Hands on Exercise for BEAM 188 or BEAM 189

Problem Description

A straight, slender cantilever beam with rectangular section has one fixed end and one free end. A load is applied to the free end. The model is analyzed using static calculations.

Problem Specifications

The following material properties are used for this problem:
Young's modulus = 1.0 x 10^4 psi
Poisson's ratio = 0.3
The following geometric properties are used for this problem:
L = 100 in
H = 5 in
B = 0.2 in
Loading for this problem is:
P = 1 lb.

Set the Analysis Title and Define Model Geometry

1. Choose menu path **Utility Menu> File> Change Title**.
2. Enter the text "A cantilever beam using Beam 188" and click on **OK**.
3. Start the model creation preprocessor and define the keypoints for the beam. Choose menu path
Main Menu> Preprocessor> Modeling> Create> Keypoints> In Active CS, and enter these keypoint numbers and the coordinates in the dialog box as indicated:

Keypoint Number	X Location	Y Location	Z Location	Click This Button to Accept Values
1	0	0	0	**Apply**
2	100.0	0	0	**Apply**
3	50	5		**OK**

4. Create a straight line through keypoints 1 and 2. Choose menu path
Main Menu>Preprocessor> Modeling> Create> Lines> Lines> Straight Line. The Create Straight Line picker appears. Select keypoints 1 and 2 in the Graphics window and click on **OK** in the Create Straight Line picker.

5. Save the model. Choose menu path **Utility Menu> File> Save As**.
Enter `geometri.db` in the Save Database to box and click on **OK**.

Define Element Type and Cross Section Information

1. Choose menu path **Main Menu> Preferences** and select the "Structural" check box.
Click on **OK** to continue.

2. Choose menu path **Main Menu> Preprocessor> Element Type> Add/Edit/Delete**. The Element Types dialog box appears.

3. Click on Add ... The Library of Element Types dialog box appears.

4. In the scroll box on the left, click on "Structural Beam" to select it.

5. In the scroll box on the right, click on "3D finite strain, 2 node 188" to select BEAM188. Click **OK**
Highlight the line with the element and Click Options… The ..Element type options dialog box appears. Choose:
Warping degree of freedom to K1 to **Restrained**
Shear stress output K4 to **Include Both**
Click Close in the Element Types dialog box.

7. Define a rectangular cross section for the beam. Choose menu path **Main Menu>Preprocessor> Sections> Beam> Common Sectns**.
The BeamTool is displayed. ANSYS sets the section ID to 1, and the subtype to RECT (signified by a rectangle on the subtype button) by default. Since you will be creating a rectangular cross section, there is no need to change the subtype.

8. In the lower half of the BeamTool, you will see a diagram of the cross section shape with dimension variables labeled. Enter the width of the cross section, 0.2, in the box labeled B. Enter the height of the cross section, 5.0, in the box labeled H. Click on Apply to set the cross section dimensions.

9. Use the BeamTool to display information about the cross section. Click on the **Preview** button on the BeamTool. A diagram and data summary of the cross section appear in the Graphics window. You can also preview the mesh of the cross section by selecting the **Meshview** button. Click on the **Close** button in the BeamTool to continue.

Define the Material Properties and Orientation Node

1. Choose menu path **Main Menu> Preprocessor> Material Props> Material Models**. The Define Material Model Behavior dialog box appears.

2. In the Material Models Available window on the right, double-click on the following: **Structural**, **Linear**, **Elastic**, **Isotropic**. A dialog box appears.

3. Enter 1E4 for EX (Young's modulus).

4. Enter 0.3 for PRXY (Poisson's ratio), and click on **OK**. Material Model Number 1 appears in the Material Models Defined window on the left.

5. Choose menu path **Material> Exit** to close the Define Material Model Behavior dialog box.

6. Replot the line by choosing menu path **Utility Menu> Plot> Lines**.

7. Select the line and define the orientation node of the line as an attribute. Choose menu path **Main Menu> Preprocessor> Meshing> Mesh Attributes> Picked Lines**. The Line Attributes picker appears. Select the line in the Graphics window and click on **Apply** in the Line Attributes picker.

