

Problem description

In this problem, we analyze a simple beam structure. Each part of this lesson shows the solution of one of the analyses. In part 1, we demonstrate the following topics:

- Starting up/shutting down the AUI
- Defining geometry points
- Defining a geometry line
- Applying a boundary condition
- Defining a material
- Defining a cross-section
- Defining and applying a load
- Generating an element
- Generating an ADINA data file
- Saving an ADINA-IN database file
- Running ADINA
- Loading an ADINA porthole file
- Drawing a mesh plot, with boundary conditions plotted
- Drawing a load plot
- Listing the extreme deflections
- Plotting bending moment and shear force diagrams

In part 2, we demonstrate the following additional topics:

- Opening an ADINA-IN database file to access a previously defined model
- Deleting and redefining a load

In part 3, we demonstrate the following additional topics:

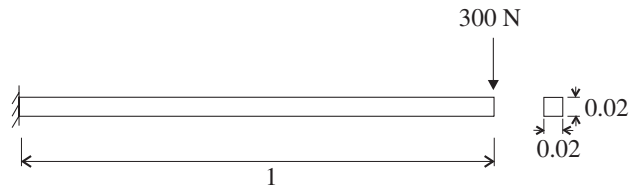
- Adding a boundary condition to an existing model
- Deleting elements from the model
- Clicking on a green column to fill in a table.
- Subdividing a line, generating elements on a subdivided line
- Magnifying the plotted displacements

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.

Part 1: Deflection due to a tip load



All lengths in meters
 $E = 2.07 \times 10^{11} \text{ N/m}^2$

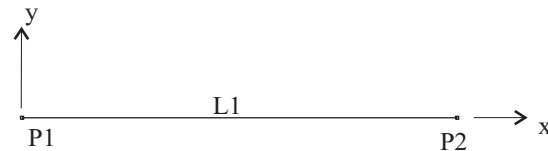
Invoking the AUI and choosing the finite element program

Invoke the AUI and choose ADINA Structures from the Program Module drop-down list.


(See the hints in the Hints chapter for more information.)

Defining model geometry

When setting up a model, it is frequently useful to make a diagram showing the key geometry used in the model definition, such as the following diagram:

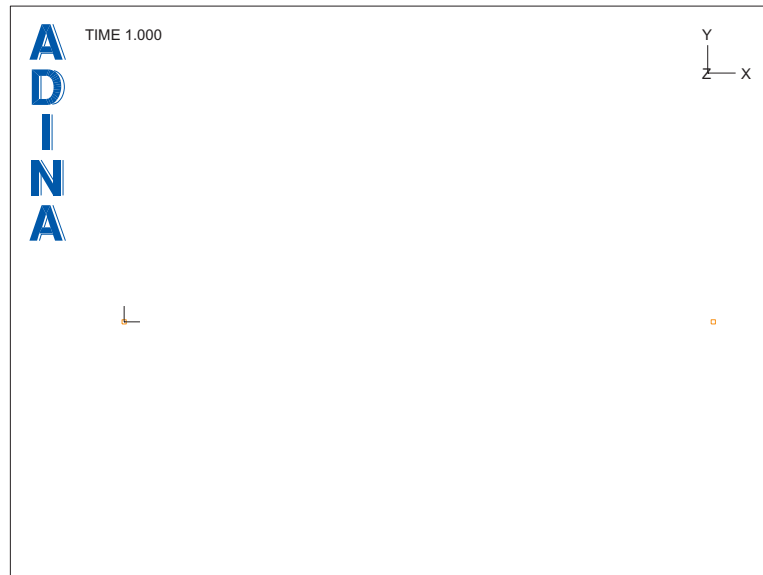



You may find it useful to refer to this diagram when working through this problem.

Click the Define Points icon  and enter the following information into the table:


Point #	X1	X2	X3
1			
2	1		

(Table cells that are left blank are treated as zeros.) Then click OK. The graphics window should look something like the figure on the next page.




Now click the Define Lines icon  and click the Add... button to add line number 1. Set the Point 1 field to 1, the Point 2 field to 2 and click OK. The graphics window should look something like the top figure on the next page.

