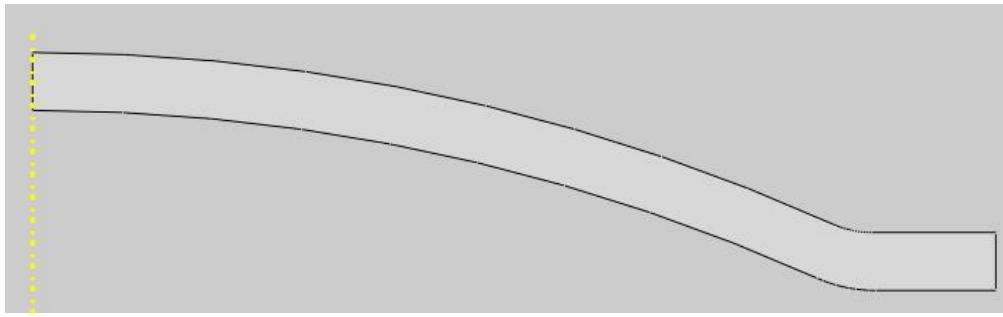
Exercise 21

In this exercise we will analyze a rubber disc subjected to pressure loading. The disc is modeled as an axisymmetric part. The geometry considered is shown in the figure below.



Pick **File > Set Work Directory** and set the work directory to the **DiscFlange** folder

Open the model database **DiscAxi.cae**.

The model consists of a single part, Disc. The disc shown in above figure has an outer radius of 0.1 m and a thickness of 0.005 m. It is assumed to be made from rubber, which is modeled as a Mooney-Rivlin material with C_{10}= 0.5 MPa and C_{01} = 0.14 MPa. The disc is meshed with bilinear axisymmetric quadrilateral elements with reduced integration and hybrid formulation (CAX4RH). Hybrid elements are used because the material is nearly incompressible material. The disc is loaded with a uniform pressure of 200 kPa.

⇨ Defining Step

We assume that load is applied slowly such that inertia effects can be neglected. So analysis will be performed using the Static, General procedure.

The step has already been defined with a total time period set to **1.0**. As the large deformations are expected to occur, Nlgeom option has been set to On. The incrementation has been set to **Automatic** type with an initial increment size of **0.01** and a maximum increment size of **0.05**

⇨ Defining Load

A uniform pressure load is applied across the bottom of the disc. The load has already been defined.

⇨ Boundary Conditions

Boundary conditions required for the analysis have already been defined.

The boundary condition “BC-Fixed” constrains all the degrees of freedom of the bottom edge of the disc. The boundary condition “BC-Symm” applies a symmetric constraint to the edge at the center of the disc.

Field Output Request

We will plot the history of applied pressure during postprocessing. This information can be plotted by requesting P, Pressure loads output variable.

Change to **Step** module.

Open the Field Output Manager by picking .

Select the F-Output-1 field and pick **Edit** as shown below.

Check the **P, Pressure loads** variable (located under the **Forces/Reactions** container).

- [x] Forces/Reactions
  - [x] RF, Reaction forces and moments
  - [ ] RT, Reaction forces
  - [ ] RM, Reaction moments
  - [x] CF, Concentrated forces and moments
  - [ ] SF, Section forces and moments
  - [ ] TF, Total forces and moments
  - [ ] VF, Viscous forces and moments due to static stabilization
  - [ ] ESF1, Effective axial force for beams and pipes subjected to pressure loading
  - [ ] NFORC, Nodal forces due to element stresses
  - [ ] NFORCSO, Nodal forces in beam section orientation
  - [ ] RBFOR, Force in rebar
  - [ ] BF, Body forces
  - [ ] CORIOMAG, Magnitude of Coriolis loads
  - [ ] ROTAMAG, Magnitude of rotary acceleration loads
  - [ ] CENTMAG, Magnitude of centrifugal loads (rho times omega squared)
  - [ ] CENTRIFMAG, Magnitude of centrifugal loads (omega squared)
  - [ ] GRAV, Uniformly distributed gravity loads
  - [x] P, Pressure loads
  - [ ] HP, Hydrostatic pressure loads

Pick **OK** to apply and exit.

Pick **Dismiss** to close the manager.

Defining Material Model

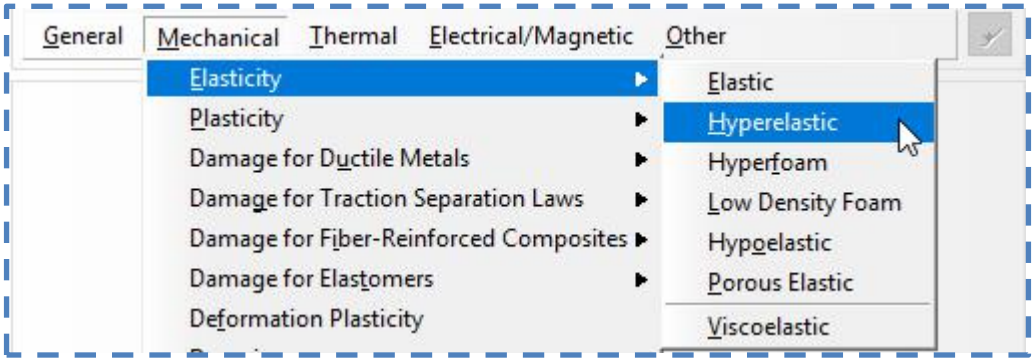
A material has already been defined, we will edit its definition to add hyperelastic properties.

Change to **Property** module.

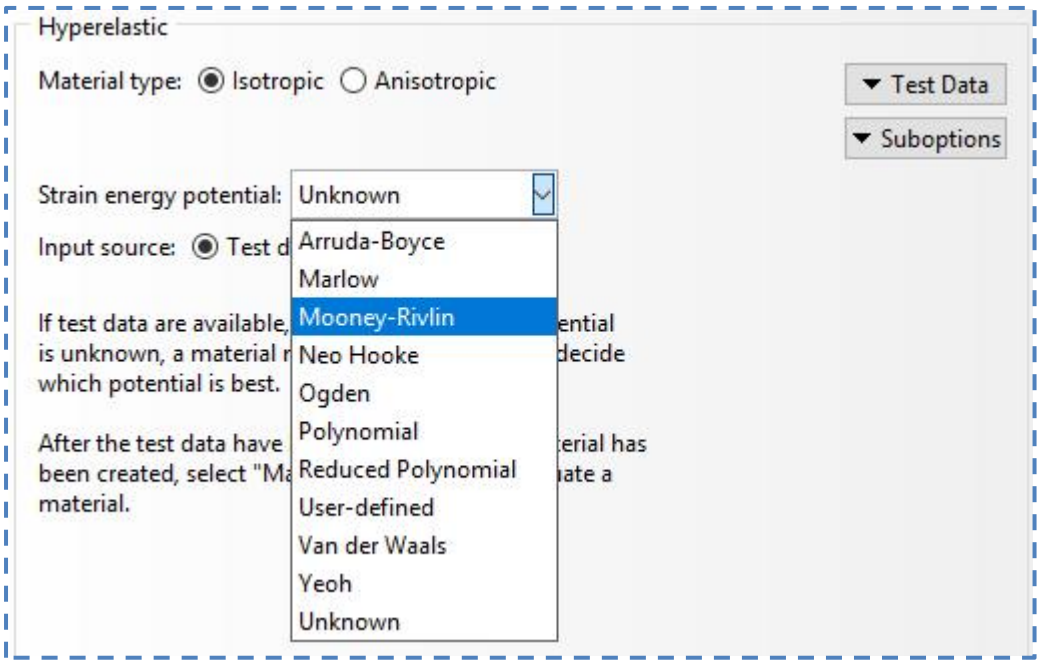
Open the Material Manager by picking .

