

Problem N

Frame-Shear Wall Interaction

Concrete Material Properties

$E = 3600$ ksi, Poissons Ratio = 0.2

Frame

Beams: 12" wide by 24" deep

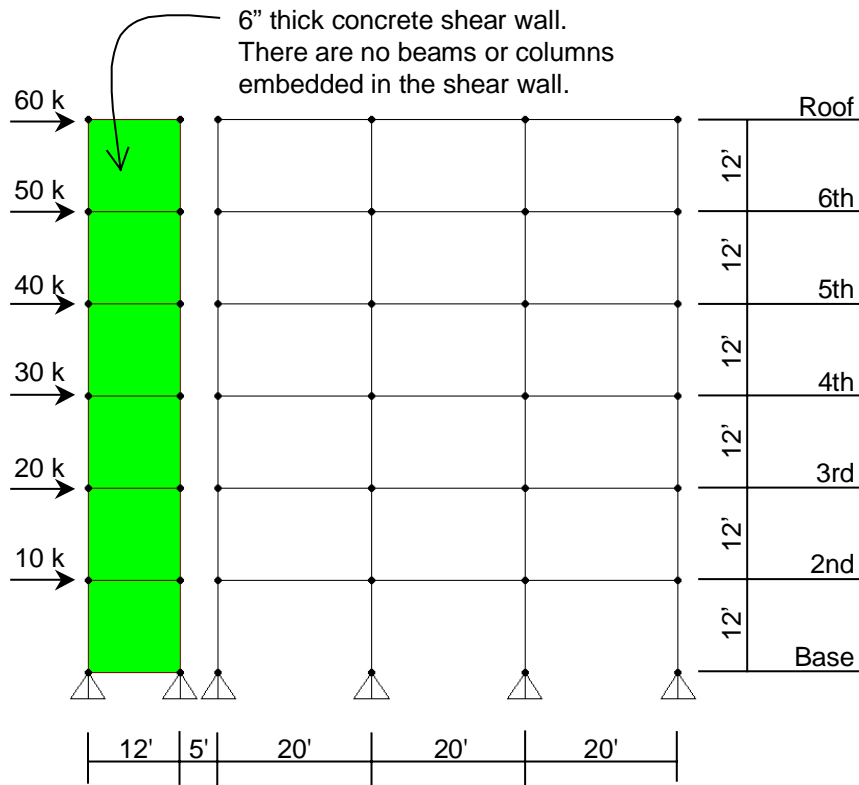
Columns: 24" by 24", pinned base

Diaphragm

Provide rigid diaphragm constraint at each level.


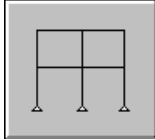


To Do

Determine shear carried by wall and by frame at 2nd level and at 6th level.



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 N 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 **6** in the Number of Stories edit box.
 - Type **3** in the Number of Bays edit box.
 - Accept the default Story Height of 12.
 - 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. From the **Draw** menu select **Edit Grid...** to display the Modify Grid Lines dialog box.
7. In this dialog box:
 - Verify that the X option is selected in the Direction area.
 - Type **-35** in the X Location edit box and click the **Add Grid Line** button.
 - Type **-47** in the X Location edit box and click the **Add Grid Line** button.
 - Click the **OK** button. The screen appears as shown in Figure N-1.
8. Click the **Quick Draw Rectangular Shell Element** button  on the side toolbar (or select **Quick Draw Rectangular Shell Element** from the **Draw** menu).
9. Click in the area labeled “A” in Figure N-1 to input the first shell element. Note that a quick Shell element is drawn by clicking in a grid space, bounded by four grid lines.
10. Click in the areas labeled “B”, “C”, “D”, “E” and “F”, in that order, in Figure N-1 to input the other shell elements.
11. Click the **Pointer** button  on the side tool bar to exit draw mode and enter select mode.

