

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

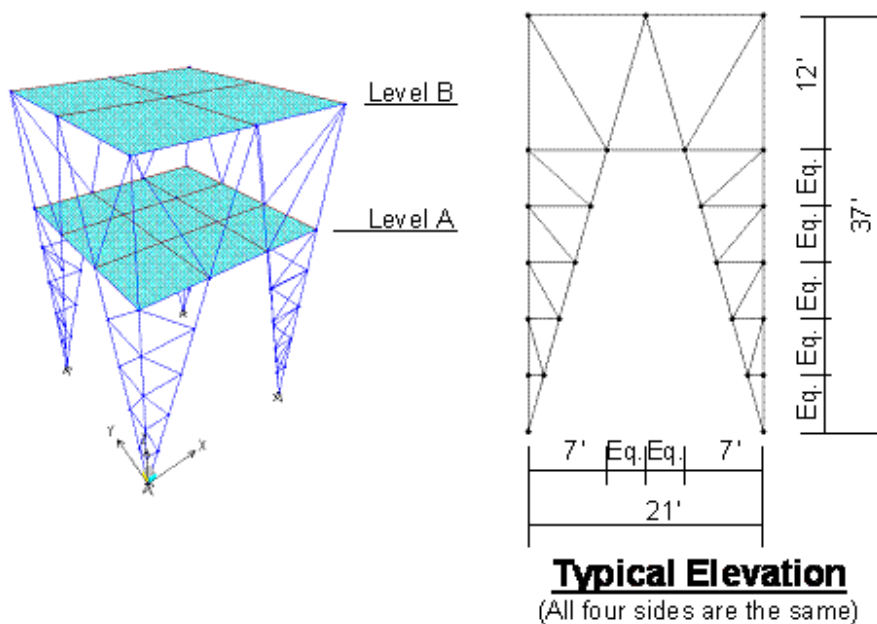
Additional dead load at each diaphragm is 50 psf

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

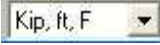


CSI Solution Demonstrates Use of These Features


- Diaphragm Constraint
- Design Optimization
- Automatic Area Mesh
- Mode Shapes
- New Model (not from template, started from scratch)
- Linear Replication

- Mirror Replication
- Radial Replication
- Steel Design

Problem C Solution

1. Click the **File menu > New Model** command to displays the **New Model** form.
2. Click the drop-down box to set the units to .



3. Click the **Blank** button .
4. Click the **Define menu > Coordinate Systems/Grids** command to display the **Coordinate/Grid Systems** form.
5. In that form click on the **Modify/Show System** button to display the **Define Grid Data** form. Fill in the form as shown in Figure C-1. Type values in the Grid ID and Ordinate cells; click in the Line Type, Visibility, and Bubble Loc. cells until the appropriate option appears:

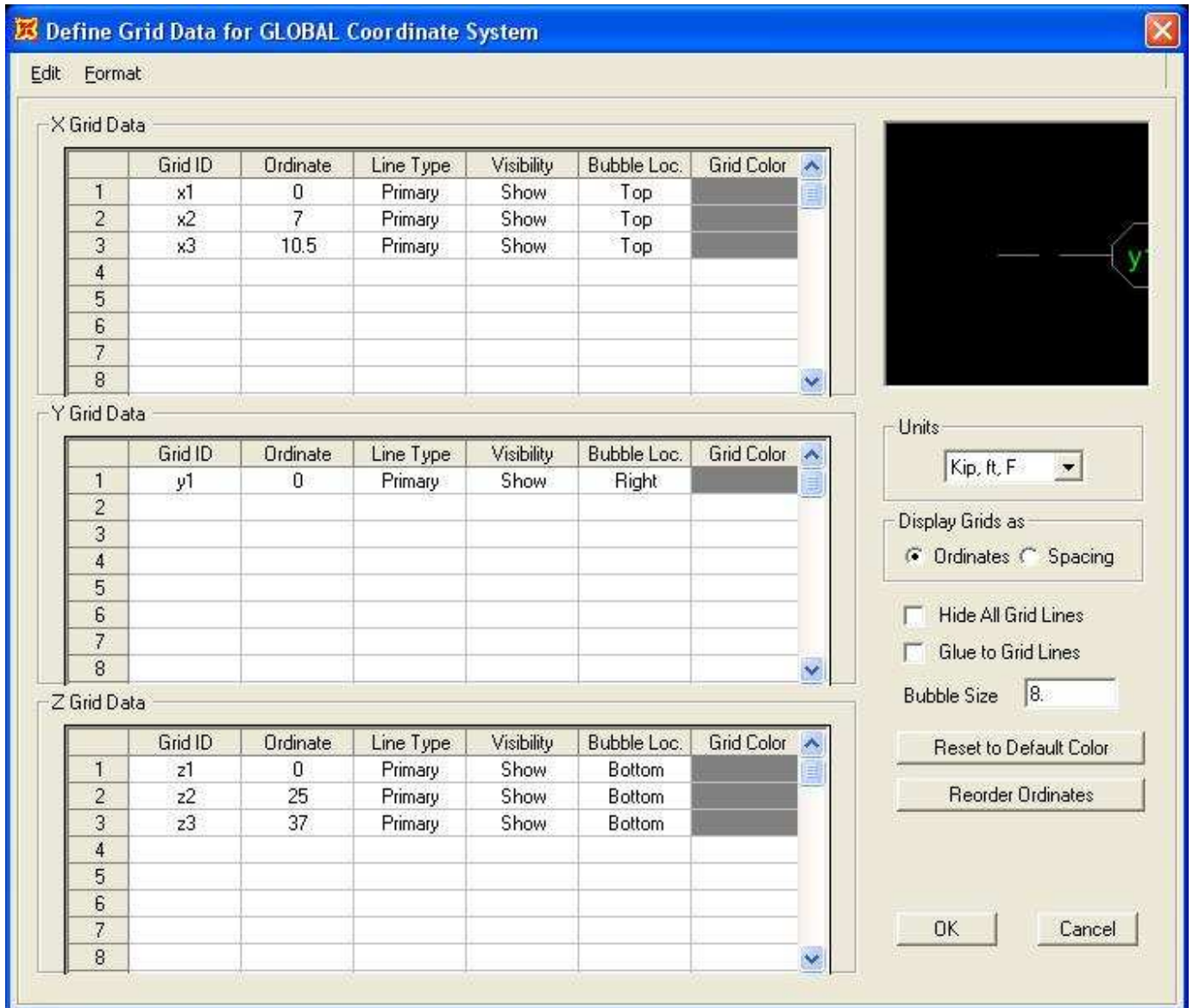





Figure C-1 Defining the Grid System

- Click the **OK** buttons on the **Coordinate/Grid System Data** and **Coordinate/Grid Systems** forms to exit the forms.
6. Click in the left window entitled 3-D View to make sure it is active. The window is highlighted when it is active.
 7. Click the **Set Default 3D View** button  to change to the default 3-d view.
 8. Click in the right window entitled 3-D View to make sure it is active. Click the **Set XZ 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.

9. Click the **Quick Draw Frame/Cable/Tendon** button  or click the **Draw menu > Quick Draw Frame/Cable/Tendon** command to display the **Properties of Object** form. By default, the **Line Object Type** should be **Straight Frame**. If it is not, click in that box to display the drop-down list and click on **Straight Frame** to select it. We will ignore the other property settings shown because other assignments will be made later in the modeling process.
10. Click on the grid line at the point labeled “A” in Figure C-2 to enter a frame object.
11. Click on the grid line at the point labeled “B” in Figure C-2 to enter another frame object.

