

Problem A

Concrete Wall and Steel Frame

Steel

$F_y = 36$ ksi, $E = 29500$ ksi, Poisson's Ratio = 0.3

Columns: W10X49, typical - pinned base

Beams: As noted, pinned ends except continuous over top of brace

Assume all W24X68 beams are braced at 1/3 points

Assume W16X36 beams braced at center only

Beams at concrete wall are not embedded in wall

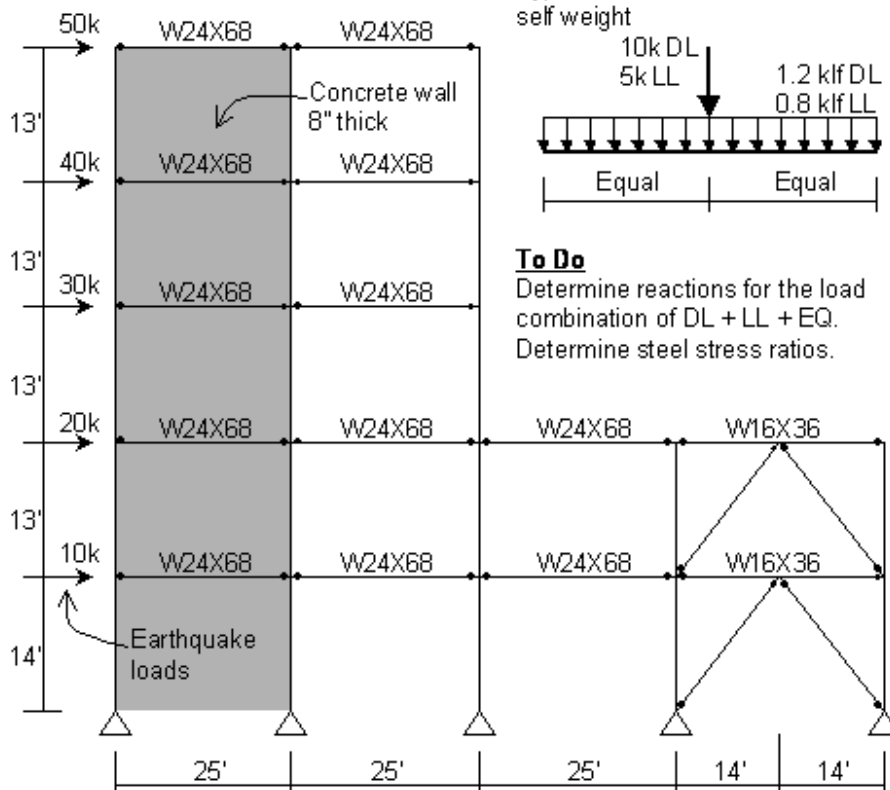
Braces: T86X6X1/4, pinned ends

Design Code: AISC-ASD89

Concrete

$E = 4000$ ksi, Poisson's Ratio = 0.22

Self weight = 150 pcf

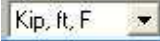


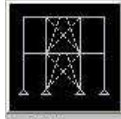
CSI Solution Demonstrates Use of These Features

- Grid lines
- Divide frames
- Frame releases
- Steel Design

Problem A Solution

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

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



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

- Select *Portal* in the *2D Frame Type* drop-down list.
- Type **5** in the *Number of Stories* edit box.
- Type **4** in the *Number of Bays* edit box.
- Type **13** in the *Story Height* edit box.
- Type **25** 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 that view.

5. Click the **Set Display Options** toolbar button  (or click 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.
- Check the *Labels* box in the *Frames/Cables* area.
- Click the **OK** button.

6. Select column objects 18, 19, 20, 23, 24 and 25 and beam objects 38, 39, 40, 43, 44 and 45. Press the delete key on the keyboard to delete those objects.

Note: Objects can be selected by clicking on each one individually, “windowing” over them, using the Intersecting Line Select Mode, or using the **Select menu > Select > Labels** command.



7. Click the **Define menu > Coordinate Systems/Grids** command to display the **Coordinate/Grid Systems** form.


8. Click on GLOBAL in the Systems list box to highlight it (select it), and then click the **Modify/Show System** button to display the **Define Grid Data for {Name} Coordinate System** form. In that form:

- Check the *Glue to Grid Lines* box in the right middle of the form.
- In the *X Grid Data* area, click in the *Ordinate* cell for *Grid ID x5* (it should read 50 now). Type **53** in the edit box.
- In the *Z Grid Data* area, click in the *Ordinate* cell for *Grid ID z1* (it should read 0 now). Type **-1** in the edit box.
- Click the **OK** button on the **Define Grid Data for {Name} Coordinate System** form and the **Coordinate/Grid Systems** form.


