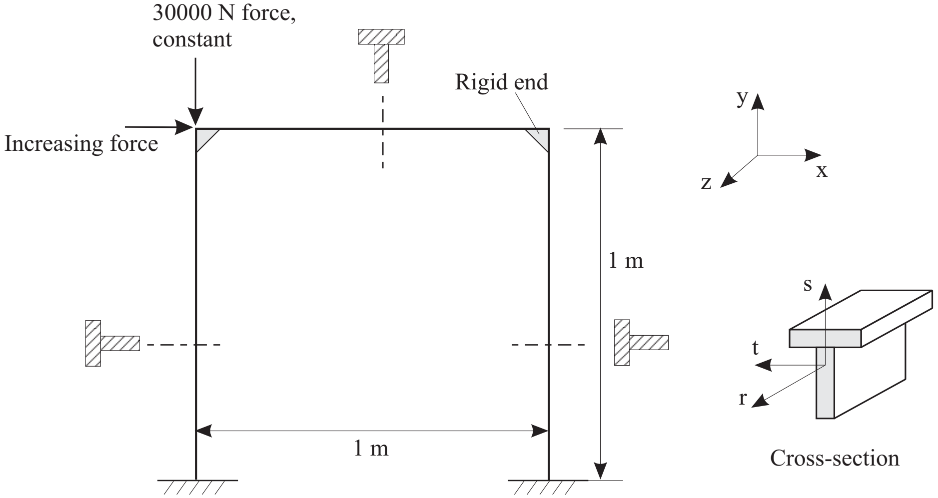
Problem 14

Pushover analysis of a frame

This page intentionally left blank.

Problem description

The frame shown in the figure below is to be analyzed to determine the horizontal force required to push the frame over.



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

- Defining multiple loads using different time functions
- Defining moment-curvature input
- Reading commands from a batch file
- Defining rigid end data
- Using the ATS (automatic time-stepping) method
- Defining an element section point
- Defining a resultant variable

We assume that you have worked through problems 1 to 13, or have equivalent experience with the AUI. Therefore we will not describe every user selection or button press.

Cross-section modeling

The beam cross-section is modeled using the moment-curvature material model in ADINA. With the moment-curvature material model, the cross-section and material are described by several curves, as shown in detail below.

The moment-curvature model accurately captures many aspects of the behavior of the cross-section. These aspects include:

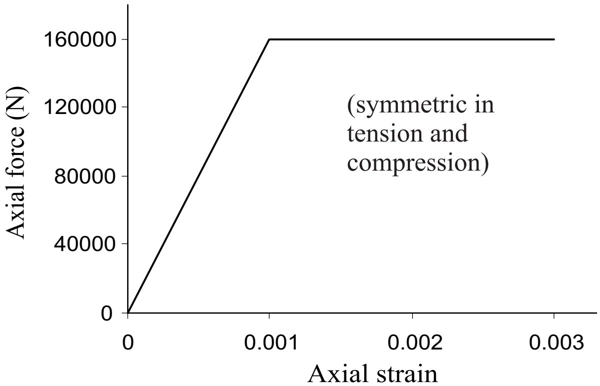
a) the dependence of the moment-curvature data on the axial force

b) the non-symmetry of the moment-curvature data (different behavior for positive curvature and negative curvature)

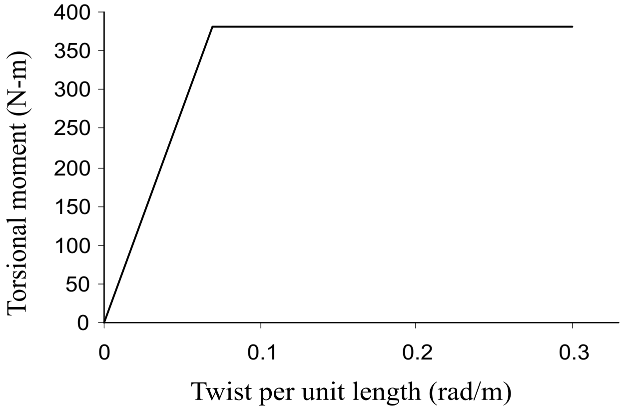
c) the dependence of rupture on the axial load

d) the multilinear nature of the yielding behavior

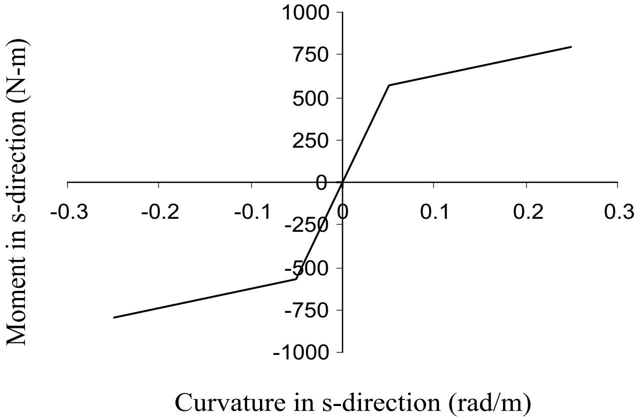
Axial force/axial strain data: Note, in ADINA, positive axial force is tensile, negative axial force is compressive.



