inside the first quadrant. Dimension it using **General** dimension as per below. (Click the z arrow of the 3D orientation located at the bottom right to make the view perpendicular to xy-plane. Make sure to click the origin when the mouse cursor is changed to “P”)



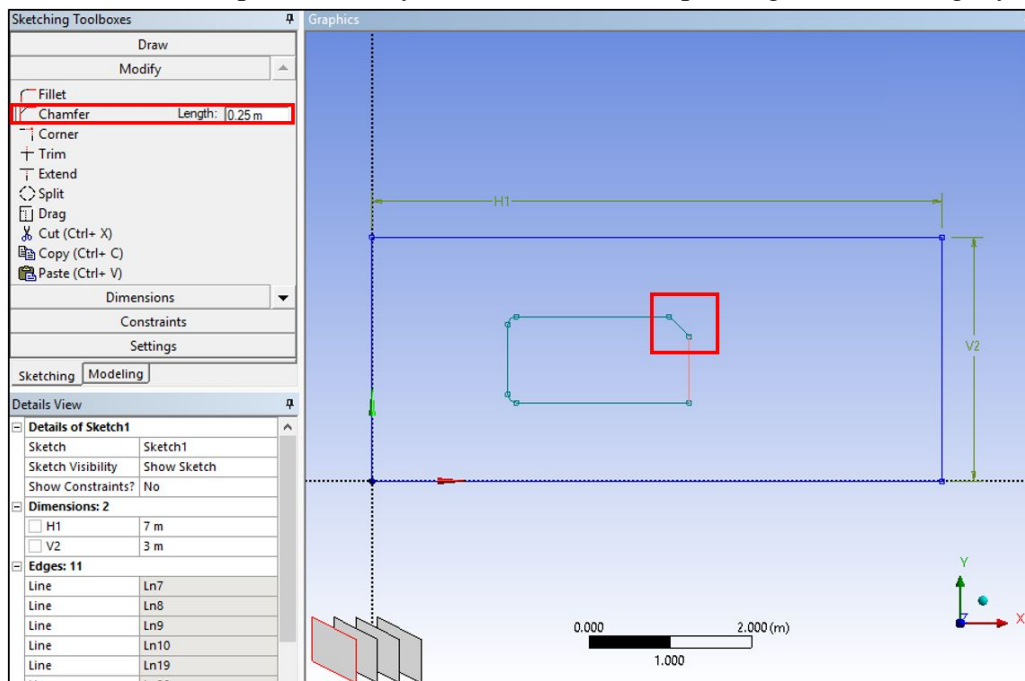
4.6. Use the **Rectangle** tool again to draw an another rectangle as per below.



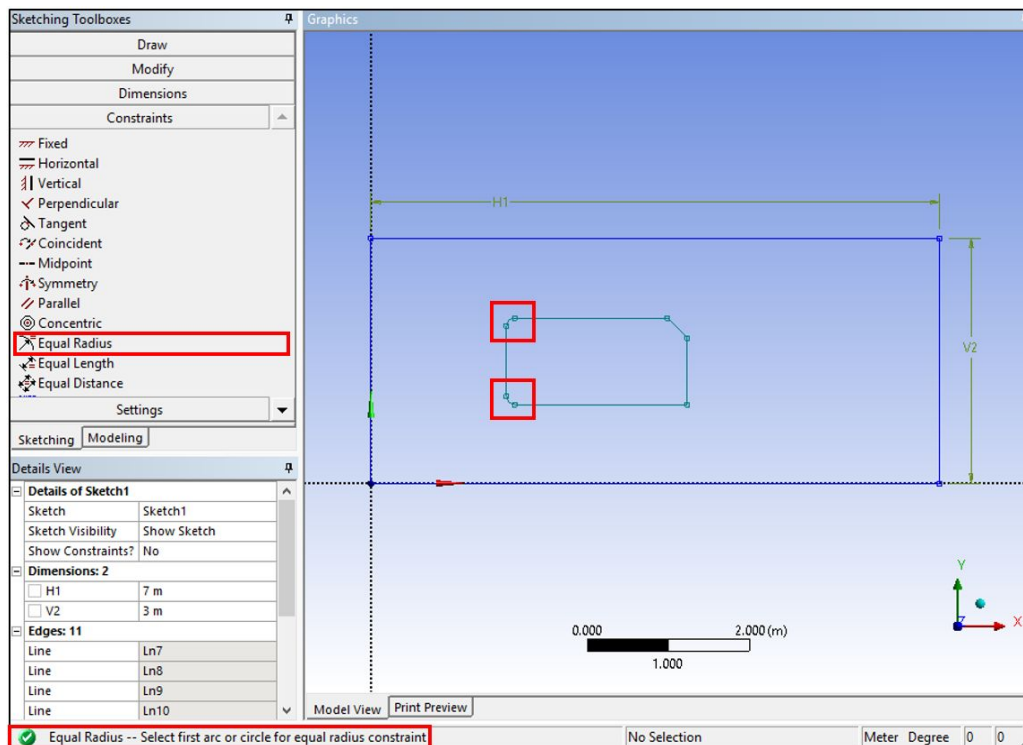
4.7. Use the **Fillet** tool in **Modify** to put a radius on the front corners of the Ahmed Car as per below. Use the **Radius** size of **0.1m**. After changing the value, click the front corners.



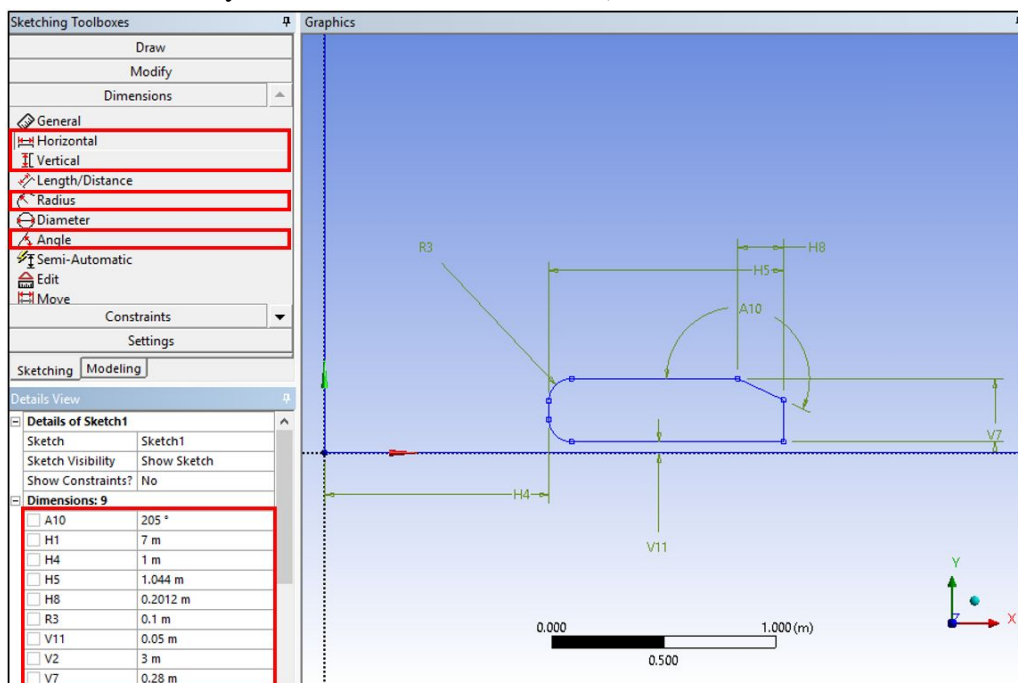
4.8. Use the **Chamfer** tool to put a chamfer on the back of the Ahmed Car as per below. Use the **Length** of **0.25m**. (Shape of the body could be different depending on the rectangle you made)



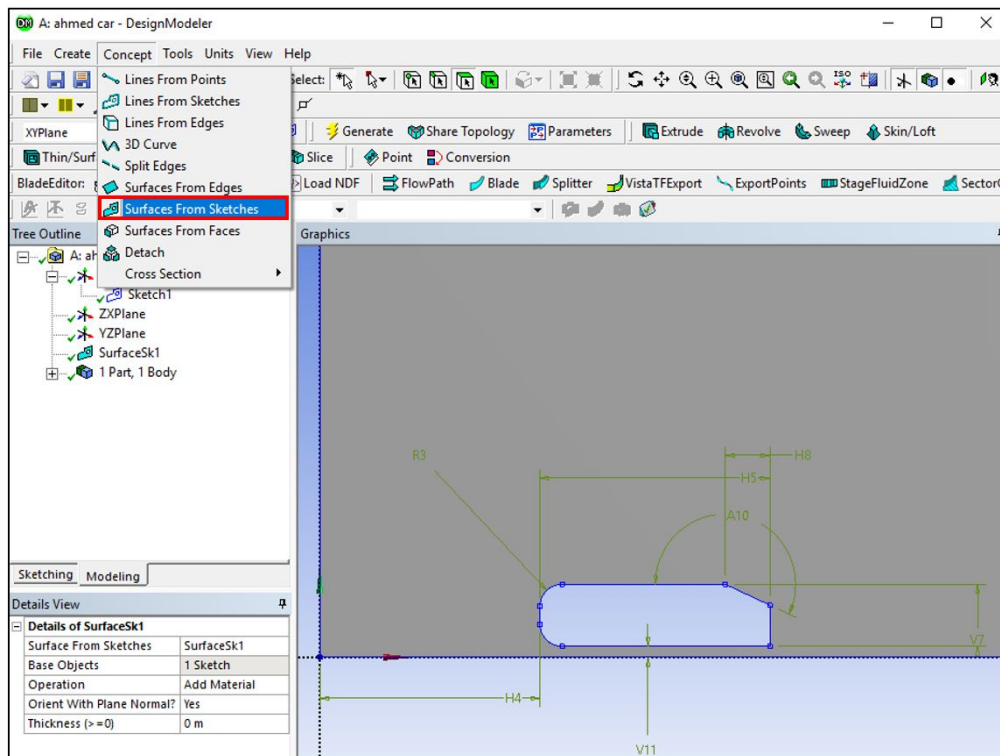
4.9. Put a constraint on the two radii using the **Equal Radius** tool in **Constraints**. (Note: in the bottom left corner next to the checkmark in a green circle is the note on how to use a tool)



