

# ANSYS - 3D Curved Beam step 5

## Problem Specification

1. Start-up and preliminary set-up
2. Specify element type and constants
3. Specify material properties
4. Specify geometry
5. **Mesh geometry**
6. Specify boundary conditions
7. Solve!
8. Postprocess the results
9. Validate the results

## Step 5: Mesh geometry

Bring up the *MeshTool*:

**Main Menu > Preprocessor > MeshTool**

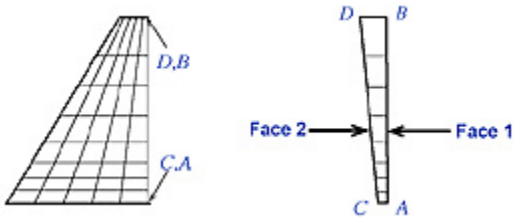
The *MeshTool* is used to control and generate the mesh.

### Set Meshing Parameters

The element type and material property set to be used in meshing are automatically selected since we have only one of each. To check this using the *Mesh Tool*, make sure **Global** is selected under **Element Attributes** and click on **Set**. You will see that the correct element type and material number are already selected in the *Meshing Attributes* menu. There is no real constant set required for the SOLID45 element with default options (which we haven't changed). Click **Cancel**.

### Set Mesh Size

Two views of the FE mesh we want to use are shown in the figure below.

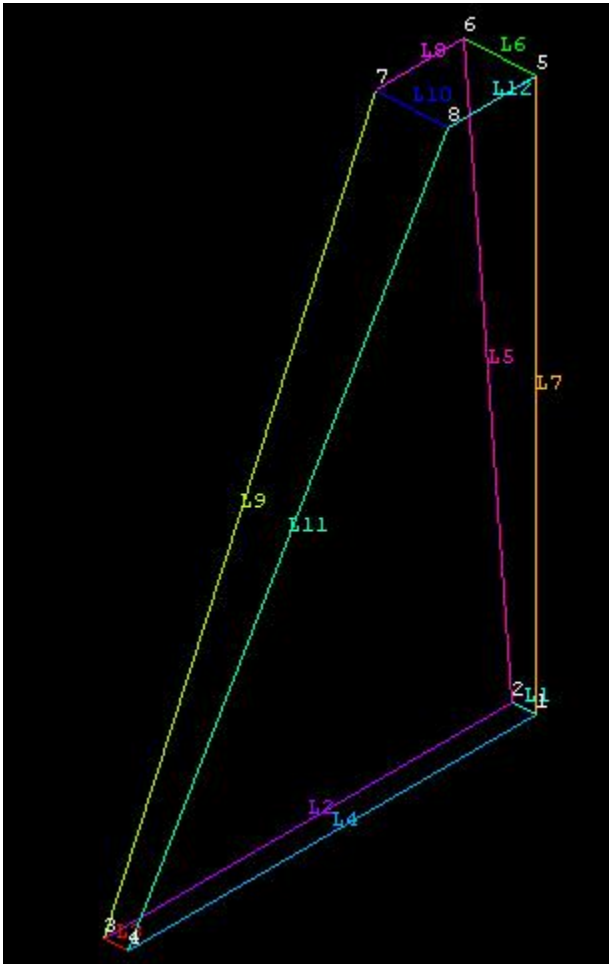


Curved beam theory predicts that the stress gradients will be highest on the edge nearest the center of curvature. So we want to use a graded mesh such that the radial length of elements increases as we go from edge AC towards edge BD. The *spacing ratio* parameter is used to control mesh grading in ANSYS. If positive, the spacing ratio is the ratio of last division size to first division size. Of course, the direction of line AB, for example, can be from A to B or B to A. We need to know the direction in order to determine which is the first division and which is the last. The way to determine the line direction is shown below. The default spacing ratio is one i.e. no mesh grading. Along lines AB and CD, we will use a spacing ratio of 0.3 (with the mesh spacing increasing from A to B and C to D).

Turn on line and keypoint numbers:

**Utility Menu > PlotCtrls > Numbering**

Turn on **Keypoint numbers** and **Line numbers** and click **OK**.



#### Utility Menu > List > Lines > OK

Looking at the table and figure, we see that the line 7 (or L7) goes from keypoint 5 (vertex *B*) to keypoint 1 (vertex *A*). So the first division is that next to *B* and the last division is that next to *A*. Since we want the radial mesh spacing to decrease from *B* to *A*, the spacing ratio is 0.3 (rather than 1/0.3). Similarly, determine the direction and spacing ratio for L5, L9 and L11.

