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
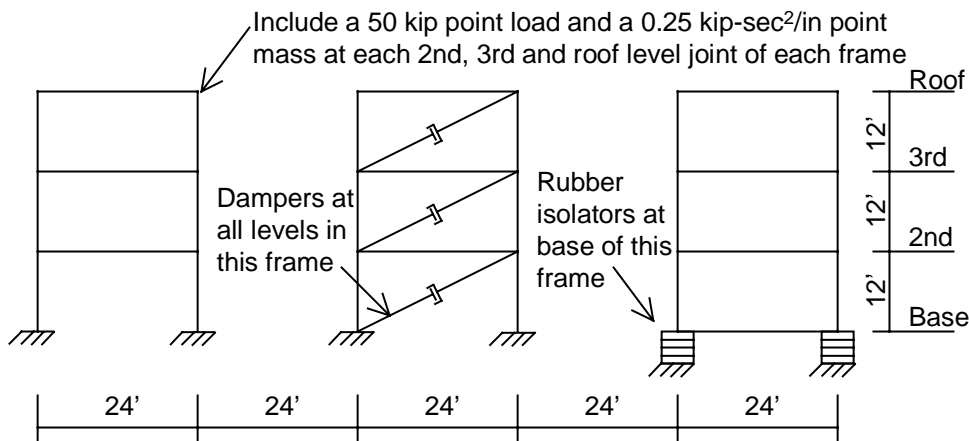Ratio of post yield shear stiffness to initial shear stiffness = 0.1

Time History
Apply the ELCENTRO record. Three time and acceleration values are given
on each line of this file.   The acceleration value is in units of g.   The length
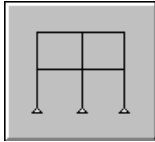of the record is 12.1 seconds.

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.

Include a 50 kip point load and a 0.25 kip-sec^2/in point
mass at each 2nd, 3rd and roof level joint of each frame

Note: Our intent is that you try this problem on your own first. After you have solved it on your
own, you can step through our solution if desired. If you have problems trying to create the
model, then follow the steps in our solution.

Problem Q Solution

1. Click the drop down box in the status bar to change the units to kip-ft. `Kip-ft ▼`

2. From the **File** menu select **New Model From Template…**. This displays the Model Templates dialog box.

3. In this dialog box click on the **Portal Frame** template button to display the Portal Frame dialog box.

4. In this dialog box

    *   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.

5. Click the "X" in the top right-hand corner of the 3-D View window to close it.

6. Click the **Set Elements** button ⊠ on the main toolbar (or select **Set Elements…** from the **View** menu) to display the Set Elements Dialog box.

7. In this dialog box:

    *   Check the Labels box in the Joints area.

    *   Check the Labels box in the Frames area.

    *   Click the **OK** button.

8. Select beam elements 22, 23, 24, 28, 29 and 30. Press the delete key on the keyboard to delete these elements.

    *Note: You could select the elements by clicking on each one individually, by using the Intersecting Line Select Mode, or by using the Select By Labels option (**Select** menu > **Select** > **Labels**).*

9. Click the **Refresh Window** button 🖉 to refresh the drawing.

10. From the **Draw** menu select **QuickDraw Frame Element**.

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

12. Click the **Pointer** button ▶ on the side tool bar to exit draw mode and enter select mode.

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

14. From the **Assign** menu, choose **Joint**, and then **Restraints…** from the submenu. This will display the Joint Restraints dialog box.

15. In this dialog box:

    • 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.

16. Click the drop down box in the status bar to change the units to kip-in. `Kip-in ▼`

17. From the **Define** menu select **Materials...** to display the Define Materials dialog box.

18. Click on CONC in the Materials area to highlight (select) it, and then click the **Modify/Show Material** button. The Material Property Data dialog box is displayed.

19. In this dialog box:

    • 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** button twice to exit all dialog boxes.

20. From the **Define** menu select **Frame Sections...** to display the Define Frame Sections dialog box.

21. In the Click To area, click the drop-down box that says Add I/Wide Flange and then click on the Add Rectangular item to display the Rectangular Section dialog box.

22. In this dialog box:

    • 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 Define Frame Sections dialog box.

23. In the Click To area, click the drop-down box that says Add Rectangular and then click on the Add Rectangular item to display the Rectangular Section dialog box.

24. In this dialog box:

   - 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** button twice to exit all dialog boxes.

25. Select all beam elements (10 total).

26. From the **Assign** menu select **Frame** and then **Sections...** from the submenu to display the Define Frame Sections dialog box.

27. In this dialog box:

   - Click on BEAM in the Frame Sections area to highlight it.

   - Click the **OK** button.

28. Select all column sections by "windowing " on each column line separately (18 total).

29. From the **Assign** menu select **Frame** and then **Sections...** from the submenu to display the Define Frame Sections dialog box.

30. In this dialog box:

   - Click on COL in the Frame Sections area to highlight it.

   - Click the **OK** button.