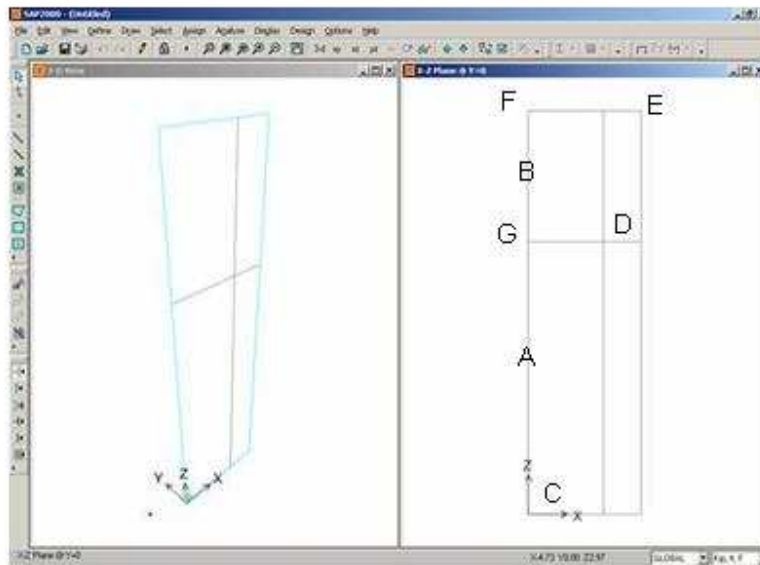





Figure C-2 Initial Grid Layout in X-Z Plane

12. Click the **Draw Frame/Cable/Tendon Element** button  or the **Draw menu > Draw Frame/Cable/Tendon** command to display the **Properties of Object** form. Again, be sure that the **Line Object Type** is **Straight Frame**. We will ignore the other property settings shown in the form because other assignments will be made later in the modeling process.
13. Click on the points labeled “C,” “D,” and “E” in Figure C-2, in that order, and then press the Enter key on the keyboard to draw two more frame objects.
14. Click on the point labeled “F” and then double click the point labeled “E” in Figure C-2 to draw the next frame object.

Note: You could have single-clicked the point labeled “E” in Figure C-2 and then pressed the Enter key on the keyboard to finish drawing the frame object.
15. Click on the point labeled “G” and then double click the point labeled “D” in Figure C-2 to draw the next frame object.
16. Click on the point labeled “D” and then double click the point labeled “F” in Figure C-2 to draw the next frame object.

17. Click the **Set Select Mode** button  to exit Draw Mode and enter Select Mode.
18. Click the **Set Display Options** button  (or the **View menu > Set Display Options** command) to display the **Display Options for Active Window** form. In that form:
 - Check the *Labels* box in the *Joints* area.
 - Check the *Labels* box in the *Frames/Cables/Tendon* area.
 - Check the *Fill Objects* box.
 - Click the **OK** button. The screen appears as shown in Figure C-3.

