ANSYS Fluent Tutorial Part 3

1 Project 3

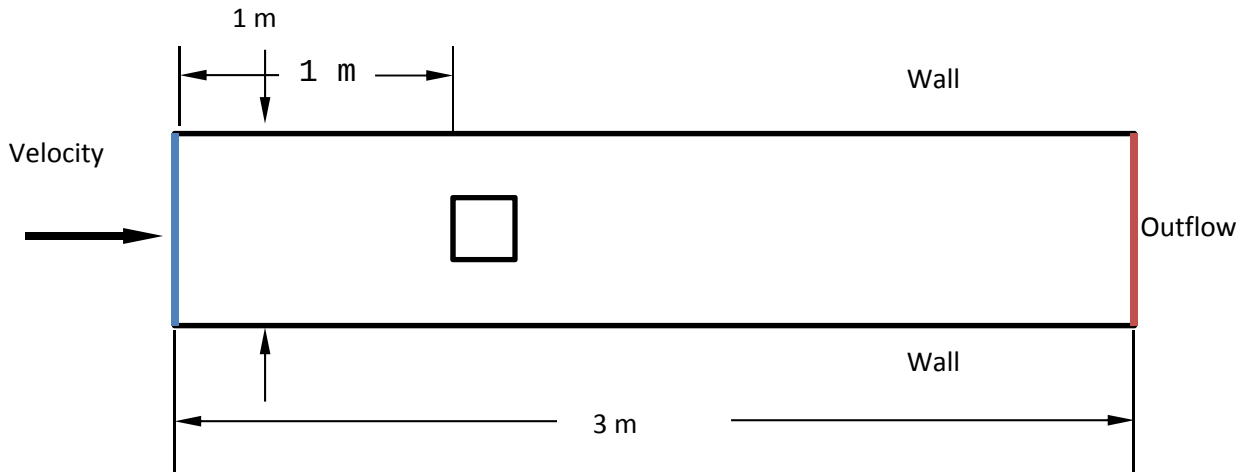


Figure 1

Simulate the flow around a block with a side of 0.3m in a wind tunnel as shown.

Note: Frequently reference Project 3 Details when plotting results and writing the report.

2 Geometry

1. Follow tutorial 1 to open Workbench and start a new project.
2. Once Design Modeler is open, create a sketch and make the External Box geometry (1m x 3m).
3. Create and generate the surface (up to step 2.5 in Tutorial 1).

Here is where things will differ:

4. Select the **XY Plane again** in the **Modeling tab** and create another **New Sketch**.
5. Draw another box and dimension it 0.3 x 0.3
6. Set its position in the channel (1m downstream of the inlet, centered on the height) by selecting the ***Horizontal*** and ***Vertical*** dimension tools, respectively and reference it to the axes.
7. Back in the Modeling tab, **Generate a surface** from the sketch. **You need to change the Operation setting from "*Add Material*" to "*Add Frozen*"** otherwise it will try to merge the surfaces.

8. Now you want to subtract your box sketch (solid) from the channel (fluid). From the **top menu**, select **Create**, **Boolean**.

