Problem O

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

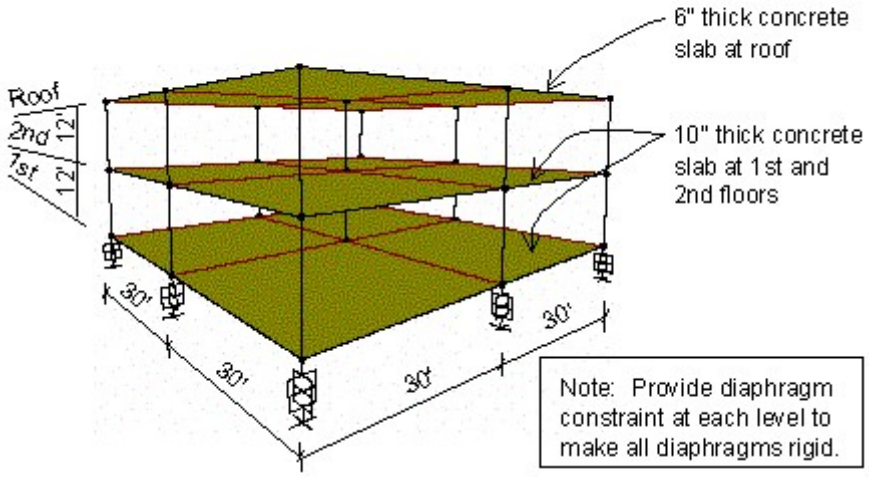
Roof: 75 psf DL
20 psf LL
Floor: 125 psf DL
100 psf LL

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

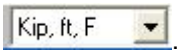
Plot time histories of Y-direction displacement at the 1st level and at the roof level. Plot a time history of the 1st level Y-direction displacement versus the Y-direction base shear.

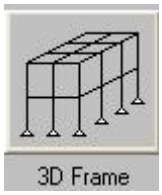


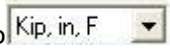
CSI Solution Demonstrates Use of These Features

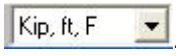
- Base (Seismic) Isolation
- Diaphragm Constraint
- Ritz Vectors
- Dynamic Analysis
- Mode Shapes
- Link Elements
- Modal Nonlinear Time History Analysis

Problem O Solution

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

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

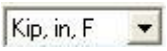
to access the ***3D Frames*** form. In that form:
   - Select *Open Frame Building* from the *3D Frame Type* drop-down list.
   - Type **2** in the *Number of Bays, X* edit box.
   - Type **30** in the *Bay Width, X* edit box.
   - Type **30** in the *Bay Width, Y* edit box.
   - Uncheck the *Restraints* check box.
   - Click the + (plus) symbol beside the drop-down list to access the ***Frame Properties*** form. In that form:
     - Click the **Import New Property** button to access the ***Import Frame Section Property*** form.
     - In the *Frame Section Property Type* drop-down list, select *Steel*.
     - Click the **I/Wide Flange** button to access the ***Section Property File*** form. Select the SECTIONS.PRO file and click the **Open** button. A ***database*** form of available sections will display. In that form:
       - Click the + (plus) symbol beside the *Material* drop-down list to access the ***Define Materials*** form. In that form:
       - Highlight the **A992Fy50** definition in the *Materials* display list and click the **Modify/Show Material** button to access the ***Material Property Data*** form. In that form:
         - Set the Units to 

.
         - Verify that the *Modulus of Elasticity is 29000* and the *Poisson's Ratio* is 0.3.
         - Click the **OK** button on the ***Material Property Data*** form to return to the ***Define Materials*** form.
       - Highlight the **4000Psi** definition in the *Materials* display list and click the **Modify/Show Material** button to access the ***Material Property Data*** form. In that form:
         - Set the Units to 

.
         - Verify that the *Weight per Unit Volume* is 0.15.
         - Click the **OK** buttons on the ***Material Property Data*** form and the ***Define Materials*** form to return to the ***database*** form.
       - In the ***database*** form, ensure that the **A992Fy50** definition is selected in the *Material* drop-down list.

- Scroll down the list of section, and while holding down the Ctrl key, click on the **W14X90** section and the **W24X55** section to select them.
- Click the **OK** button to access the ***I/Wide Flange Section*** form.
- Click the **OK** buttons on the ***I/Wide Flange Section*** form and the ***Frame Properties*** form to close those forms and return to the ***3D Frame*** form.

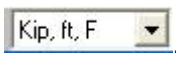
- Ensure that the **W24X55** section is selected in the *Beams* drop-down list and the **W14X90** section is selected in the *Columns* drop-down list.
- Click the **OK** button on the ***3D Frame*** form to close the form and display the template model in the program windows.

