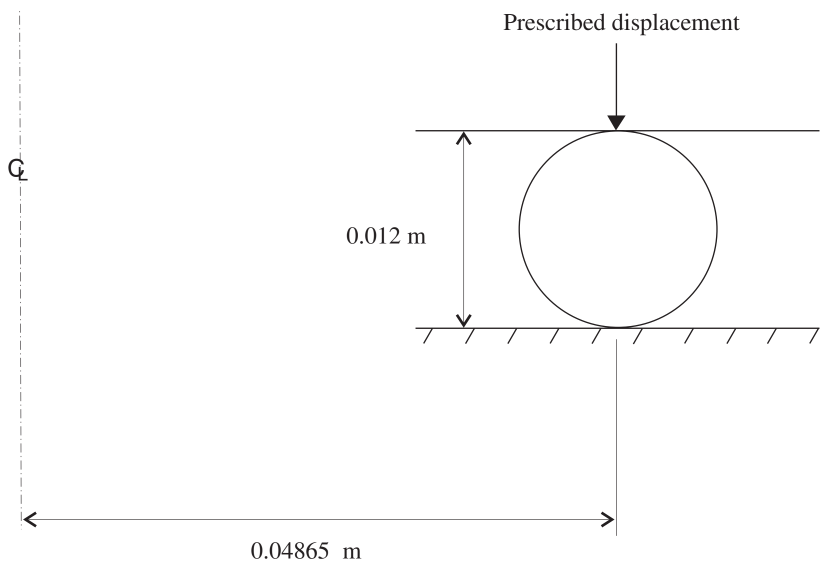
Problem description

A rubber O-ring is pressed between two frictionless plates as shown:



A two-dimensional axisymmetric analysis is appropriate here. Data points on the uniaxial stress-strain curve for the rubber are

Engineering strain (m/m)	Engineering stress (MPa)
-0.5	-0.2383
-0.3	-0.1035
-0.1	-0.0275
0.0	0.0
0.1	0.0111
0.3	0.0280
0.5	0.0409
0.7	0.0516
0.9	0.0610

In this analysis, we would like to obtain the deformations, contact forces and stress state for a prescribed displacement of 0.004 m.

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

- Input of stress-strain data for rubberlike materials
- Plotting and listing the strains.

We assume that you have worked through problems 1 to 21, 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 choose ADINA Structures from the Program Module drop-down list.

Defining model control data

Problem heading: Choose Control→Heading, enter the heading “Problem 22: Rubber O-ring pressed between two frictionless plates” and click OK.

Master degrees of freedom: Choose Control→Degrees of Freedom, uncheck the X-Translation, X-Rotation, Y-Rotation and Z-Rotation buttons 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 OK.

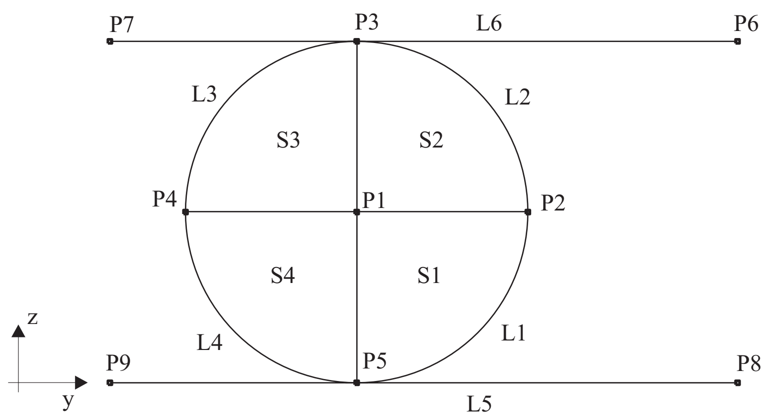
Equilibrium iteration tolerances: We will change the convergence tolerances used during equilibrium iterations. Choose Control→Solution Process, click the Iteration Tolerances... button and set the Convergence Criteria to Energy and Force. Set the Contact Force Tolerance to 1E-3, the Minimum Reference Contact Force to 1E-8, the Reference Force field in the Force Tolerances box to 3E-4 and click OK twice to close both dialog boxes.

