

Problem N

Frame-Shear Wall Interaction

Concrete Material Properties

E=3600 ksi, Poissons Ratio = 0.2

Frame

Beams: 12" wide by 24" deep

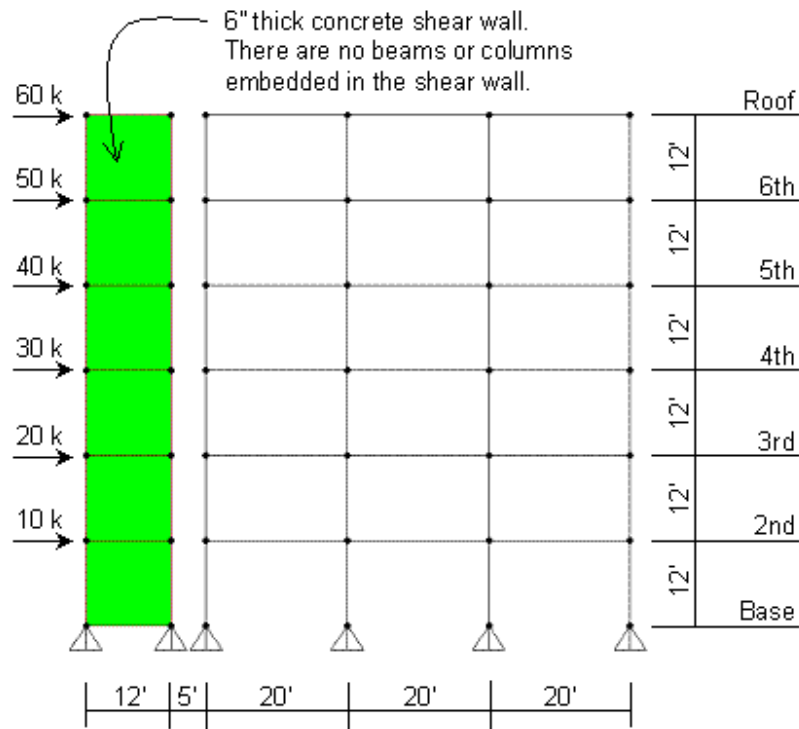
Columns: 24" by 24", pinned base

Diaphragm

Provide rigid diaphragm constraint at each level.

To Do

Determine shear carried by wall and by frame at 2nd level and at 6th level.

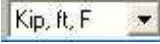


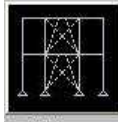
CSI Solution Demonstrates Use of These Features

- Diaphragm Constraint
- Groups
- Section Cuts

Problem N 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 Frame** button  to display the **2D Frames** form. In that form:

- Select *Portal* in the *2D Frame Type* drop-down list.
- Type **6** in the *Number of Stories* edit box.
- Type **3** in the *Number of Bays* edit box.
- Accept the default *Story Height* of 12.
- Type **20** in the *Bay Width* edit box.
- Click the **OK** button.

4. Click the “X” in the top right-hand corner of the 3-D View window to close it.

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

- Click on the **Modify/Show System** button to display the **Define Grid** form. In that form:
 - In the *X Grid Data* area, type **x5** in the *Grid ID* cell of row 5 of the spreadsheet. In that same row, type **-35** in the *Ordinates* cell and click in each of the other cells in the row (Line Type, Visibility, Bubble Loc.) to display *Primary, Show* and *Top*.
 - In the *X Grid Data* area, type **x6** in the *Grid ID* cell of row 6 of the spreadsheet. In that same row, type **-47** in the *Ordinates* cell and click in each of the other cells in the row (Line Type, Visibility, Bubble Loc.) to display *Primary, Show* and *Top*.
 - Click the **Reorder Ordinates** button to reorder the ordinates from lowest to highest.
 - Click the **OK** buttons on the **Define Grid** and **Coordinate/Grid Systems** forms to close all forms. The screen appears as shown in Figure N-1.



6. 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 shown in the *Property* drop-down list.

