Chapter 6: Modal Analysis of a Cantilevered Tapered Beam

Keywords: elastic beam, 2D elasticity, plane stress, convergence, modal analysis

Modeling Procedures: ruled surface, convert

6.1 Problem Statement and Objectives

It is required to determine the natural frequencies and mode shapes of vibration for a cantilevered tapered beam. The geometrical, material, and loading specifications for the beam are given in Figure 6.1. The geometry of the beam is the same as the structure in Chapter 3. The thickness of the beam is $2h$ inches, where $h$ is described by the equation: 

$$h = 4 - 0.6x + 0.03x^2$$

6.2 Analysis Assumptions

- Because the beam is thin in the width (out-of-plane) direction, a state of plane stress can be assumed.
- The length-to-thickness ratio of the beam is difficult to assess due to the severe taper. By almost any measure, however, the length-to-thickness ratio of the beam is less than eight. Hence, it is unclear whether thin beam theory will accurately predict the vibratory response of
the beam. Therefore, both a 2D plane stress elasticity analysis and a thin elastic beam analysis will be performed.

6.3 Mathematical Idealization

Based on the assumptions above, two different models will be developed and compared. The first model is a beam analysis. In this model, the main axis of the beam is discretized using straight two-noded 1D thin beam finite elements having a uniform cross-sectional shape and mass distribution within each element. Thus, the geometry is idealized as having a piecewise constant cross-section, as shown in Figure 6.2. The uniform thickness within each element is taken to be equal to the actual thickness of the tapered beam at the x-coordinate corresponding to the centroid of that element. Note that this type of geometry approximation also leads to an approximation of the overall mass as well as its distribution. Since the mass distribution plays a strong role in vibratory motion, the effect of this approximation should be considered carefully. As the mesh is refined, the error associated with this approximation will be reduced.

Because beam elements are designed to capture three-dimensional behavior, the beam model will predict three-dimensional modes of vibration unless additional constraints are imposed. In the present case, we are most interested in the modes of vibration that occur within the plane shown in Figures 6.1 and 6.2. Therefore, we should apply constraints to the model such that the beam cannot translate or rotate out of the plane.

![Figure 6.2 Idealized geometry for a tapered beam.](image)

The second model is a 2D plane stress model of the geometry as shown in Figure 6.1. The 2D finite element model of this structure will be developed using 2D plane stress bilinear four-noded quadrilateral finite elements. In the present analysis, the geometry and material properties are
symmetric about the mid-plane of the beam. However, the vibratory response is not symmetric about this plane. Hence, it is necessary to model the entire domain of the beam, as shown in Figure 6.1. Note that the mass distribution is accurately represented in the 2D model.

In free vibration analysis, no loads are applied. The goal of the analysis is to determine at what frequencies a structure will vibrate if it is excited by a load that is applied suddenly and then removed. These frequencies are called natural frequencies, and they are a function of the shape, the material, and the boundary constraints of the structure. Mathematically, the natural frequencies are associated with the eigenvalues of an eigenvector problem that describes harmonic motion of the structure. The eigenvectors are the mode shapes associated with each frequency.

6.4 Finite Element Model

The procedure for creating the finite element model and obtaining the finite element solution for each type of model is presented at the end of this chapter. The 1D beam analysis should be performed three times, each with a different mesh. Meshes consisting of 2, 4 and 6 elements should be developed. The 2D analysis should be performed only one time, using the mesh described within the procedure.

6.5 Model Validation

Simple analytical formulas are available for predicting the axial and bending natural frequencies of a uniform cantilevered beam. These results can be used to estimate the natural frequencies in a tapered beam and thus to assess the validity of the finite element results (i.e., to make sure that the finite element results are reasonable and do not contain any large error due to a simple mistake in the model).

For the case of axial vibratory motion, an analytical solution can be developed as follows. We assume that the axial displacement of the bar $u(x,t)$ is separable in space and time, or

$$u(x,t) = U(x)F(t)$$

where $F(t)$ is a harmonic function. When the material and geometric properties are uniform throughout the bar, the governing eigenvalue equation for axial vibratory motion is:

$$\frac{d^2U}{dx^2} + \beta^2 U = 0 \quad \beta^2 = \omega^2 \frac{m}{EA}$$
In the above equation, \( \omega = 2\pi f \) is the circular frequency measured in radians per second, \( f \) is the frequency measured in cycles per second, \( m \) is the mass per unit length of the bar, \( E \) is the modulus of elasticity, and \( A \) is the cross-sectional area.

The solution to the above equation takes the form:

\[
U = A \sin \beta x + B \cos \beta x
\]

where \( A \) and \( B \) are constants of integration that must be determined from the boundary conditions. In the present problem, the boundary conditions are of the form:

\[
U(0) = 0 \quad F \bigg|_{x=L} = EA \frac{dU}{dx} \bigg|_{x=L} = 0
\]

The first boundary condition yields \( B = 0 \). The second boundary condition yields:

\[
A \beta \cos \beta L = 0
\]

Since \( \beta \neq 0 \) and \( A = 0 \) is the trivial solution, we must have:

\[
\cos \beta L = 0
\]

which is referred to as the characteristic equation. There are many solutions to this equation:

\[
\beta_n L = \frac{n\pi}{2} \quad \text{for } n = 1, 3, 5, 7, \ldots
\]

The circular frequencies of the motion are then obtained as:

\[
\omega_n = \frac{n\pi}{2} \sqrt{\frac{EA}{mL^2}}
\]

The eigenfunctions, or mode shapes, are then found to be:

\[
U_n = A_n \sin \frac{n\pi x}{2L}
\]

where \( A_n \) are the amplitudes of the shapes, which cannot be determined uniquely. The lowest natural frequency associated with axial motion of the bar is found for \( n = 1 \):
\[ \omega_1 = \frac{\pi}{2} \sqrt{\frac{EA}{mL^2}} \]

By approximating the shape and mass distribution of the tapered bar as being uniform, this formula can be used to estimate the lowest natural frequency associated with axial motion of the tapered bar. The results will not be exact, but they should be sufficiently close to indicate whether the finite element results are reasonable, which is the goal of validation.