Select the Rubber and pick **Edit**.

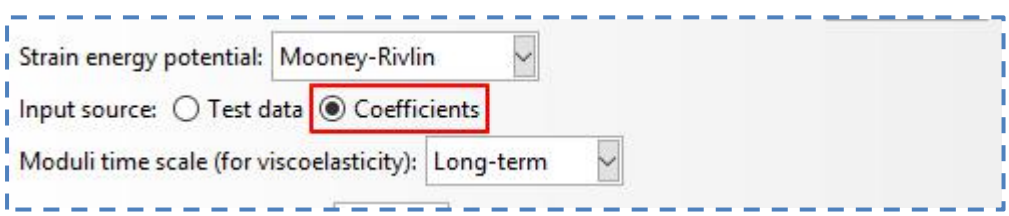
In the Edit Material dialog box, pick **Elasticity > Hyperelastic** in the Mechanical menu.



Pick Mooney-Rivlin as strain energy potential.



Select **Coefficients** as input source.



Enter C10 = 0.5E6 and C01 = 0.14E6 as shown below.

Pick **OK** to apply and exit.

Job Submission

All the information required for analysis has been set up. Now we can submit the job for analysis.

So change to **Job** module and pick  to open the job manager.

Pick **Create** and create a job named Bulge or whatever name you would like.

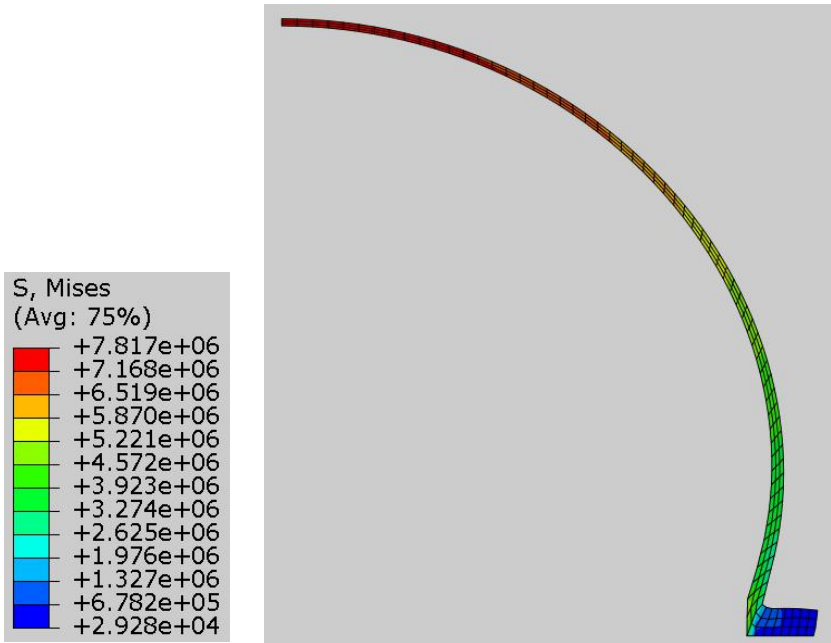
Pick **Continue** and then **OK**.

Pick **Submit** to submit the job for analysis.

Pick **Results** to view the results in Visualization module.

Postprocessing

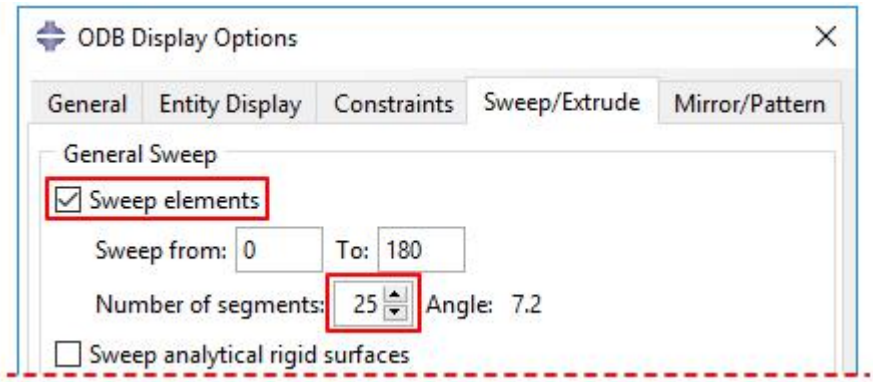
Pick  to plot contours on deformed shape. The contour plot of von Mises stress will appear as shown below.



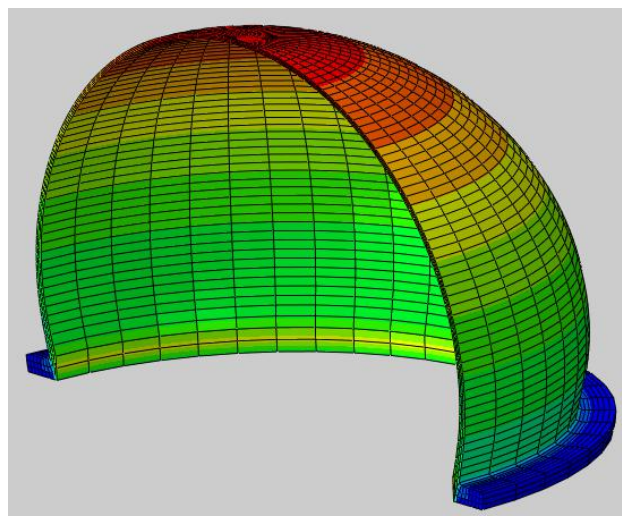
We can view the axisymmetric simulation results in three dimensions by sweeping them about the rotational axis to a specific angle. This give a three-dimensional visual effect. So first we will sweep the simulation results for better understanding.

