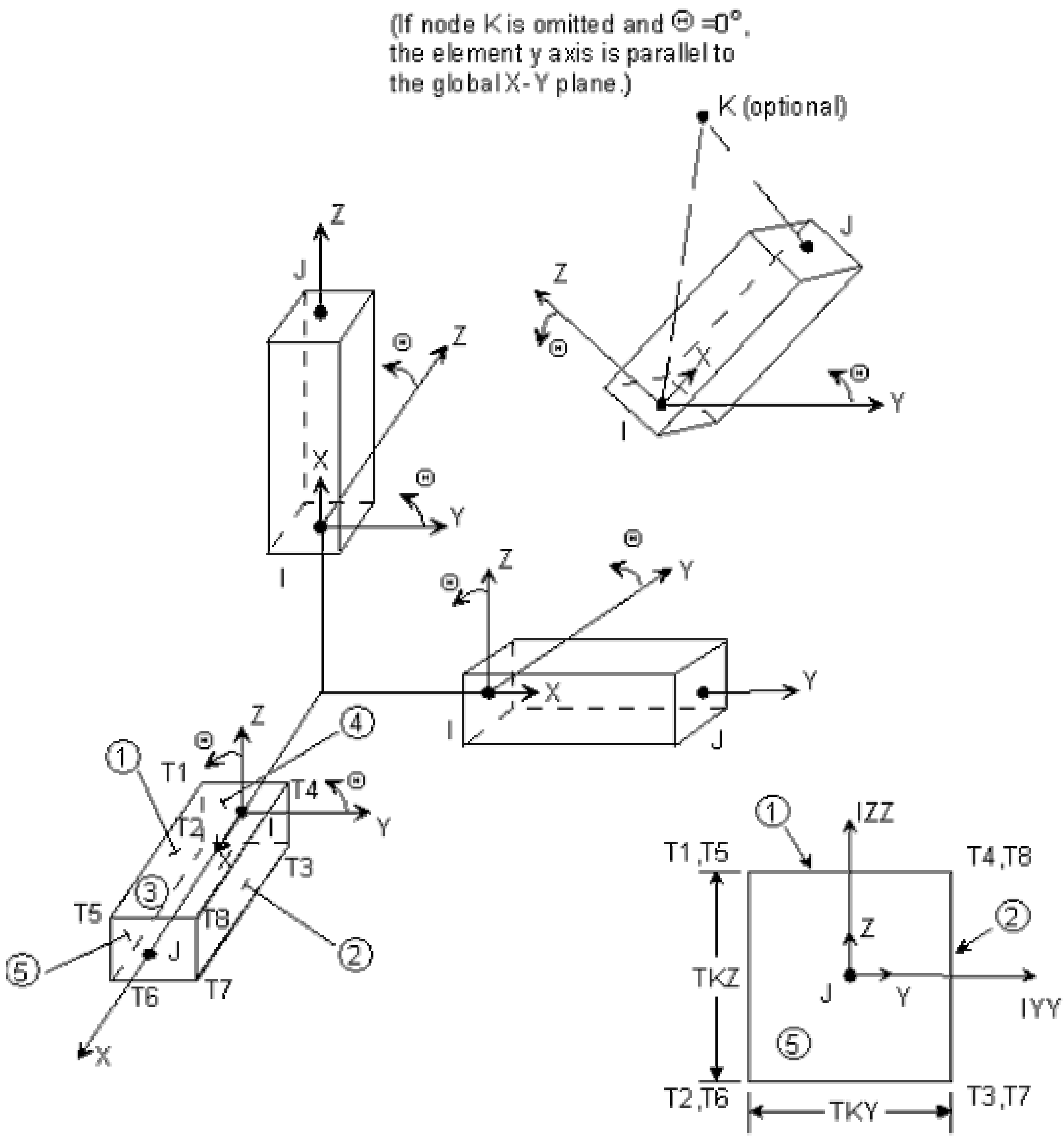
BEAM4

Element Description

BEAM4 is a uniaxial element with tension, compression, torsion, and bending capabilities. The element has six degrees of freedom at each node: translations in the nodal x, y, and z directions and rotations about the nodal x, y, and z axes. Stress stiffening and large deflection capabilities are included. A consistent tangent stiffness matrix option is available for use in large deflection (finite rotation) analyses.

