University of Utah

ME EN 6510/5510
Introduction to Finite Elements
Fall 2005

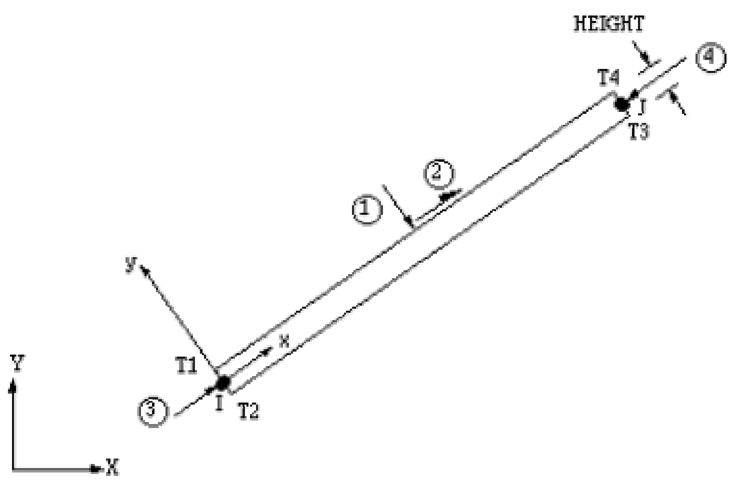
Beam Elements

snip (from ANSYS Manual)

4.3 BEAM3 2-D Elastic Beam

BEAM3 is a uniaxial element with tension, compression, and bending capabilities. The element has three degrees of freedom at each node: translations in the nodal x and y directions and rotation about the nodal z-axis. See Section 14.3 of the *ANSYS Theory Reference* for more details about this element. Other 2-D beam elements are the plastic beam (BEAM23) and the tapered unsymmetric beam (BEAM54).

Figure 4.3-1 BEAM3 2-D Elastic Beam



4.3.1

Table 4.3-1 BEAM3 Input Summary

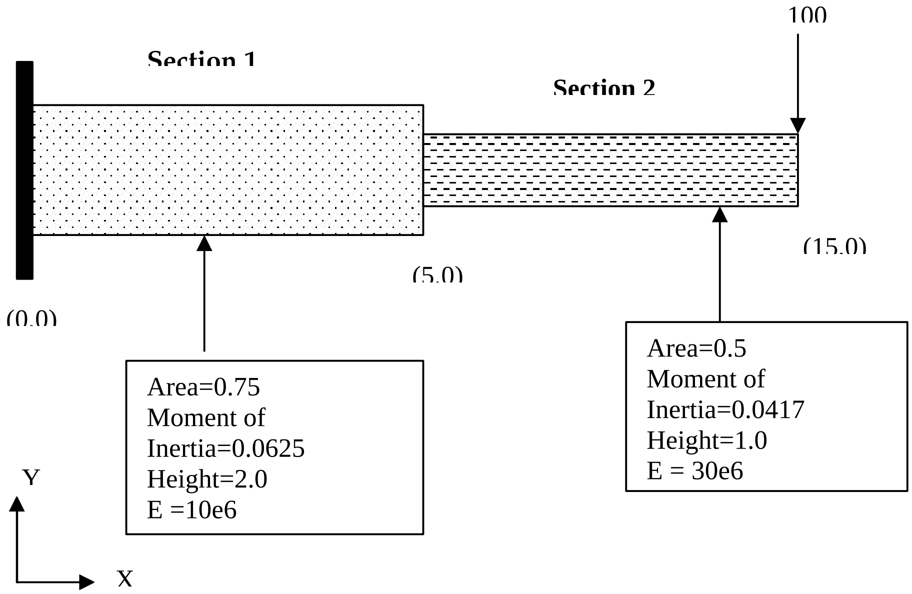
Element Name	BEAM3
Nodes	I, J
Degrees of Freedom	UX, UY, ROTZ
Real Constants	AREA, IZZ, HEIGHT, SHEARZ, ISTRN, ADDMAS
Material Properties	EX, ALPX, DENS, GXY, DAMP

4.3.3 Assumptions and Restrictions

The beam element can have any cross-sectional shape for which the moment of inertia can be computed. However, the stresses are determined as if the distance from the neutral axis to the extreme fiber is one-half of the height. The element height is used only in the bending and thermal stress calculations. The applied thermal gradient is assumed linear across the height and along the length. The beam element must lie in an X-Y plane and must not have a zero length or area. The moment of inertia may be zero if large deflections are not used.

end of snip

Beam Elements - A simple cantilever beam problem with 2 different materials and section properties will be analyzed, and an alternative way to generate nodes and elements will be used (keypoints and lines).



Analysis: A 1D analysis of the above problem with line elements will be performed.

This lab shows both the menu paths for using the ANSYS graphical user interface (GUI) as well as the code for direct input (see ALTERNATIVE TO GUI METHOD).

