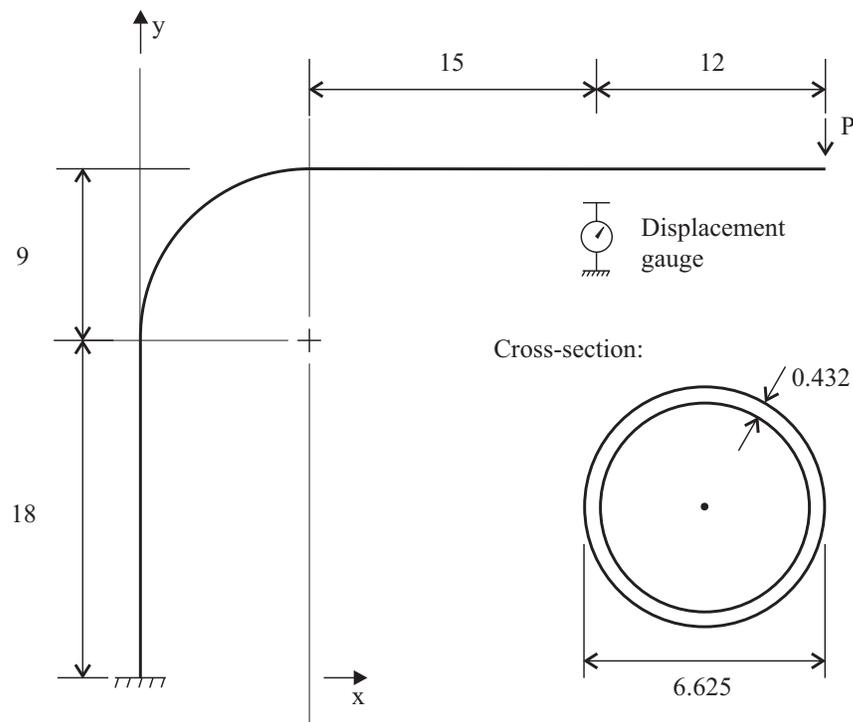


Problem description

A pipe bend is subjected to a concentrated force as shown:



All dimensions in inches. $E = 29700$ kpsi
 Material is stainless steel. $\nu = 0.27$

The material of the pipe can be idealized as an elastic-plastic material using the von Mises yield criterion with isotropic hardening, with the following points on the uniaxial stress-strain curve:

Logarithmic strain	True stress (10^3 psi)
6.06×10^{-4}	18.0
0.002	35.4
0.0077	40.8
0.02	48.9
0.04	56.5
0.1	72.2

Problem 23: Limit load analysis of a pipe bend

We would like to obtain the force-deflection curve for the pipe and, in particular, the limit load.

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

- Large displacement elastic-plastic analysis using shell elements.
- Specifying a collapse analysis.
- Adding text to the graphics window.

We assume that you have worked through problems 1 to 22, or have equivalent experience with the ADINA System. Therefore we will not describe every user selection or button press.

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 model control data

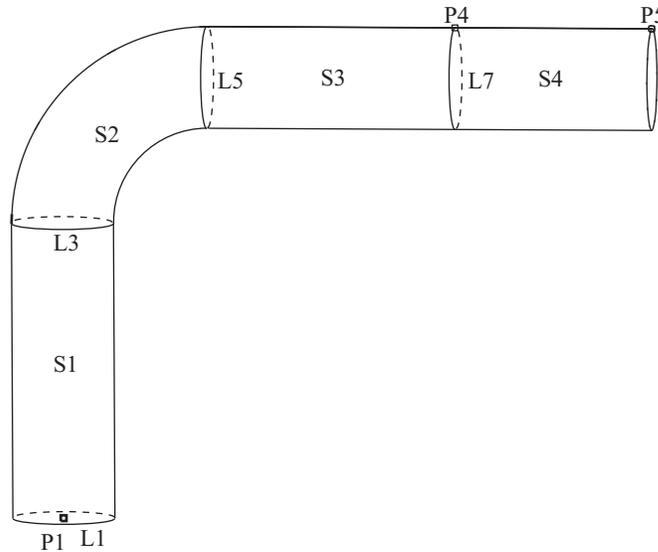
Problem heading: Choose Control→Heading, enter the heading 'Problem 23: Limit load analysis of a pipe bend' and click OK.

Overall model control data: We will perform a collapse analysis using the load-displacement control (LDC) algorithm to automatically choose the load step sizes. Set the Analysis Type drop-down list to Collapse Analysis. We will specify additional parameters needed for the LDC algorithm later.

