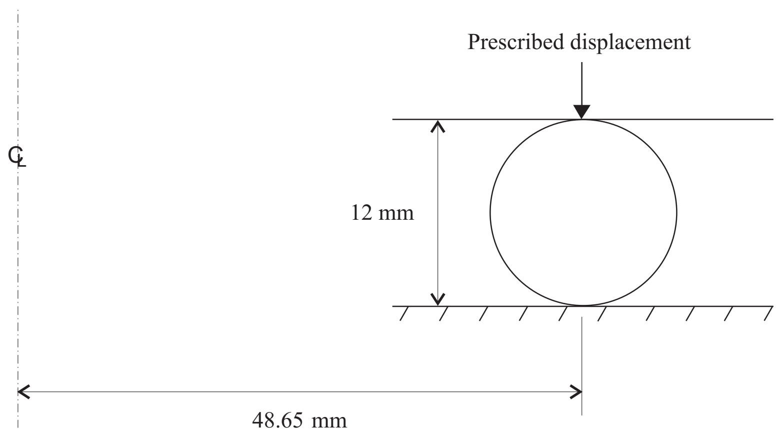
Problem description

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



This problem is in many ways similar to problem 22, and if you have not yet worked problem 22, you should work problem 22 first.

The geometry of the problem is the same as in problem 22.

The stress-strain-transverse strain characteristics for the material are as follows, when the material is subjected to uniaxial tension/compression:

Engineering strain (mm/mm)	Engineering transverse strain (mm/mm)	Engineering stress (N/mm^2)
-0.5	-0.1294	-0.15
-0.3	-0.0436	-0.09
-0.1	-0.0209	-0.04
0.0	0.0	0.0
0.1	0.0192	0.03
0.3	0.0539	0.05
0.5	0.0845	0.06

Notice that the transverse strain increases as the strain increases. This material is “auxetic”; that is, it has negative Poisson’s ratio.

We will perform three analyses:

1) Analysis assuming no viscoelastic effects. The material is modeled as a hyper-foam material.

2) Analysis assuming viscoelastic effects, no temperature effects. The Holzapfel finite strain viscoelasticity model is used, with a single viscoelastic chain. The material constants for the chain are $\beta = 2.5$, $\tau = 0.5$, and the usage flag is set to “combined” (that is, the viscoelasticity is based on the total strain energy density).

3) Analysis assuming temperature-dependent viscoelastic effects. We will assume that the material is thermorheologically simple (such a material is called a TRS material). In a TRS material, all material properties are temperature-independent, however, the viscoelasticity follows the time-temperature superposition principle. We use the shift function

$$\log_{10} a_T({}^t\theta) = -\frac{C_1\left({}^t\theta - \theta_{ref}\right)}{C_2 + {}^t\theta - \theta_{ref}}$$

in which $a_T({}^t\theta)$ gives the relationship between the actual time t and the reduced time ζ through $\dfrac{d\zeta}{dt} = \dfrac{1}{a_T({}^t\theta)}$. We will assume $C_1 = 10.86$, $C_2 = 104.8$, measured at the reference temperature $\theta_{ref} = 25$ degrees C.

As in problem 22, we will use an axisymmetric analysis.

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

- Defining a hyper-foam material
- Controlling the ATS method
- Adding viscoelastic effects to a hyper-foam material
- Specifying TRS temperature dependence for a hyper-foam material

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.

Note that you must have an ADINA-M/PS license to do this problem.

This problem cannot be solved with the 900 nodes version of the ADINA System because the 900 nodes version does not contain ADINA-M/PS.

Much of the input for this problem is stored in files `prob35_1.in`, `prob35_2.in`, `prob35_1.plo` and `prob35_2.plo`. You need to copy files `prob35_1.in`, `prob35_2.in`, `prob35_1.plo`, `prob35_2.plo` from the folder `samples\primer` into a working directory or folder before beginning this analysis.

