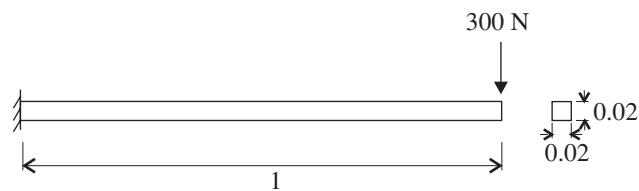


### Problem description

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.  
 $E = 2.07 \times 10^{11} \text{ N/m}^2$   $\rho = 7800 \text{ kg/m}^3$

This is the same model as problem 10, but in this problem we use mode superposition for the time integration.

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

- Setting up a mode superposition analysis
- Plotting mode shapes by changing the response type to mode-shape

We assume that you have worked through problem 1 to 10, 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 choose ADINA Structures from the Program Module drop-down list.


#### Defining the model

As the model geometry is the same as problem 10, we only briefly give the steps needed to define the model.


*Problem heading:* Choose Control→Heading, enter “Problem 11: Beam subjected to impact load - mode superposition” and click OK.


*Problem 11: Beam subjected to impact load – mode superposition*


---



*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 column 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 Define 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 table and click OK.

**Specifying the analysis options**

*Analysis type:* Choose Mode Superposition from the Analysis Type drop-down list 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. Then set the “Number of Modes to Use” to 2 and click OK to close the dialog box.



---

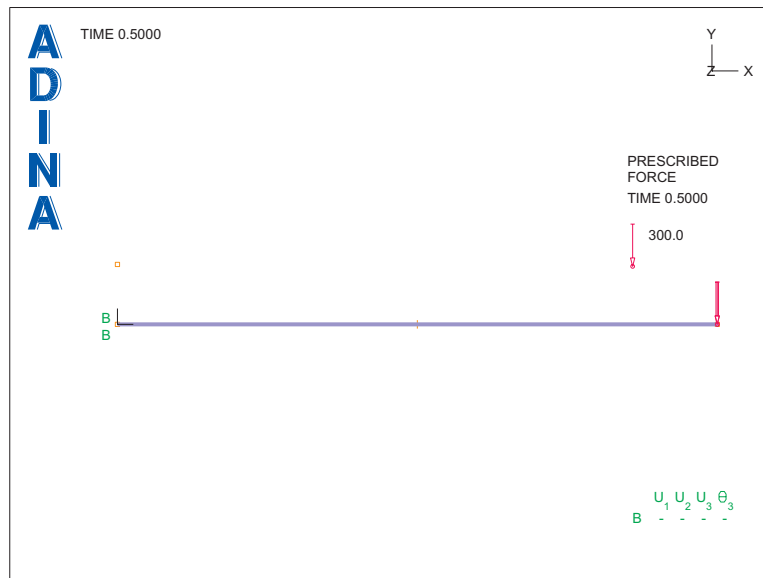
*Problem 11: Beam subjected to impact load – mode superposition*

---




*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.

*Time step:* Choose Control→Time Step, enter 200, 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, loading the porthole file

Click the Save icon  and save the database to file prob11. Click the Data File/Solution icon , set the file name to prob11, make sure that the Run Solution button is checked and click Save. When ADINA is finished, close all open dialog boxes, choose Post-Processing from the Program Module drop-down list (you can discard all changes), click the Open icon  and open porthole file prob11.

### Listing the natural frequencies

Choose List→Value List→Zone, set the Response Range to DEFAULT\_MODE-SHAPE, set Variable 1 to (Frequency/Mode:FREQUENCY) and click Apply. The frequencies should be 1.66504E+01 (Hz), 1.05131E+02. Click Close to close the dialog box.



