

Problem E

Steel Rods in Tension

Steel

$E=29000$ ksi, Poissons Ratio = 0.3

All members are 1.5" diameter steel rods.

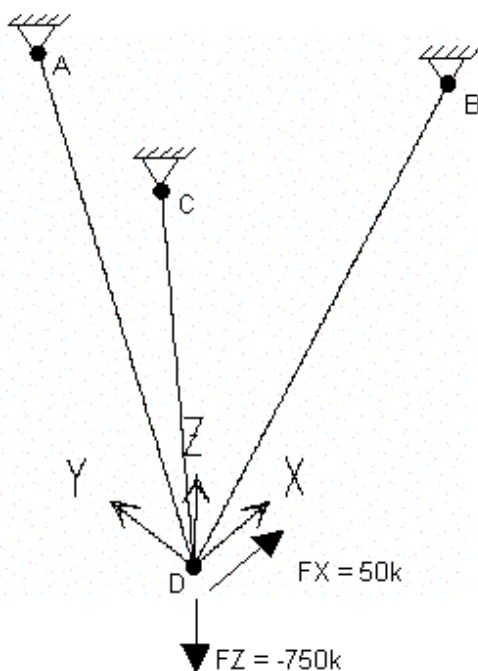
Joint Loads At Joint D:

$F_x = 50$ kips

$F_z = -750$ kips

To Do

Determine the X-direction displacements at Joint D with and without considering the stiffening effect of tension in the rods. Use P-Delta analysis to consider the stiffening effect.



**Joint Coordinates
(Feet)**

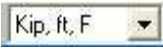
Joint	X	Y	Z
A	-3	-2	10
B	0	4	10
C	3	-2	10
D	0	0	0

CSI SOLUTION DEMONSTRATES USE OF THESE FEATURES

- Draw Special Joint
- Geometric Nonlinear P-Delta
- Move

PROBLEM E SOLUTION

1. Click the **File menu > New Model** command to display the **New Model** form.


2. Click the drop-down box to set the units to .



3. Click the **Grid Only** button to display the **New Coord/Grid System** form. In that form:

- Select the Cartesian Tab.
- In the *Number of Grid Lines* area, type **3** in the *X direction* edit box.
- In the *Number of Grid Lines* area, type **5** in the *Y direction* edit box.
- In the *Number of Grid Lines* area, type **2** in the *Z direction* edit box.
- In the *Grid Spacing* area, type **3** in the *X Direction* edit box.
- In the *Grid Spacing* area, type **2** in the *Y Direction* edit box.
- In the *Grid Spacing* area, type **10** in the *Z Direction* edit box.
- Click the **OK** button.

4. Click in the window entitled X-Y Plane @ Z=10 to make sure it is active. The window is active when the title is highlighted. The screen appears as shown in Figure E-1.

5. Click the **Draw Special Joint** button  or the **Draw menu > Draw Special Joint Draw** command to display the **Properties of Object** form.

