

Problem C

Truss Frame

Steel Frame

$E = 29000$ ksi, Poissons Ratio = 0.3

All steel members are L4x4 angles, $F_y = 36$ ksi

Base is pinned

Diaphragms

Concrete diaphragms are 8" thick with a unit weight of 150 pcf

Model as rigid diaphragm at Levels A and B

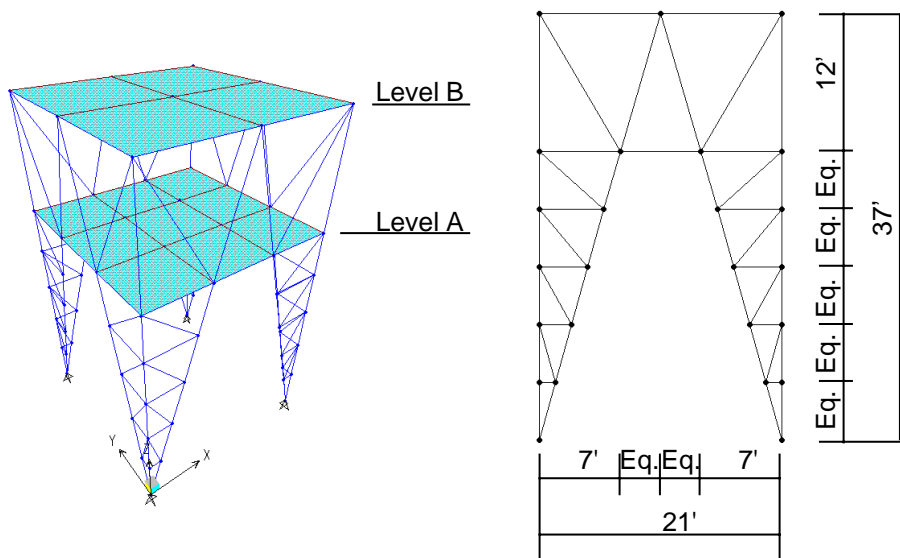
Additional dead load at each diaphragm is 50 psf

Live load at each diaphragm is 100 psf

To Do

Size steel members for DL + LL using AISC - ASD89

Determine the first three modes of vibration





Typical Elevation

(All four sides are the same)

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 C 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...** This displays the Coordinate System Definition dialog box.
3. In this dialog box
 - Select the Cartesian Tab.
 - In the Number of Grid Spaces area type **0** in the X direction edit box.
 - In the Number of Grid Spaces area type **0** in the Y direction edit box.
 - In the Number of Grid Spaces area type **0** in the Z direction edit box.
 - Click the **OK** button.
4. From the **Draw** menu select **Edit Grid...** to display the Modify Grid Lines dialog box.
5. In this dialog box:
 - Verify that the X option is selected in the Direction area.
 - Type **7** in the X Location edit box and click the **Add Grid Line** button.
 - Type **10.5** in the X Location edit box and click the **Add Grid Line** button.
 - Select the Z option in the Direction area.
 - Type **25** in the Z Location edit box and click the **Add Grid Line** button.
 - Type **37** in the Z Location edit box and click the **Add Grid Line** button.
 - Click the **OK** button.
6. Click in the window titled X-Y Plane @ Z=0 to make sure it is active. The window is highlighted when it is active.
7. Click the **xz 2D View** button  to change the view to an X-Z elevation. Note that the title of the window changes to X-Z Plane @ Y=0.