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

10. Select beams 41 and 42.
11. Click the **Edit menu > Divide Frames** command to display the **Divide Selected Frames** form.
12. Fill in the form as shown in the figure (typically the form will default to these values) and click the **OK** button.
13. Click the drop-down box in the status bar to change the units to **Kip, in, F**.
14. Click the **Define menu > Materials** command to display the **Define Materials** form.
15. Click on **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:
 - Type **0** in the *Mass per Unit Volume* edit box.
 - Type **0** in the *Weight per Unit Volume* edit box.
 - Type **29500** in the *Modulus of Elasticity* edit box.
 - Type **.3** in the *Poisson's Ratio* edit box, if it is not already entered.
 - Type **0** in the *Coeff of Thermal Expansion* edit box.
 - Type **36** in the *Minimum Yield Stress, Fy* edit box, if it is not already entered.
 - Type **58** in the *Minimum Tensile Stress, Fu* edit box.
 - Click the **OK** button.
16. Click on **CONC** 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:
 - Type **4000** in the *Modulus of Elasticity* edit box.
 - Type **.22** in the *Poisson's Ratio* edit box
 - Click the **OK** button to accept these values and the other values on the form.
17. Click the **OK** button to close the **Define Materials** form.
18. Click the drop-down box in the status bar to change the units to **Kip, ft, F**.
19. Click the **Define menu > Materials** command to display the **Define Materials** form.
20. Click on **CONC** 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 **0.15** is displayed in the *Weight per Unit Volume* edit box.
 - Click the **OK** buttons on the **Material Property Data** form and the **Define Materials** form.
21. Click the **Define menu > Frame Sections** command to display the **Frame Properties** form.
22. In the *Choose Property Type to Add* area, click the drop-down box that reads *Import I/Wide Flange*, highlight the *Import I/Wide Flange* item, and then click on the **Add New Property** button.




23. If the *Section Property File* form appears, 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.
24. A form appears with a list of all wide flange sections in the database. In that form:
 - Scroll down and click on the W16X36 section.
 - Click the **OK** button on the form and on the next form to return to the **Frame Properties** form.
25. In the *Choose Property Type to Add* area, click the drop-down box that reads *Import I/Wide Flange*, highlight the *Import Box/Tube* item, and then click on the **Add New Property** button.
26. A form appears with a list of all structural tube sections in the database. In that form:
 - Scroll up/down and click on the TS6X6X1/4 section.
 - Click the **OK** button on the **database** form, the **Box/Tube Section** form, and the **Frame Properties** form to exit all of forms.
27. Click the **Define menu > Area Sections** command to display the **Area Sections** form.
28. In the *Click To* area, click the **Add New Section** button to display the **Area Section Data** form. In that form:
 - Type **WALL** in the *Section Name* edit box.
 - In the *Thickness* area type **.6667** in both the *Membrane* and the *Bending* edit boxes.
 - Verify that the *Shell* option is selected in the *Type* area.
 - Click the **OK** button.
29. Click the **OK** button to close the **Area Sections** form.
30. Verify that the **Snap to Points and Grid Intersections** button  is depressed.
31. Click the **Draw Frame/Cable/Tendon Element** button  or select the **Draw menu > Draw Frame/Cable/Tendon** command to display the **Properties of Object** form. In that form:
 - Click in the *Property* cell to display a drop-down list. Scroll up/down and click on the TS6X6X1/4 section to assign it to the line objects that you will draw.
 - In the *Moment Releases* drop-down list, click on Pinned.
32. Draw the first brace object as follows:
 - Place the mouse pointer on joint 19. When the text box reading "Point" appears, click the left mouse button once.
 - Move the mouse pointer to joint 31. When the text box reading "Point" appears, click the left mouse button once.
 - Press the Enter key on the keyboard.
33. Click on joint 25 and then joint 31, and press the Enter key to draw the second brace element.
34. Click on joint 20 and then joint 32, and press the Enter key to draw the third brace element.



35. Click on joint 26 and then joint 32, and press the Enter key to draw the fourth and final brace element.
36. Click the **Set Select Mode** button  to exit Draw mode and enter Select mode.
37. Select all of the beams except for the braced frame beams (i.e., select beams 26 through 37 and do not select beams 46 through 49). The Intersecting Line Selection option could be useful for this.