Kinematics: We anticipate that the displacements of the pipe can be large. Therefore we select a large displacement, large strain formulation for use in the analysis. Choose Control→Analysis Assumptions→Kinematics, set the Displacements/Rotations field to Large, set the Strains field to Large, and click OK. (See the “Modeling comments” section for the reason why the Strains field should be set to Large.)

Defining the model geometry

The diagram on the next page shows the key geometry used in defining the model.



We will define the pipe midsurface by creating a circular cross-section at the base of the pipe and then extruding the cross-section along the pipe axis.

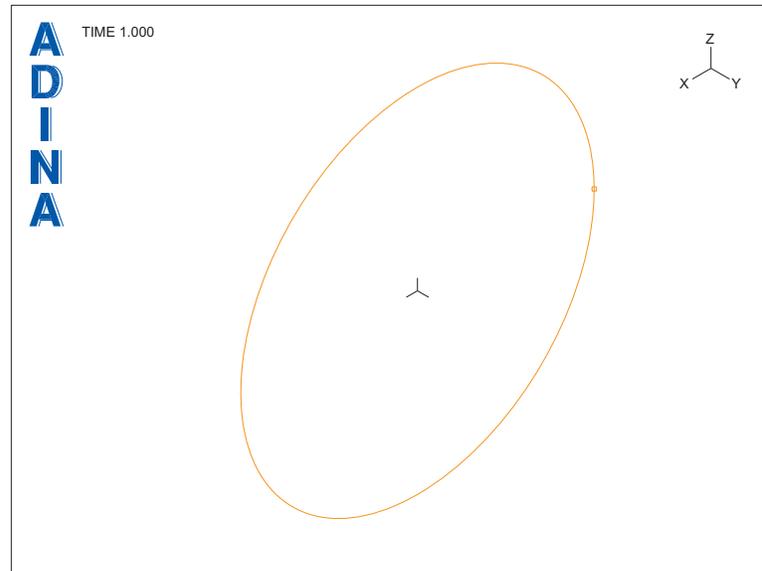
To create the pipe cross-section, we will create a point on the pipe cross-section and revolve the point around the pipe axis (at the base of the pipe, the pipe axis is coincident with the y-axis). Click the Define Points icon , enter the following point and click OK.

Point #	X1	X2	X3
1	-3.0965	0	0

(3.0965 corresponds to the radius of the midsurface of the pipe.) Now, to revolve the point, click the Define Lines icon , add line number 1 and set the Type to Revolved. Set the Initial Point to 1, the Angle of Rotation to 360, the Axis to Y and click OK. The graphics window should look something like the figure on the next page.

Now we will extrude the cross-section in the direction of the y-axis to create the surface for the first straight piece of the pipe. Click the Define Surfaces icon , add surface number 1 and set the Type to Extruded. Set the Initial Line to 1, the components of the Vector to 0.0, 18.0, 0.0 and click OK.