While in the Visualization module, pick **View > ODB Display Options** from the main menu bar and check the "Sweep elements" option located under the Sweep/Extrude tab.

Enter **25** as the number of segments for a finer view of the model.



Pick **OK** to apply and exit the dialog box. The 180^0 representation of simulation results will appear as shown below.



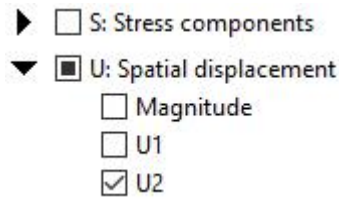
It can be seen that as the disc bulges, thickness of the disc decreases significantly in the center of the disc.

Now we will plot the thickness of the disc at the centerline .

Pick  and Create XY Data dialog box will appear.

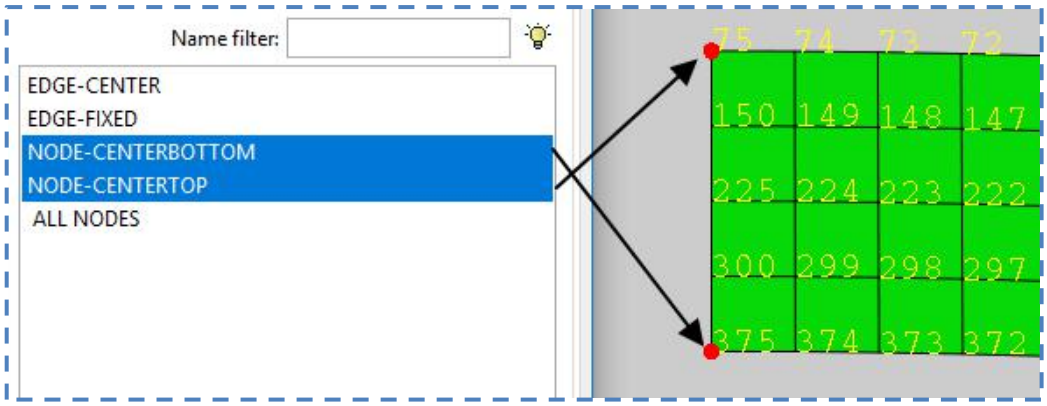
Pick **ODB field output > Continue**

In the Position field, select **Unique Nodal** and check the **U2** checkbox.



Under the Elements/Nodes tab, pick the **Node sets** as method and select the **NODE_CENTERBOTTOM** and **NODE_CENTERTOP.**

Check **Highlight items in viewport** in the dialog box to highlight the nodes in viewport. The following figure also shows the node labels for later reference.



Pick **Save** to save the plot and then pick **OK** to confirm.

The current thickness can be computed by using the following relation

Current thickness = Undeformed thickness - (Displacement of Bottom Node - Displacement of top node)

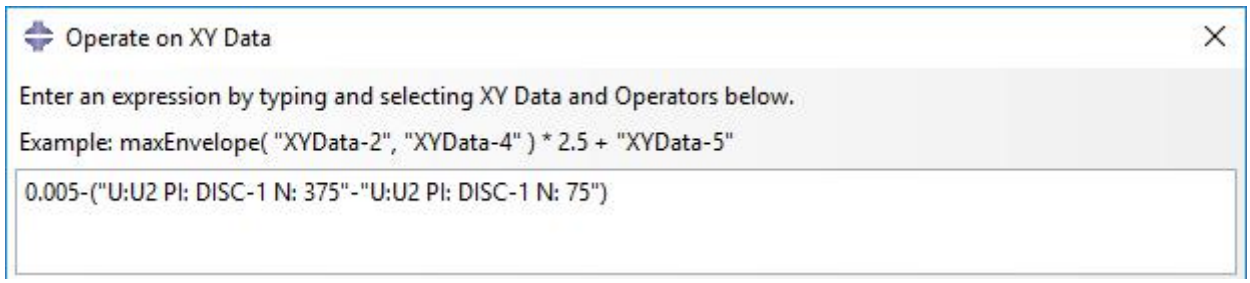
So we need to operate on the the displacement plots to obtain current thickness plot.

Again pick  and select **Operate on XY data** option.

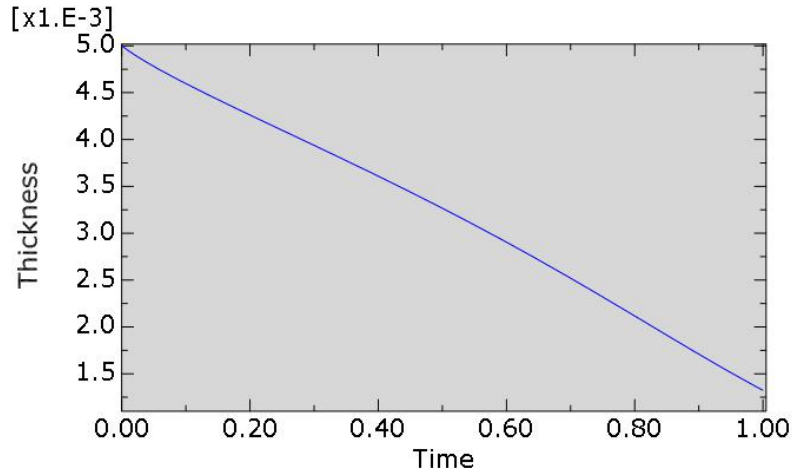
Pick **Continue** to proceed.

Enter **0.005** in the expression field. Remember this is the undeformed thickness of disc.

Then complete the expression as shown below. Notice the "-" signs according to the above relation.



Pick **Plot Expression** tab and graph will appear as shown below (after modifying the title of vertical axis).



It can be seen that thickness reduces to about 0.0013 m.

As the plot looks satisfactory so pick **Save As** to save the plot and enter **CurrentThickness** as name.

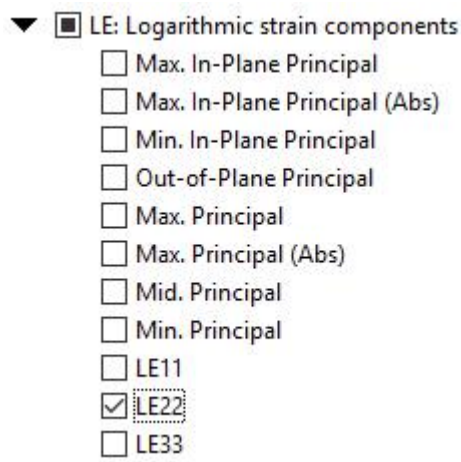
Now we will plot the thickness strain at the center of the disc versus the displacement of the center node along axis.

First we will plot the thickness strain at the centerline .

Pick  and Create XY Data dialog box will appear.