GUI METHOD

Step 1: Define Keypoint Locations
Preprocessor→*Modeling*→*Create*→*Keypoints*→*In Active CS*→(window) define keypoint 1 with label and XYZ coordinates (0,0,0), hit *Apply* and repeat for keypoints 2 at (5,0,0) and 3 at (15,0,0)→when finished, select *OK* to exit window.

Step 2: Define Lines (underlying geometry for beam element mesh)
The following commands must be typed directly into the command input window:

L,1,2,5	creates a line between keypoints 1 and 2 having 5 divisions (which will mean 5 elements).
L,2,3,10	creates a line between keypoints 2 and 3 having 10 divisions (which will mean 10 elements).

Step 3: Select Element Type
Preprocessor→*Element Type*→*Add/Edit/Delete*→(window) *Add…*→(window) highlight *Beam, 2D elastic (beam3)*→*OK*→*CLOSE*.

Step 4: Define Material Property No. 1 and No. 2
Preprocessor→*Material Properties*→*Material Models*→(window) double click *Structural/Linear/Elastic/Isotropic*→(window) input modulus and Poisson's ratio for Section 1→*OK*→(back to Material Model window) select menu item (at top) *Material*→*New Model*→(window) enter material ID (default 2), then *OK*→double click *Structural/Linear/Elastic/Isotropic*→(window) input modulus and Poisson's ratio for Section 2→*OK*→(close Material Model window).

Step 5: Define Physical Property Set 1 and Set 2
Preprocessor→*RealConstants*→*Add/Edit/Delete* →(window) *Add…*→(window with element Type Beam3 highlighted) *OK*→(window) input properties for Section 1, then select *OK*→(back to original window) *Add…*→ (back to window with element Type Beam3 highlighted) *OK*→(window—notice set ID is now No. 2) input properties for Section 2, then select *OK*→*CLOSE*.

Step 6: Mesh Lines
Mesh Line 1 First (matches current material type and real type)
Preprocessor→*Meshing*→*Lines* →*Picked Lines*→(window asking you to select line) pick line 1→*OK*.

Change Current Material and Physical Property Sets
Preprocessor→*Modeling*→*Create*→*Elements*→*Element Attributes*→(window) change Real Constant Set No. to 2, and Material No. to 2, then select *OK*. (Notice now in the bottom field that mat=2 and real=2).

Mesh Line 2
Preprocessor→*Meshing*→*Lines* →*Picked Lines*→(window asking you to select line) pick line 2→*OK*.

(Hint: you can use the *List*→*Elements*→*Attributes and Real Constants* command in the menu bar to verify that you have 15 total beam elements--5 with Material No. 1 and Real Constant No. 1, and 10 with Material No. 2 and Real Constant No. 2.)

Step 7: Apply Boundary Conditions
Preprocessor→*Loads*→*Define Loads*→*Apply*→*Structural*→*Displacement*→*On Nodes*→(window) pick node 1, then select *OK*→(window) highlight *ALL DOF*; make sure it shows *Apply As: Constant Value*; enter value as 0, select *OK*.

Step 9: Apply Loads
Preprocessor→*Loads*→*Define Loads*→*Apply*→*Structural*→*Force/Moment*→*On Nodes*→(window) pick node 7 (at end of beam), then select *OK*→(window) choose Fy direction and enter -100→ *OK*.

Step 10: Solve
Solution→*Solve*→*Current LS*→(asks you to review summary info) select *OK*→ANSYS will begin solving the problem and will post a message "Solution is done!" when it has finished. Close message windows and go to next step.

Step 11: View Results
Plot Deformation: *General Postproc*→*Plot Results*→*Contour Plot*→*Nodal Solution*→(window) highlight *DOF solution* and *Translation Uy*; pick button *Def + undeformed*; select *OK*.

List Nodal Displacements: *General Postproc*→*List Results*→ *Nodal Solution*→(window) highlight *DOF solution* and *Uy*; select *OK*. (This can also be done by simply typing *prdisp* in the command input window.) You should get the following:

NODE	Ux	Uy	ROTZ
1	0.0	0.0	0.0
2	0.0	-0.26667E-01	-0.10000E-01
3	0.0	-0.11733E-02	-0.23200E-02
4	0.0	-0.45867E-02	-0.44800E-02
5	0.0	-0.10080E-01	-0.64800E-02
6	0.0	-0.17493E-01	-0.83200E-02
7	0.0	-0.15331	-0.13997E-01
8	0.0	-0.37053E-01	-0.10759E-01
9	0.0	-0.48159E-01	-0.11439E-01
10	0.0	-0.59904E-01	-0.12038E-01
11	0.0	-0.72209E-01	-0.12558E-01
12	0.0	-0.84993E-01	-0.12998E-01
13	0.0	-0.98177E-01	-0.13357E-01
14	0.0	-0.11168	-0.13637E-01
15	0.0	-0.12542	-0.13837E-01
16	0.0	-0.13933	-0.13957E-01

