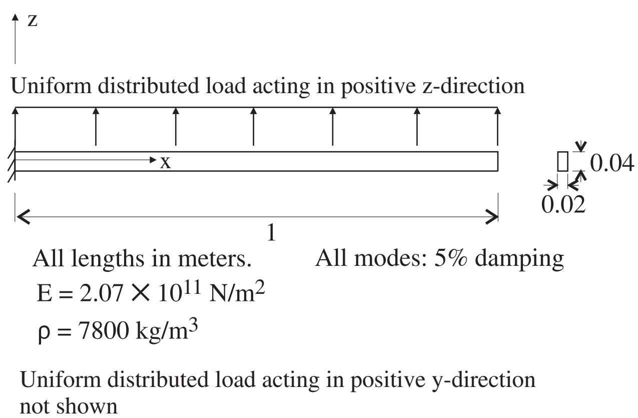
Problem description

In this problem, we subject the beam structure shown below to harmonic and random loads.



The details of the loadings are given below when they are defined.

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

- Defining out-of-plane line loads
- Setting up a modal participation factor analysis with applied loads
- Defining and using harmonic loads
- Defining and using random loads

We assume that you have worked through problem 1 to 26, 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.

Defining the model

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

Problem heading: Choose Control→Heading, enter "Problem 27: Beam subjected to harmonic and random loads" and click OK.

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

Point #	X1	X2	X3
1			
2	1		
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.

*Loads:*The loads in the y and z directions will be considered independent. We will have two load steps. In load step 1, the y-direction loads will be active; in load step 2, the z-direction loads will be active. Time function 1 will control the y loads and time function 2 will control the z loads.

Choose Control→Time Step, set the Number of Steps to 2 in the first row of the table and click OK.

Choose Control→Time Function and edit time function 1 to be

Time	Value
0	0
1	1
2	0

Then define time function 2 as

Time	Value
0	0
1	0
2	1

Click OK to close the time function dialog box.

Now we define the load applications. Click the Apply Load icon . Set the Load Type to Distributed Line Load and click the Define... button to the right of the Load Number field. In the Define Distributed Line Load dialog box, add Line Load 1, set the Magnitude [Force/Length] to -1 and click OK. In the first row of the table in the Apply Load dialog box, set the Line # to 1, the Auxiliary Point to 3, the Load Plane to "Perpendicular to Plane" and the Time Function to 1. In the second row of the table, set the Line # to 1, the Auxiliary Point to 3, the Load Plane to In-Plane and the Time Function to 2. Click OK to close the Apply Load dialog box.

Cross-section and material: Click the Cross-Sections icon , add cross-section 1, set the Width to 0.04, the Height to 0.02 and click OK. (Note: the element s-direction will lie in the x-z plane, so the Width (in the s-direction) is the larger cross-section dimension.) 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.

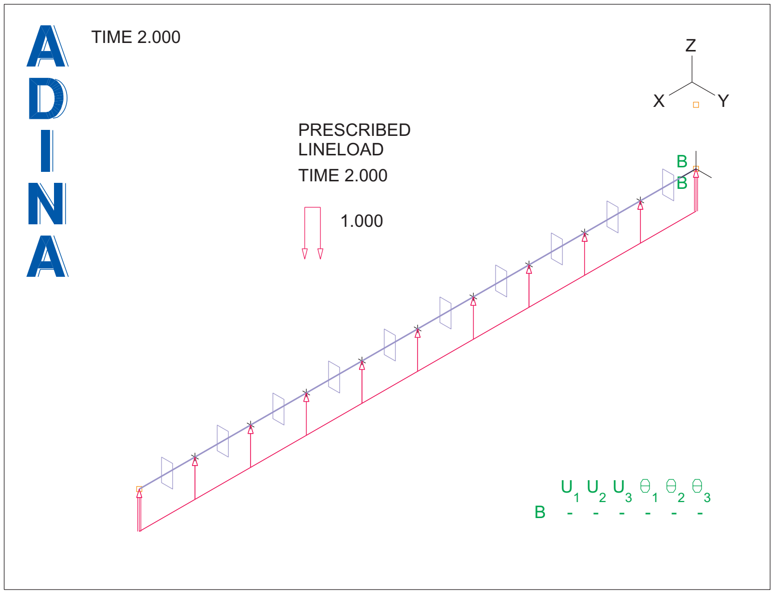
Finite elements: Click the Element Groups icon , add group 1, set the Type to Beam and click OK.

