Lesson 5

Beams

5-1 OVERVIEW

Lesson 5 treats problems in structural analysis where the most suitable modeling choice is a beam element. Like the truss and the plate, the beam is a structural element, and like the truss element it is a line element. Its formulation is based upon the engineering study of the behavior of beams as practical structural members. The lesson discusses

- Creating text file FEM beam models.
- Modeling 2-D and 3-D problems and evaluating the results.

5-2 INTRODUCTION

When long, slender bars are subjected to planar moments they bend in such a way that maximum compression is produced at one extreme surface and maximum tension at the other. Somewhere in between the strain is zero, and the strains and stresses vary linearly from this neutral axis. Such behavior is an example of pure bending.

![Figure 5-1 Beam bending by moments.](image)

The situation is not much different when transverse forces cause bending, and we can determine the state of deformation by keeping track of the displacement of the neutral axis and its slope. Thus in finite element modeling of beams, we need to utilize slope variables in addition to the displacement variables used in previous lessons.

Figure 5-2 shows a simply supported structure composed of three beam elements carrying a point force, a concentrated moment and a distributed load. The deflected shape, displacement variables, and slope variables are also shown.
ANSYS provides the engineer with an elastic beam element called BEAM3 for two-dimensional modeling. The flexural inertia of the beam cross section provides stiffness to resist bending loadings, and its cross sectional area gives it stiffness to resist axial loads just like a truss element does.

5-3 TUTORIAL 5A – CANTILEVER BEAM

Objective: Determine the end deflection and root bending stress of a steel cantilever beam modeled as a 2-D problem.

The ANSYS 2-D beam element 'beam3' is used for modeling. The ten-inch long beam is represented by ten beam3 elements connecting 11 nodes along the global X-axis. Cantilever boundary conditions at the left end prevent axial (UX), vertical (UY) and rotation about the z axis (ROTZ) deformations.

Figure 5-2 Three-element model of beam.

Figure 5-3 Cantilever beam.
The problem is formulated using a text file incorporating a consistent set of units, pounds force and inches in this case. The computed results will have deflections in inches, slopes in radians, shear forces in pounds, moments in inch-pounds, and stresses in psi.

/FILNAM,Tutorial5A
/title, 10 Element,2D Cantilever Beam
/prep7

!List of Nodes
n, 1, 0.0, 0.0 ! Node 1 is located at (0.0, 0.0) inches
n, 2, 1.0, 0.0
n, 3, 2.0, 0.0
n, 4, 3.0, 0.0
n, 5, 4.0, 0.0
n, 6, 5.0, 0.0
n, 7, 6.0, 0.0
n, 8, 7.0, 0.0
n, 9, 8.0, 0.0
n, 10, 9.0, 0.0
n, 11, 10.0, 0.0

!Material Properties
mp, ex, 1, 3.87 ! Elastic modulus for material number 1 in psi
mp, prxy, 1, 0.3 ! Poisson’s ratio

! Real constant set 1 for a 0.5 x 0.375 rectangular xsctn beam.
! Area, Izz (flexural Inertia), height 'h' as in sigma = Mc/I, c = h/2
! A = 0.1875 sq.in., Izz = 0.0022 in^4, h = 0.375 inch

r, 1, 0.1875, 0.0022, 0.375

!List of elements and nodes they connect
en, 1, 1, 2 ! Element Number 1 connects nodes 1 & 2
en, 2, 2, 3
en, 3, 3, 4
en, 4, 4, 5
en, 5, 5, 6
en, 6, 6, 7
en, 7, 7, 8
en, 8, 8, 9
en, 9, 9, 10
en, 10, 10, 11
! Displacement Boundary Conditions

d, 1, ux, 0.0  ! Displacement at node 1 in x-dir is zero

d, 1, uy, 0.0  ! Displacement at node 1 in y-dir is zero

d, 1, rotz, 0.0  ! Rotation about z axis at node 1 is zero

! Applied Force
f, 11, fy, -50.  ! Force at node 11 in negative y-direction is 50 lbf.

/pnum, elem, 1  ! Plot element numbers
eplot  ! Plot the elements
finish

/solu  ! Select static load solution
antype, static
solve
finish

/post1

1. Start ANSYS, etc., read the input file using File > Read Input from...

Plot the deformed shape and examine the computed deflections and slopes.

2. Main Menu > General Postproc > List Results > Nodal Solution > DOF Solution > Y-Component of displacement > OK

PRINT U    NODAL SOLUTION PER NODE

***** POST1 NODAL DEGREE OF FREEDOM LISTING *****

LOAD STEP= 1 SUBSTEP= 1
TIME= 1.0000  LOAD CASE= 0

THE FOLLOWING DEGREE OF FREEDOM RESULTS ARE IN THE GLOBAL COORDINATE SYSTEM

NODE   UY
1  0.0000
2 -0.36615E-02
3 -0.14141E-01
4 -0.30682E-01
5 -0.52525E-01
6 -0.78914E-01
7 -0.10909
8 -0.14230
9 -0.17778
10 -0.21477
11 -0.25253

