

Problem Q

Three Frames

Concrete

E = 5000 ksi, Poissons Ratio = 0.2
Beams: 24" wide by 36" deep
Columns: 24" by 24"

Damper Properties

Linear Properties

Effective stiffness = 0 k/in
Effective damping = 0 k-sec/in

Nonlinear Properties

Stiffness = 1000 k/in
Damping = 30 k-sec/in
Damping exponent = 0.5

Isolator Properties (Isolator1)

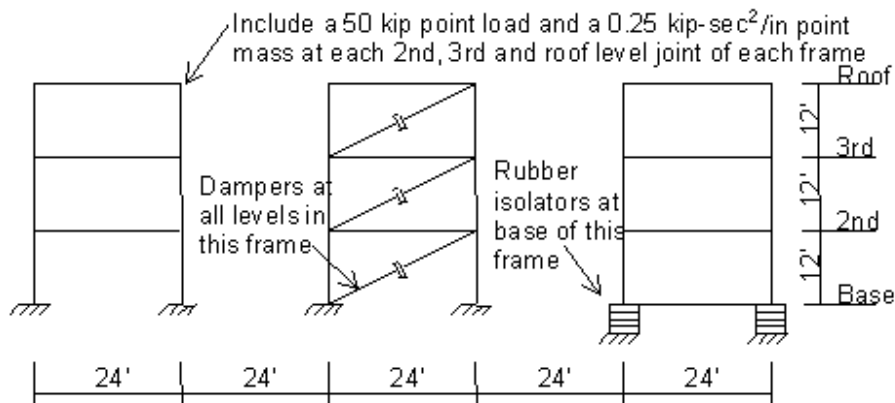
Vertical (axial) stiffness = 10,000 k/in (linear)
Initial shear stiffness = 100 k/in
Shear yield force = 40 kips
Ratio of post yield shear stiffness to initial shear stiffness = 0.1

Time History

Apply the lacc_nor-1 record. Eight acceleration values are given on each line of the file. The acceleration value is 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.

To Do

Create three frames, one bare, one with dampers, and one with isolators.
Create a video (*.avi file) of the nonlinear time history run. Review the mode shapes.



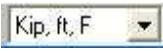
CSI Solution Demonstrates Use of These Features

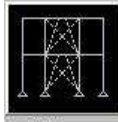
- Concrete Moment Frame
- Create Time History Video
- Dynamic Analysis
- Mode Shapes
- New Model From Template

- Link Elements
- Nonlinear Time History Analysis

Problem Q 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. On that form:

- Select *Portal* in the *2D Frame Type* drop-down list.
- Type **3** in the *Number of Stories* edit box.
- Type **5** in the *Number of Bays* edit box.
- Uncheck the *Restraints* check 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 **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 *Labels* box in the *Frames/Cables/Tendons* area.
- Click the **OK** button.

6. Select line (beam) objects 22, 23, 24, 28, 29 and 30. Press the Delete key on the keyboard to delete these objects.

Note: Select the objects by clicking on each one individually, using the **Intersecting Line Select Mode**, or using the **Select menu > Select > Labels** command.


7. Click the **Draw menu > Quick Draw Frame/Cable/Tendon** command to display the **Properties of Object** form. Verify that the *Line Object Type* is *Straight Frame*. Ignore the other values shown because the sections will be re-assigned later in the problem.

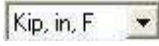
8. Click on the grid line between joints 17 and 21 to enter a line object at the base of the isolated frame.


9. Click the **Set Select Mode** button  on the side toolbar to exit Draw mode and enter Select mode.



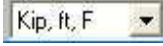
10. Select joints 1, 5, 9 and 13.



11. Click the **Assign menu > Joint > Restraints** command to display the **Joint Restraints** form. In that form:

- Click the **Fixed Base Fast Restraint** button  to set all degrees of freedom (U1, U2, U3, R1, R2 and R3) as restrained.