In a similar way, formulas for the first three flexural frequencies of a cantilevered uniform beam can be obtained as:

\[ \omega_1 = 1.875^2 \sqrt{\frac{EI}{mL^4}} \quad \omega_2 = 4.694^2 \sqrt{\frac{EI}{mL^3}} \quad \omega_3 = 7.855^2 \sqrt{\frac{EI}{mL^4}} \]

Note that Mentat presents the frequencies in Hz, so the circular frequencies calculated using the formulas above should be converted to Hz for comparison.

The above formulations are valid for a beam that has a uniform distribution of shape and mass along the length. In order to use these results for validating the finite element results of the current tapered beam, a suitable uniform shape must be assumed. As a first-order approximation, the uniform beam shape may be taken as the shape at the mid-span of the tapered beam. In other words, the cross-sectional shape at the point \( x = 5.0 \) in the tapered beam may be used as the uniform shape in the validation calculations. The mass distribution is also assumed to be uniform, and follows directly from the assumed shape and the known material density. Note, however, that the overall mass of the uniform beam may be different than that of the tapered beam. Moreover, the distribution of the mass as well as the stiffness is very different.

It should be expected that the tapered beam under consideration would behave differently than a uniform beam, so comparisons between the finite element results of the tapered beam and those of the above hand calculations must be performed thoughtfully. For example, it may be expected that the tapered beam will have a higher frequency than the uniform beam for at least two reasons. First, the tapered beam is stiffer near its root, where the loads are highest. Second, the tapered beam has less mass near its free end than does the uniform beam, so the inertial loading on the tapered beam is less than that in the uniform beam. By using this kind of intuition, a meaningful comparison between the hand calculations and the finite element results can be made.
6.6 Post Processing

A total of four finite element models are to be developed – three using 1D two-noded thin beam elements, and one using 2D four-noded bilinear plane stress elements. Based on the results of these analyses, perform and submit the following post processing steps.

(1) Complete the following table:

<table>
<thead>
<tr>
<th>Model ID</th>
<th>Frequency of First Axial Mode (Hz)</th>
<th>Frequency of First Bending Mode (Hz)</th>
<th>Frequency of Second Bending Mode (Hz)</th>
<th>Frequency of Third Bending Mode (Hz)</th>
</tr>
</thead>
<tbody>
<tr>
<td>1D – two elements</td>
<td></td>
<td></td>
<td></td>
<td></td>
</tr>
<tr>
<td>1D – four elements</td>
<td></td>
<td></td>
<td></td>
<td></td>
</tr>
<tr>
<td>1D – six elements</td>
<td></td>
<td></td>
<td></td>
<td></td>
</tr>
<tr>
<td>2D – plane stress elements</td>
<td></td>
<td></td>
<td></td>
<td></td>
</tr>
<tr>
<td>Validation hand calculation</td>
<td></td>
<td></td>
<td></td>
<td></td>
</tr>
</tbody>
</table>

(2) On a single graph, plot the first axial mode as predicted by the four models. On a single graph, plot the first bending mode as predicted by the four models. In each case, normalize the mode shapes such that the maximum displacement (amplitude) is unity. For the 2D plane stress model, select the displacements along the mid-surface of the beam for plotting purposes.

(4) Comment on the convergence of frequencies and mode shapes in the 1D beam solutions.

(5) Comment on the validity of the solutions. Show the hand calculations.

(6) For each model, include a deformed mesh plot of the first five modes of vibration.
MODAL ANALYSIS OF TAPERED BEAM -- using two elastic beam elements

1. Add points to define geometry.

1a. Add points.

   <ML> MAIN MENU / MESH GENERATION
   <ML> MAIN MENU / MESH GENERATION / PTS ADD

   Enter the coordinates at the command line, one point per line with a space separating each coordinate.
   
   >  0.0  0.0  0.0
   >  5.0  0.0  0.0
   > 10.0  0.0  0.0

   The points may not appear in the Graphics window because Mentat does not yet know the size of the model being built. When the FILL command in the static menu is executed, Mentat calculates a bounding box for the model and fits the model inside the Graphics window.

   <ML> STATIC MENU / FILL

   The points should now be visible in the Graphics window.

1b. Display point labels.

   <ML> STATIC MENU / PLOT
   <ML> STATIC MENU / PLOT / POINTS SETTINGS
   <ML> STATIC MENU / PLOT / POINTS SETTINGS / LABELS
   <ML> STATIC MENU / PLOT / POINTS SETTINGS / LABELS / REDRAW

1c. Return to MESH GENERATION menu.