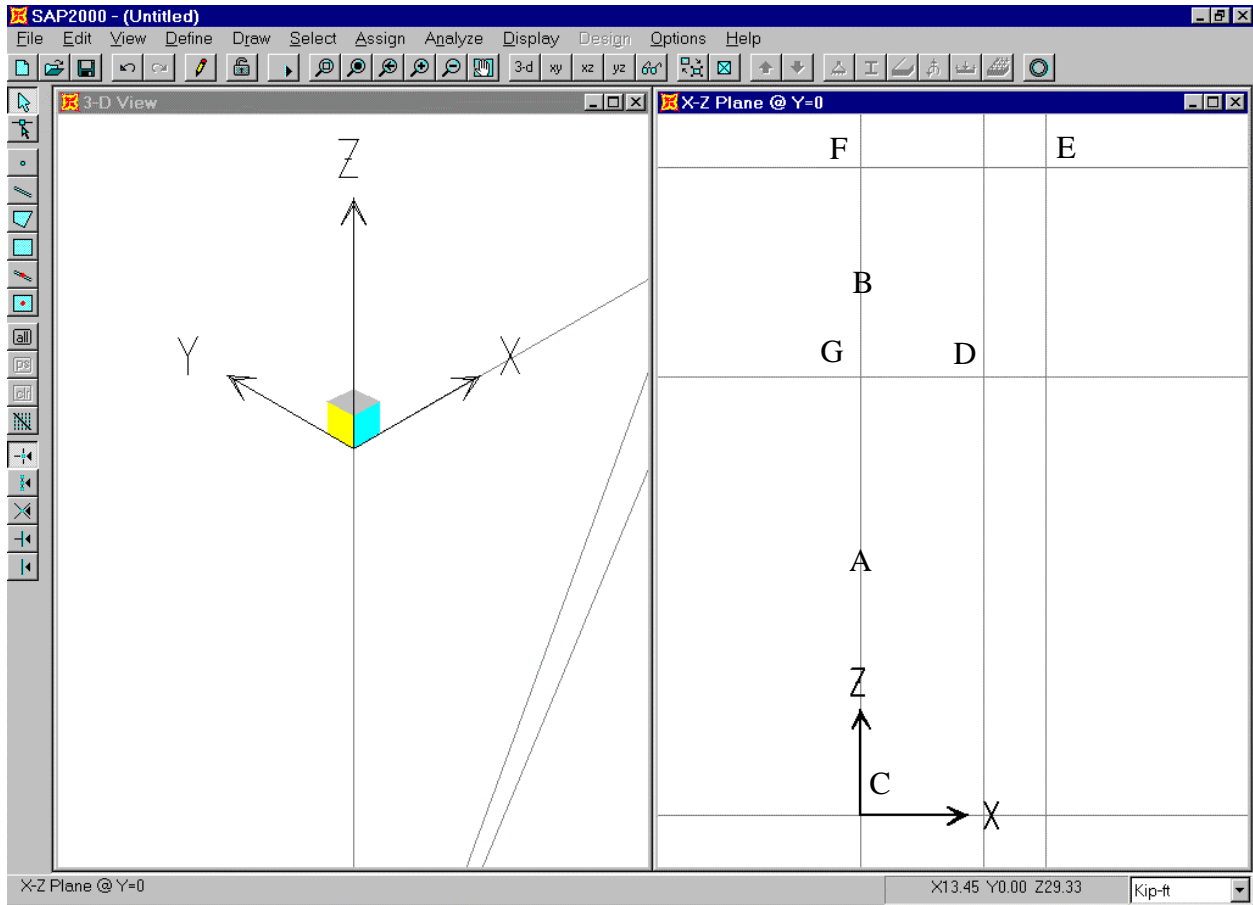






Figure C-1: Initial Grid Layout In X-Z Plane

8. Click the **Quick Draw Frame Element** button  on the side toolbar or select **Quick Draw Frame Element** from the **Draw** menu.
9. Click on the grid line at the point labeled “A” in Figure C-1 to enter a frame element.
10. Click on the grid line at the point labeled “B” in Figure C-1 to enter another frame element.
11. Click the **Draw Frame Element** button  on the side toolbar or select **Draw Frame Element** from the **Draw** menu.
12. Click on the points labeled “C”, “D”, and “E” in Figure C-1, in that order, and then press the Enter key on the keyboard to draw two more frame elements.
13. Click on the point labeled “F” and then double click the point labeled “E” in Figure C-1 to draw the next frame element.