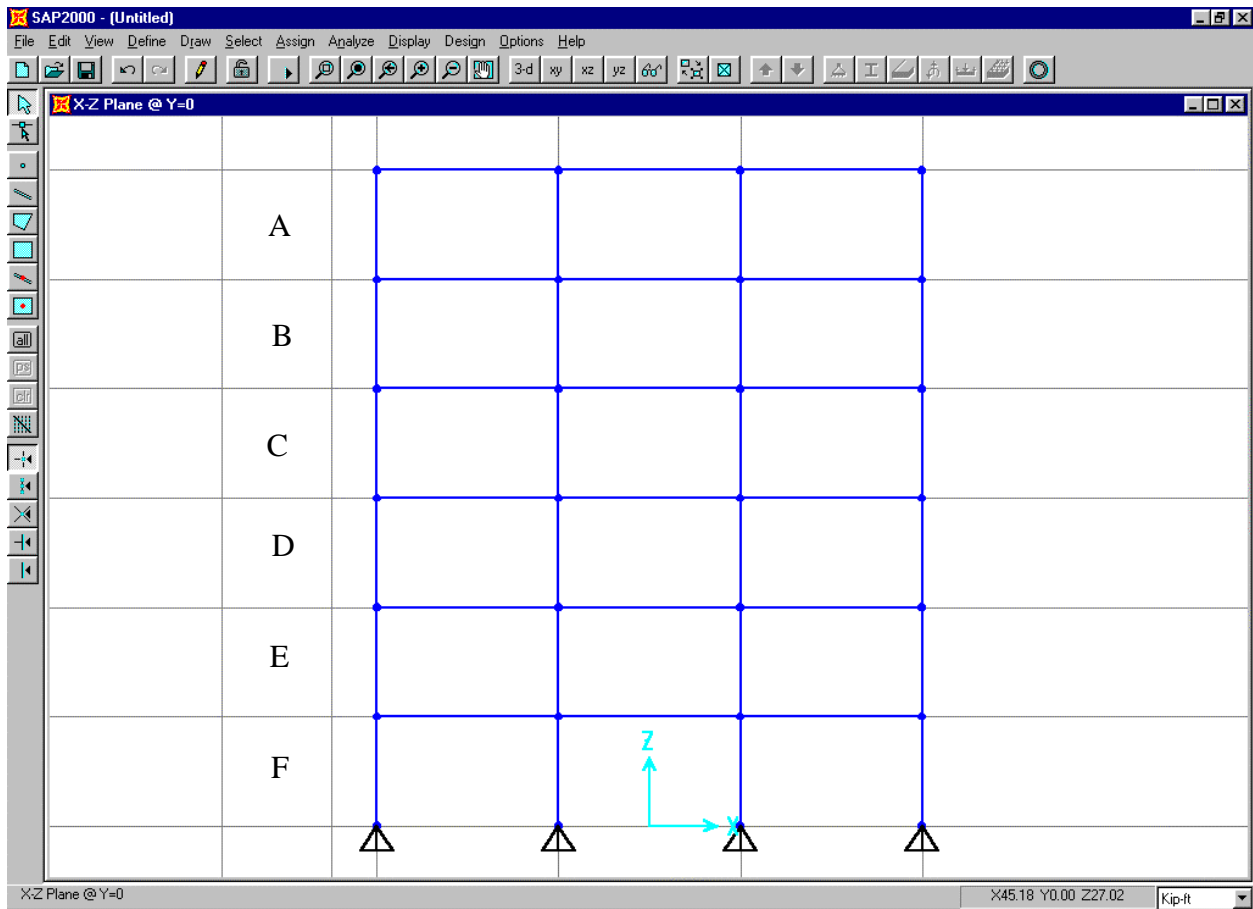







Figure N-1: Screen As It Appears After Step 7

12. Click the **Set Elements** button  on the main toolbar (or select **Set Elements...** from the **View** menu) to display the Set Elements Dialog box.
13. In this dialog box:
 - Check the Labels box in the Joints area.
 - Check the Fill Elements check box.
 - Click the **OK** button.

*Note: If the font size is too small for you to read the joint labels use the following procedure to increase the font size. From the **Options** menu select **Preferences**, click on the **Dimensions** Tab if it is not already visible, type in a new (larger) font size in the **Minimum Graphic Font Size** edit box (usually about 6 points is sufficient), click the **OK** button and then click the **Refresh Window** button  on the main toolbar.*

14. Select the joints 41 and 42 at the base of the shear wall.



15. From the **Assign** menu, choose **Joint**, and then **Restraints...** from the submenu. This will display the Joint Restraints dialog box.
16. In this dialog box:
 - Click the pinned base fast restraint button  to set all translational degrees of freedom (U1, U2 and U3) as restrained.
 - Click the **OK** button
17. Click the **Show Undeformed Shape** button  to remove the display of joint restraints and reset the window display (title).
18. Click the drop down box in the status bar to change the units to kip-in. 
19. From the **Define** menu select **Materials...** to display the Define Materials dialog box.
20. 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.
21. In this dialog box:
 - Verify that the Modulus of Elasticity is set to 3600.
 - Verify that Poisson's Ratio is set to 0.2.
 - Click the **OK** button twice to exit all dialog boxes.
22. From the **Define** menu select **Frame Sections...** to display the Define Frame Sections dialog box.
23. In this dialog box:
 - In the Click To area, click the drop-down box that says Add I/Wide Flange and then click on the Add Rectangular item to display the Rectangular Section dialog box.
 - In this dialog box:
 - Type **BEAM** in the Section name edit box.
 - Select CONC from the Material drop-down box.
 - Type **24** in the Depth (t3) edit box.
 - Type **12** in the Width (t2) edit box.
 - Click the **OK** button to return to the Define Frame Sections dialog box.