   <MR> or <ML> RETURN

   The result of this step is shown in Figure 6.3.
If the steps above were not followed precisely (e.g., if the points were entered in an order different than the order in which they appear in the above list), then the point labels will differ from those shown in Figure 6.3. These labels are simply used as identifiers in the following step, and do not affect the model. As long as the correct coordinates were entered, do not worry if the labels are not exactly as shown in Figure 6.3. Just keep track of the differences between the labels so that the appropriate procedures will be followed in the steps below.

2. Add two 2-noded line elements.

2a. Select ELEMENT CLASS.

In the MESH GENERATION menu, the currently selected type of element that can be generated is displayed to the immediate right of the ELEMENT CLASS button. Change the element type to LINE (2):

<ML> MAIN MENU / MESH GENERATION / ELEMENT CLASS
<ML> MAIN MENU / MESH GENERATION / ELEMENT CLASS / LINE (2)
<ML> MAIN MENU / MESH GENERATION / ELEMENT CLASS / RETURN

2b. Create a line element from point 1 to point 2 and from point 2 to point 3.
<ML> MAIN MENU / MESH GENERATION / ELEMS ADD

<ML> to select point 1 and then point 2 to create an element from point 1 to point 2.
<ML> to select point 2 and then point 3 to create an element from point 2 to point 3.

2c. Turn off point labels.

<ML> STATIC MENU / PLOT / POINTS SETTINGS
<ML> STATIC MENU / PLOT / POINTS SETTINGS / LABELS
<ML> STATIC MENU / PLOT / POINTS SETTINGS / LABELS / REDRAW

2d. Return to MESH GENERATION menu.

<MR> or <ML> RETURN

The result of this step is shown in Figure 6.4.

| Figure 6.4 |

3. Sweep the mesh to insure that all elements are properly connected.

<ML> MAIN MENU / MESH GENERATION / SWEEP
<ML> MAIN MENU / MESH GENERATION / SWEEP / ALL

Note: Duplicate geometrical and mesh entities will be deleted so that proper mesh connectivity is achieved.

Return to MESH GENERATION menu.
4. Add boundary conditions.

Note that only boundary constraints are imposed. No mechanical loads are applied when performing free vibration (modal) analysis.

4a. Specify the constraint condition on the left end of the model.

4a1. Set up a new boundary condition set.

<ML> MAIN MENU / BOUNDARY CONDITIONS
<ML> MAIN MENU / BOUNDARY CONDITIONS / MECHANICAL
<ML> MAIN MENU / BOUNDARY CONDITIONS / MECHANICAL / NEW
<ML> MAIN MENU / BOUNDARY CONDITIONS / MECHANICAL / NAME

At the command line, enter a name for this boundary condition set.

> FixedPoint

4a2. Define the nature of the boundary condition.

<ML> MAIN MENU / BOUNDARY CONDITIONS / MECHANICAL / FIXED-DISPLACEMENT

Note: Because beam elements have a total of six DOFs (three displacements and three rotations) at each node, it is necessary to constrain the displacements and rotations in all three directions at the left edge of the model so as to restrain all possible rigid body modes.

<ML> MAIN MENU / BOUNDARY CONDITIONS / MECHANICAL / FIXED-DISPLACEMENT / DISPLACEMENT X
<ML> MAIN MENU / BOUNDARY CONDITIONS / MECHANICAL / FIXED-DISPLACEMENT / DISPLACEMENT Y
<ML> MAIN MENU / BOUNDARY CONDITIONS / MECHANICAL / FIXED-DISPLACEMENT / DISPLACEMENT Z
<ML> MAIN MENU / BOUNDARY CONDITIONS / MECHANICAL / FIXED-DISPLACEMENT / ROTATION X
<ML> MAIN MENU / BOUNDARY CONDITIONS / MECHANICAL / FIXED-DISPLACEMENT / ROTATION Y
<ML> MAIN MENU / BOUNDARY CONDITIONS / MECHANICAL / FIXED-DISPLACEMENT / ROTATION Z

The small box to the immediate left of the DISPLACEMENT X, Y and Z and ROTATION X, Y and Z buttons should now be
highlighted. The “0” that appears to the right of these buttons is the imposed value of the displacement or rotation. If a component of displacement or rotation is non-zero, then the actual value of the displacement or rotation should be entered here.

<ML> MAIN MENU / BOUNDARY CONDITIONS / MECHANICAL / FIXED-DISPLACEMENT / OK

4a3. Apply the condition to the node on the left edge.

<ML> MAIN MENU / BOUNDARY CONDITIONS / MECHANICAL / NODES ADD
<ML> to select the node on the left edge of the model.
<MR> or END LIST

The result of this step is shown in Figure 6.5.

Figure 6.5
4b. Specify the constraint condition at remaining nodes to restrict out of plane motion. In other words, we will only allow the beam to move in the XY plane.

4b1. Set up a new boundary condition set.

<ML> MAIN MENU / BOUNDARY CONDITIONS
<ML> MAIN MENU / BOUNDARY CONDITIONS / MECHANICAL
<ML> MAIN MENU / BOUNDARY CONDITIONS / MECHANICAL / NEW
<ML> MAIN MENU / BOUNDARY CONDITIONS / MECHANICAL / NAME

At the command line, enter a name for this boundary condition set.

> OutOfPlane

4b2. Define the nature of the boundary condition.

<ML> MAIN MENU / BOUNDARY CONDITIONS / MECHANICAL / FIXED-DISPLACEMENT

Note: Out of plane motion is fully constrained by restricting the Z-translation, the X-rotation, and the Y-rotation.

