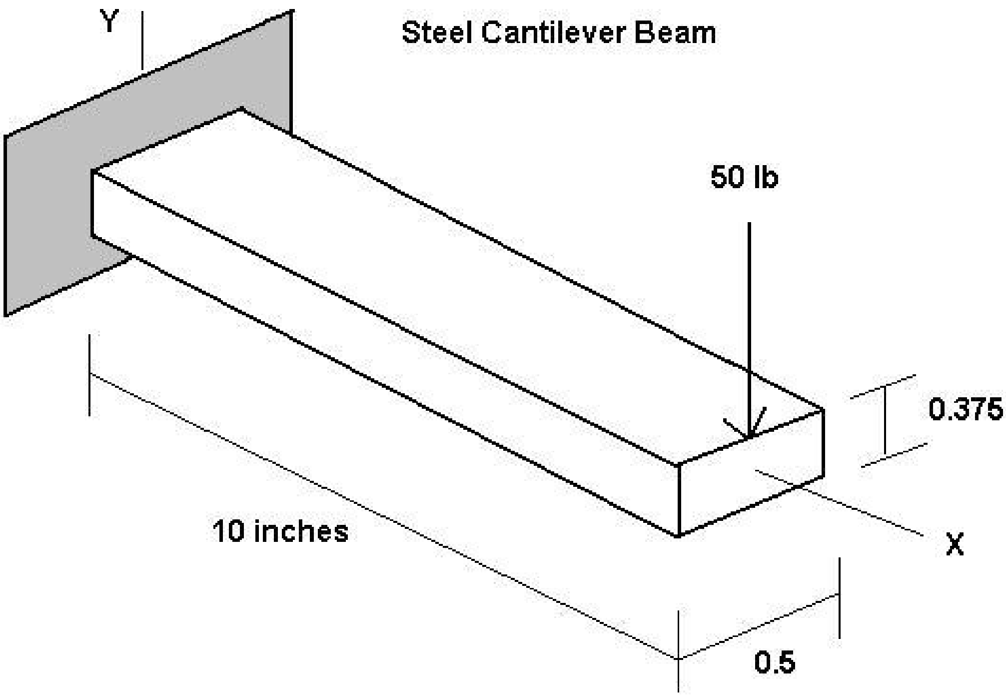
Q1. Using FEM, determine the end deflection of a steel cantilever beam shown in Figure 1. Assume the modulus of elasticity is $E = 29\, e^6$ psi.



The stiffness matrix for the beam element is : -

$$\mathbf{k} = \frac{EI}{L^3}\begin{bmatrix} 12 & 6L & -12 & 6L \\ 6L & 4L^2 & -6L & 2L^2 \\ -12 & -6L & 12 & -6L \\ 6L & 2L^2 & -6L & 4L^2 \end{bmatrix}$$

(columns: v_1, q_1, v_2, q_2)

$$\mathbf{I} = \frac{bh^3}{12} = \frac{(0.5 \times (0.375)^3)}{12} = 0.002197265$$

We put the boundary conditions : -

$$v_1 = q_1 = 0$$
$$M_2 = 0$$
$$F_{2Y} = -P$$

We now deleting v_1, q_1 (rows and columns) to get this following equation : -

$$\frac{EI}{L^3}\begin{bmatrix} 12 & -6L \\ -6L & 4L^2 \end{bmatrix}\begin{Bmatrix} v_2 \\ q_2 \end{Bmatrix} = \begin{Bmatrix} -P \\ 0 \end{Bmatrix}$$

$$\frac{EI}{L^3}\begin{bmatrix} 12 & -6L \\ -6L & 4L^2 \end{bmatrix}\begin{Bmatrix} v_2 \\ q_2 \end{Bmatrix} = \begin{Bmatrix} -50 \\ 0 \end{Bmatrix}$$