Defining boundary conditions

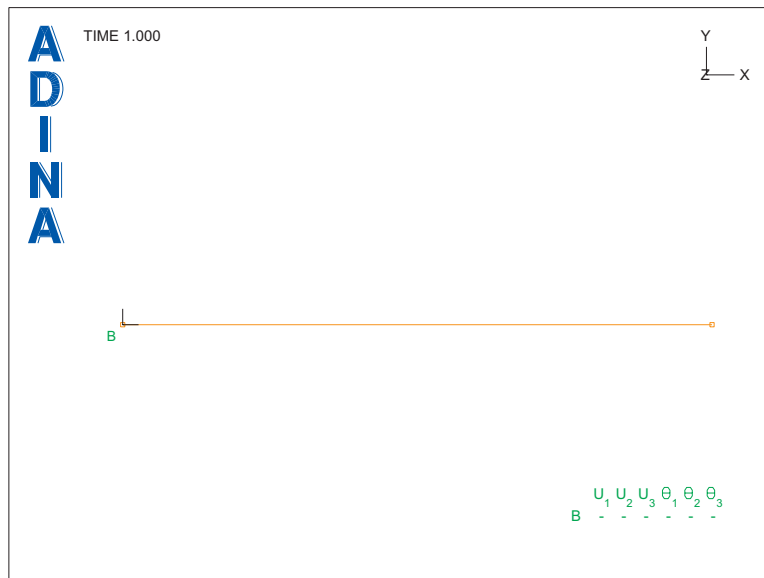
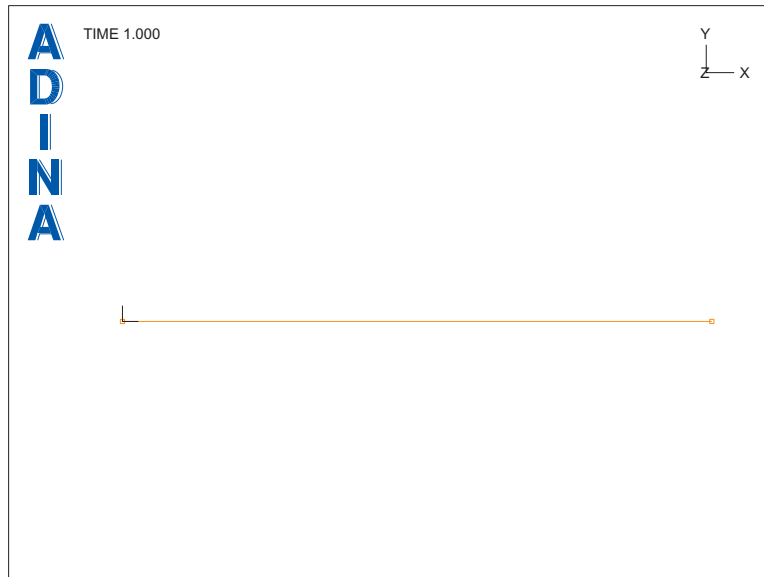
Click the Apply Fixity icon , enter 1 in the first row of the Point # column and click OK.


Click the Boundary Plot icon . The graphics window should look something like the bottom figure on the next page.

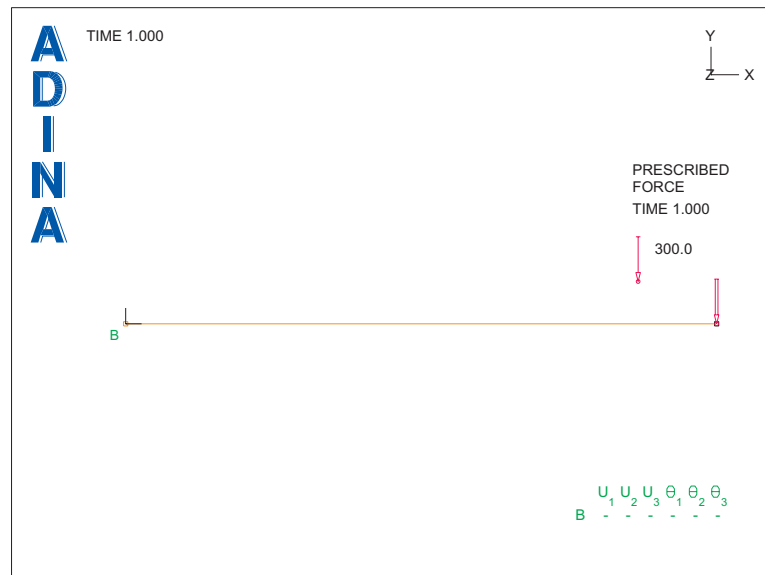
Defining loads

Click the Apply Load icon  to open the Apply Load dialog box. 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 Concentrated Force Number 1, set the Magnitude to 300, the Y Force 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 Apply Load dialog box.


Problem 1: Deflection of a beam




Click the Load Plot icon . The graphics window should look something like this:



Defining the cross-section

Click the Cross Sections icon . Then add section number 1, set the Width to 0.02, click the Square Section button and click OK.