<ML> MAIN MENU / BOUNDARY CONDITIONS / MECHANICAL / FIXED-DISPLACEMENT / DISPLACEMENT Z
<ML> MAIN MENU / BOUNDARY CONDITIONS / MECHANICAL / FIXED-DISPLACEMENT / ROTATION X
<ML> MAIN MENU / BOUNDARY CONDITIONS / MECHANICAL / FIXED-DISPLACEMENT / ROTATION Y

The small box to the immediate left of the DISPLACEMENT X, Y and Z and ROTATION X, Y and Z buttons should now be highlighted. The "0" that appears to the right of these buttons is the imposed value of the displacement or rotation.

<ML> MAIN MENU / BOUNDARY CONDITIONS / MECHANICAL / FIXED-DISPLACEMENT / OK

4b3. Apply the condition to nodes 2 and 3 (all nodes except the node on the left edge of the model).

<ML> MAIN MENU / BOUNDARY CONDITIONS / MECHANICAL / NODES ADD

<ML> to select nodes 2 and 3 of the model.

<MR> or END LIST
The result of this step is shown in Figure 6.6.

Figure 6.6 is not available at this time.

4c. Display all boundary conditions for verification.

<ML> MAIN MENU / BOUNDARY CONDITIONS / ID BOUNDARY CONDS

After verifying that boundary conditions have been applied properly, turn off the boundary condition ID's by repeating the last command.

4d. Return to the MAIN menu.

<ML> MAIN MENU / BOUNDARY CONDITIONS / MAIN

5. Specify the material properties of each element.

5a. Set up a new material property set.

<ML> MAIN MENU / MATERIAL PROPERTIES
<ML> MAIN MENU / MATERIAL PROPERTIES / NEW
<ML> MAIN MENU / MATERIAL PROPERTIES / NAME

At the command line, enter a name for this material property set.
5b. Define the properties of the material.

> Steel

<ML> MAIN MENU / MATERIAL PROPERTIES / ISOTROPIC
<ML> MAIN MENU / MATERIAL PROPERTIES / ISOTROPIC /
          YOUNG'S MODULUS
> 29.0e6

<ML> MAIN MENU / MATERIAL PROPERTIES / ISOTROPIC
<ML> MAIN MENU / MATERIAL PROPERTIES / ISOTROPIC /
          MASS DENSITY
> 0.0007356

Note: Only Young's modulus and mass density need to be specified for this problem. The mass density is input in units of lbm/in\(^3\). The specific weight is provided in units of lbf/in\(^3\), which must be converted to lbm/in\(^3\) by dividing by a factor of \((32.174)\times(12)\).

<ML> MAIN MENU / MATERIAL PROPERTIES / ISOTROPIC / OK

5c. Apply the material properties to all elements.

<ML> MAIN MENU / MATERIAL PROPERTIES / ELEMENTS ADD

Since the properties are being applied to all elements in the model, the simplest way to select the elements is to use the ALL EXISTING option.

<ML> ALL: EXIST.

5d. Display all material properties for verification.

<ML> MAIN MENU / MATERIAL PROPERTIES / ID MATERIALS

After verifying that material properties have been applied properly, turn off the material property ID's by repeating the last command.

5e. Return to the MAIN menu.

<ML> MAIN MENU / MATERIAL PROPERTIES / MAIN

6. Specify the geometrical properties of each element. For a beam element, it is necessary to specify (i) the cross-sectional area, (ii) the second moments of area (Ixx, Iyy) about the two local (principal) axes of the cross-section,
and (iii) a vector that defines the direction of the local X-axis.

Note that a local coordinate system must be defined for each beam element. All geometric properties are then defined with respect to this local coordinate system. By default, the local Z-axis is taken along the length of the element, and the local X- and Y-axes are taken in the plane of the cross-section of the beam element. The first principal axis is called the “LOCAL X-AXIS” and the second principal axis is called the “LOCAL Y-AXIS.” The local X-axis is the axis about which Ixx is calculated.

In the present analysis, the local X-axis is taken to be the same as the global Z-axis (positive out of the computer screen). According to the right-hand rule, the local Y-axis will automatically be taken as the negative global Y-axis. So the vector defining the local X-axis is (0,0,1).

6a. Specify geometrical properties for element one.

6a1. Set up a new geometric property set.

<ML> MAIN MENU / GEOMETRIC PROPERTIES
<ML> MAIN MENU / GEOMETRIC PROPERTIES / NEW
<ML> MAIN MENU / GEOMETRIC PROPERTIES / NAME

At the command line, enter a name for this geometric property set.

> X1

6a2. Define the geometric properties.

<ML> MAIN MENU / GEOMETRIC PROPERTIES / 3D
<ML> MAIN MENU / GEOMETRIC PROPERTIES / 3D / ELASTIC BEAM
<ML> MAIN MENU / GEOMETRIC PROPERTIES / 3D / ELASTIC BEAM / AREA

The cross-sectional area of element one is taken as the cross-sectional area of the bar at the geometric centroid of the element (i.e., at x=2.5).

> 5.375

<ML> MAIN MENU / GEOMETRIC PROPERTIES / 3D / ELASTIC BEAM / Ixx
The second moment of the area about the local x-axis (I_{xx}) is calculated as \( I_{xx} = (b)(h^3)/12 \), where \( b = 1 \) and \( h = 5.375 \).

\[ > 12.94 \]

<ML> MAIN MENU / GEOMETRIC PROPERTIES / 3D / ELASTIC BEAM / I_{yy} \n
The second moment of the area about the local y-axis (I_{yy}) is calculated as \( I_{yy} = (b)(h^3)/12 \), where \( b = 5.375 \) and \( h = 1 \).

\[ > 0.4479 \]

The local X-axis is defined as being parallel to the global Z-axis. So this vector is \((0,0,1)\).