Note: To use the Intersecting Line Selection option, click the **Select Using Intersecting**

Line button . Then click the left mouse button at the top of one beam bay, and while holding down the left mouse button, drag the mouse to the bottom of the beam bay. A “rubberband line” will appear and all objects that this “rubberband line” passes through will be selected. Release the left mouse button to make the selection.



38. Click the **Assign menu > Frame/Cable/Tendon > Releases/Partial Fixity** command to display the **Assign Frame Releases** form. In that form, check both the *Start* and the *End* boxes for Moment 33 (Major) and then click the **OK** button.
39. Select beam objects 46 and 48.
40. Click the **Assign menu > Frame/Cable/Tendon > Releases/Partial Fixity** command to display the **Assign Frame Releases** form. In that form, check the *Start* box for Moment 33 (Major) and then click the **OK** button.
41. Select beam objects 47 and 49.
42. Click the **Assign menu > Frame/Cable/Tendon > Releases/Partial Fixity** command to display the **Assign Frame Releases** form. In that form, check the *End* box for Moment 33 (Major) and then click the **OK** button.
43. Click the **Define menu > Load Cases** command to display the **Define Loads** form. In that form:
 - Type **LL** in the *Load Name* edit box.
 - Select *Live* from the *Type* drop-down box.
 - Type **0** in the *Self Weight Multiplier* edit 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.
44. Click the **Define menu > Combinations** command to display the **Define Response Combinations** form. In that form:
 - Click the **Add New Combo** button to display the **Response Combination Data** form. In that form:
 - Type **ALL** in the *Response Combination Name* edit box.
 - Select *Linear Add* from the *Combination Type* drop-down box if it is not already selected.
 - If not already set, select the *DEAD* load case in the *Case Name* drop-down box and type **1** in the *Scale Factor* edit box.

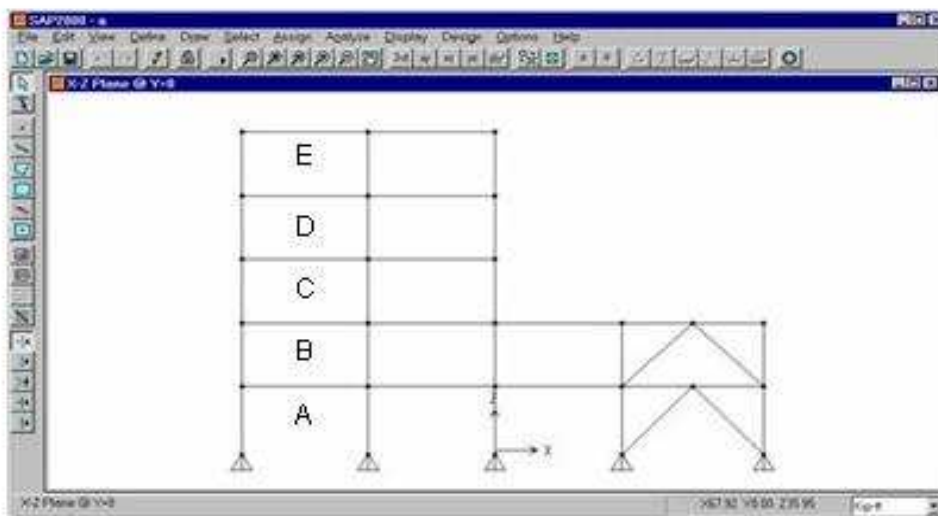
- Click the **Add** button.
 - Select the *LL* load case in the *Case Name* drop-down box.
 - Click the **Add** button.
 - Select the *EQ* load case in the *Case Name* drop-down box.
 - Click the **Add** button.
 - Click the **OK** button on the **Response Combination Data** form and the **Define Response Combinations** form.
45. Select beams 26 through 37.
46. Click the **Assign menu > Frame/Cable/Tendon Loads > Point** command to display the **Frame Point Loads** form. In that form:
- Select *DEAD* from the *Load Case Name* drop-down box.
 - In the *Load Type* and *Direction* area, make sure that *Forces* is selected and that *Gravity (-Z)* shows in the *Direction* drop-down box.
 - In the *Point Loads* area type **.5** in the first *Distance* edit box and type **10** in the first *Load* edit box.
 - Click the **OK** button.
47. Click the **Get Previous Selection** button  (or the **Select menu > Get Previous Selection** command).
48. Click the **Assign menu > Frame/Cable/Tendon Loads > Distributed** command to display the **Frame Distributed Loads** form. In that form:
- Select *DEAD* from the *Load Case Name* drop-down box.
 - In the *Load Type* and *Direction* area, make sure that *Forces* is selected and that *Gravity (-Z)* shows in the *Direction* drop-down box.
 - Type **1.2** in the *Uniform Load* area *Load* edit box.
 - Click the **OK** button.
49. Click the **Get Previous Selection** button  (or the **Select menu > Get Previous Selection** command).
50. Click the **Assign menu > Frame/Cable/Tendon Loads > Point** command to display the **Frame Point Loads** form. In that form:
- Select *LL* from the *Load Case Name* drop-down box.
 - In the *Point Loads* area type **5** in the first *Load* edit box
 - Click the **OK** button.
51. Click the **Get Previous Selection** button  (or click the **Select menu > Get Previous Selection** command).
52. Click the **Assign menu > Frame/Cable/Tendon Loads > Distributed** command to display the **Frame Distributed Loads** form. In that form:
- Select *LL* from the *Load Case Name* drop-down box.