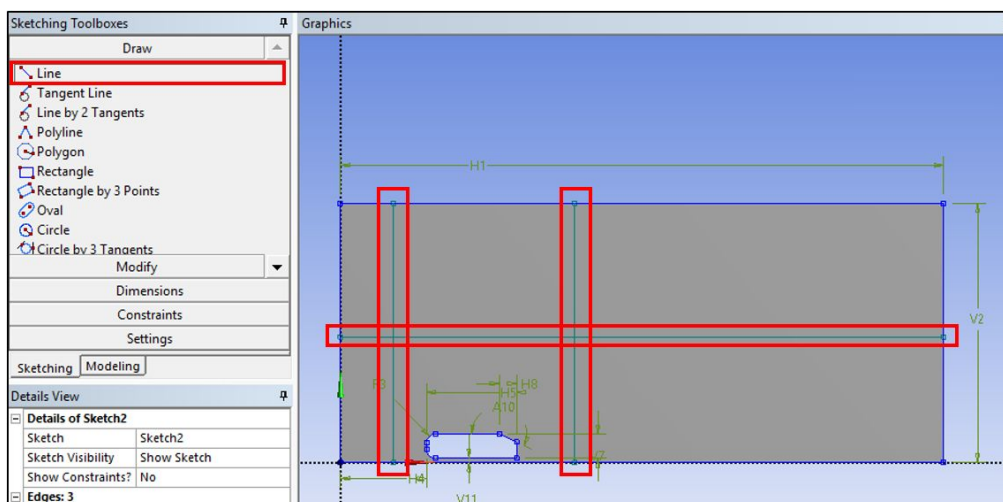
4.10. Dimension the body inside the rectangle as per below using **Horizontal**, **Vertical**, **Radius**, and **Angle** under **Dimensions**. (The name of each dimension will be followed by the order you make it, so it may be different from the manual)



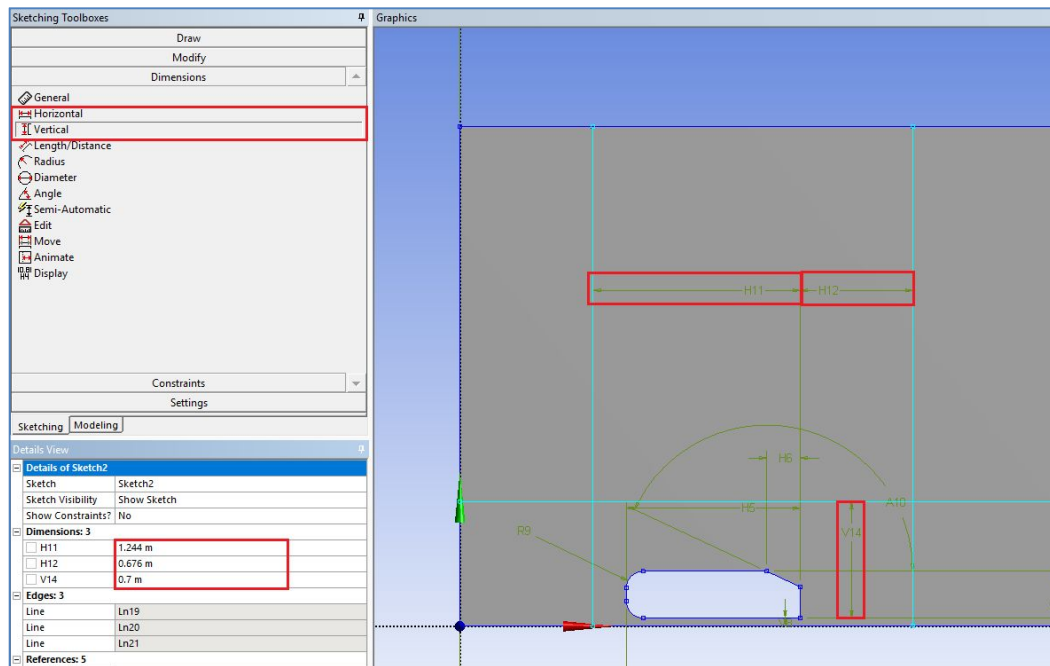
4.11. Concept > Surface From Sketches. Select the sketch you just created under the tree outline and click **Apply**. Click **Generate**.



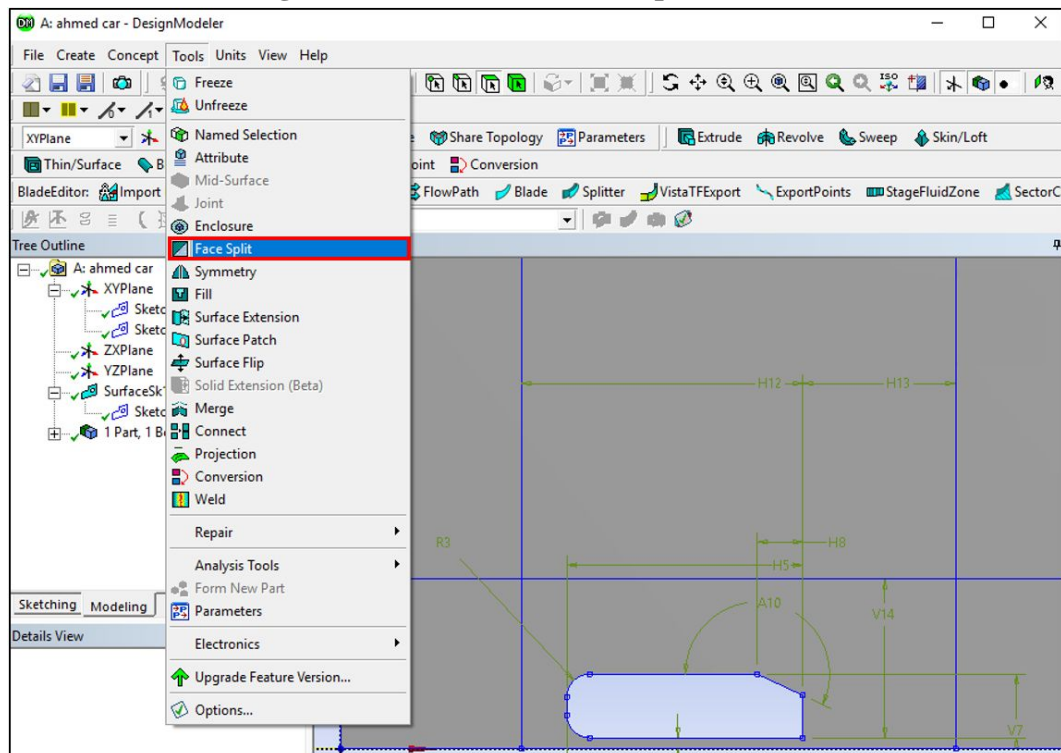
4.12. Select XYPlane and click the New Sketch button. In this new sketch, use the **Line** tool under **Draw** to make the lines as per below. Three lines will extend over the entire domain, horizontally or vertically. Make sure that the **C** appears when you are on the line and the **V/H** appears next to the line being created, ensuring that you are pointing on the edge and the line is vertical/horizontal.



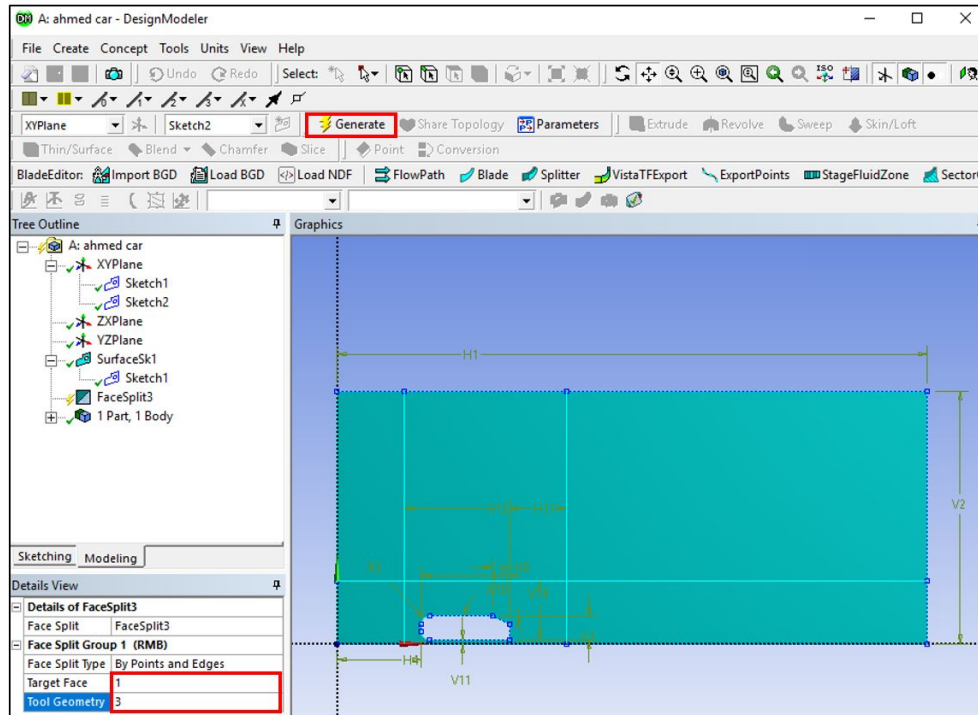
4.13. Use the **Horizontal** and **Vertical** dimension tool to dimension the lines as per below.