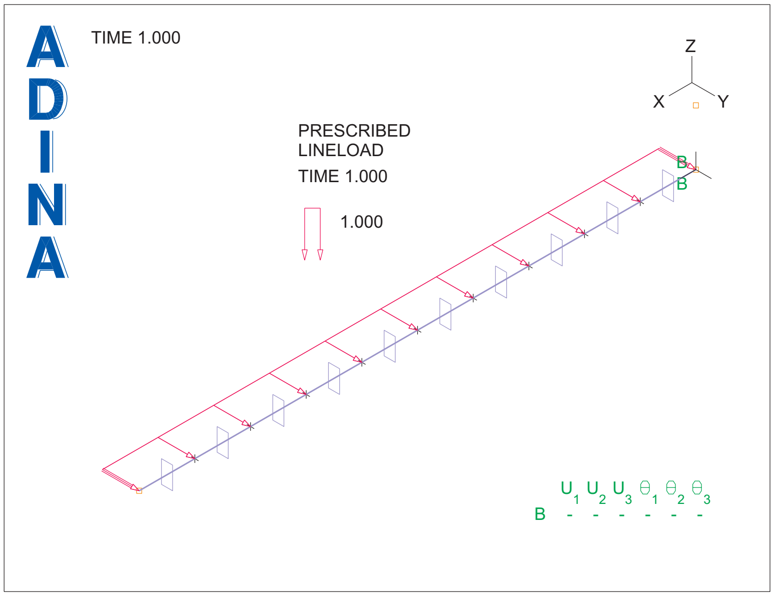
Click the Subdivide Lines icon , set the Number of Subdivisions to 10 and click OK.

Click the Mesh Lines icon , set the Auxiliary Point to 3, enter 1 in the first row of the table and click OK.

Click the Iso View 1 icon , Boundary Plot icon and Load Plot icon . Then click the Modify Mesh Plot icon , click the Element Depiction... button, click the Display Beam Cross-Section field, then click OK twice to close both dialog boxes. The graphics window should look something like the top figure on the next page. (Note: we use the Iso View 1 icon because otherwise the mesh is plotted in the x-z plane.)

Before continuing with the model definition, let's plot the loads for load step 1. Click the Previous Solution icon . The graphics window should look something like the bottom figure on the next page.


ADINA
TIME 2.000
Z
X
Y
PRESCRIBED
LINELOAD
TIME 2.000
1.000
B
B
U1 U2 U3 θ1 θ2 θ3
B - - - - - -


ADINA
TIME 1.000
Z
X
Y
PRESCRIBED
LINELOAD
TIME 1.000
1.000
B
B
U1 U2 U3 θ1 θ2 θ3
B - - - - - -

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 10 and click OK to close the dialog box. Set the "Number of Modes to Use" to 10, set the "Type of Excitation Load" to "Applied Load" and click OK to close the dialog box.

Generating the data file, running ADINA Structures, loading the porthole file

Click the Save icon and save the database to file `prob27`. Click the Data File/Solution icon , set the file name to `prob27`, 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 `prob27`.

The AUI displays the warning messages

> Node displacements not found for node 1.
> Displacement messages suppressed for 11 nodes.
> Plotted displacements set to zero for 12 nodes.

These messages appear because in this type of analysis, ADINA Structures computes only modal participation factors for solution times 1.0 and 2.0. Click OK to close the warning message box.

Listing the natural frequencies and modal participation factors

Choose List→Info→Response and verify that 2 sets of applied loading modal participation factors are loaded from times 1.0 to 2.0. The modal participation factors for time 1.0 (load step 1) are calculated from the loads at time 1.0 (which are the y loads) and the modal participation factors for time 2.0 (load step 2) are calculated from the loads at time 2.0 (which are the z loads).

Also notice that no displacements or other solution data is calculated by ADINA Structures. However ADINA Structures does calculate mode shapes, modal reactions and modal stresses. Click Close to close the dialog box.

To list the modal data, choose List→Value List→Zone, set the Response Range to DEFAULT_MODE-SHAPE, set Variable 1 to (Frequency/Mode:FREQUENCY) and click Apply. The first few frequencies should be 1.66424E+01 (Hz), 3.32770E+01. Click Close to close the dialog box.

Defining the modal damping ratios

Choose Definitions→Spectrum Definitions→Damping Table and add damping table DT1. Now click the … button to the right of the Curve Name field. In the Define Frequency Curve dialog box, add frequency curve DT1, enter 0, 5 and 10000, 5 in the first two rows of the table and click OK. ("5" corresponds to 5 % damping.) In the Define Damping Table dialog box, set the Curve Name to DT1 if necessary and click OK to close the dialog box.

Harmonic analysis

If you are not interested in harmonic analysis, you can skip to the "Random analysis" section below.

For the theory used in harmonic analysis, see the ADINA Structures Theory and Modeling Guide, Section 9.3.