Defining the material

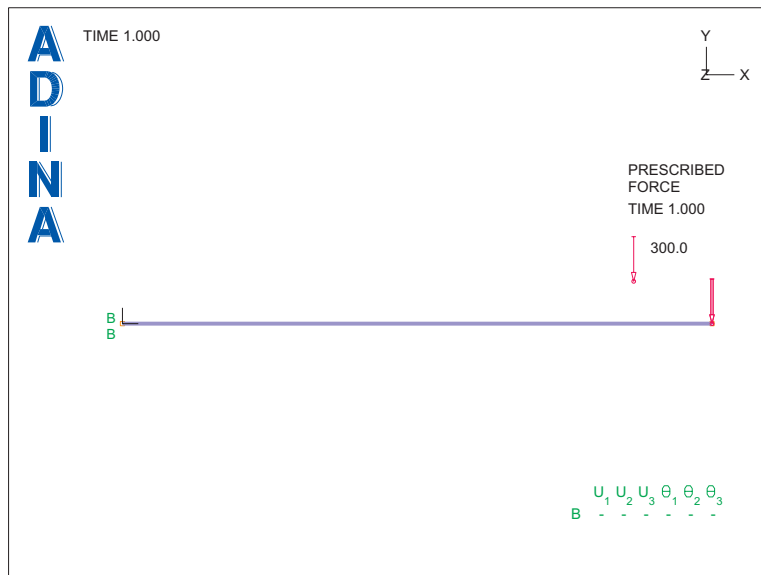
Click the Manage Materials icon  and click the Elastic Isotropic button. In the Define Isotropic Linear Elastic Material dialog box, add material 1, set the Young's Modulus to 2.07E11 and click OK. Click Close to close the Manage Material Definitions dialog box.

Defining the finite elements



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

Problem 1: Deflection of a beam


Element generation: Click the Mesh Lines icon , set the components of the Orientation Vector to (0, 1, 0), enter 1 in the first row of the Line # table and click OK. (The orientation vector is used to define the directions of the element local coordinate system; the element s-direction lies in the plane defined by the element and the orientation vector.) The graphics window should look something like this:





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

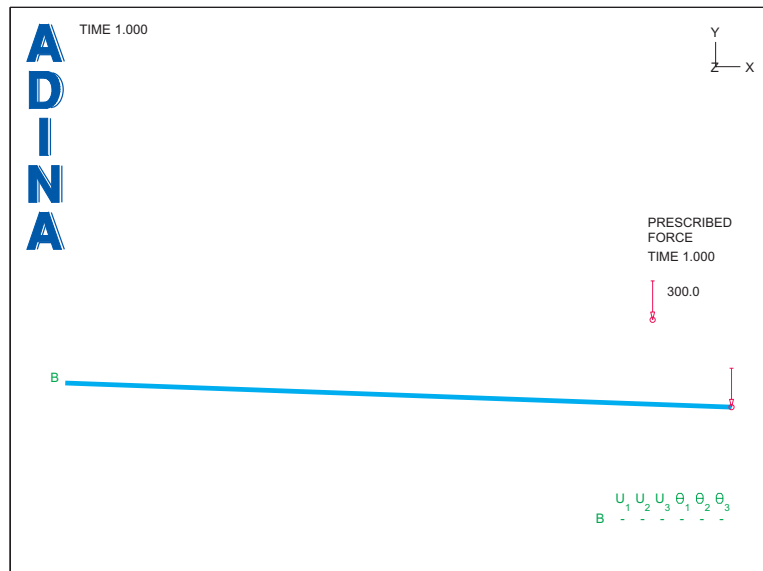
First click the Save icon , and save the database to file prob01 (the "File type" field should be "ADINA-IN Database Files (*.idb)"). To generate the ADINA data file and run ADINA, click the Data File/Solution icon , set the file name to prob01, make sure that the Run Solution button is checked and click Save.

When ADINA is finished, it displays the message "Solution successful, please check the results". Close all open dialog boxes.


Choose Post-Processing from the Program Module drop-down list, click Yes to discard all changes and continue, click the Open icon  and open porthole file prob01.



Displaying the deformed model


Click the Boundary Plot icon  to display the boundary conditions. Then click the Load Plot icon  to show the load. The graphics window should look something like this:



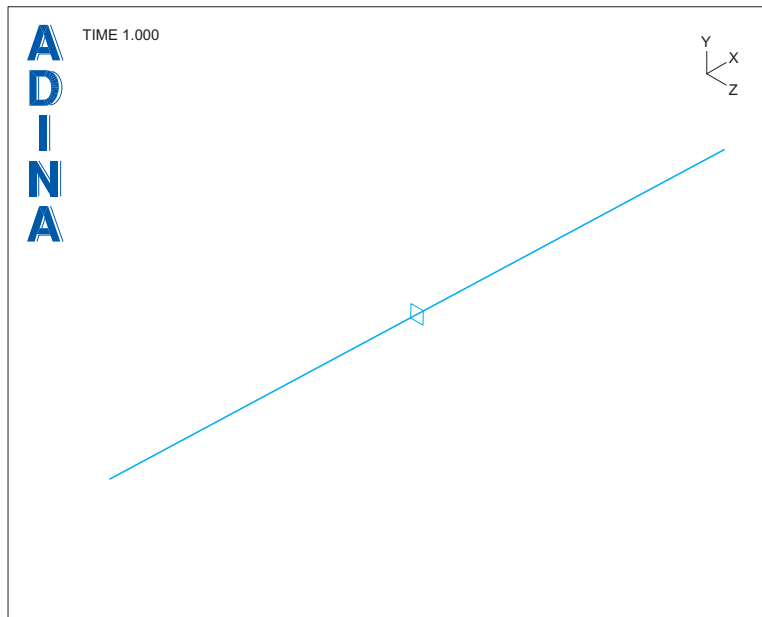
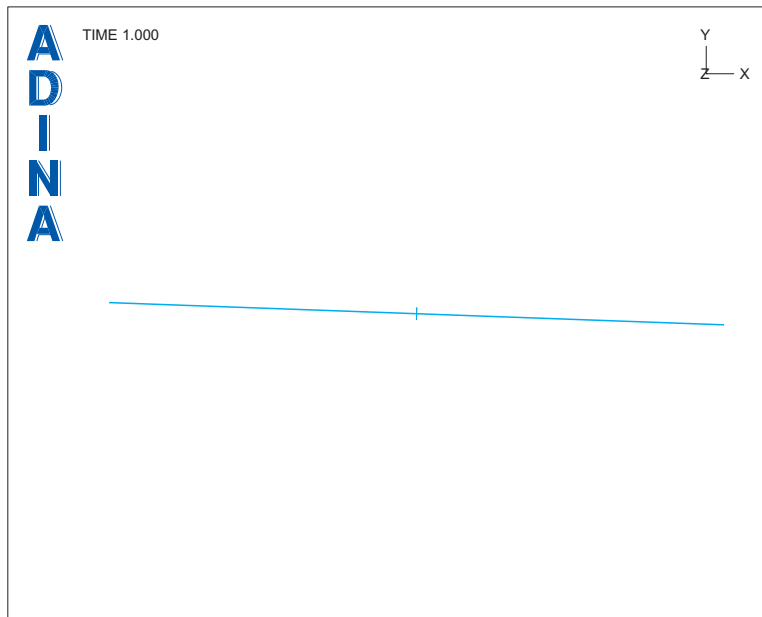
The beam cross-section can be plotted at the center of the element. Click the Modify Mesh

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.

Click the Iso View 2 icon  (located in the drop-down list next to the YZ View icon ) to plot the beam from a different angle. The graphics window should look something like the bottom figure on the next page.

Click the XY View icon  to return to the original view.


Problem 1: Deflection of a beam

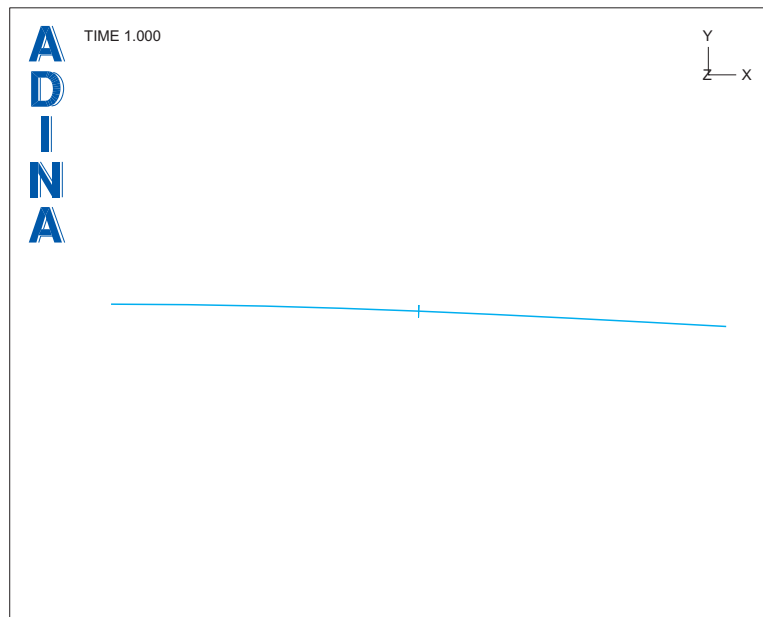


Listing the tip deflection

Choose List→Extreme Values→Zone. In the “Variables to List” box, first row, choose Y-DISPLACEMENT from the right-hand drop-down list (the list with the downwards pointing arrow). Then click Apply.

The AUI displays the value $-3.62319\text{E-}02$ for the y-displacement of node 2. Note that this is the deflection predicted by beam theory; a single beam element is sufficient in this case because the beam element contains a cubic displacement assumption and the beam theory solution requires a cubic displacement assumption. Click Close to close the dialog box.