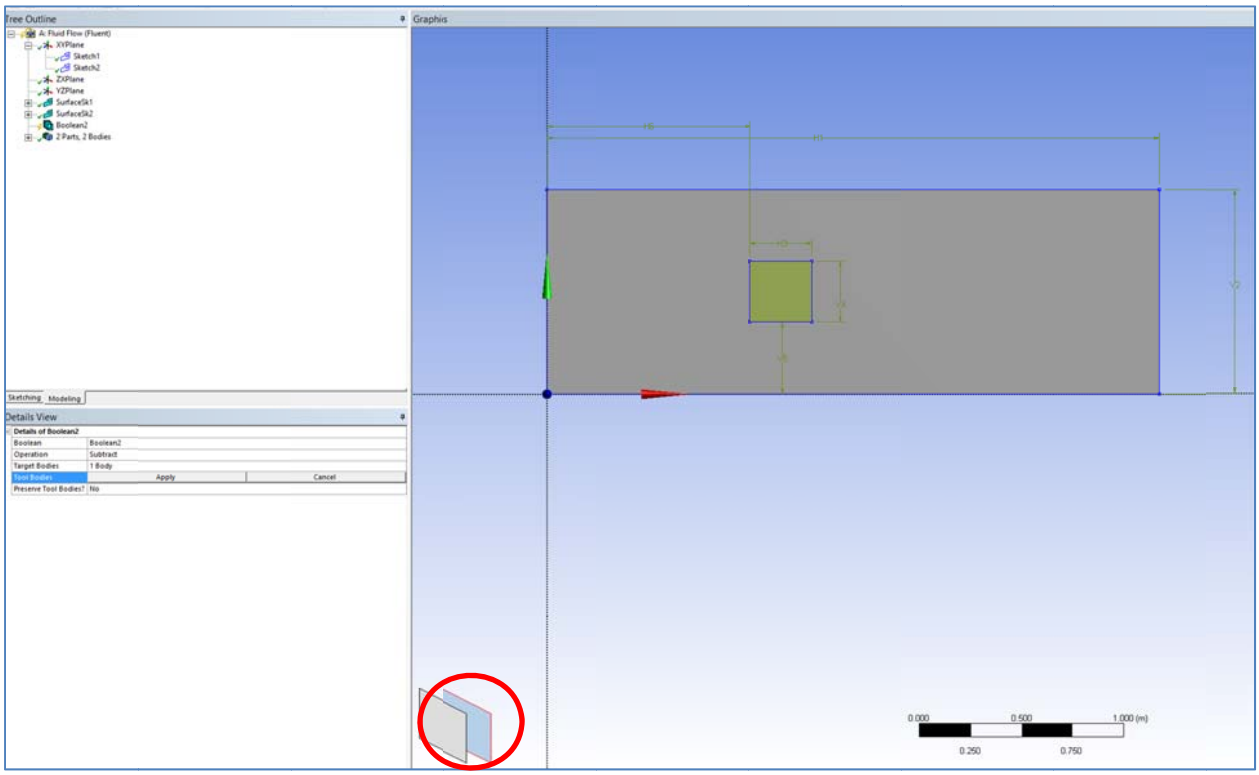
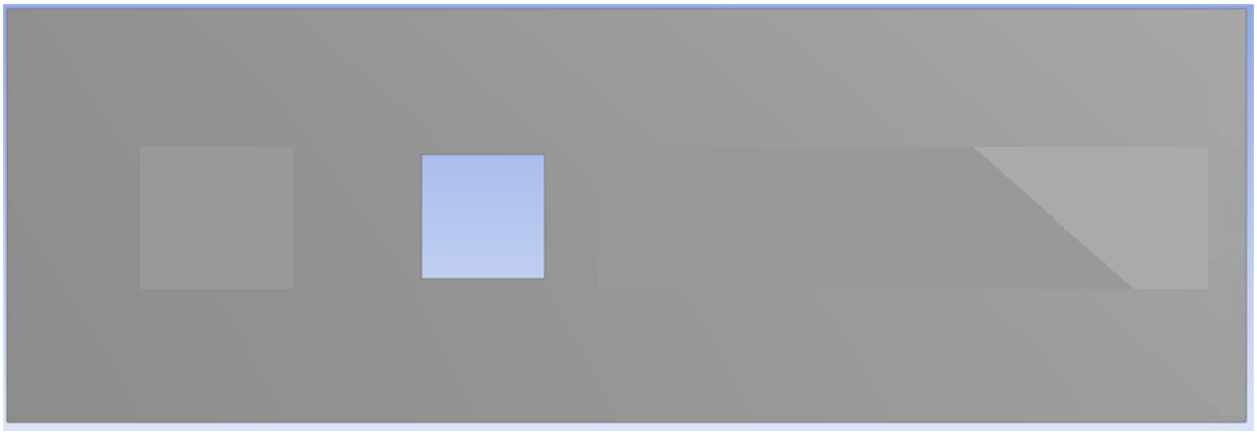