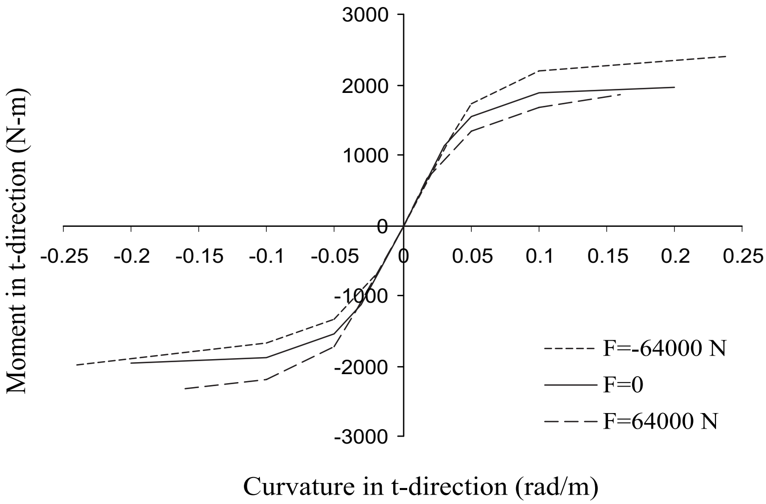
Torsional moment/twist data:



Bending moment/curvature data in the s direction:



Bending moment/curvature data in the t direction:



We will set up the model so that the *t*-direction for all beam elements coincides with the positive *z*-direction. Thus, for example, a bending moment in the *t*-direction is interpreted as a bending moment in the *z*-direction.

Also note that since the loads are in-plane, there are no torsional moments or bending in the *s*-direction. However, material input is still required for torsion and bending in the *s*-direction.

Rigid-end modeling

The rigid end feature of the beam element is used to model the assemblage of braces or struts that make up the corners of the frame.

P-Δ effect

ADINA includes the P-Δ effect when a large displacement kinematic formulation is used.

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.

The data for the bending behavior in the *t* direction is stored in a separate batch file prob14_1.in. You need to copy the file prob14_1.in from the folder samples\primer into a working directory or folder before beginning this analysis.

Invoking the AUI and choosing the finite element program

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

Defining model control data

Problem heading: Choose Control→Heading, enter the heading "Problem 14: Pushover analysis of a frame" and click OK.

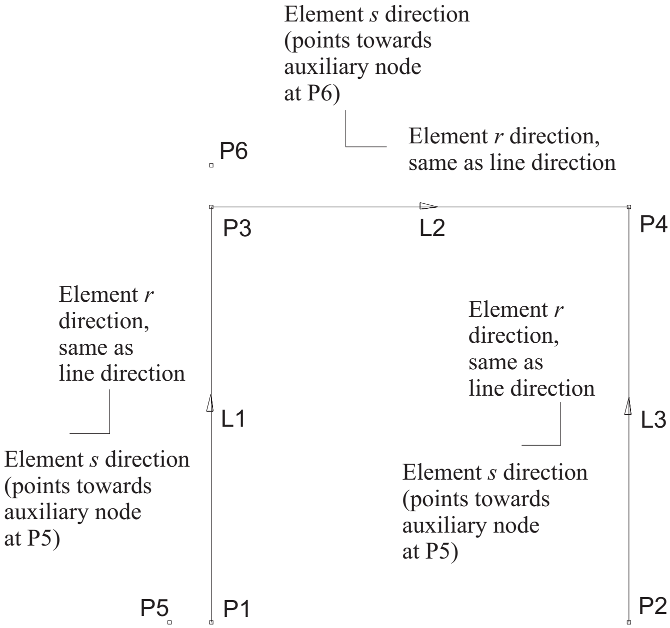
Automatic time-stepping: Click the Analysis Options icon  , set the Automatic Time Stepping Scheme to "Use Automatic Time Stepping (ATS)" and click the … button to the right of that field. In the Automatic Time-Stepping dialog box, set the "Maximum Subdivisions Allowed" to 20 and click OK twice to close both dialog boxes.

Equilibrium iteration tolerances: We will change the convergence tolerances used during equilibrium iterations. Choose Control→Solution Process, click the Iteration Tolerances... button and set the Convergence Criteria to Energy and Force. Set the Reference Force field and the Reference Moment field to 1.0 and click OK twice to close both dialog boxes.

*Kinematics:*We want to include P-Δ effects in the analysis. Choose Control→ Analysis Assumptions→Kinematics, set the "Displacements/Rotations" field to Large and click OK.

