

Problem V

Temperature Loading

Steel

$E = 29000$ ksi

Poissons Ratio = 0.3

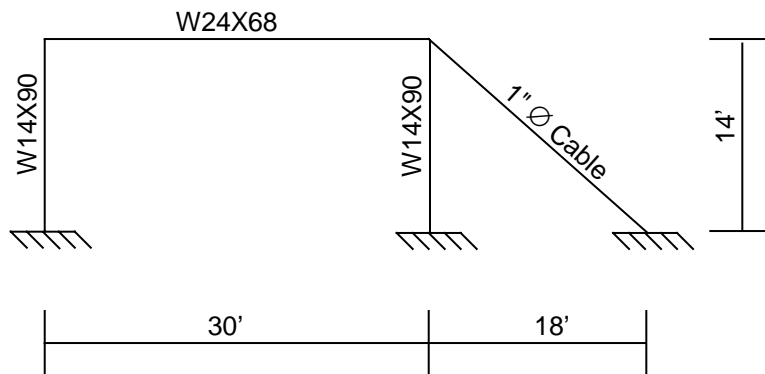
Coefficient of thermal expansion = 0.0000065 (degrees Fahrenheit)

Beam-column connections are rigid

Cable is pinned at each end


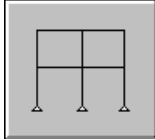


To Do






Determine support reactions and frame displacement due to a 100° Fahrenheit temperature drop in the cable only.





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.



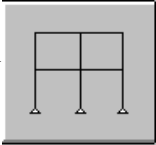




Problem V Solution

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 From Template...** This displays the Model Templates dialog box.
3. In this dialog box click on the **Portal Frame** template  button to display the Portal Frame dialog box.
4. In this dialog box:
 - Type **1** in the Number of Stories edit box.
 - Type **1** in the Number of Bays edit box.
 - Type **14** in the Story Height edit box.
 - Type **30** in the Bay Width edit box.
 - Click the **OK** button.
5. Click the “X” in the top right-hand corner of the 3-D View window to close it.
6. From the **Draw** menu select **Edit Grid...** to display the Modify Grid Lines edit box.
7. In this dialog box:
 - Verify that the X Direction option is selected.
 - Type **33** in the X Location edit box.
 - Click the **Add Grid Line** button.
 - Click the **OK** button.
8. Click the **xz 2D View** button  on the main toolbar to reset the view.
9. Click the **Set Elements** button  on the main toolbar (or select **Set Elements...** from the **View** menu) to display the Set Elements Dialog box.
10. In this dialog box:
 - Check the Labels box in the Joints area.
 - Check the Labels box in the Frames area.

- Click the **OK** button.
11. Verify that the **Snap to Joints and Grid Points** button  on the side tool bar is depressed.
 12. Click the **Draw Frame Element** button  on the side toolbar, or select **Draw Frame Element** from the **Draw** menu.
 13. Draw the cable element as follows:
 - Place the mouse pointer on joint 4. When the text box saying “Grid Intersection” appears click the left mouse button once.
 - Move the mouse pointer to to the grid intersection at the bottom of the cable (lower right-hand grid intersection). When the text box saying “Grid Intersection” appears click the left mouse button once.
 - Press the Enter key on the keyboard.
 14. Click the **Pointer** button  to exit draw mode and enter select mode.
 15. Select joints 1, 3 and 5.
 16. From the **Assign** menu, choose **Joint**, and then **Restraints...** from the submenu. This will display the Joint Restraints dialog box.
 17. In this dialog box:
 - Click the **fixed base fast restraint** button  to set all degrees of freedom (U1, U2, U3, R1, R2 and R3) as restrained.
 - Click the **OK** button.
 18. Click the drop down box in the status bar to change the units to kip-in. 
 19. From the **Define** menu select **Materials...** to display the Define Materials dialog box. Highlight the STEEL material and click the **Modify/Show Material** button to display the Material Property Data dialog box.
 20. In this dialog box:
 - Verify that the modulus of elasticity is 29000, poisson’s ratio is 0.3 and the coefficient of thermal expansion is 0.0000065.
 - Click the **OK** button twice to exit the dialog boxes.
 21. From the **Define** menu select **Frame Sections...** to display the Define Frame Sections dialog box.

