Problem Q

Isolated Building - Nonlinear Time History Analysis

Steel

E =29000 ksi, Poissons Ratio = 0.3
Beams: W24X55; Columns: W14X90

Rubber Isolator Properties

Vertical (axial) stiffness = 10,000 k/in (linear)
Initial shear stiffness in each direction = 10 k/in
Shear yield force in each direction = 5 kips
Ratio of post yield shear stiffness to initial shear stiffness = 0.2

Vertical Loading and Mass

Roof: 75 psf DL
20 psf LL

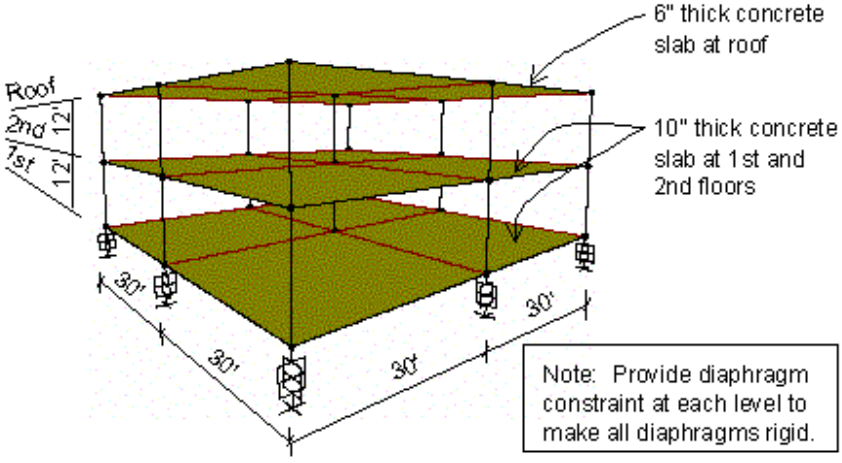
Floor: 125 psf DL
100 psf LL

Time History

Apply lacc_nor-1 in the X-direction and lacc_nor-2 in the Y-direction simultaneously. Each time history is given in units of cm/sec^2. There are 3,000 time steps, at an equal spacing of 0.02 sec, for a total of 60 sec. There are 8 accelerations points per line.

To Do

Plot time histories of Y-direction displacement at the 1st level and at the roof level. Plot a time history of the 1st level Y-direction displacement versus the Y-direction base shear.

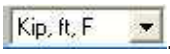


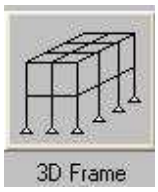
CSI Solution Demonstrates Use of These Features

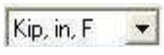
- Base (Seismic) Isolation
- Diaphragm Constraint
- Ritz Vectors
- Dynamic Analysis
- Mode Shapes
- Link Elements

- Modal Nonlinear Time History Analysis

Problem O 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 

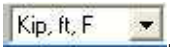
..
3. Click the **3D Frame** button 

to display the ***3D Frames*** form. In that form:
    - Select *Open Frame Building* from the *3D Frame Type* drop-down list.
    - Type **2** in the *Number of Bays, X* edit box.
    - Type **30** in the *Bay Width, X* edit box.
    - Type **30** in the *Bay Width, Y* edit box.
    - Uncheck the *Restraints* check box.
    - Accept the rest of the default values.
    - Click the **OK** button.
4. Click in the window labeled X-Y Plane @ Z=0 to make sure it is active. The window is active when its title is highlighted.
5. Click the **Move Up in List** button  until the plan display is moved up to the X-Y Plane @ Z=24.
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. We can ignore the property setting shown in this form because other assignments will be made later in the modeling process.
7. Click once in each of the four quadrants in the plan view to draw four area objects.
8. Click the **Move Down in List** button  to move the plan display down to the X-Y Plane @ Z=12.
9. Click once in each of the four quadrants in the plan view to draw four area objects.
10. Click the **Set Select Mode** button  to exit Draw mode and enter Select mode.
11. Click the **Move Down in List** button  to move the plan display down to the X-Y Plane @ Z=0.
12. Click the drop-down box in the status bar to change the units to 