MAXIMUM ABSOLUTE VALUES
NODE   11
VALUE  -0.25253
3. Main Menu > General Postproc > List Results > Nodal Solution > DOF Solution > Y-Component of displacement > OK

PRINT ROT NODAL SOLUTION PER NODE

***** POST1 NODAL DEGREE OF FREEDOM LISTING *****

LOAD STEP= 1 SUBSTEP= 1
TIME= 1.0000 LOAD CASE= 0

THE FOLLOWING DEGREE OF FREEDOM RESULTS ARE IN THE GLOBAL COORDINATE SYSTEM

NODE ROTZ
1 0.0000
2 -0.71976E-02
3 -0.13636E-01
4 -0.19318E-01
5 -0.24242E-01
6 -0.28409E-01
7 -0.31818E-01
8 -0.34470E-01
9 -0.36364E-01
10 -0.37500E-01
11 -0.37879E-01

MAXIMUM ABSOLUTE VALUES
NODE 11
VALUE -0.37879E-01

The maximum deflection UY and slope ROTZ occur at node 11 at the free end as expected.

The maximum values -0.25253 inch and -0.037879 radian agree with results you can calculate from solid mechanics beam theory. Confirm these results with a simple hand calculation. To examine the computed bending stress

4. Main Menu > General Postproc > List Results > Element Solution > Line Elements Results > Element Results > OK

PRINT ELEM ELEMENT SOLUTION PER ELEMENT

***** POST1 ELEMENT SOLUTION LISTING *****

LOAD STEP 1 SUBSTEP= 1
TIME= 1.0000 LOAD CASE= 0

EL= 1 NODES= 1 2 MAT= 1
BEAM3 TEMP = 0.00 0.00 0.00 0.00
LOCATION SDIR SBYT SBYB
1 (I) 0.0000 42614. -42614.
2 (J) 0.0000 38352. -38352.
The stresses in each beam element are given in the form shown. SDIR is the direct or axial stress and SBYT and SBYB are the bending stresses (top and bottom). SBYT and SBYB are equal for a symmetric cross section beam. SMAX = SDIR + SBYT and SMIN = SDIR - SBYT; the sum and difference of the direct and bending stress components. SDIR is zero in this example because there is no applied axial force. All quantities are given at each end of the element; that is, at first named node I and at node J.

The maximum bending stress of 42,614 psi occurs in beam element 1 at the support, and the value agrees with what you would compute using elementary beam theory, Mc/I.

The displacement and stress solution for this problem is solved just as accurately with ONE 10-inch long element connecting nodes at each end of the beam. The deformed shape plot however would show a straight line connecting the two nodes because the plotting software does not use computed slope information. If examining the deformed shape is important or if the spatial mass distribution needs to be accurately represented, use several nodes along the length as described above, otherwise two nodes will do the job, but the plotted results look funny.

Axial stiffness is included in the beam3 element formulation, so we could add an axial force to the above problem and compute the axial stress and deformation as well. Note however that in the linear approach discussed here, the axial and bending deformations are uncoupled. That is, the presence of an axial stress does not influence
the bending stiffness. If the bending deformation is small, this approach is usually completely satisfactory.

![Figure 5-5 Transverse and axial loads.](image)

To illustrate this, add an axial force of 1000 lbf to the end of the beam and solve the problem again.

Add this line to the text file `f, 11, fx, 1000.`

5. Utility Menu > File > Clear & Start new > OK > Yes

6. Utility Menu > File > Read Input from ... (read in the file with the axial load added.)

List the displacements.

7. Main Menu > General Postproc > List Results > Nodal Solution > DOF Solution > Displacement vector sum > OK

```
***** POST1 NODAL DEGREE OF FREEDOM LISTING *****

LOAD STEP=    1    SUBSTEP=    1
TIME=    1.0000    LOAD CASE=    0

THE FOLLOWING DEGREE OF FREEDOM RESULTS ARE IN GLOBAL COORDINATES

<table>
<thead>
<tr>
<th>NODE</th>
<th>UX</th>
<th>UY</th>
<th>UZ</th>
<th>USUM</th>
</tr>
</thead>
<tbody>
<tr>
<td>1</td>
<td>0.0000</td>
<td>0.0000</td>
<td>0.0000</td>
<td>0.0000</td>
</tr>
<tr>
<td>2</td>
<td>0.17778E-03</td>
<td>-0.36616E-02</td>
<td>0.0000</td>
<td>0.36659E-02</td>
</tr>
<tr>
<td>3</td>
<td>0.35556E-03</td>
<td>-0.14141E-01</td>
<td>0.0000</td>
<td>0.14146E-01</td>
</tr>
<tr>
<td>4</td>
<td>0.53333E-03</td>
<td>-0.30682E-01</td>
<td>0.0000</td>
<td>0.30686E-01</td>
</tr>
<tr>
<td>5</td>
<td>0.71111E-03</td>
<td>-0.52525E-01</td>
<td>0.0000</td>
<td>0.52530E-01</td>
</tr>
<tr>
<td>6</td>
<td>0.88889E-03</td>
<td>-0.78914E-01</td>
<td>0.0000</td>
<td>0.78919E-01</td>
</tr>
<tr>
<td>7</td>
<td>0.10667E-02</td>
<td>-0.10909</td>
<td>0.0000</td>
<td>0.10910</td>
</tr>
<tr>
<td>8</td>
<td>0.12444E-02</td>
<td>-0.14230</td>
<td>0.0000</td>
<td>0.14230</td>
</tr>
<tr>
<td>9</td>
<td>0.14222E-02</td>
<td>-0.17778</td>
<td>0.0000</td>
<td>0.17778</td>
</tr>
<tr>
<td>10</td>
<td>0.16000E-02</td>
<td>-0.21477</td>
<td>0.0000</td>
<td>0.21478</td>
</tr>
<tr>
<td>11</td>
<td>0.17778E-02</td>
<td>-0.25253</td>
<td>0.0000</td>
<td>0.25253</td>
</tr>
</tbody>
</table>

MAXIMUM ABSOLUTE VALUES

<table>
<thead>
<tr>
<th>NODE</th>
<th>VALUE</th>
</tr>
</thead>
<tbody>
<tr>
<td>11</td>
<td>0.17778E-02-0.25253</td>
</tr>
</tbody>
</table>
```
The deflections show axial deformations now, but the bending displacements and slopes are exactly the same as before indicating no interaction between the axial and bending loads.

