

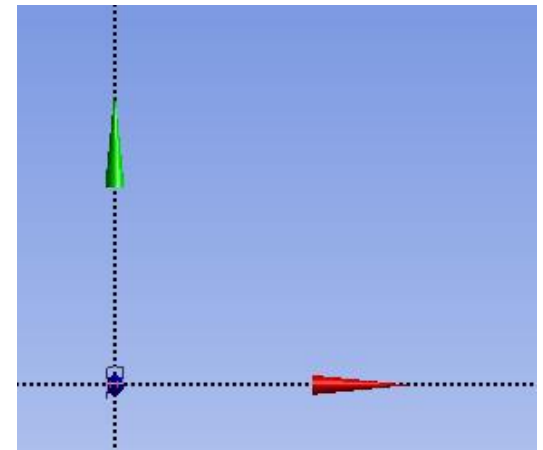
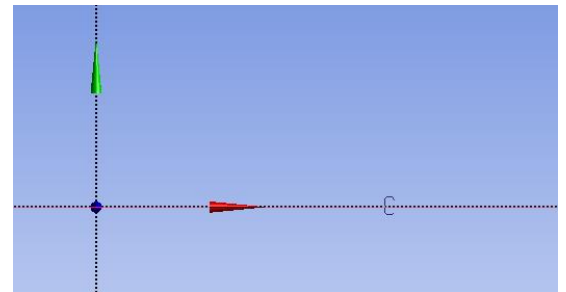
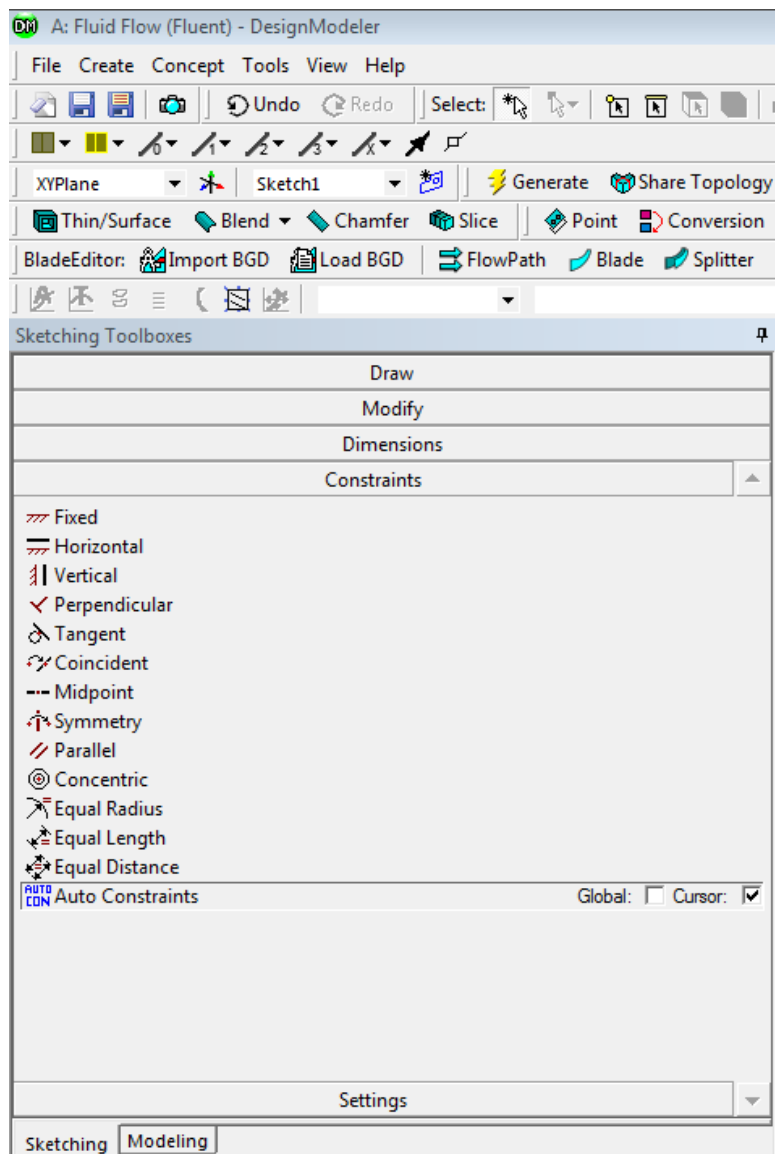
# ME 412 – Numerical Thermo-Fluid Dynamics


