Problem O

Isolated Building - Nonlinear Time History Analysis

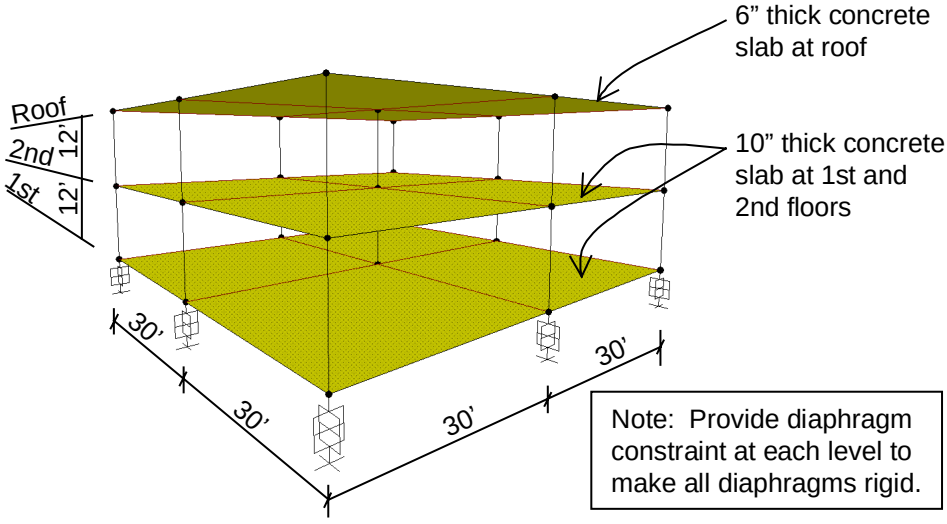
Steel
E =29000 ksi, Poissons Ratio = 0.3
Beams: W24X55; Columns: W14X90

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 Floor: 125 psf DL
20 psf LL 100 psf LL

Time History
Apply LP-TH0 in the X-direction and LP-TH90 in the Y-direction simultaneously. Each timehistory is given in units of g. There are 2000 timesteps, at an equal spacing of 0.02 sec, for a total of 40 sec. There are 5 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.

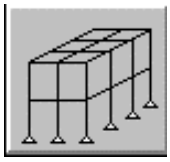


Note: Our intent is that you try this problem on your own first. After you have solved it on your own, you can step through our solution if desired. If you have problems trying to create the model, then follow the steps in our solution.

Problem O Solution

1. Click the drop down box in the status bar to change the units to 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 **Space Frame** template button  to display the Space Frame dialog box.

4. In this dialog box:

   - Type **2** in the Number of Bays Along X edit box.
   - Type **30** in the Bay Width Along X edit box.
   - Type **30** in the Bay Width Along Y edit box.
   - Uncheck the Restraints check box.
   - Accept the rest of the default values.
   - Click the **OK** button.

5. Click in the window labeled X-Y Plane @ Z=24 to make sure it is active. The window is active when its title is highlighted.

6. Click the **Quick Draw Rectangular Shell Element** button  on the side toolbar.

7. Click once in each of the four quadrants in the plan view to input four shell elements.

8. Click the **Down One Gridline** 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 input four shell elements.

10. Click the **Down One Gridline** button  to move the plan display down to the X-Y Plane @ Z=0.

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

12. In the plan view of the X-Y Plane @ Z=0 double click on each of the nine joints to draw nine NLLink elements.

13. Click the **Pointer** button  to exit draw mode and enter select mode.

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

15. From the **Define** menu select **Materials...** to display the Define Materials dialog box. Highlight the STEEL material and click the **Modify/Show Material** button to display the Material Property Data dialog box.

16. In this dialog box:

    - Verify that the modulus of elasticity is 29000 and poisson's ratio is 0.3.

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

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

18. In the Click To area, click the drop-down box that says Import I/Wide Flange and then click on the Import I/Wide Flange item.

19. If the Section Property File dialog box appears then locate the Sections.pro file which should be located in the same directory as the SAP2000 program files.

20. A dialog box appears with a list of all wide flange sections in the database. In this dialog box:

    - Scroll down and click on the W24X55 section.

    - Scroll down to the W14X90 section, and click on it while holding down the Ctrl key on the keyboard.

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

21. From the **Define** menu select **Shell Sections...** to display the Define Shell Sections dialog box.

22. In this dialog box:

    - Click the **Add New Section** button to display the Shell Sections dialog box.

    - In this dialog box:

        - Type **ROOF** in the Section Name edit box.

        - Accept the default CONC material

        - 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 Define Shell Sections dialog box.

- Click the **Add New Section** button to display the Shell Sections dialog box.
- In this dialog box:
  - Type **FLOOR** in the Section Name edit box.
  - Accept the default CONC material
  - 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** button twice to exit all dialog boxes.

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

24. In this dialog box:

- Click the **Modify/Show Property** button to display the NLLink Property Data dialog box.
- In this dialog 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 **1.5** in the Effective Stiffness edit box.
- ✓ In the Nonlinear Properties 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 rest of the default values.
- ✓ Click the **OK** button to return to the NLLink Property Data dialog box.