Problem 23: Limit load analysis of a pipe bend



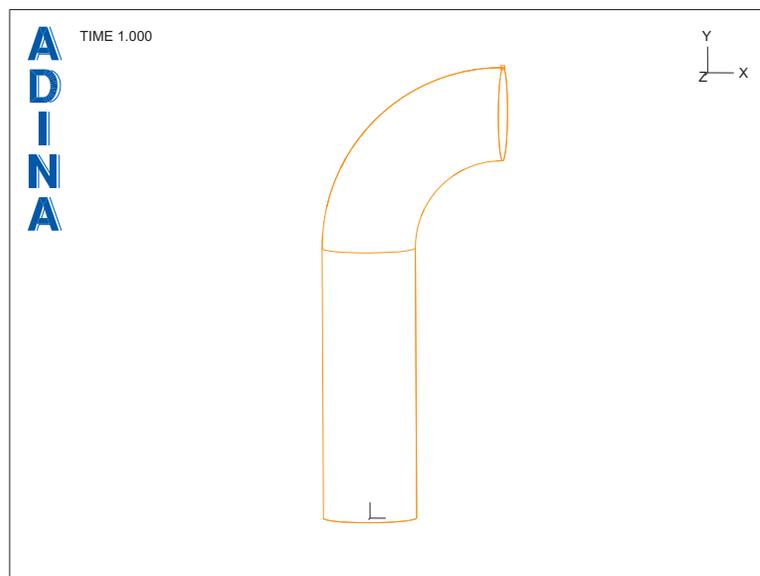
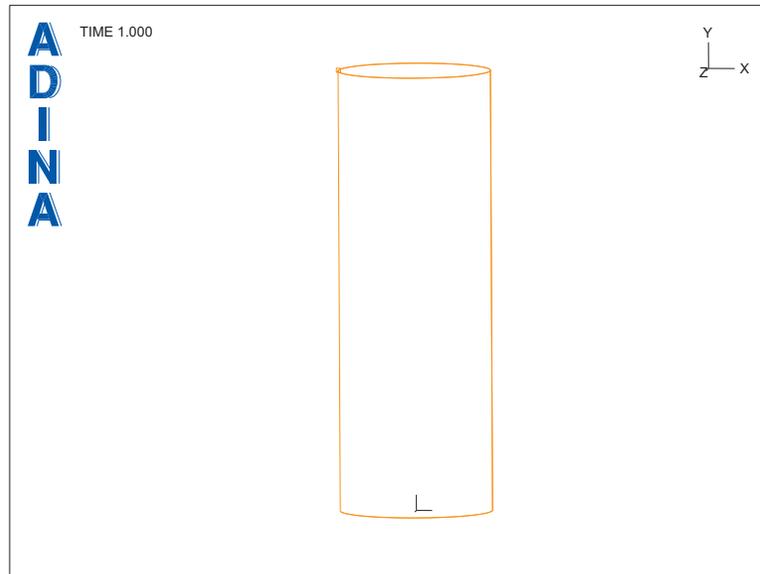
We want to choose a more convenient view for the mesh plot. Click the Modify Mesh Plot icon , click the View... button, set the View Direction (not the View Point) to 0.1, 0.1, 1.0, set the Angle of Rotation to 135 and click OK twice to close both dialog boxes. The graphics window should look something like the top figure on the next page.

We will display all successive meshes using this view, so we will change the default view.

Click the Save View icon .

Let's continue the definition of the pipe surfaces. To create the pipe bend, we need to revolve the newly-created cross-section line 90 degrees about an axis with center (9.0,18.0,0.0) and components (0.0,0.0,1.0). From the mesh plot, we observe that the line that we need to revolve is line number 3 (use the Query icon  and the mouse to confirm this). Click the Define Surfaces icon , add surface number 2 and set the Type to Revolved. Set the Initial Line to 3, the Angle of Rotation to -90, "Axis of Revolution Defined by" to Vector, the components of the Vector Origin to 9, 18, 0 and the components of the Vector Direction to 0, 0, 1. When you click OK, the graphics window should look something like the bottom figure on the next page. (You might need to use the mouse to rescale the mesh plot.)

Problem 23: Limit load analysis of a pipe bend



To create the remaining straight pipe segments, we will continue to extrude the pipe cross-section. The first extrusion will create the straight pipe up to the location where we want to measure the displacement and the second extrusion will create the remaining pipe.

Problem 23: Limit load analysis of a pipe bend

Notice that the line for the pipe cross-section that we want to extrude is line 5. To create the first extrusion, click the Define Surfaces icon , add surface number 3 and set the Type to Extruded. Set the Initial Line to 5, the components of the Vector to 15, 0, 0 and click Save. Notice that the newly created line segment for the pipe cross-section is line number 7. To create the second extrusion, return to the dialog box, add surface number 4, set the Initial Line to 7, the components of the Vector to 12, 0, 0 and click OK.

Surface thicknesses: Choose Geometry→Surfaces→Thickness, enter 0.432 for the thickness of each surface and click OK.

Defining boundary conditions and loads

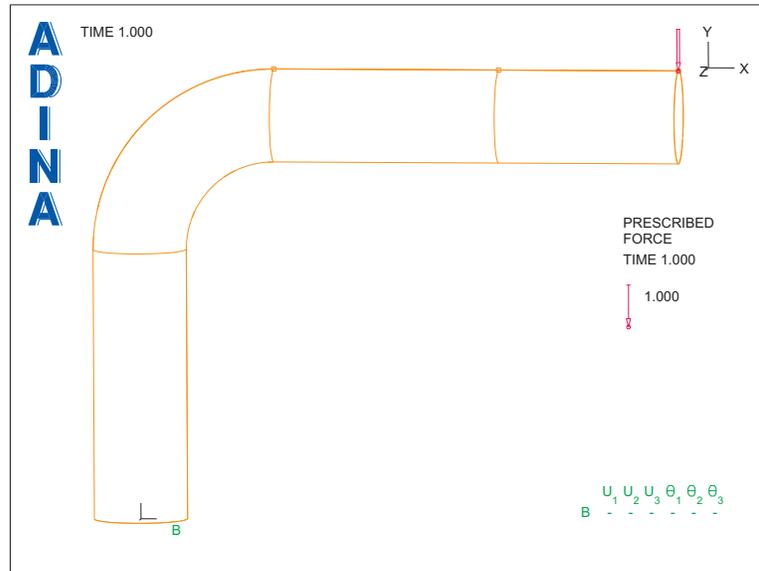
Boundary conditions: We will fix line 1. Click the Apply Fixity icon , set the “Apply to” field to Edge/Line, enter line number 1 in the first row and column of the table and click OK.