## FLUENT Lab Exercise 01 – Meshing

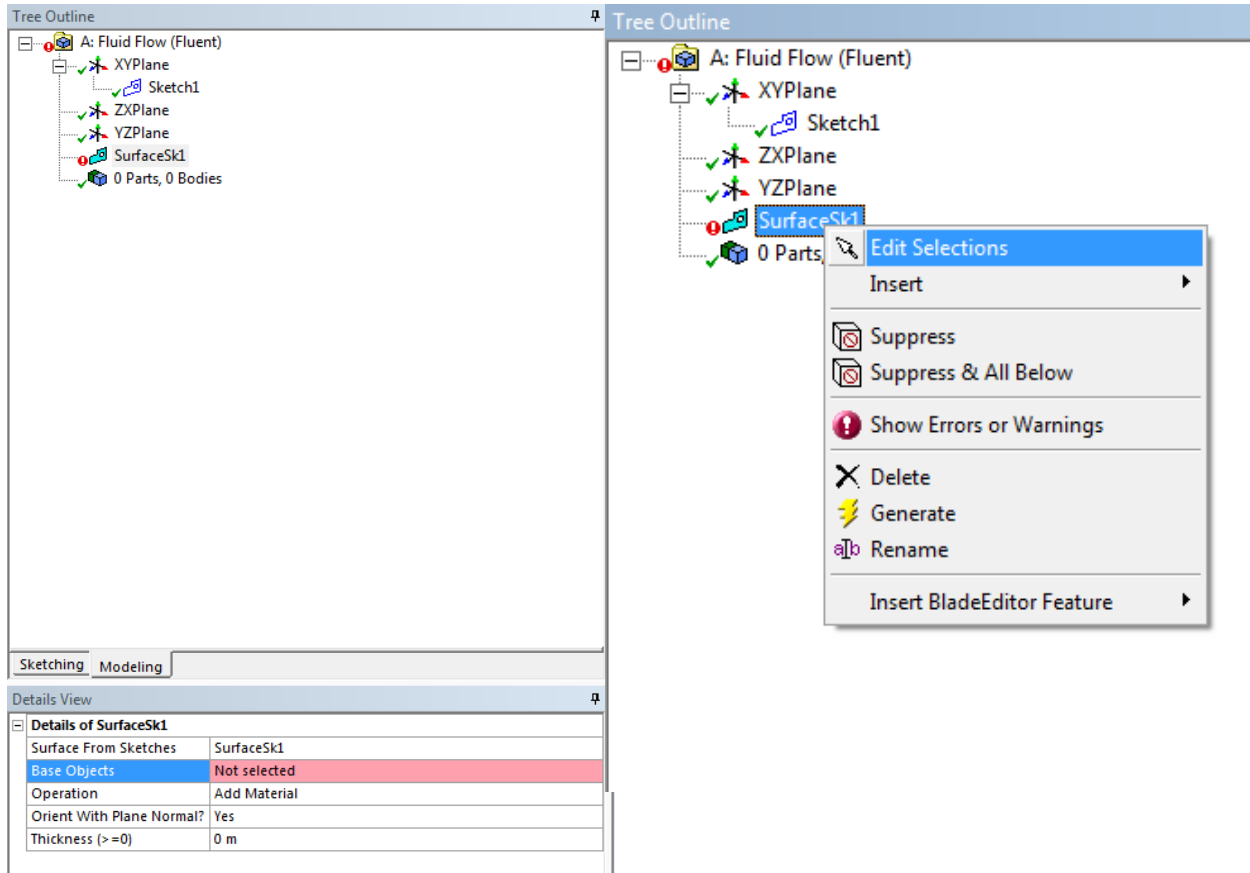
First of all, some tricks for making geometries.