6. Click on the grid intersection labeled "A" in Figure E-1 to enter a joint at (0, 2, 10).

7. Click on the grid intersection labeled "B" in Figure E-1 to enter a joint at (3, 8, 10).

8. Click on the grid intersection labeled "C" in Figure E-1 to enter a joint at (6, 2, 10).

9. Click the **Move Down in List** button  to move to the X-Y Plane @ Z=0.

10. Click on the grid intersection labeled "D" in Figure E-1 to enter a joint at (3, 4, 0).

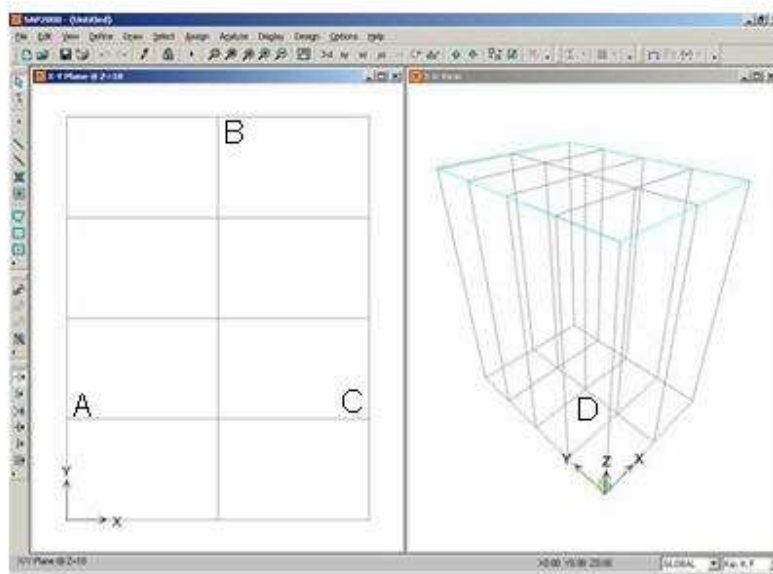



Figure E-1 Screen View Just Before Entering Joints At Elevation Z=10

11. Click the **Set Select Mode** button  on the side toolbar to exit the draw mode and enter the select mode.
12. Click the drop-down box in the status bar to change the units to .
13. Click the **Define menu > Materials** command to display the **Define Materials** form.
14. Click on STEEL in the *Materials* area to highlight (select) it, and then click the **Modify/Show Material** button to display the **Material Property Data** form. In that form:
 - Verify that the *Modulus of Elasticity* is 29000.
 - Verify that *Poisson's Ratio* is 0.3
 - Click the **OK** buttons on the **Material Property Data** and **Define Materials** forms to exit all forms.
15. Click the **Define menu > Frame Sections** command to display the **Frame Properties** form. In that form:
 - Click the drop-down box that reads *Add I/Wide Flange* and select the *Add Circle* option.
 - Click the **Add New Property** button in the *Click to:* area to display the **Circle Section** form. In that form:
 - Type **ROD** in the *Section Name* edit box.
 - Verify that the selected material in the *Material* drop-down box is STEEL.
 - Type **1.5** in the *Diameter (t3)* edit box.
 - Click the **OK** buttons on the **Circle Section** and **Frame Properties** forms to exit all forms.

16. Click in the window entitled 3-D View to activate it. The screen now appears as shown in Figure E-2.

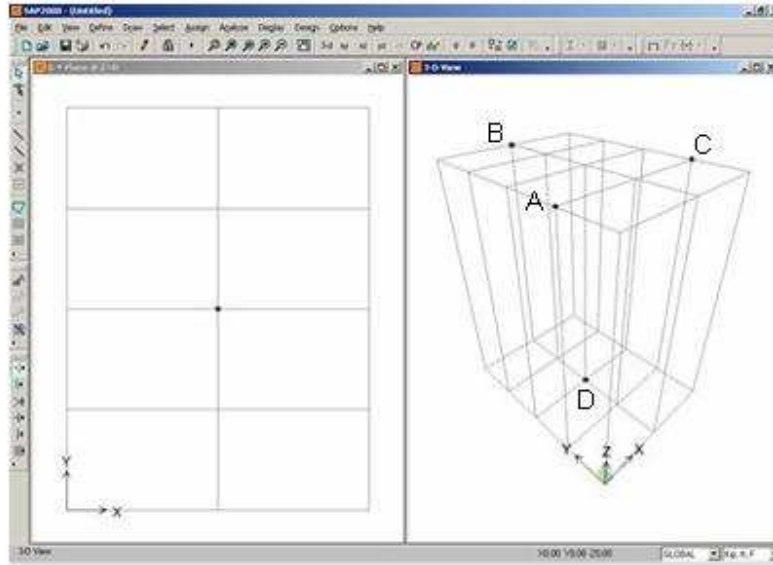



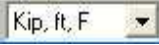


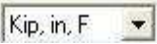



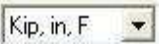



Figure E-2 Screen View After Joints Have Been Entered

17. Click the **Draw Frame/Cable/Tendon** button  or the **Draw menu > Draw Frame/Cable/Tendon** command to display the **Properties of Object** form. In that form:
- Click in the *Property* drop-down box, scroll up/down and click on the **CABLE** section.
 - In the *Moment Releases* drop-down box, click on **Pinned**.
18. Click on the point labeled “D” and then the point labeled “A” in Figure E-2 and press the Enter key on the keyboard to draw the first rod object.
19. Click on the point labeled “D” and then the point labeled “B” in Figure E-2 and press the Enter key on the keyboard to draw the next rod object.
20. Click on the point labeled “D” and then the point labeled “C” in Figure E-2 and press the Enter key on the keyboard to draw the last rod object.
21. Click the **Set Select Mode** button  to exit Draw Mode and enter Select Mode.
22. Click on the joints identified as A, B and C in Figure E-2 to select them.
23. Click the **Assign menu > Joint > Restraints** command to display the **Joint Restraints** form. In that form:
- Verify that the Translation 1, Translation 2 and Translation 3 boxes are checked.
 - Verify that the Rotation about 1, Rotation about 2 and Rotation about 3 boxes are checked.
 - Click the **OK** button.
24. Click on the joint identified as D in Figure E-2 to select it.