We solving this equation to get the deflection and the rotation at node 2 : -

$$\begin{Bmatrix} v_2 \\ q_2 \end{Bmatrix} = \begin{Bmatrix} -0.261558 \text{ in} \\ -0.0392337 \text{ rad} \end{Bmatrix}$$

To find the reaction forces and moment we must find it from the reactions equation : -

$$\begin{matrix} & & & v_2 & q_2 & & \\ \begin{Bmatrix} F_{1Y} \\ M_1 \end{Bmatrix} & = & \dfrac{EI}{L^3} & \begin{bmatrix} -12 & 6L \\ -6L & 2L^2 \end{bmatrix} & & \begin{Bmatrix} v_2 \\ q_2 \end{Bmatrix} \end{matrix}$$

The reaction force and moment is : -

$$\begin{Bmatrix} F_{1Y} \\ M_1 \end{Bmatrix} = \begin{Bmatrix} 50 \quad \text{Ib} \\ 500 \quad \text{Ib} \cdot \text{in} \end{Bmatrix}$$

Q2. For the beam shown in Figure 2, find the following:

a) deflections and rotations at node 2 and node 3.
b) the reactions.

Assume that:

$P = 50\text{ kN},\ k = 200\text{ kN/m},\ L = 3\text{ m},\ E = 210\text{ GPa},\ I = 2\times10^{-4}\text{ m}^4.$

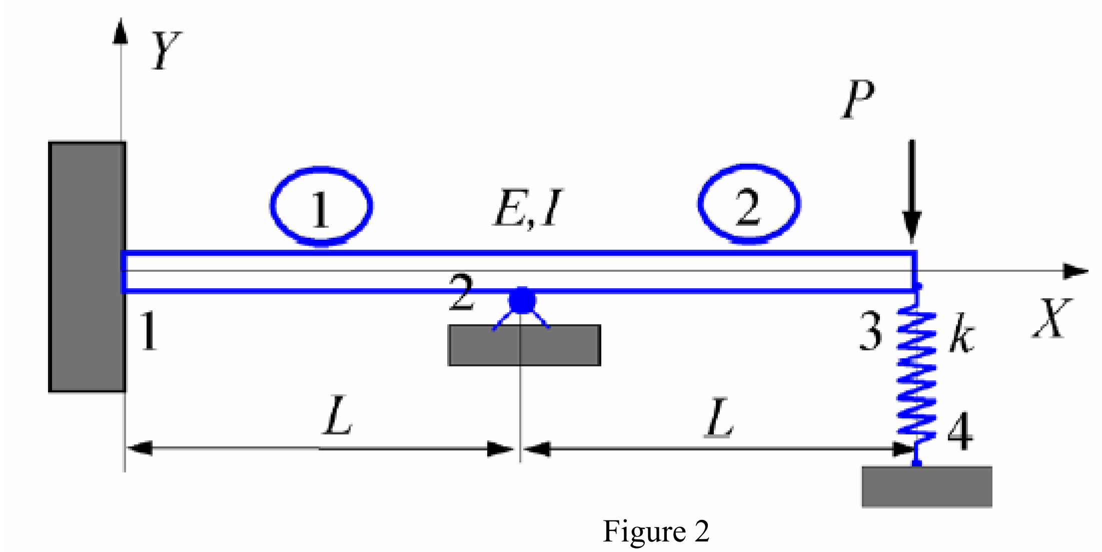


Figure 2

The stiffness matrix for the beam element is : -

$$\begin{matrix} & v_1 & q_1 & v_2 & q_2 \end{matrix}$$

$$\mathbf{k}_1 = \frac{EI}{L^3}\begin{bmatrix} 12 & 6L & -12 & 6L \\ 6L & 4L^2 & -6L & 2L^2 \\ -12 & -6L & 12 & -6L \\ 6L & 2L^2 & -6L & 4L^2 \end{bmatrix}$$

$$\begin{matrix} & v_2 & q_2 & v_3 & q_3 \end{matrix}$$

$$\mathbf{k}_2 = \frac{EI}{L^3}\begin{bmatrix} 12 & 6L & -12 & 6L \\ 6L & 4L^2 & -6L & 2L^2 \\ -12 & -6L & 12 & -6L \\ 6L & 2L^2 & -6L & 4L^2 \end{bmatrix}$$