.
13. Click the **Define menu > Link/Support Properties** command to display the ***Link/Support Properties*** form. In that form:

- Click the **Add New Property** button to display the ***Link/Support Property Data*** form. In that form:
    - Select *Rubber Isolator* from the *Link/Support Type* drop-down list.
    - Type **RUB1** in the *Property Name* edit box.
    - Type **.001** in the *Mass* edit box.
    - Check the *U1 Direction* check box in the *Directional Properties* area.
    - Click the **Modify/Show For U1** button to display the ***Link/Support Directional Properties*** form. In that form:
        - Type **10000** in the *Effective Stiffness* edit box.
        - Click the **OK** button to return to the ***Link/Support Property Data*** form.
    - Check the *U2 Direction* check box.
    - Check the *U2 Nonlinear* check box.
    - Click the **Modify/Show For U2** button to display the ***Link/Support Directional Properties*** form. In that form:
        - In the *Properties Used for Linear Analysis Cases* area, type **1.5** in the *Effective Stiffness* edit box.
        - In the *Properties Used for Nonlinear Analysis Cases* area, type **10** in the *Stiffness* edit box.
        - Type **5** in the *Yield Strength* edit box.
        - Type **.2** in the *Post Yield Stiffness Ratio* edit box.
        - Accept the other values on the form.
        - Click the **OK** button to return to the ***Link/Support Property Data*** form.
    - Check the *U3 Direction* check box.
    - Check the *U3 Nonlinear* check box.
    - Click the **Modify/Show For U3** button to display the ***Link/Support Directional Properties*** form. In that form:
        - In the *Properties Used for Linear Analysis Cases* area, type **1.5** in the *Effective Stiffness* edit box.
        - In the *Properties Used for Nonlinear Analysis Cases* area, type **10** in the *Stiffness* edit box.
        - Type **5** in the *Yield Strength* edit box.
        - Type **.2** in the *Post Yield Stiffness Ratio* edit box.
        - Accept the other values on the form.
        - Click the **OK** buttons on the ***Link/Support Directional Properties***, ***Link/Support Property Data***, and ***Link/Support Properties*** forms to exit all forms.

14. Click in the window labeled X-Y Plane @ Z=0 to make sure it is active.

15. Click the **Draw menu > Draw 1 Joint Link** command to display the ***Properties of Object*** form. Select *RUB1* from the *Property* drop-down box.
16. In the plan view of the X-Y Plane @ Z=0, click on each of the nine grid intersection points to draw nine Links.
17. Click the **Set Select Mode** button  to exit Draw mode and enter Select mode.
18. Click the **Define menu > Materials** command to display the ***Define Materials*** 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 and *Poisson's ratio* is 0.3.
    - Click the **OK** buttons on the ***Material Property Data*** and ***Define Materials*** forms to close the forms.
19. Click the **Define menu > Area Sections** command to display the ***Area Sections*** form. In that form:
    - Click the **Add New Section** button to display the ***Area Section Data*** form. In that form:
        - Type **ROOF** in the *Section Name* edit box.
        - Accept the default CONC material.
        - In the *Area Type* area, verify that the *Shell* option is selected.
        - Type **6** in the *Membrane* edit box.
        - Type **6** in the *Bending* edit box.
        - In the *Type* area verify that the *Shell* option is selected.
        - Click the **OK** button to return to the ***Area Sections*** form.
    - Click the **Add New Section** button to display the ***Area Section Data*** form. In that form:
        - Type **FLOOR** in the *Section Name* edit box.
        - Accept the default CONC material.
        - In the *Area Type* area, verify that the *Shell* option is selected.
        - Type **10** in the *Membrane* edit box.
        - Type **10** in the *Bending* edit box.
        - In the Type area verify that the *Shell* option is selected.
        - Click the **OK** buttons on the ***Area Section Data*** and ***Area Sections*** forms to close all forms.
20. Click the drop-down box in the status bar to change the units to 

.
21. Click the **Define menu > Materials** command to display the ***Define Materials*** 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 and that the *Weight per unit Volume* is 0.15.