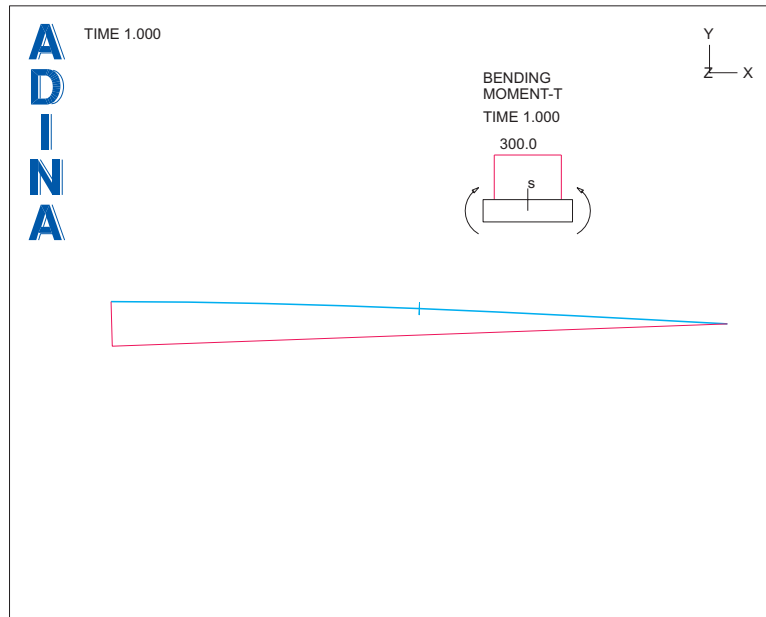
In order to see the cubic displacements of the neutral axis, click the Modify Mesh Plot icon , click the Element Depiction... button, click the Advanced tab, set the '# Segments for Neutral Axis' to 8, then click OK twice to close both dialog boxes. The graphics window should look something like this:



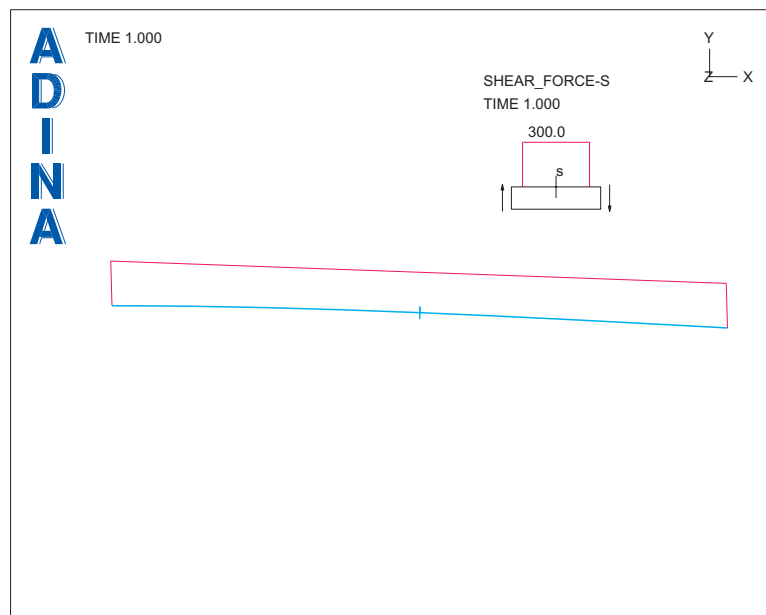
Plotting the bending moment diagram and the shear force diagram

Bending moment diagram: Choose Display→Element Line Plot→Create, set the Element Line Quantity to BENDING_MOMENT-T and click OK. The graphics window should look something like the top figure on the next page.

Problem 1: Deflection of a beam



Shear force diagram: Choose Display→Element Line Plot→Modify, set the Element Line Quantity to SHEAR_FORCE-S and click OK. The graphics window should look something like this:

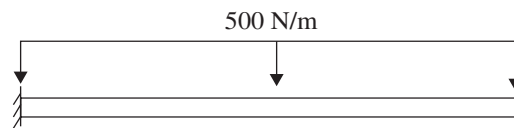


Exiting the AUI

Choose File→Exit and click Yes to discard all changes and exit ADINA-AUI.

Part 2: Deflection due to a distributed load


Now we apply a distributed load to the same cantilever as shown:




Invoke the AUI and choose ADINA Structures from the Program Module drop-down list. Then choose file prob01.idb from the recent file list near the bottom of the File menu (File→Recently Opened Files for UNIX versions).

Deleting and redefining the load


Deleting the load: In the Model Tree, click on the + next to the Loading text, right-click on the “1. Force 1 on Point 2” text, choose Delete, and click Yes to answer the prompt.