Analysis without viscoelastic effects

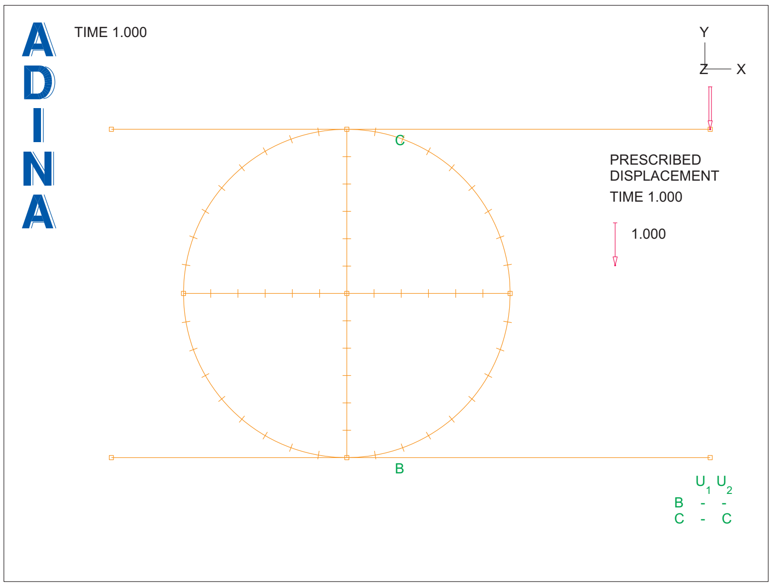
Invoke the AUI and set the Program Module drop-down list to ADINA Structures.

Defining model control data, geometry, subdivision data, boundary conditions, constraint equations and displacement loads

We have prepared a batch file (`prob35_1.in`) that performs the following operations:

- ► Sets the 2D plane to the X-Y plane, with Y as the axisymmetric axis
- ► Specifies that stiffness matrix stabilization should be used.
- ► Defines points, lines and surfaces.
- ► Subdivides the surfaces.
- ► Defines boundary conditions.
- ► Defines a constraint equation set
- ► Defines a displacement load and applies it to the model.
- ► Plots the model

Choose File→Open Batch, navigate to the working directory or folder, select the file `prob35_1.in` and click Open. The graphics window should look something like the figure on the next page.



Defining the hyper-foam material

Click the Manage Materials icon and click the Hyper-Foam 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, define it as

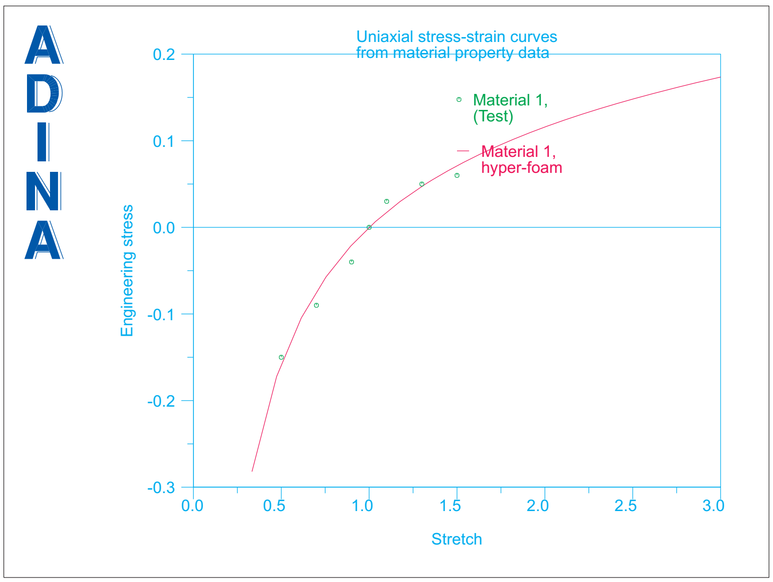
Strain	Stress	Strain2
-0.5	-0.15	-0.1294
-0.3	-0.09	-0.0436
-0.1	-0.04	-0.0209
0.0	0.0	0.0
0.1	0.03	0.0192
0.3	0.05	0.0539
0.5	0.06	0.0845

and click OK. Notice that "Strain2" is the transverse strain.

In the Define Fitting Curve dialog box, set the Approximation Order to 1, set the Simple Tension Curve to 1 and click OK.