Loads: We will apply a unit concentrated load to point 5 (the tip of the pipe). 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 1.0, set the Force Direction to 0, -1, 0 and click OK. In the Apply Load dialog box, make sure that the “Apply to” field is set to Point, and, in the first row of the table, set the Point # to 5. Click OK to close the Apply Load dialog box.

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.

Remember that because we are performing a collapse analysis, ADINA Structures will choose the load step sizes automatically. Since we have defined the force, we can now complete the input for the LDC algorithm. We will apply a displacement of 0.5 inches downward in the first solution step and we want the algorithm to terminate after a displacement of 4 inches is reached. Also we will let ADINA Structures run for a maximum of ten load steps.

Click the Analysis Options icon , make sure that the Point/Node # is set to Point, set the Label # field to 5, the Degree of Freedom to Y-Translation, the Displacement field to -0.5, the Maximum Allowed Displacement to 4, check the “Continue after the first Critical Point is reached” button and click OK. Now choose Control→Time Step, enter 10 in the first row in the Number of Steps column and click OK.



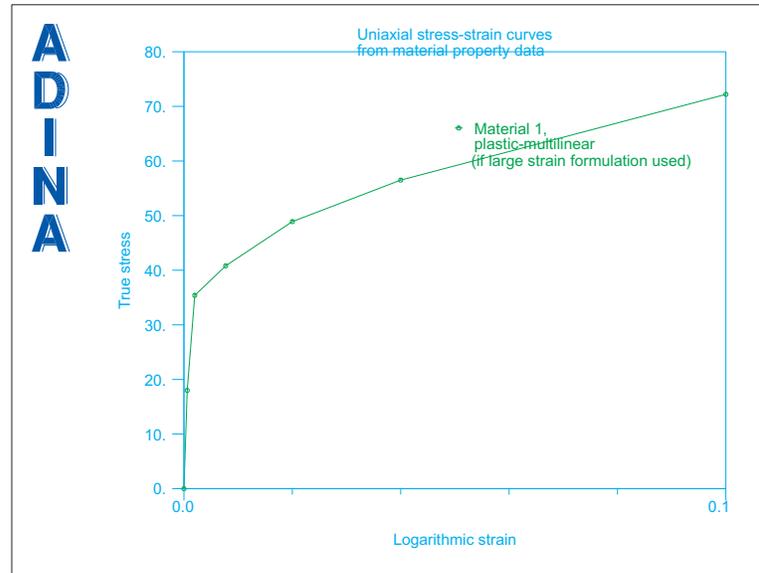
Defining the material

Click the Manage Materials icon  and click the Plastic Multilinear button. In the Define Multilinear Elastic-Plastic Material dialog box, add material 1, set the Young's Modulus to 29700, the Poisson's ratio to 0.27 and verify that the Type of Strain Hardening is Isotropic. Then enter the following stress-strain data points in the Stress-Strain curve table (these points are repeated from the problem description for convenience). Do not click OK yet.

Strain	Stress
6.06E-04	18.0
0.002	35.4
0.0077	40.8
0.02	48.9
0.04	56.5
0.1	72.2

Now click the Graph button to display the stress-strain curve. The AUI displays a new graphics window that should look something like the figure on the next page.

Problem 23: Limit load analysis of a pipe bend



Notice that the graph shows true stress vs logarithmic strain. For problems in which the strains are relatively small, as in this problem, these quantities are close to the engineering quantities.