The stiffness matrix for the spring is : -

$$\begin{matrix} & v_3 & v_4 \end{matrix}$$

$$\mathbf{k}_s = \begin{bmatrix} k & -k \\ -k & k \end{bmatrix}$$

After that we inlarge all the stiffness matrices and added together to take this global stiffness matrix : -

$$
\begin{matrix} v_1 & q_1 & v_2 & q_2 & v_3 & q_3 & v_4 \end{matrix}
$$

$$
\mathbf{k} = \frac{EI}{L^3}\begin{bmatrix} 12 & 6L & -12 & 6L & 0 & 0 & 0 \\ 6L & 4L^2 & -6L & 2L^2 & 0 & 0 & 0 \\ -12 & 6L & 24 & 0 & -12 & 6L & 0 \\ 6L & 2L^2 & 0 & 8L^2 & -6L & 2L^2 & 0 \\ 0 & 0 & -12 & -6L & 12+k' & -6L & -k' \\ 0 & 0 & 6L & 2L^2 & -6L & 4L^2 & 0 \\ 0 & 0 & 0 & 0 & -k' & 0 & k' \end{bmatrix}
$$

The equation will be : -

$$
\begin{matrix} v_1 & q_1 & v_2 & q_2 & v_3 & q_3 & v_4 \end{matrix}
$$

$$
\frac{EI}{L^3}\begin{bmatrix} 12 & 6L & -12 & 6L & 0 & 0 & 0 \\ 6L & 4L^2 & -6L & 2L^2 & 0 & 0 & 0 \\ -12 & 6L & 24 & 0 & -12 & 6L & 0 \\ 6L & 2L^2 & 0 & 8L^2 & -6L & 2L^2 & 0 \\ 0 & 0 & -12 & -6L & 12+k' & -6L & -k' \\ 0 & 0 & 6L & 2L^2 & -6L & 4L^2 & 0 \\ 0 & 0 & 0 & 0 & -k' & 0 & k' \end{bmatrix}\begin{Bmatrix} v_1 \\ q_1 \\ v_2 \\ q_2 \\ v_3 \\ q_3 \\ v_4 \end{Bmatrix} = \begin{Bmatrix} F_{1Y} \\ M_1 \\ F_{2Y} \\ M_2 \\ F_{3Y} \\ M_3 \\ F_{4Y} \end{Bmatrix}
$$

Which : -

$$
k' = \frac{L^3}{EI}k
$$

We put the boundary conditions : -

$$v_1 = q_1 = v_2 = v_4 = 0$$
$$M_2 = M_3 = 0$$
$$F_{3Y} = -P$$

We now deleting v_1, q_1, v_2, v_4 (rows and columns) to get this following equation : -

$$\frac{EI}{L^3}\begin{bmatrix} 8L^2 & -6L & 2L^2 \\ -6L & 12+k' & -6L \\ 2L^2 & -6L & 4L^2 \end{bmatrix}\begin{Bmatrix} q_2 \\ v_3 \\ q_3 \end{Bmatrix} = \begin{Bmatrix} 0 \\ -P \\ 0 \end{Bmatrix}$$

We solving this equation to get the rotation at node 2 and the deflection and the rotation at node 3 : -

$$\begin{Bmatrix} q_2 \\ v_3 \\ q_3 \end{Bmatrix} = \begin{Bmatrix} -0.002492 \text{ rad} \\ -0.01744 \text{ m} \\ -0.007475 \text{ rad} \end{Bmatrix}$$