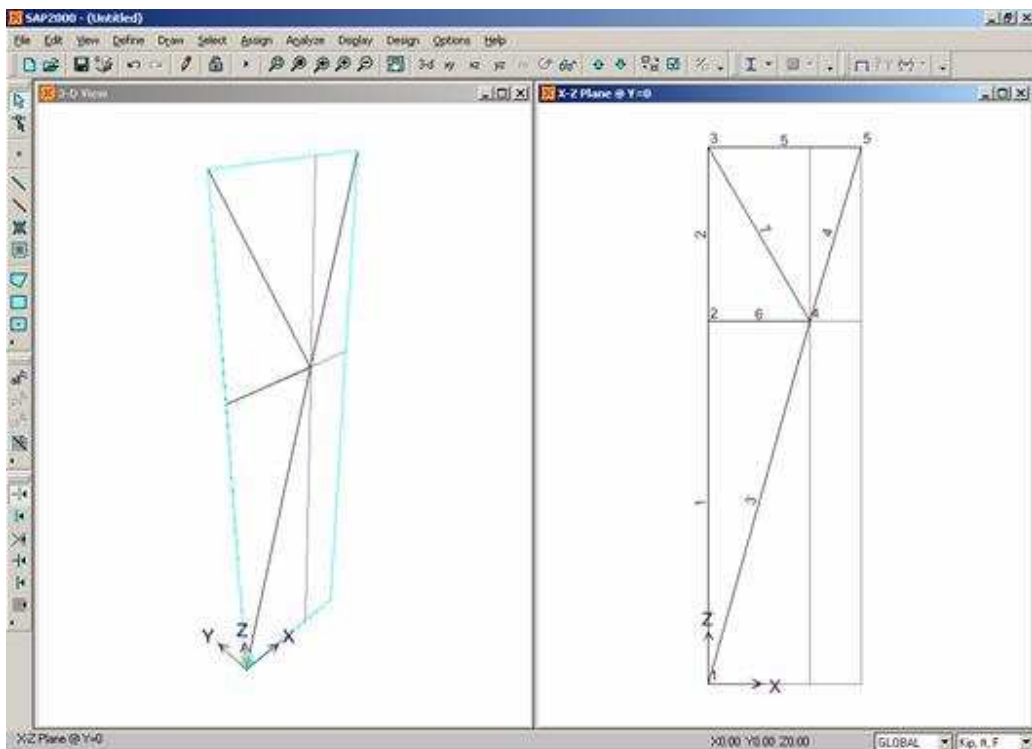



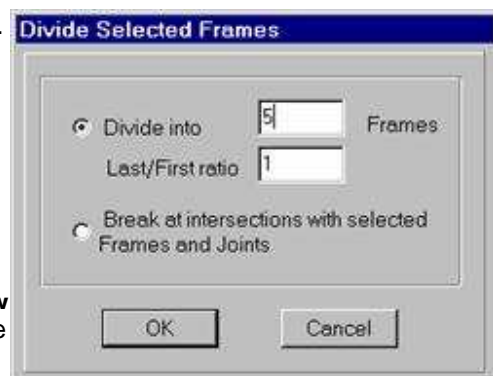
Figure C-3 Screen as It Appears After Step 18

19. Click on line objects 1 and 3 to select them.
20. Click the **Edit menu > Divide Frames** command to display the **Divide Selected Frames** form. Verify that this form is filled out as shown in the adjacent figure and click the **OK** button.
21. Click the **Draw Frame/Cable/Tendon**



button  or click the **Draw menu > Draw Frame/Cable/Tendon** command. Again, we can ignore the current property settings displayed in the **Properties of Object** form.

As an alternate method, we could have defined the sections before drawing the frame objects, and then assigned the sections when drawing using the **Properties of Objects** form.



22. Draw frame objects by single clicking then double clicking on the joints identified in the following table:

Click on This Joint	Double Click on This Joint
9	13
8	12
7	11
6	10
13	2
12	9
11	8
10	7

23. Click the **Set Select Mode** button  to exit Draw Mode and enter Select Mode.

24. Click in the Window labeled X-Z Plane @ Y=0 to activate it.


25. Click the **Select All** button  to select all objects.

26. Click the **Edit menu > Replicate** command to display the **Replicate** form. In that form:

- Select the Mirror Tab.
- In the *Mirror About Plane* area select *Parallel to Y*.
- In the *Intersection of Plane with XZ Plane* area type **10.5** in the *x1* edit box.
- Type **10.5** in the *x2* edit box.
- Click the **OK** button to proceed with the replication.





27. Click the **Draw Frame/Cable/Tendon Element** button  or the **Draw menu > Draw Frame/Cable/Tendon** command.





28. Click on joint 4 and then double click on joint 17 to draw a frame object.

Note: If the font size for the joint labels is too small, use the following procedure to increase the font size. Click the **Options menu > Preferences > Dimensions/Tolerances** command and type in a new (larger) font size in the *Minimum Graphic Font Size* edit box (usually 6 points is sufficient), click the **OK** button and then click the **Refresh Window** button .

To read a particular joint label, right click on the joint to bring up a form that displays the joint label.

29. Click the **Set Select Mode** button  to exit Draw Mode and enter Select Mode.

30. Click the **Select All** button  to select all objects.
31. Click the **Edit menu > Replicate** command to display the **Replicate** form. In that form:
 - Select the *Radial Tab*.
 - In the *Rotate About Line* area select the *Parallel to Z* option.
 - In the *Intersection of Line with XY Plane* area verify that *X* is 0 and *Y* is 0.
 - In the *Increment Data* area verify that the *Number* is 1 and the *Angle* is 90.
 - Click the **OK** button to proceed with the replication.
32. Click the **Get Previous Selection** button .
33. Click the **Edit menu > Replicate** command to display the **Replicate** form. In that form:
 - Verify the *Linear Tab* is selected.
 - In the *Increments* area type **21** in the *dy* edit box.
 - Verify that 0 is entered in the *dx* and *dz* edit boxes.
 - Verify that 1 is entered in the *Number* edit box.
 - Click the **OK** button to proceed with the replication.
34. Click in the window entitled X-Z Plane @ Y=0 to make sure it is active.
35. Click the **Set YZ View** button  to change the view to a Y-Z elevation. Note that the title of the window changes to Y-Z Plane @ X=0.
36. Select all of the elements in the Y-Z plane @ X=0 by “windowing.”
37. Click the **Edit menu > Replicate** command to display the **Replicate** form. In that form:
 - Verify the *Linear Tab* is selected.
 - In the *Increments* area type **21** in the *dx* edit box.
 - In the *Increments* area type **0** in the *dy* edit box.
 - Verify that 0 is entered in the *dz* edit box.
 - Verify that 1 is entered in the *Number* edit box.
 - Click the **OK** button to proceed with the replication.
38. Click the **Set XY 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.
39. Select the four joints at this level by “windowing” or clicking on them individually.
40. Click the **Assign menu > Joint > Restraints** command to display the **Joint Restraints** form. In that form:
 - 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.

