

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](#)

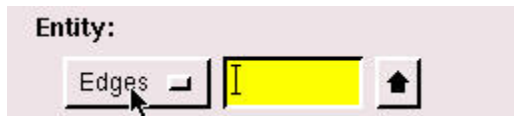
Step 3: Mesh

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.

Specify Boundary Types

FLUENT 5/6

Action:

◆ Add
▼ Modify

▼ Delete
▼ Delete all

Name	Type

☐ Show labels

Name:

Type: PRESSURE_INLET

Entity:

Edges
inlet
↑

Label	Type
inlet	Edge

Remove
Edit

Apply
Reset
Close

Repeat for the outlet, centerline, and wall edges.

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

Name	Type
inlet	PRESSURE_INL
outlet	PRESSURE_OU
wall	WALL
centerline	AXIS

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](#)