- In the *Load Type* and *Direction* area, make sure that *Forces* is selected and that *Gravity (-Z)* shows in the *Direction* drop-down box.
 - Type **.8** in the *Uniform Load* area *Load* edit box.
 - Click the **OK** button.
53. Select beams 46 through 49.
54. Click the **Assign menu > Frame/Cable/Tendon Loads > Distributed** command to display the **Frame Distributed Loads** form. In that form:
- Select *DEAD* from the *Load Case Name* drop-down box.
 - Type **1.2** in the *Uniform Load* area *Load* edit box.
 - Click the **OK** button.
55. Click the **Get Previous Selection** button  (or click the **Select menu > Get Previous Selection** command).
56. Click the **Assign menu > Frame/Cable/Tendon Loads > Distributed** command to display the **Frame Distributed Loads** form. In that form:
- Select *LL* from the *Load Case Name* drop-down box.
 - Type **.8** in the *Uniform Load* area *Load* edit box.
 - Click the **OK** button.
57. Select joints 31 and 32.
58. Click the **Assign menu > Joint Loads > Forces** command to display the **Joint Forces** form. In that form:
- Select *DEAD* from the *Load Case Name* drop-down box.
 - Type **-10** in the *Force Global Z* edit box in the *Loads* area.
 - Click the **OK** button.
59. Click the **Get Previous Selection** button  (or the **Select menu > Get Previous Selection** command).
60. Click the **Assign menu > Joint Loads > Forces** command to display the **Joint Forces** form. In that form:
- Select *LL* from the *Load Case Name* drop-down box.
 - Type **-5** in the *Force Global Z* edit box in the *Loads* area.
 - Click the **OK** button.
61. Select joints 2, 3, 4, 5 and 6 by “windowing.”
62. 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 **10** in the *Force Global X* edit box in the *Loads* area.
 - Type **0** in the *Force Global Z* edit box in the *Loads* area.
 - Click the **OK** button.





63. Select joints 3, 4, 5 and 6 (not 2) by “windowing.”
64. Click the **Assign menu > Joint Loads > Forces** command to display the **Joint Forces** form. In that form:
 - Verify *Add to Existing Loads* is selected in the *Options* area.
 - Click the **OK** button.
65. Select joints 4, 5 and 6 (not 2 and 3) by “windowing.”
66. Click the **Assign menu > Joint Loads > Forces** command to display the **Joint Forces** form. In that form:
 - Verify *Add to Existing Loads* is selected in the *Options* area.
 - Click the **OK** button.
67. Select joints 5 and 6 (not 2, 3 and 4).
68. Click the **Assign menu > Joint Loads > Forces** command to display the **Joint Forces** form. In that form:
 - Verify *Add to Existing Loads* is selected in the *Options* area.
 - Click the **OK** button.
69. Select joint 6 (not 2, 3, 4 and 5).
70. Click the **Assign menu > Joint Loads > Forces** command to display the **Joint Forces** form. In that form:
 - Verify *Add to Existing Loads* is selected in the *Options* area.
 - Click the **OK** button.
71. Select beams 26 through 37.
72. Click the **Assign menu > Frame/Cable/Tendon > Frame Sections** command to display the **Frame Properties** form. In that form:
 - Scroll down and click on W24X68 in the *Properties* area to highlight it.
 - Click the **OK** button.
73. Click the **Show Undeformed Shape** button  to remove the displayed frame section assignments and display the frame object labels again.
74. Select beams 46 through 49.
75. Click the **Assign menu > Frame/Cable/Tendon > Frame Sections** command to display the **Frame Properties** form. In that form:
 - Scroll down and click on W16X36 in the *Properties* area to highlight it.
 - Click the **OK** button.
76. Select all of the columns. An easy way to do this is to “window” each of the column lines separately.