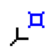
Time function: We will apply the entire load in one time step. We need a time function that reaches the maximum prescribed displacement at time 1.0. Choose Control→Time Function, edit the table as follows, and click OK.

Time	Value
0.0	0.0
1.0	0.004

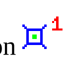
Defining the model geometry

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 (remember to leave the X1 column blank) and click OK.

Point #	X2	X3
1	0.04865	0.006
2	0.05465	0.006
3	0.04865	0.012
4	0.04265	0.006
5	0.04865	0.0
6	0.062	0.012
7	0.040	0.012
8	0.062	0.0
9	0.040	0.0

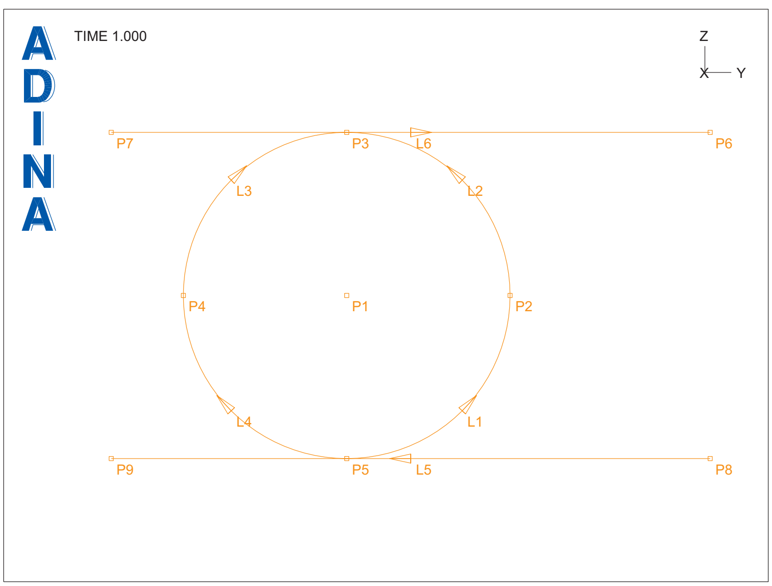
Click the Point Labels icon  to display the point numbers.

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

Line number	Type	Defined by	P1	P2	Center
1	Arc	P1, P2, Center	5	2	1
2	Arc	P1, P2, Center	2	3	1
3	Arc	P1, P2, Center	4	3	1
4	Arc	P1, P2, Center	5	4	1

Line number	Type	Point 1	Point 2
5	Straight	8	9
6	Straight	7	6

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



Geometry surfaces: Click the Define Surfaces icon , define the following surfaces and click OK:

Surface number	Type	Point 1	Point 2	Point 3	Point 4
1	Vertex	1	5	2	1
2	Vertex	1	2	3	1
3	Vertex	1	3	4	1
4	Vertex	1	4	5	1

Defining boundary conditions, constraint equations, loads

Boundary conditions: We will fix line 5. Click the Apply Fixity icon , change the "Apply to" field to Lines, enter line number 5 in the first row and column of the table, leave the fixity column blank and click Save.

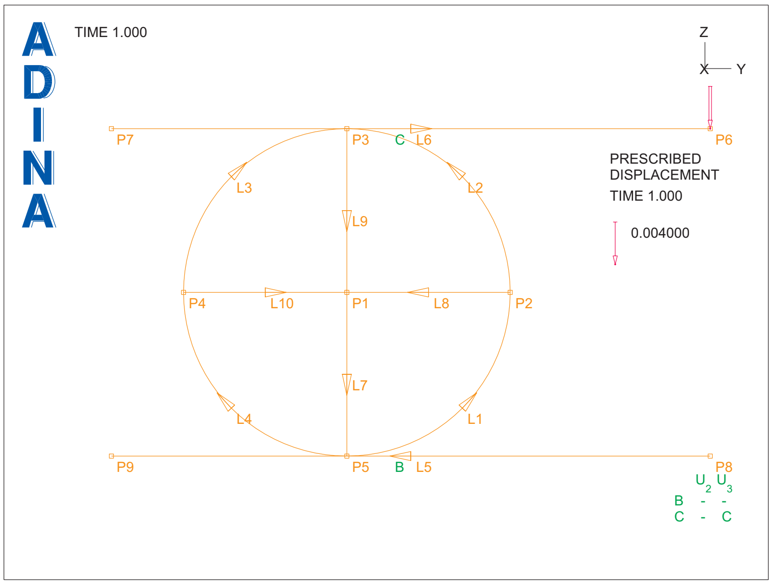
We will allow line 6 to move only in the Z direction. Here we need to define a corresponding fixity and then apply it to line 6. Click the Define... button. In the Define Fixity dialog box, add fixity name FIXY, check the Y-Translation button and click OK. In the Apply Fixity dialog box, enter line 6 and fixity FIXY in the second row of the table, then click OK.