In the Define Hyper-Foam Material dialog box, set the Fitting Curve to 1 and click Save. Notice that the value of MU(1) is set to 0.17907871 (N/mm^2), the value of ALPHA(1) is set to 1.31045910 and the value of BETA(1) is set to –0.1370987. More information about the curve fit is displayed in the Message Window.

Click the Graph button to display the curve fit. The AUI should display a new graphics window that looks something like this:



Close the new graphics window.

Click OK to close the Define Hyper-Foam Material dialog box and click Close to close the Manage Material Definitions dialog box.

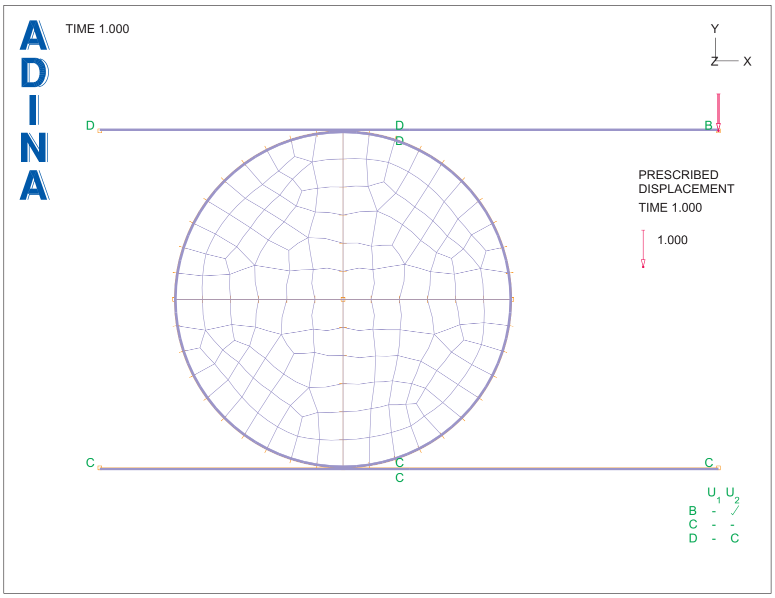
Defining the element group, meshing the geometry, defining the contact surfaces

We have prepared a batch file (`prob35_2.in`) that performs the following operations:

- ► Defines the element group
- ► Meshes the geometry (using 9-node elements generated using free-form meshing)
- ► Defines the contact group
- ► Defines contact surfaces
- ► Defines contact pairs
- ► Regenerates the graphics

Choose File→Open Batch, navigate to the working directory or folder, select the file `prob35_2.in` and click Open. The AUI processes the commands in the batch file.

The graphics window should look something like this:

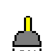


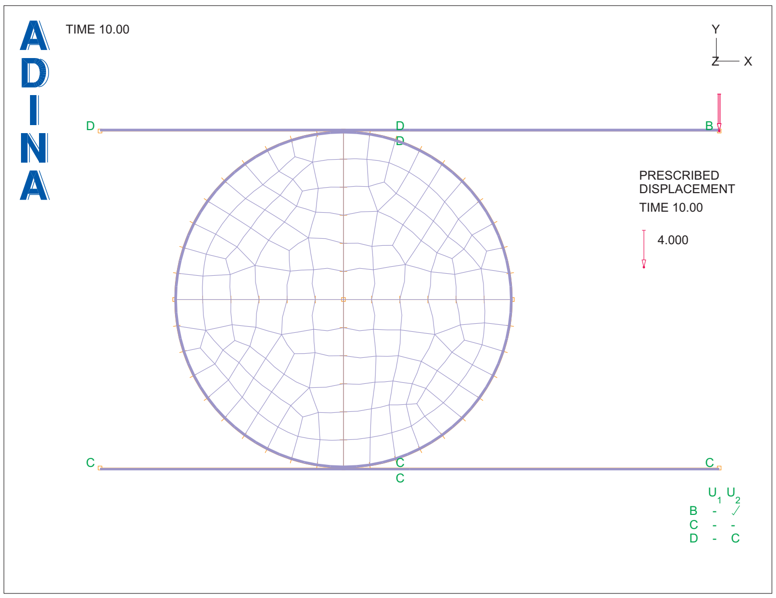
Defining the load steps

In the first run, we want to move the top plate 4 mm downwards in 10 equal steps. Choose Control→Time Step, set the Number of Steps to 10 in the first row of the table and click OK. Choose Control→Time Function, define time function 1 to be

Time	Value
0.0	0.0
10.0	4.0

and click OK.