31. Click the **Show Undeformed Shape** button ⊚ to remove the displayed frame section assignments so that you can see the frame element labels again.

32. From the **Define** menu select **NLLink Properties...** to display the Define NLLink Properties dialog box.

33. In this dialog box:

   - Click the **Add New Property** button to display the NLLink Property Data dialog box.

- In this dialog box:

  - Type **ISO** in the Property Name edit box.

  - Select Isolator1 from the Type drop-down box.

  - Type **.001** in the Mass edit box.

  - Check the U1 Direction check box.

  - Click the **Modify/Show For U1** button to display the NLLink Directional Properties dialog box.

  - In this dialog box:

    - Type **10000** in the Effective Stiffness edit box.

    - Click the **OK** button to return to the NLLink Property Data dialog box.

  - Check the U2 Direction check box.

  - Check the U2 Nonlinear check box.

  - Click the **Modify/Show For U2** button to display the NLLink Directional Properties dialog box.

  - In this dialog box:

    - In the Linear Properties area type **10** in the Effective Stiffness edit box.

    - In the Nonlinear Properties 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 rest of the default values.

    - Click the **OK** button to return to the NLLink Property Data dialog box.

  - Click the **OK** button to return to the Define NLLink Properties dialog box.

- Click the **Add New Property** button to display the NLLink Property Data dialog box.

- In this dialog box:

  - Type DAMP in the Property Name edit box.

  - Select Damper from the Type drop-down 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 NLLink Directional Properties dialog box.

- ➢ In this dialog box:

  - ✓ In the Nonlinear Properties area type **1000** in the Stiffness edit box.

  - ✓ Type **30** in the Damping edit box.

  - ✓ Type **.5** in the Damping Exponent edit box.

  - ✓ Click the **OK** button three times to exit all dialog boxes.

34. From the **Draw** menu select **Draw NLLink Element**.

35. Double click on joints 17 and 21 to draw two NLLink elements.

36. Click on joint 9 and then joint 14 to draw an NLLink element.

37. Click on joint 10 and then joint 15 to draw an NLLink element.

38. Click on joint 11 and then joint 16 to draw an NLLink element.

39. Click the **Pointer** button ▨ on the side tool bar to exit draw mode and enter select mode.

40. Click on the three NLLink element in the center frame to select them.

41. From the **Assign** menu select **NLLink** and then **Properties...** from the submenu to display the Define NLLink Properties dialog box.

42. In this dialog box:

- • Click on DAMP in the NLLink Props area to highlight it.

- • Click the **OK** button.

43. Click the **Show Undeformed Shape** button ◎ to remove the displayed NLLink property assignments so that you can see the joint labels again.

44. Hold down the Ctrl key on the keyboard and left click joint 17 to display the Selection List dialog box.

45. In the dialog box click on the NLLink 1 element. The Selection List dialog box closes and the NLLink element is selected.

46. Hold down the Ctrl key on the keyboard and left click joint 21 to display the Selection List dialog box.

47. In the dialog box click on the NLLink 2 element. The Selection List dialog box closes and the NLLink element is selected.

48. From the **Assign** menu select **NLLink** and then **Properties...** from the submenu to display the Define NLLink Properties dialog box.

49. In this dialog box:

    - Click on ISO in the NLLink Props area to highlight it.

    - Click the **OK** button.

50. Click the **Show Undeformed Shape** button ⊙ to remove the displayed NLLink property assignments.

51. Select all joints at the 2nd, 3rd and Roof levels by "windowing".

52. From the **Assign** menu select **Joint Static Loads...** and then **Forces...** from the submenu to display the Joint Forces dialog box.

53. In this dialog box:

    - Type **-50** in the Force Global Z edit box.

    - Click the **OK** button.

54. Select all joints at the 2nd, 3rd and Roof levels by "windowing".

55. From the **Assign** menu select **Joint** and then **Masses...** from the submenu to display the Joint Masses dialog box.

56. In this dialog box:

    - Type **.25** in the Direction 1 edit box.

    - Type **.25** in the Direction 3 edit box.

    - Click the **OK** button.

57. Click the drop down box in the status bar to change the units to kip-ft. [Kip-ft ▼]

58. Click the **Show Undeformed Shape** button ⊙ to remove the displayed joint mass assignments.

59. Click the **Set Elements** button ⊠ on the main toolbar (or select **Set Elements…** from the **View** menu) to display the Set Elements Dialog box.

60. In this dialog box:

    - Uncheck the Labels box in the Joints area.

    - Uncheck the Labels box in the Frames area.

    - Click the **OK** button.

    Note: Prior to defining time history functions, you should locate the time history file named Elcentro that is in the subdirectory named Examples beneath the directory where you installed SAP2000. Copy this file into the same directory as your SAP2000 input file.
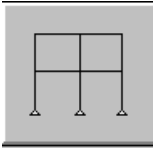
    If the Examples subdirectory does not exist you may need to reinstall SAP2000, and select to install the examples.