41. Click the **Show Undeformed Shape** button  to reset the window display from joint restraints to undeformed geometry. Note that the window title changes.
42. Click the **Move Up in List** button  to display the plan view at Z=25.
43. Click the **Draw Rectangular Area Element** button  on the side toolbar or the **Draw menu > Draw Rectangular Area** command to display the **Properties of Object** form. Again, we can ignore the current property setting.
44. Click on joint 33 and then joint 15 to draw an area object over the entire plan.
45. Click the **Set Select Mode** button  to exit Draw Mode and enter Select Mode.
46. Click on the area object to select it.
47. Click the **Assign menu > Area > Automatic Area Mesh** command to display the **Assign Automatic Area Mesh** form.
48. Fill in this form as shown in Figure C-4 and click the **OK** button.

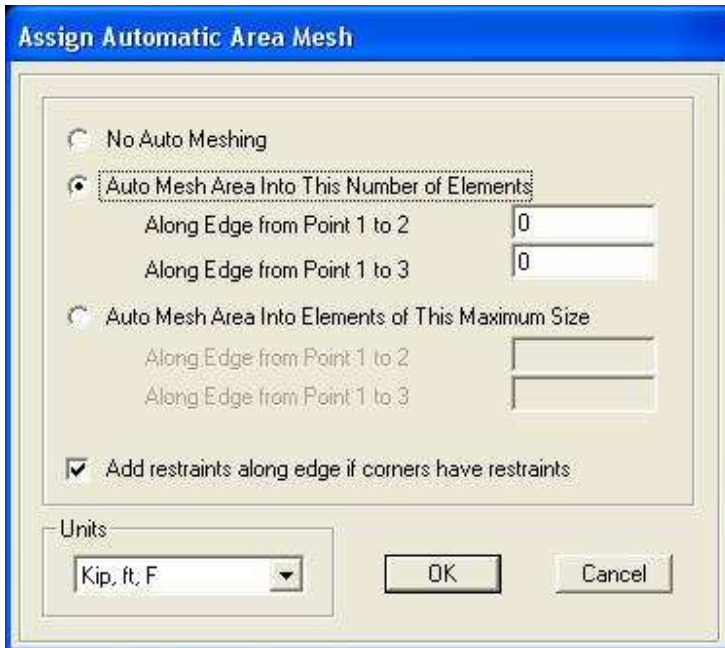








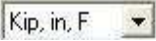
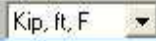
Figure C-4 Assign Automatic Area Mesh 2 x 2





49. Click the **Show Undeformed Shape** button  to reset the window display.
50. Click the **Move Up in List** button  to display the elevation view at Z=37.
51. Click the **Draw Rectangular Area Element** button  or the **Draw menu > Draw Rectangular Area** command to display the **Properties of Object** form.
52. Click on joint 34 and then joint 16 to draw an area object over the entire plan.
53. Click the **Set Select Mode** button  to exit Draw Mode and enter Select Mode.





54. Click on the area object to select it.
55. Click the **Assign menu > Area > Automatic Area Mesh** command to display the **Assign Automatic Area Mesh** form.
56. Fill in this form as shown in the adjacent figure and click the **OK** button.