Note: You could have single-clicked the point labeled “E” in Figure C-1 and then pressed the Enter key on the keyboard to finish drawing the frame element.

14. Click on the point labeled “G” and then double click the point labeled “D” in Figure C-1 to draw the next frame element.
15. Click on the point labeled “D” and then double click the point labeled “F” in Figure C-1 to draw the next frame element.
16. Click the **Pointer** button  to exit Draw Mode and enter Select Mode.
17. Click the **Set Elements** button  on the main toolbar (or select **Set Elements...** from the **View** menu) to display the Set Elements Dialog box.
18. In this dialog box:
 - Check the Labels box in the Joints area.
 - Check the Labels box in the Frames area.
 - Check the Fill Elements box.
 - Click the **OK** button. The screen appears as shown in Figure C-2.

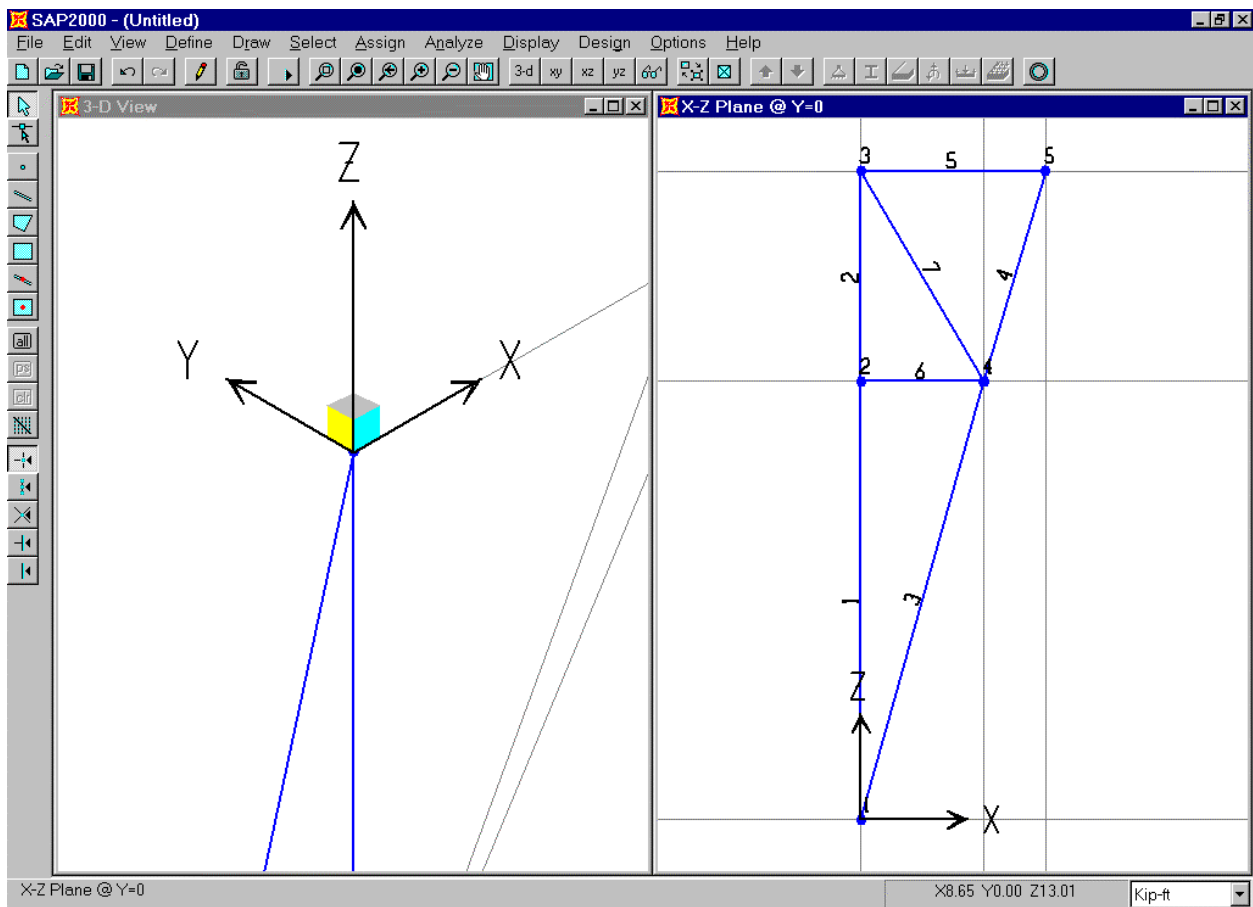

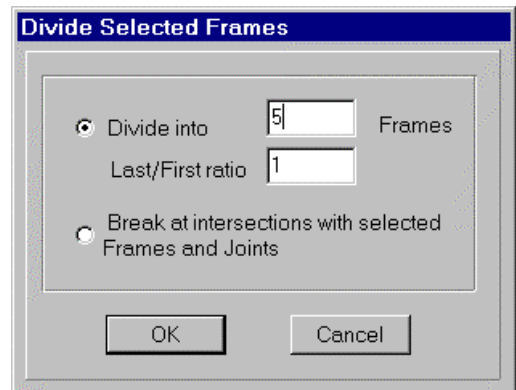






Figure C-2: Screen As It Appears After Step 18

19. Click on frame elements 1 and 3 to select them.
20. From the **Edit** menu select **Divide Frames...** to display the Divide Selected Frames dialog box.
21. Verify that this dialog box is filled out as shown in the adjacent figure and click the **OK** button.
22. Click the **Draw Frame Element** button  on the side toolbar or select **Draw Frame Element** from the **Draw** menu.
23. Click on joint 9 and then double click on joint 13 to draw a frame element.











24. Click on joint 8 and then double click on joint 12 to draw a frame element.