Click the Redraw icon  to update the graphics window. The graphics window should look something like the top figure on the next page.

Auxiliary point: In order to define the distributed load, we need to define an auxiliary point.

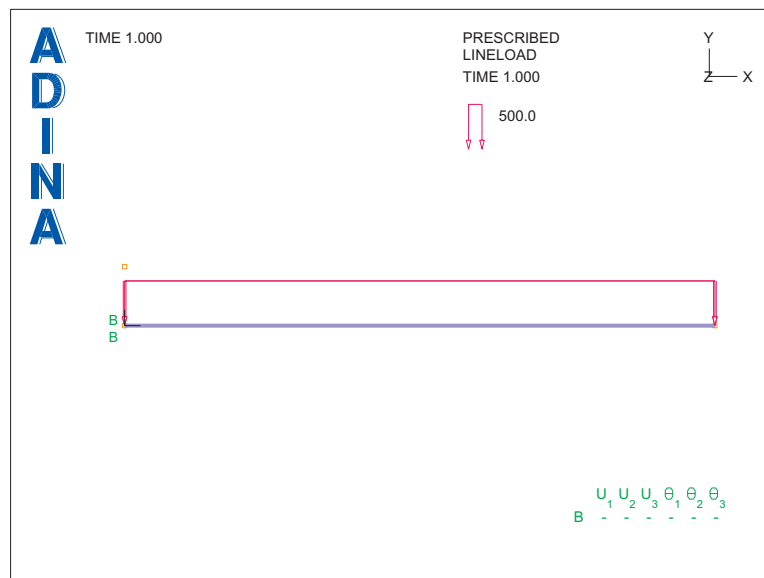
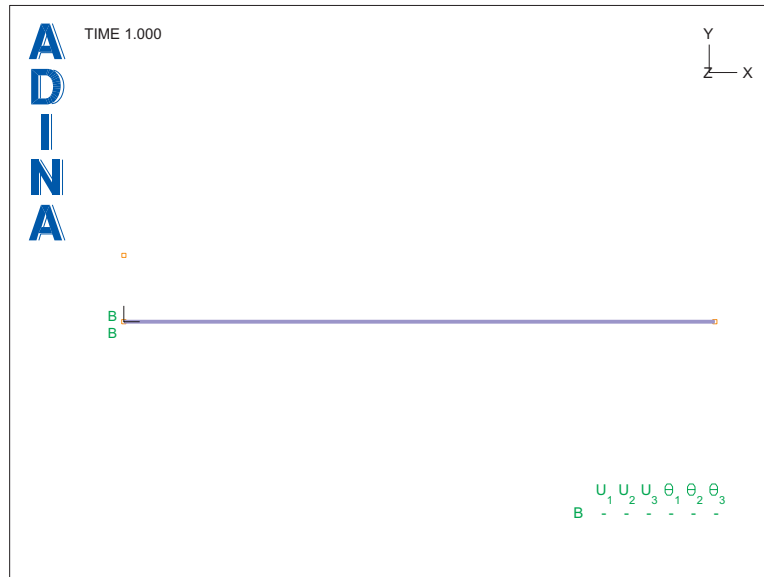
Click the Define Points icon , add point 3 and click OK:

Point #	X1	X2	X3
3		0.1	

Redefining the load: Click the Apply Load icon  and set the Load Type to Distributed Line Load. Click the Define... button to the right of the Load Number field. In the Define Distributed Line Load dialog box, add line load number 1, set the Magnitude [Force/Length] to 500 and click OK. In the first row of the table in the Apply Load dialog box, set the Line # to 1 and the Aux. Point to 3, then click OK.


When you click the Redraw icon , the graphics window should look something like the bottom figure on the next page.

Problem 1: Deflection of a beam



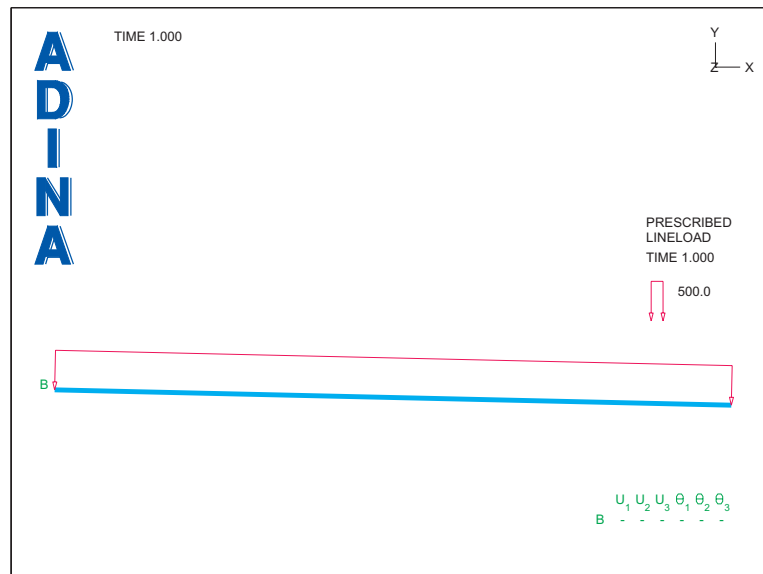
Generating the ADINA data file, running ADINA, postprocessing

To save the ADINA-IN database to a new file, choose File→Save As, set the file name to prob01a and click Save.

