

## **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

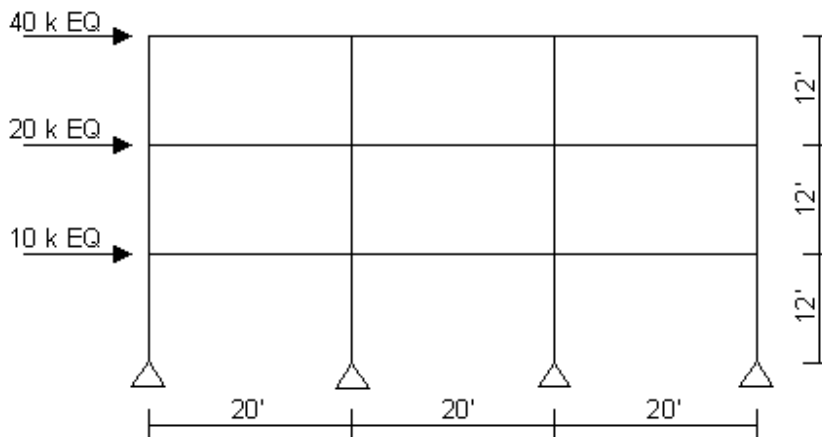
#### **Unbraced Lengths**

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.

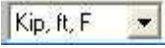


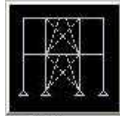
### **CSI Solution Demonstrates Use of These Features**


- New Model From Template
- Steel Design
- Unbraced Length Ratio

### **Problem K Solution**

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

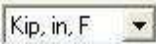
2. Click the drop-down box to set the units to .



3. Click the **2D Frame** button  to display the **2D Frame** form. In that form:

- Select *Portal* in the *2D Frame Type* drop-down list.
- 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.

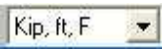
4. Click the “X” in the top right-hand corner of the 3-D View window to close it.

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

6. Click the **Define menu > Materials** command to display the **Define Materials** form.

7. Click **STEEL** in the *Materials* area to highlight it (select it), and then click the **Modify/Show Material** button to display the **Material Property Data** form. In that form:

- Verify that the *Weight per Unit Volume* is 2.836E-04.
- Verify that the *Modulus of Elasticity* is 29000.
- Verify that *Poisson’s Ratio* is 0.3
- Verify that the *Minimum Yield Stress* is 36.
- Click the **OK** buttons on the **Material Property Data** and **Define Materials** forms to close all forms.

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

9. Select all of the column objects by “windowing” each of the four column lines separately.






10. Click the **Assign menu > Frame/Cable/Tendon > Frame Sections** command to display the **Frame Properties** form. In that form:



- Click on *W14X90* in the *Properties* area to highlight it.
- Click the **OK** button.

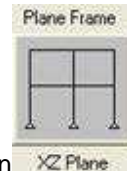
11. Select all of the beam objects using the intersecting line selection method on each of the three beam bays separately.

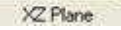

12. Click the **Assign menu > Frame/Cable/Tendon > Frame Sections** command to display the **Frame Properties** form. In that form:



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

13. Click the **Show Undeformed Shape** button  to remove the displayed frame section assignments.
14. Click the **Define menu > Load Cases** command to display the **Define Loads** form. In that form:
  - Type **LIVE** in the *Load Name* edit box.
  - Select *LIVE* from the *Type* drop-down box.
  - Click the **Add New Load** button.
  - Type **EQ** in the *Load Name* edit box.
  - Select *QUAKE* from the *Type* drop-down box.
  - Click the **Add New Load** button.
  - Click the **OK** button.
15. Click the **Get Previous Selection** button  on the side toolbar (or the **Select menu > Get Previous Selection** command) to reselect the beam objects.
16. Click the **Assign menu > Frame/Cable/Tendon Loads > Distributed** command to display the **Frame Distributed Loads** form. In that form:
  - Verify that the *Load Case Name* is *DEAD*.
  - In the *Load Type and Direction* area, verify that the *Forces* option is selected and that the *Gravity* direction is selected.
  - In the *Uniform Load* area, type **1**.
  - Click the **OK** button.
17. Click the **Get Previous Selection** button  on the side toolbar (or the **Select menu > Get Previous Selection** command) to reselect the beam objects.
18. Click the **Assign menu > Frame/Cable/Tendon Loads > Distributed** command to display the **Frame Distributed Loads** form. In that form:
  - Select *LIVE* from the *Load Case Name* drop-down box.
  - In the *Uniform Load* area type **.5**.
  - Click the **OK** button.
19. Click the **Show Undeformed Shape** button  to remove the displayed frame distributed load assignments.
20. Click the **Set Display Options** button  (or the **View menu > Set Display Options** command) to display the **Display Options for Active Window** form. In that form:
  - Check the *Labels* box in the *Joints* area.
  - Click the **OK** button.
21. Select joint 4 by clicking on it.
22. Click the **Assign menu > Joint Loads > Forces** command to display the **Joint Forces** form. In that form:

- 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.
23. Select joint 3 by clicking on it.
24. Click the **Assign menu > Joint Loads > Forces** command to display the **Joint Forces** form. In that form:
- Type **20** in the *Force Global X* edit box in the *Loads* area.
  - Click the **OK** button.
25. Select joint 2 by clicking on it.
26. Click the **Assign menu > Joint Loads > Forces** command to display the **Joint Forces** form. In that form:
- Type **10** in the *Force Global X* edit box in the *Loads* area.
  - Click the **OK** button.
27. Click the **Show Undeformed Shape** button  to remove the displayed joint load assignments.
28. Click the **Set Display Options** button  (or the **View menu > Set Display Options** command) to display the **Display Options for Active Window** form. In that form:
- Uncheck the *Labels* box in the *Joints* area.
  - Click the **OK** button.
29. Click the **Analyze menu > Set Analysis Options** command to display the **Analysis Options** form.



- In that form click the **Plane Frame XZ Plane** button  to set the available degrees of freedom.
  - Click the **OK** button.
30. Click the **Run Analysis** button  to display the **Set Analysis Cases to Run** form. In that form:
- Highlight (select) *MODAL* in the *Case Name* list and click the **Run/Do Not Run Case** button.
  - Verify that the *DEAD* analysis case is set to *Run* in the *Action* list.
  - Verify that the *LIVE* analysis case is set to *Run* in the *Action* list.
  - Verify that the *EQ* analysis case is set to *Run* in the *Action* list.
  - Click the **Run Now** button to run the analysis.

31. When the analysis is complete, check the messages in the **SAP Analysis Monitor** window (there should be no warnings or errors) and then click the **OK** button to close the window.
32. Click the **Show Undeformed Shape** button  to reset the displayed deformed shape.
33. Select all of the beam objects by using the intersecting line selection method on each of the three beam bays separately.
34. Click the **Design menu > Steel Frame Design > View/Revise Overwrites** command to display the **Steel Frame Design Overwrites for AISC-LFRD93** form. In that form:
  - Type **.5** in the *Unbraced Length Ratio (Minor, LTB) Value* edit box.
  - Click the **OK** button.
35. Click the **Clear Selection** button  on the side toolbar (or the **Select menu > Clear Selection** command).
36. Click the **Options menu > Preferences > Steel Frame Design** command to display the **Steel Frame Design Preferences for AISC-LFRD93** form. In that form:
  - Select *AISC-ASD89* from the *Design Code* drop-down list.
  - Click the **OK** button.
37. Click the **Design menu > Steel Frame Design > Start Design/Check of Structure** command to run the design check of the steel frame members.
38. When the design check completes, the steel P-M interaction ratios are displayed.