

# **Problem K**

## **Steel Moment Frame**

### **Steel**

$E = 29000$  ksi, Poissons Ratio = 0.3

Pinned base

All beam-column connections are rigid

Beams: W24X55,  $F_y = 36$  ksi

Columns: W14X90,  $F_y = 36$  ksi

### **Beam Span Loading On All Beams**

1.0 klf Dead Load (not including steel frame member self weight)

0.5 klf Live Load

### **Lateral Loading (Earthquake)**

As indicated in the figure

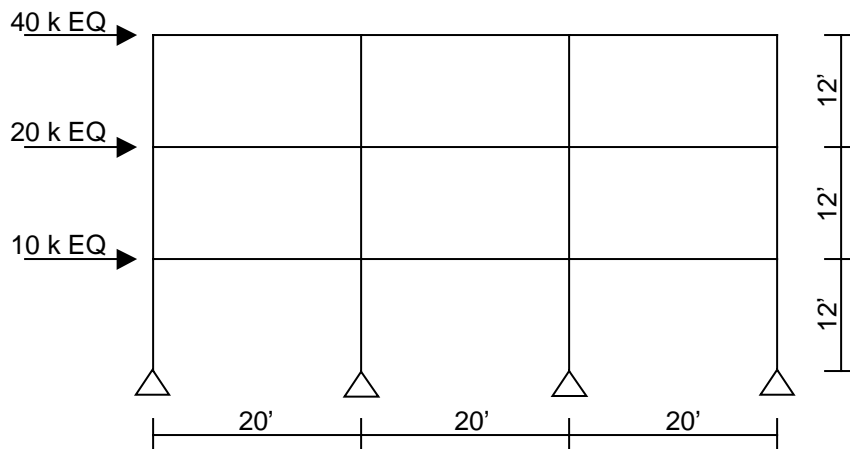
### **Unbraced Lengths**

Assume columns are laterally supported at each floor level

Assume beams are braced at 10 feet on center


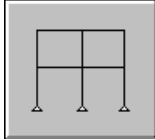


### **To Do**


Determine stress ratios using AISC-ASD89 due to DL, LL and EQ loads.







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.



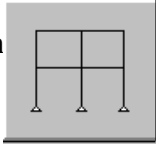


## Problem K Solution

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 **3** in the Number of Stories edit box.
  - Type **3** in the Number of Bays edit box.
  - Accept the default value of 12 in the Story Height edit box.
  - Type **20** 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.
8. 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.
9. In this dialog box:
  - Verify that the Modulus of Elasticity is 29000.
  - Verify that Poisson’s Ratio is 0.3
  - Verify that the Weight per Unit Volume is 2.830E-04.
  - Verify that the steel yield stress is 36.
  - Click the **OK** button twice to exit all dialog boxes.
10. Click the drop down box in the status bar to change the units to kip-ft. 
11. From the **Define** menu select **Frame Sections...** to display the Define Frame Sections dialog box.

12. 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.
13. 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. Highlight Sections.pro and click the **Open** button.
14. 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 W24X55 section.
  - 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.
15. Select all of the column elements by “windowing” each of the four column lines separately.
16. From the **Assign** menu select **Frame** and then **Sections...** from the submenu to display the Define Frame Sections dialog box.
17. In this dialog box:
  - Click on W14X90 in the Frame Sections area to highlight it.
  - Click the **OK** button.
18. Select all of the beam elements by using the intersecting line selection method on each of the three beam bays separately.
19. From the **Assign** menu select **Frame** and then **Sections...** from the submenu to display the Define Frame Sections dialog box.
20. In this dialog box:
  - Click on W24X55 in the Frame Sections area to highlight it.
  - Click the **OK** button.
21. Click the **Show Undeformed Shape** button  to remove the displayed frame section assignments.
22. From the **Define** menu select **Static Load Cases...** This will display the Define Static Load Case Names dialog box.
23. In this dialog box:

