

# Problem B

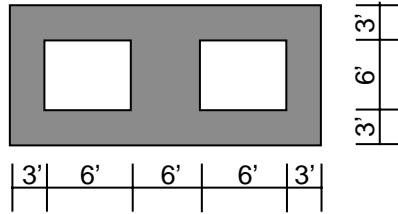
## Concrete Wall

### Concrete

$E = 3600 \text{ ksi}$ , Poissons Ratio = 0.2

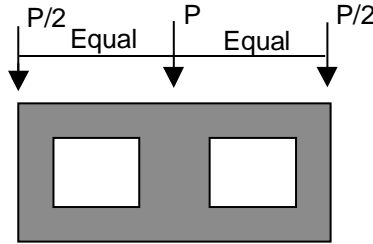
### To Do

Model wall using shell elements. Determine shear axial force and moment in Pier A, and determine total shear, moment and axial force at the sixth level for all piers combined.

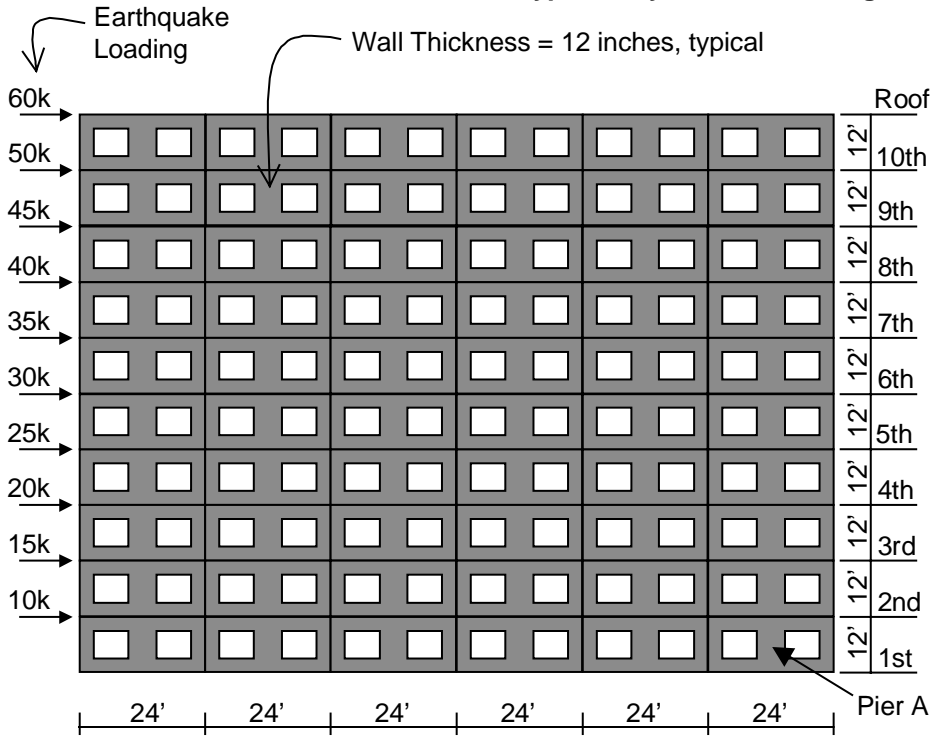


**Typical Bay Dimensions**

$P \text{ (DL)} = 21.6 \text{ k}$ ,  $P \text{ (LL)} = 7.2 \text{ k}$


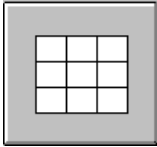




**Typical Bay Vertical Loading**








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 B 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 **Shear Wall** template  button to display the Shear Wall dialog box.
4. In this dialog box
  - Type **8** in the Number of Spaces Along X edit box.
  - Type **4** in the Number of Spaces Along Z edit box.
  - Type **3** Space Width Along X edit box.
  - Type **3** Space Width Along Z 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 Shells area.
  - Click the **OK** button.
8. Select shell elements 6, 7, 10, 11, 22, 23, 26 and 27 by clicking on them.
9. Press the Delete key on the keyboard to delete these elements.
10. Click the **Refresh Window** button  to refresh the drawing.
11. From the **Define** menu select **Static Load Cases...** to display the Define Static Load Case Names dialog box.
12. 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.
13. From the **Define** menu select **Load Combinations...** to display the Define Load Combinations dialog box.
14. 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.