The easiest way to have your rectangle coincide with the axis is to turn on the “Cursor Auto Constraints”. I’m sorry that I missed it. This used to be the default in earlier versions when I learned the software.

Go to “Constraints” tab under the “Sketching Toolboxes”, and check Cursor Auto Constraints box. Now, whenever the cursor is near an axis, a “C” will appear next to it; or whenever the cursor is near a point, say the origin, a “P” will appear next to it.







You need to click  **Generate** after making every Body, be it 2D or 3D. If anything goes wrong when generating a Body, for instance, you forget to select the Base Object, right click the erroneous Body (“SurfaceSk1”), and click “Edit Selection”. It should reset the Body to its original state.

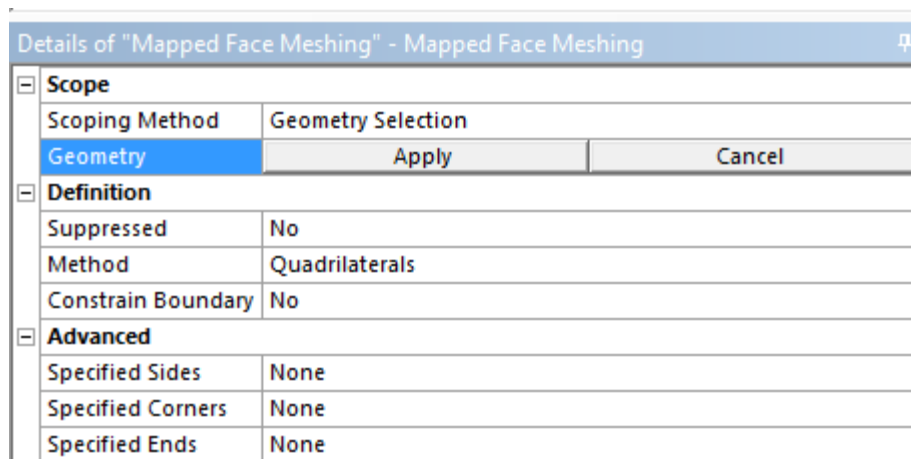
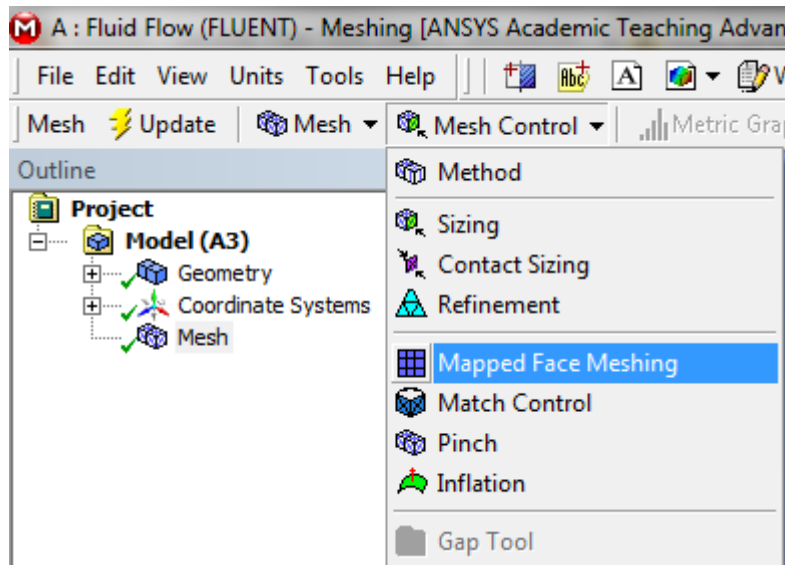


If you need more than one sketches, click “New Sketch”  to draw new ones, lest they are merged to old ones.

## Meshing

There is a number of ways of generating a mesh, depending on the geometry, physical problem, desired accuracy, etc. For this specific example – laminar flow in a rectangular domain – a uniform Cartesian grid is most appropriate. This is done by “Mapped Face Meshing”.