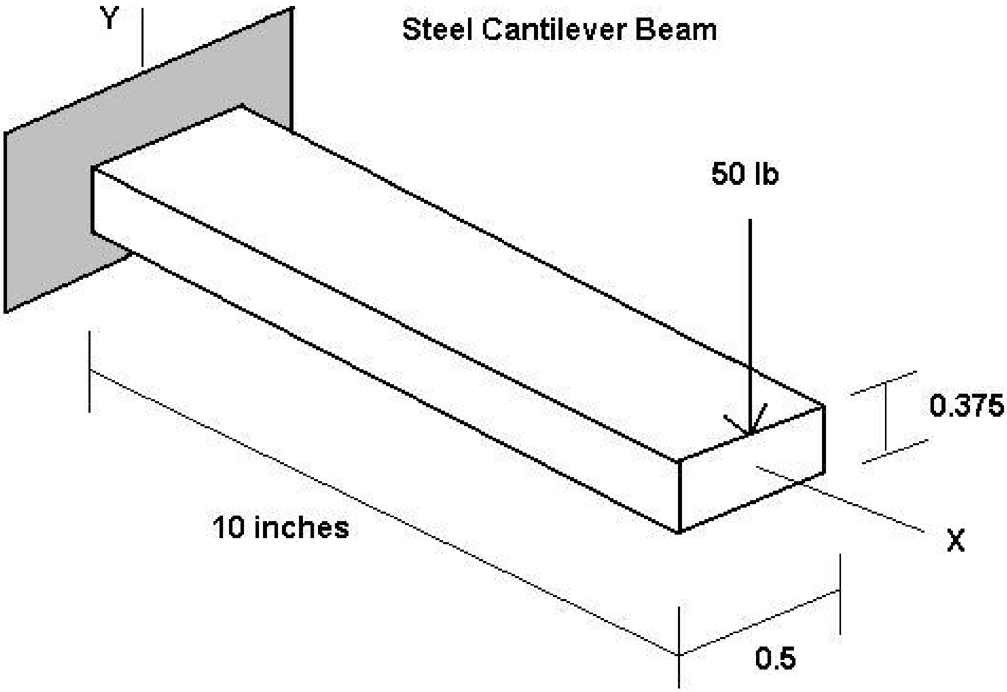
To find the reaction forces and moment we must find it from the reactions equation : -

$$\begin{array}{c} \\ \left\{\begin{array}{c} F_{1Y} \\ M_1 \\ F_{2Y} \\ F_{4Y} \end{array}\right\} \end{array} \begin{array}{c} \\ = \frac{EI}{L^3} \end{array} \begin{array}{c} \begin{array}{ccc} q_2 & v_3 & q_3 \end{array} \\ \left[\begin{array}{ccc} 6L & 0 & 0 \\ 2L^2 & 0 & 0 \\ 0 & -12 & 6L \\ 0 & -k' & 0 \end{array}\right] \end{array} \begin{array}{c} \\ \left\{\begin{array}{c} q_2 \\ v_3 \\ q_3 \end{array}\right\} \end{array}$$

The reaction forces and moment is : -

$$\left\{\begin{array}{c} F_{1Y} \\ M_1 \\ F_{2Y} \\ F_{4Y} \end{array}\right\} = \left\{\begin{array}{c} -69.78 \text{ kN} \\ -69.78 \text{ kN}\cdot\text{m} \\ 116.2 \text{ kN} \\ 3.488 \text{ kN} \end{array}\right\}$$

Q3. Using ANSYS, determine the end deflection and root bending stress of a steel cantilever beam modeled as a 2D problem. Assume the modulus of elasticity of steel is $E = 29\ e^6$ psi.