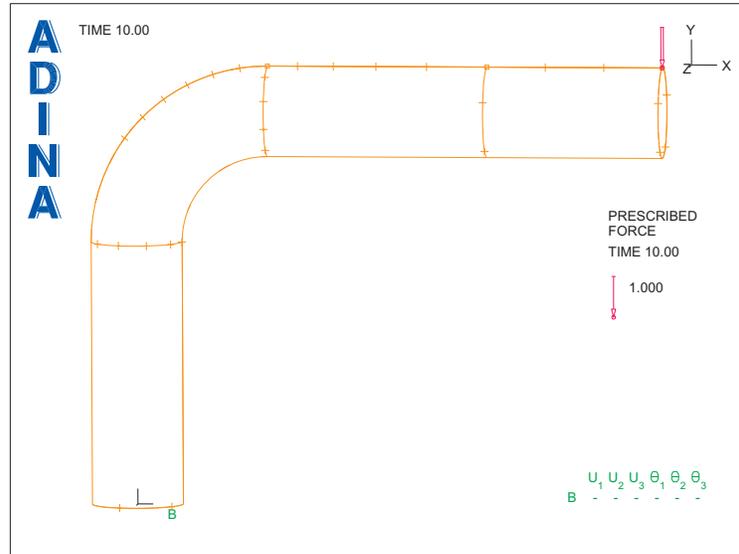
Close the new graphics window, then click OK to close the Define Multilinear Elastic-Plastic Material dialog box, and click Close to close the Manage Material Definitions dialog box.

Defining the finite elements and nodes

Subdivision data: We enter the mesh size at the geometry points, with a smaller mesh size at the pipe bend. First choose Meshing→Mesh Density→Complete Model, make sure that the “Subdivision Mode” is “Use End-Point Sizes” and click OK. Now choose Meshing→Mesh Density→Point Size, set the “Points Defined from” field to “All Geometry Points”, set the Maximum to 4 and click Apply. Change the Mesh Size for points 2 and 3 to 2.0 and click OK.

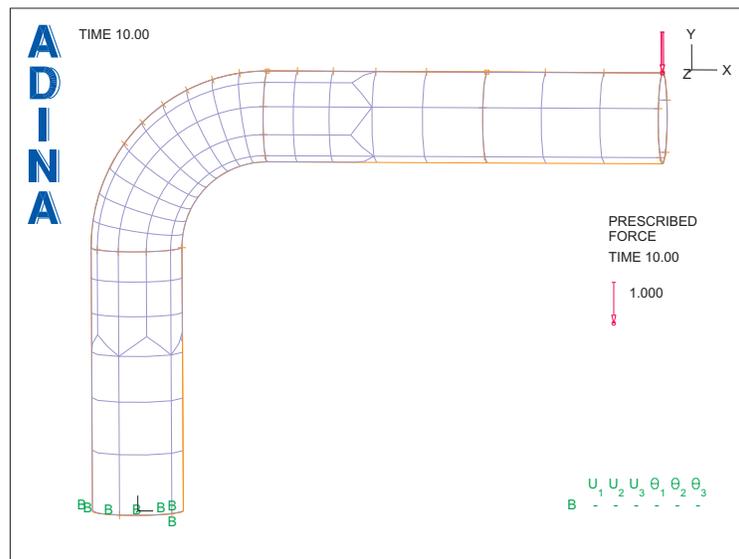
The graphics window should look something like the top figure on the next page.

Problem 23: Limit load analysis of a pipe bend



Finite elements: We will generate 9-node shell finite elements on the geometry surfaces.

Click the Mesh Surfaces icon , set the Type to Shell and click the + button to the right of the Element Group text. Now set the “Nodes per Element” to 9 and click the Options tab. In the Nodal Coincidence Checking box, set the “Check” field to “All Generated Nodes”. Click the Basic tab, enter surfaces 1, 2, 3, 4 in the first four rows of the table and click OK. The graphics window should look something like this:



Problem 23: Limit load analysis of a pipe bend

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

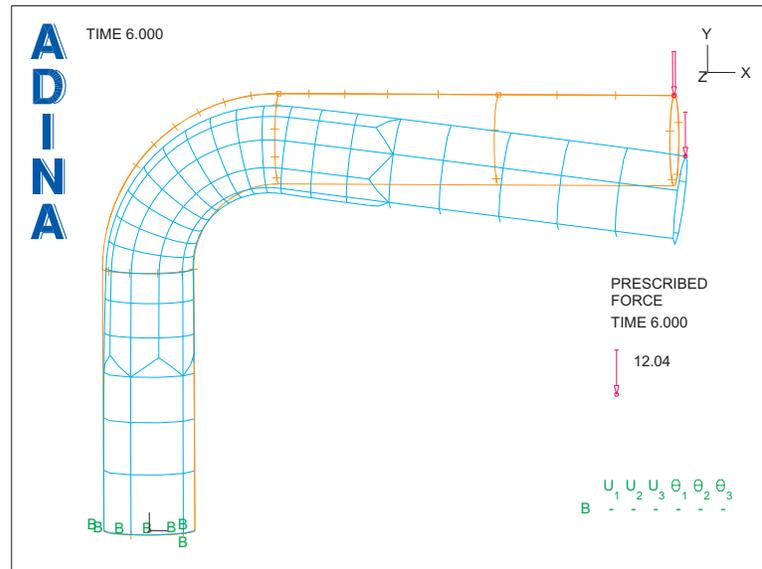
Click the Save icon  and save the database to file prob23. Click the Data File/

Solution icon , set the file name to prob23, 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 and discard all changes. Then click the Open icon , set the “File type” field to “ADINA-IN Database Files (*.idb)”, choose file prob23 and click Open. Then click the Open icon  and open porthole file prob23.

