

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

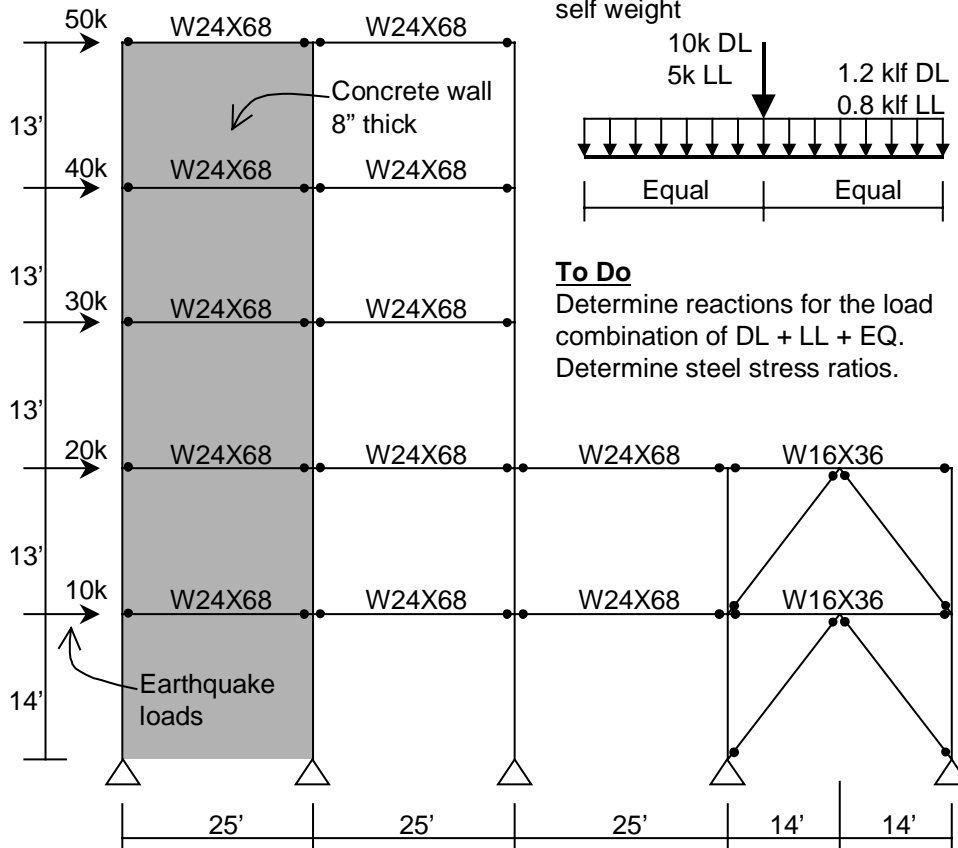
Braces: TS6X6X1/4, pinned ends

Design Code: AISC-ASD89

Concrete


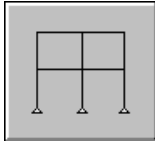

$E = 4000$ ksi, Poissons Ratio = 0.22


Self weight = 150 pcf



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 A Solution

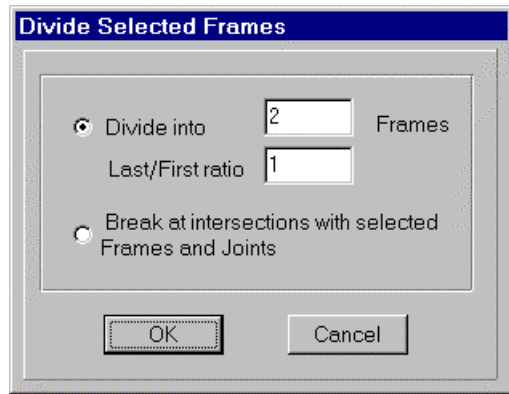
1. Click the drop down box in the status bar to change the units to kip-ft. 
2. From the **File** menu select **New Model From Template...** This displays the Model Templates dialog box.
3. In this dialog box click on the **Portal Frame** template  button to display the Portal Frame dialog box.
4. In this dialog box:
 - Type **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.
5. Click the “X” in the top right-hand corner of the 3-D View window to close it.
6. Click the **Set Elements** button  on the main toolbar (or select **Set Elements...** from the **View** menu) to display the Set Elements Dialog box.
7. In this dialog box:
 - Check the Labels box in the Joints area.
 - Check the Labels box in the Frames area.
 - Click the **OK** button.
8. Select column elements 18, 19, 20, 23, 24 and 25 and beam elements 38, 39, 40, 43, 44 and 45. Press the delete key on the keyboard to delete these elements.