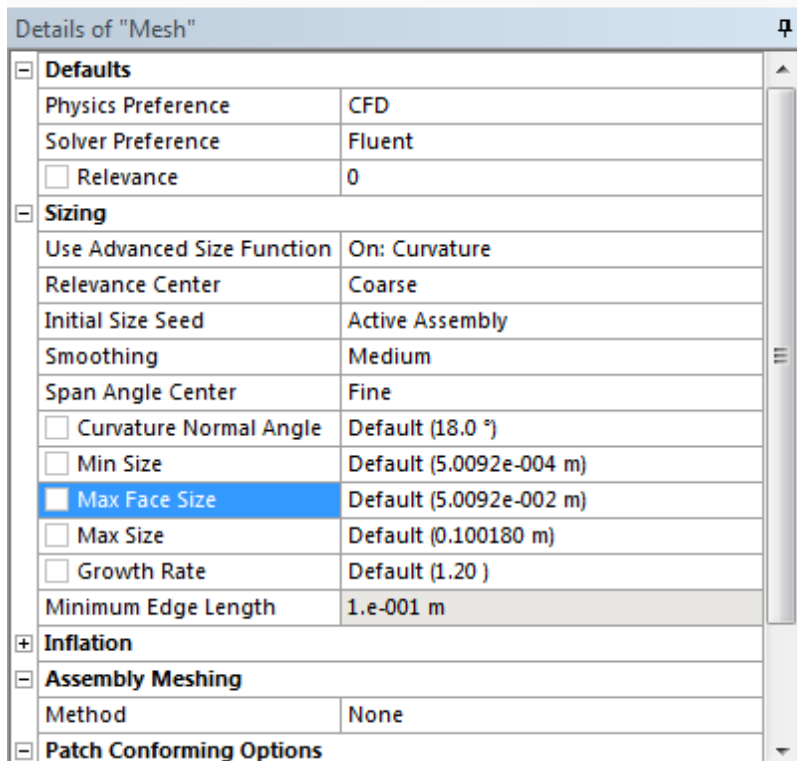
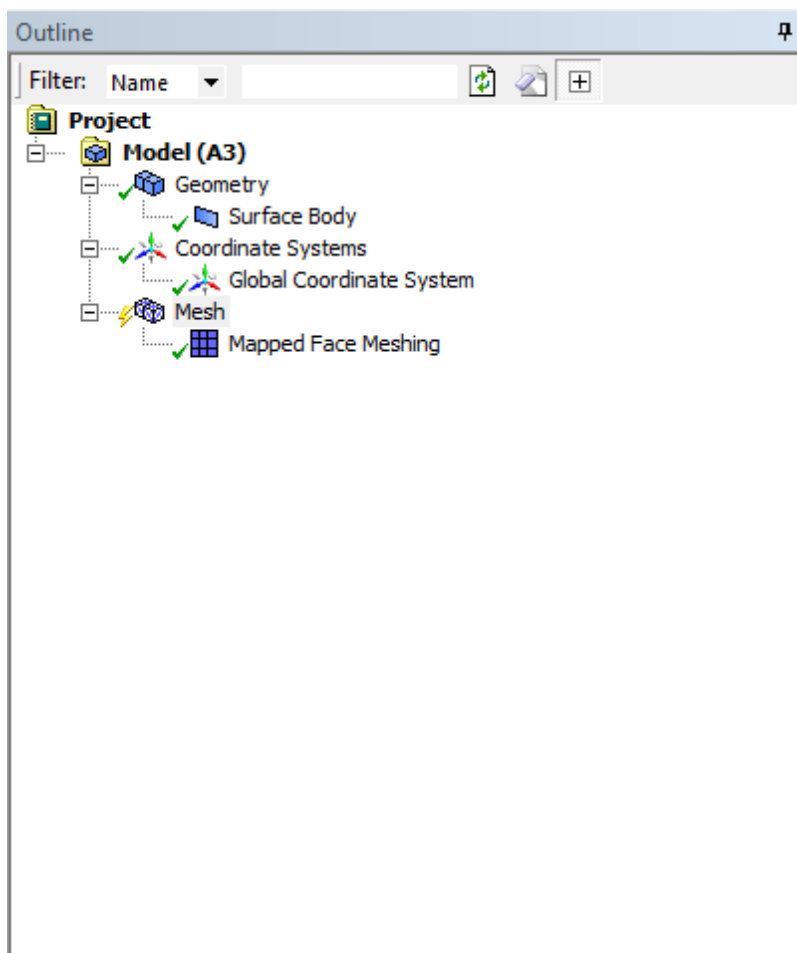
Click “Mesh Control” > “Mapped Face Meshing”. Then apply the Mapped Face Meshing to the rectangle. In order to do so, first select the surface body in the graphics window, which should be highlighted green. Next, click “Apply” in the “Details of ‘Mapped Face Meshing’ table”. You may need to toggle the selection mode among vertex, edge, face, and body    .