25. Click on joint 7 and then double click on joint 11 to draw a frame element.
26. Click on joint 6 and then double click on joint 10 to draw a frame element.
27. Click on joint 13 and then double click on joint 2 to draw a frame element.
28. Click on joint 12 and then double click on joint 9 to draw a frame element.
29. Click on joint 11 and then double click on joint 8 to draw a frame element.
30. Click on joint 10 and then double click on joint 7 to draw a frame element.
31. Click the **Pointer** button  to exit Draw Mode and enter Select Mode.
32. Click in the 3D View window to activate it.
33. From the **View** menu select **Refresh View** to rescale the view.
34. Click in the Window labeled X-Z Plane @ Y=0 to activate it.
35. Click the **Select All** button  on the side toolbar to select all elements.
36. From the **Edit** menu select **Replicate...** to display the Replicate dialog box.
37. In this dialog box:
 - Select the Mirror Tab.
 - In the Mirror About area select Y-Z plane.
 - In the Ordinate area type **10.5** in the X edit box.


- Click the **OK** button to proceed with the replication.
38. Click the **Draw Frame Element** button  on the side toolbar or select **Draw Frame Element** from the **Draw** menu.
 39. Click on joint 4 and then double click on joint 16 to draw a frame element.

*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.*

Note: If you still have difficulty reading a particular joint label you can always right click the joint to bring up a dialog box that gives you information about the joint.

