

Problem X

Through Truss Bridge

Steel

$E = 29000$ ksi

Poissons Ratio = 0.3

All members are W6X12

$F_y = 36$ ksi

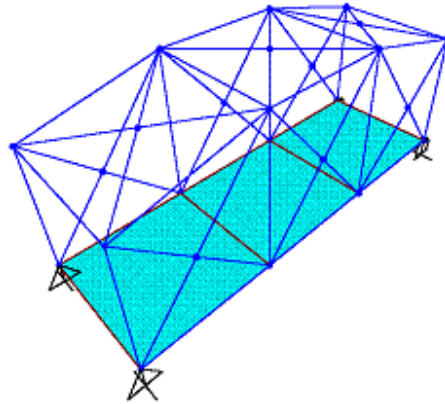
Concrete Bridge Deck

$E = 3600$ ksi

Poissons Ratio = 0.2

12 inches thick

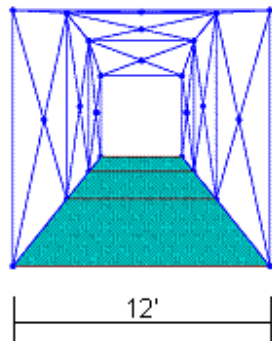
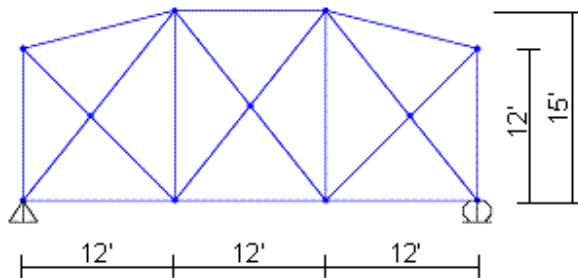
Live Load = 250 psf



To Do

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

Use AISC-ASD89.

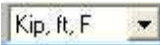


CSI SOLUTION DEMONSTRATE USE OF THESE FEATURES

- Divide Frames
- Grid Lines
- Linear Replication
- Steel Design

Problem X Solution

1. Click the **File menu > New Model** command to display the **New Model** form.

2. Click the drop-down box to set the units to .



3. Click on the **2D Trusses** button to display the **2D Truss** form. In that form:

- Select *Vertical Truss* in the *2D Truss Type* drop-down list.
- Accept the default *Number of Divisions*, 3.
- Accept the default *Height*, 12.
- Accept the default *Division Length*, 12.
- Click the **OK** button.

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


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

- Verify that the *Linear* tab is selected.
- In the *Increments* area, type **12** 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.

6. Click the **Define menu > Coordinate Systems/Grids** command to display the **Coordinate/Grid Systems** form. In that form:

- Click the **Modify/Show System** button to display the **Define Grid** form. In that form:
 - Verify that the *Ordinates* option is selected in the *Display Grids as* area in the middle right of the form.
 - In the Y Grid Data portion of the spreadsheet, type **y2** in the *Grid ID* cell of row 2. In that same row, type **12** in the *Ordinate* cell and click in the *Line Type* cell to display *Primary*, the *Visibility* cell to display *Show* and in the *Bubble Loc.* cell to display *Right*.
 - Click the **OK** buttons on the **Define Grid** and **Coordinate/Grid Systems** forms to exit all forms.

7. Click in the window entitled Y-Z Plane @ X=-18 to activate it. The window is activated when its title bar is highlighted.

8. Click the **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=12. The screen appears as shown in Figure X-1.

9. Click the **Quick Draw Frame/Cable/Tendon** button  on the side toolbar or the **Draw menu > Quick Draw Frame/Cable/Tendon** command to display the **Properties**

of Object form. We can ignore the property settings shown in this form as other assignments will be made later in the modeling process.

10. Click on the grid lines at the points labeled “A,” “B,” “C” and “D” in Figure X-1 to enter four frame objects spanning between the two vertical frames.