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



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

Click the Save icon and save the database to file `prob35`. Click the Data File/Solution icon, set the file name to `prob35`, make sure that the Run Solution button is checked and click Save.

You will notice that the times at which ADINA Structures obtains the solutions, and the step sizes, are as shown in the table on the next page. The step sizes are different than the step sizes that we requested because the ATS method is in use. We would like to use the ATS method, but not have the ATS method use larger steps than the steps that we requested.

Close all open dialog boxes. Click the Analysis Options icon, click the … button to the right of the “Use Automatic Time Stepping (ATS)” field, set the “For Next Time Step” field to “Return to Original Time Step Specified” and click OK twice to close both dialog boxes.

Click the Save icon, click the Data File/Solution icon, set the file name to `prob35`, make sure that the Run Solution button is checked and click Save.

Now the ATS method cuts back the time step for solution step 10. But the solution is obtained for all of our original time steps.

Step number	Step size	Time
1	1.0	1.0
2	1.0	2.0
3	1.0	3.0
4	1.5	4.5
5	1.0	5.5
6	1.0	6.5
6	0.5	6.0
6	1.0	7.0
7	1.0	8.0
8	1.0	9.0
9	1.0	10.0
9	0.5	9.5
9	0.5	10.0

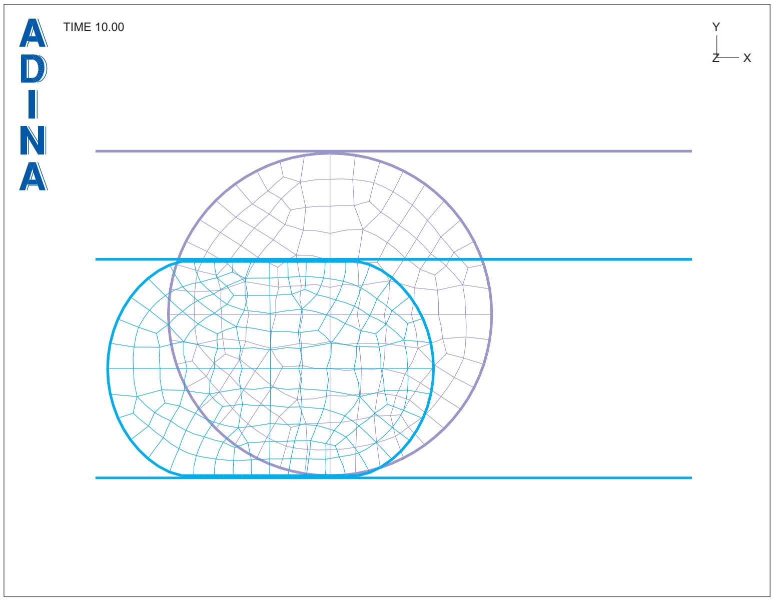
When ADINA Structures is finished, close all open dialog boxes. Set the Program Module drop-down list to Post-Processing (you can discard all changes), click the Open icon and open porthole file `prob35`.

