

ANSYS 12 - Beam (2D Element) - 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

Step 3: Mesh

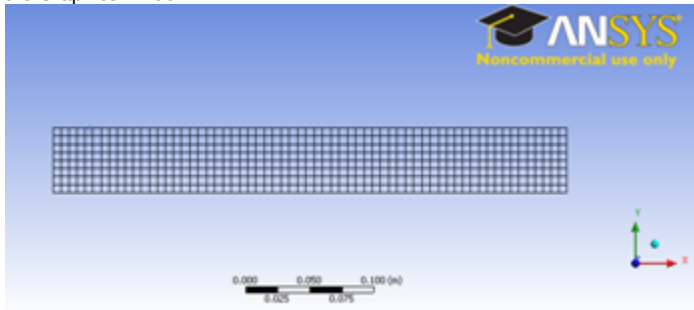
Save your work in **Workbench** window. In the **Workbench** window, right click on **Mesh**, and click **Edit**. A new **ANSYS Mesher** window will open.

We would like to create a structured mesh where the opposite edges correspond with each other. Let's insert a Mapped Face mesh.

Outline > Mesh > Insert > Mapped Face Meshing

Under **Outline**, right click on **Mesh**, move cursor to **Insert**, and select **Mapped Face Meshing**. Finally select the beam surface body in the **Graphics** window and click **Apply** next to **Geometry**.

We can now generate the mesh using the default setting. Under **Outline**, right click on **Mesh** and click **Generate Mesh**. This should be the mesh appear in the Graphics window.



[Higher Resolution Image](#)

Under **Details of "Mesh"**, you should see that we have 504 elements when you expand the **Statistics** tree.

Details of "Mesh"	
[-] Defaults	
Physics Preference	Mechanical
Relevance	0
+ Sizing	
+ Inflation	
+ Advanced	
+ Pinch	
[-] Statistics	
Nodes	1655
Elements	504
Mesh Metric	None

[Go to Step 4: Setup \(Physics\)](#)

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

[Go to all ANSYS Learning Modules](#)