List the stresses. (Results for the first element are shown.)

8. Main Menu > General Postproc > List Results > Element Solution > Line Element Results > Element Results > OK

PRINT ELEM ELEMENT SOLUTION PER ELEMENT

***** POST1 ELEMENT SOLUTION LISTING *****

LOAD STEP = 1 SUBSTEP = 1
TIME = 1.0000 LOAD CASE = 0

EL = 1 NODES = 1 2 MAT = 1

BEAM3
TEMP = 0.00 0.00 0.00 0.00
LOCATION SDIR SEBT SBYB
1(I) 5333.3 42614. -42614.
2(J) 5333.3 38352. -38352.

LOCATION SMAX SMIN
1(I) 47947. -37280.
2(J) 43686. -33019.

LOCATION EPELDIR EPELBYT EPELBYB
1(I) 0.000178 0.001420 -0.001420
2(J) 0.000178 0.001278 -0.001278

LOCATION EPTHDIR EPTHBYT EPTHBYB
1(I) 0.000000 0.000000 0.000000
2(J) 0.000000 0.000000 0.000000

EPINAXL = 0.000000

The added axial load produces a direct tensile stress (P/A) of 5333 psi that is combined with the bending stresses to give maximum and minimum values of 47,947 psi on the top of the beam and -37,280 psi on the bottom at node 1.

For problems with large deformations, the nonlinear coupling of axial stress and bending stiffness must be considered and ANSYS nonlinear solution options employed. What is large and what is small for a given problem? Some previous experience and/or numerical experimentation can help answer that question.

5-4 2-D FRAME

As another example of beam element modeling with ANSYS, we find the stress and deflection distribution in the two-dimensional frame of Figure 5-6. This model is contrived to include most of the possible situations one might encounter, including a downward force (7000 lbf), a concentrated moment (1000 in-lbf) and a wind load acting on the left side. The wind loading is developed from the load per area acting on the side of the structure that is converted to a line loading in load per unit length. It varies from 10 lbf/in at the top to zero lbf/in at the bottom.
The frame is constructed of W8x10 standard shape beams. The area of the cross section is 2.96 sq. in. and the flexural inertia is 30.8 in$^4$. The total section depth is 7.89 in. The frame is rigidly attached at its left support but allowed to pivot at its right support. If the I beams are made of A36 steel (Sy = 36,000 psi), will any yielding occur?

5-5 TUTORIAL 5B – 2-D FRAME

We use a text file to define the model, and perform the solution and postprocessing interactively. Note that the node dimensions are converted from feet to inches in the text file.

1. Start ANSYS and enter the following data using Read Input From ... Then Plot > Elements, etc.

/FILNAM,Tutorial5B
/title, Tutorial 5B – 2-D Frame

/prep7

et, 1, beam3 ! Element type; no.1 is beam3

!Material Properties
mp, ex, 1, 3.e7 ! Elastic modulus
mp, prxy, 1, 0.3 ! Poisson’s ratio
mp, dens, 1, 0.283/386. ! Mass density

Figure 5-7 Frame model.
!List of Nodes
n, 1, 0.0, 0.0    ! Node 1 is located at (0.0, 0.0) inches
! Convert from feet to inches for consistent units.
n, 2, 10.0*12, 0.0
n, 3, 0.0, 12.0*12
n, 4, 5.0*12, 12.0*12
n, 5, 10.0*12, 12.0*12
n, 6, 0.0, 24.0*12
n, 7, 10.0*12, 24.0*12

! Real constant set 1 for W8x10 I beam.
! Area, Izz (flexural Inertia), height 'h' as in sigma = Mc/I, c = h/2
! A = 2.96 sq.in., Izz = 30.8 in^4, h = 7.89 inch
r, 1, 2.96, 30.8, 7.89

!List of elements and nodes they connect
en, 1, 1, 3    ! Element Number 1 connects nodes 1 & 3
en, 2, 2, 5
en, 3, 3, 4
en, 4, 4, 5
en, 5, 3, 6
en, 6, 5, 7
en, 7, 6, 7