We will proceed as follows:

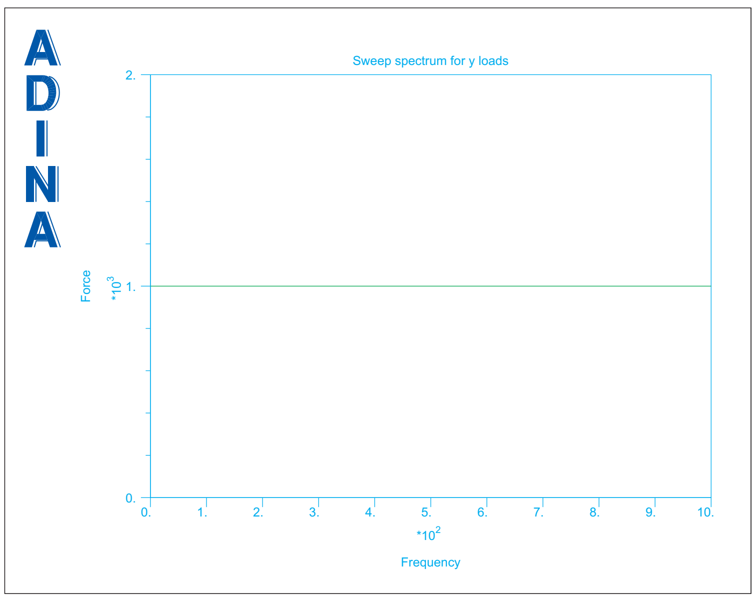
1) Analyze the beam assuming that only the y loads are applied
2) Analyze the beam assuming that both the y and z loads are applied

Analysis assuming that only the y loads are applied

Load magnitude specification: We assume that the time variation of the y loads is given by $w_y = 1000\sin(\omega t)$ where $\omega = 2\pi f$ and *f* is the frequency of the loads (in Hz). Notice that the magnitude factor 1000 is independent of the loading frequency *f* (but in general, the magnitude factor can be a function of the loading frequency). We also assume that this time variation is valid for $0 \le f \le 1000$ Hz.

To specify this information, choose Definitions→Spectrum Definitions→Sweep Spectrum, add sweep spectrum name SWEEP_Y, and click the … button to the right of the Curve Name field. In the Define Frequency Curve dialog box, add frequency curve SWEEP_Y, enter 0, 1000 and 1000, 1000 in the first two rows of the table and click OK. In the Define Sweep Spectrum dialog box, set the Curve Name to SWEEP_Y, set the "Axes Type (Frequency-Value)" to "Linear-Linear", set the Spectrum Title to "Sweep spectrum for y loads" and click Save.

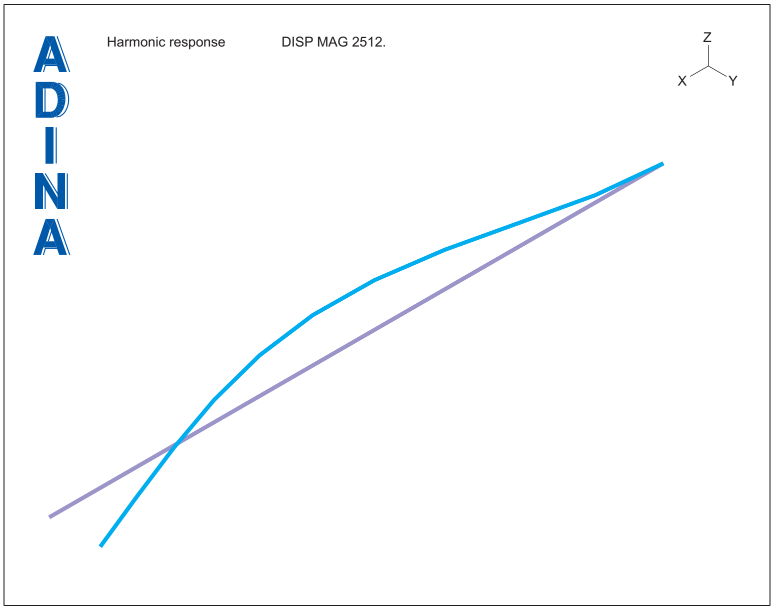
To graph the sweep spectrum, click the Clear icon, then click the Graph… button in the Define Sweep Spectrum dialog box. In the Display Sweep Spectrum dialog box, make sure that the Sweep Spectrum is set to SWEEP_Y and click OK twice to close both dialog boxes. The graphics window should look something like this:

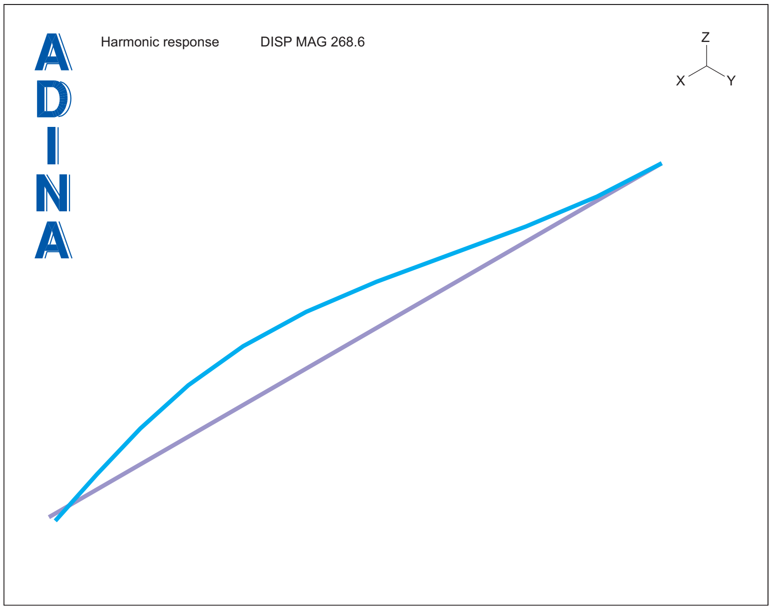


Solution for frequency of 150 Hz: Let's plot the deformations assuming a loading frequency of 150 Hz. Choose Definitions→Response, make sure that the Response Name is DEFAULT and set the Type to Harmonic. Set the Method to "Amplitude at Specified Angle", set the Loading Frequency to 150, set the Damping Table to DT1 and, in the table, enter 1, SWEEP_Y. Then click OK.

When you click the Clear icon and the Iso View 1 icon, then the Show Original Mesh icon and the Scale Displacements icon, the graphics window should look something like the top figure on the next page.

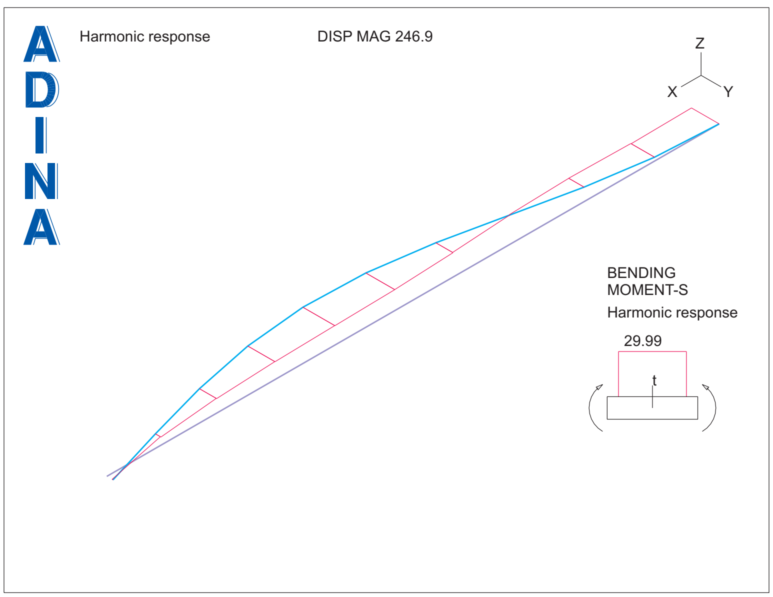
This solution is the solution when *t* in the loading equation above is 0, 1/150, 2/150, etc. We can also obtain the solution for other times by changing the specified angle in the harmonic response definition. For example, choose Definitions→Response, make sure that the Response Name is DEFAULT, set the Angle (OMEGAT) to 90 and click OK. When you click the Clear icon and the Iso View 1 icon, then the Show Original Mesh icon and the Scale Displacements icon, the graphics window should look something like the bottom figure on the next page.


ADINA
Harmonic response
DISP MAG 2512.
Z
X
Y

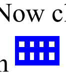

ADINA
Harmonic response
DISP MAG 268.6
Z
X
Y

This is the solution when ωt in the loading equation is 90 degrees, and therefore when t in the loading equation above is $(90/360)\times(1/150)=1.667\times10^{-3}$ seconds. You can try other angles to see the structural responses for other times.

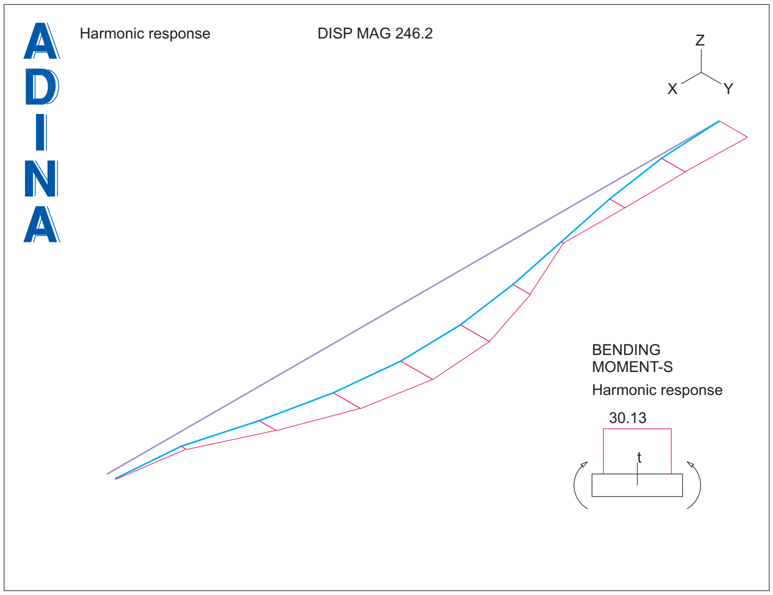
We can also plot the results, such as the bending moments. Choose Display→Element Line Plot→Create, set the Element Line Quantity to BENDING_MOMENT-S and click OK. The graphics window should look something like this:



We can also have the AUI choose ωt for each bending moment in each element so that the bending moment in each element is maximum (of course, then ωt will be different for each element). Similarly, we can have the AUI choose ωt for each displacement at each node so that the displacement is maximum (of course, then ωt will be different for each node).

For example, choose Definitions→Response, make sure that the Response Name is DEFAULT, set the Method to Maximum Amplitude and click OK. Now click the Clear icon  and the Iso View 1 icon , then the Show Original Mesh icon  and the Scale Displacements icon . The AUI chooses ωt for each displacement at each node so that the displacement at each node is maximum. Now choose Display→Element Line Plot→Create, set the Element Line Quantity to BENDING_MOMENT-S and click OK. The AUI chooses ωt for each bending moment in each element so that the bending moment in each element is maximum.

The graphics window should look something like this:



Note that this plot is similar to an envelope plot, where the envelope is taken over all solution times.

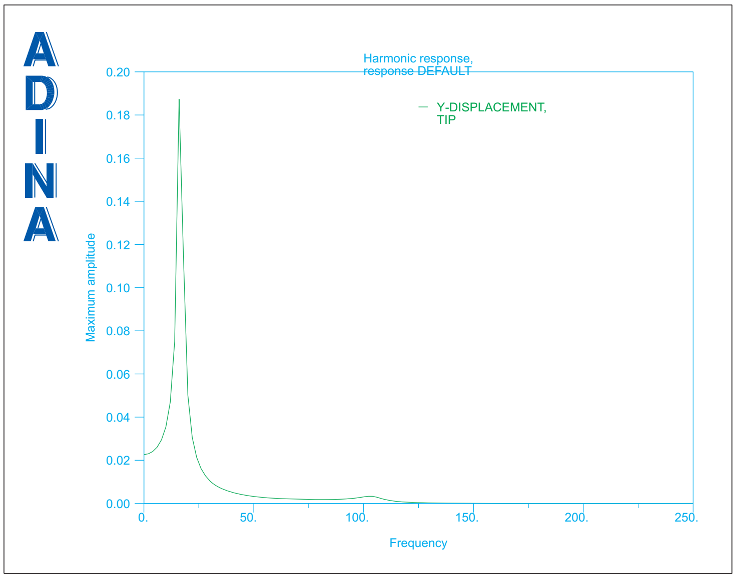
Solution for loading frequencies from 0 to 250 Hz: Of course, we could examine the results for any loading frequency using the instructions given above. But it is convenient to focus attention on one result (the tip displacement, for example) and then sweep the loading frequency over a range of frequencies.

The node at the tip is node 11. Choose Defintions→Model Point→Node, add name TIP, define it as node 11 and click OK.

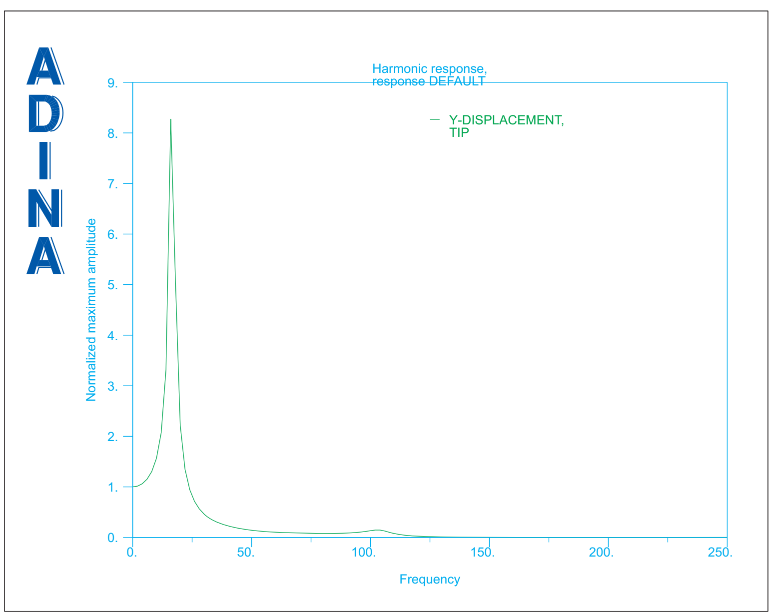
Now click the Clear icon 

and choose Graph→Harmonic Analysis. Set the Variable to (Displacement:Y-DISPLACEMENT) and make sure that the Model Point is TIP. Set the Frequency Spacing to Linear and the Number of Frequencies to 126. In the Frequency Range of the Harmonic Response box, set the Min. Frequency to 0 and the Max. Frequency to 250, then click OK. The graphics window should look something like the top figure on the next page.