Defining model geometry

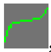
Here is a diagram showing the key geometry used in defining the model:



Geometry points: Click the Define Points icon , enter the following information into the table and click OK.

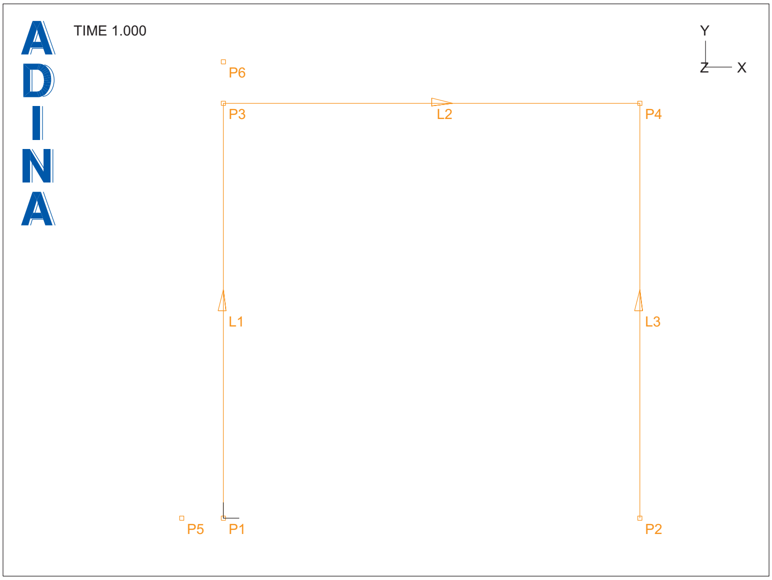
Point #	X1	X2
1	0.0	0.0
2	1.0	0.0
3	0.0	1.0
4	1.0	1.0
5	-0.1	0.0
6	0.0	1.1

Click the Point Labels icon  to display the point numbers.

Geometry lines: Click the Define Lines icon , add the following lines and click OK.

Line Number	Type	Point 1	Point 2
1	Straight	1	3
2	Straight	3	4
3	Straight	2	4

When you click the Line/Edge Labels icon  to display the line numbers, the graphics window should look something like this:



Defining subdivision data

We will use 10 elements for each line in the frame. Choose Meshing→Mesh Density→ Complete Model, set the "Subdivision Mode" to "Use Number of Divisions", set the "Number of Subdivisions" to 10 and click OK.

Defining boundary conditions and loads

Boundary conditions: We will fix the base of the frame. Click the Apply Fixity icon , make sure that the "Apply to" field is set to Points, enter 1, 2 in the first two rows of the table (leave the Fixity column blank) and click OK.

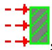
Loads: We will apply the two loads using different load applications and time functions. The vertical load will be applied using a constant time function (time function 1) and the horizontal load will be applied using an increasing time function (time function 2).

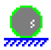
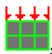
We will apply a horizontal load of 10000 N in 20 equal steps. Choose Control→Time Step, set the first row to 20, 500 and click OK.

To define the time functions, choose Control→Time Function and verify that time function 1 is a constant time function with unit magnitude. Now add time function 2, enter

Time	Value
0	0
10000	10000

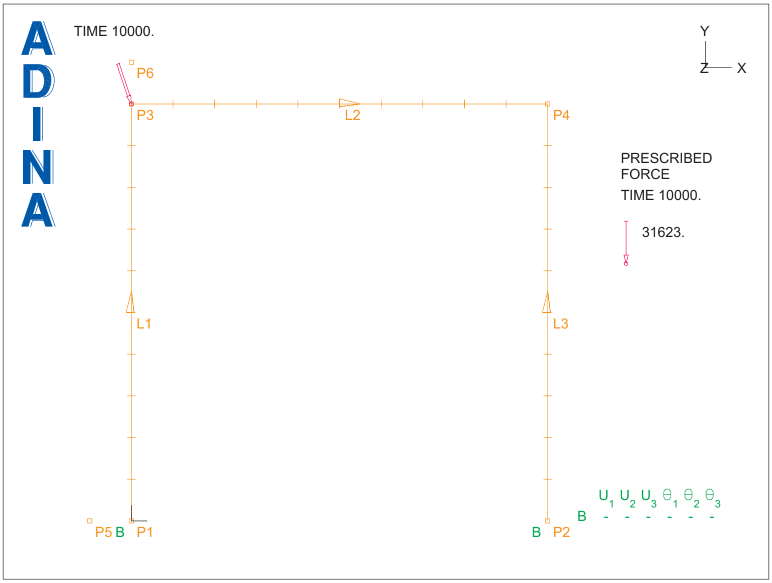
in the table and click OK. Notice that with this time function definition, the solution time represents the magnitude of the horizontal load.