- Click the **OK** buttons on the ***Material Property Data*** and ***Define Materials*** forms to close the forms.

22. Click in the window labeled X-Y Plane @ Z=0 to make sure it is active.
23. Click the **Set XZ View** button  .
24. Click the **Perspective Toggle** button  .
25. Click the **Select using Intersecting Line** button  and select all of the bottom level columns.

Note: To use the Intersecting Line Selection option, click the **Select using Intersecting Line** button  . Then click the left mouse button to the left of the first level columns, and while holding down the left mouse button, drag the mouse to the right of the first level columns. A "rubberband line" will appear and all objects that this "rubberband line" passes through will be selected. Release the left mouse button to make the selection.

26. Click the **Select using Intersecting Line** button  and select all of the top level columns.
27. Click the **Assign menu > Frame/Cable/Tendon > Frame Sections** command to display the ***Frame Properties*** form. In that form:
    - Click on *W14X90* in the *Properties* area to highlight it.
    - Click the **OK** button.
28. Click the **Show Undeformed Shape** button  to remove the displayed line object assignments.
29. Click the **Set XY View** button  . The plan view of the X-Y Plane @ Z=0 appears.
30. Click the **Move Up in List** button  to move the plan display up to the X-Y Plane @ Z=12.
31. Select all of the objects at this level by "windowing."
32. Click the **Move Up in List** button  to move the plan display up to the X-Y Plane @ Z=24.
33. Select all of the objects at this level by "windowing."
34. Click the **Assign menu > Frame/Cable/Tendon > Frame Sections** command to display the ***Frame Properties*** form. In that form:
    - Click on *W24X55* in the *Properties* area to highlight it.
    - Click the **OK** button.
35. Click the **Show Undeformed Shape** button  to remove the displayed line object assignments.
36. Select all of the objects at the Z=24 level by "windowing."

37. Click the **Assign menu > Area > Sections** command to display the ***Area Sections*** form. In that form:
    - Click on *ROOF* in the *Sections* area to highlight it.
    - Click the **OK** button.
38. Click the **Show Undeformed Shape** button  to remove the displayed area object assignments.
39. Click the **Move Down in List** button  to move the plan display down to the X-Y Plane @ Z=12.
40. Select all of the objects at this level by “windowing.”
41. Click the **Assign menu > Area > Sections** command to display the ***Area Sections*** form. In that form:
    - Click on *FLOOR* in the *Sections* area to highlight it.
    - Click the **OK** button.
42. Click the **Show Undeformed Shape** button  to remove the displayed area object assignments.
43. 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.
    - Click the **Add New Load** button.
    - Click the **OK** button.
44. Click the drop-down box in the status bar to change the units to 

.
45. Select all of the objects at the Z=12 level by “windowing.”
46. Click the **Assign menu > Area Loads > Uniform (Shell)** command to display the ***Area Uniform Loads*** form. In that form:
    - Verify that *DEAD* is selected from the *Load Case Name* drop-down box.
    - Type **125** in the *Load* edit box.
    - Verify that *Gravity* is selected in the *Direction* drop-down list.
    - Click the **OK** button.
47. Select all of the objects at the Z=12 level by “windowing.”
48. 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 list.
    - Type **100** in the *Load* edit box.
    - Click the **OK** button.

49. Click the **Show Undeformed Shape** button  to remove the displayed load assignments.

50. Click the **Move Up in List** button  to move the plan display up to the X-Y Plane @ Z=24.

51. Select all of the objects at the Z=24 level by "windowing."

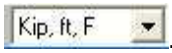
52. Click the **Assign menu > Area Loads > Uniform (Shell)** command to display the ***Area Uniform Loads*** form. In that form:

    - Select *DEAD* from the *Load Case Name* drop-down list.
    - Type **75** in the Load edit box.
    - Click the **OK** button.