!Displacement Boundary Conditions
d, 1, ux, 0.0    ! Displacement at node 1 in x-dir is zero
d, 1, uy, 0.0    ! Displacement at node 1 in y-dir is zero
d, 1, rotz, 0.0  ! Rotation about z axis at node 1 is zero
d, 2, ux, 0.0    
d, 2, uy, 0.0    

!Applied Loadings
f, 4, fy, -7000. ! Force at node 4 in negative y-direction is 7000 lbf.
f, 4, mz, -1000  ! Moment about z axis is -1000 in-lbf
sfbeam, 1, 1, pres, 0, 5  ! Surface Force on beam 1 varies from 0 to 5
sfbeam, 5, 1, pres, 5, 10
acel, 0, 386., 0    ! acceleration of gravity for weight loading
finish

Note that boundary conditions must be supplied for the slope variables and that here line loads are applied to the beam elements using the sfbeam command. Each beam element has a local coordinate axis with local x directed from node I (first named node) to node J (second named node) and with y and z axes according to the right hand rule with z directed toward the viewer. The direction of the surface loading requires some pre-analysis planning and/or interactive experimentation with the model to get things right. To view the loads and element coordinate systems.
2. Utility Menu > PlotCtrls > Symbols > [PSF] Surface Load Symbols (set to Pressures) and Show pres and convect as (set to Arrows).

Utility Menu > PlotCtrls > Symbols > ESYS Element Coordinate sys (set to ON).

To view the concentrated moment and the slope boundary condition, select ISO in the Pan, Zoom, Rotate options.

3. Main Menu > Solution > Solve > Current LS > OK

When plotting the deformed shape, notice that even though the beam element deformed shape has been correctly computed, as a plotting convenience the nodes are connected by straight lines. If you want to visualize the beam element shape, include a few extra nodes and elements between the nodes.

Figure 5-8 Deformed frame.

4. Main Menu > General Postprocess > Plot Results > Deformed Shape > Def + undeformed.

Now list the nodal deflection values.

5. Main Menu > General Postproc > List Results > Nodal Solution > DOF Solution > Displacement vector sum > OK

---

PRINT DOF  NODAL SOLUTION PER NODE

***** POST1 NODAL DEGREE OF FREEDOM LISTING *****

LOAD STEP= 1 SUBSTEP= 1
TIME= 1.0000 LOAD CASE= 0

THE FOLLOWING DEGREE OF FREEDOM RESULTS ARE IN GLOBAL COORDINATES

<table>
<thead>
<tr>
<th>NODE</th>
<th>UX</th>
<th>UY</th>
<th>UZ</th>
<th>USUM</th>
</tr>
</thead>
<tbody>
<tr>
<td>1</td>
<td>0.0000</td>
<td>0.0000</td>
<td>0.0000</td>
<td>0.0000</td>
</tr>
<tr>
<td>2</td>
<td>0.0000</td>
<td>0.0000</td>
<td>0.0000</td>
<td>0.0000</td>
</tr>
<tr>
<td>3</td>
<td>0.49528</td>
<td>-0.35297E-02</td>
<td>0.0000</td>
<td>0.49529</td>
</tr>
<tr>
<td>4</td>
<td>0.49537</td>
<td>-0.14501</td>
<td>0.0000</td>
<td>0.51616</td>
</tr>
<tr>
<td>5</td>
<td>0.49547</td>
<td>-0.87344E-02</td>
<td>0.0000</td>
<td>0.49555</td>
</tr>
<tr>
<td>6</td>
<td>0.73410</td>
<td>-0.31099E-02</td>
<td>0.0000</td>
<td>0.73411</td>
</tr>
<tr>
<td>7</td>
<td>0.73316</td>
<td>-0.95128E-02</td>
<td>0.0000</td>
<td>0.73322</td>
</tr>
</tbody>
</table>

MAXIMUM ABSOLUTE VALUES

<table>
<thead>
<tr>
<th>NODE</th>
<th>VALUE</th>
</tr>
</thead>
<tbody>
<tr>
<td>6</td>
<td>0.73410</td>
</tr>
<tr>
<td>4</td>
<td>-0.14501</td>
</tr>
<tr>
<td>0</td>
<td>0.0000</td>
</tr>
<tr>
<td>6</td>
<td>0.73411</td>
</tr>
</tbody>
</table>
As an alternative to the method used in the Tutorial 5A to examine stress results, here we will create an Element Table. Because the direct and bending stresses combine to produce different results on the top and bottom of the beam and also produce different results at each end of the beam, we will create four element table entries using interactive commands.

6. Main Menu > General Postprocess > Element Table > Define Table > Add

Enter a label you choose such as SmaxI and scroll down to find By sequence num.

Select NMISC and enter 1 (to the right of NMISC) > Apply.

![Figure 5-9 Define element table quantities.](image)

Repeat for

Label SminI, By sequence num, NMISC, 2 > Apply

Label SmaxJ, By sequence num, NMISC, 3 > Apply

Label SminJ, By sequence num, NMISC, 4 > OK > Close Element Table Data

This sets up tabular data for Smax and Smin at nodes I and J for each element.