Please notice that we first opened the ADINA-IN database, then loaded the porthole file. We did this so that we can create a force-deflection curve of the results using the geometry points.

Obtaining a deformed mesh plot

The graphics window should look something like this:



You will notice what appear to be missing lines in the finite element model. These missing lines lie on the outline (or silhouette) of the model. To display these lines, click the Modify

Mesh Plot icon  and click the Rendering... button. In the Mesh Rendering Depiction dialog box, set the “Generate Outline” field to “Geometry and Mesh”, then click OK twice to

close both dialog boxes.

Since we always want to display the outline lines in this model, click the Save Mesh Plot Style icon .

(Note: the outline lines are only important if the mesh is relatively coarse and higher-order elements are used, as in this model. So the default is for the outline lines not to be plotted.)

Obtaining a summary of model information

To view a summary of the model, choose List→Info→Model. There are 830 nodes and one element group containing 210 shell elements. Click Close to close the dialog box.

To view a summary of loaded responses, choose List→Info→Response. There are seven load steps loaded from times 0 to 6 (the first load step contains the initial conditions). Recall that you requested 10 solution load steps; ADINA Structures computed only 6 solution load steps because the maximum displacement specified for collapse analysis was exceeded in step 6. Click Close to close the dialog box.

Viewing the solutions

Use the Previous Solution icon  and Next Solution icon  to display the other solutions. Notice that the deformations increase as the load increases, as we expect. When you are finished, click the Last Solution icon .

Making a force-deflection curve graph

We will plot the applied force versus the deflection at the deflection gauge.

Result points: Before we can create the graph, we need to define result points for the node where the load is applied and for the node associated with the displacement gauge. Since we have geometry points at these node points, we will define these result points in terms of the geometry points.

Choose Definitions→Model Point (Combination)→General, add name TIP, enter “POINT 5” in the first row of the table and click Save. Define the name GAUGE to be the geometry point 4 in the same way. Click OK to close the dialog box.

Variables: We need to define variables corresponding to the force and displacement. Choose Definitions→Variable→Resultant, add resultant name FORCE, enter the expression

-<Y-PRESCRIBED_FORCE>

Problem 23: Limit load analysis of a pipe bend

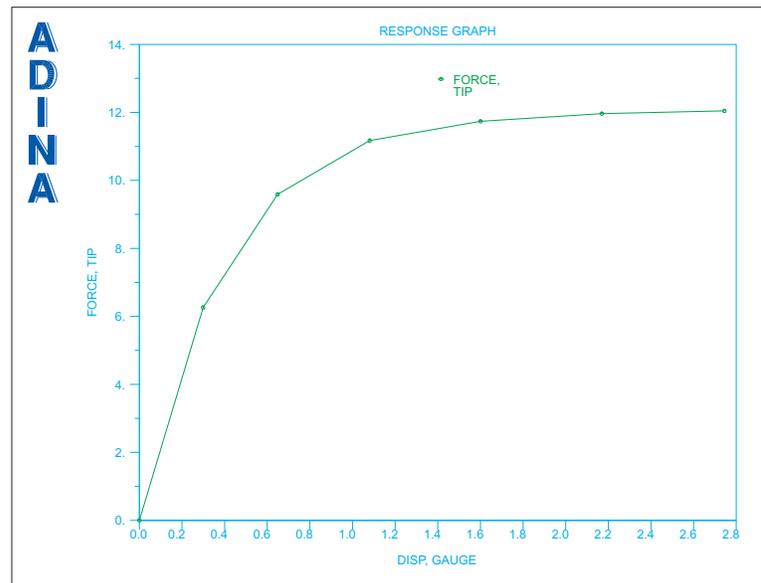
and click Save. Define the name DISP to be the expression

-<Y-DISPLACEMENT>

in the same way. Click OK to close the dialog box.

