

Problem X

Through Truss Bridge

Steel

$E = 29000$ ksi

Poissons Ratio = 0.3

All members are W6X12

$F_y = 36$ ksi

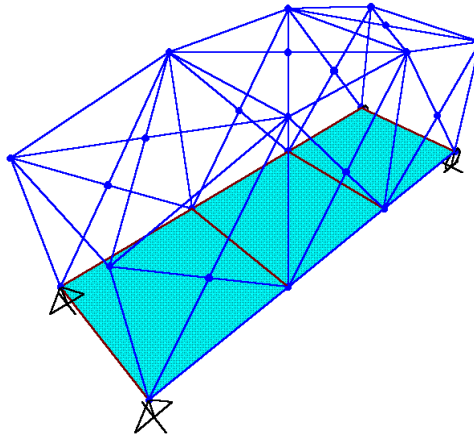
Concrete Bridge Deck

$E = 3600$ ksi

Poissons Ratio = 0.2

12 inches thick

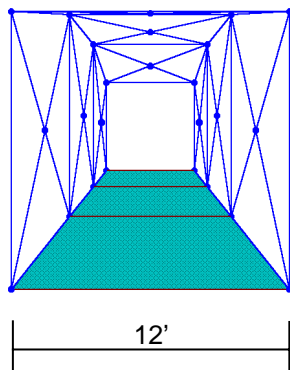
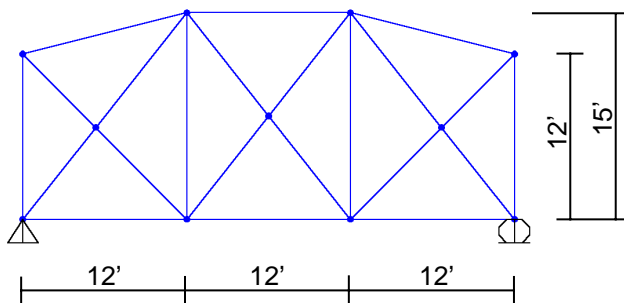
Live Load = 250 psf



To Do


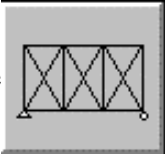

Review steel member stresses due to self weight plus live load.


Use AISC-ASD89.



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 X 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 **Vertical Truss** template  button to display the Vertical Truss dialog box.
4. In this dialog box:
 - Accept the default number of bays, 3.
 - Accept the default Height of Truss, 12.
 - Accept the default Truss Bay Length, 12.
 - Click the **OK** button.
5. Click the **Select All** button  on the side toolbar to select all elements.
6. From the **Edit** menu select **Replicate...** to display the Replicate dialog box.
7. In this dialog box:
 - Verify the Linear Tab is selected.
 - In the Distance area type **12** 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.
8. From the **Draw** menu select **Edit Grid...** to display the Modify Grid Lines dialog box.
9. In this dialog box:
 - Select the Y option in the Direction area.
 - Type **12** in the Y Location edit box and click the **Add Grid Line** button.
 - Click the **OK** button.

- 10. Click in the window titled X-Z Plane @ Y=0 to activate it. The window is activated when its title bar is highlighted.
- 11. 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=12. The screen appears as shown in Figure X-1.