7. Click in the area labeled “A” in Figure N-1 to quick draw the first area object. Note that a quick area object is drawn by clicking in a grid space, bounded by four grid lines.

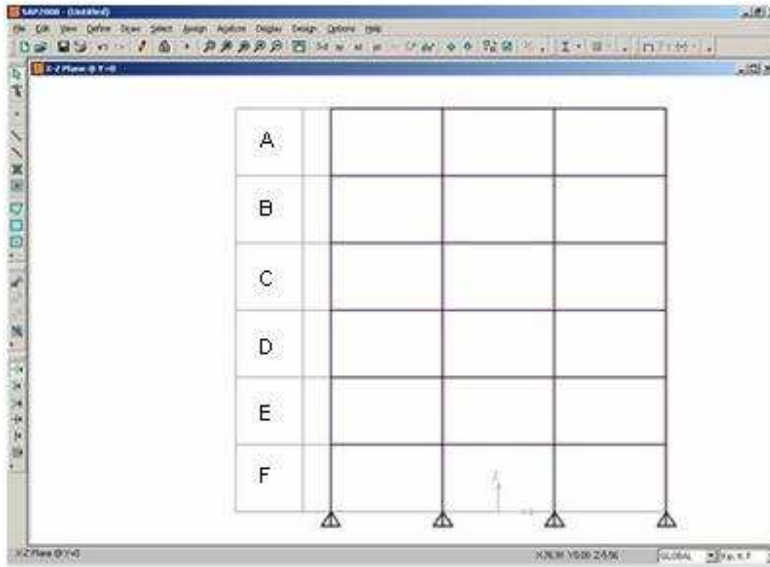




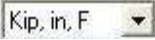



Figure N-1 Screen As It Appears After Step 7



8. Click in the areas labeled “B,” “C,” “D,” “E” and “F,” in that order, in Figure N-1 to draw the other area objects.
9. Click the **Set Select Mode** button  on the side toolbar to exit the Draw mode and enter the Select mode.
10. 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 *Fill Objects* box in the *General* area.
 - Click the **OK** button.


Note: To increase the font size, click the **Options menu > Preferences > Dimensions/Tolerances** command to display the **Dimensions/Tolerances Preferences** form. Type in a larger font size in the *Minimum Graphic Font Size* edit box (usually 6 points is sufficient), and click the **OK** button.

11. Select joints 41 and 42 at the base of the shear wall.
12. Click the **Assign menu > Joint > Restraints** command to display the **Joint Restraints** form. In that form:
 - Click the pinned base **fast restraint** button  to set all translational degrees of freedom (U1, U2 and U3) as restrained.
 - Click the **OK** button.
13. Click the **Show Undeformed Shape** button  to remove the display of joint restraints and reset the window display (title).

14. Click the drop-down box in the status bar to change the units to .
15. Click the **Define menu > Materials** command to display the **Define Materials** form.
16. Click on CONC 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 set to 3600.
 - Verify that *Poisson's Ratio* is set to 0.2.
 - Click the **OK** buttons on the **Material Property Data** and **Define Materials** forms to close all forms.
17. 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 *Add I/Wide Flange* and select the *Add Rectangular* item. Click on the **Add New Property** button to display the **Rectangular Section** form. In that form:
 - Type **BEAM** in the *Section Name* edit box.
 - Select CONC from the *Material* drop-down box.
 - Type **24** in the *Depth (t3)* edit box.
 - Type **12** in the *Width (t2)* edit box.
 - Click the **OK** button to return to the **Frame Properties** form.
 - In the *Choose Property Type to Add* area, click the *Add Rectangular* item. Click on the **Add New Property** button to display the **Rectangular Section** form. In that form:
 - Type **COL** in the *Section Name* edit box.
 - Select CONC from the *Material* drop-down box.
 - Type **24** in the *Depth (t3)* edit box.
 - Type **24** in the *Width (t2)* edit box.
 - Click the **OK** buttons on the **Rectangular Section** and **Frame Properties** forms to close all forms.
18. 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 that the selected *Material Name* is CONC.
 - Verify that the *Shell* option is selected in the *Area Type* area.
 - In the *Thickness* area type **6** in both the *Membrane* and the *Bending* edit boxes.
 - 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 close all forms.