This plot shows that the tip displacement is large for a loading frequency of 16 Hz. This is not surprising since the first natural frequency of the beam is around 16 Hz.



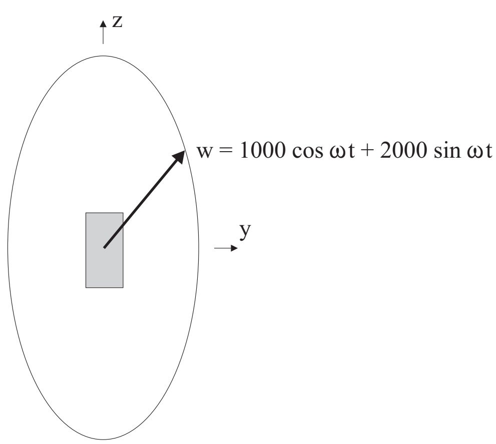
In this plot, the units of amplitude are meters. We can also plot the amplitude scaled to the quasi-static amplitude (which is the amplitude for very low loading frequencies). To make this plot, choose Definitions→Response, check the “Normalized by Quasi-Static Response” button and click OK. Then follow the instructions given above for the previous graph. The graphics window should look something like this:



Here it is easy to see that the maximum displacement is about 8 times the static displacement.

Analysis assuming that both the y and z loads are applied

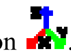
Load magnitude specification: We assume that the time variation of the y loads is $w_y = 1000\cos(\omega t)$ and that the time variation of the z loads is $w_z = 2000\sin(\omega t)$. The combination of these two loads can be interpreted as a load that traces an elliptical path around the beam, as shown:

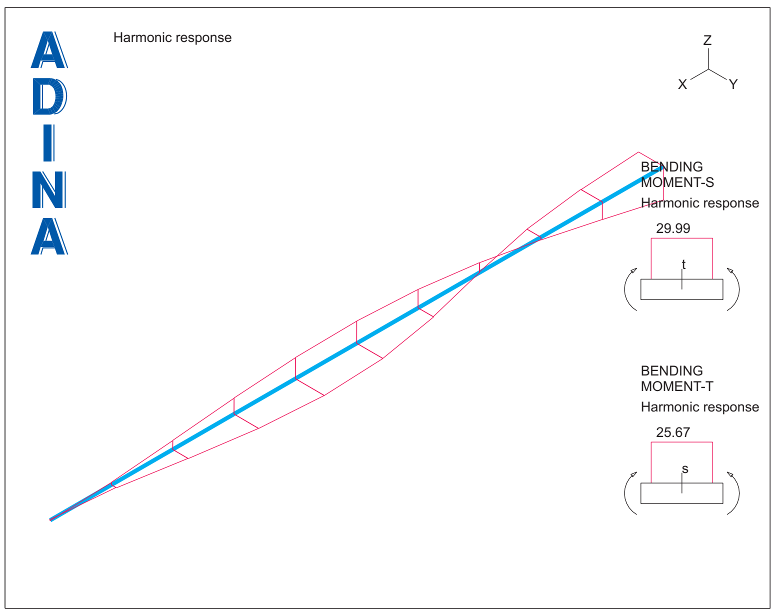


We need to define a sweep spectrum for the z load. Choose Definitions→ Spectrum Definitions→Sweep Spectrum, add sweep spectrum name SWEEP_Z, and click the … button to the right of the Curve Name field. In the Define Frequency Curve dialog box, add frequency curve SWEEP_Z, enter 0, 2000 and 1000, 2000 in the first two rows of the table and click OK. In the Define Sweep Spectrum dialog box, set the Curve Name to SWEEP_Z, set the “Axes Type (Frequency-Value)” to “Linear-Linear”, set the Spectrum Title to “Sweep spectrum for z loads” and click OK.

Solution for frequency of 150 Hz: Let’s plot the deformations assuming a frequency of 150 Hz. Choose Definitions→Response, make sure that the Response Name is DEFAULT and that the Type is Harmonic. Set the Method to “Amplitude at Specified Angle”, set the Angle (OMEGAT) to 0, the Loading Frequency to 150 and uncheck the “Normalized by Quasi-Static Response” button. Now, in the table, enter 1, SWEEP_Y, 1, -90 in the first row and 2, SWEEP_Z, 1, 0 in the second row. Click OK to close the dialog box.

(For row 1, we use the identity $\cos(\omega t) = \sin(\omega t - (-90^\circ))$ to determine the value of the Phase Angle.)