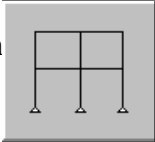
- In the Click To area, click the drop-down box that says Add Rectangular and then click on the Add Rectangular item to display the Rectangular Section dialog box.
 - In this dialog box:
 - Type **COL** in the Section name edit box.
 - Select CONC from the Material drop-down box.
 - Type **24** in the Depth (t3) edit box.
 - Type **24** in the Width (t2) edit box.
 - Click the **OK** button twice to exit all dialog boxes.
24. From the **Define** menu select **Shell Sections...** to display the Define Shell Sections dialog box.
25. In this dialog box:
- In the Click To area, click the **Add New Section** button to display the Shell Sections dialog box.
 - In this dialog box:
 - Verify that the selected material is CONC.
 - In the Thickness area type **6** in both the Membrane and the Bending edit boxes.
 - Verify that the Shell option is selected in the Type area.
 - Click the **OK** button twice to close all dialog boxes.
26. Select all of the column elements by “windowing” each of the four column lines separately.
27. From the **Assign** menu select **Frame** and then **Sections...** from the submenu to display the Define Frame Sections dialog box.
28. In this dialog box:
- Click on COL in the Frame Sections area to highlight it.
 - Click the **OK** button.
29. Select all of the beam elements by using the intersecting line method on each of the three beam bays separately.
30. From the **Assign** menu select **Frame** and then **Sections...** from the submenu to display the Define Frame Sections dialog box.


31. In this dialog box:
 - Click on **BEAM** in the Frame Sections area to highlight it.
 - Click the **OK** button.
32. Select all of the joints at the Roof level by “windowing”.
33. From the **Assign** menu select **Joint** and then **Constraints...** from the submenu to display the Constraints dialog box.
34. In this dialog box:
 - In the Click To area click the drop-down box and select Add Diaphragm to display the Diaphragm Constraint dialog box.
 - In this dialog box:
 - Type **ROOFDIA** in the Constraint Name edit box.
 - Select the Z Axis option in the Constraint Axis area.
 - Click the **OK** button twice to exit all dialog boxes.
35. Select all of the joints at the 6th level by “windowing”.
36. From the **Assign** menu select **Joint** and then **Constraints...** from the submenu to display the Constraints dialog box.
37. In this dialog box:
 - In the Click To area click the drop-down box and select Add Diaphragm to display the Diaphragm Constraint dialog box.
 - In this dialog box:
 - Type **6THDIA** in the Constraint Name edit box.
 - Select the Z Axis option in the Constraint Axis area.
 - Click the **OK** button twice to exit all dialog boxes.
38. Select all of the joints at the 5th level by “windowing”.
39. From the **Assign** menu select **Joint** and then **Constraints...** from the submenu to display the Constraints dialog box.
40. In this dialog box:

- In the Click To area click the drop-down box and select Add Diaphragm to display the Diaphragm Constraint dialog box.
 - In this dialog box:
 - Type **5THDIA** in the Constraint Name edit box.
 - Select the Z Axis option in the Constraint Axis area.
 - Click the **OK** button twice to exit all dialog boxes.
41. Select all of the joints at the 4th level by “windowing”.
42. From the **Assign** menu select **Joint** and then **Constraints...** from the submenu to display the Constraints dialog box.
43. In this dialog box:
- In the Click To area click the drop-down box and select Add Diaphragm to display the Diaphragm Constraint dialog box.
 - In this dialog box:
 - Type **4THDIA** in the Constraint Name edit box.
 - Select the Z Axis option in the Constraint Axis area.
 - Click the **OK** button twice to exit all dialog boxes.
44. Select all of the joints at the 3rd level by “windowing”.
45. From the **Assign** menu select **Joint** and then **Constraints...** from the submenu to display the Constraints dialog box.
46. In this dialog box:
- In the Click To area click the drop-down box and select Add Diaphragm to display the Diaphragm Constraint dialog box.
 - In this dialog box:
 - Type **3RDDIA** in the Constraint Name edit box.
 - Select the Z Axis option in the Constraint Axis area.
 - Click the **OK** button twice to exit all dialog boxes.
47. Select all of the joints at the 2nd level by “windowing”.