Figure 1. BEAM4 3-D Elastic Beam



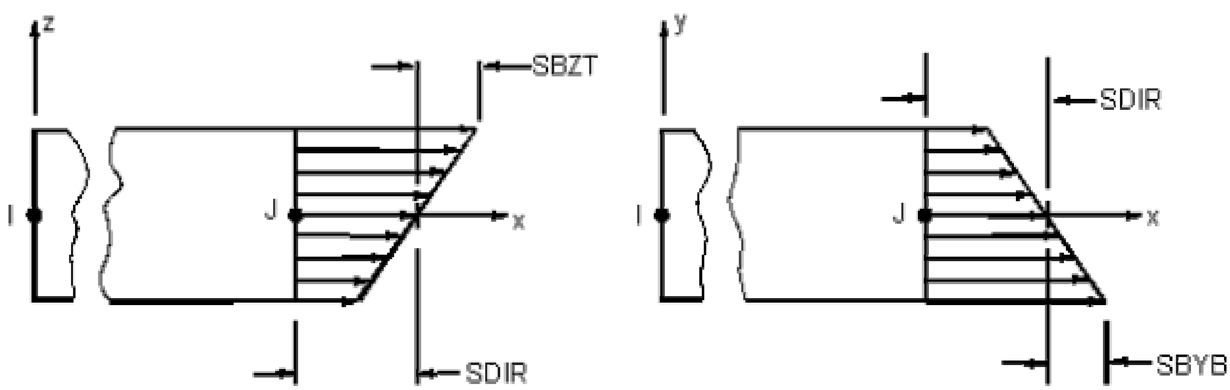
Input Data

The geometry, node locations, and coordinate systems for this element are shown in Figure 1. The element is defined by two or three nodes, the cross-sectional area, two area moments of inertia (IZZ and IYY), two thicknesses (TKY and TKZ), an angle of orientation (θ) about the element x-axis, the torsional moment of inertia (IXX), and the material properties. If IXX is not specified or is equal to 0.0, it is assumed equal to the polar moment of inertia (IYY+IZZ). IXX should be positive and is usually less than the polar moment of inertia. The element torsional stiffness decreases with decreasing values of IXX.

The element x-axis is oriented from node I toward node J. For the two-node option, the default ($\theta = 0^\circ$) orientation of the element y-axis is automatically calculated to be parallel to the global X-Y plane. Several orientations are shown in Figure 1. For the case where the element is parallel to the global Z axis (or within a 0.01 percent slope of it), the element y axis is oriented parallel to the global Y axis (as shown). For user control of the element orientation about the element x-axis, use the θ angle (THETA) or the third node option. If both are defined, the third node option takes precedence. The third node (K), if used, defines a plane (with I and J) containing the element x and z axes (as shown).