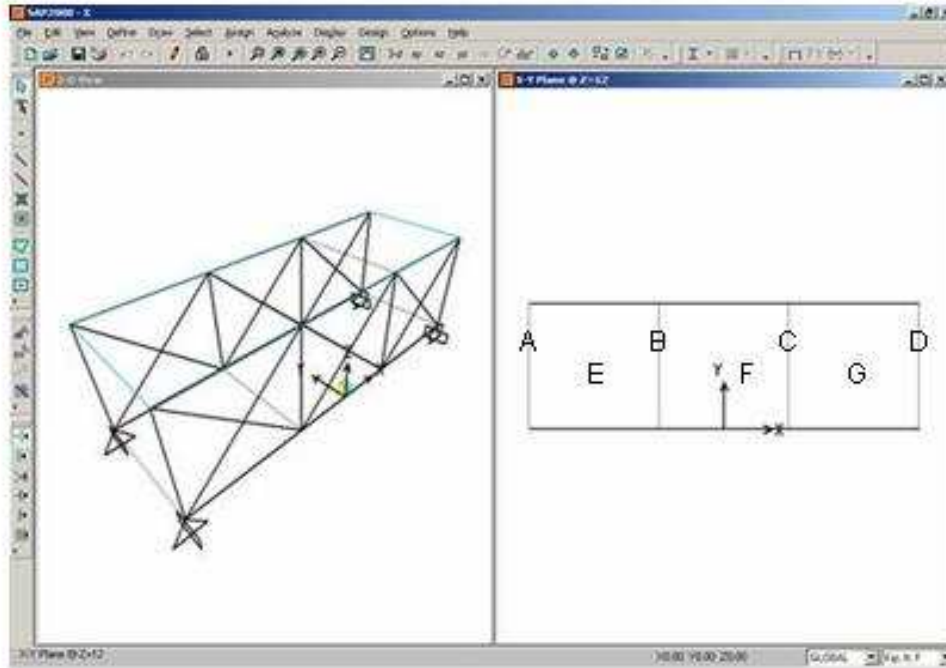





Figure X-1: View of Screen After Step 10

11. Click the **Set Select Mode** button  on the side toolbar to exit Draw mode and enter Select mode.
12. Click the **Quick Draw Braces** button  on the side toolbar or the **Draw menu > Quick Draw Braces** command to display the **Properties of Object** form. Verify that X is selected in the *Bracing* drop-down list.
13. Click on the points labeled “E,” “F” and “G” in Figure X-1 to enter three sets of diagonal frame objects spanning between the two vertical frames.
14. Click the **Move Down in List** button  to display the X-Y Plane @ Z=0. The screen appears as shown in Figure X-2.

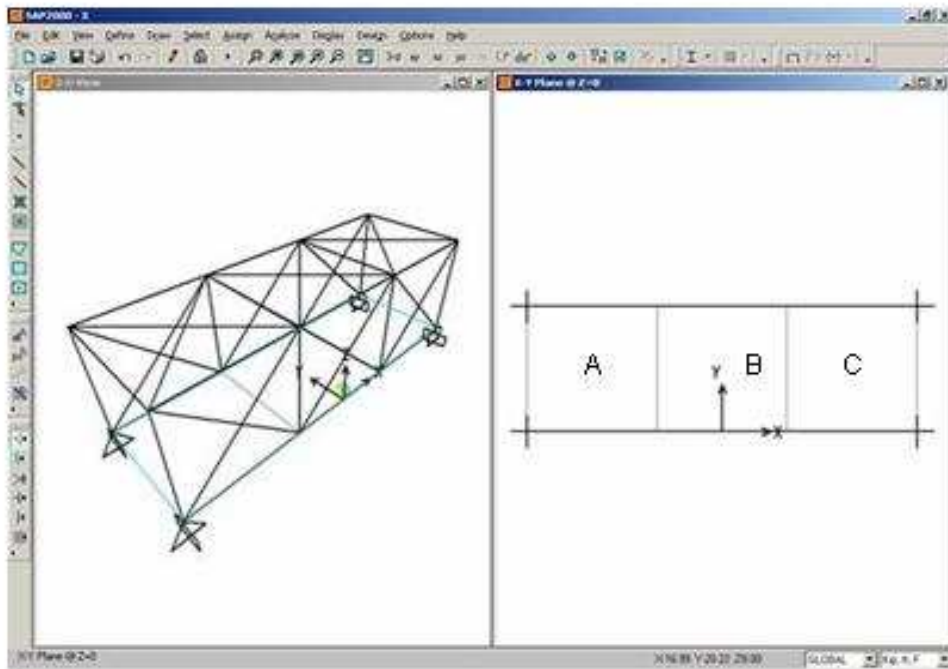









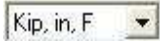
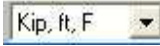







Figure X-2: View of Screen After Step 14

15. Click the **Quick Draw Area** button  on the side toolbar (or the **Draw menu > Quick Draw Area** command) to display the **Properties of Object** form. Verify that **ASEC1** is selected from the *Property* drop-down list.
16. Click on the points labeled “A,” “B” and “C” in Figure X-2 to enter three area objects spanning between the two vertical frames.
17. Click the **Set Select Mode** button  on the side toolbar to exit draw mode and enter select mode.
18. Click in the window entitled X-Y Plane @ Z=0 to make sure it is active.
19. Click the **Move Up in List** button  to display the plan view at Z=12.
20. Select the center four joints by clicking on them.
21. Click the **Edit menu > Move** command to display the **Move** form. In that form:
 - Type **3** in the *Delta Z* edit box.
 - Click the **OK** button.
22. Click the **Perspective Toggle** button . A perspective birds-eye view of the structure is displayed.
23. Click the **Select using Intersecting Line** button  and select all of the roof and floor objects by “drawing” a line through them down the centerline of the bridge. The total

number of lines and areas selected, 10 lines and 3 areas, will be displayed on the left-hand side of the status bar.