<ML> MAIN MENU / GEOMETRIC PROPERTIES / 3D / ELASTIC BEAM / X \n
\[ > 0 \]

<ML> MAIN MENU / GEOMETRIC PROPERTIES / 3D / ELASTIC BEAM / Y \n
\[ > 0 \]

<ML> MAIN MENU / GEOMETRIC PROPERTIES / 3D / ELASTIC BEAM / Z \n
\[ > 1 \]

<ML> MAIN MENU / GEOMETRIC PROPERTIES / 3D / ELASTIC BEAM / OK

6a3. Apply the geometric property to element one.

<ML> MAIN MENU / GEOMETRIC PROPERTIES / 3D / ELEMENTS ADD <ML> on element 1 (on the left side of the model).

<MR> or END LIST

6b. Specify cross-sectional area for element two.

6b1. Set up a new geometric property set.

<ML> MAIN MENU / GEOMETRIC PROPERTIES <ML> MAIN MENU / GEOMETRIC PROPERTIES / NEW
At the command line, enter a name for this geometric property set.

> X2

**6b2. Define the geometric properties.**

The cross-sectional area of element two is taken as the cross-sectional area of the bar at the geometric centroid of the element (i.e., at $x=7.5$).

> 2.375

The second moment of the area about the local $x$-axis ($I_{xx}$) is calculated as $I_{xx} = \frac{b(h^3)}{12}$, where $b = 1$ and $h = 2.375$.

> 1.116

The second moment of the area about the local $y$-axis ($I_{yy}$) is calculated as $I_{yy} = \frac{b(h^3)}{12}$, where $b = 2.375$ and $h = 1$.

> 0.1979

The local $X$-axis is defined as being parallel to the global $Z$-axis. So this vector is $(0,0,1)$.
6b3. Apply the geometric property to element two.

<ML> MAIN MENU / GEOMETRIC PROPERTIES / 3D / ELEMENTS ADD
<ML> on element 2 (on the right side of the model).
<MR> or END LIST

6c. Display all geometric properties for verification.

<ML> MAIN MENU / GEOMETRIC PROPERTIES / ID GEOMETRIES
After verifying that geometric properties have been applied properly, turn off the geometric property ID's by repeating the last command.

6d. Return to the MAIN menu.

<ML> MAIN MENU / GEOMETRIC PROPERTIES / MAIN

7. Prepare the loadcase by specifying that a modal analysis is to be performed.

<ML> MAIN MENU / LOADCASES
<ML> MAIN MENU / LOADCASES / MECHANICAL
<ML> MAIN MENU / LOADCASES / MECHANICAL / DYNAMIC MODAL

Several options are available for performing the modal analysis. The default values will be used in the current analysis.

<ML> MAIN MENU / LOADCASES / MECHANICAL / DYNAMIC MODAL / OK
<ML> MAIN MENU / LOADCASES / MECHANICAL / MAIN

8. Prepare the job for execution.

8a. Specify the analysis class and select loadcases.

<ML> MAIN MENU / JOBS
<ML> MAIN MENU / JOBS / MECHANICAL
<ML> MAIN MENU / JOBS / MECHANICAL / lcase1
8b. Select the analysis dimension.

<ML> MAIN MENU / JOBS / MECHANICAL / 3D
<ML> MAIN MENU / JOBS / MECHANICAL / OK

8c. Select the element to use in the analysis.

<ML> MAIN MENU / JOBS / ELEMENT TYPES
<ML> MAIN MENU / JOBS / ELEMENT TYPES / MECHANICAL
<ML> MAIN MENU / JOBS / ELEMENT TYPES / MECHANICAL / 3D TRUSS/BEAM

Select element number 52, a two-noded line thin elastic beam element.

<ML> MAIN MENU / JOBS / ELEMENT TYPES / MECHANICAL / 3D TRUSS/BEAM / 52
<ML> MAIN MENU / JOBS / ELEMENT TYPES / MECHANICAL / 3D TRUSS/BEAM / OK

8d. Apply the element selection to all elements.

Since the element type is being applied to all elements in the model, the simplest way to select the elements is to use the ALL EXISTING option.

<ML> ALL: EXIST.

8e. Display all element types for verification.

<ML> MAIN MENU / JOBS / ELEMENT TYPES / ID TYPES

After verifying that element types have been applied properly, turn off the element type ID's by repeating the last command.

<ML> MAIN MENU / JOBS / ELEMENT TYPES / RETURN

8f. SAVE THE MODEL!

<ML> STATIC MENU / FILES
<ML> STATIC MENU / FILES / SAVE AS

In the box to the right side of the SELECTION heading, type in the name of the file that you want to create. The name should be of the form FILENAME.mud, where FILENAME is a name that you choose. Note that you do not have to enter the extension " .mud".
8g. Execute the analysis.

<ML> MAIN MENU / JOBS / RUN
<ML> MAIN MENU / JOBS / RUN / SUBMIT 1

8h. Monitor the status of the job.

<ML> MAIN MENU / JOBS / RUN / MONITOR

When the job has completed, the STATUS will read: Complete. A successful run will have an EXIT NUMBER of 3004. Any other exit number indicates that an error occurred during the analysis, probably due to an error in the model.

