

# FLUENT - Forced Convection over a Flat Plate step 2

## Problem Specification

1. Create Geometry in GAMBIT
2. **Mesh Geometry in GAMBIT**
3. Specify Boundary Types in GAMBIT
4. Set Up Problem in FLUENT
5. Solve
6. Analyze Results
7. Refine Mesh

## Step 2: Mesh Geometry in GAMBIT

We'll now create a mesh on the rectangular face with 100 divisions in the vertical direction and 30 divisions in the horizontal direction. We'll first mesh the four edges and then the face. The desired grid spacing is specified through the edge mesh.

### Mesh Edges

Operation Toolpad > Mesh Command Button  > Edge Command Button  > Mesh Edges 

### Mesh Strategy

In creating this mesh, it is desirable to have more cells near the plate (Edge 1) because we want to resolve the turbulent boundary layer, which is very thin compared to the height of the flow field.

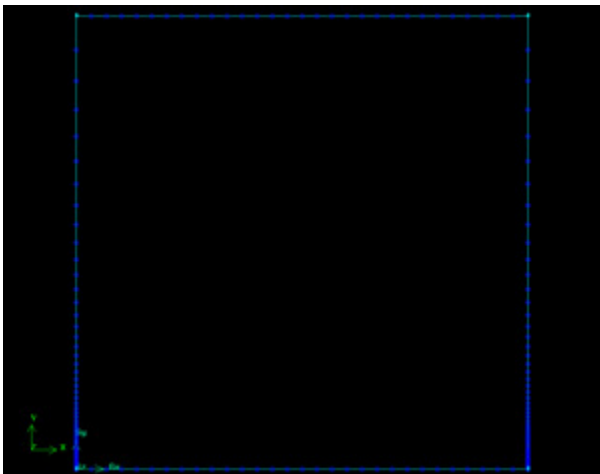
Click the **up arrow button**  next to the *Edges* box in the *Mesh Edges* window. Select edge **Edge.2**. Then push the **right arrow button**  to bring this vertex into the *Picked* column. Notice that the arrow on the selected edge should be pointing upwards. An upwards pointing arrow indicates the direction of closely spaced nodes to widely spaced nodes. Remember, we will need more closely spaced nodes near the boundary layer in order to resolve it accurately.



The proper arrow direction is necessary to ensure a proper mesh. Select **Edge.4** in the *Mesh Edges* window. The arrow on this edge is pointing downwards, which needs to be changed. **Shift + Middle-click** on the selected edge to change the direction of the arrow to upward.

Under *Type*, select **Successive Ratio**, if it is not already selected. Set *Ratio* to 1.08. Under *Spacing*, select **Interval Count**. Set Interval Count to 100 and then click **Apply**.

Select **Edge.1** and **Edge.3** in the *Mesh Edges* Window. The direction of the arrows on these edges is irrelevant because the divisions will be the same length. Leave the **Successive Ratio** set to 1 and set the **Interval Count** to 30. Click **Apply**.

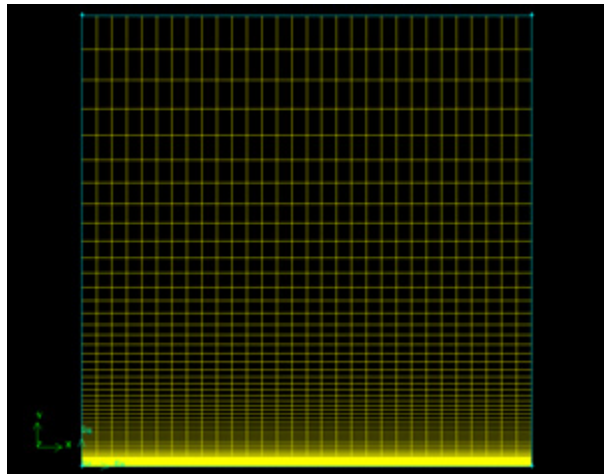


[Higher Resolution Image](#)

### Mesh Face

Operation Toolpad > Mesh Command Button  > Face Command Button  > Mesh Faces 

**Shift left-click** on the face or use the up arrow next to *Faces* to select the face. Click **Apply**.



[Higher Resolution Image](#)

[Go to Step 3: Specify Boundary Types in GAMBIT](#)

[See and rate the complete Learning Module](#)

[Go to all FLUENT Learning Modules](#)