

2D Beam - Mesh

Authors: Rajesh Bhaskaran and Vincent Prantil

[Problem Specification](#)

[1. Pre-Analysis & Start-Up](#)

[2. Geometry](#)

[3. Mesh](#)

[4. Physics Setup](#)

[5. Numerical Solution](#)

[6. Numerical Results](#)

[7. Verification & Validation](#)

[Exercises](#)

[Comments](#)

Mesh

The following video shows how to create a regular mesh for the rectangle using "Mapped face meshing".

Summary of steps in the above video:

1. In Workbench, double-click on Model cell A4.
2. To view the default mesh:
 - a. Highlight Mesh in the Project Tree.
 - b. Click Update.
3. To make a regular mesh:
 - a. Right click on Mesh > Insert > Mapped Face Meshing.
 - b. Using the Face Selection Tool, click on the rectangular geometry. Click Geometry > Apply.
 - c. Click Update.
4. To change the element size:
 - a. Highlight Mesh in the Project Tree. Right click on Mesh > Insert > Sizing.
 - b. Using the Face Selection Tool, click on the rectangular geometry. Click Geometry > Apply.
 - c. Next to Element Size, type the following to insert an expression: $= 8/3$
 - d. Hit Enter.
5. Save Project.

For this example, we would like to use "Q4" elements i.e. quadrilateral elements with 4 nodes for every element, one at each corner. It turns out that the quadrilateral elements in the mesh obtained above have mid-side nodes in addition to the corner nodes. So these elements are "Q8" elements with a total of 8 nodes for each element. In the video below, we drop mid-side nodes to get "Q4" elements (in the ANSYS element library, these are referred to as PLANE 182). Note that the mid-side nodes provide higher accuracy, so in the real world, you would not drop mid-side nodes unless you had a compelling reason to do so. Here we do it for learning purposes.

Summary of steps in the above video:

1. To double check on mid-side nodes:
 - a. Zoom in to top left corner of rectangle by holding down the right mouse button and drawing a box over the area to which you want to zoom into.
 - b. In the toolbar menu, click on Select Type (Geometry/Mesh) > Select Mesh.
 - c. In the toolbar menu, click on Select Mode > Select Box Select.
 - d. Draw a box over a few elements in the mesh by holding down the right mouse button.
2. To get rid of midside nodes:
 - a. Highlight Mesh in the Project Tree.
 - b. In Details of 'Mesh', expand Advanced. Element Midside Nodes > Dropped.
 - c. Click Update.
 - d. Repeat steps 1a-d to check that midside nodes have been dropped.
3. Save Project.

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

[Go to all ANSYS Learning Modules](#)