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

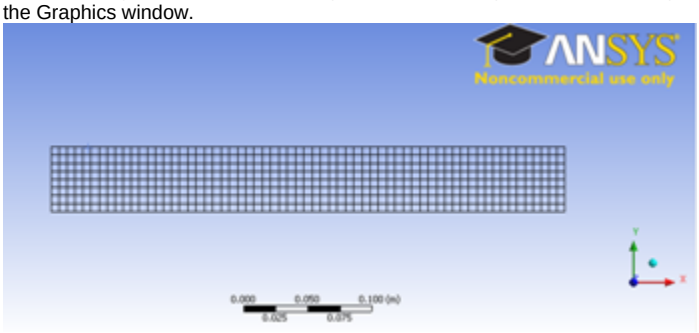
Save your work in ***Workbench*** window. In the ***Workbench*** window, right click on ***Mesh***, and click ***Edit***. A new ***ANSYS Mesher*** window will open.

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