1.Enter the ANSYS program by using the launcher : -
Click ANSYS from the launcher menu.
Type a job name in the Initial jobname like exam1.
Specify the working directory
Pick Run to apply the information

2.Specify a title for the problem :-
ANSYS Utility Menu > File
Change Title > Enter new title [banakhar] > OK

3.Preferences
structural > ok

4. Define the element type (Triangle 6 node 2)
structural solid, plate thickness, and material properties.
ANSYS Main Menu
Preprocessor > Element Type > Add/Edit/Delete..Add... > Structural Solid
> [Triangle 6 node 2] > OK
Options... > PLANE2 element type options > k3 [Plane strs w/thk] > OK > Close
Preprocessor > real Constants...> Add/Edit/Delete>Add...> OK
>RealConstant for PLANE2>THK [0.5] > OK > Close
Preprocessor > Material Props > Costant - Isotropic >EX [29e6] > OK
ANSYS Toolbar > SAVE_DB

5.Preprocessing

1. Begin the solid model by dimensions.

ANSYS Main Menu
Preprocessor > -Modeling-Create -Areas-Rectangle > By dimensions

■□□[X1,X2 = (0, 10)]
■□□[Y1,Y2 = (0,0.375)]
> OK

6. Specify an element size and mesh the solid model.
ANSYS Main Menu
Preprocessor >- Meshing-Size Control-Manual Size > -Global- Size ...> SIZE [0.5]
Preprocessor > - Meshing-Mesh - Areas - Free > Pick All

Solution

1. Apply displacement constraints at the left side

ANSYS Main Menu

Solution > -Loads- Apply-Structural-Displacements > On nodes

[The three nodes on the left vertical side.]

Apply U, ROT, on KPs > Lab2 [All DOF] > KEXPND > Yes > OK

Solution > -Loads- Apply-Structural-Force/Moment > -On nodes

[The upper right cornor] > OK > [FY] [-50] > OK

Solution> -Solve - Current LS > OK > Close

Postprocessing

Review the results using the general postprocessor (POST1).

We will view a deformed shape and the stress distribution.

ANSYS main Menu

General Postproc > Plot Results > Deformed Shape > KUND > Def + Undeformed > OK

General Postproc > Plot Results > -Contour Plot- Nodal Solu...
> Contour Nodal Solution Data > [Stress] > [X-direction SX] > OK

General Postproc > Plot Results > -Contour Plot- Nodal Solu...
> Contour Nodal Solution Data > [Stress] > [Y-direction SY] > OK

General Postproc > Plot Results > -Contour Plot- Nodal Solu...
> Contour Nodal Solution Data > [Stress] > [XY-shear SXY] > OK

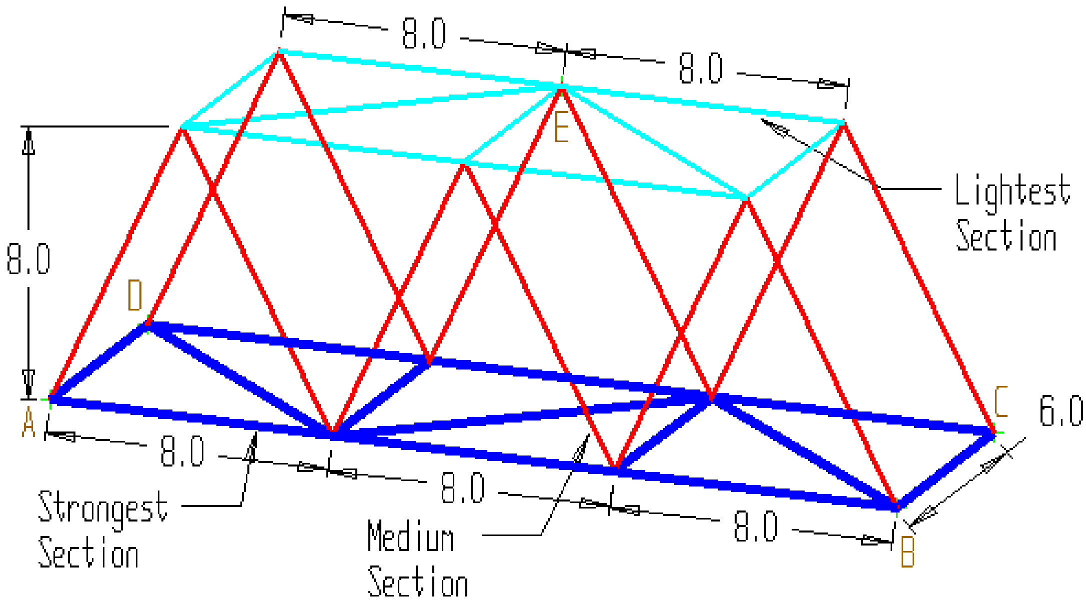
General Postproc > Plot Results > -Contour Plot- Nodal Solu...
> Contour Nodal Solution Data > [Stress] > [Von mises SEQV] > OK