- Click the **OK** button.
12. Click the drop-down box in the status bar to change the units to .
 13. Click the **Define menu > Materials** command to display the **Define Materials** form.
 14. 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:
 - Type **0** in the *Mass per Unit Volume* edit box.
 - Accept the default *Weight per Unit Volume*.
 - Type **5000** in the *Modulus of Elasticity* edit box.
 - Type **.2** in the *Poisson's Ratio* edit box, if it is not already entered.
 - Click the **OK** buttons on the **Material Property Data** and **Define Materials** forms to exit all forms.
 15. Click the **Define menu > Frame Sections** command to display the **Frame Properties** form.
 16. In the *Choose Property Type to Add* area, click the drop-down box that reads *Add I/Wide Flange* and then click on the *Add Rectangular* item.
 17. Click the **Add New Property** button to display the **Rectangular Section** form. In that form:
 - Type **BEAM** in the *Section Name* edit box.
 - Select CONC in the *Material* drop-down box.
 - Type **36** in the *Depth (t3)* edit box.
 - Type **24** in the *Width (t2)* edit box.
 - Click the **OK** button to return to the **Frame Properties** form.
 18. In the *Choose Property Type to Add* area, click the drop-down box that reads *Add Rectangular* item.
 19. Click the **Add New Property** button to display the **Rectangular Section** form. In that form:
 - Type **COL** in the *Section Name* edit box.
 - Select CONC in 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 exit all forms.
 20. Select all beam objects (10 total).
 21. 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* area to highlight it.
 - Click the **OK** button.

22. Select all column objects by “windowing“ on each column line separately (18 total).
23. 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* area to highlight it.
 - Click the **OK** button.
24. Click the **Show Undeformed Shape** button  to remove the frame section assignments and redisplay the frame object labels.
25. 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 **ISO** in the *Property Name* edit box.
 - Type **.001** in the *Mass* edit box.
 - Check the *U1 Direction* check box.
 - 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 **10** in the *Effective Stiffness* edit box.
 - In the *Properties Used for Nonlinear Analysis Cases* area, type **100** in the *Stiffness* edit box.
 - Type **40** in the *Yield Strength* edit box.
 - Type **.1** in the *Post Yield Stiffness Ratio* edit box.
 - Accept the other values on the form.
26. Click the **OK** buttons on the **Link/Support Directional Properties** and **Link/Support Property Data** forms to return to 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 *Damper* from the *Link/Support Type* drop-down list.
 - Type **DAMP** in the *Property Name* edit box.
 - Type **.001** in the *Mass* edit box.

- Check the *U1 Direction* check box.
 - Check the *U1 Nonlinear* check box.
 - Click the **Modify/Show For U1** button to display the **Link Support Direction Properties** form. In that form:
 - In the *Properties Used for Nonlinear Analysis Cases* area, type **1000** in the *Stiffness* edit box.
 - Type **30** in the *Damping Coefficient* edit box.
 - Type **.5** in the *Damping Exponent* edit box.
 - Click the **OK** buttons on the **Link/Support Directional Properties**, **Link/Support Property Data**, and **Link/Support Properties** forms to exit all forms.
27. Click the **Draw menu > Draw 1 Joint Link** command to display the **Properties of Object** form. Select *ISO* from the *Property* drop-down box.
28. Click on joints 17 and 21 to draw two Link elements.
29. Click the **Set Select Mode** button  on the side toolbar to exit Draw mode and enter Select mode.
30. Click the **Draw menu > Draw 2 Joint Link Element** command to display the **Properties of Object** form. Select *DAMP* from the *Property* drop-down box.
31. Click on joint 9 and then joint 14 to draw a Link element and then hit the Enter key on the keyboard.
32. Click on joint 10 and then joint 15 to draw a Link element and then hit the Enter key on the keyboard.
33. Click on joint 11 and then joint 16 to draw a Link element and then hit the Enter key on the keyboard.
34. Click the **Set Select Mode** button  on the side toolbar to exit Draw mode and enter Select mode.
35. Select all joints at the 2nd, 3rd and Roof levels by “windowing.”
36. Click the **Assign menu > Joint Loads > Forces** command to display the **Joint Forces** form. In that form:
 - Type **-50** in the *Force Global Z* edit box.
 - Click the **OK** button.
37. Select all joints at the 2nd, 3rd and Roof levels by “windowing.”
38. Click the **Assign menu > Joint > Masses** command to display the **Joint Masses** form. In that form:
 - Type **.25** in the *Direction 1* edit box.
 - Type **.25** in the *Direction 3* edit box.
 - Click the **OK** button.
39. Click the drop-down box in the status bar to change the units to .

