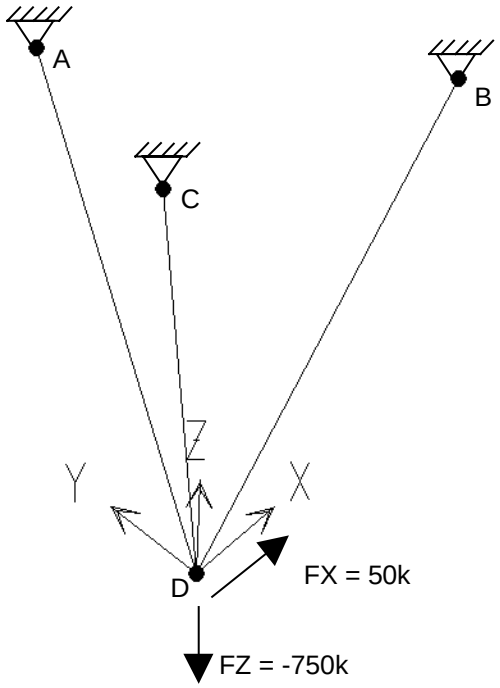
Problem E

Cables In Tension

Steel
E =29000 ksi, Poissons Ratio = 0.3
All members are 1.5" diameter steel cable

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 affect of tension in the cables. Use P-Delta analysis to consider the stiffening affect.



Joint Coordinates (Feet)			
Joint	X	Y	Z
A	-3	-2	10
B	0	4	10
C	3	-2	10
D	0	0	0

Note: Our intent is that you try this problem on your own first. After you have solved it on your own, you can step through our solution if desired. If you have problems trying to create the model, then follow the steps in our solution.

Problem E Solution

1. Click the drop down box in the status bar to change the units to kip-ft. 

2. From the **File** menu select **New Model…**. This displays the Coordinate System Definition dialog box.

3. In this dialog box

   - Select the Cartesian Tab.
   - In the Number of Grid Spaces area type **2** in the X direction edit box.
   - In the Number of Grid Spaces area type **4** in the Y direction edit box.
   - In the Number of Grid Spaces area type **1** 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 titled X-Y Plane @ Z=10 to make sure it is active. The window is highlighted when it is active. The screen appears as shown in Figure E-1.

5. Click the **Draw Special Joint** button  on the side toolbar or select **Add Special Joint** from the **Draw** menu.

6. Click on the grid intersection labeled "A" in Figure E-1 to enter a joint at (-3, -2, 10).

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

8. Click on the grid intersection labeled "C" in Figure E-1 to enter a joint at (3, -2, 10).

9. Click the **Down One Gridline** button  to move to the X-Y Plane @ Z=0.

10. Click on the origin to enter a joint at (0, 0, 0).

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

12. Click the **Draw Frame Element** button  on the side toolbar or select **Draw Frame Element** from the **Draw** menu.

