

Problem description

In this problem, we subject the beam structure of problem 1 to an impact load as shown.



All lengths in meters. Step load applied at time 0.0.
 $E = 2.07 \times 10^{11} \text{ N/m}^2$ $\rho = 7800 \text{ kg/m}^3$

The beam is initially undeformed and at rest.

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

- Setting up a transient dynamic analysis
- Using an auxiliary point to define the beam coordinate system
- Specifying a restart analysis
- Removing symbols from a graph

We assume that you have worked through problem 1 to 9, 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

As the model geometry is very similar to problem 1, we only briefly give the steps needed to define the model.

Problem heading: Choose Control→Heading, enter “Problem 10: Beam subjected to impact load” and click OK.

Problem 10: Beam subjected to impact load

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: Click the Apply Load icon , make sure that the Load Type is Force and click the Define... button to the right of the Load Number field. In the Define Concentrated Force dialog box, add force 1, set the Magnitude to 300, the Y Direction to -1 and click OK. In the first row of the table in the Apply Load dialog box, set the Point # to 2 and click OK to close the dialog box.

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 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 2 and click OK.

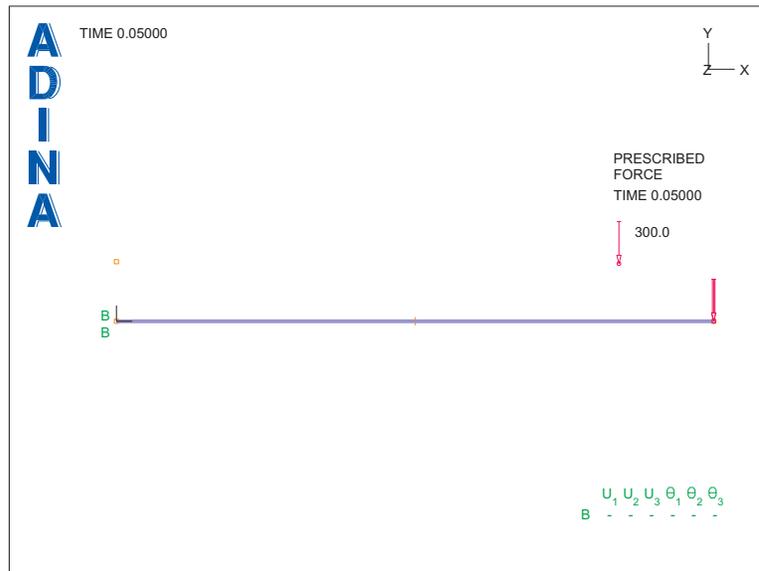
Click the Mesh Lines icon , set the Auxiliary Point to 3, enter 1 in the first row of the table and click OK. (The auxiliary point controls the orientation of the beam cross-section.)

Specifying the analysis options

Analysis type: Set the Analysis Type drop-down list to Dynamics-Implicit.

Time step: Choose Control→Time Step, enter 20, 0.0025 in the first row of the table and click OK.

When you click the Boundary Plot icon  and the Load 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 prob10a. Click the Data File/Solution icon , set the file name to prob10a, make sure that the Run Solution button is checked and click Save.

ADINA Structures runs for 20 steps. Notice that there are two lines printed for each step. These lines correspond to the two sub-steps used per solution step in the Bathe method of implicit time integration.

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 prob10a.

