# Problem Z

**Response Spectrum Analysis**

**Building Description**

The building is a four-story concrete shear wall building with concrete flat slabs supported by concrete columns. There is a 30 foot high steel flagpole on the roof at one corner of the building. A 250 pound man sits on top of the flagpole.

**Steel**

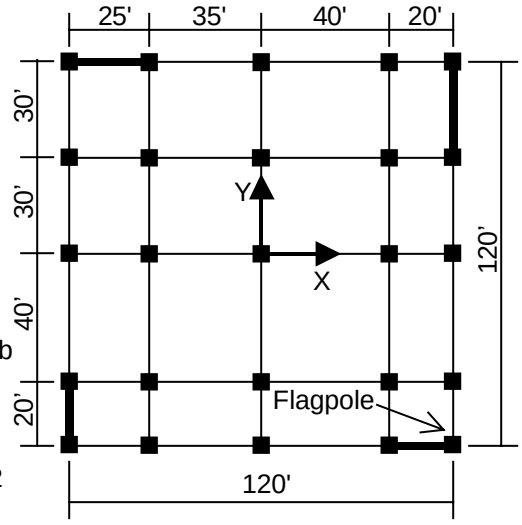
E =29000 ksi
Poissons Ratio = 0.3
Flagpole is 3" ∅ standard pipe

**Concrete**

E =3600 ksi
Poissons Ratio = 0.2
Walls are 12" thick
Columns & beams are 20" x 20"
Floors & roof are 10" thick flat slab

**Response Spectrum Loading**

X-Dir (U1): 1994 UBC S2
Y-Dir (U2): 30% of 1994 UBC S2



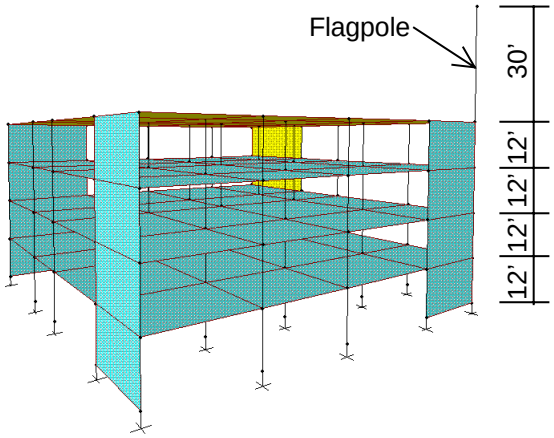

**Typical Floor and Roof Plan**

**Assumptions**

- Diaphragms are rigid in plane.
- Columns are fixed base.
- Consider the mass of the 250 pound man which is 0.00065 kip-sec2/in.

**To Do**

Determine maximum X-direction (U1) displacement at top and bottom of the flagpole for the specified response spectrum loading.



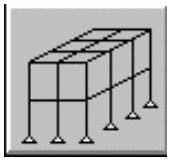

**Three Dimensional Perspective View**

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 Z 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 **Space Frame** template button  to display the Space Frame dialog box.

4. In this dialog box:

   - Type **4** in the Number of Stories edit box.
   - Type **4** in the Number of Bays Along X edit box.
   - Type **4** in the Number of Bays Along Y edit box.
   - Accept the default Story Height, 12.
   - Type **30** in the Bay Width Along X edit box.
   - Type **30** in the Bay Width Along Y edit box.
   - Verify that the Restraints and Gridlines check boxes are checked.
   - Click the **OK** button.

5. From the **Draw** menu select **Edit Grid...** to display the Modify Grid Lines dialog box.

6. In this dialog box:

   - Check the Glue Joints To Grid Lines check box.
   - Verify that the X option is selected in the Direction area.
   - Click on the -30 grid line in the X Location list box to highlight it. The -30 value appears in the X Location edit box.
   - Type **-35** in the X Location edit box and click the **Move Grid Line** button.
   - Click on the 30 grid line in the X Location list box to highlight it. The 30 value appears in the X Location edit box.
   - Type **40** in the X Location edit box and click the **Move Grid Line** button.

- Select the Y option in the Direction area.
- Click on the -30 grid line in the Y Location list box to highlight it. The -30 value appears in the Y Location edit box.
- Type **-40** in the Y Location edit box and click the **Move Grid Line** button.
- Click the **OK** button.