Pick **ODB field output > Continue**

In the Position field, select **Unique Nodal** and check the **LE22** checkbox.



Under the Elements/Nodes tab, pick the **Node sets** as method and select the **NODE_CENTERBOTTOM.**

Pick **Save** to save the plot and then pick **OK** to confirm.

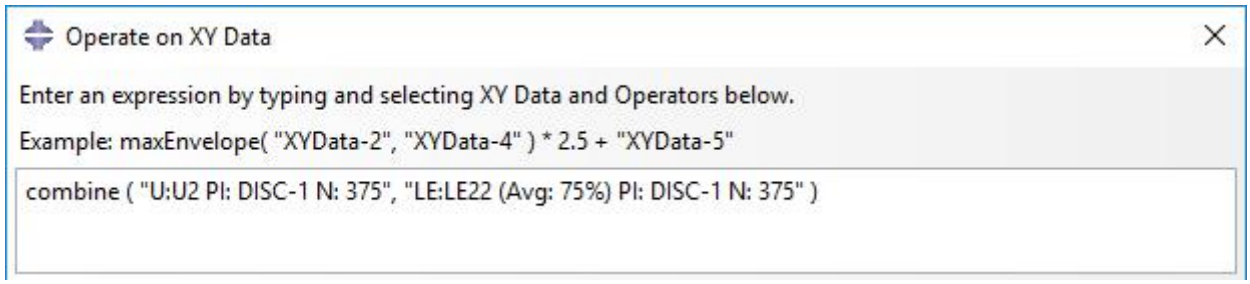
Now we will combine the plots to produce the desired strain-displacement plot.

Again pick  and select **Operate on XY data** option.

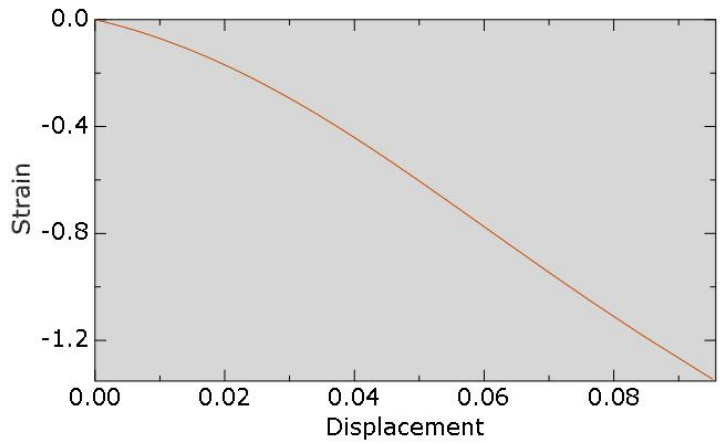
Pick **Continue** to proceed.

In the Operate on XY Data dialog box, pick **combine(X,X)** operator.

Then double-click the "U:U2……." data and then "LE:LE22……." data in the list and expression will appear as shown below.



Pick **Plot Expression** tab and graph will appear as shown below.



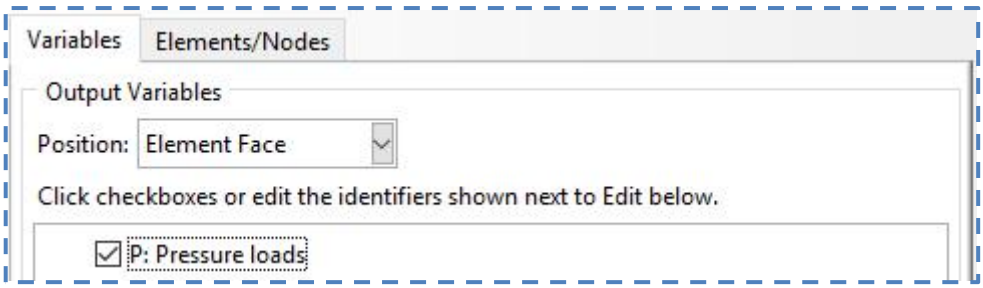
Now we will plot the applied pressure versus the displacement of the center node.

First we will plot the history of applied pressure.

Pick  and Create XY Data dialog box will appear.

Pick **ODB field output > Continue**

In the Position field, select **Element Face** and check the **P:Pressure loads** checkbox.



In the Elements/Nodes tab, pick **Element sets** as method and select the **ELEMENT_CENTER.** This element has already been defined for the ease of use.

Pick **Save** to save the plot and then pick **OK** to confirm.

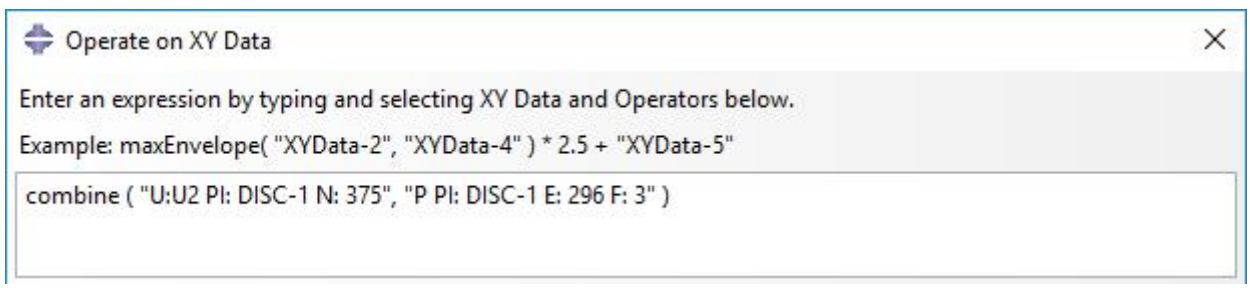
Now we will combine the plots to produce the desired pressure-displacement plot.

Again pick  and select **Operate on XY data** option.

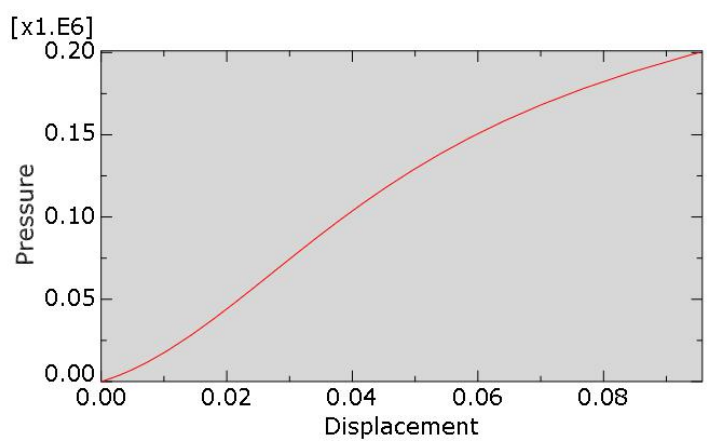
Pick **Continue** to proceed.