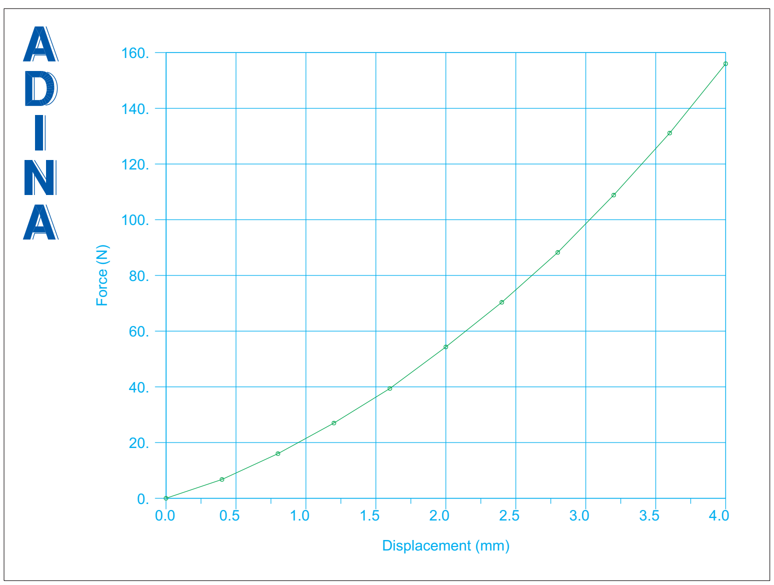
Post-processing

Click the Show Original Mesh icon and use the Pick icon and the mouse to resize the graphics. The graphics window should look something like the top figure on the next page.

Notice that the deformed mesh moves to the left (that is, towards the centerline). This is because of the negative Poisson's ratio; compression in the vertical (y) direction causes contraction of the material fibers in the hoop (z) and horizontal (x) directions.

We want to plot the force-deflection curve. We have put the necessary commands in a batch file (`prob35_1.plo`). Choose File→Open Batch, navigate to the working directory or folder, select the file `prob35_1.plo` and click Open. The AUI processes the commands in the batch file. The graphics window should look something like the bottom figure on the next page.





In this plot, we have multiplied the force by 2π, and reversed the direction of the force and displacement so that downwards forces and displacements are positive. We have also requested gridlines in the plot.

Choose Graph→List and scroll to the bottom of the list. The value of YF for YD=4.0000E+00 is 1.55996E+02 (N) (you might need to use the horizontal scrollbar to see this value).

Analysis with viscoelastic effects

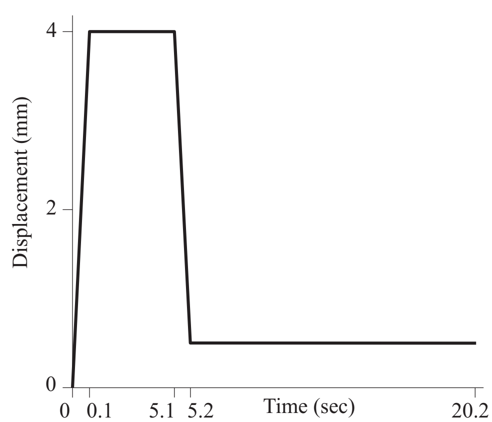
Now we will rerun the model including viscoelastic effects.

Set the Program Module drop-down list to ADINA Structures (you can discard all changes) and choose database file `prob35.idb` from the recent file list near the bottom of the File menu. In the Model Tree, click the + next to the 'Material' text, right-click on '1. Hyper-Foam' and choose Modify. In the Define Hyper-Foam Material dialog box, click the … button to the right of the Viscoelastic field. In the Define Viscoelastic Effect for Rubber Material dialog box, add viscoelastic effect 1, make sure that the Type is Holzapfel, and, in the first row of the table, set Beta to 2.5, Tau to 0.5, Usage to Combined, then click OK. In the Define Hyper-Foam Material dialog box, set the Viscoelastic field to 1, then click OK.

Time stepping

Since the model is viscoelastic, the solution response is time dependent. Therefore we must choose the time step sizes with reference to the material time dependence. Since Tau = 0.5, the material time constant is 0.5 (seconds); therefore if we load to 4 mm in 0.1 seconds, the material will not have time to relax during the loading.

After the initial loading, we want to hold the displacement of the top plate constant, then partially unload the O-ring, as shown:



Choose Control→Time Function, edit time function 1 to be

Time	Value
0.0	0.0
0.1	4.0
5.1	4.0
5.2	0.5
20.2	0.5

and click OK. Now choose Control→Time Step, edit the table to be

Number of Steps	Magnitude
10	0.01
20	0.25
10	0.01
60	0.25

and click OK.