Constraint equations: We will constrain line 6 to point 6 so that when point 6 moves in the Z direction, line 6 follows. Choose Model→Constraints→Constraint Equations and add constraint set 1. In the Slave box, set the Entity Type to Line, set the Entity # to 6 and set the Slave DOF to Z-Translation. In the first row of the table, set the Point # to 6 and the Master DOF to Z-Translation. Then click OK.

Loads: We will apply the load by moving the top contact surface downwards by the prescribed amount. Click the Apply Load icon , set the Load Type to Displacement and click the Define... button to the right of the Load Number field. In the Define Displacement dialog box, add displacement 1, set the Z field in the Prescribed Values of Translation box to -1.0 and click OK. In the Apply Load dialog box, make sure that the "Apply to" field is set to Point, then, in the first row of the table, set the Point # to 6, then click OK.

(Note, it is also possible to apply the prescribed displacement directly to the line. In that case, constraint equations are not necessary.)

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

The material of the O-ring is rubber, and we are given data points on the stress-strain curve. Therefore we use the curve fitting feature of the AUI to generate the material constants.

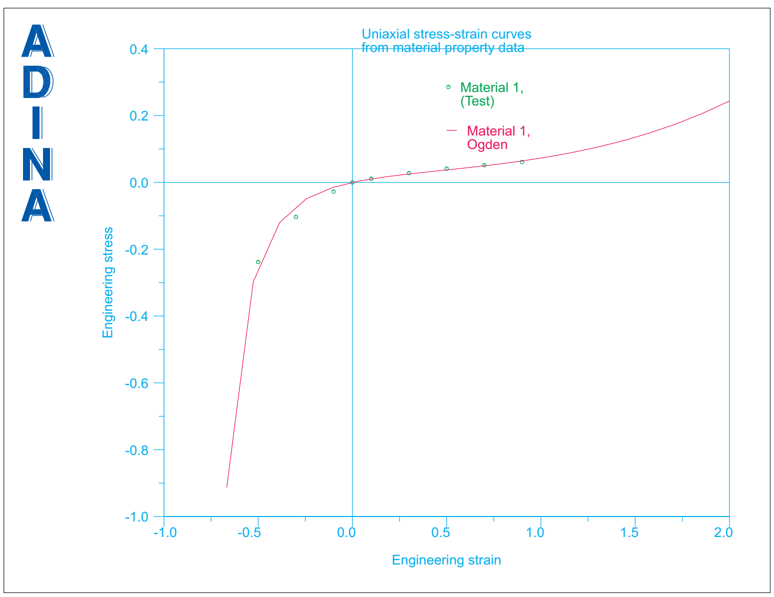
Click the Manage Materials icon M and click the Ogden button. Add material 1 and click the … button to the right of the Fitting Curve field. In the Define Fitting Curve dialog box, add Fitting Curve 1 and click the … button to the right of the Simple Tension Curve field. In the Define Stress-Strain2 Curve dialog box, add curve 1, enter the following stress-strain data points in the table (these points are repeated from the problem description for convenience) and click OK. (You can ignore the Strain2 column.)

Strain	Stress
-0.5	-0.2383
-0.3	-0.1035
-0.1	-0.0275
0.0	0.0
0.1	0.0111
0.3	0.0280
0.5	0.0409
0.7	0.0516
0.9	0.0610

In the Define Fitting Curve dialog box, set the Simple Tension Curve to 1 and click OK. In the Define Ogden Material dialog box, set the Fitting Curve to 1, and set the Bulk Modulus to 0. Now set Alpha 1= 1.3, Alpha 2 = 5.0, Alpha 3 = -2.0 and click Save.