40. Click the **Pointer** button  to exit Draw Mode and enter Select Mode.
41. Click the **Select All** button  on the side toolbar to select all elements.
42. From the **Edit** menu select **Replicate...** to display the Replicate dialog box.
43. In this dialog box:
 - Select the Radial Tab.
 - In the Rotate About area select the Z Axis option.
 - In the Increment Data area verify that the Angle is 90 and the Number is 1.
 - Click the **OK** button to proceed with the replication.
44. Click the **Restore Previous Selection** button  on the side toolbar.
45. From the **Edit** menu select **Replicate...** to display the Replicate dialog box.
46. In this dialog box:
 - Verify the Linear Tab is selected.
 - In the Distance area type **21** in the Y edit box.
 - Verify that 0 is entered in the X and Z edit boxes.
 - Verify that 1 is entered in the Number edit box.
 - Click the **OK** button to proceed with the replication.
47. Click in the window titled X-Z Plane @ Y=0 to make sure it is active.

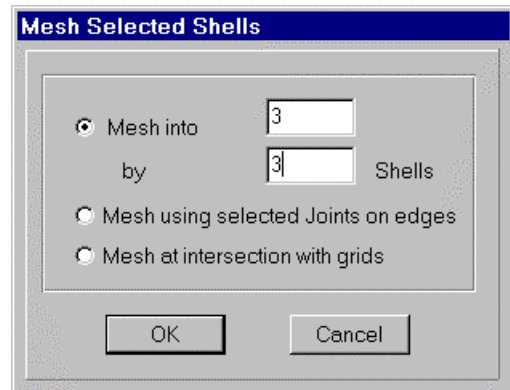
48. Click the **yz 2D View** button  to change the view to an Y-Z elevation. Note that the title of the window changes to Y-Z Plane @ X=0.
49. Select all of the elements in the Y-Z plane @ X=0 by “windowing”.
50. From the **Edit** menu select **Replicate...** to display the Replicate dialog box.
51. In this dialog box:
 - Verify the Linear Tab is selected.
 - In the Distance area type **21** in the X edit box.
 - In the Distance area type **0** in the Y edit box.
 - Verify that 0 is entered in the Z edit box.
 - Verify that 1 is entered in the Number edit box.
 - Click the **OK** button to proceed with the replication.
52. Click the **xy 2D View** button  to change the view to an X-Y plan. Note that the title of the window changes to X-Y Plane @ Z=0.
53. Select the four joints at this level either by “windowing” or by clicking on them individually.
54. From the **Assign** menu, choose **Joint**, and then **Restraints...** from the submenu. This will display the Joint Restraints dialog box.
55. In this dialog box:
 - Verify that the Translation 1, Translation 2 and Translation 3 boxes are checked.
 - Verify that the Rotation About 1, Rotation About 2 and Rotation About 3 boxes are *not* checked.
 - Click the **OK** button.
56. Click the **Show Undeformed Shape** button  to reset the window display from joint restraints to undeformed geometry. Note that the window title changes.
57. Click the **Up One Gridline** button  on the main toolbar to display the elevation view at Z=25.
58. Click the **Draw Rectangular Shell Element** button  on the side toolbar or select **Draw Rectangular Shell Element** from the **Draw** menu.
59. Click on joint 32 and then joint 14 to draw a shell element over the entire structure.


60. Click the **Pointer** button  to exit Draw Mode and enter Select Mode.

61. Click on the shell element to select it.

62. From the **Edit** menu select **Mesh Shells...** to display the Mesh Selected Shells dialog box.


63. Fill in this dialog box as shown in the adjacent figure and click the **OK** button.



64. Click the **Up One Gridline** button  on the main toolbar to display the elevation view at Z=37.

65. Click the **Draw Rectangular Shell Element** button  on the side toolbar or select **Draw Rectangular Shell Element** from the **Draw** menu.

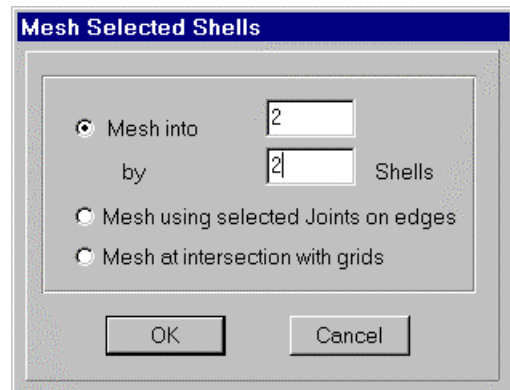
66. Click on joint 33 and then joint 15 to draw a shell element over the entire structure.