<ML> MAIN MENU / JOBS / RUN / OK
<ML> MAIN MENU / JOBS / RETURN

9. Postprocess the results.

9a. Open the results file and display the results.

<ML> MAIN MENU / RESULTS
<ML> MAIN MENU / RESULTS / OPEN DEFAULT
<ML> MAIN MENU / RESULTS / NEXT
<ML> MAIN MENU / RESULTS / BEAM CONTOURS
<ML> MAIN MENU / RESULTS / DEF ONLY
<ML> MAIN MENU / RESULTS / SETTINGS
<ML> MAIN MENU / RESULTS / SETTINGS / RANGE AUTOMATIC
<ML> MAIN MENU / RESULTS / SETTINGS / RETURN

A beam contour plot of the X-displacement should appear on a scaled deformed mesh plot that describes the shape associated with the first modal frequency.

Additional frequencies and mode shapes can be viewed by selecting NEXT, which displays the next increment. In the current modal analysis, each increment corresponds to a frequency and mode shape. The frequency is displayed in the upper left corner of the screen, and the mode shape is displayed as the deformed shape of the beam.

<ML> MAIN MENU / RESULTS / NEXT INC

9b. Display a different output variable.

<ML> MAIN MENU / RESULTS / SCALAR
<ML> MAIN MENU / RESULTS / SCALAR / Displacement Y
A beam contour plot of the Y-component of displacement should appear.

9c. Display nodal values of the output variable.

It is difficult to read the values when the entire model is displayed. To view the nodal values, zoom in on the region of interest using the zoom box on the static menu. To view the entire model again, use the FILL command on the static menu.

10. REPEAT THE ABOVE PROCEDURE FOR MESHES OF FOUR ELEMENTS AND SIX ELEMENTS.
MODAL ANALYSIS OF A TAPERED BAR -- using plane stress elements

1. Add points to define geometry.

1a. Add points.

   <ML> MAIN MENU / MESH GENERATION
   <ML> MAIN MENU / MESH GENERATION / PTS ADD

   Enter the coordinates at the command line, one point per line with a space separating each coordinate.

   > 0.0  4.0  0.0
   > 5.0  1.75 0.0
   > 10.0  1.0  0.0
   > 0.0 -4.0  0.0
   > 5.0 -1.75 0.0
   > 10.0 -1.0  0.0

   The points may not appear in the Graphics window because Mentat does not yet know the size of the model being built. When the FILL command in the static menu is executed, Mentat calculates a bounding box for the model and fits the model inside the Graphics window.

   <ML> STATIC MENU / FILL

   The points should now be visible in the Graphics window.

1b. Display point labels.

   <ML> STATIC MENU / PLOT
   <ML> STATIC MENU / PLOT / POINTS SETTINGS
   <ML> STATIC MENU / PLOT / POINTS SETTINGS / LABELS
   <ML> STATIC MENU / PLOT / POINTS SETTINGS / LABELS / REDRAW

1c. Return to MESH GENERATION menu.

   <MR> or <ML> RETURN
The result of this step is shown in Figure 6.7.

If the steps above were not followed precisely (e.g., if the points were entered in an order different than the order in which they appear in the above list), then the point labels will differ from those shown in Figure 6.7. These labels are simply used as identifiers in the following step, and do not affect the model. As long as the correct coordinates were entered, do not worry if the labels are not exactly as shown in Figure 6.7. Just keep track of the differences between the labels so that the appropriate procedures will be followed in the steps below.

2. Add lines that will be used to generate a ruled surface.

2a. Select CURVE TYPE.
In the MESH GENERATION menu, the currently selected type of curve that can be generated is displayed to the immediate right of the CURVE TYPE button. Confirm that the curve type is: INTERPLOATE. If true, then proceed to step 2b. If the curve type is not INTERPLOATE (or if you are not sure what is the selected curve type), then change the curve type as follows:

<ML> MAIN MENU / MESH GENERATION / CURVE TYPE
<ML> MAIN MENU / MESH GENERATION / CURVE TYPE / INTERPOLATE
<ML> MAIN MENU / MESH GENERATION / CURVE TYPE / RETURN

2b. Add an interpolated curve to create the upper boundary of the bar.

<ML> MAIN MENU / MESH GENERATION / CRVS ADD
<ML> to select point 1.
<ML> to select point 2.
<ML> to select point 3.
<MR> or END LIST

Note: The curve that appears on the screen looks like a polyline, but the curve shape that is stored internally is a mathematically-defined smooth quadratic curve. This will be confirmed later when the mesh is developed.

2c. Add an interpolated curve to create the lower boundary of the bar.

<ML> MAIN MENU / MESH GENERATION / CRVS ADD
<ML> to select point 4.
<ML> to select point 5.
<ML> to select point 6.
<MR> or END LIST

2d. Turn off point labels.

<ML> STATIC MENU / PLOT
<ML> STATIC MENU / PLOT / POINTS SETTINGS
<ML> STATIC MENU / PLOT / POINTS SETTINGS / LABELS
<ML> STATIC MENU / PLOT / POINTS SETTINGS / LABELS / REDRAW

2e. Turn on curve labels.

<ML> STATIC MENU / PLOT / CURVES SETTINGS
<ML> STATIC MENU / PLOT / CURVES SETTINGS / LABELS
2f. Return to MESH GENERATION menu.

<MR> or <ML> RETURN

The result of this step is shown in Figure 6.8.