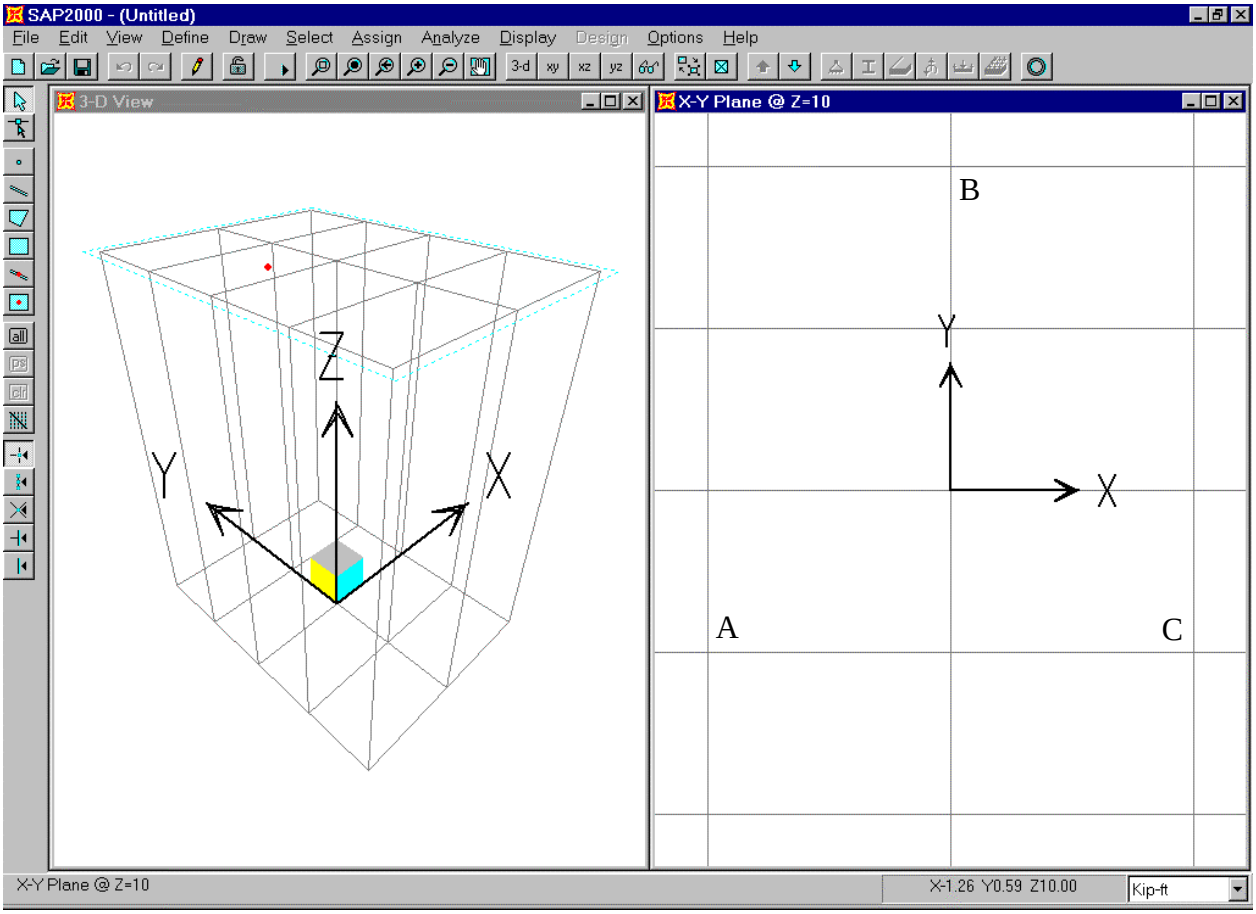


Figure E-1: Screen View Just Before Entering Joints At Elevation Z=10

13. 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 frame (cable) element.

14. 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 frame (cable) element.

15. 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 frame (cable) element.

16. Click the **Pointer** button  to exit Draw Mode and enter Select Mode.

17. Click on the joints identified as A, B and C in Figure E-2 to select them.

18. From the **Assign** menu, choose **Joint**, and then **Restraints…**from the submenu. This will display the Joint Restraints dialog box.

19. In this dialog box:

    - Verify that the Translation 1, Translation 2 and Translation 3 boxes are checked.