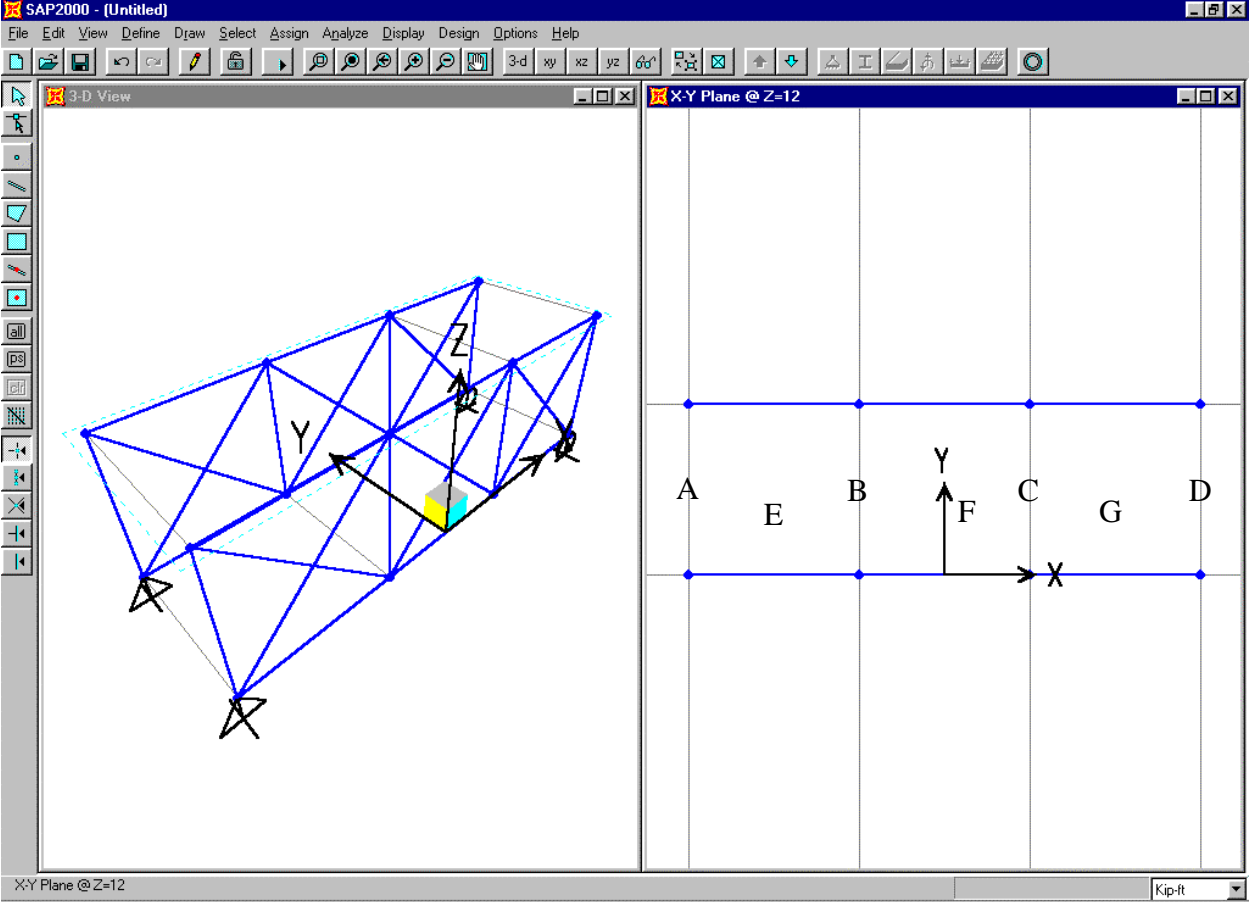





Figure X-1: View of Screen After Step 11

- 12. Click the **Quick Draw Frame Element** button  on the side toolbar or select **Quick Draw Frame Element** from the **Draw** menu.
- 13. Click on the grid lines at the points labeled “A”, “B”, “C” and “D” in Figure X-1 to enter four frame elements spanning between the two vertical frames.
- 14. Click on the grid lines at the points labeled “E”, “F” and “G” in Figure X-1 to enter three sets of diagonal frame elements spanning between the two vertical frames.
- 15. Click the **Down One Gridline** button  on the main toolbar to display the X-Y Plane @ Z=0. The screen appears as shown in Figure X-2.
- 16. Click the **Quick Draw Rectangular Shell Element** button  on the side toolbar (or select **Quick Draw Rectangular Shell Element** from the **Draw** menu).