25. Click the **Assign menu > Joint > Restraints** command to display the **Joint Restraints** form. In that form:
 - Uncheck the boxes for Translation 1, Translation 2 and Translation 3.
 - Click the **OK** button.
26. Click the **Define menu > Load Cases** command to display the **Define Loads** form. In that form:
 - Type **VERT** in the *Load Name* edit box.
 - Type **0** in the *Self Weight Multiplier* edit box.
 - Click the **Modify Load** button
 - Type **LAT** in the *Load Name* edit box.
 - Select **QUAKE** from the *Type* drop-down box.
 - Click the **Add New Load** button.
 - Click the **OK** button.
27. Select the joint identified as D in Figure E-2 by clicking on it.
28. Click the **Assign menu > Joint Loads > Forces** command to display the **Joint Forces** form. In that form:
 - Verify VERT is selected in the *Load Case Name* drop-down box.
 - Type **-750** in the *Force Global Z* edit box.
 - Click the **OK** button.
29. Select the joint identified as D in Figure E-2 by clicking on it.
30. Click the **Assign menu > Joint Loads > Forces** command to display the **Joint Forces** form. In that form:
 - Select LAT from the *Load Case Name* drop-down box.
 - Type **50** in the *Force Global X* edit box.
 - Type **0** in the *Force Global Z* edit box.
 - Click the **OK** button.
31. Click the **Show Undeformed Shape** button  to remove the display of joint loads.
32. Click in the window titled 3-D View to activate it.
33. Click the drop-down box in the status bar to change the units to .
34. Click the **Select All** button  on the side toolbar.
35. Click the **Edit menu > Move** command to display the **Move** form. In that form:
 - Type **-3** in the *Delta X* edit box.
 - Type **-4** in the *Delta Y* edit box.
 - Click the **OK** button.
36. Click the **View menu > Show Grid** command to turn off the grid display in the 3-D view.

37. Click the **Define menu > Analysis Cases** to display the **Analysis Cases** form. In that form:
- Highlight (select) DEAD in the *Case Name* list box.
 - Click the **Modify/Show Case** button to display the **Analysis Case Data** form. In that form:
 - Type **VERT** in the *Analysis Case Name* edit box
 - Click the **OK** buttons on the **Analysis Case Data** and **Analysis Cases** forms to exit all of the forms.
38. Click the **Run Analysis** button  to display the **Set Analysis Cases to Run** form. In that form, click the **Run Now** button.
39. When the analysis is complete, check the messages in the **SAP Analysis Monitor** window and then click the **OK** button to close the window.
40. Click the drop-down box in the status bar to change the units to .
41. Click in the window with the 3-D View to make sure it is active.
42. Click the **Show Deformed Shape** button  (or the **Display menu > Show Deformed Shape** command) to display the **Deformed Shape** form. In that form:
- Select LAT from the *Case/Combo Name* drop-down box.
 - Click the **OK** button.
43. Right click on the bottom joint (the one labeled “D” in the problem statement) to see its displacement. Note the X-direction displacement of this joint. This is the displacement without considering the stiffening effect of the tension in the rods.
44. Click the **Lock/Unlock Model** button  to unlock the model. Click the **OK** button when asked if it is OK to delete.
45. Click the **Define menu > Analysis Cases** command to display the **Analysis Cases** form. In that form:
- Highlight (select) VERT in the *Case Name* list box.
 - Click the **Modify/Show Case** button to display the **Analysis Case Data** form. In that form:
 - In the *Analysis Type* area select the *Nonlinear* option.
 - In the *Other Parameters* area, click on the **Modify/Show** button for the *Nonlinear Parameters* edit box to display the **Nonlinear Parameters** form. In that form:
 - Select the *P-Delta* option in the *Geometric Nonlinearity Parameters* area.
 - Click the **OK** buttons on the **Nonlinear Parameters** and **Analysis Case Data** forms to return to the **Analysis Cases** form. In that form:
 - Highlight (select) LAT in the *Case Name* list box.
 - Click the **Modify/Show Case** button to display the **Analysis Case Data** form. In that form:

- In the *Stiffness to Use* area, select the *Stiffness at End of Nonlinear Case* option.
 - Verify that VERT is selected in the *Stiffness at End of Nonlinear Case* drop-down box.
 - Click the **OK** buttons on the **Analysis Case Data** and **Analysis Cases** forms to close all forms.
46. Click the **Run Analysis** button  to display the **Set Analysis Cases to Run** form. In that form, click the **Run Now** button.
47. When the analysis is complete, check the messages in the **SAP Analysis Monitor** window and then click the **OK** button to close the window.
48. Click the drop-down box in the status bar to change the units to .
49. Click in the window with the 3-D View to make sure it is active.
50. Click the **Show Deformed Shape** button  (or the **Display menu > Show Deformed Shape** command) to display the **Deformed Shape** form. In that form:
 - Select LAT from the *Case/Combo Name* drop-down box.
 - Click the **OK** button.
51. Right click on the bottom joint (the one labeled “D” in the problem statement) to view its displacement. This is the displacement with the stiffening effect of the tension in the rods considered.