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

Click the Save icon. Click the Data File/Solution icon, set the file name to `prob35b`, make sure that the Run Solution button is checked and click Save.

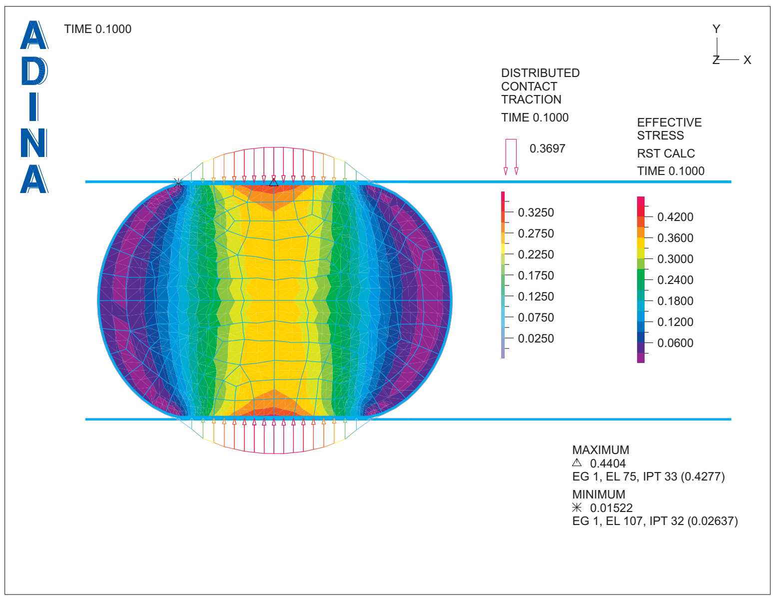
When ADINA Structures is finished, close all open dialog boxes. Set the Program Module drop-down list to Post-Processing (you can discard all changes), click the Open icon and open porthole file `prob35b`.

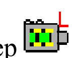
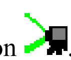
Post-processing

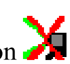
Click the Movie Load Step icon, then the Animate icon. Notice that the when the top plate moves upwards, the O-ring loses contact with the top plate.

Click the Refresh icon to clear the animation, then click the First Solution icon and click the Next Solution icon 9 times, until the solution time is 0.1. Now click the Model Outline icon, click the Quick Band Plot icon, click the Create Reaction Plot icon, set the Reaction Quantity to DISTRIBUTED_CONTACT_TRACTION and click OK.

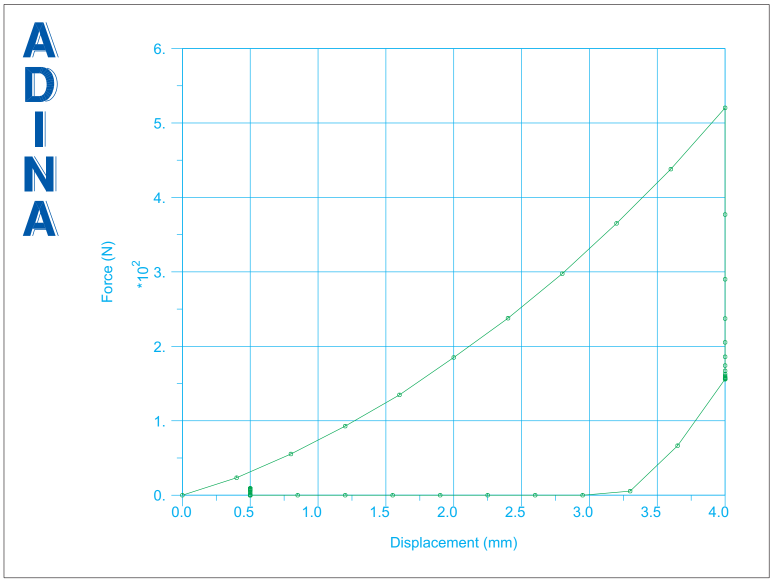
Use the mouse to rearrange the graphics until the graphics window looks something like this:



Now click the Movie Load Step  icon, then the Animate icon . As the top plate moves downwards, the stresses and contact forces increase. Then, when the top plate remains stationary, the stresses and contact forces decrease and eventually reach a relaxed state. When the top plate moves upwards, the stresses and contact forces drop to zero, and when the material relaxes and recontacts the top plate, the stresses and contact forces increase. However, since the deformation at the end of the solution is much less than the deformation corresponding to the stress and contact force scaling, there is no visual indication of the stresses and contact forces at the end of the solution.