4.14. Go back to **Modeling** tab and then **Tools > Face Split**.

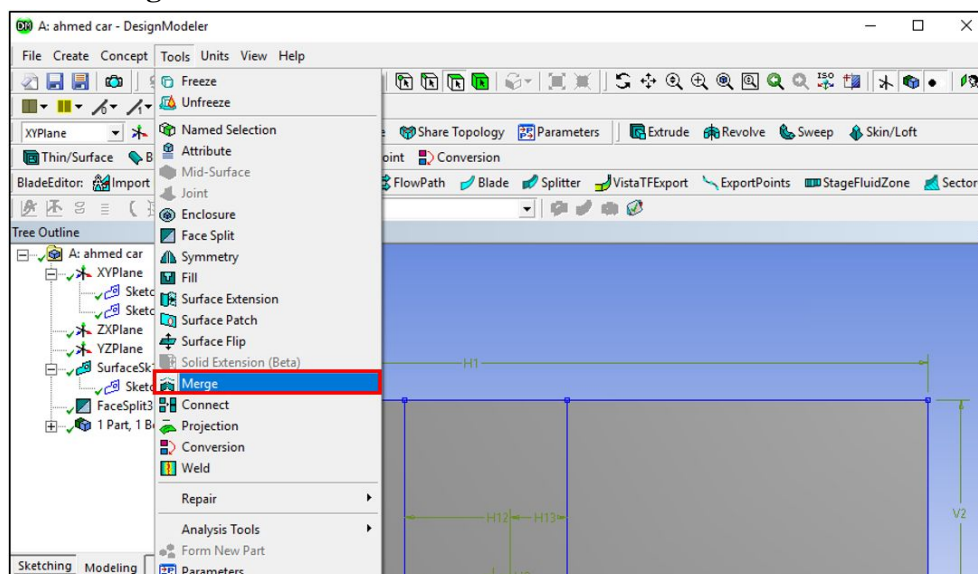


- 4.15. Select the gray surface for **Target Face** and click **Apply**. For **Tool Geometry** select two endpoints of the one line you just created while holding **Ctrl** then click **Apply**. Select **Tool Geometry** again and select two more endpoints of another line and click **Apply**. Repeat this process for the last line and click **Apply**. Click **Generate**. This splits the surface into six pieces.

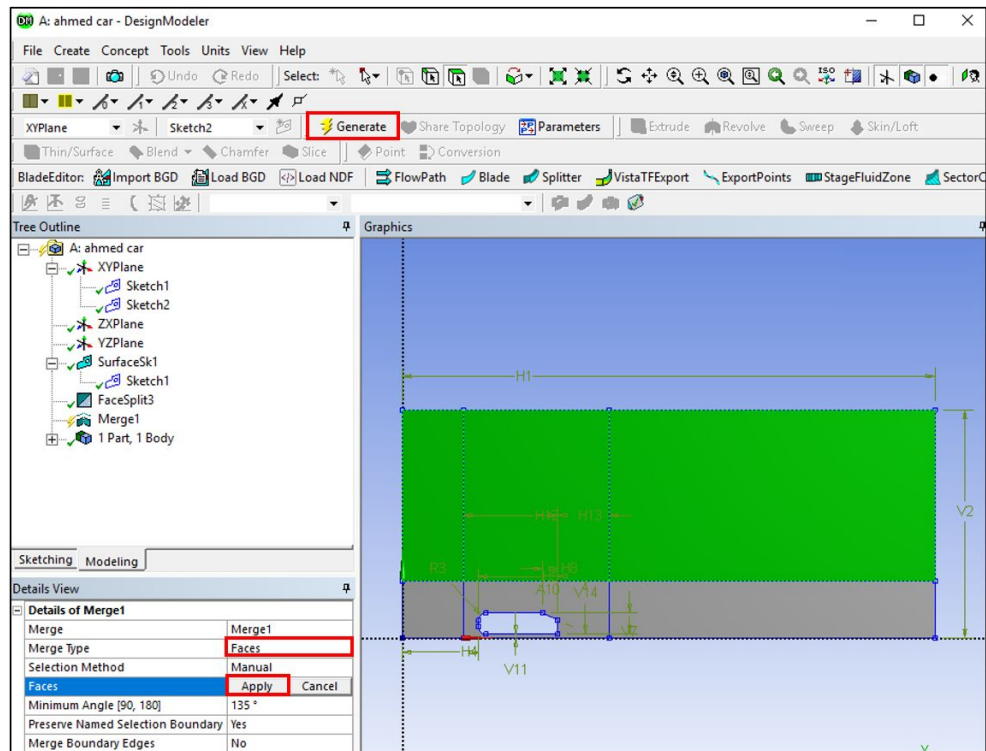


(The picture above was taken right before clicking 'Generate')

4.16. Tools > Merge.



4.17. Change the **Merge Type** to **Faces** and select the top three faces. Click **Apply** then **Generate**.

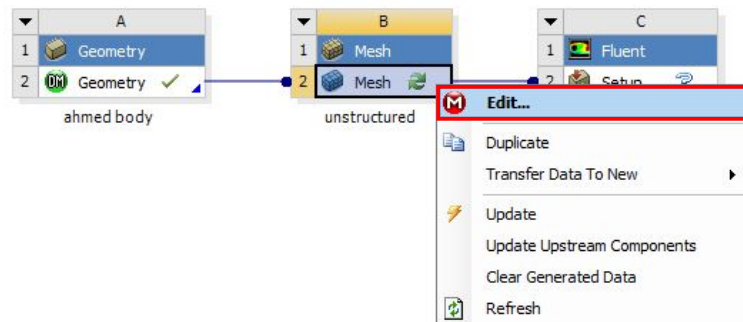


(The picture above was taken right before clicking 'Apply')

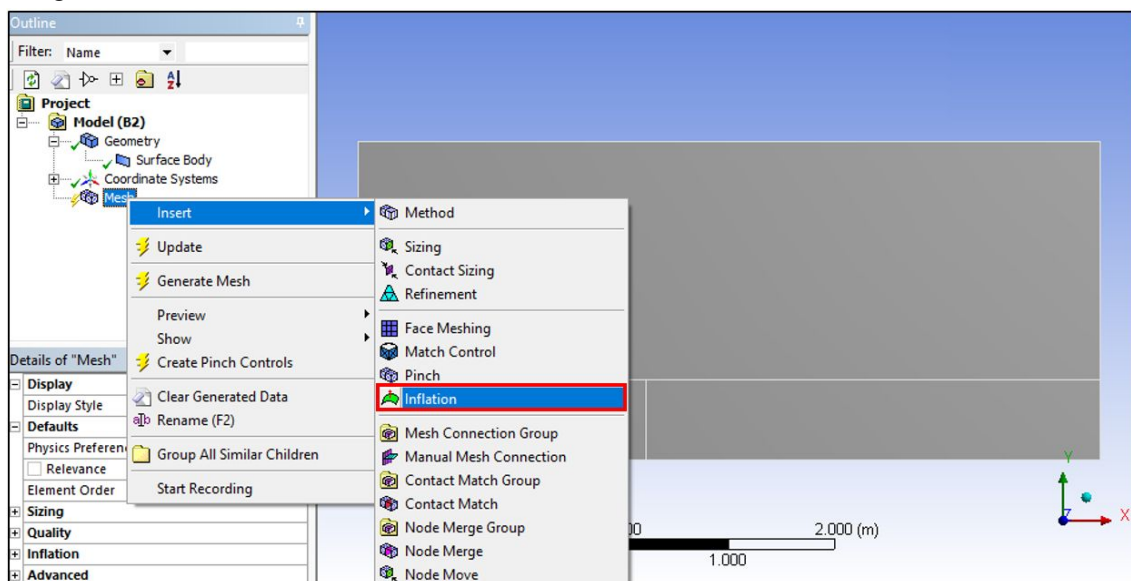
4.18. **File > Save Project.** Close the **Design Modeler** window.

5. Mesh

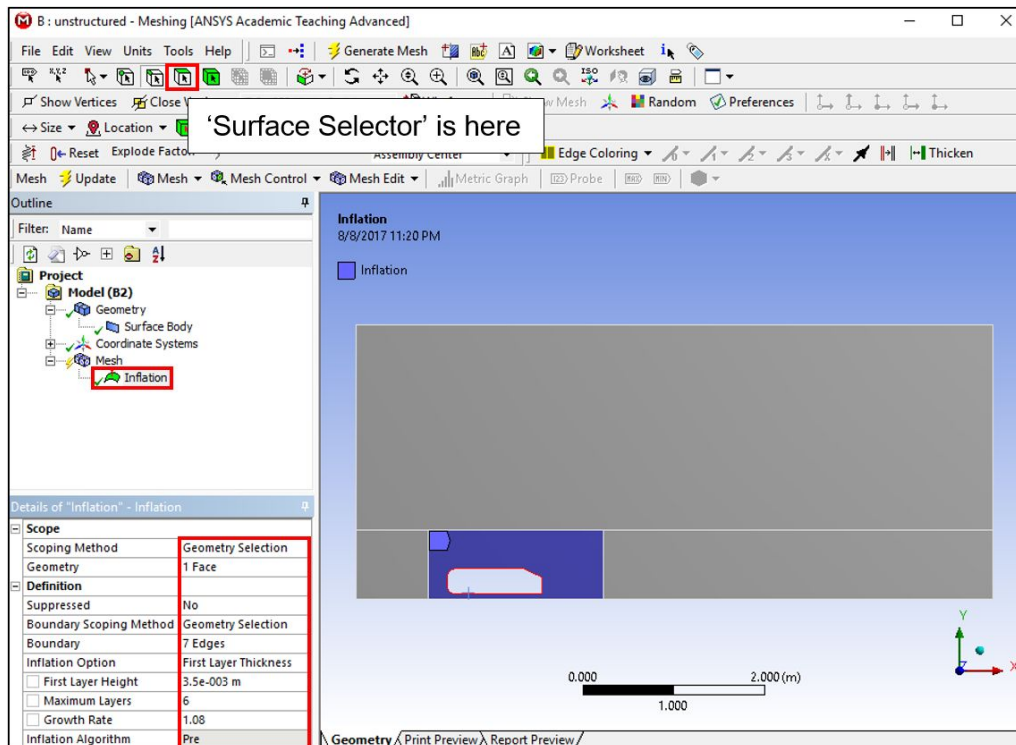
5.1. Right click **Mesh** and from the dropdown menu then select **Edit...**



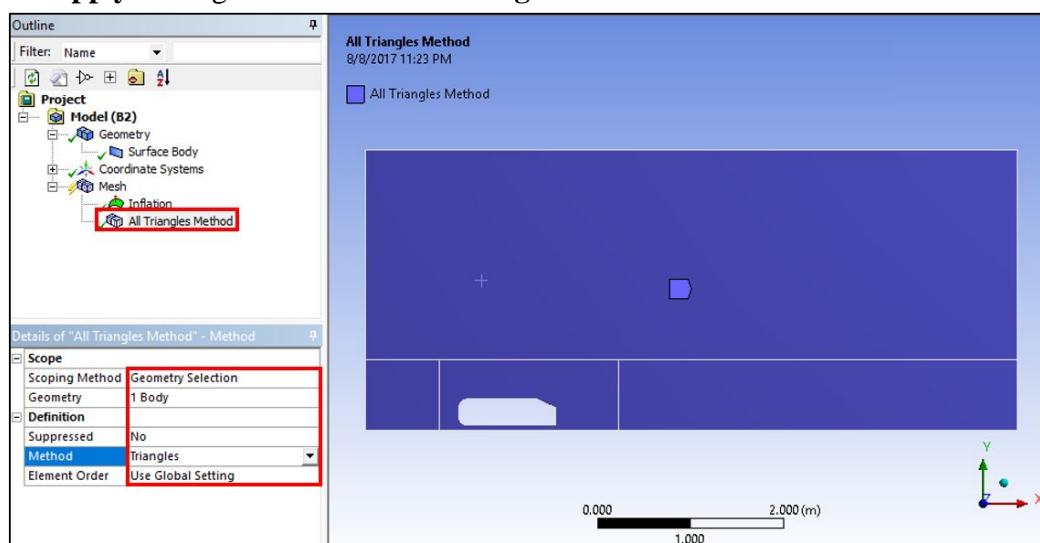
5.2. Right click on **Mesh** > **Insert** > **Inflation**.