- Type **DL** in the Load edit box.
  - Click the **Change Load** button
  - Type **LL** in the Load edit box.
  - Select LIVE from the Type drop-down box.
  - Type **0** in the Self weight Multiplier box.
  - Click the **Add New Load** button.
  - Type **EQ** in the Load edit box.
  - Select QUAKE from the Type drop-down box.
  - Click the **Add New Load** button.
  - Click the **OK** button.
24. Click the **Restore Previous Selection** button  on the side toolbar (or select **Get Previous Selection** from the **Select** menu) to reselect the beam elements.
25. From the **Assign** menu select **Frame Static Loads...** and then **Point and Uniform...** from the submenu to display the Point and Uniform Span Loads dialog box.
26. In this dialog box:
- Verify that the Load Case Name is DL.
  - In the Load Type and Direction area verify that the Forces option is selected and that the Global Z direction is selected.
  - In the Uniform Load area type **-1**.
  - Click the **OK** button.
27. Click the **Restore Previous Selection** button  on the side toolbar (or select Get Previous Selection from the Select menu).
28. From the **Assign** menu select **Frame Static Loads...** and then **Point and Uniform...** from the submenu to display the Point and Uniform Span Loads dialog box.
29. In this dialog box:
- Select LL from the Load Case Name drop-down box.
  - In the Uniform Load area type **-.5**.

- Click the **OK** button.
30. Click the **Show Undeformed Shape** button  to remove the displayed frame uniform load assignments.
  31. Click the **Set Elements** button  on the main toolbar (or select **Set Elements...** from the **View** menu) to display the Set Elements Dialog box.
  32. In this dialog box:
    - Check the Labels box in the Joints area.
    - Click the **OK** button.
  33. Select joint 4 by clicking on it.
  34. From the **Assign** menu select **Joint Static Loads...** and then **Forces...** from the submenu to display the Joint Forces dialog box.
  35. In this dialog box:
    - Select EQ from the Load Case Name drop-down box.
    - Type **40** in the Force Global X edit box in the Loads area.
    - Click the **OK** button.
  36. Select joint 3 by clicking on it.
  37. From the **Assign** menu select **Joint Static Loads...** and then **Forces...** from the submenu to display the Joint Forces dialog box.
  38. In this dialog box:
    - Type **20** in the Force Global X edit box in the Loads area.
    - Click the **OK** button.
  39. Select joint 2 by clicking on it.
  40. From the **Assign** menu select **Joint Static Loads...** and then **Forces...** from the submenu to display the Joint Forces dialog box.
  41. In this dialog box:
    - Type **10** in the Force Global X edit box in the Loads area.
    - Click the **OK** button.

42. Click the **Show Undeformed Shape** button  to remove the displayed joint load assignments.
43. Click the **Set Elements** button  on the main toolbar (or select **Set Elements...** from the **View** menu) to display the Set Elements Dialog box.
44. In this dialog box:
  - Uncheck the Labels box in the Joints area.
  - Click the **OK** button.
45. 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.
46. Click the **Run Analysis** button  to run the analysis.
47. 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.
48. Click the **Show Undeformed Shape** button  to reset the displayed deformed shape.
49. Select all of the beam elements by using the intersecting line selection method on each of the three beam bays separately.
50. From the **Design** menu select **Redefine Element Design Data** to display the Element Overwrite Assignments dialog box.
51. In this dialog box:
  - Check the Unbraced Length Ratio (Minor, LTB) check box.
  - Type **.5** in the Unbraced Length Ratio (Minor, LTB) edit box.
  - Click the **OK** button.
52. From the **Options** menu select **Preferences...** to display the Preferences dialog box.
53. In this dialog box:
  - Click on the Steel Tab
  - Select AISC-ASD89 from the Steel Design Code drop-down box if it is not already selected.

- Click the **OK** button.
54. From the **Design** menu click **Start Design/Check Of Structure** to run the design check of the steel frame elements.
  55. When the design check completes, the stress ratios are displayed.