Click the Refresh icon  to clear the animation.

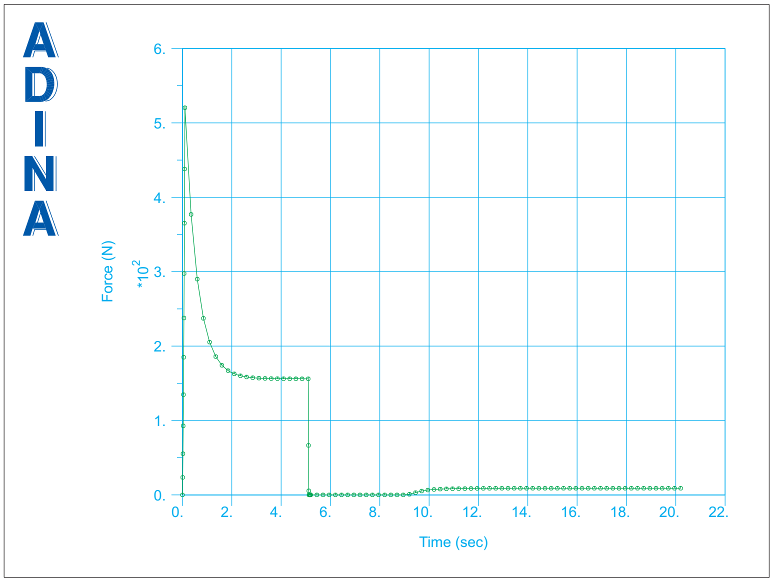
Now let's plot the force-deflection curve. We can use the same batch file that we previously used. Choose File→Open Batch, navigate to the working directory or folder, select the file `prob35_1.plo` and click Open. The graphics window should look something like the figure on the next page.



Use Graph→List to look at the numerical values in the graph. During initial loading, the force reaches 5.20331E+02, then when the top plate is held constant, the force decreases to 1.56014E+02 N (nearly the same value as was obtained in the analysis without viscoelastic effects). When the top plate is moved upwards, the force drops to zero, then after recontact, the force increases to 8.91276E+00 N.

Lets plot the force time history. We have set up the necessary plotting commands in file `prob35_2.plo`. Choose File→Open Batch, navigate to the working directory or folder, select the file `prob35_2.plo` and click Open. The graphics window should look something like the figure on the next page.

The relaxation process is clearly visible between 0.1 and 5 seconds.



Analysis with temperature-dependent viscoelastic effects

Now we will rerun the model including temperature-dependent viscoelastic effects.

Set the Program Module drop-down list to ADINA Structures (you can discard all changes) and choose database file `prob35.idb` from the recent file list near the bottom of the File menu.

In this analysis, we will set the temperature of the material to 15.0 degrees C. Choose Control→Analysis Assumptions→Default Temperature Settings, set the Initial Temperature to 15.0, set the Prescribed Temperature to 15.0 and click OK.

In the Model Tree, click the + next to the 'Material' text, right-click on '1. Hyper-Foam' and choose Modify. In the Define Hyper-Foam Material dialog box, set the Temperature Dependence to TRS, set the Reference Temperature to 25.0 and click the … button in the Temperature Dependence box. In the Define Temperature-Dependent Rubber Material Properties dialog box, add Rubber Table 1, set the Type to TRS, edit the table to be

Temperature	Thermal Expansion Coef
0.0	0.0
100.0	0.0

and click OK. In the Define Hyper-Foam Material dialog box, set the Table to 1 and click

Save (we do not want to close the dialog box yet). Now click the … button to the right of the Viscoelastic field, set 'Use Shift Function' to 'WLF (Williams-Landel-Ferry)', set Constant C1 to 10.86, set Constant C2 to 104.8 and click OK to close both dialog boxes.