Now we define the load applications. Click the Apply Load icon , set the Load Type to 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 30000, the Y Force Direction to –1.0 and click Save. Now add force 2, set the Magnitude to 1, the X Force Direction to 1.0 and click OK. In the Apply Load dialog box, make sure that the Load Number is 1 and that the "Apply to" field is set to Point, then, in the first row of the table, set the Site # to 3, then click Apply. Now set the Load Number to 2, make sure that the "Apply to" field is set to Point, then, in the first row of the table, set the Site # to 3, the Time Function to 2 and click OK.

When you click the Boundary Plot icon  and the Load Plot icon , the graphics window should look something like the figure on the next page.

Defining the material data

Choose Model→Materials→Beam Rigidity, add rigidity number 1 and set the Type to Multilinear Plastic. Check the "Unsymmetric" button between the Bending fields and click Save.



Axial strain/axial force data: Click the … button to the right of the Axial Force vs Strain Curve field. In the Define Axial Strain-Force Curve dialog box, add number 1, enter

Axial Strain	Axial Force
0.001	160000
0.003	160000

in the table, then click OK. In the Define Rigidity dialog box, set the Curve # field to 1 and click Save.

Torsional data: Click the … button to the right of the Torsion (r) field. In the Define Force vs Twist-Moment Curve dialog box, add number 1, then right-click in one of the cells in the Twist-Moment Curve column and click Define. In the Define Twist-Moment Curve dialog box, add number 1, enter

Twisted Angle per Unit Length	Moment
0.07	380
0.3	380

in the table and click OK to close the Twist-Moment Curve dialog box. In the Define Force vs Twist-Moment Curve dialog box, enter

Axial Force	Twist-Moment Curve
-1E6	1
1E6	1

in the table and click OK to close the Force/Twist Moment Curve dialog box. In the Define Rigidity dialog box, set the Torsion (r) field to 1 and click Save.

Bending data in the s direction: Click the … button to the right of the Bending (s) field. In the Define Force vs Curvature-Moment Curve dialog box, add number 1, then right-click in one of the cells in the Curvature-Moment Curve column and click Define. In the Define Curvature-Moment Curve dialog box, add number 1, enter

Curvature	Moment
-0.25	-800
-0.05	-567
0	0
0.05	567
0.25	800

in the table and click OK to close the Define Curvature-Moment Curve dialog box. In the Define Force vs Curvature-Moment Curve dialog box, enter

Axial Force	Curvature-Moment Curve
-1E6	1
0	1
1E6	1

in the table and click OK to close the Define Force vs Curvature-Moment Curve dialog box. In the Define Rigidity dialog box, set the Bending (s) field to 1 and click OK to close the Define Rigidity dialog box.

Bending data in the t direction: There is too much data to conveniently enter using dialog boxes, so we have put the equivalent commands in the batch file prob14_1.in. Read these commands as follows: Click the Open icon , navigate to the working directory or folder, set the 'Files of type' field to 'ADINA-IN Command Files (*.in)', select the file prob14_1.in and click Open.

The AUI processes the commands in the batch file.

Let's verify that the data is entered correctly. Choose Model→Materials→Beam Rigidity and click the … button to the right of the Bending (t) field. In the Define Force vs Curvature-Moment Curve dialog box, choose curve number 2. The table should be

Axial Force	Curvature-Moment Curve
-164000	3
-64000	3
0	2
64000	4
164000	4

Now right-click in one of the cells in the Curvature-Moment Curve column and click Define. In the Define Curvature-Moment Curve dialog box, choose curve number 2. The table should be

Curvature	Moment
-0.2	-1970
-0.1	-1881
-0.05	-1556
-0.0308	-1118
0	0
0.0308	1118
0.05	1556
0.1	1881
0.2	1970

When you choose curve number 3, the table should be

Curvature	Moment
-0.24	-1975
-0.1	-1686
-0.05	-1337
-0.0185	-671
0	0
0.0343	1246
0.05	1731
0.1	2189
0.24	2394

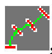
and when you choose curve number 4, the table should be

Curvature	Moment
-0.16	-2326
-0.1	-2189
-0.05	-1731
-0.0343	-1246
0	0
0.0185	671
0.05	1337
0.1	1686
0.16	1866

Click OK twice to close the Define Curvature-Moment Curve dialog box and the Define Force vs Curvature-Moment Curve dialog boxes. In the Define Rigidity dialog box, set the Bending (t) field to 2 and click OK.