The AUI performs a curve fit to determine the constants in the Ogden material model and fills in the Bulk Modulus, Mu1, Mu2 and Mu3 fields of the Define Ogden Material dialog box. The Bulk Modulus is 20.0911, Mu1 is 0.00741697, Mu2 is 0.00254953 and Mu3 is –0.0289875. The AUI also writes some information about the curve fit to the message window. Use the scrollbar of the message window to review the information (if necessary, choose View→Message Window to open the message window).

To display the stress-strain curve, click the Graph button in the Define Ogden Material dialog box. A new graphics window is displayed, that should look something like this:



Close the new graphics window. Click OK to close the Define Ogden Material dialog box and click Close to close the Manage Material Definitions dialog box.

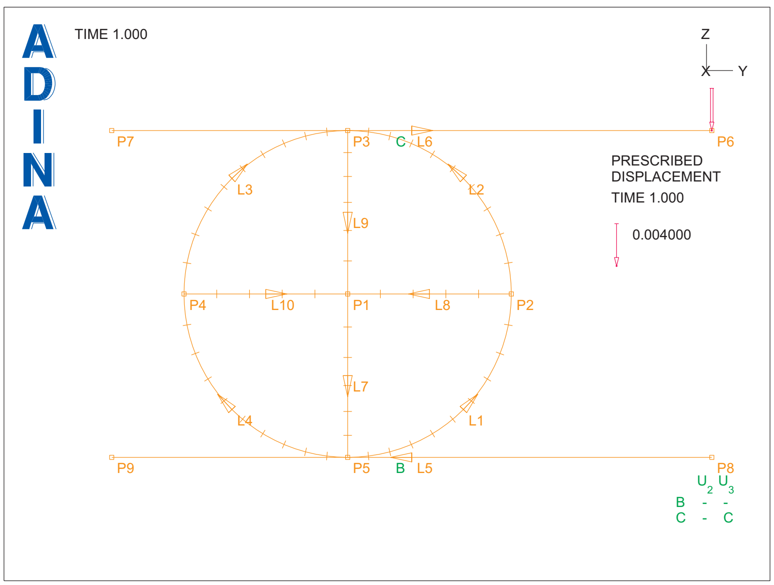
Defining subdivision data

We will use point-sizes to define the subdivision data. Choose Meshing→Mesh Density→ Complete Model, make sure that the "Subdivision Mode" is set to "Use End-Point Sizes" and click OK.

We enter the element density at five points in the O-ring. Choose Meshing→Mesh Density→ Point Size, and, in the table, set the mesh size for points 1 to 5 as follows, then click OK.

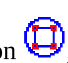
Point #	Mesh Size
1	0.0012
2	0.0012
3	0.0008
4	0.0012
5	0.0008

The graphics window should look something like this:

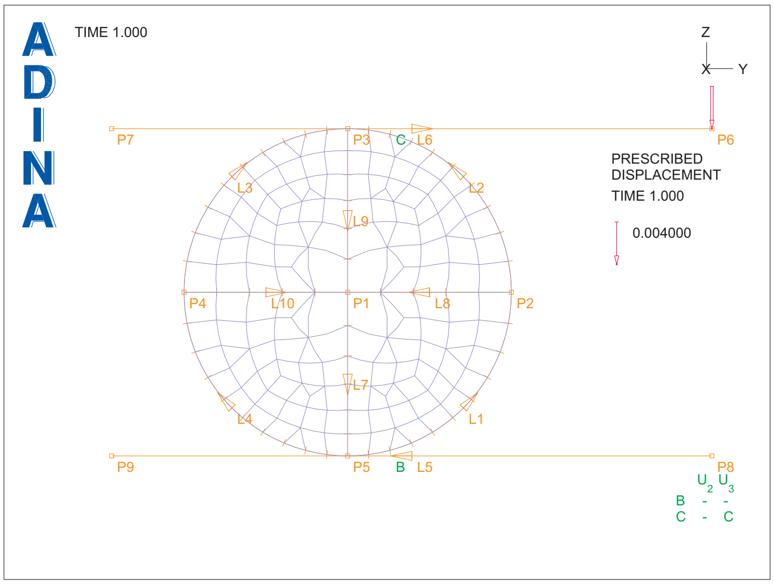


Notice that these short perpendicular lines are more closely spaced near points 3 and 5 and are more widely spaced near point 1. This is because the point size is smallest at points 3 and 5 and largest at point 1.

