

# Turbulent Pipe Flow - Mesh

Author: Rajesh Bhaskaran, 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

You should have completed the [Laminar Pipe Flow](#) tutorial before continuing with this one. The starting point for this tutorial is the ending point of the one before it. If you bring up the project we have already completed, you can follow the next steps.

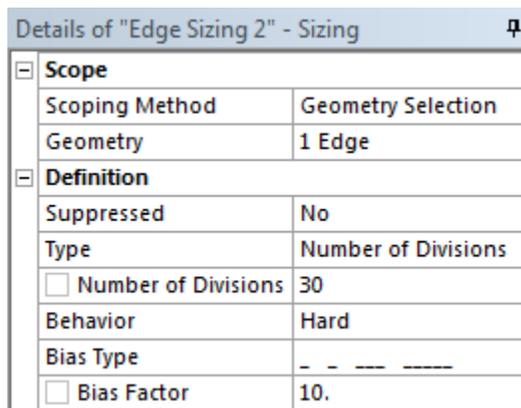
Right click on **Mesh**. Then click on **Duplicate**, which will duplicate the mesh from the previous tutorial. Enter "Turbulent Flow" in the highlighted field to rename it. At this point your project schematic window should appear as below:



Next, double click on the **Mesh** cell so we can edit the mesh.

We need to change the edge sizing, as we did in the previous tutorial, to 100 by 30 (instead of 100 by 5). We are also going to need to bias it. This is because we want smaller divisions the closer you get to the wall. First, right click on **Edge Sizing 2** in the Project tree on the left, and click **Delete** to remove the existing edge sizing on the inlet and outlet.

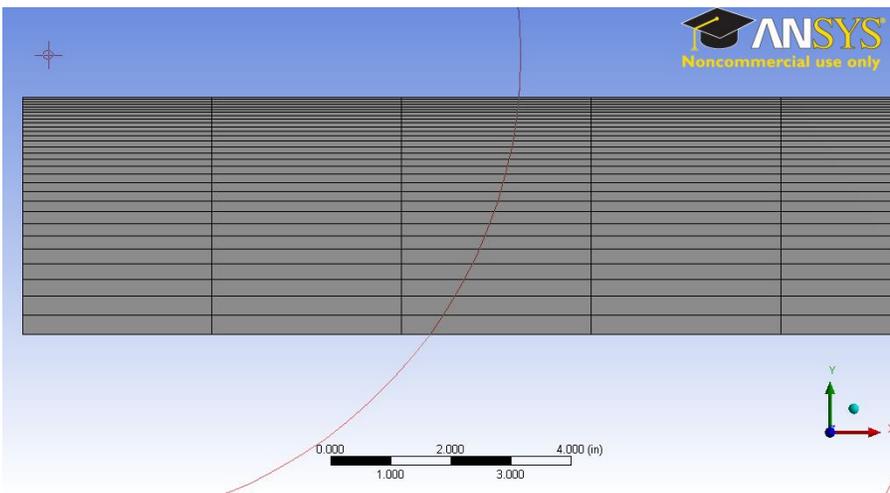
Next, we'll apply an edge sizing with bias to the inlet, the left end of the pipe. Click **Mesh Control > Sizing**. Using the edge selection tool, highlight the inlet (left end) of the pipe and click **Apply** next to **Geometry**. As in the Laminar Pipe Flow tutorial, change **Type** to **Number of Divisions**, and enter 30. Change **Behavior** to **Hard**. Now, let's apply a bias to the edge sizing. Under **Bias Type**, select the second option, - - - - -. Enter a **Bias Factor** of 10. Your Details of "Edge Sizing 2" should now appear like the image below.



Now we would like to apply an edge sizing to the outlet, the right end of the pipe. Once again, we'll use 30 divisions, with a bias factor of 10 and with the smaller divisions at the top, near the wall. This time, when selecting **Bias Type**, choose the first option, - - - - -. This will put the smallest divisions at the top. Other than this, the procedure is the same as for the inlet. When complete, your Details of "Edge Sizing 3" should look like this:

Details of "Edge Sizing 3" - Sizing	
<b>Scope</b>	
Scoping Method	Geometry Selection
Geometry	1 Edge
<b>Definition</b>	
Suppressed	No
Type	Number of Divisions
<input type="checkbox"/> Number of Divisions	30
Behavior	Hard
Bias Type	-----
<input type="checkbox"/> Bias Factor	10.

Right click on Mesh and select "Generate Mesh". The bias factor generates a finer mesh near the pipe wall. This is done to compute the small fluctuation in fluid property near the wall.



Next, close the meshing window to return to the main project view.

Recall that we created the following boundary types for the 100x5 mesh in the Laminar Pipe Flow tutorial:

Edge Position	Name	Type
Left	inlet	VELOCITY_INLET
Right	outlet	PRESSURE_OUTLET
Top	wall	WALL
Bottom	centerline	AXIS

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

[Go to all FLUENT Learning Modules](#)