Plot and Print Tensile and Compressive Stresses at the Top and Bottom of Beam: This needs to be done using the command input window. Type the following in the window:

```
etable,sigtop,ls,2      ! enter the element  stresses on the top into a user defined table 'sigtop'*
etable,sigbot,ls,3      ! enter the element  stresses on the bottom into a user defined table 'sigbot'*
pletab,sigtop           ! plots the stress values in the 'sigtop' table
pletab,sigbot           ! plots the stress values in the 'sigbot'  table
pretab,sigtop           ! prints the stress values in the 'sigtop' table
pretab,sigbot           ! prints the stress values in the 'sigbot'  table
```

***Q:** How do we know which end of the beam element(s) are being used to build the stress tables sigtop and sigbot when we use the **etable** command? Although not terribly important for this lab, you may have a situation where it is not intuitive which side of the beam element is experiencing the maximum moment, and therefore the maximum stress. You need to understand how to access element output at both nodes. Open the ANSYS Help menu and type "beam3" in the index search. This will pull up literature describing everything you want to know about the beam3 element, including how it calculates stress, what element output is accessible in the etable command, etc. Hopefully what you'll realize is that the ls,2 and ls,3 commands give the top stress and bottom stress at node i (the first node defining the element). If you needed these stresses at the other side of the beam element (node j), then you would have to use

```
etable,sigtopnodej,ls,7
etable,sigbotnodej,ls,8
pretab,sigtopnodej          !or pletab,sigtopnodej if you want to show contour plot.
pretab,sigtop               !or pletab,sigbotnodej if you want to show contour plot.
```

Other Questions: What other output can you get from the element? Moments? Torques? Maximum Combined Stress? If the ANSYS software assumes the beam is rectangular, how does this affect stress output if you are trying to model circular beams like tubes?

ALTERNATIVE TO GUI METHOD:

ANSYS **Input file**	**Comments**
/title, *your_ title*	
/prep7	
k,1,0,0	! sets a keypoint at (0,0)
k,2,5,0	! sets a keypoint at (5,0)
k,3,15,0	! sets a keypoint at (15,0)
l,1,2,5	! creates a line with 5 divisions from keypoints 1 to 2
	! you can remove lines with "ldel"
l,2,3,10	! line with 10 divisions from point 2 to 3
et,1,3	! sets element type 1 to beam3, the 2d beam element in ANSYS
mp,ex,1,10e6	! sets modulus of mtl 1
mp,prxy,1,0.3	! sets poissons ratio of mtl 1
mp,ex,2,30e6	! sets modulus of mtl 2
mp,prxy,2,0.3	!sets poissons ratio of mtl 2
r,1,0.75,0.0625,2.0	! defines property set 1 for the beam element (A,I,height)
r,2,0.5,0.0417,1.0	! defines property set 2 for the beam element (A,I,height)
mat,1	! sets material to 1, not needed as it defaults to 1
real,1	! sets the property set to 1 (this is the default)
lmesh,1	! creates a mesh of elements on line1 (undo with lclear)
mat,2	! sets current material to 2
real,2	! sets element properties to set 2
lmesh,2	! creates a mesh of elements on line 2
d,1,all,0	! constrains displacements at node 1 to zero
f,7,fy,-100.	! applies a force of 100 units in the (–ve) Y direction at node 7

Once ANSYS is up and running, move your cursor within the direct command input window and type the following.

```
/prep7                  ! Puts ANSYS into the preprocessor module
/show, x11              ! directs output to the screen
/input,filename.dat     ! Filename.dat is the name of your input file
                        ! All of the commands in the input file will be
                        ! executed. Remember 8 chars max for filename, 3 for extension.
```

The following commands are then used to examine your model to see that it is correct and what you want.

```
nlist                   ! gives a list of the nodes. Note the numbering and locations.
elist                   ! lists the elements
eplot                   ! plots the elements
```

If the model is not correct, go to the editor and fix the input file, and re-enter it as above. Remember to clear out the old one before attempting to read in the new one. If correct, proceed as follows.

```
finish                  exits prep7

/solu                   ! enters the solution phase
solve                   ! runs the solution
finish                  ! exits the solution phase
```

Now enter the postprocessor and examine the results:

```
/post1                  ! enter the postprocessing  phase
pldisp,1                ! plot  displacements  over original  geometry
etable,sigtop,ls,2      ! enter the element  stresses on the top into a user defined table 'sigtop'
etable,sigbot,ls,3      ! enter the element  stresses on the bottom into a user defined table 'sigbot'
etable,smax,nmisc,1     ! make table 'smax' using maximum stress
etable,smin,nmisc,2     ! make table 'smin' using maximum stress
pletab, smax            ! plots the stress values in the 'smax'  table
pletab, smin            ! plots the stress values in the 'smin' table
pletab, sigtop          ! plots the stress values in the 'sigtop' table
pletab, sigbot          ! plots the stress values in the 'sigbot'  table
prdisp                  ! prints the displacements
finish                  ! exits post1, the postprocessor
```