

ANSYS Design Modeler and Meshing Tutorial: A Simple Duct Mesh

R1.00

Arpan R. Singh
Scott J. Ormiston

Department of Mechanical Engineering
University of Manitoba
Canada

2019-09-10

1 Introduction

This tutorial covers the steps necessary to create a simple rectangular duct mesh using ANSYS Workbench. The geometry has lengths of $L_x = 2.0$ [m], $L_y = 0.2$ [m], and $L_z = 0.3$ [m]. The geometry of the domain is created in ANSYS DesignModeler. A uniform orthogonal mesh is then created in the domain using ANSYS Meshing. The completed mesh is saved as file that is used in another tutorial: *ANSYS CFX Tutorial: Laminar Flow in a Rectangular Duct*.

2 Geometry

1. Launch ANSYS Workbench.
2. At the top of the Workbench window, click File → Save As...
Select an appropriate location for this project and set the file name to duct.
Click Save.
3. In the Toolbox pane on the left side of the window, expand Component Systems if it has not already been expanded, then double click Geometry.

The Project Schematic pane will have a new object named Geometry. Rename the label at the bottom of the object to “duct”. The screen should look similar to Figure 1.

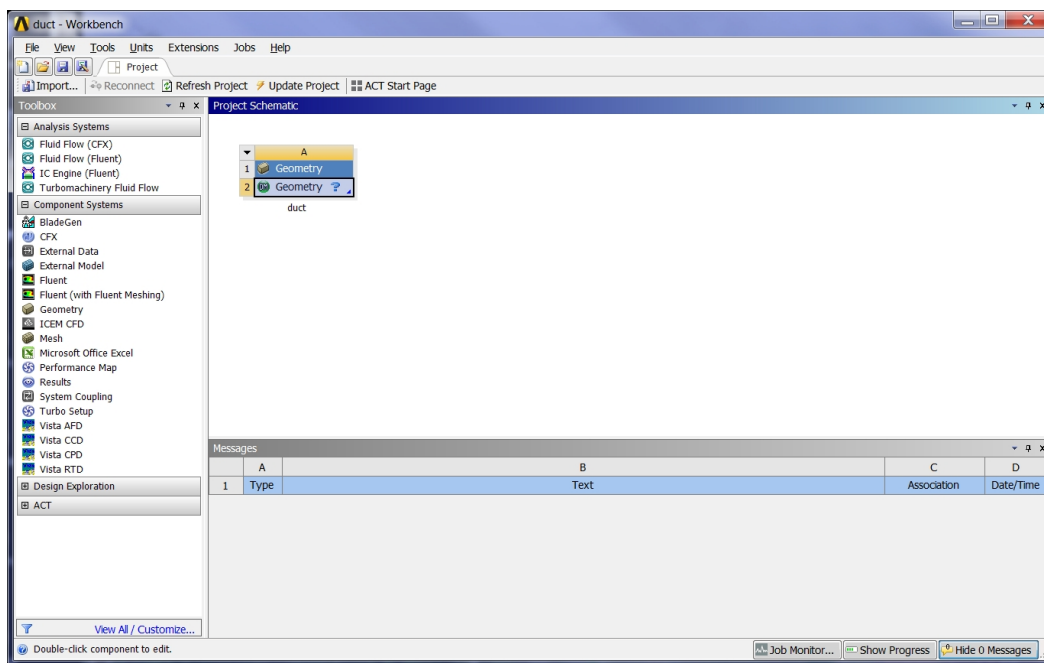


Figure 1: How the Workbench window should appear at the end of Step 3 of Geometry

4. Right-click cell A2 (Labelled Geometry ?) in the newly shown Geometry object.
In the new context menu, click New DesignModeler Geometry... A new DesignModeler window will launch as shown in Figure 2.