7. Main Menu > General Postprocess > Element Table > List Elem Table (Click on the first four items SMAXI, SMINI, etc.) > OK
The resulting element stresses are shown in the table below.

PRINT ELEMENT TABLE ITEMS PER ELEMENT

***** POST1 ELEMENT TABLE LISTING *****

<table>
<thead>
<tr>
<th>STAT</th>
<th>CURRENT ELEM</th>
<th>CURRENT MAXI</th>
<th>CURRENT MINI</th>
<th>CURRENT MAXJ</th>
<th>CURRENT MINJ</th>
</tr>
</thead>
<tbody>
<tr>
<td>1</td>
<td>10119.</td>
<td>-11631.</td>
<td>2525.3</td>
<td>-3955.3</td>
<td></td>
</tr>
<tr>
<td>2</td>
<td>-1840.0</td>
<td>-1840.0</td>
<td>8431.5</td>
<td>-12030.</td>
<td></td>
</tr>
<tr>
<td>3</td>
<td>515.64</td>
<td>-419.78</td>
<td>18113.</td>
<td>-18017.</td>
<td></td>
</tr>
<tr>
<td>4</td>
<td>18241.</td>
<td>-18145.</td>
<td>18439.</td>
<td>-18343.</td>
<td></td>
</tr>
<tr>
<td>5</td>
<td>2839.7</td>
<td>-2705.5</td>
<td>1099.5</td>
<td>-883.83</td>
<td></td>
</tr>
<tr>
<td>6</td>
<td>7978.0</td>
<td>-8343.0</td>
<td>4545.1</td>
<td>-4828.7</td>
<td></td>
</tr>
<tr>
<td>7</td>
<td>756.35</td>
<td>-1227.0</td>
<td>4451.6</td>
<td>-4922.2</td>
<td></td>
</tr>
</tbody>
</table>

MINIMUM VALUES
ELEM 2 4 5 4
VALUE -1840.0 -18145. 1099.5 -18343.

MAXIMUM VALUES
ELEM 4 3 4 5
VALUE 18241. -419.78 18439. -883.83

A maximum stress of 18,439 psi occurs in element 4, but there will be no yielding in the frame for a material yield stress of 36 kpsi.

You can also plot each of these table data items for a visual indication of the stress distribution in the frame. See below.

Figure 5-11
Plot of SMAXI.

8. Main Menu > General Postprocess > Element Table > Plot Elem Table (Select the one to plot.) > OK
5-6 BEAM MODELS IN 3-D

When a beam element is incorporated in a 3-dimensional model, the full 3-D flexibility of the beam must be considered. It can have axial deformation, torsional deformation, and bending deformations in two principal bending planes. Each beam of the model must be positioned in space to reflect the proper orientation of the element cross section.

The correct orientation can be defined by specifying the angular rotation of the element about its longitudinal axis or by employing three node points to define a principal plane of bending for the element. The next tutorial uses the approach with 3 nodes.

5-7 TUTORIAL 5C – ‘L’ BEAM

Objective: Find the stresses and deflections of a simple ‘L’-shaped aluminum beam with one end cantilevered and a point load at the other end.

The ANSYS 3D beam element BEAM4 is used in modeling this problem. A typical element located in a global coordinate system XYZ is shown below. It connects two
nodes I and J, and has its local or element x-axis defined by these two nodes. The local y and z axes are aligned with the principal cross section flexural inertia planes.

![Diagram of global and local axes](image)

**Figure 5-14** Global and local axes.

A third node, K, is used to define the local x-z plane which is one of the principal planes of bending of the beam. Node K can be another node in the model or a dummy node (with all DOF set to zero) that is used just for orientation purposes. The beam cross sectional properties are defined using an ANSYS ‘Real Constants’ set. The values for Izz and Iyy entered into the program must correspond to the orientation specified by the three nodes. Check your work carefully. It’s easy to get Izz and Iyy reversed.

The Global axis (X,Y,Z) and local axis (x,y,z) definitions for the two elements of the simple problem of this tutorial are illustrated below.
Figure 5-15 Orientation of element 1; I = node 1, J = node 2, K = node 3.

Figure 5-16 Orientation of element 2; I = node 2, J = node 3, K = node 1.

**Important:** The geometric property constant $I_{xx}$ is the torsional stiffness constant for the cross section. For circular sections $I_{xx}$ is equal to the polar moment of inertia ($I_{yy} + I_{zz}$). For non-circular sections the torsional stiffness constant is **not** equal to the polar moment of inertia (see torsion of non-circular section in a solid mechanics reference). If no value is entered for $I_{xx}$, however, ANSYS will compute the torsional stiffness...
constant as Iyy + Izz which is correct for circular sections but not correct for non-circular sections.

The text file below defines the simple 'L' – Beam ANSYS model.

/FILLM, Tutorial5C
/title, 3D Beam Sample Problem - An 'L' - Beam
/prep7

n, 1, 0.0, 0.0, 0.0 ! Node 1 is located at (X=0.0, Y=0.0, Z=0.0)
n, 2, 0.0, 0.0, 12.0
n, 3, 6.0, 0.0, 12.0

et, 1, beam4 ! Element type; no.1 is 3D beam

!Material Properties
mp, ex, 1, 1.e7 ! Elastic modulus, psi
mp, prxy, 1, 0.3 ! Poisson's ratio

! Real properties for beam with 0.25 x 1.0 cross section
! (6 values on first line. 'rmore' for additional values.)