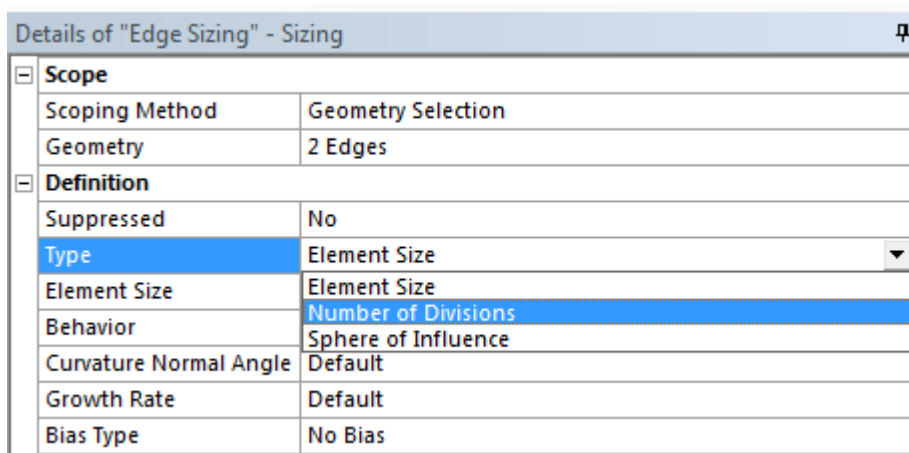
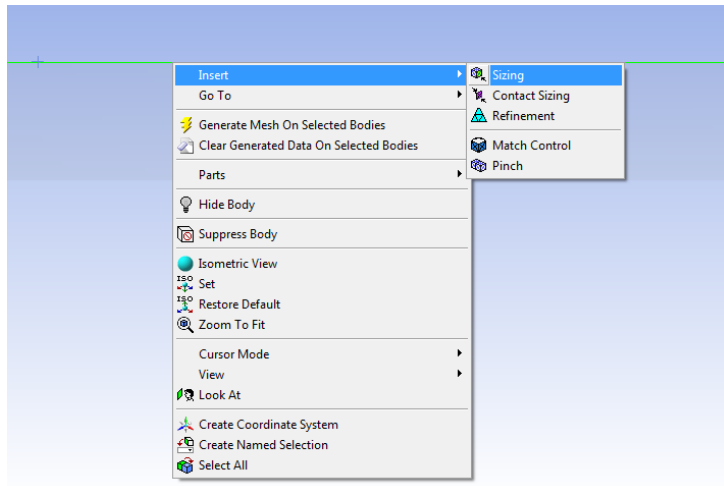
## Sizing

Now that the Mapped Face Meshing has been applied, you need to specify the element sizes. This can be done either globally or locally.

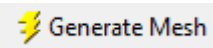
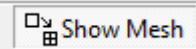
When you select "Mesh" under the "Outline" tree, the details of it will appear in the lower window. Expand the "Sizing" to enter the "Max Face Size". For example, entering 0.1 means the largest surface element will be no larger than 0.1 by 0.1. Since it is a mapped face meshing, all elements will have the same size.