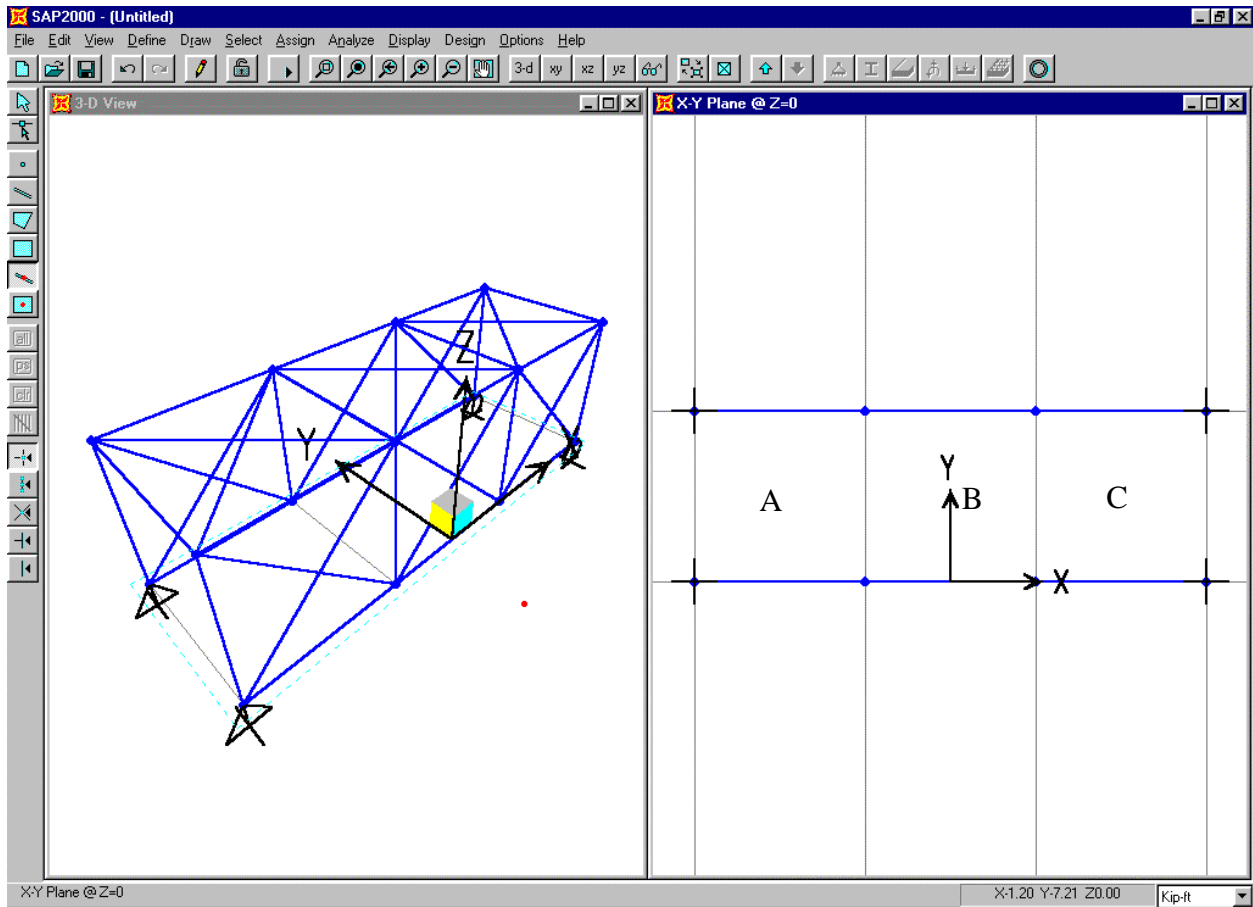

















Figure X-2: View of Screen After Step 15

17. Click on the points labeled “A”, “B” and “C” in Figure X-2 to enter three shell elements spanning between the two vertical frames.
18. Click the **Pointer** button  on the side tool bar to exit draw mode and enter select mode.
19. Click in the window titled X-Y Plane @ Z=0 to make sure it is active.
20. Click the **Up One Gridline** button  on the main toolbar to display the plan view at Z=12.
21. Select the center four joints by clicking on them.
22. From the **Edit** menu select **Move...** to display the Move Selected Points dialog box.
23. In this dialog box:
 - Type **3** in the Delta Z edit box.
 - Click the **OK** button.