Defining the elements

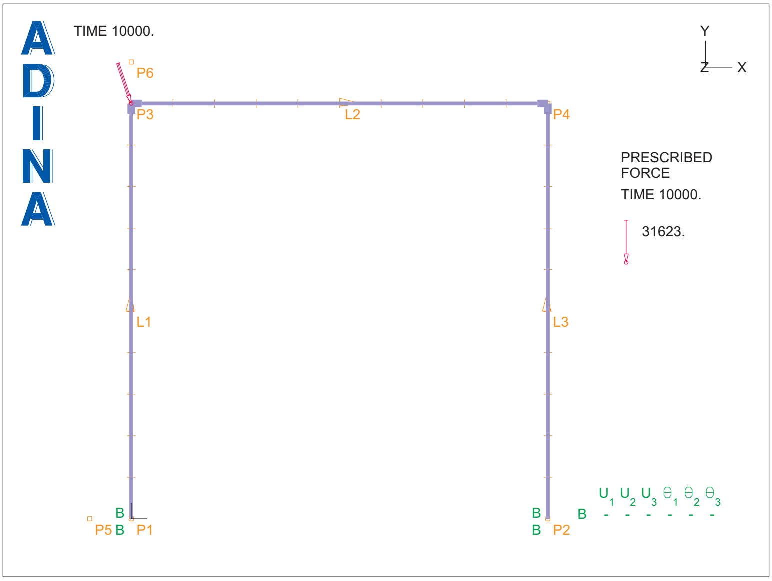
Element group: Click the Define Element Groups icon , add group number 1 and set the Type to Beam. Check the Use Moment-Curvature Rigidity button in the Stiffness Definition box and make sure that the Rigidity is 1. Now click the Advanced tab, set the “Rigid End-Zones” field to “Defined by Length with Infinite Stiffness” and click OK.

Element generation: Click the Mesh Lines icon , set the Auxiliary Point to 5, enter 1, 3 in the first two rows of the table and click Apply. Then set the Auxiliary Point to 6, enter 2 in the first row of the table and click OK.

Rigid-end data: Choose Meshing→Elements→Element Data, enter the following data for the indicated elements, and click OK.

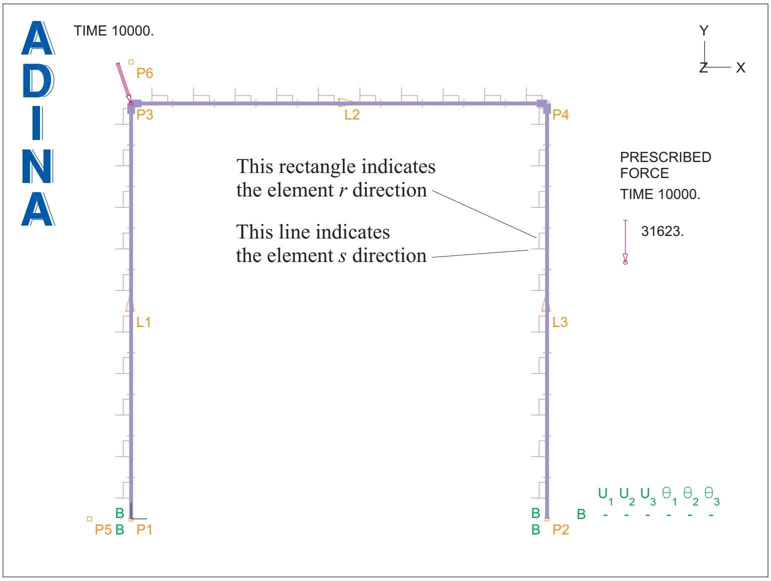
Beam Element	Rigid End-Zone (Length from Start)	Rigid End-Zone (Length from End)
10	0	0.025
20	0	0.025
21	0.025	0
30	0	0.025

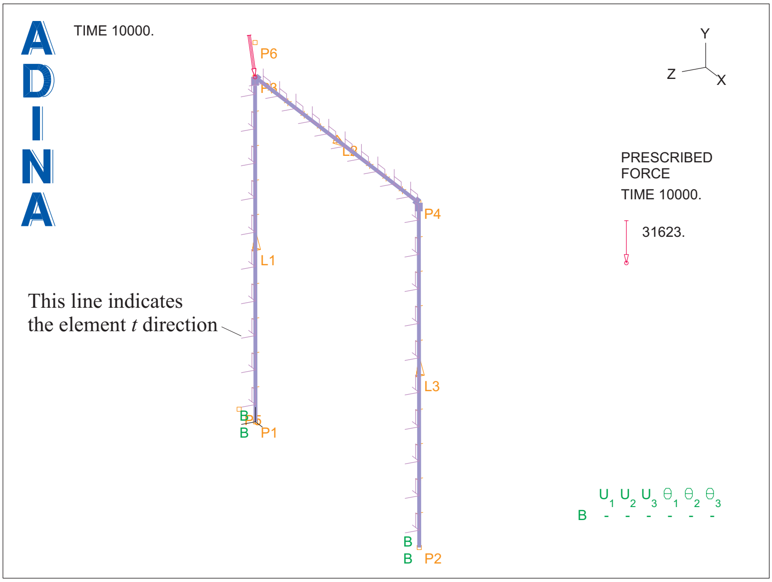
When you click the Redraw icon , the graphics window should look something like the figure on the next page.