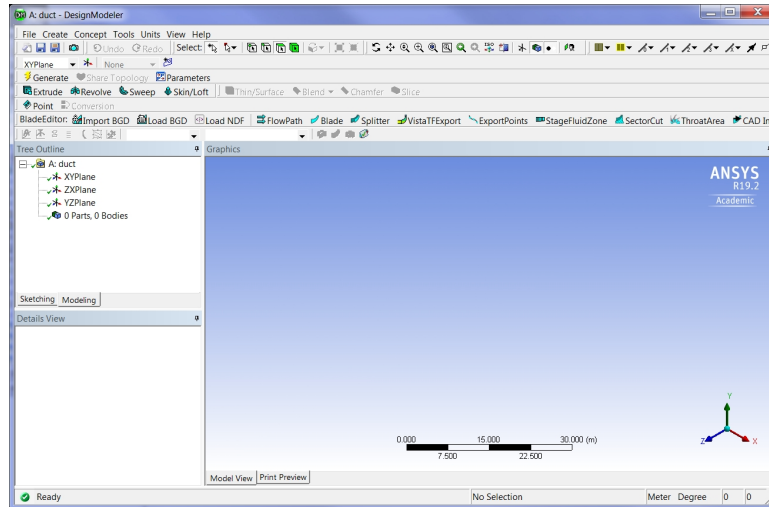


Figure 2: A new DesignModeller window

5. As shown in Figure 3, click XYPlane ▼ → YZPlane.

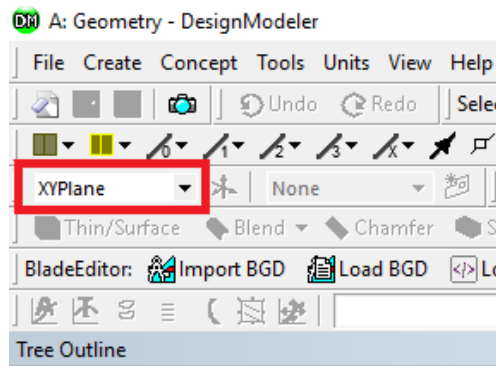


Figure 3: Location of Change Plane box

6. To look normal to the YZ plane, click Look At Face/Plane/Sketch. This button can be found along the top ribbon of buttons and its symbol is shown in Figure 4.

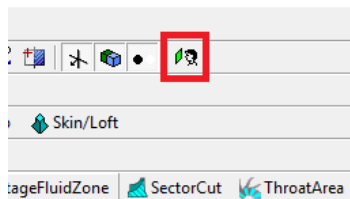


Figure 4: Look At Face/Plane/Sketch

7. In the Tree Outline pane on the left side of the window, click Sketching. The Tree Outline pane is now replaced by the Sketching Toolboxes pane.

8. In the **Sketching Toolboxes** pane, click **Draw** → **Rectangle**.
9. Sketch a rectangle with its bottom left corner coincident with the coordinate system's origin (a **P** should appear as the cursor passes over the origin) and the opposing corner in the positive y and z quadrant. Make the rectangle relatively small. It should look similar to Figure 5.

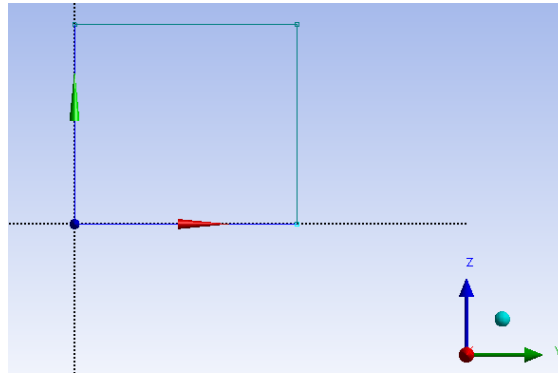


Figure 5: A rectangle with one of its corners on the origin

10. In the **Sketching Toolboxes** pane, click **Dimensions** → **General**.
Click the line along the y axis of the rectangle you just sketched, drag the mouse pointer away from the rectangle, then click again to lock the position of the dimensional annotation. Repeat this for the line along the z axis.

Your screen will contain something similar to Figure 6.

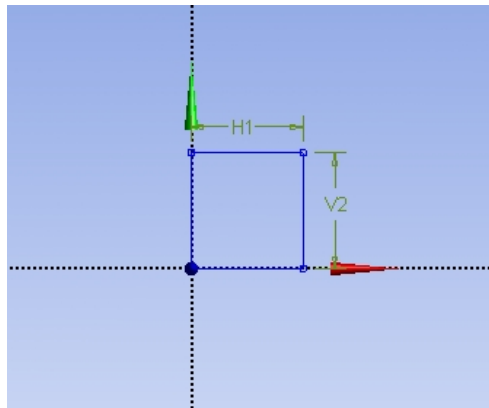


Figure 6: Sketch of rectangle with annotations for dimensions

11. In the **Details View** pane on the left side of the window, set the value of **H1** to 0.2 [m] in the box beside it. Also set the value of **V2** to 0.3 [m]. The result should look like Figure 7.