67. Click the **Pointer** button  to exit Draw Mode and enter Select Mode.

68. Click on the shell element to select it.

69. From the **Edit** menu select **Mesh Shells...** to display the Mesh Selected Shells dialog box.

70. Fill in this dialog box as shown in the adjacent figure and click the **OK** button.



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


72. In this dialog box:

- Uncheck the Labels box in the Joints area.
- Uncheck the Labels box in the Frames area.
- Click the **OK** button.


73. From the **Define** menu select **Static Load Cases...** This will display the Define Static Load Case Names dialog box.

74. 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 box.
 - Click the **Add New Load** button.
 - Click the **OK** button.
75. From the **Define** menu select **Materials...** to display the Define Materials dialog box.
76. 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 twice to exit all dialog boxes.
77. Click the drop down box in the status bar to change the units to kip-in. 
78. From the **Define** menu select **Materials...** to display the Define Materials dialog box.
79. In this 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.
 - Verify that the steel yield stress is 36.
 - Click the **OK** button twice to exit the dialog boxes.
80. From the **Define** menu select **Frame Sections...** to display the Define Frame Sections dialog box.
81. In this dialog box:





- Click the drop-down box that says Import I/Wide Flange and select the Import Angle option.
- 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.
- A dialog box appears with a list of all wide flange sections in the database. In this dialog box:
 - Scroll down and highlight the L4x4x3/4 by clicking on it.
 - Hold down the Shift key on the keyboard and click on the L4x4x1/4 angle. All of the L4x4 angles will now be selected (seven total).
 - Click the **OK** button twice to return to the Define Frame sections dialog box.
- Click the drop-down box that says Add I/Wide Flange and select the Add Auto Select option to display the Auto Selection Sections dialog box.
- In this dialog box:
 - Highlight all of the angles in the List of Sections list box by clicking on the top angle, pressing and holding down the shift key on the keyboard, and clicking on the bottom angle.
 - Click the **Add** button to add the angles to the Auto selections list box.
 - Click the **OK** button twice to exit all dialog boxes.




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








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





84. In this dialog box:

- Click the **Modify/Show Section** button to display the Shell Sections dialog box.
- In this dialog box:
 - Verify the Material specified is CONC.
 - In the Thickness area type **.6667** in both the Membrane and Bending edit boxes.
 - Verify that the Shell option is chosen in the Type area.
 - Click the **OK** button twice to exit all dialog boxes.

85. Click in the 3D View window to make sure it is active.
86. Click the **Select All** button  on the side toolbar to select all elements.
87. From the **Assign** menu select **Frame** and then **Sections...** from the submenu to display the Define Frame Sections dialog box.
88. In this dialog box:
 - Highlight the AUTO1 section.
 - Click the **OK** button.
89. Click the **Select All** button  on the side toolbar to select all elements.
90. From the **Assign** menu select **Shell Static Loads...** and then **Uniform...** from the submenu to display the Shell Uniform Loads dialog box.
91. In this dialog box:
 - Verify DL is selected in the Load Case Name drop-down box.
 - In the Uniform Load area type **-.05** (50 psf) in the Load edit box.
 - In the Uniform Load area verify that the Dir item is set to Global Z.
 - Click the **OK** button.
92. Click the **Select All** button  on the side toolbar to select all elements.
93. From the **Assign** menu select **Shell Static Loads...** and then **Uniform...** from the submenu to display the Shell Uniform Loads dialog box.
94. In this dialog box:
 - Select LL in the Load Case Name drop-down box.
 - In the Uniform Load area type **-.1** (100 psf) in the Load edit box.
 - Click the **OK** button.
95. Click the **Show Undeformed Shape** button  to remove the display of the shell static loads.
96. Click in the window labeled X-Y Plane @ Z=37 to activate it.
97. Select all of the elements in the plan view by “windowing”

