FLUENT - Forced Convection over a Flat Plate step 3

Step 3: Specify Boundary Types in GAMBIT
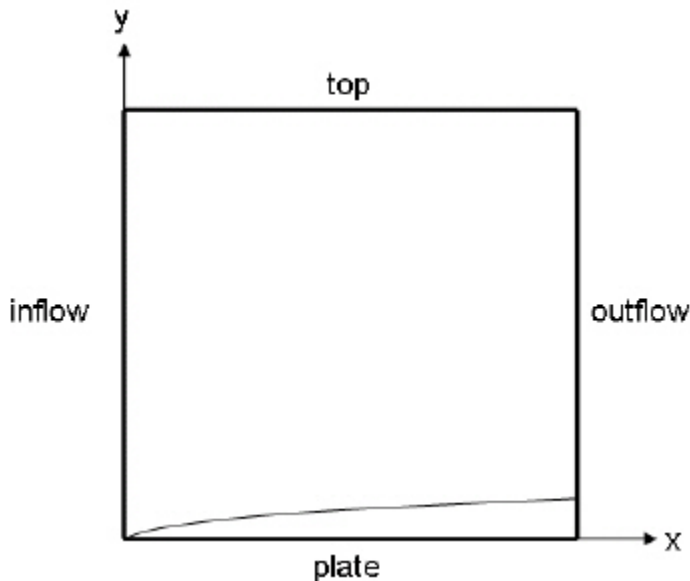
Create Boundary Types

We'll next set the boundary types in *GAMBIT*. The left edge is the inflow of the flow field, the right edge the outflow, the top edge the open top of the flow field, and the bottom edge the plate.

**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 inflow. Under *Entity:*, pick **Edges** so that *GAMBIT* knows we want to pick an edge (face is default).
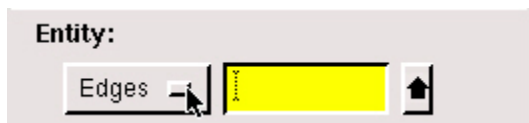
Now select the left edge by **Shift-clicking** on it. The selected edge should appear in the yellow box next to the **Edges** box as well as the **Label/Type** list under the **Edges** box.

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

For *Type:*, select **VELOCITY_INLET**. You may have to move the *Specify Boundary Types* box up in order to see the bottom of the list and select **VELOCITY_INLET**.
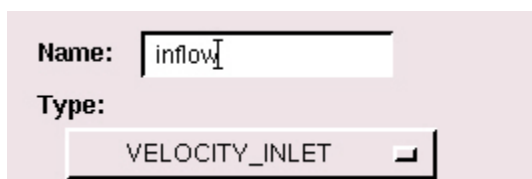
Click **Apply**. You should see the new entry appear under **Name/Type** box near the top of the window.

Repeat this process for the other three edges according to the following table:

Edge Position	Name	Type
Left	inflow	VELOCITY_INLET
Right	outflow	PRESSURE_OUTLET
Top	top	SYMMETRY
Bottom	plate	WALL

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 `plate.msh` for the *File Name*:. Select *Export 2d Mesh* because this is a 2 dimensional mesh. Click *Accept*.

It is important to check that `plate.msh` has been created in your working directory. *GAMBIT* may periodically fail to write the `.msh` file. If this should happen, simply try writing the `.msh` file to another directory and then coping it into your working directory.

Go to

See and rate the complete Learning Module

Go to all FLUENT Learning Modules