24. Click the **Edit menu > Divide Frames** command to display the **Divide Selected Frames** form. In that form:
 - 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.
25. Click on the **XZ View** button  to view an elevation in the X-Z plane. Note the title of the window is X-Z Plane @ Y=0.
26. Click the **Perspective Toggle** button . A perspective view of the structure is displayed.
27. Click the **Select using Intersecting Line** button  and select all of the vertical and diagonal objects by “drawing” a line through them. The total number of lines, 20 lines, will be displayed on the left-hand side of the status bar.
28. Click the **Edit menu > Divide Frames** command to display the **Divide Selected Frames** form. In that form:
 - 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.
29. Click on the **XZ View** button  to view an elevation in the X-Z plane.
30. Click the drop-down box in the status bar to change the units to .
31. Click the **Define menu > Materials** command to display the **Define Materials** form.
32. Click on STEEL in the *Materials* area to highlight it (select it), and then click the **Modify/Show Material** button. The **Material Property Data** form is displayed. In that form:
 - Verify that the *Modulus of Elasticity* is 29000.
 - Verify that *Poisson's Ratio* is 0.3.
 - Verify that the *Minimum Yield Stress, Fy* is 36.
 - Click the **OK** button.
33. Click on CONC in the *Materials* area to highlight (select) it, and then click the **Modify/Show Material** button to display the **Material Property Data** form. In that form:
 - Verify that the *Modulus of Elasticity* is 3600.
 - Verify that *Poisson's Ratio* is 0.2.
 - Click the **OK** buttons on the **Material Property Data** and **Define Materials** forms to exit all forms.
34. Click the drop-down box in the status bar to change the units to .

35. Click the **Define menu > Materials** command to display the **Define Materials** form.
36. Click on CONC in the *Materials* area to highlight (select) it, and then click the **Modify/Show Material** button to display the **Material Property Data** form. In that form:
 - 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.
37. Click the **Define menu > Frame Sections** command to display the **Frame Properties** form. In that form:
 - In the *Choose Property Type to Add* area, click the drop-down box that reads *Import I/Wide Flange* and then click on the *Import I/Wide Flange* item.
 - Click the **Add New Property** button to display the **Section Property File** form. In that 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 wide flange sections in the database. In that form:
 - Scroll down and click on the W6X12 section.
 - Click the **OK** buttons on the **database** form, the **I/Wide Flange Section**, and **Frame Properties** forms to exit all forms.
38. Click the **Define menu > Area Sections** command to display the **Area Sections** form.
 - In the *Click To* area, click the **Modify/Show Section** button to display the **Area Section Data** form. In that form:
 - Verify that the *Material* chosen is CONC.
 - Verify that the *Shell* option is selected in the *Area Type* area.
 - 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** buttons on the **Area Section Data** and **Area Sections** forms to exit all forms.
39. Click the **Select All** button  to select all objects.
40. Click the **Assign menu > Frame/Cable/Tendon > Frame Sections** command to display the **Frame Properties** form. In that form:
 - Click on W6X12 in the *Properties* area to highlight it.
 - Click the **OK** button.
41. Click the **Show Undeformed Shape** button  to remove the displayed frame section assignments.
42. 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 the *Self Weight Multiplier* is 0.
 - Click the **Add New Load** button.
 - Click the **OK** button.
43. Click the **Select All** button  to select all objects.
44. Click the **Assign menu > Area Loads > Uniform (Shell)** command to display the **Area Uniform Loads** form. In that form:
- Select *LIVE* from the *Load Case Name* drop-down box.
 - In the *Uniform Load* area, type **.25** in the *Load* edit box.
 - Verify that the *Gravity* shows in the *Direction* drop-down box.
 - In the *Options* area, verify that the *Replace Existing Loads* option is selected.
 - Click the **OK** button to apply the load.
45. Click the **Show Undeformed Shape** button  to reset the window display.
46. Click the “X” in the upper right-hand corner of the window labeled X-Z Plane @ Y=0 to close it.
47. Click the **Run Analysis** button  to display the **Set Analysis Cases to Run** form. In that form:
- Click on MODAL in the *Case Name* list to highlight it.
 - Click the **Run/Do Not Run Case** button.
 - Click the **Run Now** button.
48. 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 window.
49. 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 list if it is not already selected.
 - Click the **OK** button.
50. Click the **Design menu > Steel Frame Design > Start Design/Check of Structure** command to run the design check of the steel frame objects.
51. When the design check completes, the stress ratios are displayed.