Graph: Click the Clear icon , then choose Graph→Response Curve (Model Point). Set the X variable to (User Defined:DISP) and set the X model point to GAUGE. Set the Y variable to (User Defined:FORCE) and set the Y model point to TIP. Then click OK.

The graphics window should look something like this. It is, of course, possible to change the graph title, axis labels and curve legends as was shown in problem 2.



Drawing bands of plastic strain

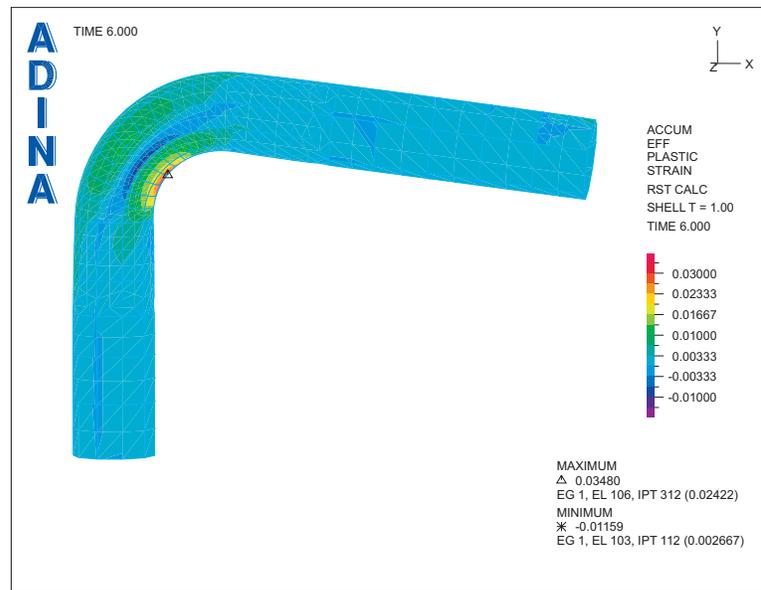
Let's draw bands corresponding to the accumulated effective plastic strain. This strain will show us which areas of the pipe are most damaged.

Click the Clear icon , then the Mesh Plot icon .

In the band plots, we do not want to show the mesh geometry or the boundary conditions.

Click the Show Geometry icon  and the Boundary Plot icon . Then click the Save Mesh Plot Style icon  to update the defaults.

To draw the bands, click the Create Band Plot icon , set the Band Plot Variable to (Strain: ACCUM_EFF_PLASTIC_STRAIN) and click OK. Use the Pick icon  and the mouse to resize the mesh plot and rearrange the annotations until the graphics window looks something like this:



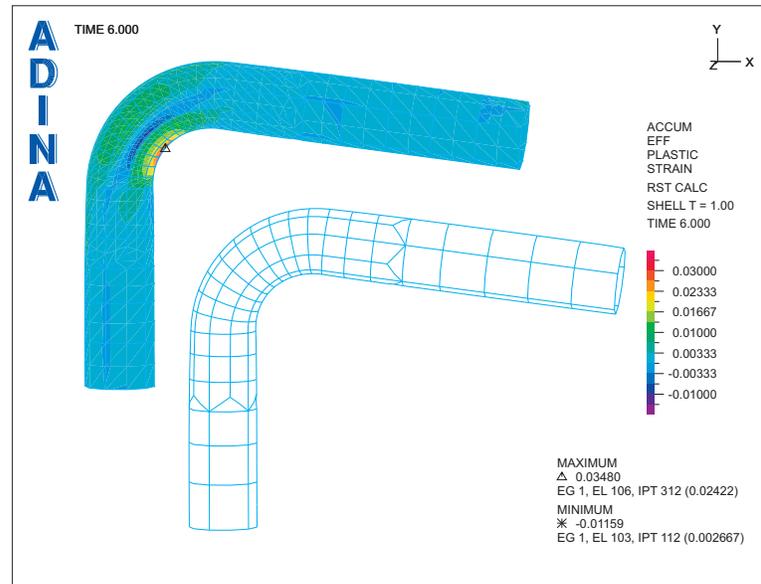
We are now observing the plastic strains on the top surface of the pipe skin (which is the outer surface in this model). We would also like to observe the plastic strains on the bottom (inner) surface of the pipe skin. So we will display another mesh plot, then we will plot plastic strains of the inner surface onto this mesh plot.

Click the Mesh Plot icon  and use the Pick icon  to move the mesh plot to a position under and to the right of the first mesh plot. Shrink both mesh plots so that they both fit in the graphics window. Also delete any duplicate text and axes. The graphics window should look something like the figure on the next page.