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

Click the Save icon . Click the Data File/Solution icon , set the file name to `prob35c`, make sure that the Run Solution button is checked and click Save.

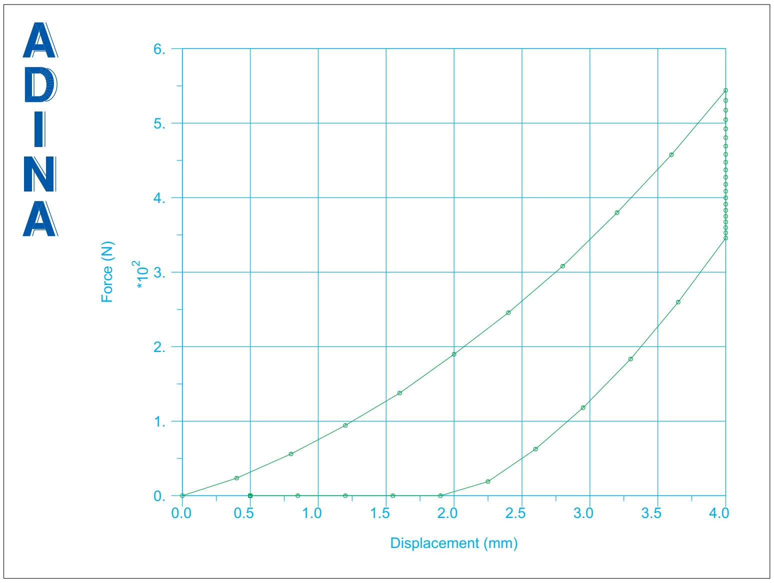
When ADINA Structures is finished, close all open dialog boxes. Set the Program Module drop-down list to Post-Processing (you can discard all changes), click the Open icon and open porthole file `prob35c`.

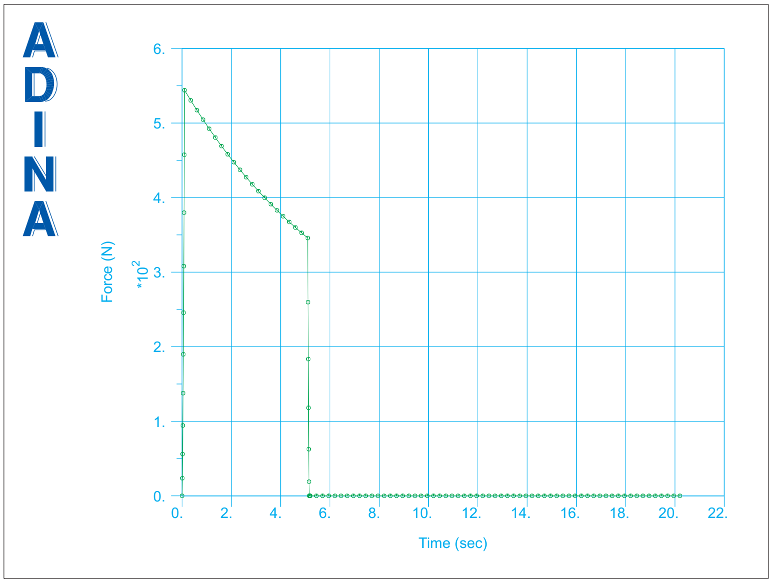
Post-processing

Click the Movie Load Step icon, then the Animate icon . Notice that the material relaxes much more slowly compared with the previous analysis. Click the Refresh icon to clear the animation.

You can plot the stresses and contact forces just as in the previous analysis. You can also plot the force-deflection curve and time history curve. The plots should look something like the figures on the next page.

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


ADINA
Force (N)
*10^2
6.
5.
4.
3.
2.
1.
0.
0.0
0.5
1.0
1.5
2.0
2.5
3.0
3.5
4.0
Displacement (mm)


ADINA
Force (N)
*10^2
6.
5.
4.
3.
2.
1.
0.
0.
2.
4.
6.
8.
10.
12.
14.
16.
18.
20.
22.
Time (sec)