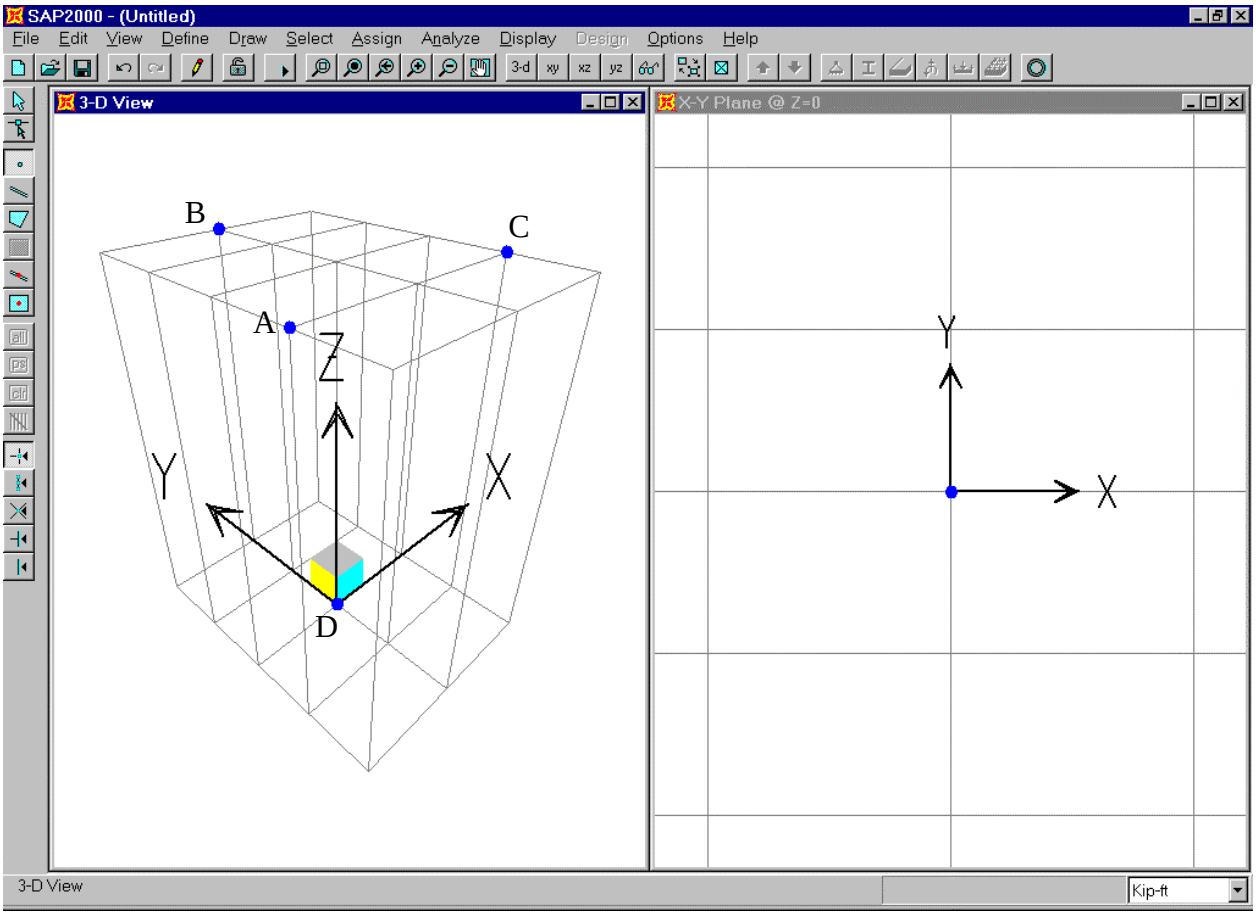


Figure E-2: Screen View After Joints Have Been Entered

- Verify that the Rotation About 1, Rotation About 2 and Rotation About 3 boxes are *not* checked.
- Click the **OK** button.

20. From the **Define** menu select **Static Load Cases…**. This will display the Define Static Load Case Names dialog box.

21. In this dialog box:

- Type **VERT** in the Load edit box.
- Type **0** in the Self weight Multiplier box.
- Click the **Change Load** button
- Type **LAT** in the Load edit box.
- Select QUAKE from the Type drop-down box.

- Click the **Add New Load** button.
- Click the **OK** button.

22. Select the joint identified as D in Figure E-2 by clicking on it.

23. From the **Assign** menu select **Joint Static Loads...** and then **Forces...** from the submenu to display the Joint Forces dialog box.

24. In this dialog box:

    - 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.

25. Select the joint identified as D in Figure E-2 by clicking on it.

26. From the **Assign** menu select **Joint Static Loads...** and then **Forces...** from the submenu to display the Joint Forces dialog box.

27. In this dialog box:

    - 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.

28. Click the **Show Undeformed Shape** button  to remove the display of joint static loads.

29. Click the drop down box in the status bar to change the units to kip-in. 

30. From the **Define** menu select **Materials...** to display the Define Materials dialog box.

31. Click on STEEL in the Materials area to highlight (select) it, and then click the **Modify/Show Material** button. The Material Property Data dialog box is displayed.

32. In this dialog box:

    - Verify that the Modulus of Elasticity is 29000.
    - Verify that Poisson's Ratio is 0.3
    - Click the **OK** button twice to exit all dialog boxes.

33. From the **Define** menu select **Frame Sections...** to display the Define Frame Sections dialog box.

34. In this dialog box:

    - Click the drop-down box that says Add I/Wide Flange and select the Add Circle option. This displays the Circle Section dialog box.

    - In this dialog box:

        - Type **CABLE** 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** button twice to exit all dialog boxes.

35. Select the three frame elements in the 3-D View window by clicking on them.

36. From the **Assign** menu select **Frame** and then **Sections...** from the submenu to display the Define Frame Sections dialog box.

37. In this dialog box:

    - Click on CABLE in the Frame Sections area to highlight it.

    - Click the **OK** button.

38. Click the **Show Undeformed Shape** button  to remove the display of frame sections.

39. Click the **Run Analysis** button  to run the analysis.

40. When the analysis is complete check the messages in the Analysis window (there should be no warnings or errors) and then click the **OK** button to close the Analysis window.

41. Click the drop down box in the status bar to change the units to kip-in. 

42. Click in the window with the 3-D View to make sure it is active.

43. Click the **Display Static Deformed Shape** button  (or select **Show Deformed Shape…** from the **Display** menu). The Deformed Shape dialog box appears.

44. In this dialog box:

    - Select LAT Load Case from the Load drop-down box.

    - Click the **OK** button.

45. 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 affect of the tension in the cables.

46. Click the **Lock/Unlock Model** button  on the main toolbar to unlock the model. Click the **OK** button when asked if it is OK to delete.

47. From the **Analyze** menu select **Set Options...** to display the Analysis Options dialog box.

48. In this dialog box:

    - Check the Include P-Delta check box.
    - Click the **Set P-Delta Parameters** button to display the P-Delta Parameters dialog box.
    - In this dialog box:
        - In the Iteration Controls area type **5** in the Maximum Iterations edit box.
        - Accept the other default values in the Iteration Controls area.
        - In the P-Delta Load Combination area verify that the Load Case list box says VERT and the Scale Factor list box says 1.
        - Click the **OK** button twice to exit all dialog boxes.

49. Click the **Run Analysis** button  to run the analysis.

50. When the analysis is complete check the messages in the Analysis window (there should be no warnings or errors) and then click the **OK** button to close the Analysis window.

51. Click the drop down box in the status bar to change the units to kip-in. 

52. Click in the window with the 3-D View to make sure it is active.

53. Click the **Display Static Deformed Shape** button  (or select **Show Deformed Shape…** from the **Display** menu). The Deformed Shape dialog box appears.

54. In this dialog box:

    - Select LAT Load Case from the Load drop-down box.
    - Click the **OK** button.

55. 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 with the stiffening affect of the tension in the cables considered. Notice the difference between this displacement and that noted in step 45.