In the Operate on XY Data dialog box, pick **combine(X,X)** operator.

Then double-click the "U:U2……." data and then "P P1:……." data in the list and expression will appear as shown below.



Pick **Plot Expression** tab and graph will appear as shown below.

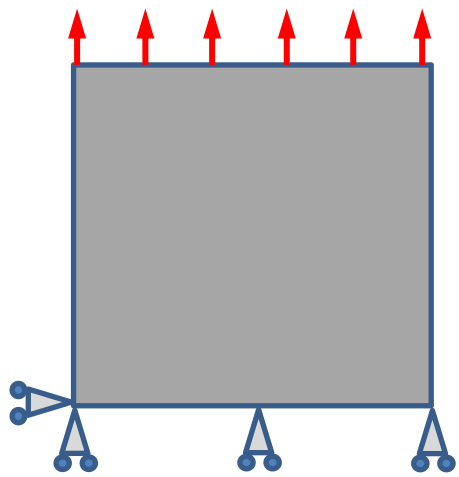


Such graphs can be useful to compare the results to physical test.

Select **File > Save** to save the changes we made so for.

Exercise 22

In this exercise we will analyze a viscoelastic sheet subjected to constant pressure load. The sheet is modeled assuming plane stress conditions. The material is assumed linear viscoelastic. The geometry considered is shown in the figure below.



Pick **File > Set Work Directory** and set the work directory to the **SheetVisco** folder

Open the model database **Sheet.cae**.

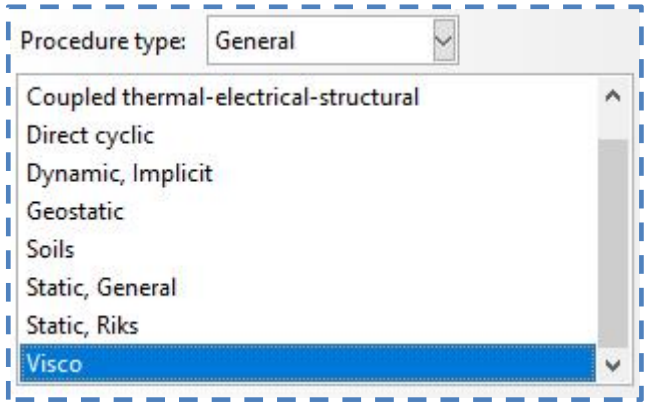
The model consists of a single part, Sheet. It is assumed to be made from polyethylene, which is modeled as a linear elastic material with a Young's modulus of 1.08 GPa and a Poisson's ratio of 0.4. It is meshed using second order plane stress quadrilateral elements with reduced integration (CPS8R). The sheet is loaded with a pressure of 10 MN/m.

Defining Step

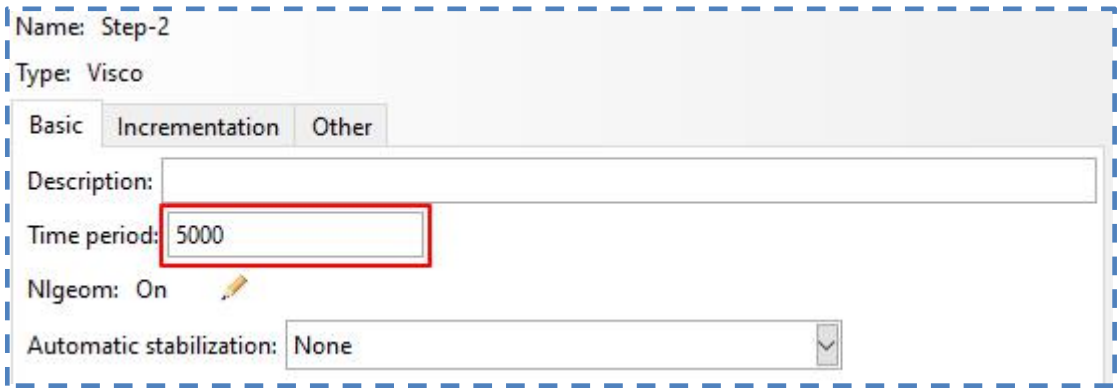
The analysis will be performed in two steps. The load is applied in the first step. We assume that load is applied slowly such that inertia effects can be neglected. So analysis will be performed using the Static, General procedure. This step has already been defined. The static procedure does not allow viscous material behavior so the response is purely elastic during first step. To simulate the viscoelastic response a second step using the quasi-static procedure (Visco procedure) will be defined. In this step load stays constant and the material is allowed to creep for 5000 seconds.

Change to **Step** module.

Pick  to create a new step and select the **Visco** procedure.

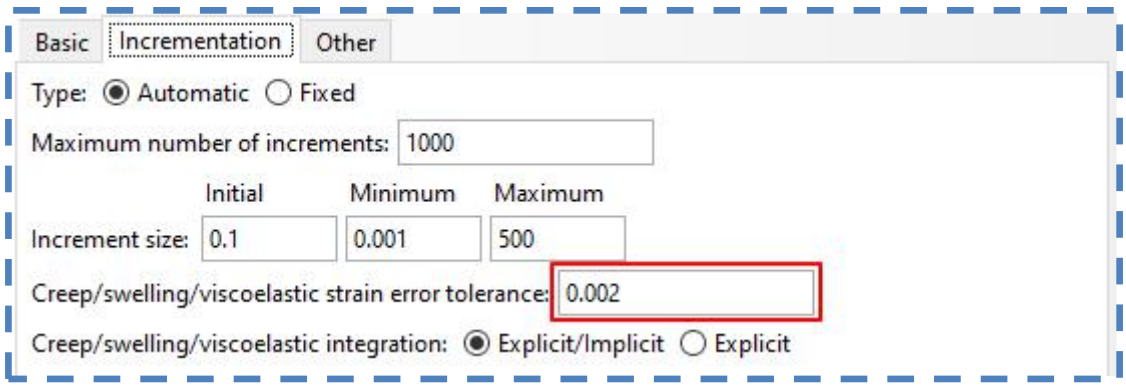


Pick **Continue** and Edit Step dialog box will appear. Notice that by default NLgeom option is set to **On**. Enter **5000** as the time timer period.



Under the Incrementation tab, set the initial increment size to **0.1**, minimum to **0.001** and maximum to **500**

Enter **0.002** as viscoelastic strain error tolerance.



Strain error tolerance is the tolerance value for the maximum difference in the creep strain increment over a time increment.

Strain error tolerance is usually the same order of magnitude as the maximum elastic strain. (Maximum elastic strain at the end of first step is 9.3 x 10^{-3} in this exercise.)