48. From the **Assign** menu select **Joint** and then **Constraints...** from the submenu to display the Constraints dialog box.
49. In this dialog box:
 - In the Click To area click the drop-down box and select Add Diaphragm to display the Diaphragm Constraint dialog box.
 - In this dialog box:
 - Type **2NDDIA** in the Constraint Name edit box.
 - Select the Z Axis option in the Constraint Axis area.
 - Click the **OK** button twice to exit all dialog boxes.
50. Select joints 39, 37, 35, 33, 29, and 32 (left-hand side of shear wall from 2nd level to roof level) by “windowing”.
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 **10** in the Force Global X edit box in the Loads area.
 - Click the **OK** button.
53. Select joints 37, 35, 33, 29, and 32 (left-hand side of shear wall from 3rd level to roof level) by “windowing”.
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 click the **OK** button.
56. Select joints 35, 33, 29, and 32 (left-hand side of shear wall from 4th level to roof level) by “windowing”.
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 click the **OK** button.
59. Select joints 33, 29, and 32 (left-hand side of shear wall from 5th level to roof level) by “windowing”.
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 click the **OK** button.
62. Select joints 29, and 32 (left-hand side of shear wall from 6th level to roof level) by “windowing”.
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 click the **OK** button.
65. Select joint 32 (left-hand side of shear wall at roof level).
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 click the **OK** button.
68. Click the **Show Undeformed Shape** button  to remove the display of joint forces.
69. Click the **Set Elements** button  on the main toolbar (or select **Set Elements...** from the **View** menu) to display the Set Elements Dialog box.
70. In this dialog box:
 - Check the Labels box in the Frames area.
 - Check the Labels box in the Shells area.
 - Click the **OK** button.
71. Select joints 29 and 30 and select shell element 1.
72. From the **Assign** menu select **Group Names...** to display the Assign Group dialog box.
73. In this dialog box:
 - Type **6THWALL** in the Groups edit box.
 - Click the **Add New Group Name** button.
 - Click the **OK** button.
74. Select joints 6, 13, 20 and 27 by clicking on each joint individually and select frame (column) elements 6, 12, 18 and 24 using the intersecting line method.
75. From the **Assign** menu select **Group Names...** to display the Assign Group dialog box.
76. In this dialog box:

- Type **6THFRAME** in the Groups edit box.
 - Click the **Add New Group Name** button.
 - Click the **OK** button.
77. Select joints 39 and 40 and select shell element 5.
78. From the **Assign** menu select **Group Names...** to display the Assign Group dialog box.
79. In this dialog box:
- Type **2NDWALL** in the Groups edit box.
 - Click the **Add New Group Name** button.
 - Click the **OK** button.
80. Select joints 2, 9, 16 and 23 by clicking on each joint individually and select frame (column) elements 2, 8, 14 and 20 using the intersecting line method.
81. From the **Assign** menu select **Group Names...** to display the Assign Group dialog box.
82. In this dialog box:
- Type **2NDFRAME** in the Groups edit box.
 - Click the **Add New Group Name** button.
 - Click the **OK** button.
83. Click the **Set Elements** button  on the main toolbar (or select **Set Elements...** from the **View** menu) to display the Set Elements Dialog box.
84. In this dialog box:
- Uncheck the Labels box in the Joints area.
 - Uncheck the Labels box in the Frames area.
 - Uncheck the Labels box in the Shells area.
 - Click the **OK** button.
85. 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.
86. Click the **Run Analysis** button  to run the analysis.
 87. 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.
 88. From the **Display** menu select **Show Group Joint Force Sums...** to display the Select Groups dialog box.
 89. In this dialog box:
 - Click on 2NDFRAME to highlight it.
 - Hold down the shift key on the keyboard and click on 6THWALL. All of the available groups, except the default ALL group should now be highlighted.
 - Click the **OK** button to display the a window with the Group Joint Force Summations tabulated.

Note: Notice the direction of the shear (F-X force) in the shear wall at the 6th level.

- When finished viewing the window click the “X” in its upper right-hand corner to close it.

*Note: If you want to print the group joint force sum tables click on the **File** menu in the Group Joint Force Summations widow and select either **Print Tables** or **Print Tables to File...***