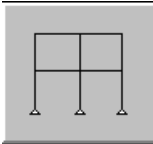

15. Select joints 10, 25 and 45.
16. From the **Assign** menu select **Joint Static Loads...** and then **Forces...** from the submenu to display the Joint Forces dialog box.
17. In this dialog box:
  - Select DL from the Load Case Name drop-down box.
  - Type **-10.8** in the Force Global Z edit box in the Loads area.
  - Click the **OK** button.
18. Select joint 25.
19. From the **Assign** menu select **Joint Static Loads...** and then **Forces...** from the submenu to display the Joint Forces dialog box. Click the **OK** button in this dialog box.
20. Select joints 10, 25 and 45.
21. From the **Assign** menu select **Joint Static Loads...** and then **Forces...** from the submenu to display the Joint Forces dialog box.
22. In this dialog box:
  - Select LL from the Load Case Name drop-down box.
  - Type **-3.6** in the Force Global Z edit box in the Loads area.
  - Click the **OK** button.
23. Select joint 25.
24. From the **Assign** menu select **Joint Static Loads...** and then **Forces...** from the submenu to display the Joint Forces dialog box. Click the **OK** button in this dialog box.
25. Click the drop down box in the status bar to change the units to kip-in. 
26. From the **Define** menu select **Materials...** to display the Define Materials dialog box. Highlight the CONC material and click the **Modify/Show Material** button to display the Material Property Data dialog box.
27. In this dialog box:
  - Verify that the modulus of elasticity is 3600 and poisson's ratio is 0.2.
  - Click the **OK** button twice to exit the dialog boxes.
28. Click the drop down box in the status bar to change the units to kip-ft. 

29. From the **Define** menu select **Shell Sections...** to display the Define Shell Sections dialog box. Click the **Modify/Show Section** button to display the Shell Sections dialog box.
30. In this dialog box:
  - Accept all of the default values.
  - Click the **OK** button twice to exit the dialog boxes.
31. Click the **Select All** button  on the side tool bar.
32. From the **Edit** menu select **Replicate...** to display the Replicate dialog box.
33. In this dialog box:
  - Click the Linear Tab if it is not already selected.
  - In the Distance area type **24** in the X edit box.
  - Type **5** in the Number edit box.
  - Click the **OK** button
34. Click the **Set Elements** button  on the main toolbar (or select **Set Elements...** from the **View** menu) to display the Set Elements Dialog box.
35. In this dialog box:
  - Check the Hide box in the Joints area.
  - Uncheck the Labels box in the Shells area.
  - Click the **OK** button.
36. Click the **Select All** button  on the side tool bar.
37. From the **Edit** menu select **Replicate...** to display the Replicate dialog box.
38. In this dialog box:
  - Click the Linear Tab if it is not already selected.
  - In the Distance area type **0** in the X edit box.
  - Type **12** in the **Z** edit box.
  - Type **9** in the Number edit box.
  - Click the **OK** button.

39. Click the **Set Elements** button  on the main toolbar (or select **Set Elements...** from the **View** menu) to display the Set Elements Dialog box.
40. In this dialog box:
  - Uncheck the Hide box in the Joints area.
  - Check the Labels box in the Joints area.
  - Check the Restraints box in the Joints area.
  - Check the Fill Elements box in the Options area.
  - Click the **OK** button.
41. Select joint 10. You may need to zoom in to distinguish it.
42. From the **Assign** menu select **Joint Static Loads...** and then **Forces...** from the submenu to display the Joint Forces dialog box.
43. 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.
44. Select joint 243.
45. From the **Assign** menu select **Joint Static Loads...** and then **Forces...** from the submenu to display the Joint Forces dialog box.
46. In this dialog box:
  - Type **15** in the Force Global X edit box in the Loads area.
  - Click the **OK** button.
47. Select joint 427.
48. From the **Assign** menu select **Joint Static Loads...** and then **Forces...** from the submenu to display the Joint Forces dialog box.
49. In this dialog box:
  - Type **20** in the Force Global X edit box in the Loads area.