Note: You could select the elements by clicking on each one individually, by “windowing”, by using the Intersecting Line Select Mode, or by using the Select By Labels option (Select menu > Select > Labels).
9. Click the **Refresh Window** button  to refresh the drawing.
10. From the **Draw** menu choose **Edit Grid...** to display the Modify Grid Lines dialog box.
11. In this dialog box:


- Verify that the X option is chosen in the Direction area.
- Check the Glue Joints To Grid Lines box.
- Click on the 50 grid line in the X Location list box to highlight it. Note that the 50 appears in the X Location edit box.
- Change the 50 in the X Location edit box to **53** and click the **Move Grid Line** button.
- Select the Z option in the Direction area.
- Click on the 0 grid line in the Z Location list box to highlight it. Note that the 0 appears in the Z Location edit box.
- Change the 0 in the Z Location edit box to **-1** and click the **Move Grid Line** button.
- Click the **OK** button.


12. Select beams 41 and 42.

13. From the **Edit** menu select **Divide Frames...** to display the Divide Selected Frames dialog box.



14. Fill in the dialog box as shown in the figure (typically the dialog box will default to these values) and click the **OK** button.

15. Verify that the **Snap to Joints and Grid Points** button  on the side tool bar is depressed.




16. Click the **Draw Frame Element** button  on the side toolbar, or select **Draw Frame Element** from the **Draw** menu.

17. Draw the first brace element as follows:

- Place the mouse pointer on joint 19. When the text box saying “Grid Intersection” appears click the left mouse button once.
- Move the mouse pointer to joint 31. When the text box saying “Point” appears click the left mouse button once.
- Press the Enter key on the keyboard.


18. Click on joint 25 and then joint 31, and press the Enter key to draw the second brace element.

19. Click on joint 20 and then joint 32, and press the Enter key to draw the third brace element.


20. Click on joint 26 and then joint 32, and press the Enter key to draw the fourth and final brace element.
21. Click the **Pointer** button  on the side tool bar to exit draw mode and enter select mode.
22. Click the drop down box in the status bar to change the units to kip-in. 
23. From the **Define** menu select **Materials...** to display the Define Materials dialog box.
24. 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.
25. In this dialog box:
 - 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 **36** in the Steel Yield Stress, Fy edit box, if it is not already entered.
 - Click the **OK** button.
26. Click on CONC in the Materials area to highlight (select) it, and then click the **Modify/Show Material** button. The Material Property Data dialog box is displayed.
27. In this dialog box:
 - Type **4000** in the Modulus of Elasticity edit box.
 - Type **.22** in the Poisson's Ratio edit box
 - Accept the other values in the dialog box.
 - Click the **OK** button.
28. Click the **OK** button to close the Define Materials dialog box.
29. Click the drop down box in the status bar to change the units to kip-ft. 
30. From the **Define** menu select **Materials...** to display the Define Materials dialog box.
31. Click on CONC in the Materials area to highlight (select) it, and then click the **Modify/Show Material** button. The Material Property Data dialog box is displayed.
32. In this dialog box:



- Verify **0.15** is entered in the Weight per Unit Volume edit box.
 - Click the **OK** button.
33. From the **Define** menu select **Frame Sections...** to display the Define Frame Sections dialog box.
 34. 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.
 35. 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.
 36. 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 W16X36 section, and click on it while holding down the Ctrl key on the keyboard.
 - Scroll down to the W10X49 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.
 37. In the Click To area, click the drop-down box that says Import I/Wide Flange and then click on the Import Box/Tube item.
 38. A dialog box appears with a list of all structural tube sections in the database. In this dialog box:
 - Scroll down and click on the TS6X6X1/4 section.
 - Click the **OK** button three times to exit all dialog boxes.
 39. From the **Define** menu select **Shell Sections...** to display the Define Shell Sections dialog box.
 40. In the Click To area, click the **Add New Section** button to display the Shell Sections dialog box.
 41. In this dialog box:
 - 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.
42. Click the **OK** button to close the Define Shells dialog box.
 43. 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 **Set Intersecting Line Select Mode** button  on the side tool bar. 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 elements that this “rubberband line” passes through will be selected. Release the left mouse button to make the selection.*
 44. From the **Assign** menu select **Frame** and then **Releases...** from the submenu to display the Frame Releases dialog box.
 45. In this dialog box check both the Start and the End boxes for Moment 33 (Major) and then click the **OK** button.
 46. Select beam elements 46 and 48.
 47. From the **Assign** menu select **Frame** and then **Releases...** from the submenu to display the Frame Releases dialog box.
 48. In this dialog box check the Start box for Moment 33 (Major) and then click the OK button.
 49. Select beam elements 47 and 49.
 50. From the **Assign** menu select **Frame** and then **Releases...** from the submenu to display the Frame Releases dialog box.
 51. In this dialog box check the End box for Moment 33 (Major) and then click the OK button.
 52. Select all of the braces (i.e., select braces 50 through 53).
 53. From the **Assign** menu select **Frame** and then **Releases...** from the submenu to display the Frame Releases dialog box.
 54. In this dialog box check both the Start and the End boxes for Moment 33 (Major) and then click the OK button.
 55. From the **Define** menu select **Static Load Cases...** to display the Define Static Load Case Names dialog box.
 56. 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 edit 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.
57. From the **Define** menu select **Load Combinations...** to display the Define Load Combinations dialog box.
58. In this dialog box:
- Click the **Add New Combo** button to display the Load Combination Data dialog box.
 - In this dialog box:
 - Type **ALL** in the Load Combination Name edit box.
 - Select ADD from the Load Combination Type drop-down box if it is not already selected.
 - Type **DL + LL + EQ** in the Title edit box.
 - Select DL Load Case in the Case Name drop down box (if it is not already selected) and type **1** in the Scale Factor edit box (if it is not already there).
 - Click the **Add** button.
 - Select LL Load Case in the Case Name drop down box.
 - Click the **Add** button.
 - Select EQ Load Case in the Case Name drop down box.
 - Click the **Add** button.