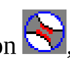
Defining the finite elements and nodes for the O-ring

Element group control data: Click the Define Element Groups icon , add element group 1, set the Type to 2-D Solid, verify that the Element Sub-Type is Axisymmetric and click OK.

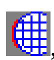
Finite elements: Click the Mesh Surfaces icon , enter 1, 2, 3, 4 in the first four rows of the table and click OK. The graphics window should look something like the figure on the next page.

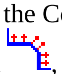


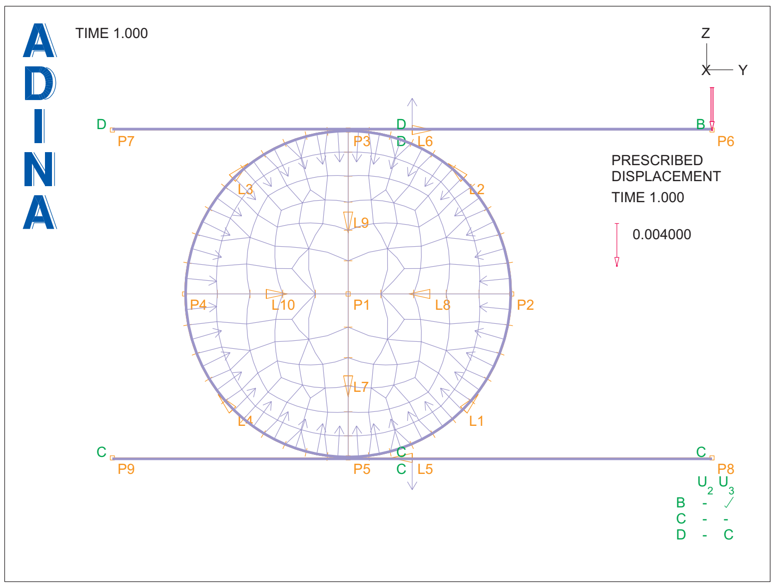
Defining the contact surfaces

Contact group control data: Click the Define Contact Groups icon , add contact group 1, and verify that the contact group type is 2-D Contact. Now click the Advanced tab, set the "Normal Contact w-Function Parameter" to 1E-15 and click OK.

Contact surfaces: We use three contact surfaces. Contact surface 1 surrounds the entire O-ring, contact surface 2 represents the lower plate and contact surface 3 represents the upper plate.

Click the Define Contact Surfaces icon , add contact surface number 1, enter the line numbers 1, 2, 3, 4 in the first column for the first four rows of the table, then click Save. Then add contact surface number 2, enter line 5 in the first column and row of the table and click Save. Define contact surface number 3 onto line 6 in the same way, then click OK.

We need to have nodes on contact surfaces 2 and 3. Click the Mesh Rigid Contact Surface icon , set the Contact Surface field to 2 and click Apply. This places one contact segment and three nodes onto contact surface 2. To place nodes and contact segments onto contact surface 3, set the Contact Surface field to 3 and click OK. When you click the Show Segment Normals icon , the graphics window should look something like the figure on the next page.



Contact pairs: To finish the contact modeling, we need to define which pairs of surfaces can come into contact and the associated friction coefficients. Contact surface 1 can contact both contact surface 2 and contact surface 3, so we have two contact pairs. In the first pair contact surface 1 is the contactor and contact surface 2 is the target, and in the second pair contact surface 1 is the contactor and contact surface 3 is the target.

Click the Define Contact Pairs icon and add contact pair 1. Set the Target Surface to 2, set the Contactor Surface to 1, verify that the Coulomb Friction Coefficient is 0.0 and click Save. Add contact pair 2, with contact surface 3 as the Target Surface and 1 as the Contactor Surface in the same way. Click OK.

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

Click the Save icon and save the database to file prob22. Click the Data File/Solution icon, set the file name to prob22, make sure that the Run Solution button is checked and click Save.

