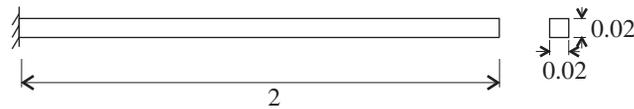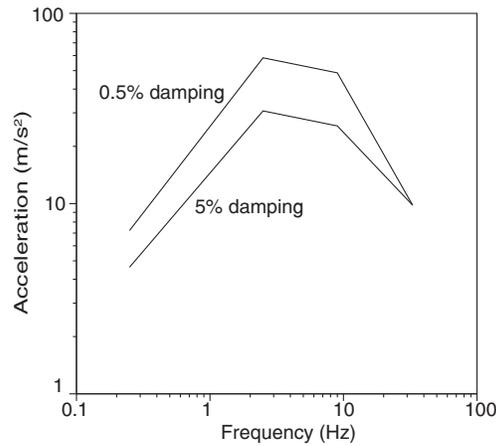## Problem description

In this problem, we subject the beam structure shown below to an earthquake load.

All lengths in meters.   Mode 1: 1% damping
$E = 2.07 \times 10^{11}$ N/m^2 Mode 2: 3% damping
$\rho = 7800$ kg/m^3

The loading response spectrum is shown below. The earthquake load acts in the vertical direction only.

We make the beam 2 meters long in this problem so that the natural frequencies are in the range of frequencies amplified by the earthquake.

In this problem solution, we demonstrate the following topics that have not been presented in previous problems:

• Setting up a response spectrum analysis
• Defining an element result point
• Listing the modal masses

We assume that you have worked through problem 1 to 11, or have equivalent experience with the ADINA System.

Before you begin

Please refer to the Icon Locator Tables chapter of the Primer for the locations of all of the AUI icons. Please refer to the Hints chapter of the Primer for useful hints.

This problem can be solved with the 900 nodes version of the ADINA System.

Invoking the AUI and choosing the finite element program

Invoke the AUI and set the Program Module drop-down list to ADINA Structures.

Defining the model

Problem heading: Choose Control→Heading, enter "Problem 12: Beam subjected to earthquake load" and click OK.

Geometry: Click the Define Points icon , define the following points and click OK.

Point #	X1	X2	X3
1			
2	2		
3		0.1	

Click the Define Lines icon , add line 1, define a straight line with Point 1 = 1 and Point 2 = 2 and click OK.

Boundary conditions: Click the Apply Fixity icon , enter 1 in the first row of the Point table and click OK.

Cross-section and material: Click the Cross Sections icon , add cross-section 1, set the Width field to 0.02, click the Square Section button and click OK. Click the Manage Materials icon and click the Elastic Isotropic button. In the Define Isotropic Linear Elastic Material dialog box, add material 1, set the Young's Modulus to 2.07E11, the Density to 7800 and click OK. Click Close to close the Manage Material Definitions dialog box.

Finite elements: Click the Subdivide Lines icon , set the Number of Subdivisions to 2 and click OK.

Click the Mesh Lines icon , set the Type to Beam, click the + button to the right of the Element Group text, set the Auxiliary Point to 3, enter 1 in the first row of the table and click OK.

Specifying the analysis options

Analysis type: Set the Analysis Type drop-down list to Modal Participation Factors and click the Analysis Options icon ⬛ . Click the Settings button, set the Number of Frequencies/Mode Shapes to 2 and click OK to close the dialog box. In the Modal Participation Factors dialog box, set the "Number of Modes to Use" to 2, verify that the "Type of Excitation Load" is set to Ground Motion and click OK to close the dialog box.

Degrees of freedom: In order to suppress all out-of-plane modes, we force the model to vibrate in the X-Y plane. Choose Control→Degrees of Freedom, uncheck the Z-Translation, X-Rotation and Y-Rotation buttons and click OK.

When you click the Boundary Plot icon ▆, the graphics window should look something like this:
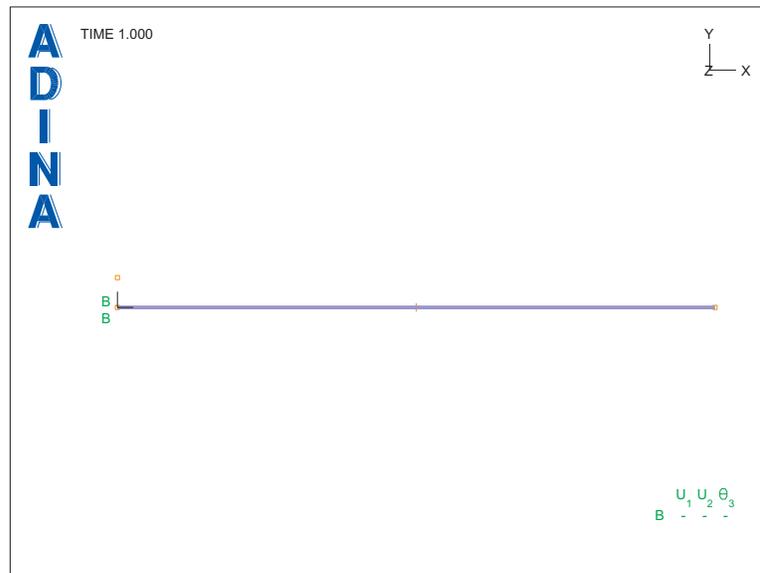
Generating the data file, running ADINA Structures, loading the porthole file

Click the Save icon 🖫 and save the database to file `prob12`. Click the Data File/Solution icon 📄❗, set the file name to `prob12`, make sure that the Run Solution button is checked and click Save. When ADINA Structures is finished, close all open dialog boxes, set the Program Module drop-down list to Post-Processing (you can discard all changes), click the Open icon 📂 and open porthole file `prob12`.