*Problem 11: Beam subjected to impact load – mode superposition*


---

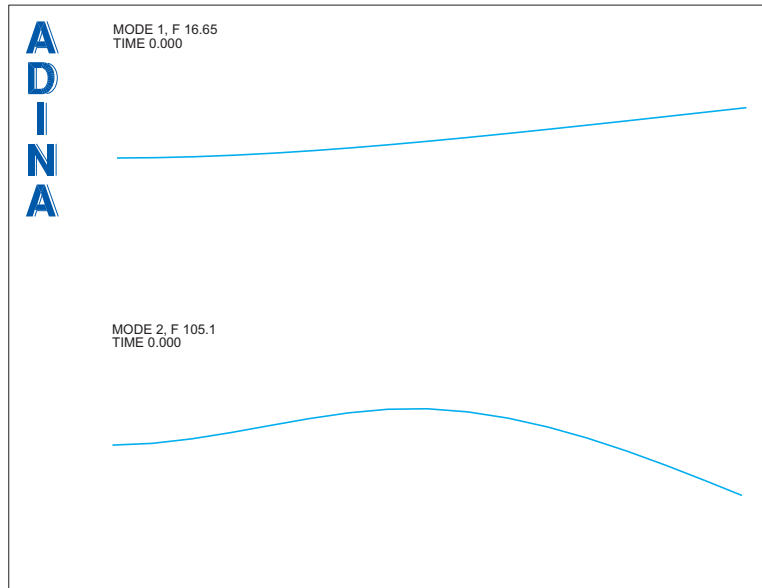
**Plotting the mode shapes**

To prepare for plotting mode shape 1, choose Definitions→Response, set the Type to Mode Shape and click OK.

In all plots, we would like to plot the neutral axes using curved lines. Choose Display→Geometry/Mesh Plot→Define Style, click the ... button to the right of the Element Depiction field, click the Advanced tab, set the '# Segments for Neutral Axis' to 8, then click OK twice to close both dialog boxes.

Click the Clear icon  and the Mesh Plot icon  and move the resulting mesh plot to the upper half of the graphics window.

To prepare for plotting mode shape 2, choose Definitions→Response, make sure that the Response Name is set to DEFAULT, set the Mode Shape Number to 2 and click OK. Then click the Mesh Plot icon  and move the resulting mesh plot to the lower half of the graphics window. Rearrange and delete the mesh plot annotations until the graphics window looks something like this:




**Graphing the time history response**

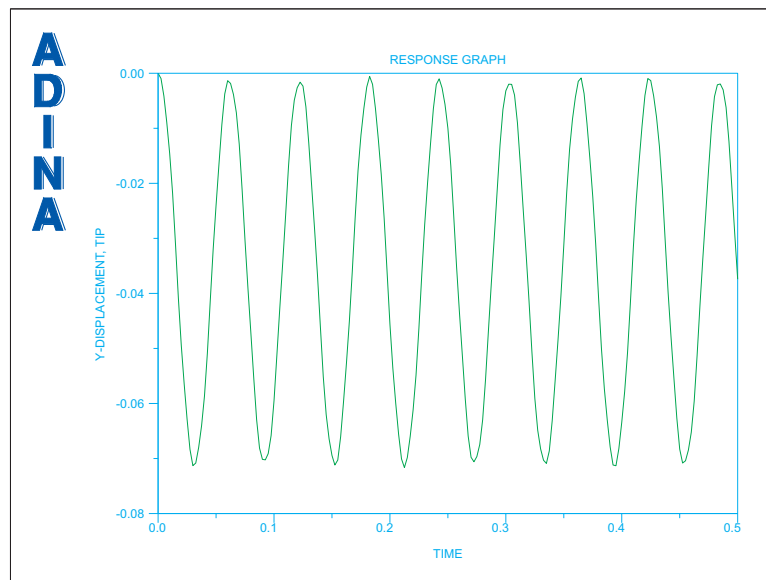
We need to name the node point at the tip of the beam. Choose Definitions→Model Point→Node, add name TIP, set the Node Number to 3 and click OK.

---

*Problem 11: Beam subjected to impact load – mode superposition*

For this graph, we know in advance that we do not want to see the curve symbols. Choose Graph→Define Style and 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.

Now click the Clear icon . 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:



The response is very similar to the response seen in problem 10.

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

*Problem 11: Beam subjected to impact load – mode superposition*

---

This page intentionally left blank.