Now click the Clear icon  and the Iso View 1 icon . Choose Display→ Element Line Plot→Create, set the Element Line Quantity to BENDING_MOMENT-S, click Apply, set the Element Line Quantity to BENDING_MOMENT-T and click OK. The graphics window should look something like this:



This is the solution for ωt in the loading equation = 0 degrees.

Random analysis

If you are not interested in random analysis, you can exit the AUI now.

For the theory used in random analysis, see the ADINA Structures Theory and Modeling Guide, Section 9.4.

We will proceed as follows:

1) Analyze the beam assuming that only the y loads are applied
2) Analyze the beam assuming that both the y and z loads are applied

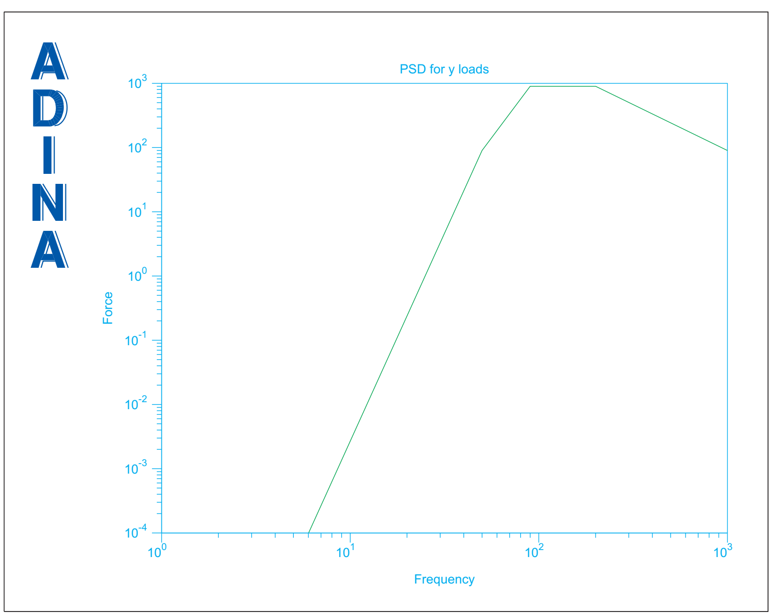
Analysis assuming that only the y loads are applied

Load magnitude specification: In random vibration analysis, we specify the power-spectral-density (PSD) of the load. For example, suppose that the PSD of the y load is given by the following table:

Frequency (Hz)	PSD $(N/m)^2/Hz$
1	9E-10
50	90
90	900
200	900
1000	90

To specify this information, choose Definitions→Spectrum Definitions→Random Spectrum, add random spectrum name PSD_Y, and click the … button to the right of the Curve Name field. In the Define Frequency Curve dialog box, add frequency curve PSD_Y, enter the above table and click OK. In the Define Random Spectrum dialog box, set the Curve Name to PSD_Y, set the Spectrum Title to "PSD for y loads" and click Save.

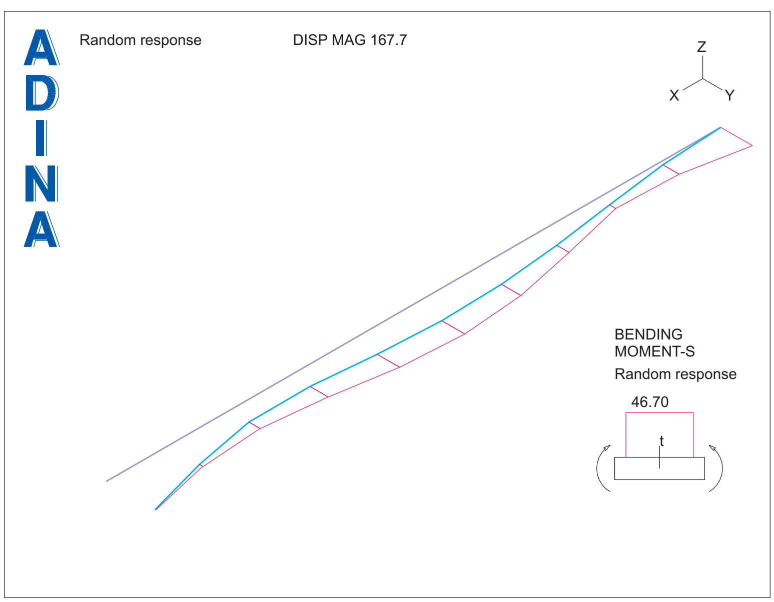
To graph the random spectrum, click the Clear icon CLEAR, then click the Graph… button in the Define Random Spectrum dialog box. In the Display Random Spectrum dialog box, make sure that the Random Spectrum is set to PSD_Y and click OK twice to close both dialog boxes. The graphics window should look something like this:



RMS solution: Let's list the rms (root-mean-square) values of the displacements. Choose Definitions→Response, make sure that the Response Name is DEFAULT and set the Type to Random. Then set the Damping Table to DT1 and, in the table, enter 1, PSD_Y. Then click OK to close the dialog box.