98. From the **Assign** menu select **Joints** and then **Constraints...** from the submenu to display the Constraints dialog box.
99. In this dialog box:
 - In the Click To area click the drop-down box that says Add Body and select Add Diaphragm to display the Diaphragm Constraint dialog box.
 - In this dialog box:
 - Type **ROOF** in the Constraint Name edit box.
 - Verify that the Z Axis option is selected in the Constraint Axis area.
 - Click the **OK** button twice to exit all dialog boxes.
100. Click the **Show Undeformed Shape** button  to remove the display of the joint constraints and reset the window display.
101. Click the **Down One Gridline** button  to move to the X-Y Plane @ Z=25.
102. Select all of the elements in the plan view by “windowing”
103. From the Assign menu select Joints and the Constraints... from the submenu to display the Constraints dialog box.
104. In this dialog box:
 - In the Click To area click the drop-down box that says Add Body and select Add Diaphragm to display the Diaphragm Constraint dialog box.
 - In this dialog box:
 - Type **SECOND** in the Constraint Name edit box.
 - Verify that the Z Axis option is selected in the Constraint Axis area.
 - Click the **OK** button twice to exit all dialog boxes.
105. Click the **Show Undeformed Shape** button  to remove the display of the joint constraints and reset the window display.
106. From the **Analyze** menu select **Set Options...** to display the Analysis Options dialog box.
 - Check the Dynamic Analysis check box.
 - Click the **Set Dynamic Parameters** button to display the Dynamic Analysis Parameters dialog box.

- In this dialog box:
 - Type 3 in the Number of Modes edit box.
 - Click the **OK** button twice to exit all dialog boxes.
107. From the **Options** menu select **Preferences...** to display the Preferences dialog box.
108. In this dialog box:
- Select the Steel Tab.
 - Select AISC-ASD89 from the Steel Design Code drop-down box.
 - Click the **OK** button.
109. Click the **Run Analysis** button  to run the analysis.
110. 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.
111. From the **Design** menu select **Start Design/Check of Structure** to initiate the design. The design proceeds and when it is complete P-M interaction ratios are displayed.
112. Click the **Set Elements** button  on the main toolbar (or select **Set Elements...** from the **View** menu) to display the Set Elements Dialog box.
113. In this dialog box:
- Check the Sections box in the Frames area.
 - Click the **OK** button. The sections chosen by the program are displayed.
- Note: You may want to zoom in using the Rubber Band Zoom button  on the main toolbar to see the chosen sections better.*
114. Click the **Show Undeformed Shape** button  to remove the display of frame sections and interaction values.
115. If you have zoomed in for a better view of the chosen sections, then click the Restore Full View button  on the main toolbar.
116. Click the **Select All** button  on the side toolbar to select all elements.
117. From the **Design** menu select **Replace Auto W/ Optimal Sections** to update the frame sections from Auto sections to the chosen angle sizes. Click **OK** when it says it will unlock the model and asks if it is OK to update.
118. Click the **Run Analysis** button  to run the analysis using the optimal sections.

119. 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. Note that the 3-D window now shows the first mode shape.
120. Click the **Start Animation** button  , located in the status bar at the bottom of the SAP2000 window, to animate the mode shape.
121. Click the **Right Arrow** button  , located in the status bar at the bottom of the screen, to view the second mode shape.
122. Click the **Right Arrow** button  again to view the third mode shape.
123. Click the **Stop Animation** button  , located in the status bar at the bottom of the SAP2000 window, to stop the mode shape animation.
124. From the **Design** menu select **Start Design/Check of Structure** to initiate a final design check of the structure based on the analysis results using the optimal sections. The design proceeds and when it is complete the final P-M interaction ratios are displayed.