Figure C-5 Assign Automatic Area Mesh 3 x 3

57. Click the **Show Undeformed Shape** button  to reset the window display.
58. Click the **Set Display Options** button  (or the **View menu > Set Display Options** command) to display the **Display Options for Active Window** form. In that form:
 - Uncheck the *Labels* box in the *Joints* area.
 - Uncheck the *Labels* box in the *Frames/Cables/Tendon* area.
 - Click the **OK** button.
59. Click the **Define menu > Load Cases** command to display the **Define Loads** form. In that form:
 - Type **LIVE** in the *Load Name* edit box.
 - Select *LIVE* from the *Type* drop-down box.
 - Verify that **0** is in the *Self Weight Multiplier* edit box.
 - Click the **Add New Load** button.
 - Click the **OK** button.
60. Click the **Define menu > Materials** command to display the **Define Materials** form. In that form:
 - Highlight the **CONC** material and click the **Modify/Show Material** button to display the **Material Property Data** form. In that form:




- Verify that the Mass per Unit Volume is 4.662E-03.
 - Verify that the Weight per Unit Volume is 0.15.
 - Click the **OK** buttons on the **Material Property Data** and **Define Materials** forms to exit all forms.
61. Click the drop-down box in the status bar to change the units to .
62. Click the **Define menu > Materials** command to display the **Define Materials** form. In that form:
- Highlight the STEEL material and click the **Modify/Show Material** button to display the **Material Property Data** form. In that form:
 - 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** buttons on the **Material Property Data** and **Define Materials** forms to exit the forms.
63. Click the **Define menu > Frame Sections** command to display the **Frame Properties** form. In that form:
- Click the drop-down box that reads *Import I/Wide Flange* and select the *Import Angle* option.
 - Click the **Add New Property** button to display the **Section Property File** form. 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 form appears with a list of all angle sections in the database. In that form:
 - Scroll down and highlight the *L4x4x1/2* by clicking on it.
 - Hold down the Shift key on the keyboard and click on the *L4x4x7/16* angle. All of the *L4x4* angles will now be selected (seven total).
 - Click the **OK** buttons on the two forms to return to the **Frame Properties** form.
 - Click the drop-down box that reads *Add I/Wide Flange* and select the *Add Auto Select* option.
 - Click the **Add New Property** button to display the **Auto Selection Sections** form. In that form:
 - 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** buttons on the **Auto Selection Sections** and **Frame Properties** forms to exit all forms.
64. Click the drop-down box in the status bar to change the units to .
65. Click the **Define menu > Area Sections** command to display the **Area Sections** form. In that form:

- Click the **Modify/Show Section** button to display the **Area Section Data** form. In that form:
 - Verify the *Material* specified is *CONC*.
 - Verify that the *Shell* option is selected in the *Area Type* area.
 - 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** buttons on the **Area Section Data** and **Area Sections** forms to exit all forms.
66. Click in the 3D View window to make sure it is active.
67. Click the **Select All** button  on the side toolbar to select all objects.
68. Click the **Assign menu > Frame/Cable/Tendon > Frame Sections** command to display the **Frame Properties** form. In that form:
 - Highlight the *AUTO1* section.
 - Click the **OK** button.
69. Click the **Select All** button  to select all elements.
70. Click the **Assign menu > Area Loads > Uniform (Shell)** command to display the **Area Uniform Loads** form. In that form:
 - Verify *DEAD* 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 *Direction* item is set to *Gravity*.
 - Click the **OK** button.
71. Click the **Select All** button  on the side toolbar to select all objects.
72. Click the **Assign menu > Area Loads > Uniform (Shell)** command to display the **Area Uniform Loads** form. In that form:
 - Select *LIVE* 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.
73. Click the **Show Undeformed Shape** button  to remove the display of the area loads.
74. Click in the window labeled X-Y Plane @ Z=37 to activate it.
75. Select all of the objects in the plan view by “windowing.”
76. Click the **Assign menu > Joint > Constraints** command to display the **Assign/Define Constraints** form. In that form:
 - In the *Choose Constraint Type for Add* area, click the drop-down box that reads *Body* and select *Diaphragm*.