4. Click the drop-down list in the status bar to change the units to 

.
5. Click the **Define menu > Section Properties > Area Sections** command to access the ***Area Sections*** form.
   - In the *Select Section Type to Add* drop-down list, select the *Shell* option.
   - Click the **Add New Section** button to access the ***Shell Section Data*** form. In that form:
     - Type **ROOF** in the *Section Name* edit box.
     - In the *Type* area verify that the *Shell-Thin* option is selected.
     - Ensure that *4000Psi* is selected in the *Material Name* drop-down list.
     - Type **6** in the *Membrane* edit box.
     - Type **6** in the *Bending* edit box.
     - Click the **OK** button to return to the ***Area Sections*** form.
   - Click the **Add New Section** button to access the ***Shell Section Data*** form. In that form:
     - Type **FLOOR** in the *Section Name* edit box.
     - In the *Area Type* area, verify that the *Shell-Thin* option is selected.
     - Ensure that *4000Psi* is selected in the *Material Name* drop-down list.
     - Type **10** in the *Membrane* edit box.
     - Type **10** in the *Bending* edit box.
     - Click the **OK** buttons on the ***Shell Section Data*** and ***Area Sections*** forms to close all forms.
6. Click the **Define menu > Section Properties > Link/Support Properties** command to access the ***Link/Support Properties*** form. In that form:
   - Click the **Add New Property** button to access 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 access the ***Link/Support Directional Properties*** form. In that form:
       - Type **10000** in the *Effective Stiffness* edit box.

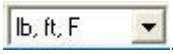
- Click the **OK** button to return to the ***Link/Support Property Data*** form.

- Check the *U2 Direction* check box.
- Check the *U2 Nonlinear* check box.
- Click the **Modify/Show For U2** button to access 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 access 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.

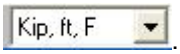
7. Click the **Define menu > Load Patterns** command to display the ***Define Load Patterns*** form. In that form:
   - Type **LIVE** in the *Load Pattern Name* edit box.
   - Select *LIVE* from the *Type* drop-down list.
   - Click the **Add New Load Pattern** button.
   - Click the **OK** button.
8. Click the drop-down list in the status bar to change the units to 

.
9. 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.
10. Click the **Move Up in List** button  until the plan display is moved up to the X-Y Plane @ Z=24.
    - Click the **Quick Draw Area Element** button  (or the **Draw menu > Quick Draw Area** command) to access the ***Properties of Object*** form. Select **Roof** from the *Section* drop-down list.

- Click once in each of the four quadrants in the plan view to draw four roof area objects.

11. Click the **Move Down in List** button  to move the plan display down to the X-Y Plane @ Z=12.
    - In the ***Properties of Object*** form, Select **Floor** from the *Section* drop-down list.
    - Click once in each of the four quadrants in the plan view to draw the four floor area objects.
12. Click the **Set Select Mode** button  to exit Draw mode and enter Select mode.
13. 11. Click the **Move Down in List** button  to move the plan display down to the X-Y Plane @ Z=0.
14. Click the **Draw menu > Draw 1 Joint Link** command to access the ***Properties of Object*** form. Select **RUB1** from the *Property* drop-down list.
15. In the plan view of the X-Y Plane @ Z=0, click on each of the nine grid intersection points to draw nine Links.
16. Click the **Set Select Mode** button  to exit Draw mode and enter Select mode.
17. Click the drop-down list in the status bar to change the units to  .
18. Click the **Move Up in List** button  until the plan display is moved up to the X-Y Plane @ Z=12.
19. Select all of the objects at the Z=12 level by “windowing.”
20. Click the **Assign menu > Area Loads > Uniform (Shell)** command to access the ***Area Uniform Loads*** form. In that form:
    - Verify that *DEAD* is selected from the *Load Pattern Name* drop-down list.
    - Type **125** in the *Load* edit box.
    - Verify that *Gravity* is selected in the *Direction* drop-down list.
    - Click the **OK** button.
21. Select all of the objects at the Z=12 level by “windowing.” As necessary, click the **Show Undeformed Shape** button  to remove the display of gravity load applied in Step 20 and reapply the title for the window.
22. Click the **Assign menu > Area Loads > Uniform (Shell)** command to access the ***Area Uniform Loads*** form. In that form:
    - Select *LIVE* from the *Load Pattern Name* drop-down list.
    - Type **100** in the *Load* edit box.
    - Click the **OK** button.
23. Click the **Show Undeformed Shape** button  to remove the displayed load assignments.
24. Click the **Move Up in List** button  to move the plan display up to the X-Y Plane @ Z=24.
25. Select all of the objects at the Z=24 level by “windowing.”
26. Click the **Assign menu > Area Loads > Uniform (Shell)** command to access the ***Area Uniform Loads*** form. In that form:
    - Select *DEAD* from the *Load Pattern Name* drop-down list.