Now, before we draw the bands on the second mesh plot, we instruct the AUI to compute plastic strains on the bottom surface of the shell elements. Choose Definitions→Result Control, set the Result Control Name to DEFAULT, set the t Coordinate field to -1.0 and

click OK. Click the Create Band Plot icon , set the Band Plot Variable to (Strain: ACCUM_EFF_PLASTIC_STRAIN) and click OK.

Problem 23: Limit load analysis of a pipe bend



It is difficult to compare the two pictures because the band tables are different. So we will use the same band table for both band plots. Click the Modify Band Plot icon , set the Band Plot Name to BANDPLOT00001, click the Band Table... button, set the Value Range to Maximum = 0.05 and Minimum = 0.0, then click OK twice to close both dialog boxes. Repeat this procedure for band plot BANDPLOT00002.

We are also not interested in the minimum value of the plastic strain. Click the Modify Band Plot icon , set the Band Plot Name to BANDPLOT00001, click the Band Rendering... button, set the “Extreme Values” field to “Plot the Maximum” and click OK twice to close both dialog boxes. Repeat this procedure for band plot BANDPLOT00002.

Since we now have two band tables, each with almost the same information, use the Pick icon  and the mouse to delete one of them.

Let's add some text to label the two plots. Choose Display→Text→Draw, enter the text

Outer surface

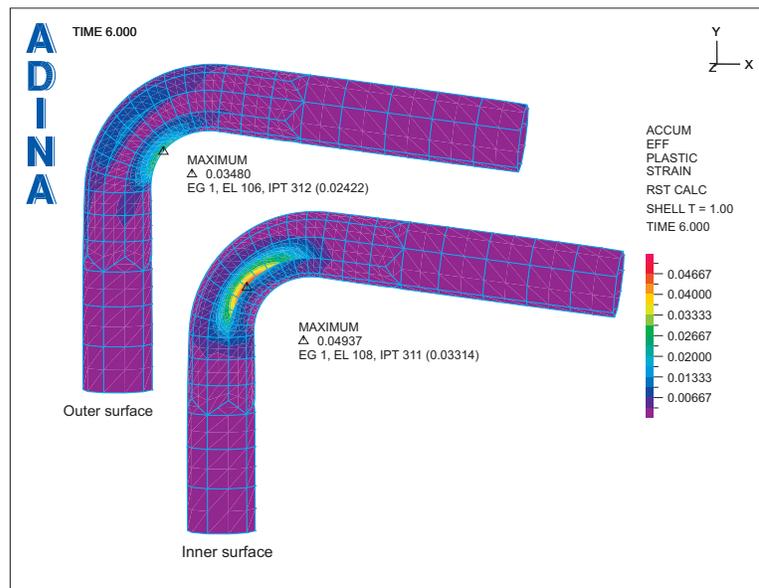
in the Text box and click OK. The AUI draws the text near the center of the graphics

window. Use the Pick icon  to move and resize the text so that it is below the base of the upper mesh plot. Repeat these steps for the text

Inner surface

and place this text below the base of the lower mesh plot.

The graphics window should look something like this:



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

Modeling comments

1. A large strain formulation is chosen because the displacements are large and the material is elastic-plastic. A large displacement / small strain formulation could have been chosen for this analysis, because the strains are anticipated to be small. However, it turns out that the use of a large displacement / small strain formulation is not appropriate for an elastic-plastic material, even when the strains are small. The reason is as follows:

The choice of formulation affects the choice of stresses / strains used:

Small displacement / small strain (MNO): Engineering stresses and small strains.

Large displacement / small strain: 2nd Piola-Kirchhoff stresses and Green-Lagrange strains.

Large displacement / large strain: Cauchy stresses and Hencky strains.

When the strains are numerically small, the engineering, 2nd Piola Kirchhoff and Cauchy stresses are all numerically close to each other, also the small, Green-Lagrange and Hencky strains are all numerically close to each other. Hence it would appear that, in particular, the

Problem 23: Limit load analysis of a pipe bend

large displacement / small strain formulation can be used when the strains are numerically small.

However, if a single elastic-plastic element is put into uniaxial tension and analyzed with the three formulations, it is found that the slope of the force-deflection curve is significantly different for all three formulations, with the large displacement / small strain formulation giving a stiffer response than the other formulations.

This phenomenon occurs when the slope of the stress-strain curve is of the same order of magnitude as the stress, which is frequently the case in plasticity.

Due to the above reasoning, we recommend that the large displacement / large strain formulation be used whenever large displacement effects are to be included in the analysis, even when the strains are numerically small.