Checking the element orientations: Let's confirm the element local coordinate system directions. Click the Modify Mesh Plot icon , then click the Element Depiction… button. In the Element Depiction dialog box, check the Display Local System Triad button, make sure that the Type is "Element Coordinate System" and click OK twice to close both dialog boxes. The graphics window should look something like the top figure on the next page.

Use the mouse to rotate the model out-of-plane until the graphics window looks something like the bottom figure on the next page.


ADINA
TIME 10000.
Y
Z X
P6
P3
L2
P4
This rectangle indicates
the element r direction
This line indicates
the element s direction
PRESCRIBED
FORCE
TIME 10000.
31623.
L1
L3
U1 U2 U3 θ1 θ2 θ3
B - - - - - -
B
P5 B P1
B
B P2


ADINA
TIME 10000.
Y
Z X
P6
P4
L1
PRESCRIBED
FORCE
TIME 10000.
31623.
This line indicates
the element t direction
L3
B
B P1
U1 U2 U3 θ1 θ2 θ3
B - - - - - -
B
B P2

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

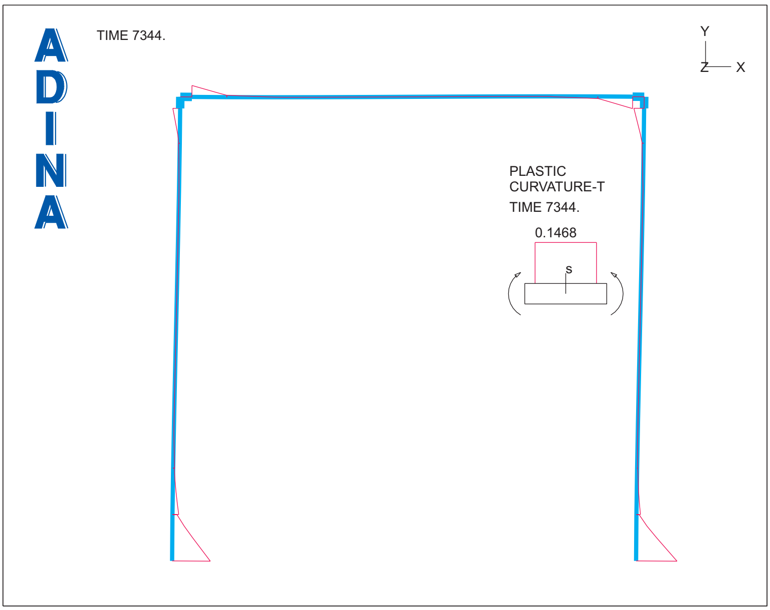
Click the Save icon and save the database to file prob14. Click the Data File/Solution icon, set the file name to prob14, make sure that the Run Solution button is checked and click Save.

ADINA reports "*** Program stopped abnormally ***…". This is OK, as we shall see, ADINA does not converge in step 15 because the pushover load is exceeded. See the comments at the end of this problem description for more information.

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 prob14.

Displaying the plastic curvatures

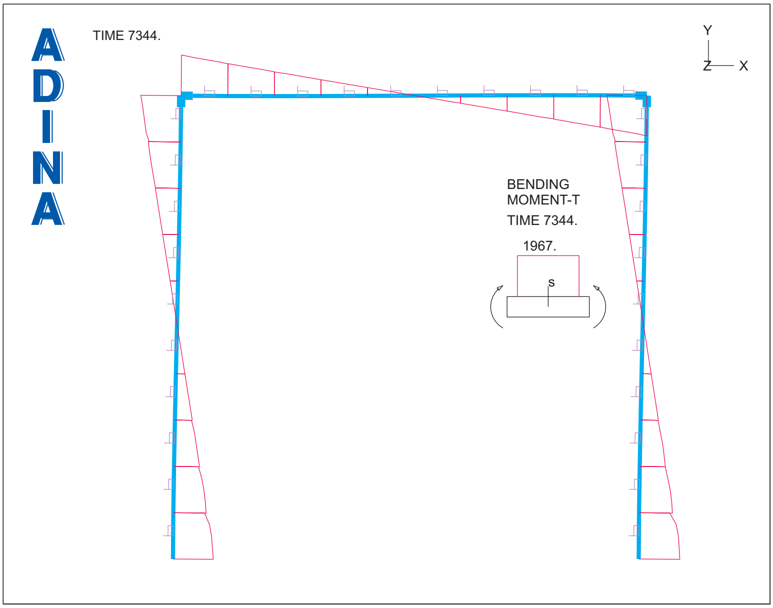
Choose Display→Element Line Plot→Create, set the Element Line Quantity to PLASTIC_CURVATURE-T and click OK. The graphics window should look something like this:



We see that the highest horizontal load reached was 7344 (N) and that there is significant plastic deformation at the bases and at the corners. Use the Previous Solution icon and the Next Solution icon to examine how the plasticity develops as the horizontal load is increased. When you are finished, click the Last Solution icon to return to the last solution.