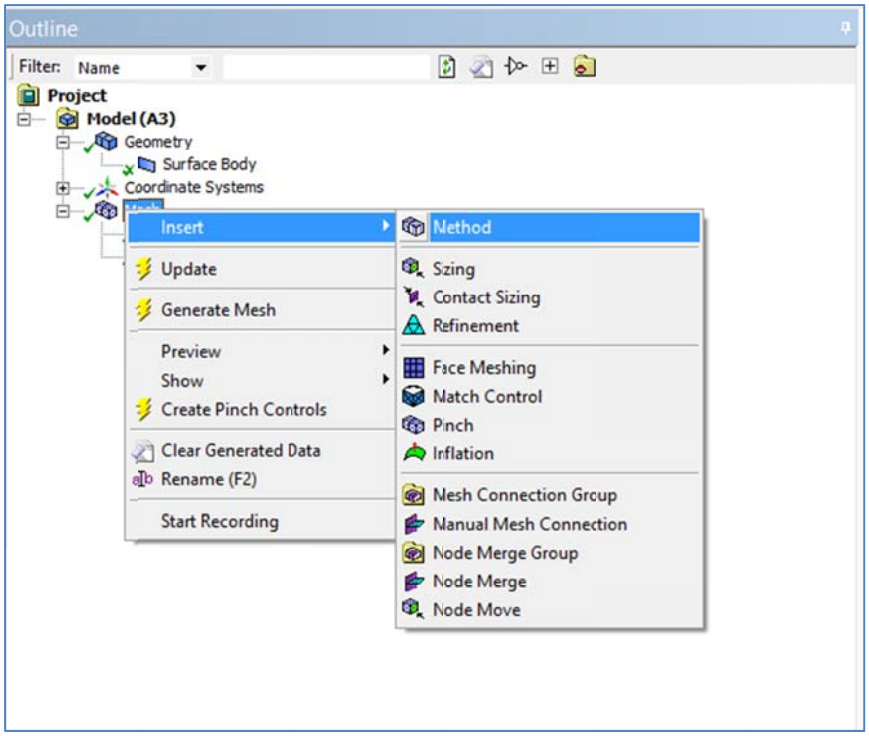
Figure 2

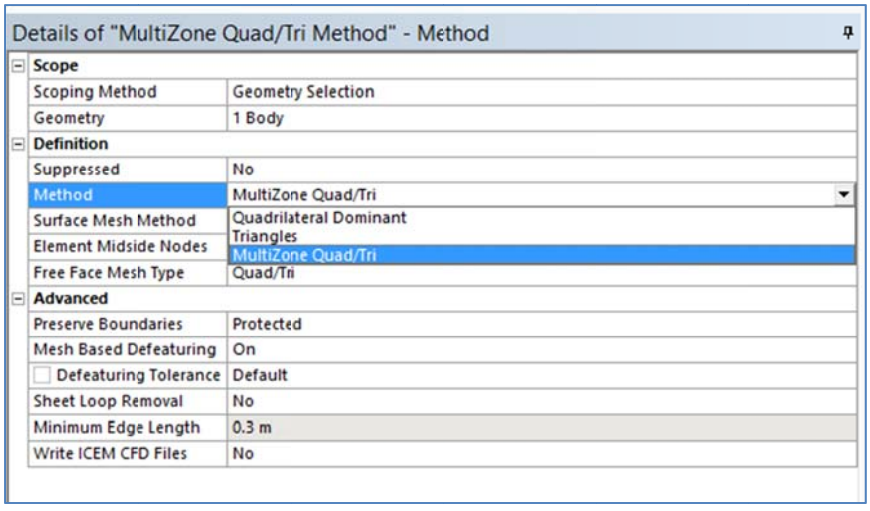
9. In the **Boolean menu**, change the **Operation** to "***Subtract***", select the **Target Body** and click on the **Large domain**, then select **Tool Body** and click on the **small box**. When selecting the small box, it may select the large box because it is the top layer. **<u>You can select the layer behind it by selecting the back plane in the bottom left corner</u>** (circled in Figure 2).
10. Click ***Generate***
11. You should see your channel with a square cutout.