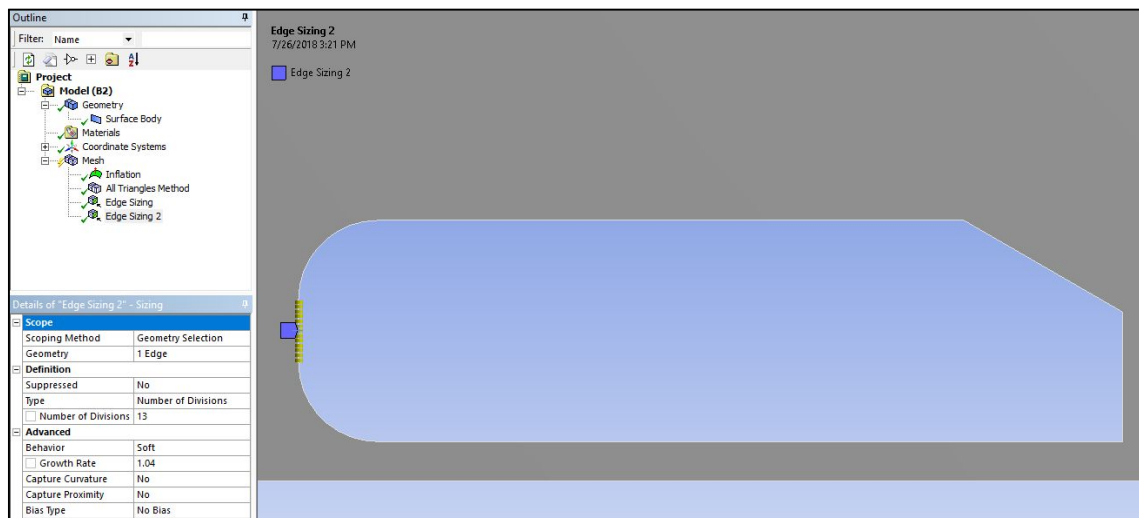
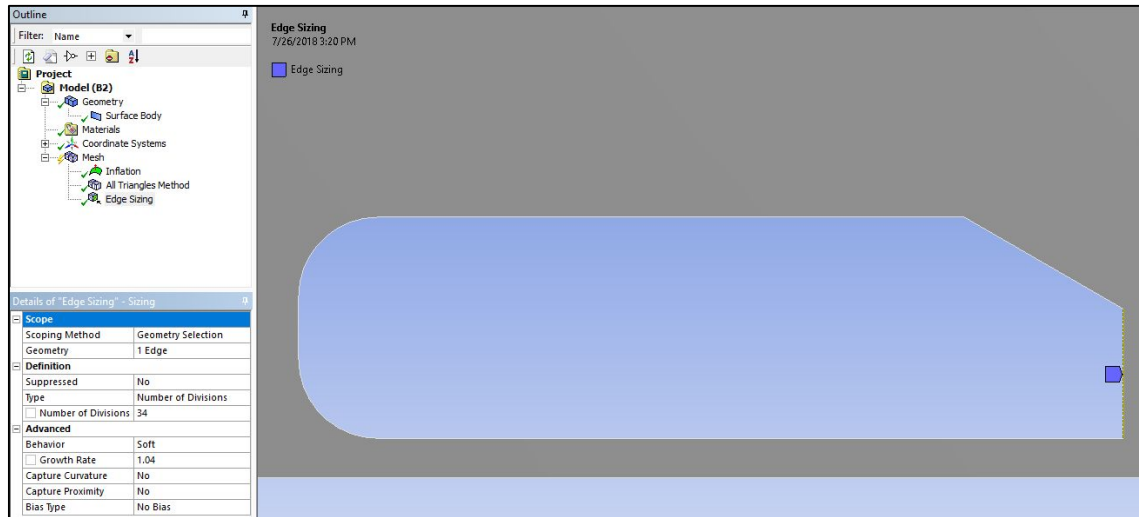
5.3. For **Geometry** option, select the surface of the domain which borders the Ahmed Car and click **Apply** (Change the cursor to 'Surface Selector' at upper region to select the surface). For the **Boundary**, select the edges of the Ahmed Car by holding **Ctrl** and selecting the edges and then click **Apply**. There should be seven edges selected for the **Boundary**. Change the parameters in **Details of "Inflation"** – **Inflation** as per below.

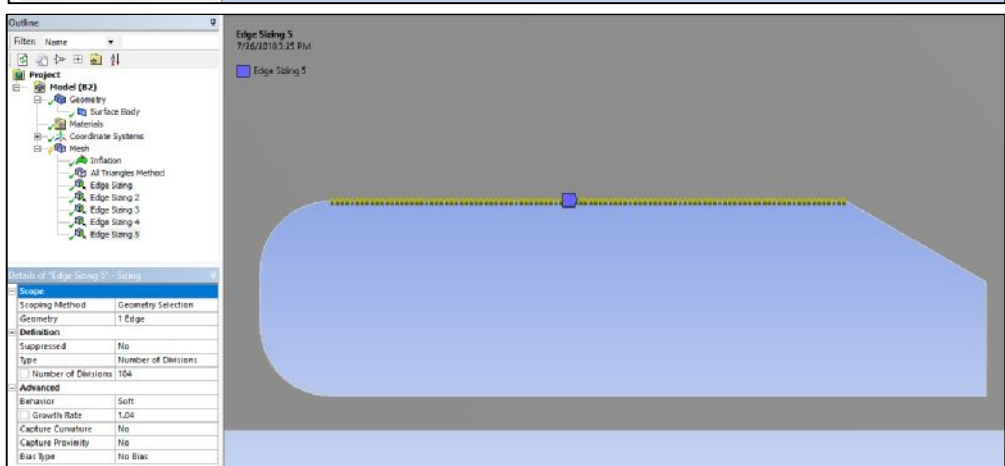


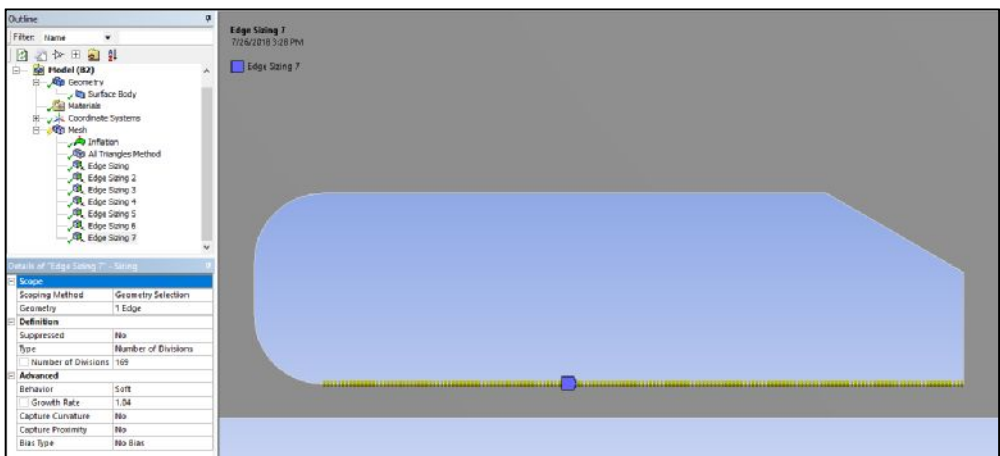
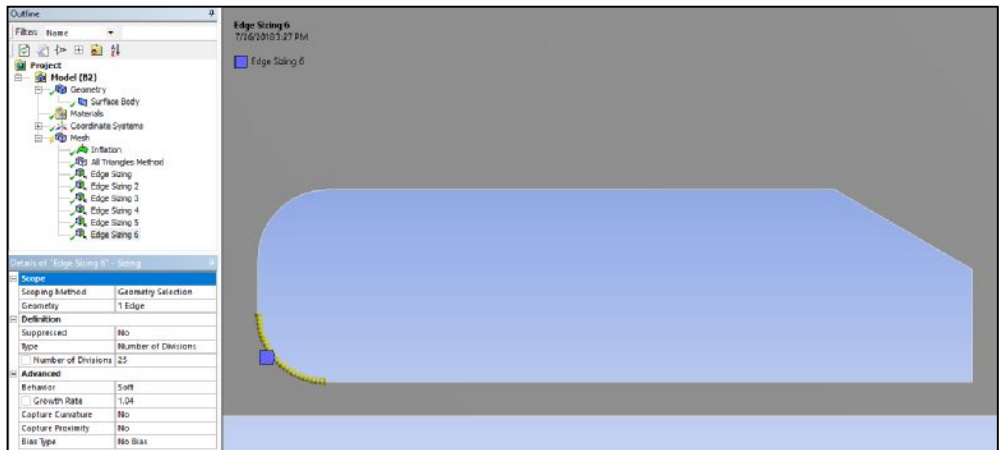
5.4. Right click **Mesh** > **Insert** > **Method**. Select the whole domain (surfaces) for Geometry and click **Apply**. Change the **Method** to **Triangles**.