7. Verify that the 3-D View window is active. The window is active when its title is highlighted.
8. Click the **Set Elements** button  on the main toolbar (or select **Set Elements…** from the **View** menu) to display the Set Elements Dialog box.
9. In this dialog box:
   - Check the Fill Elements check box.
   - Click the **OK** button.
10. From the **View** menu select **Refresh View** to update the 3-D view.
11. Click in the window labeled X-Y Plane @ Z=48 to activate it.
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 Fill Elements check box.
    - Click the **OK** button.
14. Click the **xz 2D View** button  on the main toolbar. The view switches to the X-Z plane @ Z=60 and appears as shown in Figure Z-1.
15. Click the **Quick Draw Rectangular Shell Element** button  on the side toolbar.
16. Click once in each of the area labeled “A”, “B”, “C” and “D” in Figure Z-1 to enter four shell elements.
17. From the **View** menu select **Set 2D View...** to display the Set 2D View dialog box.
18. In this dialog box:
    - Verify that the X-Z Plane option is selected.
    - Type **-60** in the Y= edit box.

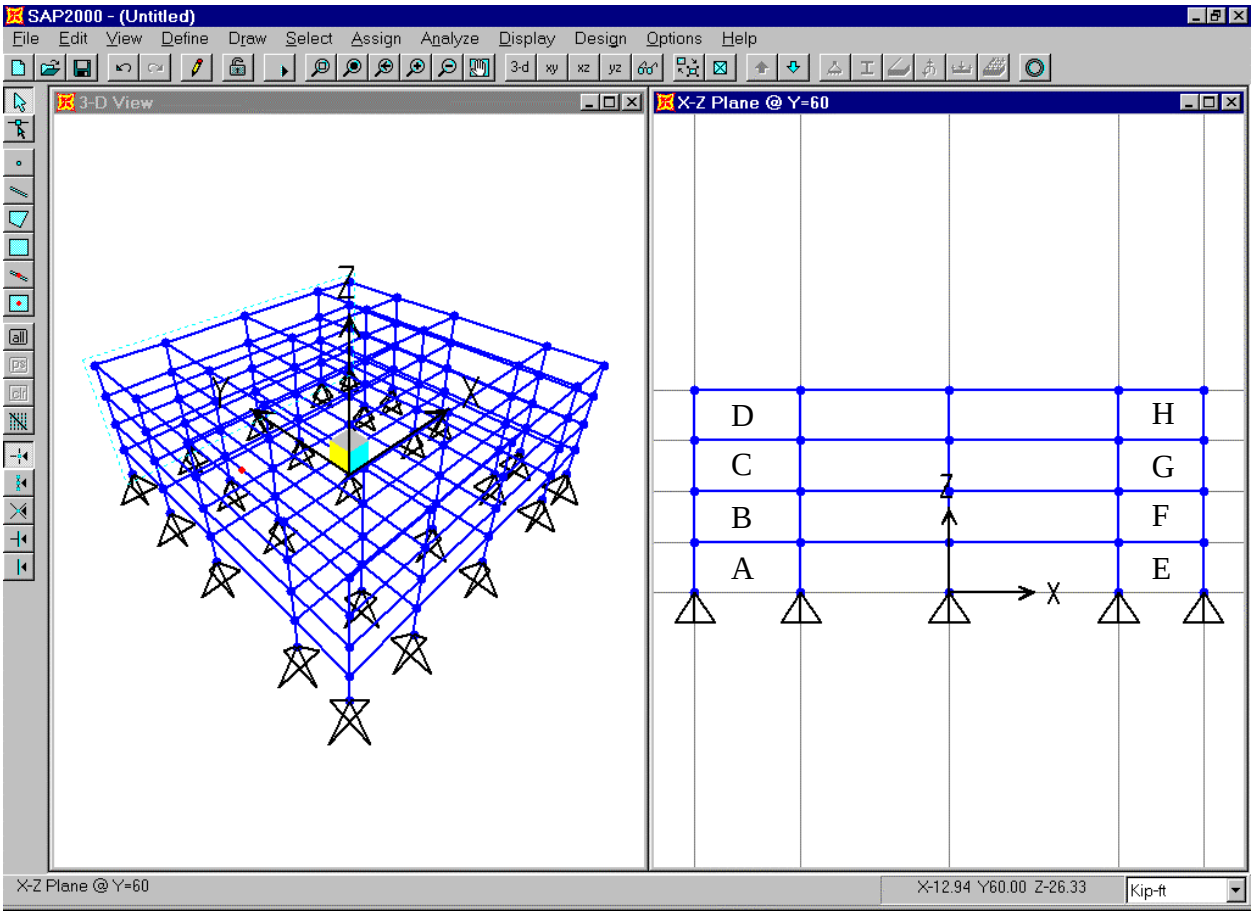



Figure Z-1: Screen After Step 14

- Click the **OK** button. The screen appears similar to that shown in Figure Z-1 (except that the location is now at Y=-60).

19. Click once in each of the area labeled “E”, “F”, “G” and “H” in Figure Z-1 to enter four shell elements.

20. From the **View** menu select **Set 2D View...** to display the Set 2D View dialog box.

21. In this dialog box:

    - Select the Y-Z Plane option.
    - Verify that 60 is entered in the X= edit box.
    - Click the **OK** button. The screen appears as shown in Figure Z-2.