3. Create a ruled surface.

3a. Change the SURFACE TYPE to RULED:

<ML> MAIN MENU / MESH GENERATION / SURFACE TYPE
<ML> MAIN MENU / MESH GENERATION / SURFACE TYPE / RULED
<ML> MAIN MENU / MESH GENERATION / SURFACE TYPE / RETURN

3b. Create the ruled surface.

<ML> MAIN MENU / MESH GENERATION / SRFS ADD

<ML> to select curve 2 and then curve 1 to create a ruled surface from curve 2 to curve 1.
3c. Turn off curve labels.

<ML> STATIC MENU / PLOT
<ML> STATIC MENU / PLOT / CURVES SETTINGS
<ML> STATIC MENU / PLOT / CURVES SETTINGS / LABELS
<ML> STATIC MENU / PLOT / CURVES SETTINGS / LABELS / REDRAW

3d. Return to MESH GENERATION menu.

<MR> or <ML> RETURN

The result of this step is shown in Figure 6.9.

Figure 6.9

4. Mesh the ruled surface using the CONVERT option.

4a. Mesh surface 1.

<ML> MAIN MENU / MESH GENERATION / CONVERT

4b. Select the mesh divisions.
4b. Select the mesh bias factors.

Enter the mesh bias factors at the command line, with a space separating each value.

> 0.0 0.0

4c. Mesh the surface.

Enter the mesh bias factors at the command line, with a space separating each value.

> 30 16

4d. Turn off surface displays.

4e. Turn off point displays.

4f. Turn off curve displays.

4g. Exit the PLOT menu.

> or <RETURN

4h. Return to MESH GENERATION menu.
5. Sweep the mesh to insure that all elements are properly connected.

<ML> MAIN MENU / MESH GENERATION / SWEEP
<ML> MAIN MENU / MESH GENERATION / SWEEP / ALL

Note: Duplicate geometrical and mesh entities will be deleted so that proper mesh connectivity is achieved.

Return to MESH GENERATION menu.

<MR> or <ML> RETURN

6. Check for upside down elements.

<ML> MAIN MENU / MESH GENERATION / CHECK
<ML> MAIN MENU / MESH GENERATION / CHECK / UPSIDE DOWN

Note: All elements should be numbered locally in a counterclockwise direction. Those elements numbered locally in a clockwise fashion are defined as upside down, and are highlighted when the above command is issued. These elements should be flipped by executing the FLIP ELEMENTS command.

If the procedure has been followed accurately to this point, the number of upside down elements should be zero. If so, then proceed to step 7. If not, then do the following to flip the elements.

<ML> MAIN MENU / MESH GENERATION / CHECK / FLIP ELEMENTS

Note: The upside down elements are already selected.

<ML> MAIN MENU / MESH GENERATION / CHECK / ALL: SELECT.

Note: Verify that all elements are now oriented correctly.

<ML> MAIN MENU / MESH GENERATION / CHECK / UPSIDE DOWN

Return to the MAIN menu.

<ML> MAIN MENU / MESH GENERATION / CHECK / MAIN

The result of this step is shown in Figure 6.10.
7. Add boundary conditions.

7a. Specify the constraint condition (zero horizontal and vertical displacement) on the left edge. Note that plane stress elements, as used in this example, have only two degrees of freedom per node – translation in the x- and y-directions. There is no out of plane movement.

7a1. Set up a new boundary condition set.

<ML> MAIN MENU / BOUNDARY CONDITIONS
<ML> MAIN MENU / BOUNDARY CONDITIONS / MECHANICAL
<ML> MAIN MENU / BOUNDARY CONDITIONS / MECHANICAL / NEW
<ML> MAIN MENU / BOUNDARY CONDITIONS / MECHANICAL / NAME

At the command line, enter a name for this boundary condition set.

> FixedEdge
7a2. Define the nature of the boundary condition.

<ML> MAIN MENU / BOUNDARY CONDITIONS / MECHANICAL / FIXED DISPLACEMENT
<ML> MAIN MENU / BOUNDARY CONDITIONS / MECHANICAL / FIXED DISPLACEMENT / DISPLACEMENT X
<ML> MAIN MENU / BOUNDARY CONDITIONS / MECHANICAL / FIXED DISPLACEMENT / DISPLACEMENT Y

The small box to the immediate left of the button for DISPLACEMENT X and DISPLACEMENT Y should now be highlighted.

<ML> MAIN MENU / BOUNDARY CONDITIONS / MECHANICAL / FIXED DISPLACEMENT / OK

7a3. Apply the condition to nodes along the left edge.

<ML> MAIN MENU / BOUNDARY CONDITIONS / MECHANICAL / NODES ADD

Box pick the nodes lying on the left edge of the model, or <ML> to select each node individually.

<MR> or END LIST

The result of this step is shown in Figure 6.11.
7b. Display all boundary conditions for verification.

<ML> MAIN MENU / BOUNDARY CONDITIONS / ID BOUNDARY CONDS

After verifying that boundary conditions have been applied properly, turn off the boundary condition ID's by repeating the last command.

7c. Return to the MAIN menu.

<ML> MAIN MENU / BOUNDARY CONDITIONS / MAIN

8. Specify the material properties of each element.

8a. Set up a new material property set.

<ML> MAIN MENU / MATERIAL PROPERTIES
<ML> MAIN MENU / MATERIAL PROPERTIES / NEW
<ML> MAIN MENU / MATERIAL PROPERTIES / NAME

At the command line, enter a name for this material property set.