24. Click the **Perspective Toggle** button  on the main toolbar. A perspective birds-eye view of the structure is displayed.
25. Click the **Set Intersecting Line Select Mode** button  and select all of the roof and floor elements by “drawing” a line through them. There should now be 10 frames and 3 shells selected. You can verify this by looking at the left-hand side of the status bar at the bottom of the SAP2000 window.
26. From the **Edit** menu select **Divide Frames...** to display the Divide Selected Frames dialog box.
27. In this dialog box:
 - Click on the Break At Intersections With Selected Frames and Joints option to select it.
 - Click the **OK** button to add a center joint at each set of cross braces in the roof.
28. Click on the **xz 2D View** button  on the main toolbar to view an elevation in the X-Z plane. Note the title of the window is X-Z Plane @ Y=0.
29. Click the **Perspective Toggle** button  on the main toolbar. A perspective view of the structure is displayed.
30. Click the **Set Intersecting Line Select Mode** button  and select all of the vertical and diagonal elements by “drawing” a line through them. There should now be 20 frames selected. You can verify this by looking at the left-hand side of the status bar at the bottom of the SAP2000 window.
31. From the **Edit** menu select **Divide Frames...** to display the Divide Selected Frames dialog box.
32. In this dialog box:
 - Verify that the Break At Intersections With Selected Frames and Joints option is selected.
 - Click the **OK** button to add a center joint at each of the six sets of vertical cross braces.
33. Click on the **xz 2D View** button  on the main toolbar to view an elevation in the X-Z plane.
34. Click the drop down box in the status bar to change the units to kip-in. 
35. From the **Define** menu select **Materials...** to display the Define Materials dialog box.
36. Click on STEEL in the Materials area to highlight (select) it, and then click the **Modify/Show Material** button. The Material Property Data dialog box is displayed.

37. In this dialog box:
 - Verify that the Modulus of Elasticity is 29000.
 - Verify that Poisson's Ratio is 0.3.
 - Verify that the steel stress, F_y is 36.
 - Click the **OK** button.
38. 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.
39. In this dialog box:
 - Verify that the Modulus of Elasticity is 3600.
 - Verify that Poisson's Ratio is 0.2.
 - Click the **OK** button twice to exit all dialog boxes.
40. Click the drop down box in the status bar to change the units to kip-ft. 
41. From the **Define** menu select **Materials...** to display the Define Materials dialog box.
42. 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.
43. In this dialog box:
 - Verify that the Weight per Unit Volume is 0.15.
 - Click the **OK** button twice to exit all dialog boxes.
44. From the **Define** menu select **Frame Sections...** to display the Define Frame Sections dialog box.
45. 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 I/Wide Flange 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. 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 click on the W6X12 section.
 - Click the **OK** button three times to exit all dialog boxes.
46. From the **Define** menu select **Shell Sections...** to display the Define Shell Sections dialog box.
 47. In the Click To area, click the **Modify/Show Section** button to display the Shell Sections dialog box.
 48. In this dialog box:
 - Verify that the Material chosen is CONC.
 - Verify that both the Membrane and the Bending thicknesses are 1.
 - Verify that the Shell option is selected in the Type area.
 - Click the **OK** button twice to exit all dialog boxes.
 49. Click the **Select All** button  on the side toolbar to select all elements.
 50. From the **Assign** menu select **Frame** and then **Sections...** from the submenu to display the Define Frame Sections dialog box.
 51. In this dialog box:
 - Click on W6X12 in the Frame Sections area to highlight it.
 - Click the **OK** button.
 52. Click the **Show Undeformed Shape** button  to remove the displayed frame section assignments.
 53. From the **Define** menu select **Static Load Cases...** to display the Define Static Load Case Names dialog box.
 54. 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.
 - Click the **OK** button.
55. Click the **Select All** button  on the side toolbar to select all elements.
 56. From the **Assign** menu select **Shell Static Loads...** and then **Uniform...** from the submenu. This will display the Shell Uniform Loads dialog box.
 57. In this dialog box:
 - Select LL from the Load Case Name drop-down box.
 - In the Uniform Load area type **-.25** in the Load box and select Global Z from the drop-down Dir box.
 - In the Options area verify that the Add To Existing Loads option is selected.
 - Click the **OK** button to apply the load.
 58. Click the **Show Undeformed Shape** button  to reset the window display.
 59. Click the “X” in the upper right-hand corner of the window labeled X-Z Plane @ Y=0 to close it.
 60. Click the **Run Analysis** button  to run the analysis.
 61. 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.
 62. From the **Options** menu select **Preferences...** to display the Preferences dialog box.
 63. In this dialog box.
 - Click on the Steel Tab
 - Select AISC-ASD89 from the Steel Design Code drop-down box if it is not already selected.
 - Click the **OK** button.
 64. From the **Design** menu click **Start Design/Check Of Structure** to run the design check of the steel frame elements.
 65. When the design check completes, the stress ratios are displayed.