- Click the **Add New Constraint** button to display the **Diaphragm Constraint** form. In that form:
 - Type **ROOF** in the *Constraint Name* edit box.
 - Verify that the *Z Axis* option is selected in the *Constraint Axis* area.
 - Click the **OK** buttons on the **Diaphragm Constraint** and the **Assign/Define Constraints** forms to exit all forms.
77. Click the **Show Undeformed Shape** button  to remove the display of the joint constraints and reset the window display.
78. Click the **Move Down in List** button  to move to the X-Y Plane @ Z=25.
79. Select all of the objects in the plan view by “windowing.”
80. Click the **Assign menu > Joint > Constraints** command to display the **Assign/Define Constraints** form. In that form:
- In the *Choose Constraint Type for Add* area, click the drop-down box that reads *Body* and select *Diaphragm*.
 - Click the **Add New Constraint** button to display the **Diaphragm Constraint** form. In that form:
 - Type **SECOND** in the *Constraint Name* edit box.
 - Verify that the *Z Axis* option is selected in the *Constraint Axis* area.
 - Click the **OK** buttons on the **Diaphragm Constraint** and the **Assign/Define Constraints** forms to exit all forms.
81. Click the **Show Undeformed Shape** button  to remove the display of the joint constraints and reset the window display.
82. Click the **Define menu > Analysis Cases** command to display the **Analysis Cases** form. In that form:
- Click on *MODAL* in the *Case Name* list to highlight it.
 - Click the **Modify/Show Case** button to display the **Analysis Case Data** form. In that form:
 - In the *Number of Modes* area, type **3** in the *Maximum Number of Modes* edit box.
 - Click the **OK** buttons on the **Analysis Case Data** and **Analysis Cases** forms to exit all forms.
83. Click the **Options menu > Preferences > Steel Frame Design** command to display the **Steel Frame Design Preferences** form. In that form:
- Select *AISC-ASD89* from the *Design Code* drop-down box.
 - Click the **OK** button.
84. Click the **Run Analysis** button  to display the **Set Analysis Cases to Run** form. In that form:
- Verify that the *DEAD*, *MODAL* and *LIVE* analysis cases are all set to *Run* in the *Action* list.


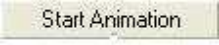


- Click the **Run Now** button to run the analysis.
85. When the analysis is complete, check the messages in the **SAP Analysis Monitor** window (there should be no warnings or errors) and then click the **OK** button to close the **Analysis** window.
 86. Click the **Design menu > Steel Frame Design > Start Design/Check of Structure** command to initiate the design. The design proceeds and when it is complete P-M interaction ratios are displayed.
 87. Click the **Design menu > Steel Frame Design > Display Design Info** command to display the **Display Steel Design Results** form. In that form:
 - Select the *Design Input* option. Verify that *Design Sections* is displayed in the drop-down box.
 - Click the **OK** button. The sections chosen by the program are displayed.

Note: Zoom in using the **Rubber Band Zoom** button  to view the chosen sections better.

88. Click the **Show Undeformed Shape** button  to remove the display of frame sections and interaction colors.
89. If you have zoomed in for a better view of the chosen sections, then click the **Restore Full View** button .
90. Click the **Design menu > Steel Frame Design > Verify Analysis vs Design Section** command to display the number of members that changed sections between analysis and design. If this number is greater than 1, the analysis should be re-run with the sections chosen during design. Click **NO** to close the form.
91. Click the **Run Analysis** button  to display the **Set Analysis Cases to Run** form. In that form:
 - Verify that the DEAD, MODAL and LIVE analysis cases are all set to *Run in the Action* list.
 - Click the **Run Now** button to run the analysis with the updated design sections.
93. When the analysis is complete check the messages in the SAP Analysis Monitor window (there should be no warnings or errors) and then click the **OK** button to close the Analysis window.
94. Click the **Design menu > Steel Frame Design > Start Design/Check of Structure** command to initiate the design.
95. Click the **Design menu > Steel Frame Design > Verify Analysis vs Design Section** command to display the number of members that changed sections between analysis and design. Steps 91 through 94 should be repeated until the adjacent form appears. Click **OK** to close the form.



match, the [Auto Select Sections list](#) will need to be altered to include larger sections. Click the **NO** button to close the form.

97. Click the **Show Deformed Shape** button  (or the **Display menu > Show Deformed Shape** command) to display the *Deformed Shape* form. In that form:
 - Verify that *MODAL* is selected in the *Case/Combo Name* drop-down box.
 - Type **1** in the *Mode Number* edit box.
 - Click the **OK** button.
98. Click the **Start Animation** button , located in the status bar at the bottom of the SAP2000 window, to animate the mode shape.
99. Click the **Right Arrow** button , located in the status bar at the bottom of the screen, to view the second mode shape.
100. Click the **Right Arrow** button again to view the third mode shape.
101. Click the **Stop Animation** button , located in the status bar at the bottom of the SAP2000 window, to stop the mode shape animation.