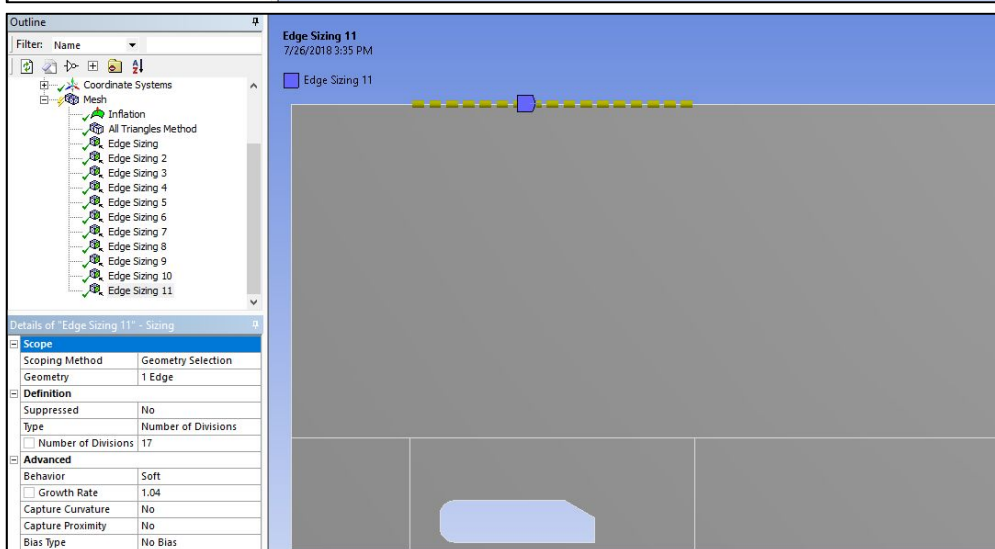
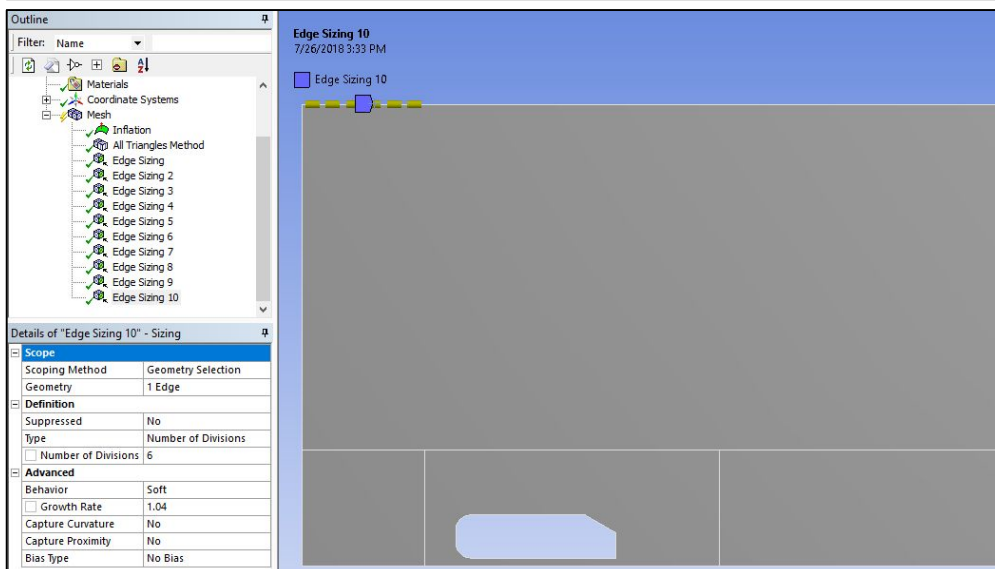
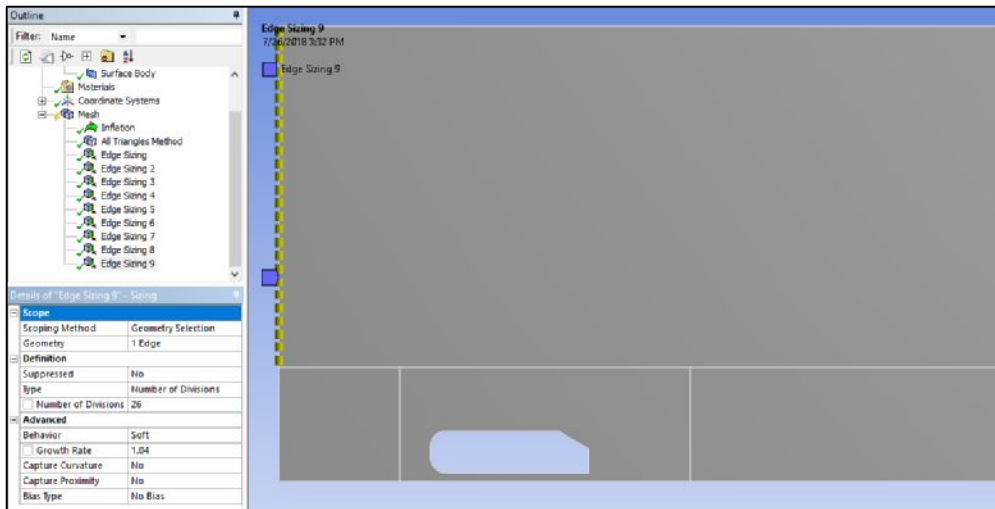


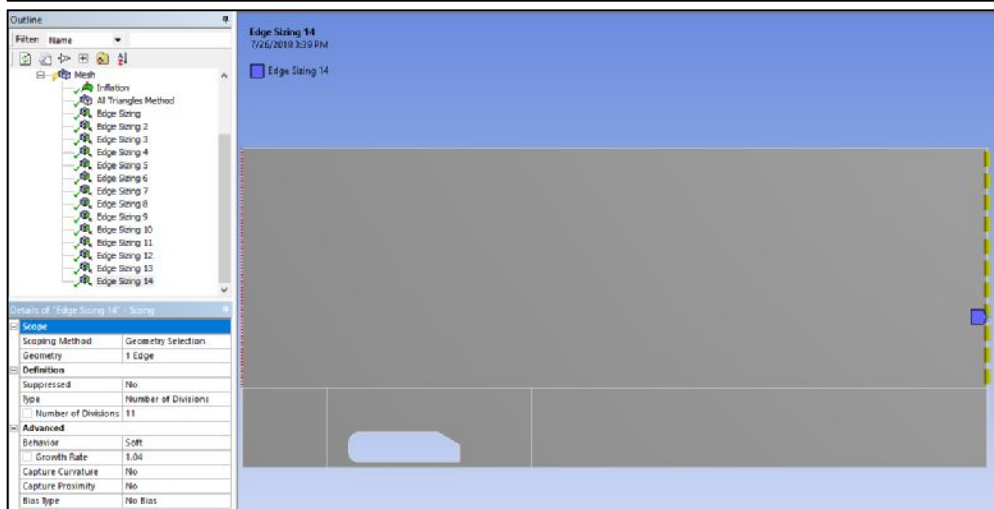
5.5. Right click **Mesh** > **Insert** > **Sizing**. Select the line as per below and click **Apply**. Change the parameters of sizing as per below. Repeat this for the following figures below. There should be **22 edge sizings** in total. Change the cursor to “Edge Selector” to select the edges.

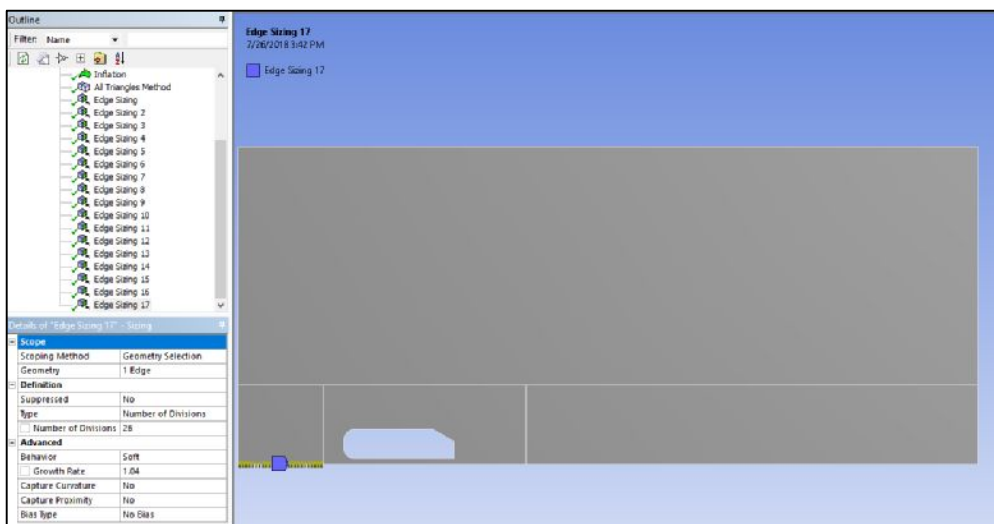
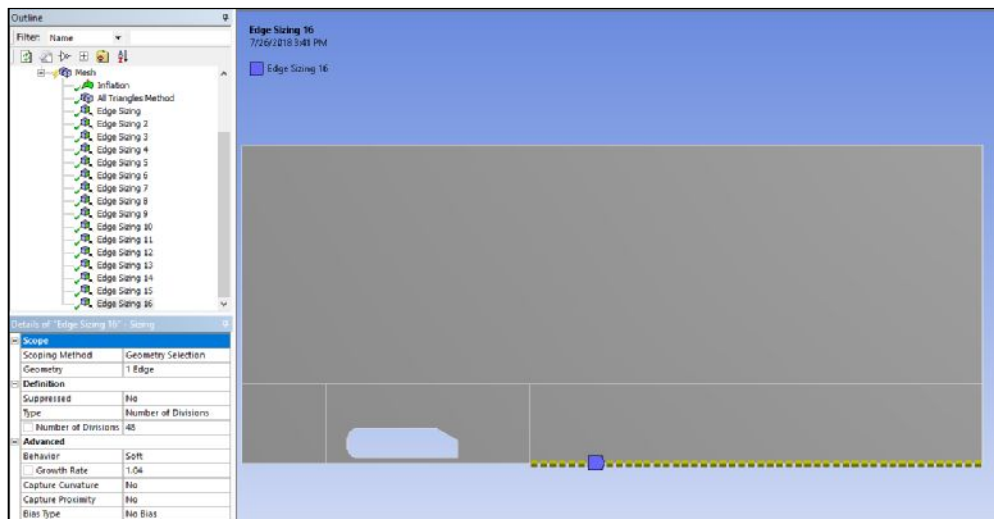
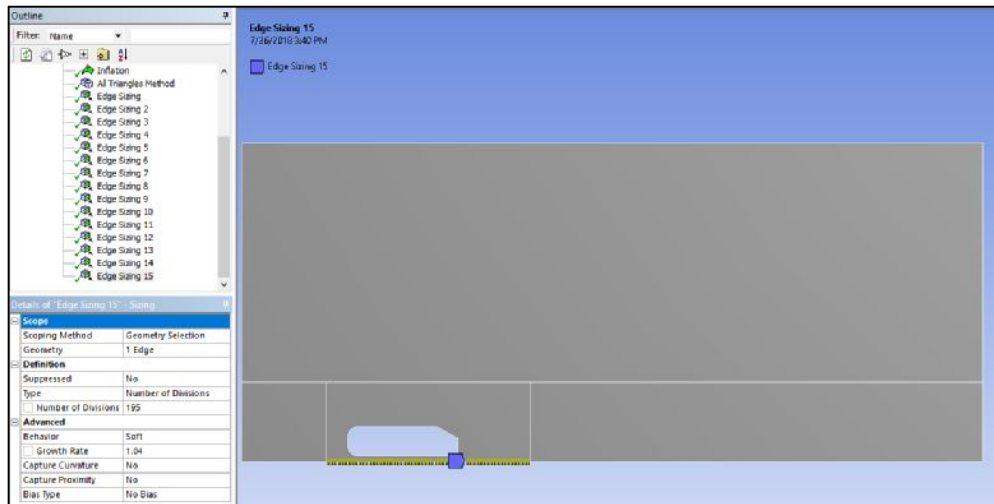