3 Meshing

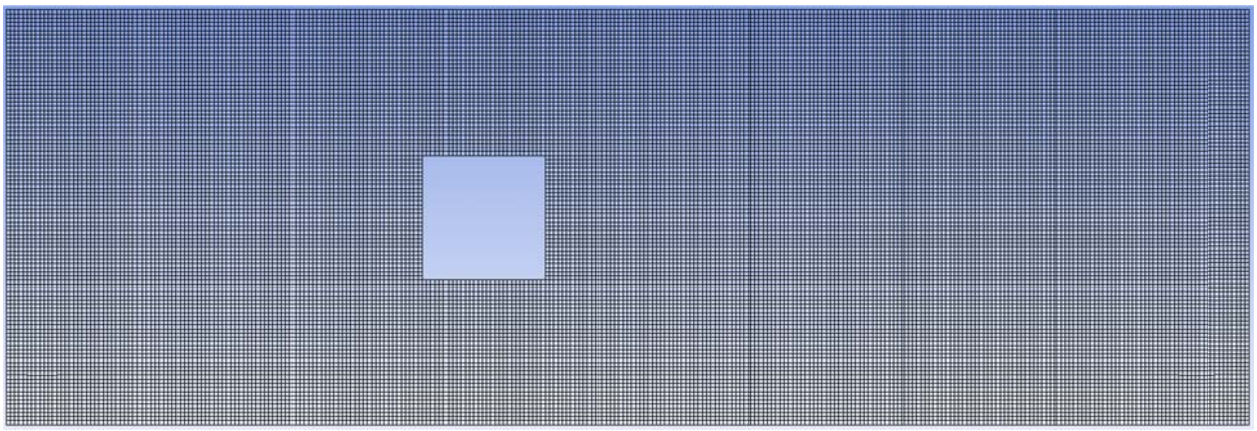
1. **SAVE** and **Open the meshing program**
2. In the **Outline Tree**, expand out **Geometry** and select **Surface Body**. In the **Details**, **make sure *Thickness* is set to *0 m* and the *Material* is set to *Fluid***.
3. Back in the **Outline Tree**, *right click* **Mesh** and **Insert Method**. In the **Details**, select your **Geometry** and **Apply**. Now under **Method**, **change *Quadrilateral Dominant* to *MultiZone***. For the reason, please see the chart listed in the Appendix.





4. Use the **Face Selection Tool** to select your **Geometry**, then *right-click* and **insert Sizing**. Change ***element size*** to **1e-2** and set **Behavior** to **Hard**.

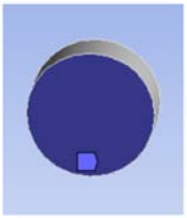
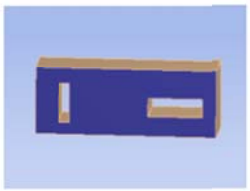
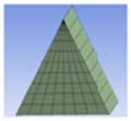
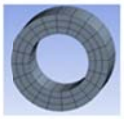
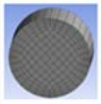
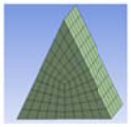
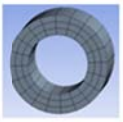
5. Click **Generate Mesh** in the top menu. Click on your Mesh again to view.



6. Using the **edge selection tool**, select an edge of your geometry, right-click, and **insert named section** to **label** your: Inlet, Outflow, Wall_top, Wall_bottom, Wall_boxfront, Wall_boxback, Wall_boxtop, Wall_boxbottom.
7. Back in ANSYS, update your Mesh, SAVE, and proceed to Setup Fluent.
8. Follow Tutorial 1 to run your Fluent simulations and Tutorial 2 to illustrate your results.

4 Appendix

This table is very helpful in determining meshing methods for various geometries:

Mesh Method	Face Characteristic			
	Circular 	**Triangle** 	**Annular** 	**Internal Loops** 
• Sweep • Thin Sweep • Hex Dominant • Patch Conforming Tetrahedron • Quadrilateral Dominant • Triangles	Not supported. You must free mesh these faces.	Not supported. You must free mesh these faces. As side faces, triangle faces can be mapped to obtain a wedge mesh at one corner, depending on source face selection. 	Supported. In the Details View, the **Internal Number of Divisions** option is activated so you can specify the number of divisions across the annular region. The option is set to **3** in the example below: 	Not supported. You must free mesh these faces, but note the following: If there is just one internal loop, it is treated as an annular case. For example, if the model above were split in half, you would have a square annulus which would mesh similar to the circular annulus model.
• MultiZone • MultiZone Quad/Tri	Supported, but mesh quality is poor.  You can use inflation to obtain an O-Grid for better quality in corners. 	Supported. Triangle faces are submapped to tri primitives.  As side faces, triangle faces can be mapped to obtain a wedge mesh at one corner, depending on source face selection. 	Supported. In the Details View, the **Internal Number of Divisions** option is activated so you can specify the number of divisions across the annular region. The option is set to **3** in the example below: 	Ignored for source faces. However, internal loops are supported for side faces. If this example is meshed top to bottom, and has mapped faces defined on the sides, it meshes as follows:  For more information on using mapped Face Meshing with side faces, see *Side Face Handling of Imprinted Regions*.

Source: https://www.sharcnet.ca/Software/Ansys/17.0/en-us/help/wb_msh/msh_Notes_Map_Face.html