➢ Check the U3 Direction check box.

➢ Check the U3 Nonlinear check box.

➢ Click the **Modify/Show For U3** button to display the NLLink Directional Properties dialog box.

➢ In this dialog box:

- ✓ In the Linear Properties area type **1.5** in the Effective Stiffness edit box.
- ✓ In the Nonlinear Properties 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 rest of the default values.
- ✓ Click the **OK** button three times to exit all dialog boxes.

25. Click the drop down box in the status bar to change the units to kip-ft. 

26. From the **Define** menu select **Materials...** to display the Define Materials dialog box. Highlight the CONC material and click the **Modify/Show Section** button to display the Material Property Data dialog box.

27. In this dialog box:

- Verify that the mass per unit volume is 4.657E-03 and that the weight per unit volume is 0.15.
- Click the **OK** button twice to exit the dialog boxes.

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

29. Click the **xz 2D View** button 

on the main toolbar.

30. Click the **Perspective Toggle** button  on the main toolbar.

31. Click the Set Intersecting Line select Mode button  on the side toolbar and select all of the bottom level columns.

    *Note: To use the Intersecting Line Selection option, click the **Set Intersecting Line Select Mode** button  on the side tool bar. 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 elements that this "rubberband line" passes through will be selected. Release the left mouse button to make the selection.*

32. Click the Set Intersecting Line select Mode button  on the side toolbar and select all of the top level columns.

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

34. In this dialog box:

    - Click on W14X90 in the Frame Sections area to highlight it.
    - Click the **OK** button.

35. Click the **Show Undeformed Shape** button  to remove the displayed frame element assignments.

36. Click the **xy 2D View** button  on the main toolbar. The plan view of the X-Y Plane @ Z=0 appears.

37. Click the **Up One Gridline** button  to move the plan display up to the X-Y Plane @ Z=12.

38. Select all of the elements at this level by "windowing".

39. Click the **Up One Gridline** button  to move the plan display up to the X-Y Plane @ Z=24.

40. Select all of the elements at this level by "windowing".

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

42. In this dialog box:

    - Click on W24X55 in the Frame Sections area to highlight it.
    - Click the **OK** button.

43. Click the **Show Undeformed Shape** button  to remove the displayed frame element assignments.

44. Select all of the elements at the Z=24 level by “windowing”.

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

46. In this dialog box:

    - Click on ROOF in the Shell Sections area to highlight it.
    - Click the **OK** button.

47. Click the **Down One Gridline** button  to move the plan display down to the X-Y Plane @ Z=12.

48. Select all of the elements at this level by “windowing”.

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

50. In this dialog box:

    - Click on FLOOR in the Shell Sections area to highlight it.
    - Click the **OK** button.

51. From the **Define** menu select **Static Load Cases...** to display the Define Static Load Case Names dialog box.

52. In this dialog box:

    - Type **DL** in the Load edit box.
    - Click the **Change Load** button.
    - Type **LL** in the Load edit box.
    - Select LIVE from the Type drop-down box.
    - Type **0** in the Self Weight Multiplier edit box.
    - Click the **Add New Load** button.
    - Click the **OK** button.

53. Click the drop down box in the status bar to change the units to lb-ft. 

54. Select all of the elements at the Z=12 level by “windowing”.

55. From the **Assign** menu select **Shell Static Loads…** and then **Uniform...** from the submenu to display the Shell Uniform Loads dialog box.

56. In this dialog box:

    - Select LL from the Load Case Name drop-down box.
    - Type **-100** in the Load edit box.
    - Click the **OK** button.

57. Click the **Up One Gridline** button  to move the plan display up to the X-Y Plane @ Z=24.

58. Select all of the elements at the Z=24 level by “windowing”.

59. From the **Assign** menu select **Shell Static Loads…** and then **Uniform...** from the submenu to display the Shell Uniform Loads dialog box.

60. In this dialog box:

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

61. Click the drop down box in the status bar to change the units to kip-ft. 

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

63. Click the **Down One Gridline** button  to move the plan display down to the X-Y Plane @ Z=12.

64. Select all of the elements at the Z=12 level by “windowing”.

65. From the **Edit** menu select **Replicate…** to display the Replicate dialog box.

66. In this dialog box:

    - Select the Linear Tab.
    - Type **-12** in the Z Distance edit box.
    - Type **1** in the Number edit box.
    - Click the **OK** button.

Note: Prior to defining time history functions, you should locate the time history files named Lp-th0 and Lp-th90 that are in the subdirectory named Examples beneath the directory where you installed SAP2000. Copy these files 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.

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

68. 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 **LPTH0** 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 LP-TH0
    - ✓ Click the **Open** button to return to the Time History Function Definition dialog box.
    - ✓ Type **5** in the Number Of Points Per Line edit box.
    - ✓ Select the Function At Equal Time Step option.
    - ✓ Type **.02** in the Function At Equal Time Step edit box.
    - ✓ Click the **OK** button to return to the Define Time History Functions dialog box.
- Click the **Add Function From File** button to display the Time History Function Definition dialog box.
- In this dialog box:
  - Type LPTH90 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 LP-TH90.