General Postproc >List Results > Element solu... > stress > Components S

General Postproc >List Results > Nodal solu... > DOF solution > All DOFs DOF

General Postproc >List Results > Reaction solu... > All Items

Q4. A three-dimensional approximation of a bridge frame is shown in Figure 3.



A three-dimensional approximation of a bridge frame is shown in the figure.

All dimensions in the figure are given in **feet**. The members of this structure are wood:

Modulus	Poisson's Ratio	Density
1.4×10^6 psi	0.21	0.0266 lb_m/in^3

Three different sizes of construction timber are used in this frame. Each cross-sectional area is:

Strongest Section (4x4)	Medium Section (2x4)	Lightest Section (2x2)
12.25 in^2	5.25 in^2	2.25 in^2

Solution

Analyze this structure for a static loading condition:

Pin Constraints in all DOF (UX, UY, UZ) at the four base corners (A, B, C, D in the figure)

One lateral (or transverse) constraint along the top (at point E in the figure) which gives support to the frame in the side-to-side direction.

To simulate the wood decking load on the base, apply a 200 lb_f downward load on each of the four unconstrained pins of the base.

Then, to account for a heavy load crossing the bridge, add 1000 lb_f to each of the two unconstrained pins nearest B and C on the figure (making the total load 1200 lb_f downward on each of those two pins).

Also, include the weight of the frame structure (by using a vertical gravity loading)

Enter the ANSYS program by using the launcher

Enter a job name, say banakhar > Run

File > Change Title > [3-D Beam]

Preferences

structural > OK

Preprocessing

1. Define the element type, element real constants, and material properties. 3D elastic beam is the appropriate element for analyzing plane frames.

Preprocessor > Element Type > Add/Edit/Delete... > Add... > Structural Beam [3D spar 8] > OK > Close

Preprocessor > Real Constants >Add/Edit/Delete... > Add... > OK

Real Constant Set No. [1]: AREA [12.25] > OK

Real Constant Set No. [2]: AREA [5.25] > OK

Real Constant Set No. [3]: AREA [2.25] > OK > Close

Preprocessor > Material Props > - constant - Isotropic >EX [1.4e6] >DENS [0.0266] >PRXY [0.21] > OK

ANSYS Toolbar > SAVE_DB

Preprocessor > -Modeling- Create Nodes > In active CS

[WP = (0, 0, 0)] node 1
[WP = (8, 0, 0)] node 2
[WP = (16, 0, 0)] node 3
[WP = (24, 0, 0)] node 4
[WP = (24, 0, -6)] node 5
[WP = (16, 0, -6)] node 6
[WP = (8, 0, -6)] node 7
[WP = (0, 0, -6)] node 8
[WP = (4, 8, 0)] node 9
[WP = (12, 8, 0)] node 10
[WP = (20, 8, 0)] node 11
[WP = (20, 8, -6)] node 12
[WP = (12, 8, -6)] node 13
[WP = (4, 8, -6)] node 14