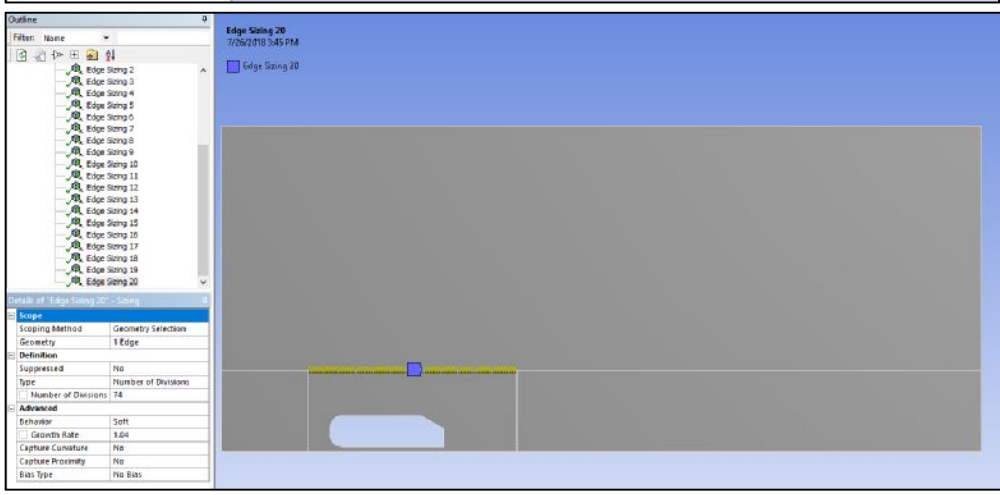
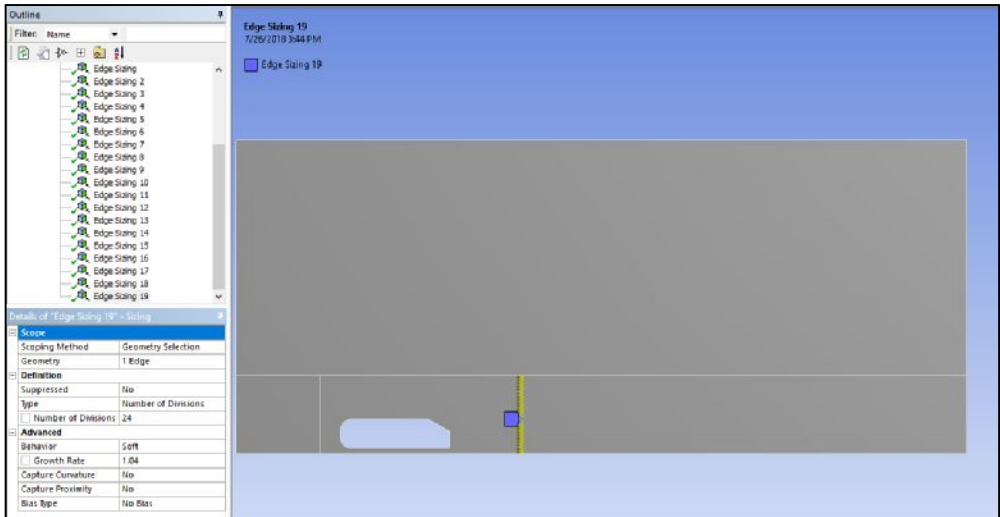
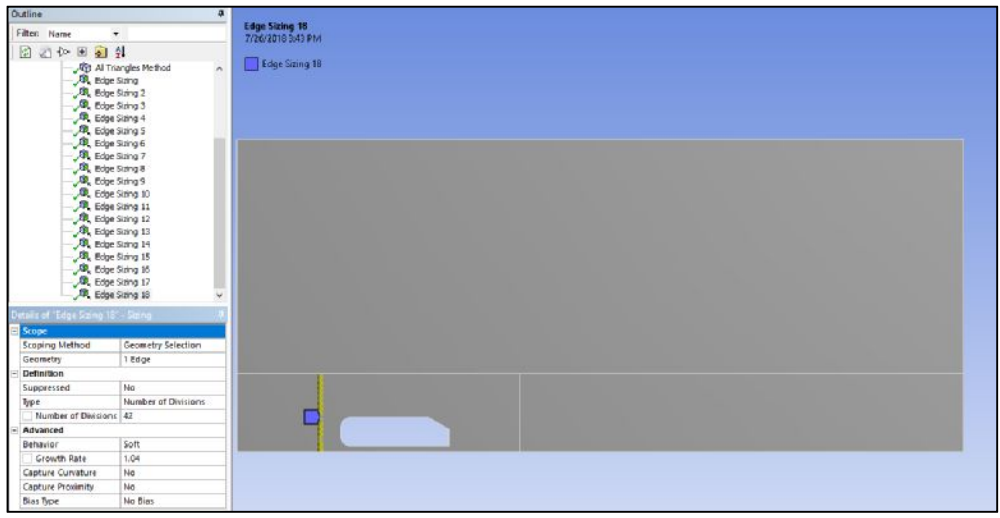


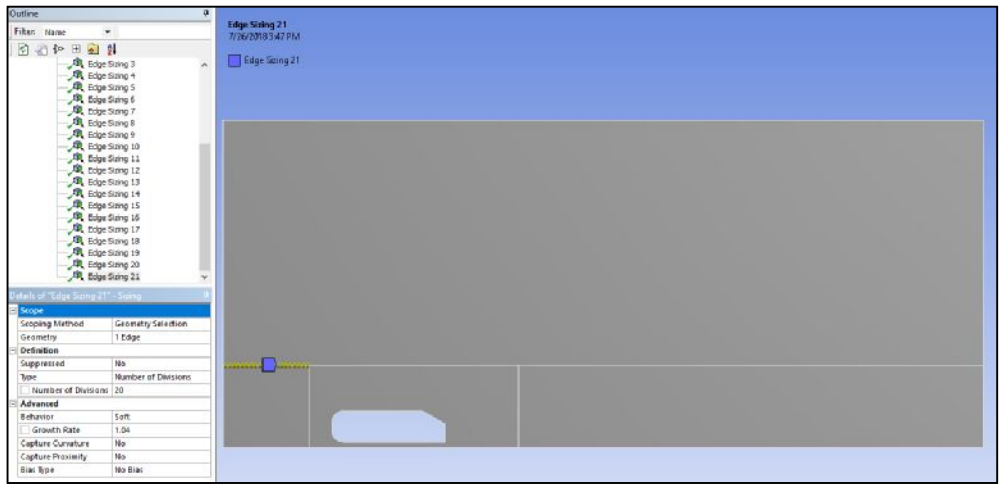






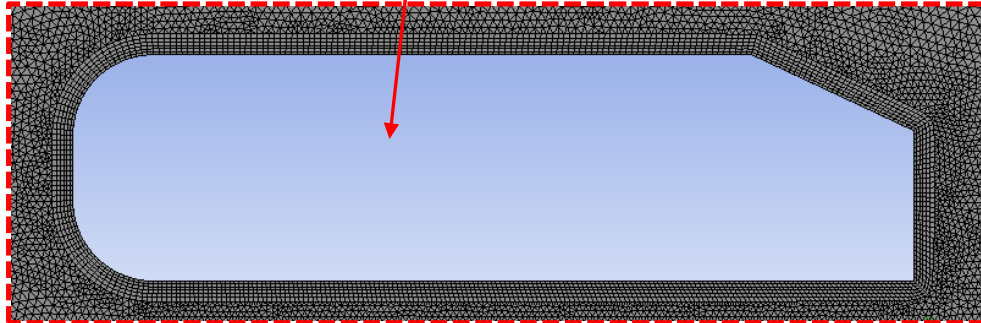
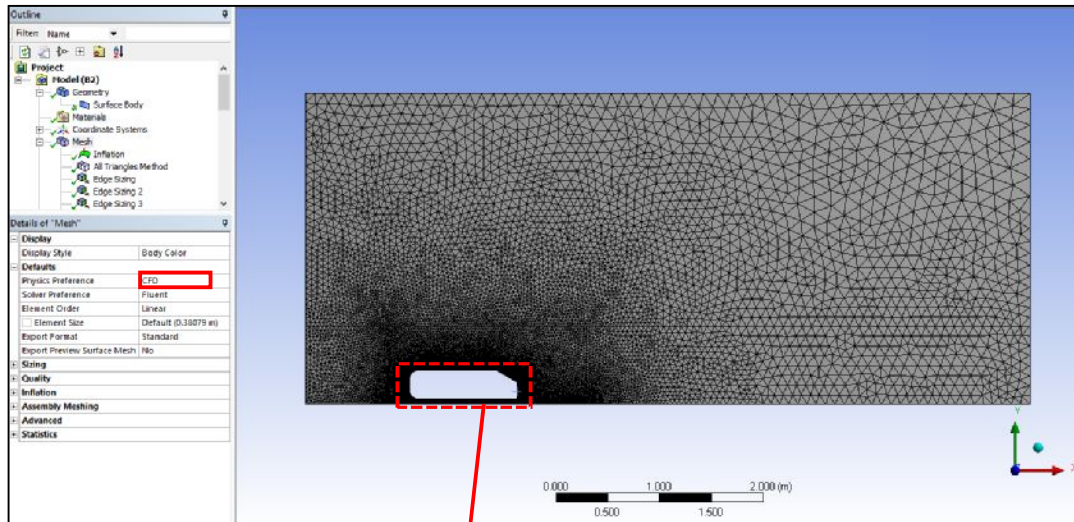