Let's make a table summarizing the number of divisions (*NDIV*) and spacing ratio (*SPACE*) for each line.

Line no.	<i>NDIV</i>	<i>SPACE</i>
L1,L3,L6,L10	1	1
L2,L4,L8,L12	5	1
L7,L9,L11	8	0.3
L5	8	1/0.3

Your keypoint and line numbers may be different from the above and you should make your own table to account for that. Refer to the table while following the instructions given below for specifying no. of divisions and spacing ratio for each line.



If you have trouble selecting the correct line below, **hold down** the left mouse button until the line is selected and then release the left button. If you want to **deselect** a line, right-click to go into deselect mode, left-click on the line to be deselected and right-click again to go back into select mode.

In *MeshTool*, under **Size Controls** and **Lines**, click **Set**. This brings up a pick menu.

Pick lines L1,L3,L6, and L10 and click **OK** in the pick menu. Enter 1 for **No. of element divisions** and click **Apply**. (The **Spacing Ratio** field can be left blank since the default value is one.)

Pick lines L2,L4,L8, and L12 and click **OK** in the pick menu. Enter 5 for **No. of element divisions** and click **Apply**.

Pick lines L7,L9, and L11 in the *Graphics* window and click **OK** in the pick menu. Enter 8 for **No. of element divisions**, 0.3 for **Spacing Ratio** and click **Apply**.

Pick line L5 in the *Graphics* window and click **OK** in the pick menu. Enter 8 for **No. of element divisions**, 1/0.3 for **Spacing Ratio** and click **OK**.

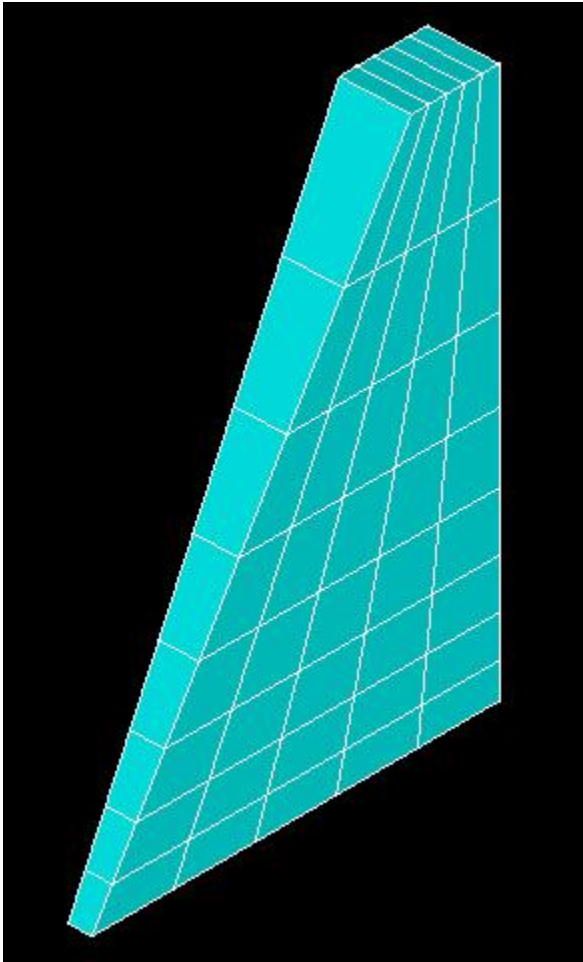
## Mesh Volume

We'll mesh the volume using hexahedral elements (rather than tetrahedral elements). So choose **Hex** under **Shape** in the *MeshTool*.

Make sure **Volumes** is selected in the drop-down list next to **Mesh** in the *MeshTool*. This means the geometry component to be meshed is a volume. Click

on  on .

Click on **Pick All** in the pick menu. The volume is meshed.



**Utility Menu > PlotCtrls > Pan, Zoom, Rotate**

Look at various views of the mesh and satisfy yourself that it looks right.

Close the *MeshTool*.

Save your work: **Toolbar > SAVE\_DB**

[Go to Step 6: Specify boundary conditions](#)

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

[Go to all ANSYS Learning Modules](#)