! "r", Set number, Area, Izz, Iyy, thick-z,thick-y,theta,
r, 1, 0.25, 0.001302, 0.0208, 1.0, 0.25, 0

! "rmore", intr strain, Ixx,shear-z,shear-y,spin,addmass
rmore, 0, .004388, . . .

! element connection and orientation
en, 1, 1, 2, 3 ! Element #1 connects nodes 1 & 2, and
! uses node 3 to define the element local x-z plane.
en, 2, 2, 3, 1 ! Element #2 connects nodes 2 & 3, and
! uses node 1 to define the element local x-z plane.

! Displacement boundary conditions
d, 1, ux, 0. ! Displacement at node 1 in x-dir is zero
d, 1, uy, 0.
d, 1, uz, 0.
d, 1, rotx, 0. ! Rotation at node 1 about x-axis is zero
d, 1, roty, 0.
d, 1, rotz, 0.

! (We could have used d, 1, all, 0.0 to define all root restraints.)

! Applied force
f, 3, fy, -5.0 ! Force at node 3 in the negative y-direction
finish

/solu ! Select static load solution
antype, static
solve
save
finish

/post1
1. Read the problem definition file into ANSYS, and once it's solved, plot the deformed shape.

The deformed plus undeformed beam is shown in the figure below. Nodes were only used at the ends of the 'L' segments so the deformed shape shows a straight-line connection between nodes. The calculated deformations are correct however. You can check the results using superposition or energy methods.

![Deformed two-element beam](image)

**Figure 5-17** Deformed two-element beam.

The rotation boundary conditions ROTX, ROTY, ROTZ are indicated by double-headed arrows, positive by the right-hand rule. They overlay the displacement conditions UX, UY, UZ at the cantilever support; on screen the two conditions are shown in different colors. Now list the deflection, slope and stress results.

2. **Main Menu > General Postproc > List Results > Nodal Solution > DOF Solution > Displacement vector sum > OK**

```
PRINT U NODAL SOLUTION PER NODE

***** POST1 NODAL DEGREE OF FREEDOM LISTING *****

LOAD STEP= 1 SUBSTEP= 1
TIME= 1.0000 LOAD CASE= 0

THE FOLLOWING DEGREE OF FREEDOM RESULTS ARE IN GLOBAL COORDINATES

<table>
<thead>
<tr>
<th>NODE</th>
<th>UX</th>
<th>UY</th>
<th>UZ</th>
<th>USUM</th>
</tr>
</thead>
<tbody>
<tr>
<td>1</td>
<td>0.0000</td>
<td>0.0000</td>
<td>0.0000</td>
<td>0.0000</td>
</tr>
<tr>
<td>2</td>
<td>0.0000</td>
<td>-0.22120</td>
<td>0.0000</td>
<td>0.22120</td>
</tr>
<tr>
<td>3</td>
<td>0.0000</td>
<td>-0.37683</td>
<td>0.0000</td>
<td>0.37683</td>
</tr>
</tbody>
</table>

MAXIMUM ABSOLUTE VALUES

<table>
<thead>
<tr>
<th>NODE</th>
<th>VALUE</th>
</tr>
</thead>
<tbody>
<tr>
<td>0</td>
<td>0.0000</td>
</tr>
<tr>
<td>3</td>
<td>-0.37683</td>
</tr>
<tr>
<td>0</td>
<td>0.0000</td>
</tr>
<tr>
<td>3</td>
<td>0.37683</td>
</tr>
</tbody>
</table>
```
3. Main Menu > General Postproc > List Results > Nodal Solution > DOF Solution > Displacement vector sum > OK

PRINT ROT NODAL SOLUTION PER NODE

***** POST1 NODAL DEGREE OF FREEDOM LISTING *****

LOAD STEP= 0 SUBSTEP= 1
TIME= 1.0000 LOAD CASE= 0

THE FOLLOWING DEGREE OF FREEDOM RESULTS ARE IN THE GLOBAL COORDINATE SYSTEM

<table>
<thead>
<tr>
<th>NODE</th>
<th>ROTX</th>
<th>ROTY</th>
<th>ROTZ</th>
<th>RSUM</th>
</tr>
</thead>
<tbody>
<tr>
<td>1</td>
<td>0.0000</td>
<td>0.0000</td>
<td>0.0000</td>
<td>0.0000</td>
</tr>
<tr>
<td>2</td>
<td>0.27650E-01</td>
<td>0.0000</td>
<td>-0.21331E-01</td>
<td>0.34922E-01</td>
</tr>
<tr>
<td>3</td>
<td>0.27650E-01</td>
<td>0.0000</td>
<td>-0.28243E-01</td>
<td>0.39525E-01</td>
</tr>
</tbody>
</table>