61. From the **Define** menu select **Time History Functions...** to display the Define Time History Functions dialog box.

62. In this dialog box:

    - Click the Add Function From File button to display the Time History Function Definition dialog box.

    - In this dialog box:

        ➢ Type **ELCEN** in the Function Name edit box.

        ➢ Click the **Open File** button to display the Pick Function Data File dialog box.

        ➢ In this dialog box:

            ✓ Locate and highlight the file named Elcentro

            ✓ Click the **Open** button to return to the Time History Function Definition dialog box.

            ✓ Type **3** in the Number Of Points Per Line edit box.

            ✓ Select the Time and Function Values option.

            ✓ Click the **OK** button twice to exit all dialog boxes.

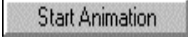63. From the **Analyze** menu select **Set Options...** to display the Analysis Options dialog box.

- In this dialog box click the **Plane Frame XZ Plane** button to set the available degrees of freedom.

- Check the Dynamic Analysis check box, if it is not already checked.

- Click the **Set Dynamic Parameters** button to display the Dynamic Analysis Parameters dialog box.

- In this dialog box:

  ➢ Type **30** in the Number of Modes edit box.

  ➢ In the Type Of Analysis area select the Ritz Vectors option.

  ➢ Verify that ACCEL X and ACCEL Z are in the Ritz Load Vectors box in the Starting Ritz Vectors area. Move any other vectors out of the Ritz Load Vectors box

  ➢ Confirm that the Include NLLink Vectors box is checked.

  ➢ Click the **OK** button twice to exit all dialog boxes.

64. From the **Define** menu select **Time History Cases...** to display the Define Time History Cases dialog box.

65. In this dialog box:

- Click the **Add New History** button to display the Time History Case Data dialog box.

- In this dialog box:

  ➢ Type **GRAV** in the History Case Name edit box.

  ➢ Select Nonlinear from the Analysis Type drop-down box.

  ➢ Click the **Modify/Show** button for modal damping to display the Modal Damping dialog box.

  ➢ In this dialog box:

    ✓ Type **.02** in the Damping For All Modes edit box.

    ✓ Click the **OK** button.

  ➢ Type **100** in the Number of Output Time Steps edit box.

  ➢ Type **.1** in the Output Time Step Size edit box.

  ➢ Check the Envelopes check box.

- ➢ In the Load drop-down box select GRAV.

- ➢ In the Function drop-down box, select RAMP.

- ➢ Type **1** in the Scale Factor edit box.

- ➢ Click the **Add** button.

- ➢ Click the **OK** button to return to the Define Time History Cases dialog box.

- • Click the **Add New History** button to display the Time History Case Data dialog box.

- • In this dialog box:

  - ➢ Type **ELCN** in the History Case Name edit box.

  - ➢ Select Nonlinear from the Analysis Type drop-down box.

  - ➢ Click the **Modify/Show** button for modal damping to display the Modal Damping dialog box.

  - ➢ In this dialog box:

    - ✓ Type **.02** in the Damping For All Modes edit box.

    - ✓ Click the **OK** button.

  - ➢ Type **1210** in the Number of Output Time Steps edit box.

  - ➢ Type **.01** in the Output Time Step Size edit box.

  - ➢ Check the Envelopes check box.

  - ➢ In the Load drop-down box select acc dir 1.

  - ➢ In the Function drop-down box, select ELCEN.

  - ➢ Type **32.2** in the Scale Factor edit box.

  - ➢ Click the **Add** button.

  - ➢ Click the **OK** button twice to exit all dialog boxes.

66. Click the **Run Analysis** button ▶ to run the analysis.

67. When the analysis is complete check the messages in the Analysis window (there should be no warnings or errors). Click the **OK** button to close the Analysis window.

68. Click the **Start Animation** button [Start Animation] located on the status bar at the bottom of the screen to animate the first mode shape.

69. Click the **Right Arrow** button ⇨ located on the status bar at the bottom of the screen to view the next mode shape.

70. Continuing clicking the **Right Arrow** button ⇨ to step through all of the mode shapes.

71. Click the **Show Undeformed Shape** button ◎ to remove the displayed mode shape.

72. From the **File** menu select **Create Video…** and then select **Create Time History Animation Video…** to display the Video File dialog box.

73. In this dialog box select the name and location for the video file (*.avi) and click the **Save** button. The Time History Video File Creation dialog box is displayed.

74. In this dialog box:

    - Type **50** in the Magnification Factor edit box.

    - In the Frame Size (Pixels) edit boxes type **640** by **480**.

    - Click the **OK** button. The *.avi file is created.

    *Note: Once the *.avi file has been created it can be played on any multimedia player that supports *.avi files. A program called Media Player is supplied with Windows 95 that will play *.avi files. This program can often be found by clicking the **Start** button, selecting **Programs**, then selecting **Accessories** and finally selecting **Multimedia**. No multimedia player is provided with SAP2000.*

    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.