22. Click once in each of the area labeled “A”, “B”, “C” and “D” in Figure Z-2 to enter four shell elements.

23. From the **View** menu select **Set 2D View...** to display the Set 2D View dialog box.

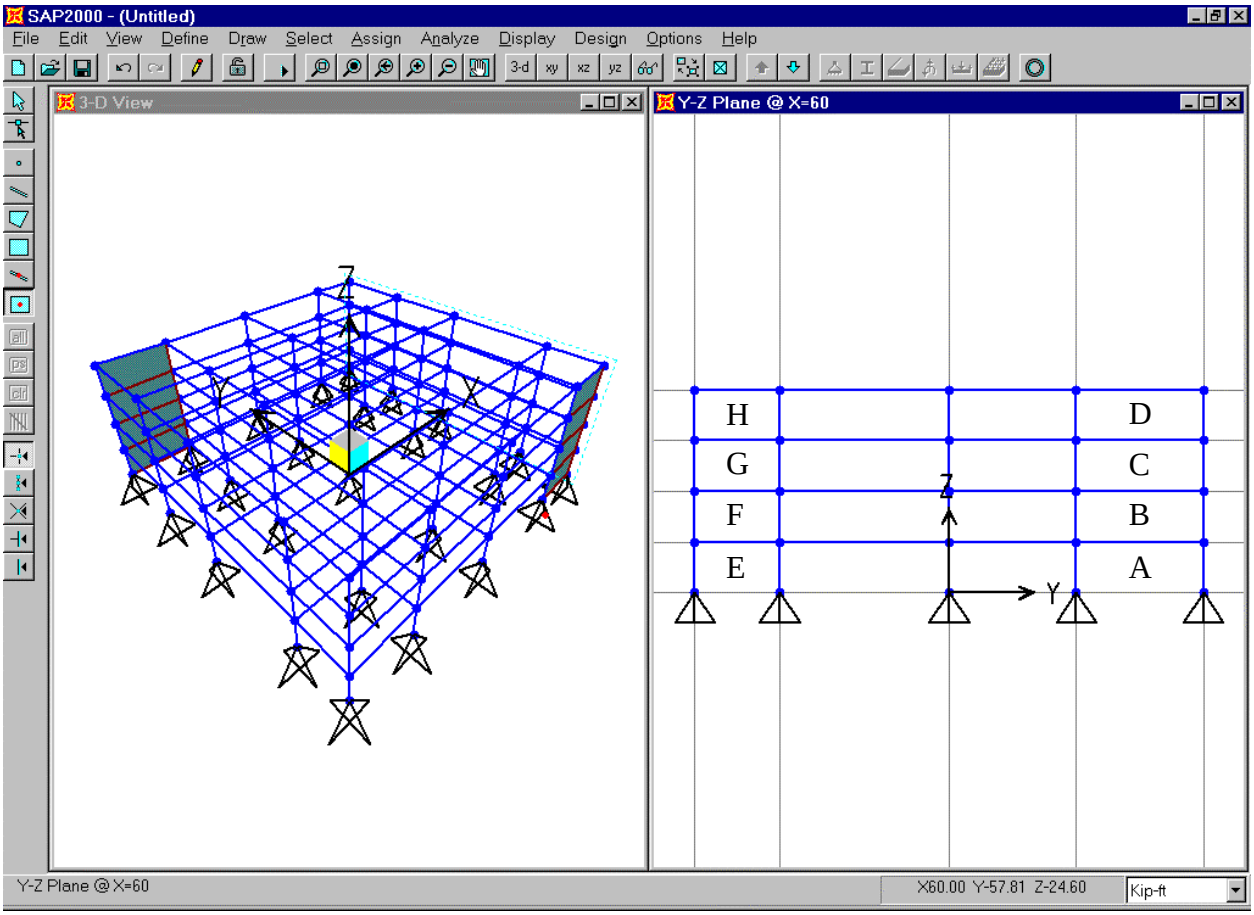



Figure Z-2: Screen After Step 21

24. In this dialog box:

    - Verify that the Y-Z Plane option is selected.
    - Type **-60** in the X= edit box.
    - Click the **OK** button. The screen appears similar to that shown in Figure Z-2 (except that the location is now at X=-60).

25. Click once in each of the area labeled “E”, “F”, “G” and “H” in Figure Z-2 to enter four shell elements. This completes the drawing of the shear walls.

26. Click the **Pointer** button to exit draw mode and enter select mode.

27. From the **Define** menu select **Materials...** to display the Define Materials dialog box.

28. In this dialog box:

    - Highlight the CONC material and click the **Modify/Show Material** button to display the Material Property Data dialog box.

- In this dialog box:
  - Verify that the Mass per Unit Volume is 4.658E-03.
  - Verify that the Weight per Unit Volume is 0.15.
  - Click the **OK** button to return to the Define Materials dialog box.
- Highlight the STEEL material and click the **Modify/Show Material** button to display the Material Property Data dialog box.
- In this dialog box:
  - Verify that the Mass per Unit Volume is 0.0152.
  - Verify that the Weight per Unit Volume is 0.489.
  - Click the **OK** button twice to exit all dialog boxes.

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


30. From the **Define** menu select **Materials...** to display the Define Materials dialog box.

31. In this dialog box:
    - Highlight the CONC material and click the **Modify/Show Material** button to display the Material Property Data dialog box.
    - In this dialog box:
      - Verify that the Modulus of Elasticity is 3600.
      - Verify that Poisson's ratio is 0.2.