Listing the natural frequencies, modal participation factors and modal masses

To list the modal data, choose List→Info→MPF. In the first table, the frequencies should be 4.16284E+00 (Hz), 2.62936E+01 and the Y modal participation factors should be 1.89159E+00 and 8.14598E-01. In the second table, the Y modal masses should be 3.57811 (kg) and 6.63570E-01 (kg). In the third table, the accumulated Y modal masses should be 3.57811 and 4.24168. Note that only two modes are required to pick up a significant amount of the total mass (which is 6.24 kg). Click Close to close the dialog box.

Defining the loading response spectrum

Frequency curves: The loading response spectrum consists of two frequency curves, one for a damping of 0.5 % and the other for a damping of 5.0 %. Each frequency curve gives the acceleration as a function of frequency. Choose Definitions→Spectrum Definitions→ Frequency Curve, add frequency curve F05 and define it as follows:

Frequency	Value
0.25	7.22
2.5	58.37
9.0	48.66
33.0	9.81

Then add frequency curve F50 and define it as follows:

Frequency	Value
0.25	4.63
2.5	30.71
9.0	25.60
33.0	9.81

Click OK to close the dialog box.

Response spectrum: Choose Definitions→Spectrum Definitions→Response Spectrum, add response spectrum RS1, enter F05, 0.5 in the first row of the table, F50, 5.0 in the second row of the table. Then click Save and click the Graph... button. In the Display Response Spectrum dialog box, set the Response Spectrum to RS1 and click OK. Click Cancel to close the other dialog box.

To delete the mesh plot, click the Pick icon ➤, highlight the mesh plot and press the Delete key. The graphics window should look something like the figure on the next page.
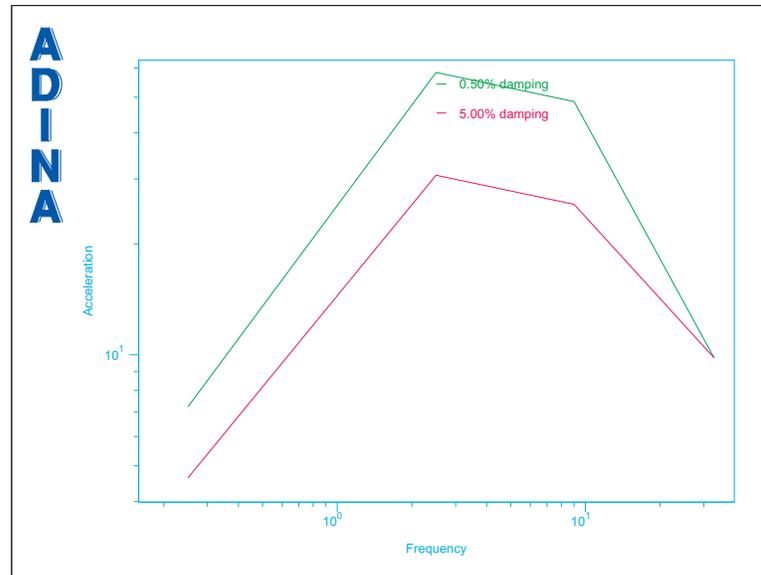
Defining the modal damping ratios

Choose Definitions→Spectrum Definitions→Damping Table, add damping table name DAMPING, set "Defined by" to Tabular Input, set the first two rows of the table to 1, 1.0; 2, 3.0 and click OK.

Computing the response due to the earthquake load

Response definition: Choose Definitions→Response, add response EARTHQUAKE and set the Type to Response Spectrum. Set the Spectrum to RS1, the Damping Table to DAMPING, the Ground Motion Direction to Y and click OK.

Computations: Choose List→Value List→Zone, set the Response Option to Single Response and the Response to EARTHQUAKE. Then set Variable 1 to (Displacement: Y-DISPLACEMENT), variable 2 to (Velocity: Y-VELOCITY), variable 3 to (Acceleration:Y-ACCELERATION), variable 4 to (Reaction:Z-MOMENT_REACTION) and click Apply.

The results for node 1 should be reaction = 2.74963E+02 (N-m) and the results for node 3 should be displacement = 1.13969E-01 (m), velocity = 2.98134E+00 (m/s), acceleration = 7.83830E+01 (m/s^2). Click Close to close the dialog box.

To compute the internal bending moments at the built-in end, we use an element result point corresponding to the built-in end. Results for the built-in end are computed at element 1, local node 1. Choose Definitions→Model Point→Element, add point BUILT-IN, verify that the

Element Number is 1, set "Defined By" to Label Number, verify that the Label # is 1 and click OK.

Now choose List→Value List→Model Point, set the Response Option to Single Response and the Response to EARTHQUAKE. Then set Variable 1 to (Force:NODAL_MOMENT-T) and click Apply. The result should be 2.74963E+02 (N-m). Click Close to close the dialog box.

Exiting the AUI: Choose File→Exit to exit the AUI (you can discard all changes).

Additional note

It is easier to define the response spectrum in a batch file than it is to enter the response spectrum using the dialog boxes. In primer problem 26, we define the response spectrum in a batch file and then read the batch file into the AUI.