Output Data

Figure 14. 3-D BEAM4 Stress Output

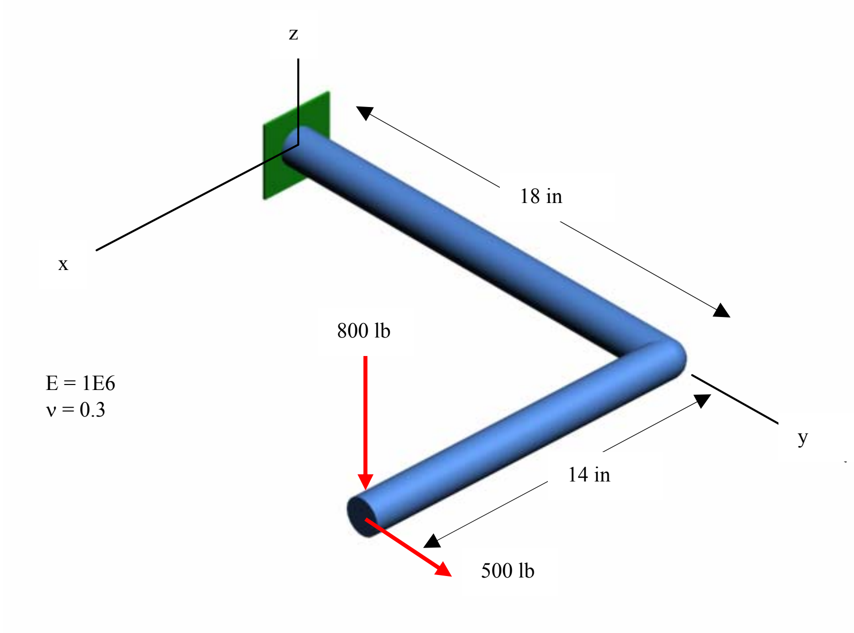


Assumptions and Restrictions

The beam must not have a zero length or area. The moments of inertia, however, may be zero if large deflections are not used. The beam can have any cross-sectional shape for which the moments of inertia can be computed. The stresses, however, will be determined as if the distance between the neutral axis and the extreme fiber is one-half of the corresponding thickness. The element thicknesses are used only in the bending and thermal stress calculations. The applied thermal gradients are assumed to be linear across the thickness in both directions and along the length of the element.

Example 1

The rod shown below has a radius of 0.75 in. If it is subjected to the loading shown, determine the state of stress.