      - Click the **OK** button to return to the Define Materials dialog box.
    - Highlight the STEEL material and click the **Modify/Show Material** button to display the Material Property Data dialog box.
    - In this dialog box:
      - Verify that the Modulus of Elasticity is 29000.
      - Verify that Poisson's ratio is 0.3.
      - Click the **OK** button twice to exit all dialog boxes.

32. From the **Define** menu select **Frame Sections...** to display the Define Frame Sections dialog box.

33. In this dialog box:

    - In the Click To area, click the drop-down box that says Import I/Wide Flange and then click on the Import Pipe item.
    - 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.
    - A dialog box appears with a list of all pipe sections in the database. In this dialog box:
        - Scroll down and click on the P3 (3" diameter standard pipe section) item.
        - Click the OK button twice to return to the Define Frame Sections 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. The Rectangular Section dialog box is displayed.
    - In this dialog box:
        - Type **BMCOL** in the Section Name edit box.
        - Select CONC from the Material drop-down box.
        - Type **20** in the Depth (t3) edit box.
        - Type **20** in the Width (t2) edit box.
        - Click the OK button twice to exit all dialog boxes.

34. From the **Define** menu select **Shell Sections...** to display the Define Shell Sections dialog box.

35. In this dialog box:

    - Click the **Modify/Show Section** button to display the Shell Sections dialog box.
    - In this dialog box:
        - Type **WALL** in the Section Name edit box.
        - Accept the default CONC material
        - In the Thickness area verify that both the Membrane and the Bending thicknesses are 12.
        - In the Type area verify that the Shell option is selected.
        - Click the **OK** button to return to the Define Shell Sections dialog box.