Pick **OK** to complete the definition of step.

Defining Load

A uniform pressure load is applied on the top edge of the sheet. The load has already been defined.

Boundary Conditions

Boundary conditions required for the analysis have already been defined.

The boundary condition “BC-BottomEdge” applies the symmetry boundary condition to the bottom edge of the sheet. The boundary condition “BC-Node” constrains the movement of the bottom left node along x-axis.

Defining Material Model

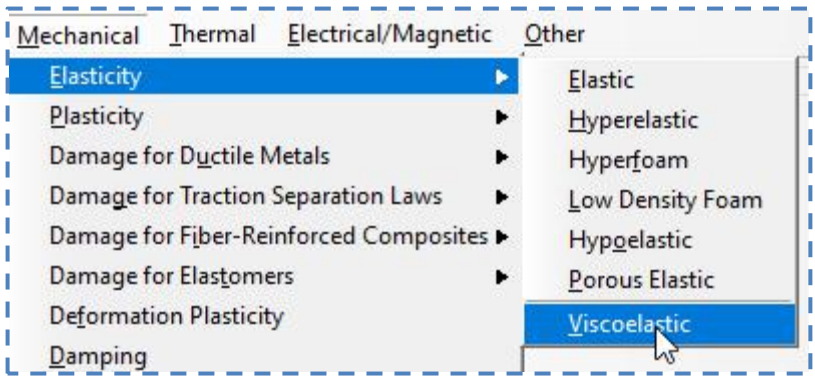
A material has already been defined, we will edit its definition to add viscoelastic properties.

Change to **Property** module.

Open the Material Manager by picking  .

Select the Polyethylene and pick **Edit**.

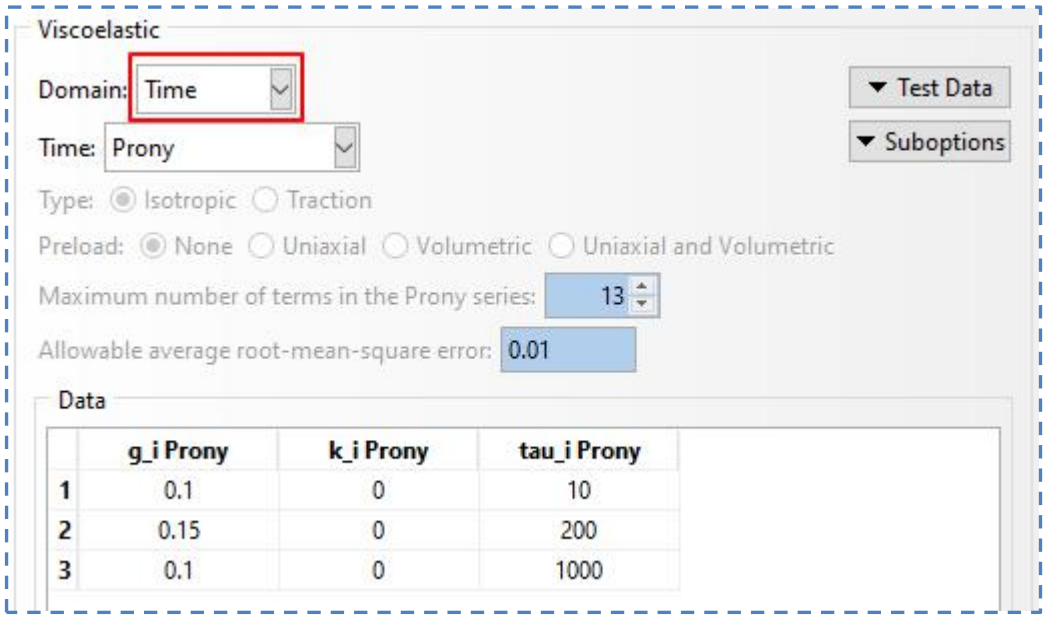
In the Edit Material dialog box, pick **Elasticity > Viscoelastic** in the Mechanical menu.



The viscous behavior will be modeled by a time dependent shear relaxation modulus which is expanded in a Prony series with three terms. It is assumed that no volumetric relaxation occurs and hence bulk modulus is independent of time. Therefor relative moduli for bulk modulus are set to 0. The relative moduli and time constants to specify the Prony series are as follow:

i	g_i^P	k_i^P	τ_i^P
1	0.1	0	10
2	0.15	0	200
3	0.1	0	1000

Pick **Time** as domain and enter the relative moduli and time constants as shown below.



Pick **OK** to apply and exit.

Job Submission

All the information required for analysis has been set up. Now we can submit the job for analysis.

So change to **Job** module and pick  to open the job manager.

Pick **Create** and create a job named Creep or whatever name you would like.

Pick **Continue** and then **OK**.

Pick **Submit** to submit the job for analysis.

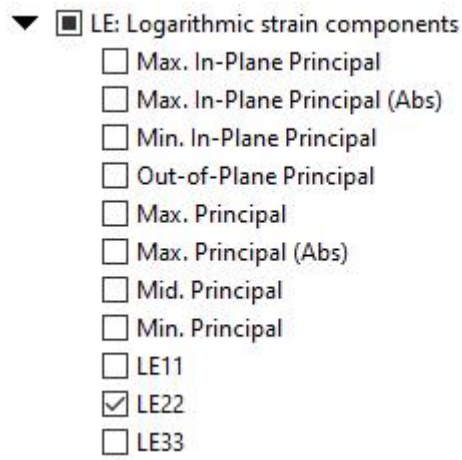
Pick **Results** to view the results in Visualization module.

Postprocessing

Pick  and Create XY Data dialog box will appear.

Pick **ODB field output > Continue**

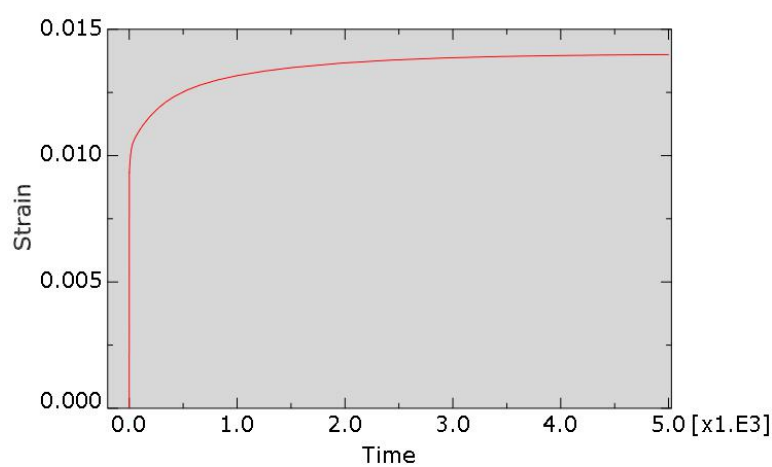
In the Position field, select **Unique Nodal** and check the **LE22** checkbox.