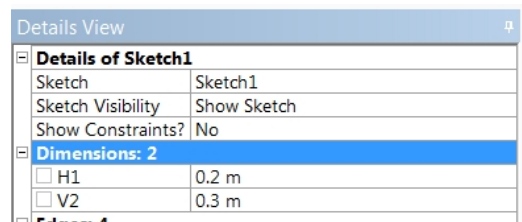


Figure 7: Rectangle dimensions

12. With the 2D profile of the duct profile complete, extrude the feature in the x direction by clicking Extrude (Located as shown in Figure 8) and setting the depth found in the Details View pane to 2.00m.

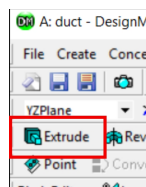


Figure 8: The Extrude button shown inside a red box

13. Click Generate, which is shown in Figure 9.

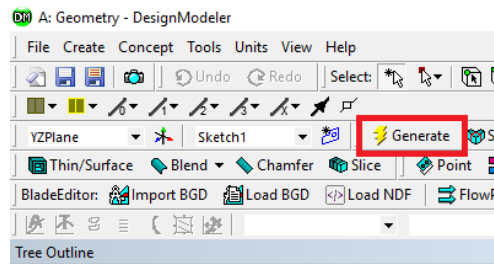


Figure 9: The Generate button shown inside a red box

14. Click on Sketching. Now adjust the location of the dimension labels relative to the body by using Dimensions → Move. After clicking on Move, you can click on the dimension label (H2 or V1) and drag it closer to the rectangle. You can also use the Zoom to fit button (as shown in Figure 10) to fit the rectangle to the screen.

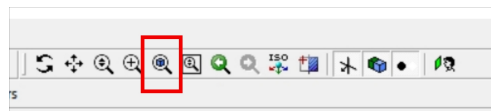


Figure 10: The Zoom to fit button shown inside a red box

After the adjustments to the dimensions and the scaling, your screen should look like Figure 11.

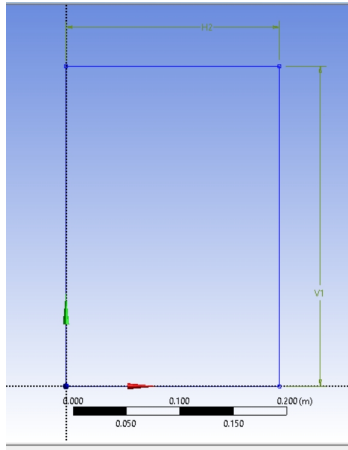


Figure 11: The adjusted view of the rectangle and dimension labels

15. Click on **ISO** in the top ribbon of buttons (seen to the right of **Zoom to fit** in Figure 10). The screen should look like Figure 12.

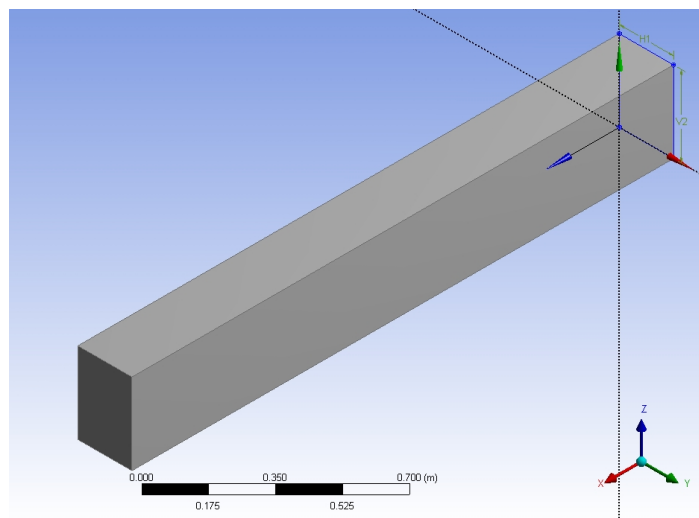


Figure 12: Isometric view of the geometry

16. Now, assign names to all the faces of the domain. This step involves selecting and generating a face, followed by naming the face. To select the face, click on the **Select Face** button in the top ribbon of buttons (the location is shown in Figure 13). If this button is greyed-out, click on **Modeling** in the bottom of the Sketching Toolbox panel.