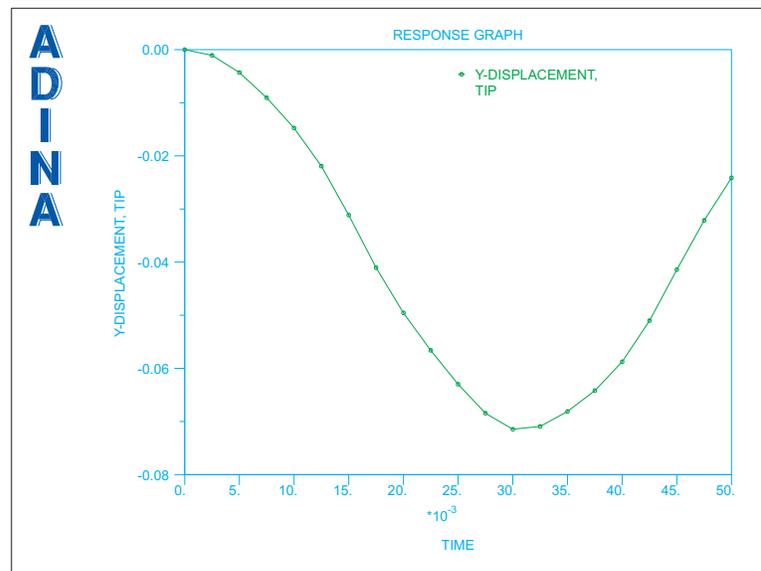
Graphing the time history response

We need to name the node point at the tip of the beam. Choose Definitions→Model Point→Node, add name TIP, set the Node Number to 3 and click OK.

Problem 10: Beam subjected to impact load

Click the Clear icon . Now choose Graph→Response Curve (Model Point), set the Y Variable to (Displacement:Y-DISPLACEMENT) and click OK.

The graphics window should look something like this:



Restart analysis

We will continue the dynamic analysis with the same time step size. We will use the restart feature in ADINA Structures for this purpose.

Set the Program Module drop-down list to ADINA Structures (you can discard all changes). Choose database file prob10a from the recent file list near the bottom of the File menu.

Choose Control→Solution Process, set the "Analysis Mode" to "Restart Run" and click OK. Now choose Control→Time Step, change the number of steps in the first row of the table to 180 and click OK.

Running ADINA Structures: Choose File→Save As and save the database to file prob10b.

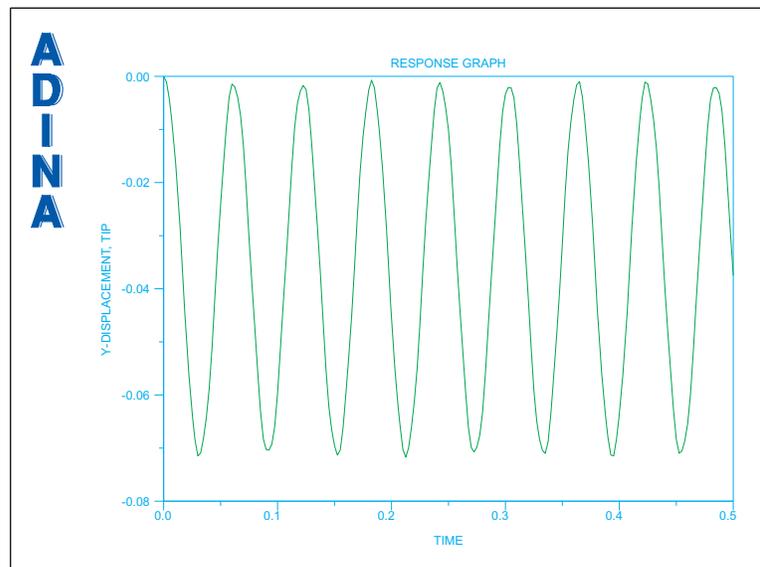
Click the Data File/Solution icon , set the file name to prob10b, make sure that the Run Solution button is checked and click Save. The AUI opens a window in which you specify the restart file from the first analysis. Enter restart file prob10a and click Copy.

Now we load both porthole files. Close all open dialog boxes, set the Program Module drop-down list to Post-Processing (you can discard all changes) and choose File→Open Porthole.

Choose file prob10a, set “Load” to “Entire Sequence of Files starting with Specified File” and click Open. Both porthole files are loaded.

Follow the instructions given above to graph the time history response.

This time, we will remove the symbols and legend from the plot. Choose Graph→Modify, set the Action to “Modify the Curve Depiction”, click the P button and highlight the graph curve. Then click the ... button to the right of the Curve Depiction field. In the Curve Depiction dialog box, uncheck the Display Curve Symbol button, click the Legend tab, and, in the Legend Attributes box, set the Type to No Legend. Then click OK twice to close both dialog boxes. The graphics window should look something like this:



Listing of the points in the graph: Choose Graph→List. The y displacement for the last solution step should be $-4.44120E-02$ (m). Click Close to close the dialog box.

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

Problem 10: Beam subjected to impact load

This page intentionally left blank.