22. In the Click To area, click the drop-down box that says Import I/Wide Flange and then click on the Import I/Wide Flange item.
23. If the Section Property File dialog box appears then locate the Sections.pro file which should be located in the same directory as the SAP2000 program files.
24. A dialog box appears with a list of all wide flange sections in the database. In this dialog box:
 - Scroll down and click on the W24X68 section.
 - Scroll down to the W14X90 section, and click on it while holding down the Ctrl key on the keyboard.
 - Click the **OK** button twice to return to the Define Frame Sections dialog box.
25. In the Click To area, click the drop-down box that says Add I/Wide Flange and then click on the Add Circle item.
26. The Circle Section dialog box appears. In this dialog box:
 - Type **CABLE** in the Section Name edit box.
 - Type **1** in the Diameter (t3) edit box.
 - Click the **OK** button twice to exit all dialog boxes.
27. Click the drop down box in the status bar to change the units to kip-ft. 
28. Select the beam element (frame element 3).
29. From the **Assign** menu select **Frame** and then **Sections...** from the submenu to display the Define Frame Sections dialog box.
30. In this dialog box:
 - Click on W24X68 in the Frame Sections area to highlight it.
 - Click the **OK** button.
31. Select the two column elements (frame elements 1 and 2).
32. From the **Assign** menu select **Frame** and then **Sections...** from the submenu to display the Define Frame Sections dialog box.
33. In this dialog box:
 - Click on W14X90 in the Frame Sections area to highlight it.

- Click the **OK** button.
34. Select the cable element (frame element 4).
 35. From the **Assign** menu select **Frame** and then **Sections...** from the submenu to display the Define Frame Sections dialog box.
 36. In this dialog box:
 - Click on **CABLE** in the Frame Sections area to highlight it.
 - Click the **OK** button.
 37. Click the **Show Undeformed Shape** button  to remove the displayed frame section assignments.
 38. Select the cable element (frame element 4).
 39. From the **Assign** menu select **Frame** and then **Releases...** from the submenu to display the Frame Releases dialog box.
 40. In this dialog box check both the Start and the End boxes for **Moment 33 (Major)** and then click the **OK** button.
 41. From the **Define** menu select **Static Load Cases...** to display the Define Static Load Case Names dialog box.
 42. In this dialog box:
 - Type **0** in the Self Weight Multiplier edit box.
 - Click the **Change Load** button.
 - Click the **OK** button.
 43. Select the cable element (frame element 4).
 44. From the **Assign** menu select **Frame Static Loads...** and then **Temperature...** from the submenu to display the Frame Temperature Loading dialog box.
 45. In this dialog box:
 - Verify that the **Temperature** option is selected in the Type area.
 - Verify that the **By Element** option is selected in the Temperature area.
 - Type **-100** in the Temperature edit box.
 - Click the **OK** button.

46. Click the **Show Undeformed Shape** button  to remove the displayed temperature assignments.
47. Click the **Set Elements** button  on the main toolbar (or select **Set Elements...** from the **View** menu) to display the Set Elements Dialog box.
48. In this dialog box:
- Uncheck the Labels box in the Joints area.
 - Uncheck the Labels box in the Frames area.
 - Click the **OK** button.
49. From the **Analyze** menu select **Set Options...** to display the Analysis Options dialog box.
- In this dialog box click the **Plane Frame XZ Plane** button  to set the available degrees of freedom.
 - Click the **OK** button.
50. Click the **Run Analysis** button  to run the analysis.
51. When the analysis is complete check the messages in the Analysis window (there should be no warnings or errors). Click the **OK** button to close the Analysis window.
52. Click the **Joint Reaction Forces** button  to display the Joint Reaction Forces dialog box.
53. In this dialog box:
- Verify that the Reactions option is selected in the Type area.
 - Click the **OK** button.
54. The reactions are displayed on the screen. You can right click on any joint to see the reactions at that joint or you can just read the reactions on the screen. If the text goes off of the screen, you can use click the **Pan** button  on the main toolbar to move the display, or you can click the **Zoom Out One Step** button  on the main toolbar as many times as required to resize the display. If the text is too small to read, you can zoom in on the joint, or you can change the minimum font size as described in the note below.

*Note: To change the minimum font size select **Preferences** from the **Options** menu and make sure the **Dimensions Tab** is selected. In the Minimum Graphic font Size edit box input a new size, maybe 5 or 6 points. Click the **OK** button.*