19. Select all of the column objects by “windowing” each of the four column lines separately.
20. Click the **Assign menu > Frame/Cable/Tendon > Frame Sections** command to display the **Frame Properties** form. In that form:
 - Click on COL in the *Properties* list box to highlight it.
 - Click the **OK** button.
21. Select all of the beam objects by using the intersecting line method on each of the three beam bays separately.
22. Click the **Assign menu > Frame/Cable/Tendon > Frame Sections** command to display the **Frame Properties** form. In that form:
 - Click on BEAM in the *Properties* list box to highlight it.
 - Click the **OK** button.
23. Select all of the joints by “windowing” or by clicking the **Select All** button  on the side toolbar.
24. 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 **DIAPH** in the *Constraint Name* edit box.
 - Select the *Z Axis* option in the *Constraint Axis* area.
 - Check the *Assign a different diaphragm constraint to each different selected Z level* check box. This will assign a diaphragm constraint to each floor level with the label *DIAPH_el*, where *el* is the elevation of the floor in the units when assigned.
 - Click the **OK** buttons on the **Diaphragm Constraint** and **Assign/Define Constraints** forms to close all forms.
25. Select joints 39, 37, 35, 33, 29, and 32 (left-hand side of shear wall from 2nd level to roof level) by “windowing.”
26. Click the **Assign menu > Joint Loads > Forces** command to display the **Joint Forces** form. In that form:
 - Type **10** in the *Force Global X* edit box in the *Loads* area.
 - In the *Options* area, select the *Add to Existing Loads* option.
 - Click the **OK** button.
27. Select joints 37, 35, 33, 29, and 32 (left-hand side of shear wall from 3rd level to roof level) by “windowing.”
28. Click the **Assign menu > Joint Loads > Forces** command to display the **Joint Forces** form. In that form:
 - In the *Options* area, select the *Add to Existing Loads* option.
 - Click the **OK** button.

29. Select joints 35, 33, 29, and 32 (left-hand side of shear wall from 4th level to roof level) by “windowing.”
30. Click the **Assign menu > Joint Loads > Forces** command to display the **Joint Forces** form. In that form:
 - In the *Options* area, select the *Add to Existing Loads* option.
 - Click the **OK** button.
31. Select joints 33, 29, and 32 (left-hand side of shear wall from 5th level to roof level) by “windowing.”
32. Click the **Assign menu > Joint Loads > Forces** command to display the **Joint Forces** form. In that form:
 - In the *Options* area, select the *Add to Existing Loads* option.
 - Click the **OK** button.
33. Select joints 29 and 32 (left-hand side of shear wall from 6th level to roof level) by “windowing.”
34. Click the **Assign > Joint Loads > Forces** command to display the **Joint Forces** form. In that form:
 - In the *Options* area, select the *Add to Existing Loads* option.
 - Click the **OK** button.
35. Select joint 32 (left-hand side of shear wall at roof level).
36. Click the **Assign menu > Joint Loads > Forces** command to display the **Joint Forces** form. In that form:
 - In the *Options* area, select the *Add to Existing Loads* option.
 - Click the **OK** button.
37. Click the **Show Undeformed Shape** button  to remove the display of joint forces.
38. Click the **Set Display Options** button  on the main toolbar (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 *Frames/Cables/Tendons* area.
 - Check the *Labels* box in the *Areas* area.
 - Click the **OK** button.
39. Select joints 29 and 30 and select area object 1.
40. Click the **Assign menu > Assign to Group** command to display the **Assign/Define Group Names** form. In that form:
 - Click the **Add New Group** button to display the **Group Definition** form. In that form:
 - Type **6THWALL** in the *Group Name* edit box.
 - Click the **OK** buttons on the **Group Definition** and **Assign/Define Group Names** form to close all forms.