53. Select all of the objects at the Z=24 level by "windowing."

54. 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 list.
    - Type **20** in the *Load* edit box.
    - Click the **OK** button.

55. Click the drop-down box in the status bar to change the units to 

.

56. Click the **Show Undeformed Shape** button  to remove the displayed load assignments.

57. Click the **Move Down in List** button  to move the plan display down to the X-Y Plane @ Z=12.

58. Select all of the objects at the Z=12 level by "windowing."

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

    - Select the *Linear Tab.*
    - Type **-12** in the *dz* edit box in the *Increments* area.
    - Verify that the *Number* is 1.
    - Click the **OK** button.

60. Click in the window labeled X-Y Plane @ Z=12 to make sure it is active.

61. Click the **Move Up in List** button  to move the plan display up to the X-Y Plane @ Z=24.

62. Select all objects at the Z=24 level by "windowing."

63. Click the **Assign menu > Joint > Constraints** command to display the ***Assign/Define Constraints*** form. In that form:

    - In the *Choose Constraint Type to Add* area, click the drop-down box that reads *Body* and then click *Diaphragm*. Click the **Add New Constraint** button to display the ***Diaphragm Constraint*** form. In that form:

- Type **ROOF** in the *Constraint Name* edit box.
- Select the *Z axis* option in the *Constraint Axis* area if it is not already selected.
- Click the **OK** buttons on the **Diaphragm Constraint** and **Assign/Define Constraints** forms to assign the diaphragm constraint.

64. Click the **Show Undeformed Shape** button  to remove the displayed constraint assignments.
65. Click the **Move Down in List** button  to move the plan display down to the X-Y Plane @ Z=12.
66. Select all objects at the Z=12 level by "windowing."
67. Click the **Assign menu > Joint > Constraints** command to display the ***Assign/Define Constraints*** form. In that form:
    - In the *Choose Constraint Type to Add* area, click the drop-down box that reads *Body* and then click *Diaphragm*. Click the **Add New Constraint** button to display the ***Diaphragm Constraint*** form. In that form:
        - Type **2ND** in the *Constraint Name* edit box.
        - Select the *Z axis* option in the *Constraint Axis* area if it is not already selected.
        - Click the **OK** buttons on the ***Diaphragm Constraint*** and ***Assign/Define Constraints*** forms to assign the diaphragm constraint.
68. Click the **Show Undeformed Shape** button  to remove the displayed constraint assignments.
69. Click the **Move Down in List** button  to move the plan display down to the X-Y Plane @ Z=0.
70. Select all objects at the Z=0 level by "windowing."
71. Click the **Assign menu > Joint > Constraints** command to display the ***Assign/Define Constraints*** form. In that form:
    - In the *Choose Constraint Type to Add* area, click the drop-down box that reads *Body* and then click *Diaphragm*. Click the **Add New Constraint** button to display the ***Diaphragm Constraint*** form. In that form:
        - Type **1ST** in the *Constraint Name* edit box.
        - Select the *Z axis* option in the *Constraint Axis* area if it is not already selected.
        - Click the **OK** buttons on the ***Diaphragm Constraint*** and ***Assign/Define Constraints*** forms to assign the diaphragm constraint.
72. Click the **Show Undeformed Shape** button  to remove the displayed diaphragm constraint assignments.

Note: Before defining time history functions, locate the time history files that you wish to use. For this problem, we are using files named lacc_nor-1.th and Lacc_nor-2.th, but any time history files may be used. A number of sample files are included with SAP2000.

