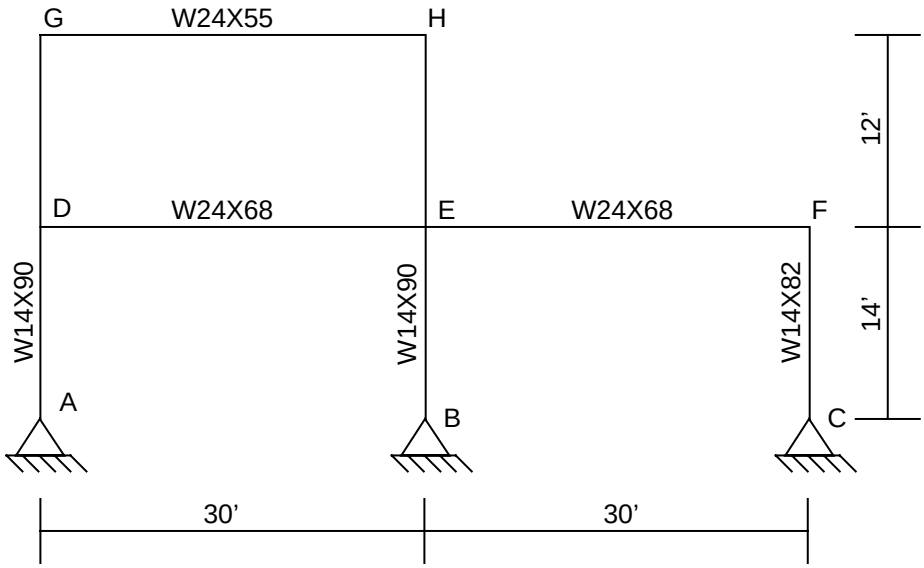
Problem G

Frame With Support Displacement

Steel
E =29000 ksi, Poissons Ratio = 0.3
Pinned base
All beam-column connections are rigid

To Do
Determine support reactions due to a
1" downward displacement of joint B.

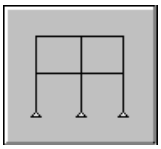


Note: Our intent is that you try this problem on your own first. After you have solved it on your own, you can step through our solution if desired. If you have problems trying to create the model, then follow the steps in our solution.

Problem G 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 **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. Click the drop down box in the status bar to change the units to kip-in. 

7. 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.

8. In this dialog box:

   - Verify that the modulus of elasticity is 29000 and poisson's ratio is 0.3.
   - Click the **OK** button twice to exit the dialog boxes.

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

10. Click the **Set Elements** button  on the main toolbar (or select **Set Elements…** from the **View** menu) to display the Set Elements Dialog box.

11. In this dialog box:

    - Check the Labels box in the Frames area.
    - Click the **OK** button.

12. Select frame elements 6 and 10. Press the delete key on the keyboard to delete these elements.

13. Click the **Refresh Window** button  to refresh the drawing.

14. From the **Draw** menu choose **Edit Grid...** to display the Modify Grid Lines dialog box.

15. In this dialog box:

    - Verify that the Z option is chosen in the Direction area.
    - Check the Glue Joints To Grid Lines box.
    - Click on the 28 grid line in the Z Location list box to highlight it. Note that the 28 appears in the Z Location edit box.
    - Change the 28 in the Z Location edit box to **26** and click the **Move Grid Line** button.
    - Click the **OK** button.

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

17. 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.

18. 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.

19. 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 W24X55 section, and click on it while holding down the Ctrl key on the keyboard.
    - Scroll down to the W14X90 section, and click on it while holding down the Ctrl key on the keyboard.
    - Click the **OK** button three times to exit all dialog boxes.

20. Select frame member 8.

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

22. In this dialog box:

    - Click on W24X55 in the Frame Sections area to highlight it.
    - Click the **OK** button.

23. Click the **Show Undeformed Shape** button  to remove the displayed frame section assignments so that you can see the frame element labels again.

24. Select frame members 7 and 9.

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

26. In this dialog box:

    - Click on W24X68 in the Frame Sections area to highlight it.
    - Click the **OK** button.

27. Click the **Show Undeformed Shape** button  to remove the displayed frame section assignments so that you can see the frame element labels again.

28. Select frame members 1 through 5.

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 W14X90 in the Frame Sections area to highlight it.
    - Click the **OK** button.

31. Click the **Show Undeformed Shape** button  to remove the displayed frame section assignments.

32. Click the **Set Elements** button  on the main toolbar (or select **Set Elements…** from the **View** menu) to display the Set Elements Dialog box.

33. In this dialog box:

    - Check the Labels box in the Joints area.
    - Uncheck the Labels box in the Frames area.
    - Click the **OK** button.

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

35. Select joint 4 (labeled B in the problem statement).

36. From the **Assign** menu select **Joint Static Loads...** and then **Displacements...** from the submenu to display the Ground Displacements dialog box.

37. In this dialog box:

    - Type **-1** in the Translation Z edit box.
    - Click the **OK** button.

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

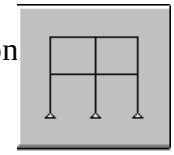
39. Click the **Show Undeformed Shape** button  to remove the displayed joint displacement assignments.

40. Click the **Set Elements** button  on the main toolbar (or select **Set Elements…** from the **View** menu) to display the Set Elements Dialog box.

41. In this dialog box:

    - Uncheck the Labels box in the Joints area.
    - Click the **OK** button.

42. 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.

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

44. 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.

45. Click the **Joint Reaction Forces** button  to display the Joint Reaction Forces dialog box.

46. In this dialog box:

    - Verify that the Reactions option is selected in the Type area.
    - Click the **OK** button.

47. 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 font size for the joint reactions shown on the screen is too small then read 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.*