Click the Data File/Solution icon , set the file name to prob01a, make sure that the Run Solution button is checked and click Save.

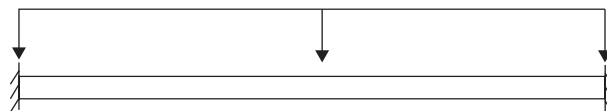
When ADINA is finished, close all open dialog boxes.

Postprocess this model using exactly the same steps as with the previous model (except that you will load porthole file prob01a this time). This time the tip displacement is $-2.26449E-02$. Again this is the same tip displacement as predicted by beam theory. The deformed mesh with loads and boundary conditions should look something like this:



Part 3: Deflection of a beam built-in at both ends


Now we change the problem to that of a beam built-in at both ends subjected to a distributed load as shown:



Problem 1: Deflection of a beam

Choose ADINA Structures from the Program Module drop-down list and click Yes to discard all changes and continue. Choose prob01a.idb from the recent file list near the bottom of the File menu.

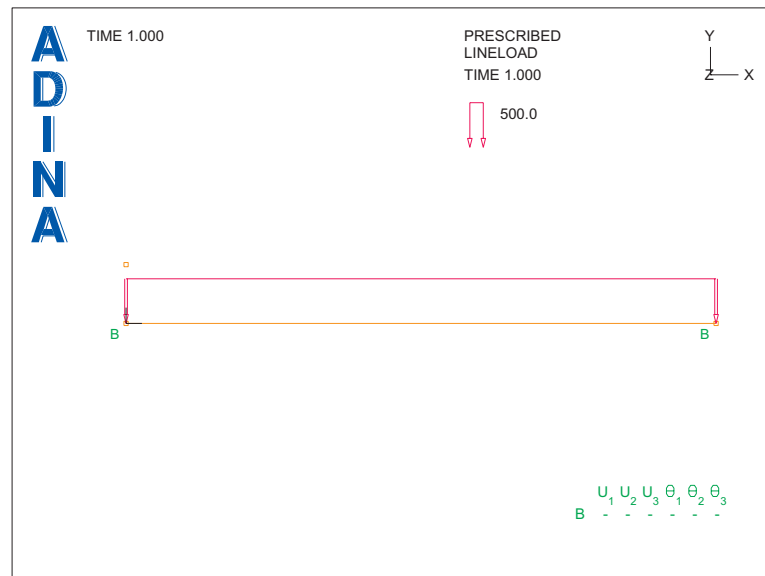
Adding a boundary condition, refining the mesh


We need to add a boundary condition to the right end of the beam. In the Model Tree, highlight and right-click the Fixity text, then choose Apply... . In the Apply Fixity dialog box, add point 2 to the second row of the table and click OK. Click the Redraw icon  to update the graphics.

Now we need to use more elements to solve the model.

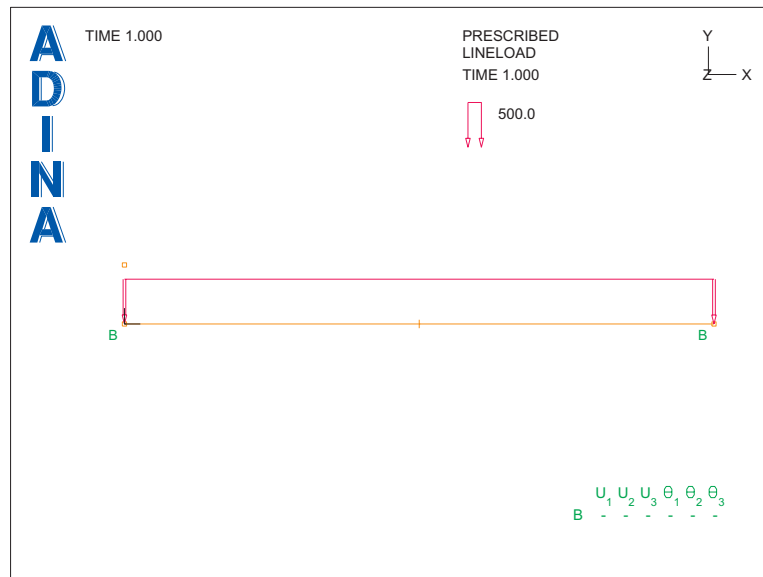
Deleting the existing element: Click the Delete Mesh icon , set the first row of the Line # table to 1 and click OK.