- ✓ Click the **Open** button to return to the Time History Function Definition dialog box.
- ✓ Type **5** in the Number Of Points Per Line edit box.
- ✓ Select the Function At Equal Time Step option.
- ✓ Type **.02** in the Function At Equal Time Step edit box.
- ✓ Click the **OK** button twice to exit all dialog boxes.

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

- 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, ACCEL Y and ACCEL Z are in the Ritz Load Vectors box in the Starting Ritz Vectors area.
  - Confirm that the Include NLLink Vectors box is checked.
  - Click the **OK** button twice to exit all dialog boxes.

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

71. 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 **.05** 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 DL.

➢ 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 **LP** 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 **.05** in the Damping For All Modes edit box.
- ✓ In the Modal Damping Overrides area type 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 Type 2 in the Mode box and click the **Add** button.
- ✓ In the Modal Damping Overrides area type Type 3 in the Mode box and click the **Add** button.
- ✓ Click the **OK** button.

➢ Type **2000** in the Number of Output Time Steps edit box.

➢ Type **.02** in the Output Time Step Size edit box.

- In the Start From Previous History drop-down box select GRAV.
- Check the Envelopes check box.
- In the Load drop-down box, select acc dir 1.
- In the Function drop-down box, select LPTH0.
- Type **32.2** in the Scale Factor edit box.
- Click the **Add** button.
- In the Load drop-down box, select acc dir 2.
- In the Function drop-down box, select LPTH90.
- Click the **Add** button.
- Click the **OK** button twice to exit all dialog boxes.

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

73. Click the **Up One Gridline** button  to move the plan display up to the X-Y Plane @ Z=24.

74. Select all elements at the Z=24 level by “windowing”.

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

76. In this dialog box:

    - Click the drop-down box in the Click To area, and click Add Diaphragm to display the Diaphragm Constraint dialog box.
    - In this dialog box:
        - 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** button twice to assign the diaphragm constraint.

77. Click the **Down One Gridline** button  to move the plan display down to the X-Y Plane @ Z=12.

78. Select all elements at the Z=12 level by “windowing”.

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

80. In this dialog box:

    - Click the drop-down box in the Click To area, and click Add Diaphragm to display the Diaphragm Constraint dialog box.

    - In this dialog box:

        - 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** button twice to assign the diaphragm constraint.

81. Click the **Down One Gridline** button  to move the plan display down to the X-Y Plane @ Z=0.

82. Select all elements at the Z=0 level by "windowing".

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

84. In this dialog box:

    - Click the drop-down box in the Click To area, and click Add Diaphragm to display the Diaphragm Constraint dialog box.

    - In this dialog box:

        - 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** button twice to assign the diaphragm constraint.

85. Click the **Show Undeformed Shape** button  to remove the displayed diaphragm constraint assignments.

86. Click the **Run Analysis** button  to run the analysis.

    Note: The analysis would run even quicker if we had not requested envelopes in the time history case data.

87. 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.

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

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

90. In this dialog box:

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

91. Click on the center joint, joint 13, in the plan at Z=0 to select it.

92. Click the **Up One Gridline** button  twice to move the plan display up to the X-Y Plane @ Z=24.

93. Click on the center joint, joint 15, in the plan at Z=24 to select it.

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

95. In this dialog box:

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

96. From the **Display** menu select **Show Time History Traces…** to display the Time History Display Definition dialog box.

97. In this dialog box:

    - Click the **Define Functions** button to display the Time History Functions dialog box.
    - In this dialog box:
        - Highlight Joint 13.
        - Click the **Modify/Show TH Function** button to display the Time History Joint Function dialog box.
        - In this dialog box:
            - ✓ Verify that the Displ option is selected in the Vector Type area.
            - ✓ Select the UY option is selected in the Component area.
            - ✓ Click the **OK** button to return to the Time History Functions dialog box.
        - Highlight Joint 15.

- Click the **Modify/Show TH Function** button to display the Time History Joint Function dialog box.
- In this dialog box:
  - ✓ Verify that the Displ option is selected in the Vector Type area.
  - ✓ Select the UY option is selected in the Component area.
  - ✓ Click the **OK** button to return to the Time History Functions dialog box.
- In the Click To area select Add Base Functions from the drop-down box to display the Base Functions dialog box.
- In this dialog box:
  - ✓ Check the Base Shear Y check box.
  - ✓ Click the **OK** button twice to return to the Time History Display Definition dialog box.

- Select LP from The Time History Case drop-down box.
- Click on Joint 13 in the List of Functions to highlight (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 Plot 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 time history display and return to the Time History Display Definition dialog box.
- Click the F(t) vs F(t) tab.
- Select Joint 13 from the Horizontal drop-down box.
- Select Base Shear Y from the Vertical drop-down box.
- Click the **Display** button to display the force-displacement plot.
- Click the **OK** button to close the Time History Functions display and return to the Time History Display Definition dialog box.
- Click the **Done** button to close the Time History Display Definition dialog box.