8. The Line Attributes dialog box appears. ANSYS includes the material attribute pointer to the material set 1, the element type attribute pointer to the local element type 1, and the section attribute pointer to the section ID 1 by default. Click on the radio button beside the Pick Orientation Keypoint(s) label to change it to Yes and click on **OK**.

9. The Line Attributes picker reappears. Type 3 (Orientation Point) in the picker, press the **Enter** key, and click on **OK**.

10. Save the model. Choose menu path **Utility Menu> File> Save As**. `model.db`

Mesh the Line and Verify Beam Orientation

1. Define the mesh size and number of divisions. Choose menu path **Main Menu> Preprocessor> Meshing> Size Cntrls> ManualSize> Lines> All Lines**.
Enter 10 in the No. of Element Divisions box and click on **OK**.

2. Mesh the line. Choose menu path **Main Menu> Preprocessor> Meshing> MeshTool**. Click on MESH on the MeshTool and the Mesh Lines picker appears.

Pick the line in the Graphics window, and then click on **OK** in the Mesh Lines picker. Click on Close in the MeshTool to close it.

3. Rotate the meshed line. Choose menu path **Utility Menu> PlotCtrls> Pan, Zoom, Rotate**. The Pan, Zoom, Rotate tool appears. Select ISO and click on Close. The beam is rotated in the Graphics window.

4. Verify the beam orientation. Choose menu path **Utility Menu> PlotCtrls> Style> Size and Shape**. Select the radio button next to the **/ESHAPE** label to turn **/ESHAPE** on and click on **OK**.

Define the Boundary Conditions

1. Define a boundary condition to the fixed end. Choose menu path **Main Menu> Solution> Define Loads> Apply> Structural> Displacement> On Keypoints**. The Apply U,ROT on KPs picker appears.

2. Define keypoint 1 as the fixed end. In the picker, type 1, press the **Enter** key, then click on **OK**. The Apply U,ROT on KPs dialog box appears.

3. Click on "All DOF" to select it, and click on **OK**. The boundary condition information appears in the ANSYS Graphics window at keypoint 1.

4. Apply a force to the free end. Choose menu path **Main Menu> Solution> Define Loads> Apply> Structural> Force/Moment> On Keypoints**. The Apply F/M on KPs picker appears.

5. Identify keypoint 2 as the free end. Type 2 in the picker, press the **Enter** key, and then click on **OK**. The Apply F/M on KPs dialog box appears.

6. In the drop down list for Direction of force/mom, select FY.

7. Enter 1 for the Force/moment value in the Apply F/M on KPs dialog box, and click on **OK**.
The force symbol appears in the ANSYS Graphics window at keypoint 2.

8. Save the model. Choose menu path **Utility Menu> File> Save As**. `solve.db`
Click on **OK**

Solve the Static Analysis

1. Choose menu path **Main Menu> Solution> Solve> Current LS**. Review the summary information in the /STAT command window, then select Close from its menu bar. Click on **OK** in the Solve Current Load Step window to begin the solution.

2. When the Solution is Done! window appears, click on Close to close it.

3. Choose menu path **Main Menu> Finish**.

Plot and Review the Results

1. Replot the beam. Choose menu path **Utility Menu> Plot> Elements**.

2. Choose menu path **Utility Menu> PlotCtrls> Style> Size and Shape**.
Be sure the radio button beside the label Display of element shapes ... (**/ESHAPE**) is set to On, and click on **OK**.

3. Plot the shape of the beam. Choose menu path
Main Menu> General Postproc> Plot Results> Deformed Shape.
The Plot Deformed Shape dialog box appears. Select Def + undef edge and click on **OK**.

4. **Main Menu> General Postproc> Plot Results > Contur Plot > Nodal Solu**
The Contour Nodal Solution Data dialog box appears.

Select DOF Solution > select the displacement you want to see

4 **Main Menu> General Postproc> Plot Results > Contur Plot > Nodal Solu**
The Contour Nodal Solution Data dialog box appears.

Select Stress > select the stress you want to see and click on **OK**.