The graphics window should look something like this:



Specifying the mesh refinement: Click the Subdivide Lines icon , make sure that the “Method” is set to “Use Number of Divisions”, set the Number of Subdivisions to 2 and click OK.

The graphics window should look something like this. A short vertical line has been added to the middle of the geometry line, indicating how the line will be subdivided into elements.




Adding the elements: Click the Mesh Lines icon , set the components of the Orientation Vector to (0, 1, 0), enter 1 in the first row of the Line # table and click OK. (You can select the Line # using the mouse and the P button.) The graphics window should look something like the top figure on the next page.

Generating the ADINA data file, running ADINA, postprocessing

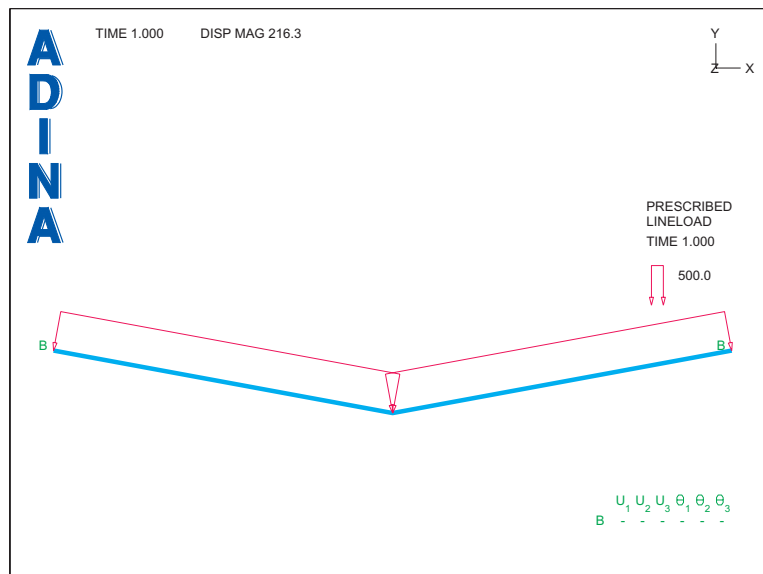
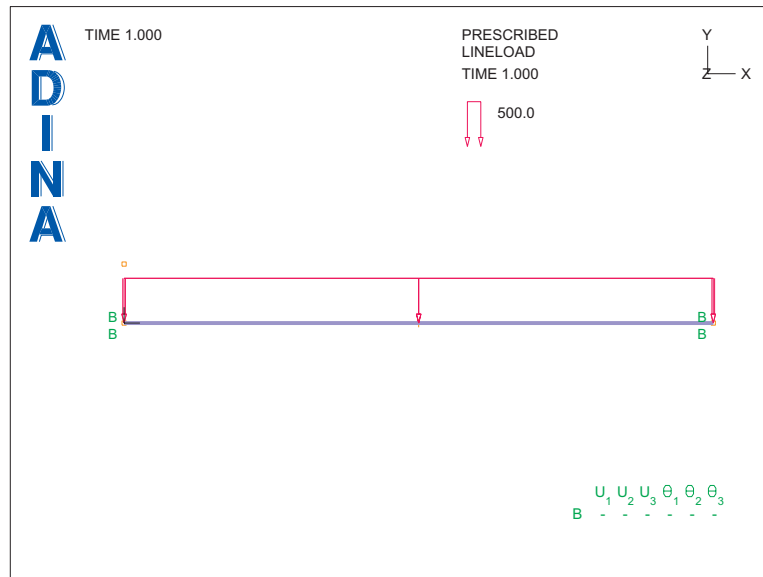
Save the database to file prob01b.idb, create ADINA data file prob01b.dat and run ADINA using file prob01b.dat using the same steps as with the previous models.

Postprocess this model using exactly the same steps as with the previous models (except that you will load porthole file prob01b this time. This time, when you plot the deformed mesh, you will not be able to see the deformed shape because the displacements are too small.

Because the displacements are so small, we need to magnify them so that they are visible on the plot. Click the Scale Displacements icon  10%. The graphics window should look something like the bottom figure on the next page. (The plot can be improved by increasing the number of segments used to plot the neutral axis, as we demonstrated above.)

When you make the listing, note that the maximum displacement is -4.71769E-4 at the center of the model. Again this is the result predicted by beam theory.

Problem 1: Deflection of a beam



Exiting the AUI

Choose File→Exit to exit the AUI. You can discard all changes.