Under the Elements/Nodes tab, pick the **Node sets** as method and select the **SHEET-1.TOPNODE**

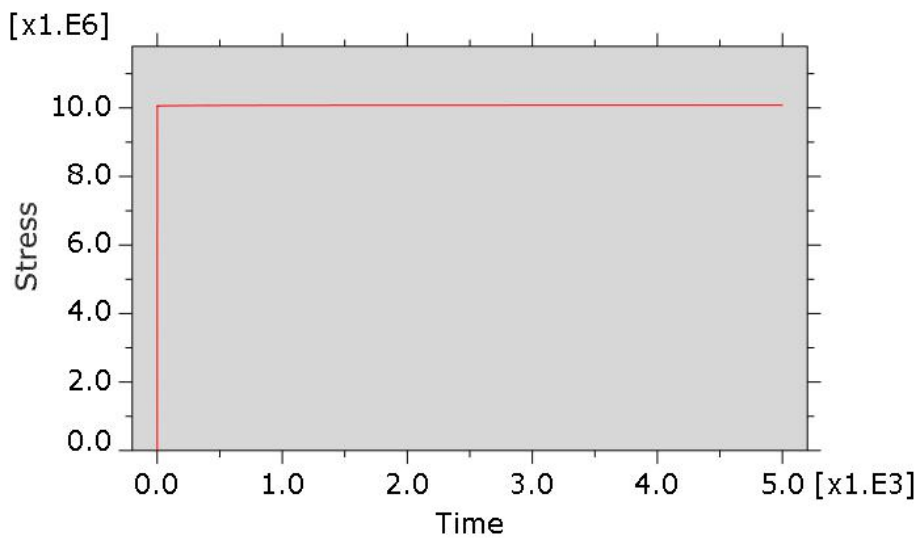
Check **Highlight items in viewport** in the dialog box to highlight the nodes in viewport.

Pick **Plot** tab and graph will appear as shown below (after modifying the axis limits).



It can be seen that sheet creeps for 5000 seconds as the load is kept constant during second step. Furthermore solution appears to be steady state as 5000 seconds is a long time compared with material time constants.

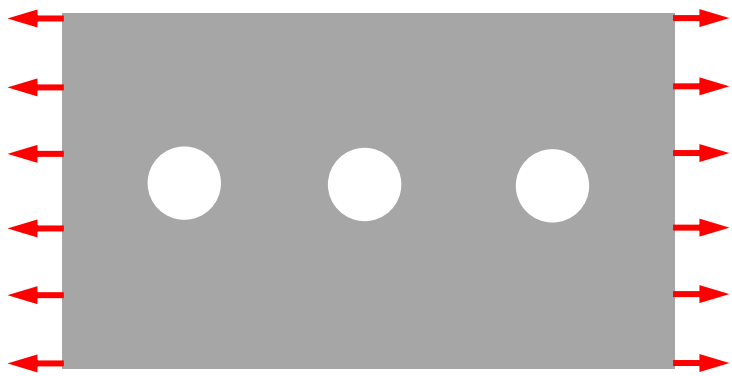
By plotting the S22, it can be seen that stress almost stays constant as shown below.



Select **File > Save** to save the changes we made so for.

Exercise 23

In this exercise we will investigate stress relaxation in a viscoelastic sheet subjected to constant strain. The sheet is modeled assuming plane stress conditions. The material is assumed linear viscoelastic. The geometry considered is shown in the figure below.



The sheet is symmetric so only quarter of the geometry is considered.

Pick **File > Set Work Directory** and set the work directory to the **SheetVisco** folder

Open the model database **Sheet_Hole.cae**.

The model consists of a single part, Sheet. It is assumed to be made from polyethylene, which is modeled as a Neo-Hookean material with C_{10}= 186 MPa and D=0.001 MPa^{-1}. It is meshed using second order plane stress quadrilateral elements with reduced integration (CPS8R).

Defining Step

The analysis will be performed in two steps. A strain is imposed using displacement boundary condition in the first step. This loading is performed using the Static, General procedure. The static procedure does not allow viscous material behavior so the response is purely elastic during first step. To simulate the viscoelastic response a second step using the quasi-static procedure (Visco procedure) has been defined. In this step strain stays constant and the material relaxes for 2E6 seconds. Both steps have already been defined.

Boundary Conditions

Two boundary conditions required for the analysis have already been defined.

The boundary condition “BC-BottomEdge” applies the symmetry boundary condition to the bottom edge of the sheet. The boundary condition “BC-LeftEdge” applies the symmetry boundary condition to the left edge of the sheet. We will create another boundary condition to impose the fixed displacement to the right edge of the sheet.

To define boundary condition, change to **Load** module.

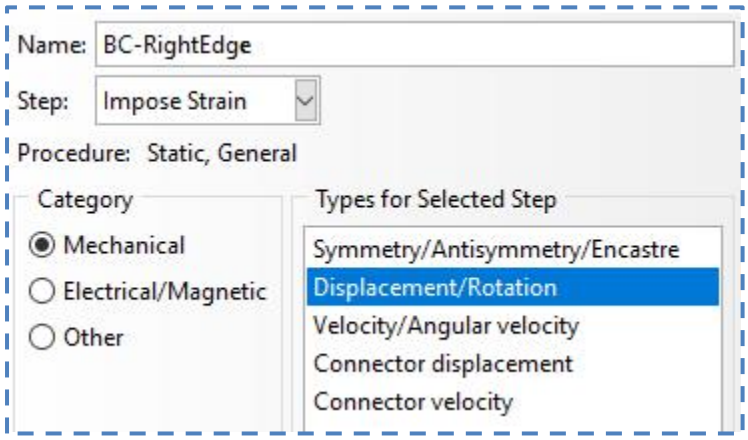
Pick  to open Boundary Condition Manager.

Pick **Create** to define a new boundary condition.

Enter BC-RightEdge as name.