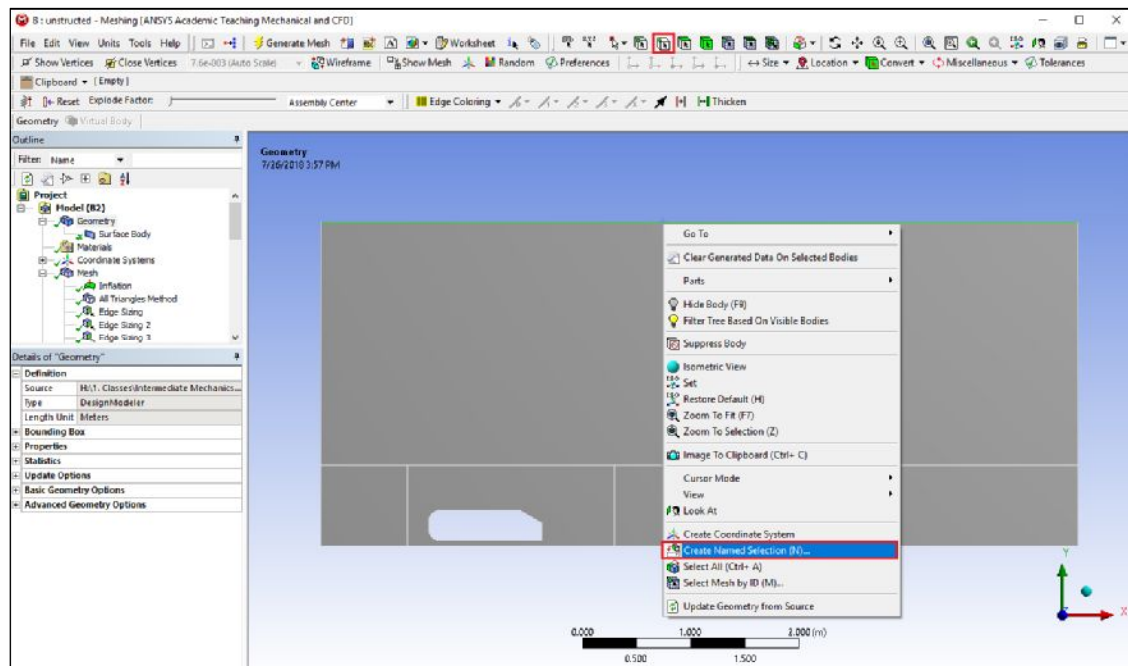


5.6. Click on **Mesh** under the **Outline** and change the **Physics Preference** to **CFD**.

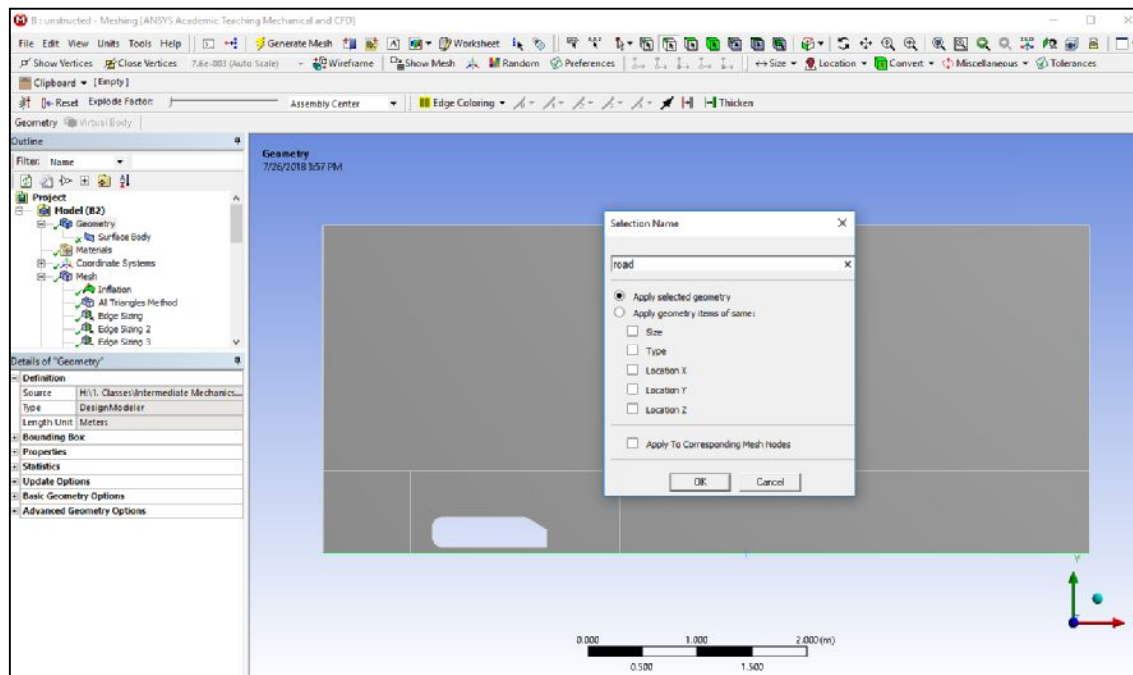
5.7. Click **Generate Mesh**.



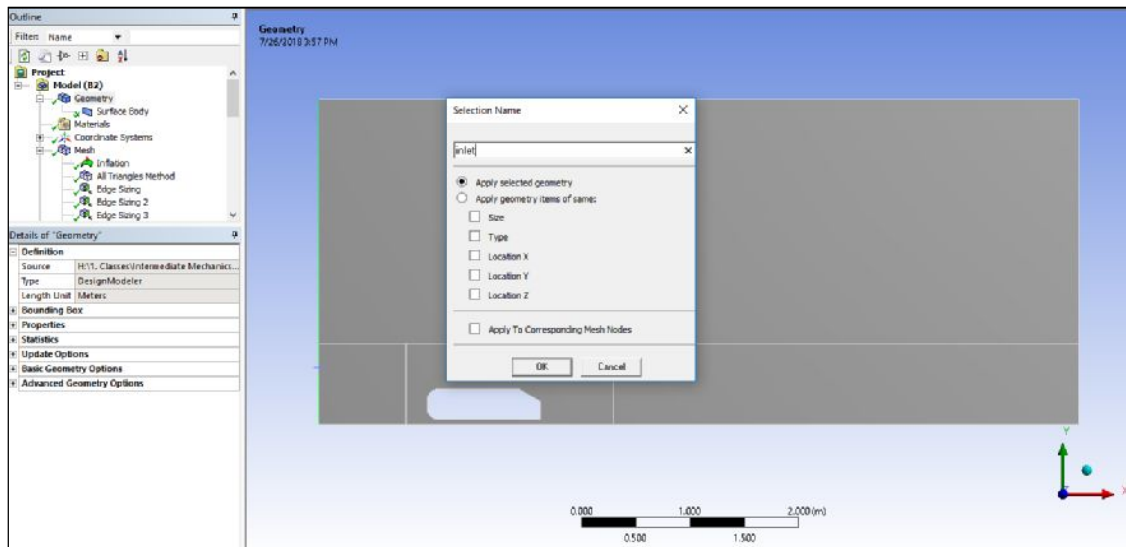
5.8. Select 'Edge Selector'. Select the top edges of the domain by holding **Ctrl** while selecting, right click, select **Create Named Selection** from the dropdown menu. Name the top edge '*symmetry*'.



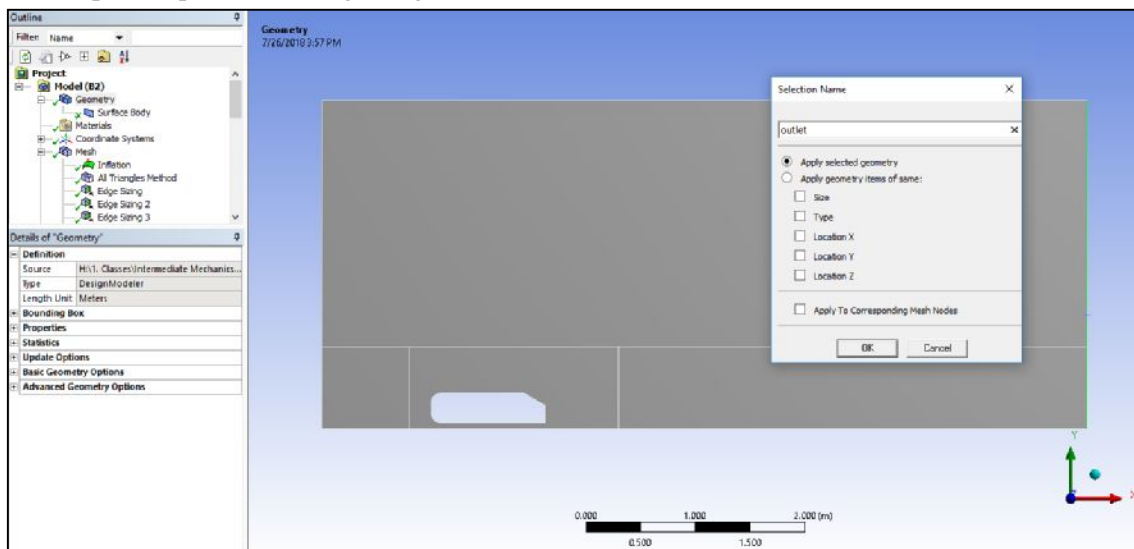
5.9. Repeat step 5.8 for the bottom edges and name them '*road*'.



