

FLUENT - Compressible Flow in a Nozzle- Step 2

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 2: Geometry

Now that we have the basic geometry of the nozzle created, we need to mesh it. We would like to create a 50x20 grid for this geometry.

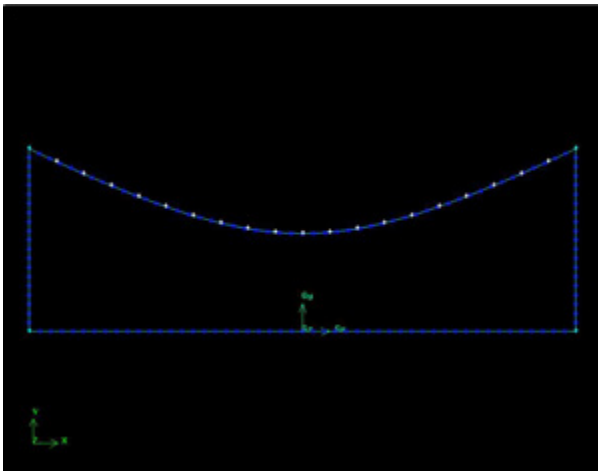
Mesh Edges

As in the previous tutorials, we will first start by meshing the edges.

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

Like the [Laminar Pipe Flow Tutorial](#), we are going to use even spacing between each of the mesh points. We won't be using the Grading this time, so deselect the box next to *Grading* that says *Apply*.


Then, change *Interval Count* to 20 for the side edges and *Interval Count* to 50 for the top and bottom edges.



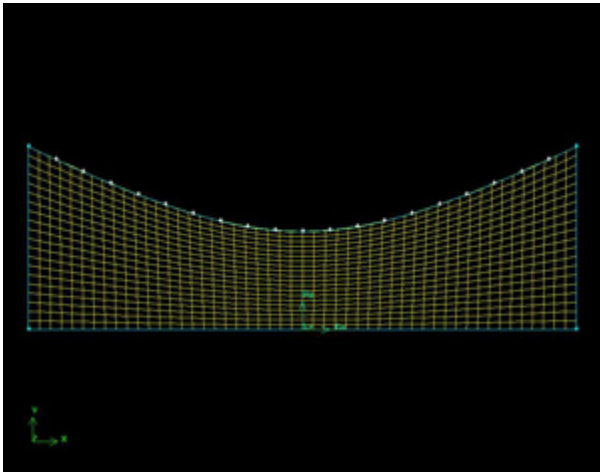
[Higher Resolution Image](#)

Mesh Face

Now that we have the edges meshed, we need to mesh the face.

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

As before, select the face and click the *Apply* button.



[Higher Resolution Image](#)

Save Your Work

Main Menu > File > Save

Go to [Step 3: Mesh](#)

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

[Go to all FLUENT Learning Modules](#)