41. Select joints 6, 13, 20 and 27 by clicking on each joint individually and select frame (column) objects 6, 12, 18 and 24 using the intersecting line method.
42. Click the **Assign menu > Assign to Group** command to display the **Assign/Define Group Names** form. In that form:
 - Click the **Add New Group** button to display the **Group Definition** form. In that form:
 - Type **6THFRAME** in the *Group Name* edit box.
 - Click the **OK** buttons on the **Group Definition** and **Assign/Define Group Names** forms to close all forms.
43. Select joints 39 and 40 and select area object 5.
44. Click the **Assign menu > Assign to Group** command to display the **Assign/Define Group Names** form. In that form:
 - Click the **Add New Group** button to display the **Group Definition** form. In that form:
 - Type **2NDWALL** in the *Group Name* edit box.
 - Click the **OK** buttons on the **Group Definition** and **Assign/Define Group Names** forms to close all forms.
45. Select joints 2, 9, 16 and 23 by clicking on each joint individually and select frame (column) objects 2, 8, 14 and 20 using the intersecting line method.
46. Click the **Assign menu > Assign to Group** command to display the **Assign/Define Group Names** form. In that form:
 - Click the **Add New Group** button to display the **Group Definition** form. In that form:
 - Type **2NDFRAME** in the *Groups* edit box.
 - Click the **OK** buttons on the **Group Definition** and **Assign/Define Group Names** forms to close all forms.
47. 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/Tendons* area.
 - Uncheck the *Labels* box in the *Areas* area.
 - Click the **OK** button.
48. Click the **Define menu > Section Cuts** command to display the **Section Cuts** form. In that form:
 - Click the **Add Section Cut** button to display the **Section Cut Data** form. In that form:
 - Type **2NDFRAME** in the *Section Cut Name* edit box.
 - Select **2NDFRAME** from the *Group* drop-down list.
 - Click the **OK** button to return to the **Section Cuts** form.
49. On the **Section Cuts** form:

- Click the **Add Section Cut** button to display the **Section Cut Data** form. In that form:
 - Type **2NDWALL** in the *Section Cut Name* edit box.
 - Select **2NDWALL** from the *Group* drop-down list.
 - Click the **OK** button to return to the **Section Cuts** form.

50. On the **Section Cuts** form:

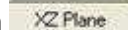
- Click the **Add Section Cut** button to display the **Section Cut Data** form. In that form:
 - Type **6THFRAME** in the *Section Cut Name* edit box.
 - Select **6THFRAME** from the *Group* drop-down list.
 - Click the **OK** button to return to the **Section Cuts** form.


51. On the **Section Cuts** form:

- Click the **Add Section Cut** button to display the **Section Cut Data** form. In that form:
 - Type **6THWALL** in the *Section Cut Name* edit list.
 - Select **6THWALL** from the *Group* drop-down box.
 - Click the **OK** buttons on the **Section Cut Data** and **Section Cuts** form to exit all forms.

52. Click the **Analyze menu > Set Analysis Options** command to display the **Analysis Options** form. In that form:



- Click the **Plane Frame XZ Plane** button  to set the available degrees of freedom.
- Click the **OK** button.

53. Click the **Run Analysis** button  to display the **Set Analysis Cases to Run** form. In that form:

- Highlight (select) **MODAL** in the *Case Name* list and click the **Run/Do Not Run Case** button.
- Verify that the **DEAD** analysis case is set to *Run* in the *Action* list.
- Click the **Run Now** button to run the analysis.

54. 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.

55. Click the **Display menu > Show Tables** command to display the **Choose Tables to Display** form. In that form:

- In the *Analysis Results* area of the form, click on the + (plus) associated with the *Structure Output* item.

- Click the + (plus) associated with *Other Output Items*.
- Click on the *Table: Section Cut Forces* item.
- Click the **OK** button to display the **Section Cut Forces** table.

Note: Notice the direction of the shear (F1 force) in the shear wall at the 6th level.

Note: To print the section cut forces table, click the **File menu > Print Current Table as Text File** command on the **Section Cut Forces** table.

56. When finished viewing the table, click the **Done** button to close the form.