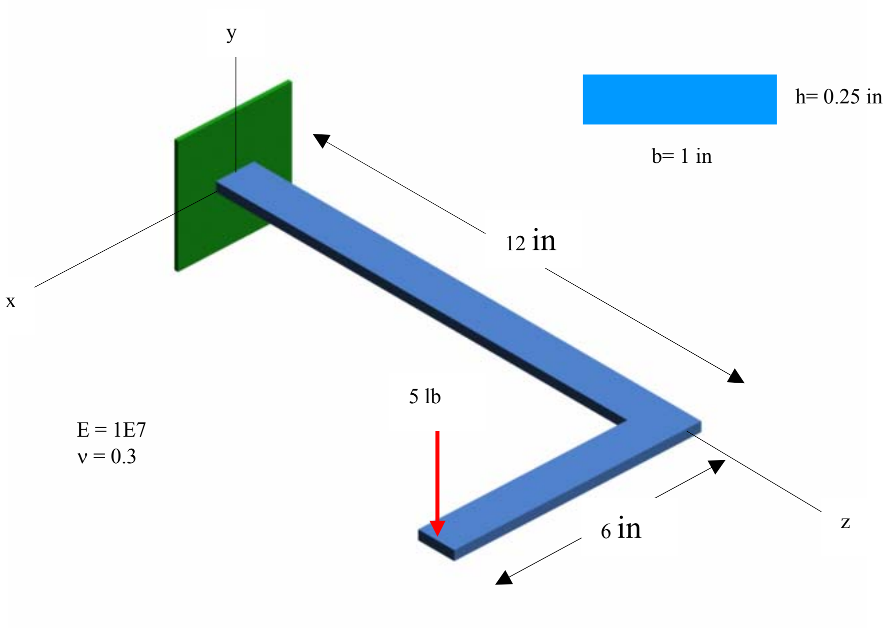


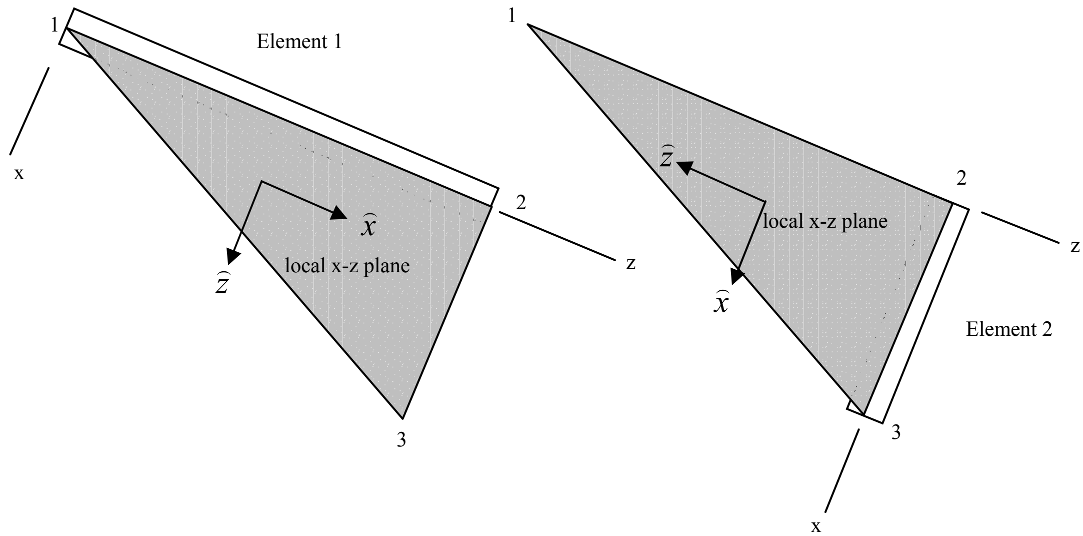

```
/prep7
pi = 4*atan(1)                !define value for pi
R = 0.75                      !radius
A = pi*R**2                   !cross-sectional area
Izz = 1/4*pi*R**4             !moment of inertia about z
Iyy = 1/4*pi*R**4             ! moment of inertia about y
Ixx = Izz+Iyy                 !moment of inertia about x (only valid for circular cross-section)
et,1,beam4                    !set element type to 3D elastic beam
mp,ex,1,1e6                   !young modulus
mp,prxy,1,0.3                 !poisson's ratio

r,1,A,Izz,Iyy,2*R,2*R,0,0,Ixx !geometric attribute: area,Izz,Iyy,z-thickness,y-thickness,orientation
                              !with x, initial strain, Ixx
k,1,0,0,0
k,2,0,18,0
k,3,14,18,0
l,1,2,18                      !create line between k1 and k2 with 18 division
l,2,3,14                      !create line between k2 and k3 with 14 division
lmesh,all                     !create element for all lines
dk,1,all,0                    !constrain dof at k1
fk,3,fy,500                   !apply load in y-direction at k3
fk,3,fz,-800                  !apply load in z-direction at k3
/pbc,f,1                      !display loads
/pbc,u,1                      !display dof
eplot
fini
```

```
/solu
solve
fini

/post1
etable,s1,ls,2                  !assign stress at +y = R to s1
etable,s2,ls,3                  !assign stress at –y = R to s2
etable,s3,ls,4                  !assign stress at +z = R to s3
etable,s4,ls,5                  !assign stress at –z = R to s4
/output,ex1,out                 !direct output to a file called ex1.out
pretab,s1                       !output s1
pretab,s2                       !output s2
pretab,s3                       !output s3
pretab,s4                       !out put s4
/output,,                       !switch output back to screen
fini
```

The next example shows the case when the cross-section of the beam is rectangle. Here, a local **x-z** planes are created for elements in order to have the correct orientation for I_{xx}, I_{yy}, and I_{zz}.

Example 2






```
/prep7

b = 1
h = 0.25
A = b*h
Izz = 1/12*b*h**3
Iyy = 1/12*h*b**3
Ixx = 1/12*b*h*(b**2+h**2)

et,1,beam4
mp,ex,1,1e7
mp,prxy,1,0.3

r,1,A,Izz,Iyy,b,h,0,0,Ixx

k,1,0,0,0
k,2,0,0,12
k,3,6,0,12

l,1,2,10                  !line 1
l,2,3,10                  !line 2

lsel,s,line,,1            !select line 1
latt,1,1,1,,3,3,1         !create x-z plane from line 1 to k3
lsel,s,line,,2            !select line 2
latt,1,1,1,,1,1,1         !create x-z plane from line 1 to k4

lsel,all                  !reselect all the lines

lmesh,all

dk,1,all,0
fk,3,fy,-5
/pbc,f,1
/pbc,u,1
eplot
fini
```