PlotCtrls > Numbering > Plot Numbering Controls > NODE > On > OK
PlotCtrls > Pan-Zoom-Rotate > Iso > Close
Utility menu > list > nodes ...
ANSYS Toolbar > SAVE_DB
Preprocessor > -Modeling- Create > Elements
> Element Attributes > [REAL] [1] >OK
Preprocessor > -Modeling- Create > Elements > -Auto Numbered- Thru Nodes

[Node 1 and 2] > OK
[Node 2 and 3] > OK
[Node 3 and 4] > OK
[Node 4 and 5] > OK
[Node 5 and 6] > OK
[Node 6 and 7] > OK
[Node 7 and 8] > OK
[Node 8 and 1] > OK
[Node 2 and 8] > OK
[Node 2 and 7] > OK
[Node 2 and 6] > OK
[Node 3 and 6] > OK
[Node 4 and 6] > OK

Preprocessor > -Modeling- Create >Elements
> Element Attributes > [REAL] [2] > OK
Preprocessor > -Modeling- Create >Elements > -Auto Numbered- Thru Nodes

[Node 1 and 9] > OK
[Node 9 and 2] > OK
[Node 2 and 10] > OK
[Node 10 and 3] > OK
[Node 3 and 11] > OK
[Node 11 and 4] > OK
[Node 8 and 14] > OK
[Node 14 and 7] > OK
[Node 7 and 13] > OK
[Node 13 and 6] > OK
[Node 6 and 12] > OK
[Node 12 and 5] > OK

Preprocessor > -Modeling- Create >Elements
> Element Attributes > [REAL] [3] > OK
Preprocessor > -Modeling- Create >Elements > -Auto Numbered- Thru Nodes

[Node 9 and 10] > OK
[Node 10 and 11] > OK
[Node 11 and 12] > OK
[Node 12 and 13] > OK
[Node 13 and 14] > OK
[Node 14 and 9] > OK
[Node 9 and 13] > OK
[Node 10 and 13] > OK
[Node 11 and 13] > OK

Solution

Solution > -Loads > Apply > -Structural- Displacement > On Nodes
Pick [Node 1] > OK > Apply U.ROT on Nodes > [ALL DOF] > OK
Pick [Node 4] > OK > Apply U.ROT on Nodes > [ALL DOF] > OK
Pick [Node 5] > OK > Apply U.ROT on Nodes > [ALL DOF] > OK
Pick [Node 8] > OK > Apply U.ROT on Nodes > [ALL DOF] > OK
Pick [Node 13] > OK > Apply U.ROT on Nodes > [UZ] > OK
Apply > -Structural- Force / Moment >
On Nodes > [Node 1] [Node 8] > OK > Direction of force/mom [FY] >
Value [-200] > OK
On Nodes > [Node 4] [Node 5] > OK > Direction of force/mom [FY] >
Value [-1200] > OK
Solution > -Loads > Apply > -Structural- Gravity ...
ACELX [0]
ACELY [-8]
ACELZ [0]
ANSYS Toolbar > SAVE_DB
Initiate the solution
Solution > -Solve- Current LS

Postprocessing

Plot deformed mesh and bending moment

ANSYS main Menu >General Postproc > Plot Results > Deformed Shape >
KIND > Def + Undeformed

ANSYS main Menu > General Postproc > Plot Results >
Element solu ... > stress > x- direction SX
Element solu ... > stress > y - direction SY
Element solu ... > stress > z- direction SZ
Element solu ... > stress > xy - shear SXY
Element solu ... > stress > yz- shear SYZ
Element solu ... > stress > xz- shear SXZ > OK

ANSYS main Menu >General Postproc > list Results >
Element solu ... > stress > components S > OK

ANSYS main Menu >General Postproc > list Results >
Nodal solu ... > DOF Solution > All DOFs DOF > OK

ANSYS main Menu >General Postproc > Reaction solu ... > All items > OK