73. Click the **Define menu > Functions > Time History** command to display the ***Define Time History Functions*** form. In that form:
    - In the *Choose Function Type to Add* area, click the drop-down list that reads *Sine Function* and then click *Function from File*.
    - Click the **Add New Function** button to display the ***Time History Function Definition*** form. In that form:
        - Type **LACC0** in the *Function Name* edit box.
        - Click the **Browse** button in the *Function File* area to display the ***Pick Function Data File*** form. In that form:
            - Locate and highlight the first time history file that you wish to use (*we are using lacc_nor-1.th*).
            - Click the **Open** button to return to the ***Time History Function Definition*** form.
        - Type **2** in the *Header Lines to Skip* edit box.
        - Type **8** in the *Number of Points Per Line* edit box.
        - Select the *Values At Equal Intervals of* option and type **.02** in the edit box.
        - Click the **OK** button to return to the ***Define Time History Functions*** form.
    - Click the **Add New Function** button to display the ***Time History Function Definition*** form. In that form:
        - Type **LACC90** in the *Function Name* edit box.
        - Click the **Browse** button in the *Function File* area to display the ***Pick Function Data File*** form. In that form:
            - Locate and highlight the second time history file that you wish to use (*we are using lacc_nor-2.th*).
            - Click the **Open** button to return to the ***Time History Function Definition*** form.
        - Type **2** in the *Header Lines to Skip* edit box.
        - Type **8** in the *Number of Points Per Line* edit box.
        - Select the *Values At Equal Intervals of* option and type **.02** in the edit box.
        - Click the **OK** buttons on the ***Time History Function Definition*** and ***Define Time History Functions*** form to close all forms.
74. 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 - Modal*** form. In that form:
        - Type **30** in the *Maximum Number of Modes* edit box.
        - In the *Type of Modes* area select the *Ritz Vectors* option.

- In the *Loads Applied* area, verify that *Load* shows in the *Load Type* drop-down list and that *DEAD* shows in the *Load Name* drop-down list. Click the **Add** button.
- In the *Loads Applied* area, select *Accel* from the *Load Type* drop-down list and *UX* from the *Load Name* drop-down list. Click the **Add** button.
- In the *Loads Applied* area, select *UY* from the *Load Name* drop-down list. Click the **Add** button.
- In the *Loads Applied* area, select *Link* from the *Load Type* drop-down list. Click the **Add** button.
- Click the **OK** button to return to the ***Analysis Cases*** form.

- Click the **Add New Case** button to display the ***Analysis Case Data*** form. In that form:
  - Type **GRAV** in the *Analysis Case Name* edit box.
  - Select *Time History* from the *Analysis Case Type* drop-down list.
  - Select the *Nonlinear* option in the *Analysis Type* area.
  - In the *Loads Applied* area, verify that *Load* shows in the *Load Type* drop-down box and that *DEAD* shows in the *Load Name* drop-down box. Select *RAMPTH* from the *Function* drop-down box. Click the **Add** button.
  - Type **100** in the *Number of Output Time Steps* edit box.
  - Type **.1** in the *Output Time Step Size* edit box.
  - Click the **OK** button to return to the ***Analysis Cases*** form.
- Click the **Add New Case** button to display the ***Analysis Case Data*** form. In that form:
  - Type **LAC** in the *Analysis Case Name* edit box.
  - Select *Time History* from the *Analysis Case Type* drop-down list.
  - Select the *Nonlinear* option in the *Analysis Type* area.
  - In the *Initial Conditions* area, select the *Continue from State at End of Modal History* option.
  - In the *Loads Applied* area, select *Accel* from the *Load Type* drop-down box and *U1* from the *Load Name* drop-down box. Select *LACC0* from the *Function* drop-down box, and type **0.0328** in the *Scale Factor* edit box. Click the **Add** button.
  - In the *Loads Applied* area, select *U2* from the *Load Name* drop-down box and select *LACC90* from the *Function* drop-down box. Click the **Add** button.
  - Type 3**000** in the *Number of Output Time Steps* edit box.
  - Type **.02** in the *Output Time Step Size* edit box.
  - In the *Other Parameters* area of the form, click the **Modify/Show** button for Modal Damping to display the ***Modal Damping*** form. In that form:
    - Verify that **.05** shows in the *Constant Damping For All Modes* edit box.