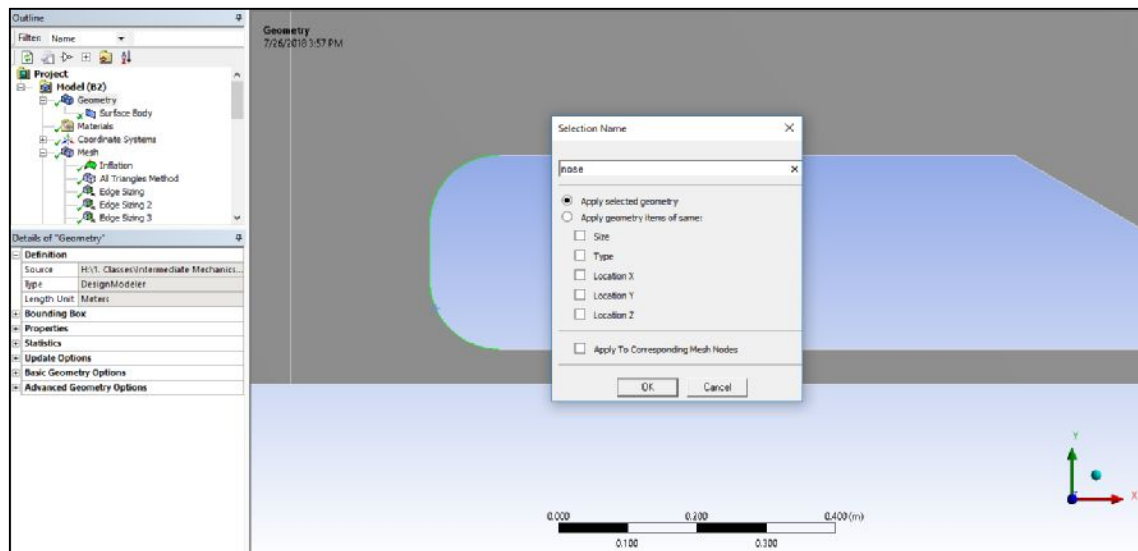
5.10. Repeat step 5.8 for the left edges and name them 'inlet'.



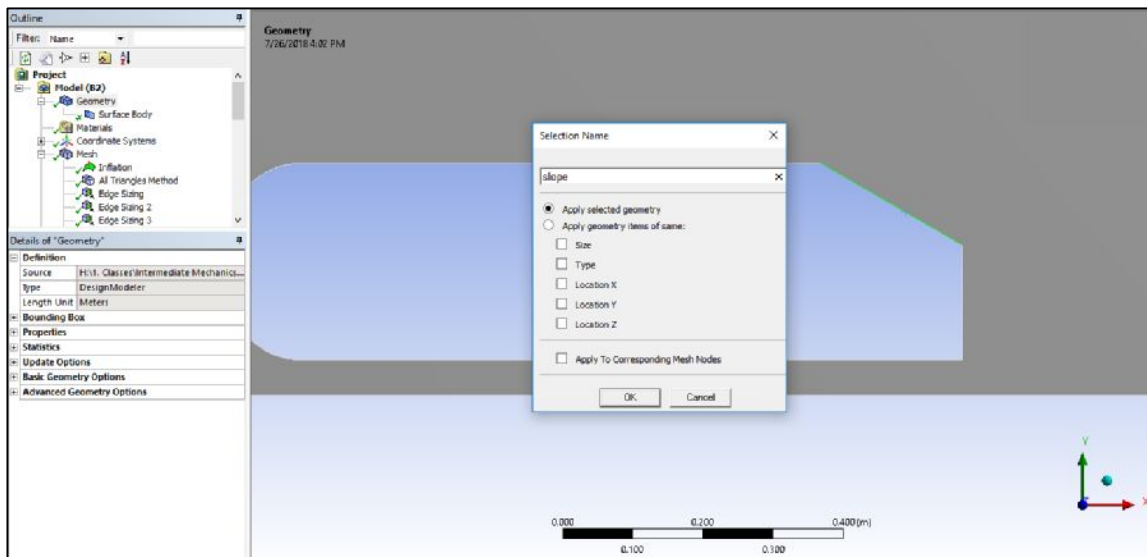
5.11. Repeat step 5.8 for the right edges and name them 'outlet'.



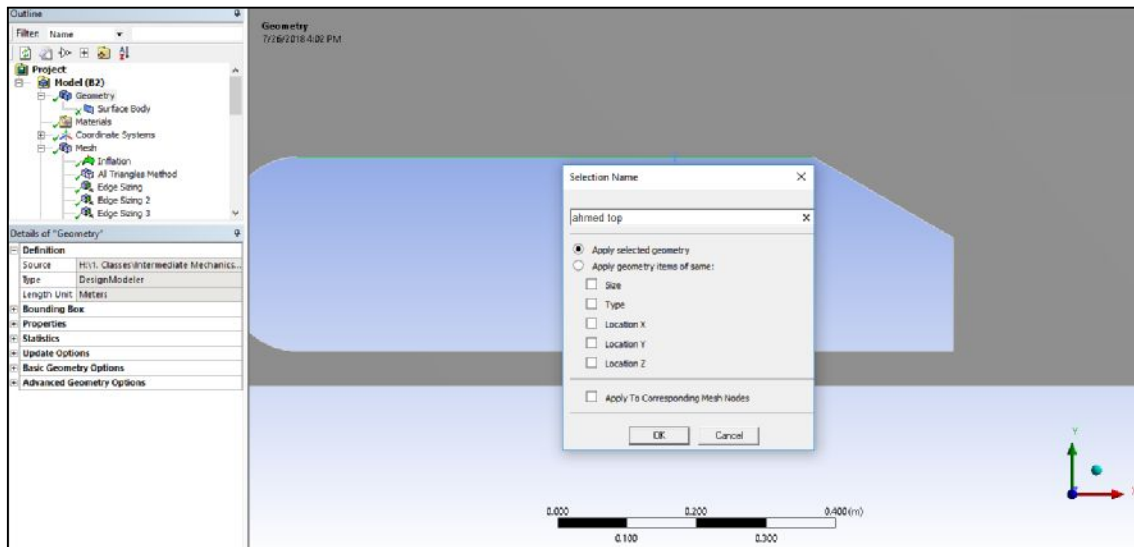
5.12. Repeat step 5.8 for the filleted corners and the straight segment that connects them and name them 'nose'.



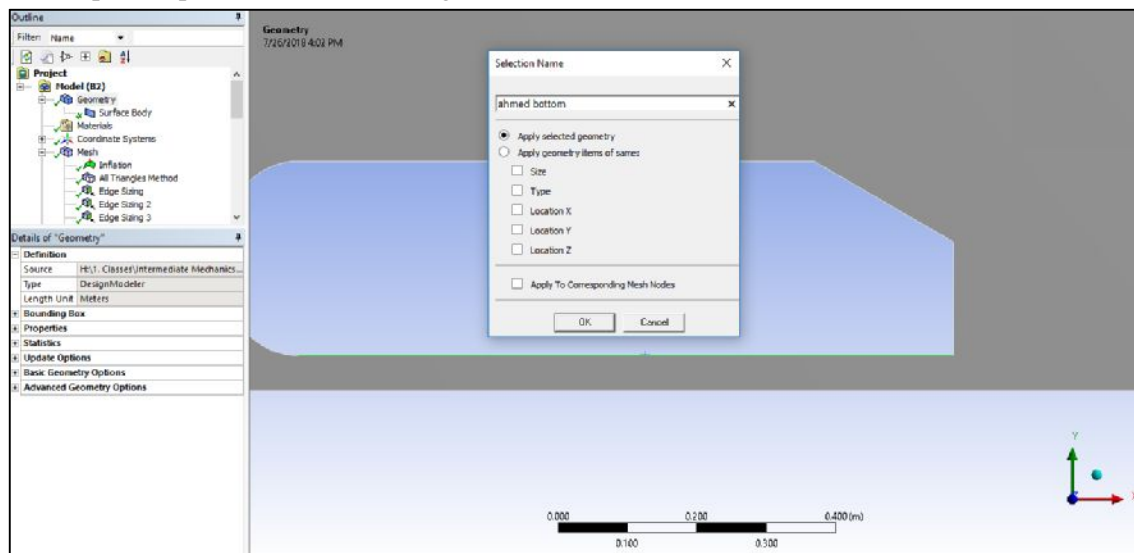
5.13. Repeat step 5.8 for the sloped edge of the Ahmed Car and name it 'slope'.



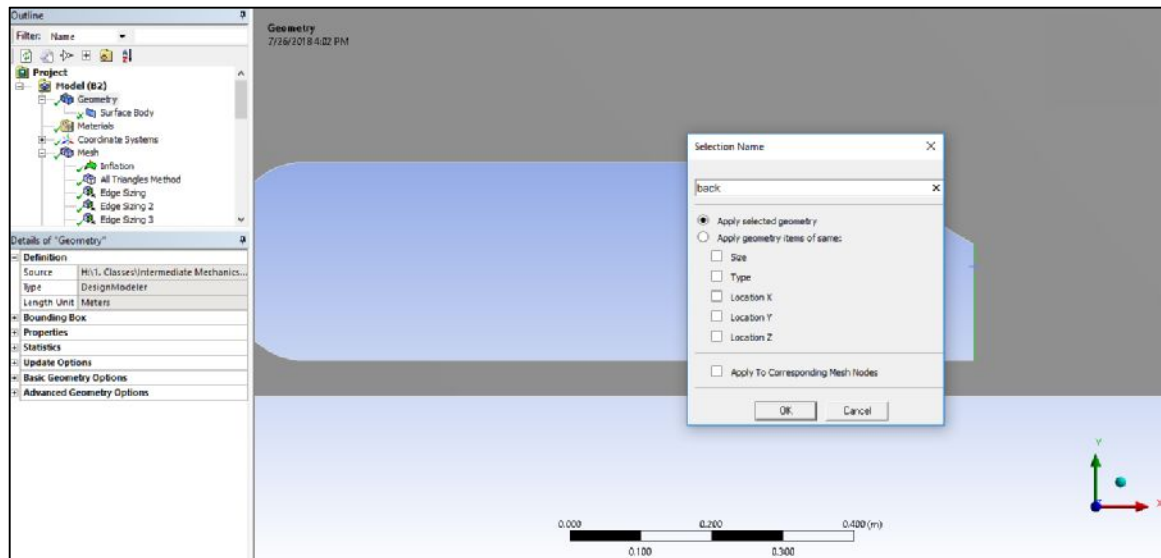
5.14. Repeat step 5.8 for the top edge of the Ahmed Car and name it '*ahmed top*'.



5.15. Repeat step 5.8 for the bottom edge of the Ahmed Car and name it '*ahmed bottom*'.



5.16. Repeat step 5.8 for the right vertical edge of the Ahmed Car and name it *back*.



5.17. **File > Save Project.** Close **Meshing** window.

5.18. Update the mesh by right clicking **Mesh** and from the dropdown menu select **Update**.

