

Forced Convection - Mesh

Author: Rajesh Bhaskaran & Yong Sheng Khoo, Cornell University

[Problem Specification](#)

[1. Pre-Analysis & Start-Up](#)

[2. Geometry](#)

[3. Mesh](#)

[4. Physics Setup](#)

[5. Numerical Solution](#)

[6. Numerical Results](#)

[7. Verification & Validation](#)

[Exercises](#)

[Comments](#)

Mesh

Save your work in **Workbench** window. In the **Workbench** window, right click on **Mesh**, and click **Edit**. A new **ANSYS Mesher** window will open. We will create a mesh with 200x30 elements along the pipe. This means that we will divide the pipe with 200 elements in axial direction and 30 elements along the radial direction.

In **ANSYS Mesher**, make sure that the unit we are working on is meter Metric unit. On the top menu, click on **Units** and make sure that **Metric (m, kg, N, s, V, A)** is selected.

Since we are going to manually specify meshing type and element size, we should turn off ANSYS build-in advanced sizing function. Under **Details of "Mesh"**, expand **Sizing**, select **Off** next to **Use Advanced Size Function**. [Turn off advanced size demo](#)

Meshing Method

We would also like to create a structured mesh where the opposite edges correspond with each other. Let's insert a Mapped Face mesh. Under **Outline**, right click on **Mesh**, move cursor to **Insert**, and select **Mapped Face Meshing**. Alternatively, you can click on Mesh Control on the third menu and select **Mapped Face Meshing**. Finally select the pipe surface body in the **Graphics** window and click **Apply** next to **Geometry**. [Mapped Face Demo](#)

Edge Sizing

Now let us move on to specify the element sizing along the pipe radial direction.

Outline > Mesh > Insert > Sizing

In the Graphics window, select both the left and right edge of the geometry (click on the **Edge** tab on the Fluid flow Fluent - Mesh window and then press Ctrl + mouse click to multiple select). Under **Details of "Edge Sizing"**, click **Apply** next to **Geometry**. Change the edge sizing definition **Type** to **Number of Divisions**. Enter 30 for **Number of Divisions**. Next to **Behaviour**, change **Soft** to **Hard** (This is to overwrite the sizing function employed by ANSYS Mesher. Try meshing with soft behavior and see what you get). [Radial Sizing](#)

Now continue with the sizing in the axial direction.

Outline > Mesh > Insert > Sizing

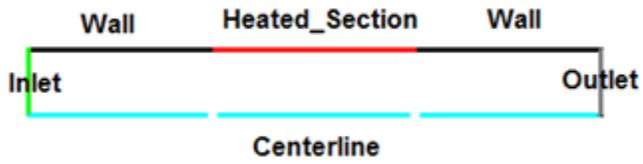
In the Graphics window, select all the top and bottom edge of the geometry (press Ctrl + mouse click to multiple select). Under **Details of "Edge Sizing"**, click **Apply** next to **Geometry**. Enter 0.03 for **Element Size** (this will give us roughly 200 divisions). Next to **Behaviour**, change **Soft** to **Hard** (This is to overwrite the sizing function employed by ANSYS Mesher. Try meshing with soft behavior and see what you get). [Axial Sizing Demo](#)

We have specified all the meshing conditions. Click Update on the third menu to see the mesh.

Click on **Mesh** and look under **Details of "Mesh"**, next to **Statistics**, you should see that we have approximately 6000 **Elements** for our mesh (the exact number might vary slightly depending on the version you are using).

Create Named Selection

Next, we will name the edges accordingly so that we can specify the appropriate boundary conditions in the later step. We know the bottom edges of the geometry are the centerline of the pipe, the left edge is the inlet of the pipe, the right edge is the outlet of the pipe, top side edges are the unheated wall sections and the top middle edge is the heated wall section. Let's name the edges according to the diagram below. **Remember** to click on the **Edge** icon and then press Ctrl + mouse click to multiple select the 3 line sections that make up the center line before naming it.



Select the left edge and right click and select **Create Named Selection**. Enter Inlet and click **OK**. Under **Outline**, you will see the name **Inlet** under **Named Selections**. [Named Selection Demo](#) Finish naming rest of the edges. Finally, click **Update**  **Update**.

[Go to Step 4: Physics Setup](#)

[Go to all FLUENT Learning Modules](#)