- Type **75** in the Load edit box.
- Click the **OK** button.

27. Select all of the objects at the Z=24 level by “windowing.”
28. Click the **Assign menu > Area Loads > Uniform (Shell)** command to access the ***Area Uniform Loads*** form. In that form:
    - Select *LIVE* from the *Load Pattern Name* drop-down list.
    - Type **20** in the *Load* edit box.
    - Click the **OK** button.
29. Click the drop-down list in the status bar to change the units to 

.
30. Click the **Show Undeformed Shape** button  to remove the displayed load assignments.
31. Click the **Move Down in List** button  to move the plan display down to the X-Y Plane @ Z=12.
32. Select all of the objects at the Z=12 level by “windowing.”
33. Click the **Edit menu > Replicate** command to access 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.
34. Click in the window labeled X-Y Plane @ Z=12 to make sure it is active.
35. Click the **Move Up in List** button  to move the plan display up to the X-Y Plane @ Z=24.
36. Select all objects at the Z=24 level by “windowing.”
37. Click the **Assign menu > Joint > Constraints** command to access the ***Assign/Define Constraints*** form. In that form:
    - In the *Choose Constraint Type to Add* area, click the drop-down list that reads *Body* and then click *Diaphragm*. Click the **Add New Constraint** button to access 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.
38. Click the **Show Undeformed Shape** button  to remove the displayed constraint 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 objects at the Z=12 level by “windowing.”
41. Click the **Assign menu > Joint > Constraints** command to access the ***Assign/Define Constraints*** form. In that form:
    - In the *Choose Constraint Type to Add* area, click the drop-down list that reads *Body* and then click *Diaphragm*. Click the **Add New Constraint** button to access 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.

42. Click the **Show Undeformed Shape** button  to remove the displayed constraint assignments.
43. Click the **Move Down in List** button  to move the plan display down to the X-Y Plane @ Z=0.
44. Select all objects at the Z=0 level by "windowing."
45. Click the **Assign menu > Joint > Constraints** command to access the ***Assign/Define Constraints*** form. In that form:
    - In the *Choose Constraint Type to Add* area, click the drop-down list that reads *Body* and then click *Diaphragm*. Click the **Add New Constraint** button to access 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.
46. 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.

47. Click the **Define menu > Functions > Time History** command to access 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 access 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 access 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 access 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 access 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.

48. Click the **Define menu > Load Cases** command to access the ***Define Load Cases*** form. In that form:
    - Click on *Modal* in the *Load Case Name* list to highlight it.
    - Click the **Modify/Show Case** button to access the ***Load 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 Pattern* 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 access 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 list and that *DEAD* shows in the *Load Name* drop-down list. Select *RAMPTH* from the *Function* drop-down list. 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 ***Define Load Cases*** form.
    - Click the **Add New Load Case** button to access the ***Load Case Data*** form. In that form:

- Type **LAC** in the *Load Case Name* edit box.
- Select *Time History* from the *Load 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 list and *U1* from the *Load Name* drop-down list. Select *LACC0* from the *Function* drop-down list, 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 list and select *LACC90* from the *Function* drop-down list. 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 access 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***, ***Load Case Data***, and ***Define Load Cases*** forms to close all forms.

49. Click the **Run Analysis** button to display the ***Set Load Cases to Run*** form. In that form:
    - Verify that all load cases are set to *Run* in the *Action* list.
    - Click the **Run Now** button to run the analysis.
50. When the analysis is complete, check the messages in the ***SAP Analysis Monitor*** window and then click the **OK** button to close the window.
51. Click in the window labeled X-Y Plane @ Z=0 to make sure it is active.
52. Click the **Set Display Options** button  (or the **View menu > Set Display Options** command) to access the ***Display Options for Active Window*** form. In that form:
    - Check the *Labels* box in the *Joints* area.
    - Click the **OK** button.
53. Click on the center joint, joint 13, in the plan at Z=0 to select it.
54. Click the **Move Up in List** button  twice to move the plan display up to the X-Y Plane @ Z=24.
55. Click on the center joint, joint 15, in the plan at Z=24 to select it.
56. Click the **Set Display Options** button  (or the **View menu > Set Display Options** command) to access the ***Display Options on Active Window*** form. In that form:

- Uncheck the *Labels* box in the *Joints* area.
- Click the **OK** button.

57. Click the **Display menu > Show Plot Functions** command to access the ***Plot Function Trace Display Definition*** form. In that form:
    - Select *LAC* from the *Load Case* drop-down list.
    - Click the **Define Plot Functions** button in the *Choose Plot Functions* area to access the ***Plot Functions*** form. In that form:
        - Highlight Joint 13.
        - Click the **Modify/Show Plot Function** button to access 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 access 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 list and click **Add Plot Function** button to access 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 list, 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.