Alternatively, you can assign local sizing to edges (or faces in a 3D geometry). Select the target edge (hold Control key to select multiply edges), and right click on it > “Insert” > “Sizing”. In the “Details of ‘Sizing’” window, set “Type” to “Number of Divisions”. Then set “Number of Divisions” to 100. It means you want 100 divisions along that edge.



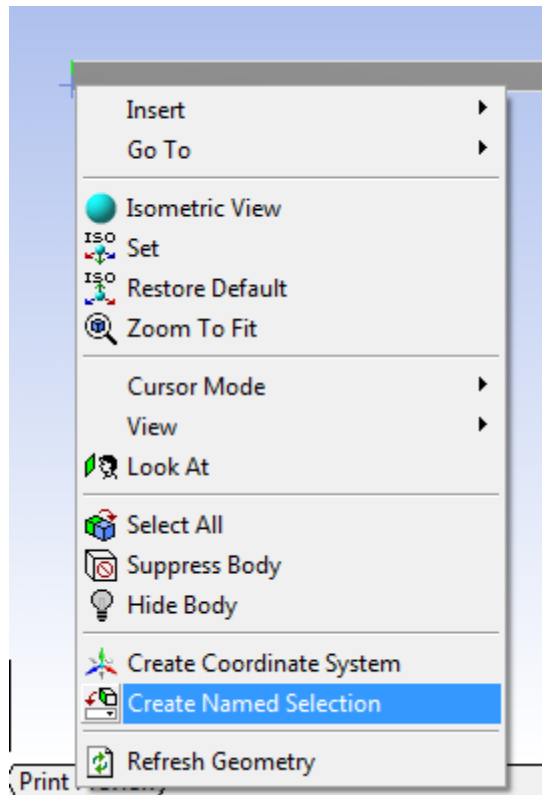
## Generating Mesh

Again, you need to click on “Generate”  to finish the process. After that, you can click on “Show Mesh”  to preview the mesh.

## Creating Named Selection

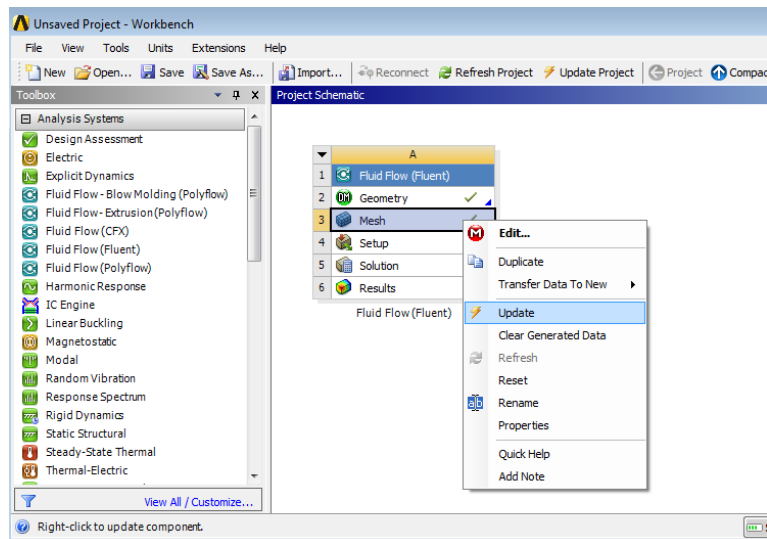
It is convenient to name the boundaries in the meshing step so that they can be recognized by FLUENT.

Right click on the edge you want to name and choose “Create Named Selection”. Enter the name and click OK. Name the four edges as “left”, “right”, “top”, and “bottom”, respectively.



## Updating the mesh

After exiting the mesh module, right click on the “Mesh” cell, and choose update. That should get the mesh updated and FLUENT ready to launch.



Note: if you need to go back to change the named selections after launching FLUENT, you also need to reset FLUENT. Otherwise it will give you errors complaining no recognizing the new named selections.