- Click the **OK** button.
50. Select joint 611.
  51. From the **Assign** menu select **Joint Static Loads...** and then **Forces...** from the submenu to display the Joint Forces dialog box.
  52. In this dialog box:
    - Type **25** in the Force Global X edit box in the Loads area.
    - Click the **OK** button.
  53. Select joint 795.
  54. From the **Assign** menu select **Joint Static Loads...** and then **Forces...** from the submenu to display the Joint Forces dialog box.
  55. In this dialog box:
    - Type **30** in the Force Global X edit box in the Loads area.
    - Click the **OK** button.
  56. Select joint 979.
  57. From the **Assign** menu select **Joint Static Loads...** and then **Forces...** from the submenu to display the Joint Forces dialog box.
  58. In this dialog box:
    - Type **35** in the Force Global X edit box in the Loads area.
    - Click the **OK** button.
  59. Select joint 1163.
  60. From the **Assign** menu select **Joint Static Loads...** and then **Forces...** from the submenu to display the Joint Forces dialog box.
  61. In this dialog box:
    - Type **40** in the Force Global X edit box in the Loads area.
    - Click the **OK** button.
  62. Select joint 1347.

63. From the **Assign** menu select **Joint Static Loads...** and then **Forces...** from the submenu to display the Joint Forces dialog box.
64. In this dialog box:
  - Type **45** in the Force Global X edit box in the Loads area.
  - Click the **OK** button.
65. Select joint 1531.
66. From the **Assign** menu select **Joint Static Loads...** and then **Forces...** from the submenu to display the Joint Forces dialog box.
67. In this dialog box:
  - Type **50** in the Force Global X edit box in the Loads area.
  - Click the **OK** button.
68. Select joint 1715.
69. From the **Assign** menu select **Joint Static Loads...** and then **Forces...** from the submenu to display the Joint Forces dialog box.
70. In this dialog box:
  - Type **60** in the Force Global X edit box in the Loads area.
  - Click the **OK** button.
71. Click the **Show Undeformed Shape** button  to remove the displayed joint force assignments.
72. Click the **Set Elements** button  on the main toolbar (or select **Set Elements...** from the **View** menu) to display the Set Elements Dialog box.
73. In this dialog box:
  - Check the Labels box in the Shells area.
  - Click the **OK** button.
74. Zoom in on the pier labeled Pier A in the problem statement.
75. Select joints 208, 212 and 218.
76. Select shell elements 138 and 142.

77. From the **Assign** menu select **Group Name...** to display the Assign Group dialog box.
78. In this dialog box:
- Type **PIERA** in the Groups edit box.
  - Click the **Add New Group Name** button.
  - Click the **OK** button.
79. From the **View** menu select **Restore Full View**.
80. Select all joints level with the bottom of the sixth floor windows by “windowing” (joints 972, 973, 980, etc., 49 joints total).
81. Select all shell elements level with the bottom half of the sixth floor windows by using the intersecting line selection method (shell elements 730, 738, 742, etc., 24 shells total).
82. From the **Assign** menu select **Group Name...** to display the Assign Group dialog box.
83. In this dialog box:
- Type 6TH in the Groups edit box.
  - Click the **Add New Group Name** button.
  - Click the **OK** button.
84. Click the **Set Elements** button  on the main toolbar (or select **Set Elements...** from the **View** menu) to display the Set Elements Dialog box.
85. In this dialog box:
- Uncheck the Labels box in the Joints area.
  - Uncheck the Labels box in the Shells area.
  - Click the **OK** button.
86. 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.
87. Click the **Run Analysis** button  to run the analysis.

88. 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.
89. From the **Display** menu select **Show Group Joint Force Sums** to display the Select Groups dialog box.
90. In this dialog box:
  - Click on the group named 6TH to highlight it.
  - Hold down the Ctrl key on the keyboard and click on the group named PIERA to add it to the selection
  - Click the **OK** button.
91. When finished viewing the group joint force sums press the “X” in the upper right-hand corner of the Group joint Force Summation window to close it.