- Click the **Add New Section** button to display the Shell Sections dialog box.
- In this dialog box:
  - Type **FLOOR** in the Section Name edit box.
  - Accept the default CONC material.
  - Type **10** in the Membrane edit box.
  - Type **10** in the Bending edit box.
  - In the Type area verify that the Shell option is selected.
  - Click the **OK** button twice to exit all dialog boxes.

36. Click the drop down box in the status bar to change the units to kip-ft. 

37. Click in the window labeled Y-Z Plane @ X=-60 to verify it is active.
38. From the **View** menu select **Set 2D View...** to display the Set 2D View dialog box.
39. In this dialog box:
    - Select the X-Y Plane option.
    - Verify that 48 is entered in the Z= edit box.
    - Click the **OK** button.
40. Click the **Set Elements** button  on the main toolbar (or select **Set Elements…** from the **View** menu) to display the Set Elements Dialog box.
41. In this dialog box:
    - Check the Labels box in the Joints area.
    - Click the **OK** button.
42. Click the **Draw Rectangular Shell Element** button  on the side toolbar or select **Draw Rectangular Shell Element** from the **Draw** menu.
43. Click on joint 25 and then joint 105 to enter a single shell element for the entire floor.
44. Click the **Pointer** button  to exit draw mode and enter select mode.
45. Select all elements in the X-Y Plane @ Z=48 by “windowing”.
46. From the **Edit** menu select **Mesh Shells...** to display the Mesh Selected Shells dialog box.

47. In this dialog box:

    - Select the mesh Using Selected Joints on Edges option.

      *Note: The Mesh At Intersections With Grids option would work equally well.*

    - Click the **OK** button.

48. Select all elements in the X-Y Plane @ Z=48 by "windowing".

49. From the **Assign** menu select **Shell** and then **Sections...** from the submenu to display the Define Shell Sections dialog box.

50. In this dialog box:

    - Highlight the FLOOR section by clicking on it.

    - Click the **OK** button.

51. Click the **Show Undeformed Shape** button  to remove the displayed shell section assignments.

52. Click the **Restore Previous Selection** button  on the side toolbar (or select **Get Previous Selection** from the **Select** menu).

53. From the **Edit** menu select **Replicate...** to display the Replicate dialog box.

54. In this dialog box:

    - Verify the Linear Tab is selected.

    - In the Distance area type **-12** in the Z edit box.

    - Verify that 0 is entered in the X and Y edit boxes.

    - Type **3** in the Number edit box.

    - Click the **OK** button to proceed with the replication and create the other floor diaphragms.

55. Click the **Restore Previous Selection** button  on the side toolbar (or select **Get Previous Selection** from the **Select** menu).

56. From the **Assign** menu select **Joint** and then **Constraints...** from the submenu to display the Constraints dialog box.

57. 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.

58. Click the **Down One Gridline** button to move the plan display down to the X-Y Plane @ Z=36. You can confirm the elevation by looking on the right-hand side of the staus bar at the bottom of the SAP2000 window.

59. Select all elements at this level by "windowing".

60. From the **Assign** menu select **Joint** and then **Constraints...** from the submenu to display the Constraints dialog box.

61. 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.

62. Click the **Down One Gridline** button to move the plan display down to the X-Y Plane @ Z=24.

63. Select all elements at this level by "windowing".

64. From the **Assign** menu select **Joint** and then **Constraints...** from the submenu to display the Constraints dialog box.

65. 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.

66. Click the **Down One Gridline** button  to move the plan display down to the X-Y Plane @ Z=12.

67. Select all elements at this level by “windowing”.

68. From the **Assign** menu select **Joint** and then **Constraints...** from the submenu to display the Constraints dialog box.

69. 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.

70. Click the **Down One Gridline** button  to move the plan display down to the X-Y Plane @ Z=0.

71. Select all elements at this level by “windowing”.

72. From the **Assign** menu select **Joint** and then **Restraints...** from the submenu to display the Joint Restraints dialog box.

73. In this dialog box:

- Click the **Fixed Support** button  in the Fast Restraints area.
- Click the **OK** button.

74. Click the **Select All** button  on the side toolbar to select all elements.

75. From the **Assign** menu select **Frame** and then **Sections...** from the submenu to display the Define Frame Sections dialog box.