MAXIMUM ABSOLUTE VALUES

<table>
<thead>
<tr>
<th>NODE</th>
<th>VALUE</th>
</tr>
</thead>
<tbody>
<tr>
<td>2</td>
<td>0.27650E-01</td>
</tr>
<tr>
<td>3</td>
<td>0.28243E-01</td>
</tr>
</tbody>
</table>

4. Main Menu > General Postproc > List Results > Element Solution > Linie Element Results > Element Results > OK

PRINT ELEM ELEMENT SOLUTION PER ELEMENT

***** POST1 ELEMENT SOLUTION LISTING *****

LOAD STEP= 1 SUBSTEP= 1
TIME= 1.0000 LOAD CASE= 0

EL= 1 NODES= 1 2 3 MAT= 1 BEAM4

<table>
<thead>
<tr>
<th>LOCATION</th>
<th>SDIR</th>
<th>SBYT</th>
<th>SBZT</th>
</tr>
</thead>
<tbody>
<tr>
<td>1 (I)</td>
<td>0.0000</td>
<td>-5760.4</td>
<td>5760.4</td>
</tr>
<tr>
<td>2 (J)</td>
<td>0.0000</td>
<td>0.0000</td>
<td>0.0000</td>
</tr>
</tbody>
</table>

LOCATION EPELDIR EPELBYT EPELBYB EPELZT EPELBZB

<table>
<thead>
<tr>
<th>LOCATION</th>
<th>EPDTHDIR</th>
<th>EPELZT</th>
<th>EPELBZB</th>
</tr>
</thead>
<tbody>
<tr>
<td>1 (I)</td>
<td>0.000000</td>
<td>0.000000</td>
<td>0.000000</td>
</tr>
<tr>
<td>2 (J)</td>
<td>0.000000</td>
<td>0.000000</td>
<td>0.000000</td>
</tr>
</tbody>
</table>

LOCATION SMAX SMIN

<table>
<thead>
<tr>
<th>LOCATION</th>
<th>SMAX</th>
<th>SMIN</th>
</tr>
</thead>
<tbody>
<tr>
<td>1 (I)</td>
<td>5760.4</td>
<td>-5760.4</td>
</tr>
<tr>
<td>2 (J)</td>
<td>0.0000</td>
<td>0.0000</td>
</tr>
</tbody>
</table>

LOCATION EPELDIR EPELBYT EPELBYB EPELZT EPELBZB

<table>
<thead>
<tr>
<th>LOCATION</th>
<th>EPDTHDIR</th>
<th>EPELZT</th>
<th>EPELBZB</th>
</tr>
</thead>
<tbody>
<tr>
<td>1 (I)</td>
<td>0.000000</td>
<td>0.000000</td>
<td>0.000000</td>
</tr>
<tr>
<td>2 (J)</td>
<td>0.000000</td>
<td>0.000000</td>
<td>0.000000</td>
</tr>
</tbody>
</table>

EPINAXL = 0.000000

EL= 2 NODES= 2 3 1 MAT= 1 BEAM4

<table>
<thead>
<tr>
<th>LOCATION</th>
<th>SDIR</th>
<th>SBYT</th>
<th>SBZT</th>
</tr>
</thead>
<tbody>
<tr>
<td>1 (I)</td>
<td>0.0000</td>
<td>-2880.2</td>
<td>2880.2</td>
</tr>
<tr>
<td>2 (J)</td>
<td>0.0000</td>
<td>0.000000</td>
<td>0.54573E-11</td>
</tr>
</tbody>
</table>

LOCATION SMAX SMIN

<table>
<thead>
<tr>
<th>LOCATION</th>
<th>SMAX</th>
<th>SMIN</th>
</tr>
</thead>
<tbody>
<tr>
<td>1 (I)</td>
<td>2880.2</td>
<td>-2880.2</td>
</tr>
<tr>
<td>2 (J)</td>
<td>0.54573E-11</td>
<td>0.54573E-11</td>
</tr>
</tbody>
</table>

LOCATION EPELDIR EPELBYT EPELBYB EPELZT EPELBZB

<table>
<thead>
<tr>
<th>LOCATION</th>
<th>EPDTHDIR</th>
<th>EPELZT</th>
<th>EPELBZB</th>
</tr>
</thead>
<tbody>
<tr>
<td>1 (I)</td>
<td>0.000000</td>
<td>0.000000</td>
<td>0.000000</td>
</tr>
<tr>
<td>2 (J)</td>
<td>0.000000</td>
<td>0.000000</td>
<td>0.000000</td>
</tr>
</tbody>
</table>

EPINAXL = 0.000000
These stress values can be checked easily by hand since the simple 'L'-beam is statically determinate. (SBYT = the bending stress on the +y local axis side of the beam.)

Alternatively the orientation of beams in space can be specified using only nodes I and J if one also gives the **angle of rotation** \( \theta \) of the cross section about the **element axis** \( x \). (See Figure 5-14.) If \( \theta = 0 \), the element local \( y \)-**axis** is automatically calculated to be parallel to the global X-Y plane. In the previous example we get the same results if the third node is eliminated from the element definition since \( \theta = 0 \) corresponds to the desired orientation of the two elements in this case.