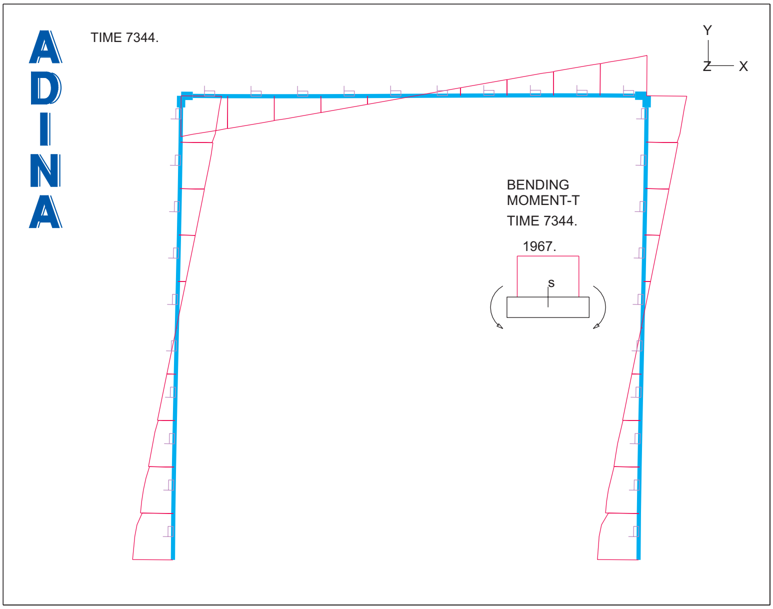
Displaying the moments

Choose Display→Element Line Plot→Modify, set the Element Line Quantity to BENDING_MOMENT-T and click the Rendering… button. In the Define Element Line Rendering Depiction dialog box, set the Scale Factor to Automatic and click OK twice to close both dialog boxes. Click the Modify Mesh Plot icon, then click the Element Depiction… button. In the Element Depiction dialog box, check the Display Local System Triad button, make sure that the Type is "Element Coordinate System" and click OK twice to close both dialog boxes. The graphics window should look something like this:



This plot shows that the bending moments at the base are negative (the directions of the bending moment lines are opposite to the s-directions of the elements at the base), and the bending moments of the vertical members at the rigid corners are positive (the directions of the bending moment lines are the same as the s-directions of these elements). You can change the sign convention used in the element line plot as follows. Choose Display→ Element Line Plot→Modify, click the Rendering… button, set the Positive Moment Convention to Clockwise and click OK twice to close both dialog boxes.

The graphics window should look something like this:



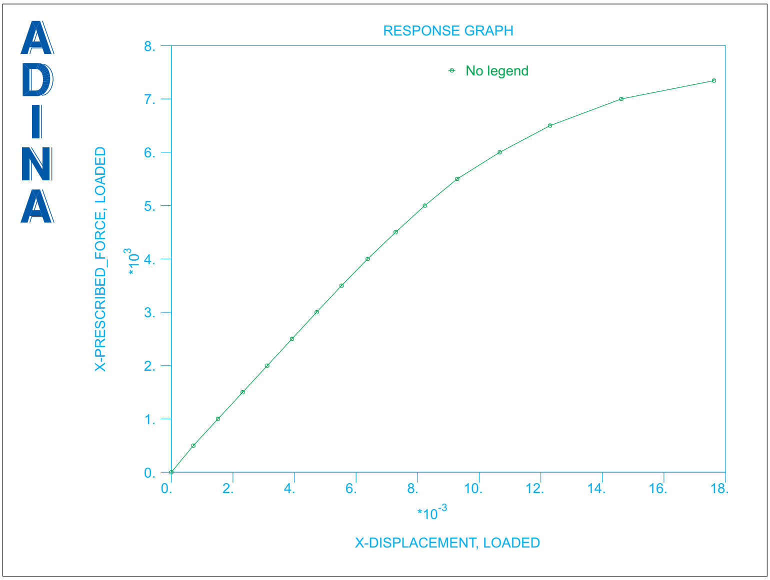
Graphing the force-deflection curve

We will create a force-deflection graph. To determine the node number where the load is applied, click the Node Labels icon (the node should be node 11). Then choose Definitions→Model Point→Node, define point LOADED as node 11, then click OK. Now click the Clear icon, choose Graph→Response Curve (Model Point), set the X variable to (Displacement:X-DISPLACEMENT), set the Y variable to (Prescribed Load: X-PRESCRIBED_FORCE), verify that the model point is LOADED, then click OK.

The graphics window should look something like the figure on the next page.

Graphing the moment-curvature curves at the base

We will graph the moment-curvature curves at the base. First we define two element section points corresponding to the nodes at the base. Click the Clear icon, then click the Node Labels icon and the Element Labels icon to display the node numbers and element numbers.