76. In this dialog box:

- Click on BMCOL in the Frame Sections area to highlight it.
- Click the **OK** button.

77. Click the **Show Undeformed Shape** button  to remove the displayed frame section assignments.

78. From the **View** menu select **Set 2D View...** to display the Set 2D View dialog box.

79. In this dialog box:

    - Select the X-Z Plane option.
    - Verify -60 is entered in the Y= edit box.
    - Click the **OK** button.

80. From the **Draw** menu select **Edit Grid...** to display the Modify Grid Lines dialog box.

81. In this dialog box:

    - Select the Z option in the Direction area.
    - Type **78** in the Z Location edit box and click the **Add Grid Line** button.
    - Click the **OK** button. The screen appears as shown in Figure Z-3.

82. Click the **Quick Draw Frame Element** button  on the side toolbar or select **Quick Draw Frame Element** from the **Draw** menu.

83. Click on the grid line at the point labeled "A" in Figure Z-3 to enter the flagpole frame element.

84. Click the **Pointer** button  to exit draw mode and enter select mode.

85. Click on the frame element to select it.

86. From the **Assign** menu select **Frame** and then **Sections...** from the submenu to display the Define Frame Sections dialog box.

87. In this dialog box:

    - Click on P3 in the Frame Sections area to highlight it.
    - Click the **OK** button.