Now choose List→Value List→Zone, set the Response Option to Single Response, set Variable 1 to (Displacement:Y-DISPLACEMENT) and click Apply. The listing shows that the y-displacement at node 11 is 4.74897E-04. Actually, because this is a random vibration analysis, the y-displacement is interpreted as the RMS value of the y displacement, which is interpreted as the standard deviation of the y displacement (the mean value of the y displacement is zero). Therefore, the probability that the y displacement exceeds 4.74897E-04 m is about 32%. Click Close to close the dialog box.

Let's plot the rms displacements and the bending moments. Click the Clear icon and the Iso View 1 icon, then the Show Original Mesh icon and the Scale Displacements icon. Now choose Display→Element Line Plot→Create, set the Element Line Quantity to BENDING_MOMENT-S and click OK. The graphics window should look something like this:

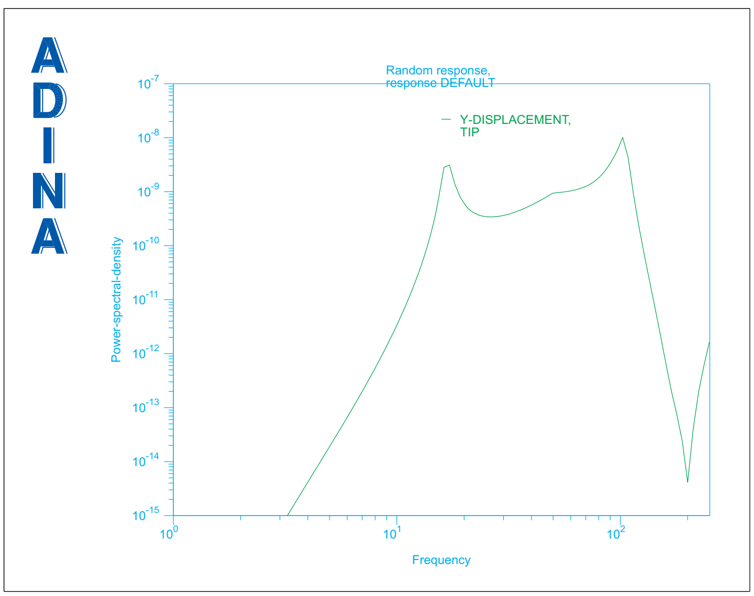


So the probability that the bending moment exceeds 46.7 (N-m) is about 32%.

PSD of the solution: We can plot the PSD of the tip displacements.

The node at the tip is node 11. If you have not already done so in the harmonic analysis above, choose Defintions→Model Point→Node, add name TIP, define it as node 11 and click OK.

Now click the Clear icon  and choose Graph→Random Analysis. Set the Variable to (Displacement:Y-DISPLACEMENT) and make sure that the Model Point is TIP. In the Frequency Spacing box, set the Number of Frequencies to 100. In the Frequency Range of the Random Response box, set the Min. Frequency to 1 and the Max. Frequency to 250, then click OK. The graphics window should look something like this:



Analysis assuming that both the y and z loads are applied

Load magnitude specification: We assume that the PSD of the y loads is the same as was used above, and that the PSD of the z loads is

Frequency (Hz)	PSD $(N/m)^2/Hz$
1	2E-10
50	100
90	200
200	200
1000	20

To specify this information, choose Definitions→Spectrum Definitions→Random Spectrum, add random spectrum name PSD_Z, and click the … button to the right of the Curve Name field. In the Define Frequency Curve dialog box, add frequency curve PSD_Z, enter the above table and click OK. In the Define Random Spectrum dialog box, set the Curve Name to PSD_Z, set the Spectrum Title to "PSD for z loads" and click OK.

RMS solution: Let's list the rms values of the displacements. Choose Definitions→Response, make sure that the Response Name is DEFAULT and make sure that the Response Type is set to Random. In the table, make sure that the first row is 1, PSD_Y and in the second row of the table, enter 2, PSD_Z. Then click OK.

Now choose List→Value List→Zone, set the Response Option to Single Response, set Variable 1 to (Displacement:Y-DISPLACEMENT), Variable 2 to (Displacement: Z-DISPLACEMENT), and click Apply. The listing shows that the standard deviation of the y-displacement at node 11 is 4.74897E-04 m, and that the standard deviation of the z displacement at node 11 is 3.93917E-04 m. Click Close to close the dialog box.

Note: the AUI assumes that the loads are uncorrelated (the AUI neglects the cross-spectral densities of the loading combinations).

Also note, in this problem, the y displacements depend only on the y loads, and the z displacements depend only on the z loads. However, in general, each computed result depends on all of the applied loads.

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

This page intentionally left blank.