> Steel

8b. Define the nature of the material.

<ML> MAIN MENU / MATERIAL PROPERTIES / ISOTROPIC
<ML> MAIN MENU / MATERIAL PROPERTIES / ISOTROPIC / YOUNG'S MODULUS

> 29.0e6

<ML> MAIN MENU / MATERIAL PROPERTIES / ISOTROPIC / POISSON'S RATIO

> 0.30

<ML> MAIN MENU / MATERIAL PROPERTIES / ISOTROPIC / MASS DENSITY

> 0.0007356

Note: Only Young's modulus, Poisson’s ratio and mass density need to be specified for this problem. The mass density is input in units of lbm/in^3. The specific weight is provided in units of lbf/in', which must be converted to lbm/in^3 by dividing by a factor of (32.174)*(12).
8c. Apply the material properties to all elements.

Since the properties are being applied to all elements in the model, the simplest way to select the elements is to use the ALL EXISTING option.

8d. Display all material properties for verification.

After verifying that material properties have been applied properly, turn off the material property ID's by repeating the last command.

8e. Return to the MAIN menu.

9. Specify the thickness of each element.

9a. Set up a new geometric property set.

At the command line, enter a name for this geometric property set.

> Thickness

9b. Define the nature of the geometric property.
9c. Apply the geometric property to all elements.

<ML> MAIN MENU / GEOMETRIC PROPERTIES / PLANAR / ELEMENTS ADD

Since the property is being applied to all elements in the model, the simplest way to select the elements is to use the ALL EXISTING option.

<ML> ALL: EXIST.

9d. Display all geometric properties for verification.

<ML> MAIN MENU / GEOMETRIC PROPERTIES / ID GEOMETRIES

After verifying that geometric properties have been applied properly, turn off the geometric property ID’s by repeating the last command.

9e. Return to the MAIN menu.

<ML> MAIN MENU / GEOMETRIC PROPERTIES / MAIN

10. Prepare the loadcase.

<ML> MAIN MENU / LOADCASES
<ML> MAIN MENU / LOADCASES / MECHANICAL
<ML> MAIN MENU / LOADCASES / MECHANICAL / DYNAMIC MODAL

Several options are available for performing the modal analysis. The default values will be used in the current analysis.

<ML> MAIN MENU / LOADCASES / MECHANICAL / DYNAMIC MODAL / OK
<ML> MAIN MENU / LOADCASES / MECHANICAL / MAIN

11. Prepare the job for execution.

11a. Specify the analysis class and select loadcases.

<ML> MAIN MENU / JOBS
<ML> MAIN MENU / JOBS / MECHANICAL
<ML> MAIN MENU / JOBS / MECHANICAL / lcase1

11b. Select the analysis dimension.

<ML> MAIN MENU / JOBS / MECHANICAL / PLANE STRESS
<ML> MAIN MENU / JOBS / MECHANICAL / OK

11c. Select the element to use in the analysis.
Select element number 3, a fully-integrated, four-noded quadrilateral.

11d. Apply the element selection to all elements.

Since the element type is being applied to all elements in the model, the simplest way to select the elements is to use the ALL EXISTING option.

11e. Display all element types for verification.

After verifying that element types have been applied properly, turn off the element type ID's by repeating the last command.

11f. SAVE THE MODEL!

In the box to the right side of the SELECTION heading, type in the name of the file that you want to create. The name should be of the form FILENAME.mud, where FILENAME is a name that you choose.

11g. Execute the analysis.

11h. Monitor the status of the job.

<ML> MAIN MENU / JOBS / RUN / MONITOR

When the job has completed, the STATUS will read: Complete. A successful run will have an EXIT NUMBER of 3004. Any other exit number indicates that an error occurred during the analysis, probably due to an error in the model.

<ML> MAIN MENU / JOBS / RUN / OK
<ML> MAIN MENU / JOBS / RETURN

9. Postprocess the results.

9a. Open the results file and display the results.

<ML> MAIN MENU / RESULTS
<ML> MAIN MENU / RESULTS / OPEN DEFAULT
<ML> MAIN MENU / RESULTS / NEXT
<ML> MAIN MENU / RESULTS / BEAM CONTOURS
<ML> MAIN MENU / RESULTS / DEF ONLY
<ML> MAIN MENU / RESULTS / SETTINGS
<ML> MAIN MENU / RESULTS / SETTINGS / RANGE AUTOMATIC
<ML> MAIN MENU / RESULTS / SETTINGS / RETURN

A beam contour plot of the X-displacement should appear on a scaled deformed mesh plot that describes the shape associated with the first modal frequency.

Additional frequencies and mode shapes can be viewed by selecting NEXT, which displays the next increment. In the current modal analysis, each increment corresponds to a frequency and mode shape. The frequency is displayed in the upper left corner of the screen, and the mode shape is displayed as the deformed shape of the beam.

<ML> MAIN MENU / RESULTS / NEXT INC

9b. Display a different output variable.

<ML> MAIN MENU / RESULTS / SCALAR
<ML> MAIN MENU / RESULTS / SCALAR / Displacement Y
<ML> MAIN MENU / RESULTS / SCALAR / OK

A contour plot of the Y-component of displacement should appear.

9c. Display nodal values of the output variable.

<ML> MAIN MENU / RESULTS / NUMERICS
It is difficult to read the values when the entire model is displayed. To view the nodal values, zoom in on the region of interest using the zoom box on the static menu. To view the entire model again, use the FILL command on the static menu.