FLUENT - Compressible Flow in a Nozzle- Step 3

Author: Rajesh Bhaskaran & Yong Sheng Khoo, Cornell University

Problem Specification
1. Pre-Analysis & Start-up
2. Geometry
3. Mesh
4. Setup (Physics)
5. Solution
6. Results
7. Verification & Validation
Problem 1
Problem 2

Step 3: Mesh

Specify Boundary Types

Now that we have the mesh, we would like to specify the boundary conditions here in *GAMBIT*.

Operation Toolpad > Zones Command Button **> Specify Boundary Types Command Button**

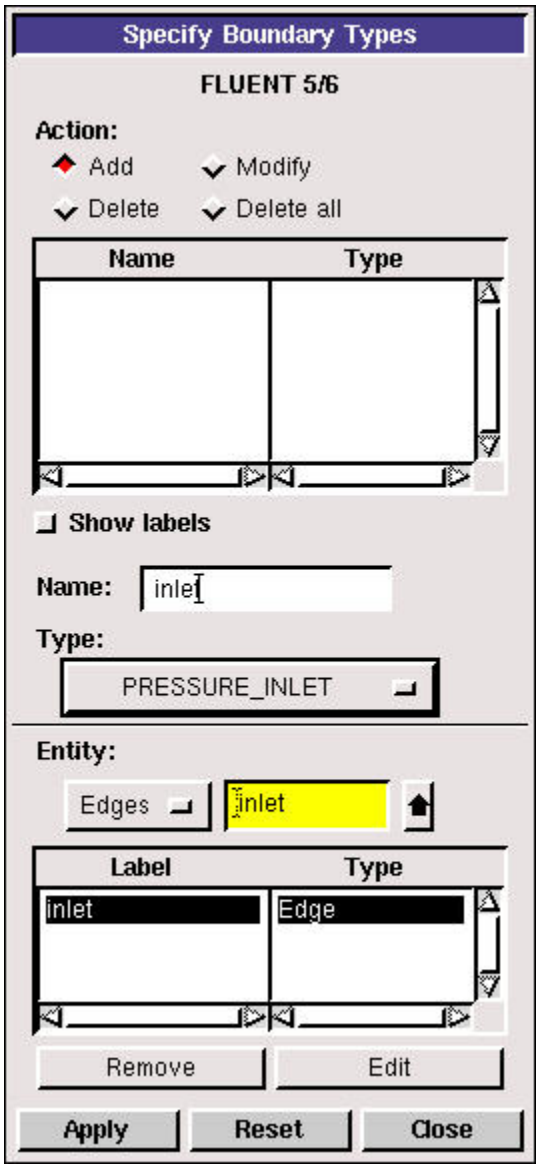
This will bring up the *Specify Boundary Types* window on the *Operation Panel.* We will first specify that the left edge is the inlet. Under ***Entity***:, pick ***Edges*** so that *GAMBIT* knows we want to pick an edge (face is default).

Now select the left edge by ***Shift-clicking*** on it. The selected edge should appear in the yellow box next to the ***Edges*** box you just worked with as well as the ***Label/Type*** list right under the Edges box.

Next to ***Name***:, enter `inlet`.

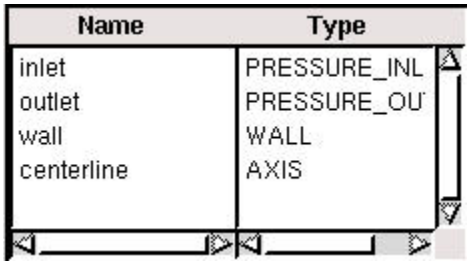
For ***Type:***, select ***WALL***.

Click ***Apply***. You should see the new entry appear under ***Name/Type*** box near the top of the window.



Repeat for the outlet, centerline, and wall edges.

You should have the following edges in the ***Name/Type*** list when finished:



Save and Export

Main Menu > File > Save

Main Menu > File > Export > Mesh...

Type in `nozzle.msh` for the ***File Name:***. Select ***Export 2d Mesh*** since this is a 2 dimensional mesh. Click ***Accept***.

Check `nozzle.msh` has been created in your working directory.

Go to Step 4: Setup (Physics)

See and rate the complete Learning Module

Go to all FLUENT Learning Modules