40. Click the **Show Undeformed Shape** button  to remove the displayed joint mass assignments.
41. 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.
 - Click the **OK** button.

Note: Before defining time history functions, locate the time history files to be used. For this problem, we are using a file named *lacc_nor-1.th*, but any time history file may be used. A number of sample files are included with SAP2000.






42. 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 box 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 time history file to be used (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* option in the *Values are* area and type **.02** in the edit box.
 - Click the **OK** buttons on the **Time History Function Definition** and **Define Time History Functions** forms to exit all forms.
43. Click the **Define menu > Analysis Cases** command to display the **Analysis Cases** form. In that form:
- Highlight (select) MODAL in the *Case Name* list.
 - Click the **Modify/Show Case** button to display the **Analysis Case Data** 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 box and that *DEAD* shows in the *Load Name* drop-down box. Click the **Add** button.

- In the *Loads Applied* area, select *Accel* from the *Load Type* drop-down box and *UX* from the *Load Name* drop-down box. Click the **Add** button.
- In the *Loads Applied* area, select *Link* from the *Load Type* drop-down box. 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 box.
 - 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 *RAMP* 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 **Modify/Show** button for Modal Damping in the *Other Parameters* area to display the **Modal Damping** form. In that form:
 - Type **.02** in the *Constant Damping For All Modes* edit box.
 - Click the **OK** buttons on the **Modal Damping** and **Analysis Case Data** forms 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 box.
 - 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 *LACCO* from the *Function* drop-down box, and type **0.0328** in the *Scale Factor* edit box. Click the **Add** button.
 - Type **1210** in the *Number of Output Time Steps* edit box.
 - Type **.01** in the *Output Time Step Size* edit box.
 - Click the **Modify/Show** button for Modal Damping in the *Other Parameters* area to display the **Modal Damping** form. In that form:
 - Type **.02** in the *Constant Damping For All Modes* edit box.
 - Click the **OK** buttons on the **Modal Damping**, **Analysis Case Data** and **Analysis Cases** forms to exit all forms.

44. 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.
45. 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.
46. When the analysis is complete, check the messages in the **SAP Analysis Monitor** window and then click the **OK** button to close the window.
47. Click the **Start Animation** button  on the status bar at the bottom of the screen to animate the first mode shape.
48. Click the **Right Arrow** button  on the status bar at the bottom of the screen to view the next mode shape.
49. Continue clicking the **Right Arrow** button  to step through all of the mode shapes.
50. Click the **Show Undeformed Shape** button  to remove the displayed mode shape.
51. Click the **File menu > Create Video > Create Multi-step Animation Video** command to display the **Video File** form. In that form, select the name and location for the video file (*.avi) and click the **Save** button. The **Multi-step Video File Creation** form is displayed. In that form:
- Select LAC from the *Case Name* drop-down box.
 - Type **50** in the *Magnification Factor* edit box.
 - Verify that 640 by 480 is entered in the *Frame Size (Pixels)* edit boxes.
 - Click the **OK** button. The *.avi file is created.

Note: After the *.avi file has been created, it can be played on any multimedia player that supports *.avi files.

Note: The purpose of this example is to display the capabilities of SAP2000. The structural properties of each frame have not been optimized in this example. Therefore, great care should be taken in drawing any conclusions about the relative performance of different structural systems based on this example.