- In the *Modal Damping Overrides* area type **1** in the *Mode* box, type **0.02** in the *Damping* box and click the **Add** button.
- In the *Modal Damping Overrides* area type **2** in the *Mode* box and click the **Add** button.
- In the *Modal Damping Overrides* area type **3** in the *Mode* box and click the **Add** button.
- Click the **OK** buttons on the ***Model Damping***, ***Analysis Case Data***, and ***Analysis Cases*** forms to close all forms.

75. Click the **Run Analysis** button to display the ***Set Analysis Cases to Run*** form. In that form:
    - Verify that all analysis cases are set to *Run* in the *Action* list.
    - Click the **Run Now** button to run the analysis.
76. When the analysis is complete, check the messages in the ***SAP Analysis Monitor*** window and then click the **OK** button to close the window.
77. Click in the window labeled X-Y Plane @ Z=0 to make sure it is active.
78. 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.
    - Click the **OK** button.
79. Click on the center joint, joint 13, in the plan at Z=0 to select it.
80. Click the **Move Up in List** button  twice to move the plan display up to the X-Y Plane @ Z=24.
81. Click on the center joint, joint 15, in the plan at Z=24 to select it.
82. Click the **Set Display Options** button  (or the **View menu > Set Display Options** command) to display the ***Display Options on Active Window*** form. In that form:
    - Uncheck the *Labels* box in the *Joints* area.
    - Click the **OK** button.
83. Click the **Display menu > Show Plot Functions** command to display the ***Plot Function Trace Display Definition*** form. In that form:
    - Select *LAC* from the *Analysis Case* drop-down box.
    - Click the **Define Plot Functions** button in the *Choose Plot Functions* area to display the ***Plot Functions*** form. In that form:
        - Highlight Joint 13.
        - Click the **Modify/Show Plot Function** button to display the ***Joint Plot Function*** form. In that form:
            - Verify that the *Displ* option is selected in the *Vector Type* area.
            - Select the *UY* option in the *Component* area.

        - Click the **OK** button to return to the ***Plot Functions*** form.
    - Highlight *Joint 15*.
    - Click the **Modify/Show Plot Function** button to display the ***Joint Plot Function*** form. In that form:
        - Verify that the *Displ* option is selected in the *Vector Type* area.
        - Select the UY option in the Component area.
        - Click the **OK** button to return to the ***Plot Functions*** form.
    - In the *Choose Function Type to Add* area select *Add Base Functions* from the drop-down box and click *Add Plot Function* to display the ***Base Functions*** form. In that form:
        - Check the *Base Shear Y* check box.
        - Click the **OK** buttons on the ***Base Functions*** and ***Plot Functions*** forms to return to the ***Plot Function Trace Display Definition*** form.
    - Click on *Joint 13* in the *List of Functions* to highlight it (select it).
    - Hold down the Ctrl key on the keyboard and click on *Joint 15* to add it to the selection.
    - Click the **Add** button to move Joints 13 and 15 into the *Vertical Functions* list.
    - Click the **Display** button to display the displacement time histories. Note that there is very little difference between the 1st and roof level displacements. The structure is essentially moving as a rigid body on top of the isolators.
    - Click the **OK** button to close the ***Display Plot Function Traces*** form and return to the ***Plot Function Trace Display Definition*** form.
- Click on *Joint 15* in the *Vertical Functions* list to highlight it.
- Hold down the Ctrl key on the keyboard and click on *Joint 13* to add it to the selection.
- Click the **Remove** button to move Joints 15 and 13 back into the *List of Functions* list.
- Click on *Base Shear Y* in the *List of Functions* to highlight it.
- Click the **Add** button to move Base Shear Y into the *Vertical Function*s list.
- In the *Horizontal Plot Function* drop-down box, select *Joint 13*.
- Click the **Display** button to display the force-displacement plot.
- Click the **OK** button to close the ***Display Plot Function Traces*** form and return to the ***Plot Function Trace Display Definition*** form.
- Click the **Done** button to close the ***Plot Function Trace Display Definition*** form.