Choose Definitions→Model Point→Element Section, add point LEFT_BASE, set the Element Number to 1, set “Defined by” to Node Number, set the Node Number to 1 and click Save. Now add point RIGHT_BASE, set the Element Number to 11, set “Defined by” to Node Number, set the Node Number to 13 and click OK.

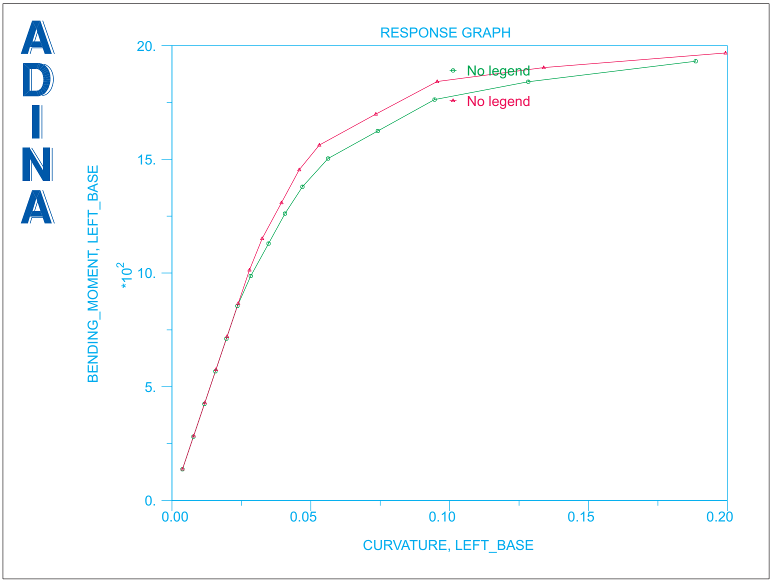
Now click the Clear icon, choose Graph→Response Curve (Model Point), set the X Variable to (Strain:CURVATURE-T), set the Y Variable to (Force:BENDING_MOMENT-T), verify that the model point is LEFT_BASE, then click OK.

Notice that the bending moment and curvature are negative. We will define resultants to switch the signs of the bending moment and curvature. Choose Definitions→Variable→Resultant, add resultant BENDING_MOMENT, define it as -<BENDING_MOMENT-T> and click Save. Now add resultant CURVATURE, define it as -<CURVATURE-T> and click OK. (Hint, you can type the resultant in upper, lower or mixed upper and lower case.)

Now click the Clear icon, choose Graph→Response Curve (Model Point), set the X Variable to (User Defined:CURVATURE), set the Y Variable to (User Defined: BENDING_MOMENT), verify that the model point is LEFT_BASE, then click OK.

To add the curve for the right base, choose Graph→Response Curve (Model Point), set the X Variable to (User Defined:CURVATURE), set the X Model Point to RIGHT_BASE, set the Y Variable to (User Defined:BENDING_MOMENT), set the Y Model Point to RIGHT_BASE, set the Plot Name to PREVIOUS and click OK.

The graphics window should look something like this:



The graph title, axes and curves can be customized as in Primer Problem 2.

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

Examining the output file

We will examine the ADINA output file to learn how the ATS algorithm determined a converged solution near the pushover load. Click the View Output button in the ADINA control window to view the output file, or use a text editor to edit file prob14.out. Go to the end of the file and back up until you see the text

```
RESTART DATA IS SAVED FOR STEP     14    AT TIME EQUALS  0.700000000000E+04
```

Step 14 corresponds to time 7000.0. For step 15, ADINA first attempts to find the solution for time 7500.0. ADINA performs 6 equilibrium iterations, then reports

```
RESTART FOR AUTOMATIC-TIME-STEPPING AFTER ITERATION    7

OUT-OF-BALANCE LOADS LARGER THAN INCREMENTAL LOADS
```

ADINA then tries to find the solution for time 7250, and succeeds after 4 equilibrium iterations.

Now ADINA tries to find the solution for time 7500 (starting from the solution for time 7250) and reports

```
RESTART FOR AUTOMATIC-TIME-STEPPING AFTER ITERATION    7

OUT-OF-BALANCE LOADS LARGER THAN INCREMENTAL LOADS
```

In this manner, ADINA continues to cut back the time step, as shown in the following table:

Current time	Time step size	Trial solution time	Result
7000	500	7500	No convergence
7000	250	7250	Convergence
7250	250	7500	No convergence
7250	125	7375	No convergence
7250	62.5	7312.5	Convergence
7312.5	62.5	7375	No convergence
7312.5	31.25	7343.75	Convergence
7343.75	31.25	7375	No convergence
7343.75	25	7368.75	No convergence
7343.75	156.25	7500	No convergence

ADINA stops when the last time step size is smaller than or equal to the original time step size divided by the maximum number of subdivisions (in this case, the maximum number of subdivisions allowed was 20).

So the pushover load is probably between 7343.75 N and 7368.75 N.

This page intentionally left blank.