Note: To “window” a column line, left click the mouse above and to the left of the column line. While holding the left mouse button down, drag the mouse so that it is below and to the right of the column line. A “rubberband window” will appear surrounding the column line. Release the left mouse button to select all objects that are fully enclosed by the “rubberband window.”

77. Click the **Assign menu > Frame/Cable/Tendon > Frame Sections** command to display the **Frame Properties** form. In that form:
- Scroll down and click on W10X49 in the *Properties* area to highlight it.
 - Click the **OK** button.
78. Click the **Show Undeformed Shape** button  to remove the displayed frame section assignments.
79. Click the **Set Display Options** button  (or select 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.
 - Uncheck the *Labels* box in the *Frames/Cables/Tendon* area.
 - Click the **OK** button.
80. Click the **Quick Draw Area Element** button  to display the **Properties of Object** form. In that form:
- Scroll down and select WALL from the *Property* drop-down box.
81. Click the mouse pointer once in the areas labeled A, B, C, D and E in the figure below to draw the area objects.



82. Click the **Set Select Mode** button  to exit Draw mode and enter Select mode.
83. Select the five area objects just entered by clicking on them.
84. Click the **Assign menu > Area > Automatic Area Mesh** command to display the **Assign Automatic Area Mesh** form.
85. Fill out that form as shown in the figure to mesh each area object into twelve elements (4 by 3) and click the **OK** button.

86. Click the **Set Display Options** button  (or select the **View menu > Set Display Options** menu) to display the **Display Options for Active Window** form. In that form:
- Check the *Not in View* box in the *Joints* area.
 - Check the *Sections* box in the *Frames/Cables/Tendon* area.
 - Check the *Not in View* box in the *Areas* area.
 - Click the **OK** button.
87. Select all of the W24X68 beam sections (12 total).
88. Click the **Design menu > Steel Frame Design > View/Revise Overwrites** command to display the **Steel Frame Design Overwrites** form. In that form:
- Select the *Unbraced Length Ratio (Minor, LTB)* box and type **.3333** in the associated edit box.
 - Click the **OK** button.
89. Click the **Clear Selection** button  (or click the **Select menu > Clear Selection** command).
90. Select all of the W16X36 beam sections (4 total).
91. Click the **Design menu > Steel Frame Design > View/Revise Overwrites** command to display the **Steel Frame Design Overwrites** form. In that form:
- Select the *Unbraced Length Ratio (Minor, LTB)* box and type **.5** in the associated edit box.
 - Click the **OK** button.
92. Click the **Set Display Options** button  (or select the **View menu > Set Display Options** command) to display the **Display Options for Active Window** form. In that form:
- Uncheck the *Not in View* box in the *Joints* area.
 - Check the *Restraints* box in the *Joints* area.
 - Uncheck the *Sections* box in the *Frames/Cables/Tendon* area.
 - Uncheck the *Not in View* box in the *Areas* area.
 - Click the **OK** button.
93. Click the **Options menu > Preferences > Steel Frame Design** command to display the **Steel Frame Design Preferences** form. In that form:
- Select *AISC-ASD89* from the *Design Code* drop-down box if it is not already selected.
 - Click the **OK** button.
94. 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 and click the **OK** button.
95. Click the **Run Analysis** button  to display the **Set Analysis Cases to Run** form. In that form:

- Click on *Modal* in the *Case Name* list to highlight it.
 - Click the **Run/Do Not Run Case** button.
 - Click the **Run Now** button.
96. 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.
97. Click the **Display menu > Show Forces/Stresses > Joints** command to display the **Joint Reaction Forces** form. In that form:
- Select *ALL* from the *Case/Combo Name* drop-down box.
 - Verify that the *Reactions* option is selected in the *Type* area.
 - Click the **OK** button.
98. The reactions are displayed on the screen. If the text is too small to read, zoom in, or change the minimum font size as described in the following Note.
- Note:** To change the minimum font size, click the **Options menu > Preferences > Dimensions/Tolerances** command. In the *Minimum Graphic Font Size* edit box input a new size, for example 5 or 6 points. Click the **OK** button.
99. Click the **Design menu > Steel Frame Design > Start Design/Check of Structure** command to run the design check of the steel frame objects.
100. When the design check completes, a color map of the stress ratios is displayed.