### 5-8 SUMMARY

ANSYS modeling of two and three-dimensional beam bending problems has been presented in this lesson and tutorials given to provide hands-on experience with this kind of analysis. The engineering theory of beam bending is the basis for the elements utilized here, and angular degrees of freedom (ROTX, etc.) are introduced for the first time since these variables are required for the calculation of the beam element neutral axis slope.

Transverse loads that cause bending also cause shearing deformations that may become important for short beams. The real property set for beams allows specification of the effective shear deformation area so that this behavior can be modeled when necessary.

Because they connect two node points, beam and truss elements are called ‘line’ elements in ANSYS. The Shell (plate) elements of the next lesson are the two-dimensional or surface equivalents of beams, connecting three or four nodes in a plane, they are loaded by forces transverse to their surface and experience bending in ways similar to beams.

### 5-9 PROBLEMS

Use 2-D beam models to find the maximum deflection (mm or inches) and bending stress (N/m\(^2\) or psi) for the rectangular cross section single span beams of Problems 5-1 through 5-5 below. Use metric system units, **M**, or British system units, **B**, for problem formulation. Compare your computed results with those that you calculate from elementary beam theory (Use the uniform beam module in www.etbx.com, for example).

<table>
<thead>
<tr>
<th>Description</th>
<th>Metric</th>
<th>English</th>
</tr>
</thead>
<tbody>
<tr>
<td>Length</td>
<td>( L = 3 ) m</td>
<td>9.8 ft.</td>
</tr>
<tr>
<td>Elastic modulus</td>
<td>( E = 2.11 \times 10^8 ) N/m(^2)</td>
<td>2.9E7 psi</td>
</tr>
<tr>
<td>Cross section Base</td>
<td>( b = 50 ) mm</td>
<td>1.97 in</td>
</tr>
<tr>
<td>Cross section Height</td>
<td>( h = 150 ) mm</td>
<td>5.9 in</td>
</tr>
<tr>
<td>Distributed load</td>
<td>( w = 30 ) kN/m</td>
<td>2055 lbf/ft</td>
</tr>
<tr>
<td>Point load</td>
<td>( P = 25 ) kN</td>
<td>5620 lbf</td>
</tr>
<tr>
<td>Point moment</td>
<td>( M = 12 ) kN-m</td>
<td>8852 ft-lbf</td>
</tr>
</tbody>
</table>

5-1 Simply supported beam (a) point load \( P \) at center, (b) uniformly distributed load \( w \).
5-2 Fixed (cantilevered) at one end, free at the other, (a) point load \( P \) at free end, (b) uniformly distributed load \( w \), (c) point moment \( M \) at free end, (d) linearly distributed load varying from \( w \) at the fixed end to zero at the free end.

5-3 Fixed (cantilevered) at both ends, (a) point load \( P \) at center, (b) uniformly distributed load \( w \).

5-4 Fixed (cantilevered) at one end, simply supported at the other, (a) point load \( P \) at center, (b) uniformly distributed load \( w \).

5-5 Find the maximum deflection and stress in the stepped steel cantilevered beam below with a point load on the free end. The segments are of equal length and the larger inertia is twice the smaller. Select your own material, geometric, and load values. Check the maximum bending stress with a simple hand calculation. The ANSYS calculated end deflection should be somewhere between that for a single beam of inertia I and that for one with an inertia 2I.

Figure P5-5

5-6 A stepped steel shaft with equal length segments of 16 inches, and 1.5 inch, 1 inch diameter cross sections is loaded with a 1000 lbf force in the middle and considered restrained from deflection and slope by the bearings at either end. (a) Compute the mid-span deflection and moment. (b) Compute the maximum bending stress at the step if the fillet radius is 0.25 inch. Do this by developing a 3D model of a segment of the shaft near the step. Apply the moment found in (a) to the segment using equal and opposite forces. Assume that the 1000 lbf does not contribute significantly to the local stress distribution and can be ignored. Check your results against published stress concentration factors for this geometry.

Figure P5-6

5-7 Find the maximum stress and deflection of the 2-D frame shown if it has both columns fixed at ground level. The geometric, material, and distributed load parameters are the same as for the beam of Problem 5-1. There is a side load of \( w \) and also a uniformly distributed downward load \( w \) (not shown) on each horizontal beam.
5-8 Determine the maximum stress in the ‘L’-beam of Figure 5-11 if a 10 lbf force is applied in the global Z direction in addition to the 5.0 lbf Y-direction force shown. Confirm the ANSYS result with a hand calculation.

5-9 Create a 3-dimensional model of a building frame using two frames like that of Problem 5-7, the second displaced 3 m in the Z-direction from the first. The second frame is loaded like the first and connected to it by with beams at the 4 and 8 m levels. The connecting beams (same properties as the others) have distributed loads 2w N/m. Find maximum stress and displacement.

5-10 The tower in the figure has a right triangle base and is loaded by two 5000 lbf forces parallel to the direction of the 8 ft side of the triangle. Find the maximum displacement and stress if the tower is made from steel pipe with a 4.5 inch OD and a wall thickness of 0.337 inches. The three legs are firmly anchored at ground level, so the three base triangle lines do not need to be elements of the model. Use BEAM4 or PIPE16 (needs only two nodes to define) elements.