88. Click the **Show Undeformed Shape** button  to remove the displayed frame section assignments.

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


90. Click on the joint at the top of the flagpole to select it.

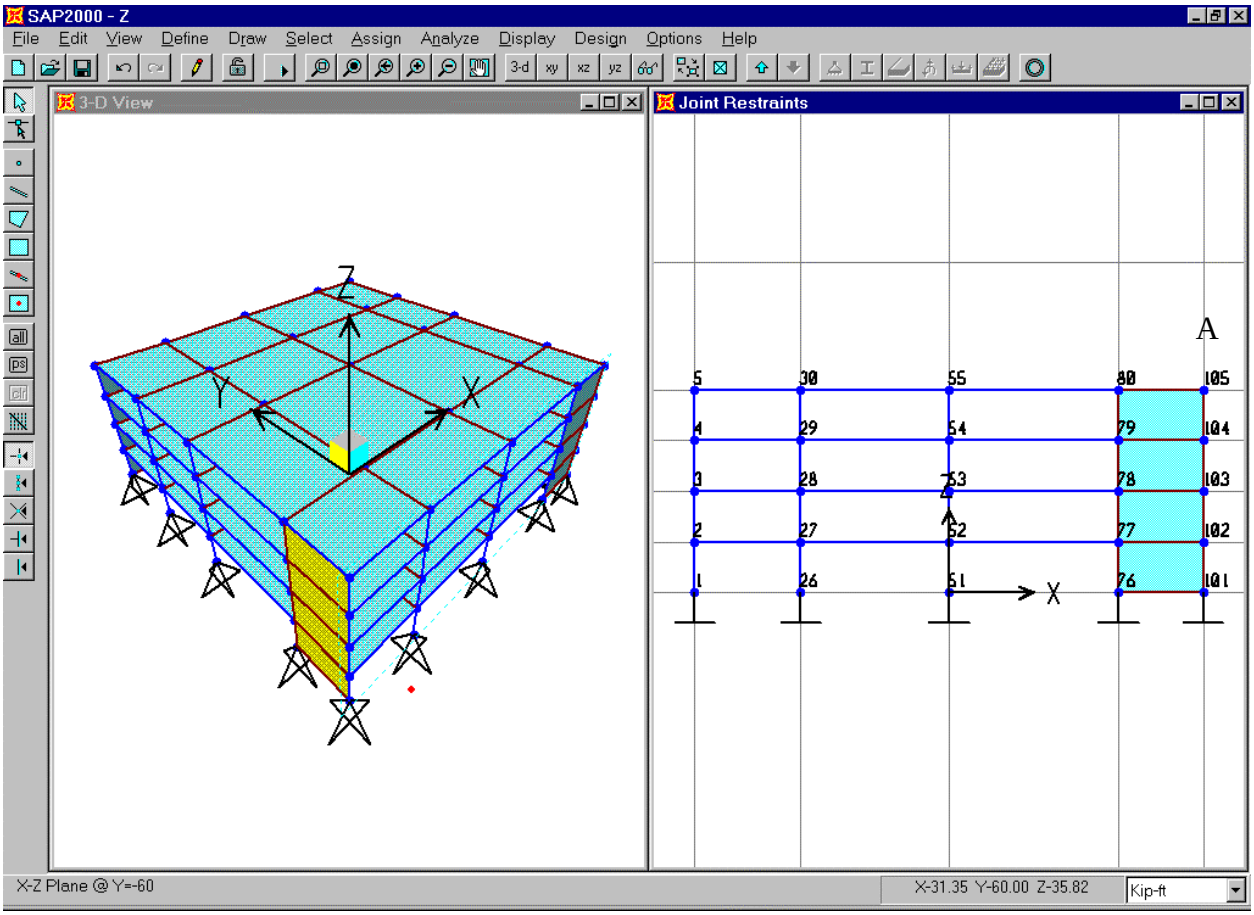



Figure Z-3: Screen After Step 81

91. From the **Assign** menu select **Joint** and then **Masses...** from the submenu to display the Joint Masses dialog box.

92. In this dialog box:

    - Type .**00065** in the Direction 1, Direction 2 and Direction 3 edit boxes.
    - Click the **OK** button.

93. Click the **Show Undeformed Shape** button  to remove the displayed joint mass assignments.

94. Click the drop down box in the status bar to change the units to kip-ft. 


95. Click the **Set Elements** button  on the main toolbar (or select **Set Elements…** from the **View** menu) to display the Set Elements Dialog box.

96. In this dialog box:

    - Uncheck the Labels box in the Joints area.

- Click the **OK** button.

97. From the **Define** menu select **Response Spectrum Cases…** to display the Define Response Spectra dialog box.

98. In this dialog box:

    - Click the Add New Spectra button to display the Response Spectrum Case Data dialog box.

    - In this dialog box:

        - Accept the default Spectrum case name, SPEC1.

        - Accept the default Excitation Angle, 0.

        - Accept the default Modal Combination option, CQC.

        - Type **.05** in the Damping edit box.

        - Accept the default Directional Combination option, SRSS.

        - In the Input Response Spectra area select UBC94S2 from the U1 Function drop-down box.

        - In the Input Response Spectra area type **32.2** in the U1 Scale factor edit box.

        - In the Input Response Spectra area select UBC94S2 from the U2 Function drop-down box.

        - In the Input Response Spectra area type **9.66** (0.3 * 32.2 = 9.66) in the U2 Scale factor edit box.

        - Click the **OK** button twice to exit all dialog boxes.

99. From the **Analyze** menu select **Set Options...** to display the Analysis Options dialog box.

    - Verify the the Dynamic Analysis check box is checked.

    - Click the **Set Dynamic Parameters** button to display the Dynamic Analysis Parameters dialog box.

    - In this dialog box:

        - Type **20** in the Number of Modes edit box.

        - In the Type of Analysis area select Ritz Vectors.

        - Click on ACCEL Z in the Ritz Load Vectors list box to highlight it.

- Click the **Remove** button to remove ACCEL Z from the Ritz Load Vectors list box. ACCEL X and ACCEL Y should remain in the Ritz Load Vectors list box.

- Click the **OK** button twice to exit all dialog boxes.

100. Click the **Run Analysis** button  to run the analysis.

101. When the analysis is complete check the messages in the Analysis window (there should be no warnings or errors). Click the **OK** button to close the Analysis window.

102. Click in the window labeled X-Z Plane @ Y=-60 to make sure it is active.

103. Click the **Display Static Deformed Shape** button  (or select **Show Deformed Shape…** from the **Display** menu). The Deformed Shape dialog box appears.

104. In this dialog box:

    - Select SPEC1 Spectra from the Load drop-down box.

    - Click the **OK** button.

105. Right click the joints at the top and bottom of the flagpole to see their displacements.

    *Note: If the top of the flagpole goes off of the screen when the deformed shape is displayed, then click the **Zoom Out One Step** button*  *on the main toolbar as many times as required to bring the joint back on to the screen.*