You will notice that ADINA uses the ATS method to obtain the solution as shown in the following table:

Current time	Time step size	Trial solution time	Result
0.0	1.0	1.0	No convergence
0.0	0.5	0.5	No convergence
0.0	0.25	0.25	Convergence
0.25	0.75	1.0	No convergence
0.25	0.375	0.625	Convergence
0.625	0.375	1.0	No convergence
0.625	0.1875	0.8125	Convergence
0.8125	0.1875	1.0	Convergence

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 prob22.

Obtaining a summary of model information

We will view a summary of the model. Choose List→Info→Model and read the information in the window that appears, using the scrollbars if necessary. There are 635 nodes, 1 element group and one contact surface group. In element group 1, there are 152 axisymmetric solid elements. Close this window.

To learn which solutions are loaded, choose List→Info→Response and read the information in the window that appears. There are two load steps loaded from times 0 to 1 (the first load step contains the initial conditions and the second contains the computed response). There are no mode shape results since this is not a frequency analysis. Close this window.

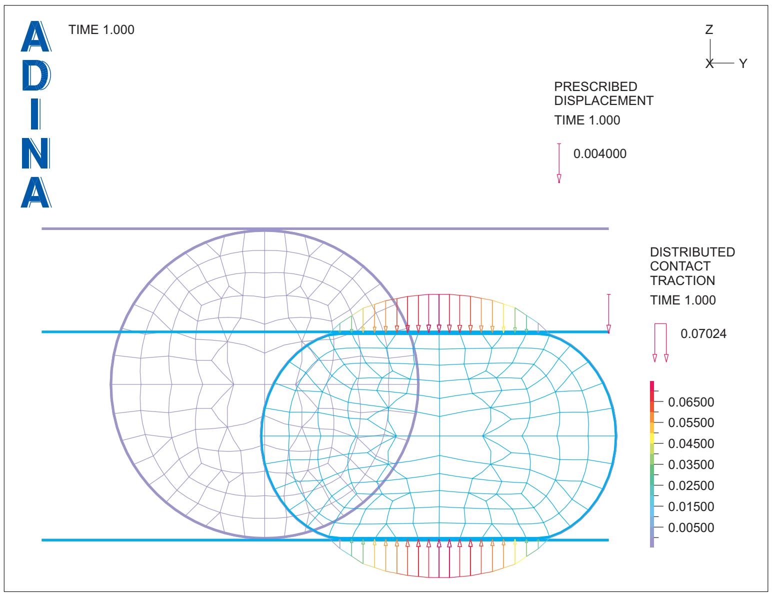
To learn which variables can be used in post-processing, choose List→Info→Variable and read the information in the window that appears.

Obtaining a deformed mesh plot with loads and contact forces

Click the Show Original Mesh icon and the Load Plot icon.

To add the distributed contact tractions to the mesh plot, choose Display→Reaction Plot→Create, set the Reaction Quantity to DISTRIBUTED_CONTACT_TRACTION and click OK.

Using the Pick icon and the mouse, shrink the mesh plot and move the annotations until the graphics window looks something like the figure on the next page.

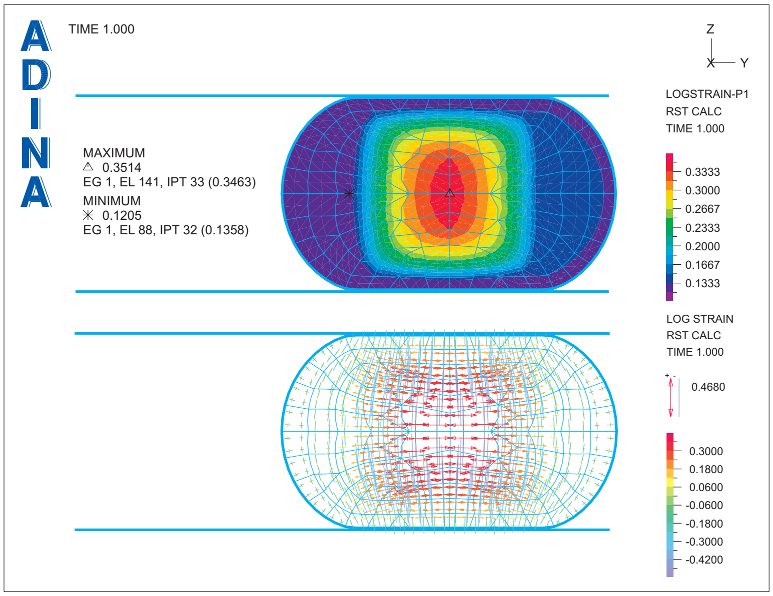


Plotting strains as bands and vectors

We will plot bands of the maximum principal strain. Click the Clear icon and the Mesh Plot icon, then click the Create Band Plot icon, choose variable (Strain:LOGSTRAIN-P1) and click OK. Move the mesh plot to the upper half of the graphics window.