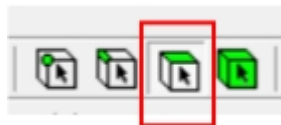


Figure 13: Face button

To name a face:

- (a) Click once on a face.
- (b) Right click and select **Generate**.
- (c) Click on the face again, then right click and select **Named Selection**.
- (d) Type in the face name in the box beside **Named Selection** in the panel on the left side of the screen.

Note that in some versions of **DesignModeler**, you may need to reverse the order of Steps (b) and (c) above.

Follow the procedure above for all six faces of the geometry using the naming convention in Table 1. The naming convention uses **RCT** followed by an underscore and then a letter related to a compass point for x and y directions and bottom and top for the z direction. It is used here for consistency with a pre-existing CFX tutorial.

Table 1: Naming convention for geometry faces

Location	Name
$x = 0$	RCT_W
$x = L_x$	RCT_E
$y = 0$	RCT_S
$y = L_y$	RCT_N
$z = 0$	RCT_B
$z = L_z$	RCT_T

To accomplish this naming, it will be necessary to rotate the geometry to access faces that are not visible in Figure 12.

17. Change the body name. Click on the beside **1 Part, 1 Body** in the tree at the left. Then right click on **solid** and rename it to “duct”.
18. The geometry is now complete. Close **DesignModeler**.

3 Meshing

1. From **Workbench**, double click **Mesh**, which is found under **Component Systems**. Rename this object label from “Mesh” to “duct”.
2. Click and hold cell A2. Drag mouse over cell B2. Release the mouse button. The screen should look like Figure 14.
3. Right-click cell B3.
Click **Edit...** in the new context menu. The **Meshing** application will launch.

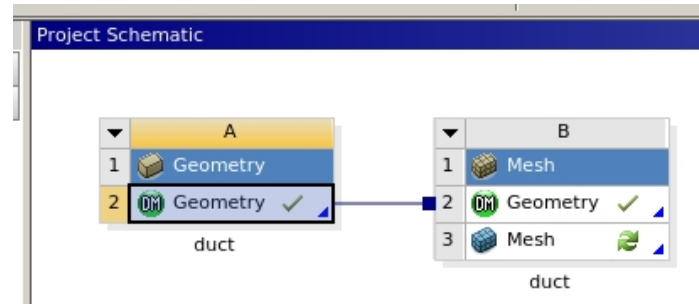


Figure 14: How the Workbench window should appear at the end of Step 2 of Meshing

4. In the Outline pane, click Mesh.

In the Details of Mesh pane, make the following changes:

- (a) Physics Preference: CFD
- (b) Solver Preference: CFX

If you see a next to Element Size, click on it so it becomes a blank square ().

5. Set up sizing rules to control the element size in each direction. The steps to follow for the x direction are:
 - (a) Right click on Mesh in the tree on the left.
 - (b) Select **Insert** → **Sizing**. Make sure that “Scoping Method” is set to “Geometry Selection” in the panel on the left.
 - (c) Click on the box to the right of **Geometry**. It should have buttons that say **Apply** and **Cancel**.
 - (d) Click on the **Edge** selection button in the top ribbon of buttons. This button is to the left of the **Face** button in Figure 13.
 - (e) Click on one of the edges of the domain that is parallel to the x axis.
 - (f) Hold down the **Ctrl** key and click on the other three edges parallel to the first edge.
 - (g) Click **Apply**. It should now say “4 Edges” and highlight the edges on the geometry.
 - (h) Beside **Type** click “Element Size”. This should create a pull-down menu. Select “Number of Divisions”. Enter 40 in the box beside **Number of Divisions** and hit **Enter**.
 - (i) Click in the box to the right of **Behavior**. In the pull-down menu, select **Hard**.
6. Repeat Step 5 but select the four y -direction edges and use 10 divisions.
7. Repeat Step 5 but select the four z -direction edges and use 15 divisions.
8. Click **Generate Mesh** near the top.
9. Click **File** → **Export**. In the dialog box, beside **Save as type:**, select **FLUENT Input Files (*.msh)**. Find a suitable location for the file and name it **duct**. Click **Save**.
10. Close **Meshing**.
11. Exit **Workbench**. Save before closing.