- Click the **OK** button twice.
59. Select beams 26 through 37.
60. 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.
61. In this dialog box:
- Select DL from the Load Case Name 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
 - Type **-1.2** in the Uniform Load area edit box.
 - Click the **OK** button.
62. Click the **Restore Previous Selection** button  on the side toolbar (or select **Get Previous Selection** from the **Select** menu).
63. 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.
64. In this dialog box:
- Select LL from the Load Case Name drop-down box.
 - In the Point Loads area type **-5** in the first Load edit box
 - Type **-.8** in the Uniform Load area edit box.
 - Click the **OK** button.
65. Select beams 46 through 49.
66. 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.
67. In this dialog box:
- Select DL from the Load Case Name drop-down box.
 - In the Point Loads area type **0** in the first Distance edit box and type **0** in the first Load edit box
 - Type **-1.2** in the Uniform Load area edit box.

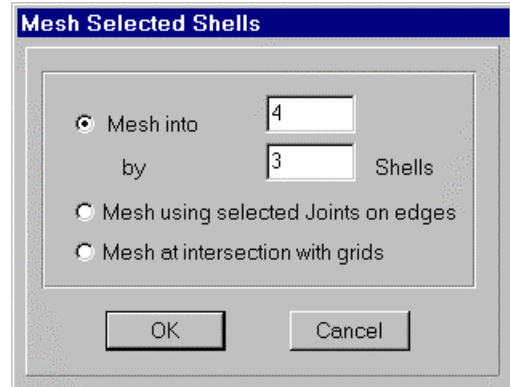
- Click the **OK** button.
68. Click the **Restore Previous Selection** button  on the side toolbar (or select Get Previous Selection from the Select menu).
 69. From the **Assign** menu select **Frame Static Loads...** and then **Point and Uniform...** from the submenu to display the Point and Uniform Span Loads dialog box.
 70. In this dialog box:
 - Select LL from the Load Case Name drop-down box.
 - Type **-.8** in the Uniform Load area edit box.
 - Click the **OK** button.
 71. Select joints 31 and 32.
 72. From the **Assign** menu select **Joint Static Loads...** and then **Forces...** from the submenu to display the Joint Forces dialog box.
 73. In this dialog box:
 - Select DL 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.
 74. Click the **Restore Previous Selection** button  on the side toolbar (or select Get Previous Selection from the Select menu).
 75. From the **Assign** menu select **Joint Static Loads...** and then **Forces...** from the submenu to display the Joint Forces dialog box.
 76. In this dialog box:
 - 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.
 77. Select joints 2, 3, 4, 5 and 6 by “windowing”.
 78. From the **Assign** menu select **Joint Static Loads...** and then **Forces...** from the submenu to display the Joint Forces dialog box.
 79. In this dialog box:


- 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.
80. Select joints 3, 4, 5 and 6 (not 2) by “windowing”.
81. From the **Assign** menu select **Joint Static Loads...** and then **Forces...** from the submenu to display the Joint Forces dialog box.
82. In this dialog box:
- Verify Add To Existing Loads is selected in the Options area.
 - Click the **OK** button.
83. Select joints 4, 5 and 6 (not 2 and 3) by “windowing”.
84. From the **Assign** menu select **Joint Static Loads...** and then **Forces...** from the submenu to display the Joint Forces dialog box.
85. In this dialog box click the **OK** button.
86. Select joints 5 and 6 (not 2, 3 and 4).
87. From the **Assign** menu select **Joint Static Loads...** and then **Forces...** from the submenu to display the Joint Forces dialog box.
88. In this dialog box click the **OK** button.
89. Select joint 6 (not 2, 3, 4 and 5).
90. From the **Assign** menu select **Joint Static Loads...** and then **Forces...** from the submenu to display the Joint Forces dialog box.
91. In this dialog box click the **OK** button.
92. Select beams 26 through 37.
93. From the **Assign** menu select **Frame** and then **Sections...** from the submenu to display the Define Frame Sections dialog box.
94. In this dialog box:
- Click on W24X68 in the Frame Sections area to highlight it.


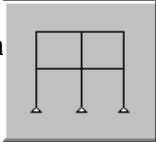


- Click the **OK** button.
95. Click the **Show Undeformed Shape** button  to remove the displayed frame section assignments so that you can see the frame element labels again.
 96. Select beams 46 through 49.
 97. From the **Assign** menu select **Frame** and then **Sections...** from the submenu to display the Define Frame Sections dialog box.
 98. In this dialog box:
 - Click on W16X36 in the Frame Sections area to highlight it.
 - Click the **OK** button.
 99. 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 elements that are fully enclosed by the “rubberband window”.
 100. From the **Assign** menu select **Frame** and then **Sections...** from the submenu to display the Define Frame Sections dialog box.
 101. In this dialog box:
 - Click on W10X49 in the Frame Sections area to highlight it.
 - Click the **OK** button.
 102. Select the four brace elements.
 103. From the **Assign** menu select **Frame** and then **Sections...** from the submenu to display the Define Frame Sections dialog box.
 104. In this dialog box:
 - Click on TS6X6X1/4 in the Frame Sections area to highlight it.
 - Click the **OK** button.
 105. Click the **Show Undeformed Shape** button  to remove the displayed frame section assignments.


114. Click the **Show Undeformed Shape** button  to remove the displayed shell section assignments.
115. Click the **Restore Previous Selection** button  on the side toolbar (or select Get Previous Selection from the Select menu).
116. From the **Edit** menu select **Mesh Shells...** to display the Mesh Selected Shells dialog box.
117. Fill out this dialog box as shown in the figure to mesh each shell into twelve elements (4 by 3) and click the **OK** button.



118. Click the **Set Elements** button  on the main toolbar (or select **Set Elements...** from the **View** menu) to display the Set Elements Dialog box.
119. In this dialog box:
 - Check the Hide box in the Shells area.
 - Check the Hide box in the Joints area.
 - Check the Sections box in the Frames area.
 - Click the **OK** button.
120. Select all of the W24X68 beam sections (12 total).
121. From the **Design** menu select **Redefine Element Design Data...** to display the Element Overwrite Assignments dialog box.
122. In this dialog box:
 - Check the Unbraced Length Ratio (Minor, LTB) box and type **.3333** in the associated edit box.
 - Click the **OK** button.
123. Select all of the W16X36 beam sections (4 total).
124. From the **Design** menu select **Redefine Element Design Data...** to display the Element Overwrite Assignments dialog box.
125. In this dialog box:
 - Check the Unbraced Length Ratio (Minor, LTB) box and type **.5** in the associated edit box.

- Click the **OK** button.
126. Click the **Set Elements** button  on the main toolbar (or select **Set Elements...** from the **View** menu) to display the Set Elements Dialog box.
127. In this dialog box:
- Uncheck the Hide box in the Joints area.
 - Check the Restraints box in the Joints area.
 - Uncheck the Sections box in the Frames area.
 - Uncheck the Hide box in the Shells area.
 - Click the **OK** button.
128. From the **Options** menu select **Preferences...** to display the Preferences dialog box.
129. 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.
130. 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.
131. Click the **Run Analysis** button  to run the analysis.
132. 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.
133. Click the **Joint Reaction Forces** button  on the main toolbar to display the Joint Reaction Forces dialog box.
134. In this dialog box:
- Select All Combo from the Load drop-down box.
 - Verify that the Reactions option is selected in the Type area.

- Click the **OK** button.
135. 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 is too small to read, you can zoom in, 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. Click the **Refresh Window** button  located on the main toolbar to see the changes.*

136. From the **Design** menu click **Start Design/Check Of Structure** to run the design check of the steel frame elements.
137. When the design check completes, the stress ratios are displayed.