Now we will add a strain vector plot to the graphics window. Click the Mesh Plot icon, then shrink the newly created mesh plot so that it is about the same size as the previous mesh plot and move it to a position directly under the previous mesh plot. Use the Pick icon and the mouse to remove the extra text and axes.

Click the Create Vector Plot icon, set the Vector Quantity to STRAIN, click OK, then move the vector table to a position to the right of the mesh plot. You may want to shrink the band table and vector table so that they do not overlap. The graphics window should look something like the figure on the next page.



We want to get a closer look at the strain vectors near the center of the O-ring. We could click the zoom icon and then select a zoom bounding box to enclose just the center of the O-ring, but then the strain vector table would not be visible and the strain vectors would be correspondingly enlarged.

So instead, click the Mesh Zoom icon  and then make a rubber-band box that encloses just the center of the O-ring. The graphics window should look something like the figure on the next page.

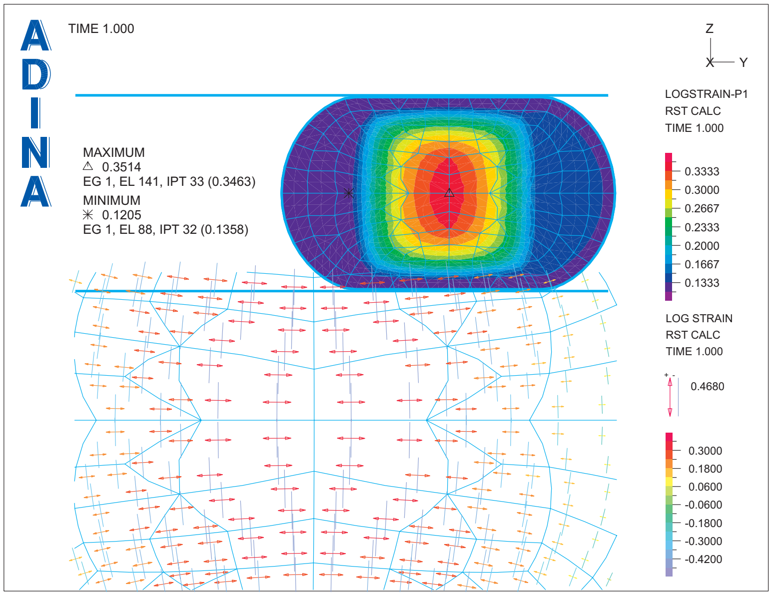
The AUI only enlarges the mesh plot enclosed in the rubber-band box and does not enlarge the strain vectors. To restore the original picture, click the Refit icon .

Making listings

We will list the strains within the model. Choose List→Value List→Zone. Set variable 1 to (Strain:LOGSTRAIN-P1), variable 2 to (Strain:LOGSTRAIN-P2) and variable 3 to (Strain: LOGSTRAIN-P3), then click Apply. Use the scrollbars to examine the listing.

Notice that the strains are output at the integration points. To get a listing with smoothed strains at the node points, set the Smoothing Technique field to AVERAGED and click Apply. The dialog box displays the new listing.

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


ADINA
TIME 1.000
Z
X
Y
LOGSTRAIN-P1
RST CALC
TIME 1.000
0.3333
0.3000
0.2667
0.2333
0.2000
0.1667
0.1333
MAXIMUM
△ 0.3514
EG 1, EL 141, IPT 33 (0.3463)
MINIMUM
✳ 0.1205
EG 1, EL 88, IPT 32 (0.1358)
LOG STRAIN
RST CALC
TIME 1.000
0.4680
0.3000
0.1800
0.0600
-0.0600
-0.1800
-0.3000
-0.4200