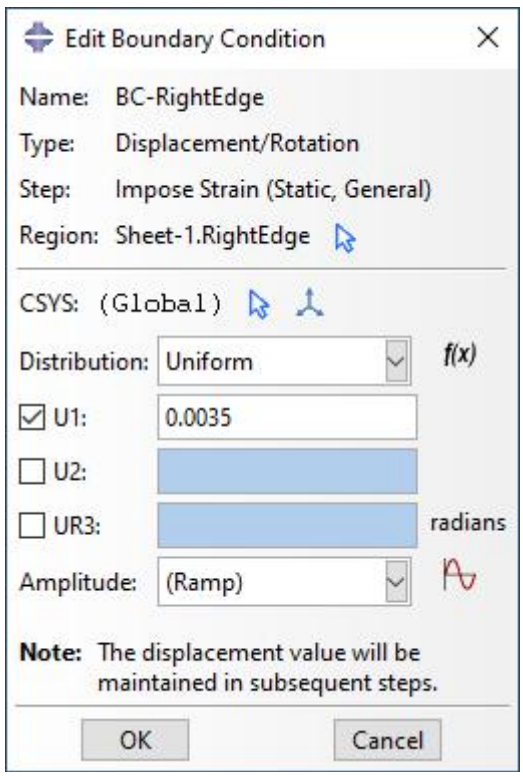
Pick **Impose Strain** as step. This is the step defined with static procedure.

Pick Displacement/Rotation as type.



Pick **Continue** and system will ask you to pick the region. As we have already defined a set, so pick **Sets** and select the Sheet-1.RightEdge.

Pick **Continue** to proceed and enter ***0.0035*** as magnitude of displacement in U1 direction as shown below.



Pick **OK** in the Edit Constraint dialog box to apply changes and exit.

Pick **Dismiss** to close the manager.

Material Model

A material model has already been defined where the instantaneous behavior of the viscoelastic material is defined by hyperelastic properties. The viscous behavior is modeled by a time-dependent shear modulus which is expanded in a Prony series with five terms. It is assumed that no volumetric relaxation occurs and hence bulk modulus is independent of time. Therefor relative moduli for bulk modulus are set to 0. The relative moduli and time constants to specify the Prony series are as follow:

i	g_i^P	k_i^P	τ_i^P
1	0.001	0	10
2	0.0015	0	200
3	0.003	0	1000
4	0.04	0	10000
5	0.05	0	1000000

Job Submission

All the information required for analysis has been set up. Now we can submit the job for analysis.

So change to **Job** module and pick  to open the job manager.

Pick **Create** and create a job named Relaxation or whatever name you would like.

Pick **Continue** and then **OK**.

Pick **Submit** to submit the job for analysis.

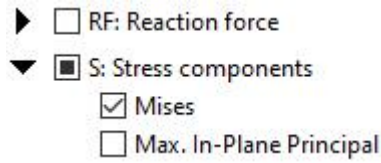
Pick **Results** to view the results in Visualization module.

Postprocessing

Pick  and Create XY Data dialog box will appear.

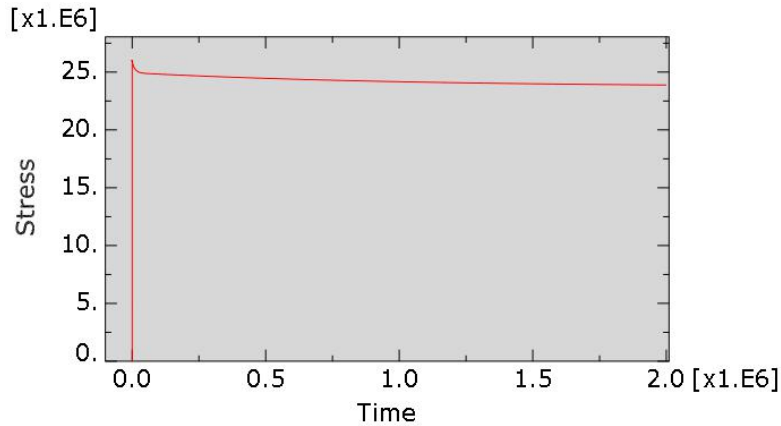
Pick **ODB field output > Continue**

In the Position field, select **Unique Nodal** and check the **Mises** checkbox.



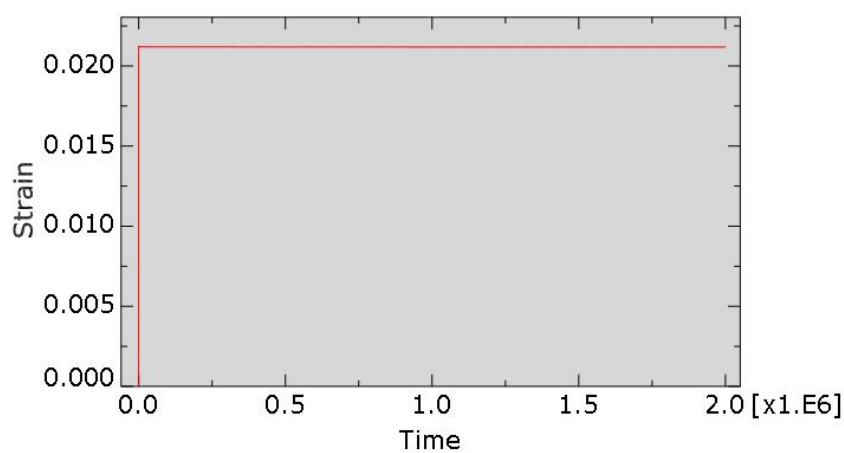
Under the Elements/Nodes tab, pick the **Node sets** as method and select the **SHEET-1.NODERIGHT**

Pick **Plot** tab and graph will appear as shown below (after modifying the axis limits).



It can be seen that stress relaxes for 2E6 seconds as the strain is kept constant during second step.

By plotting the LE11, it can be seen that strain stays constant as shown below.

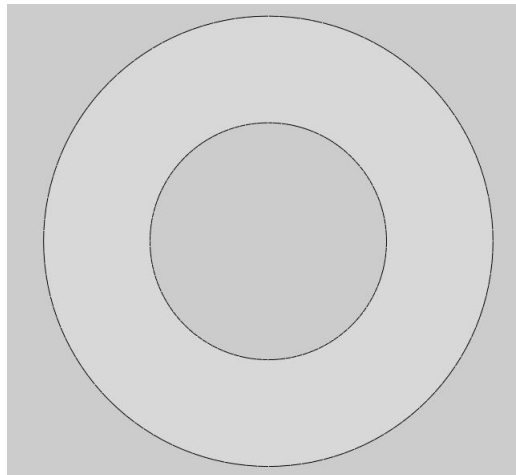


Select **File > Save** to save the changes we made so for.

Note: Generally it is not recommended to plot stress values at nodes as those are extrapolated from integration points.

Exercise 24

In this exercise we will investigate stress relaxation in a viscoelastic bushing using a nonlinear viscoelastic model. The bushing is modeled assuming plane strain conditions. The geometry considered is shown in the figure below.



Pick **File > Set Work Directory** and set the work directory to the **BushingVisco** folder.

Open the model database **Sheet_Hole.cae**.

The model consists of a single part, Bushing. It is assumed to be made from EPDM rubber, which is modeled as a Neo-Hookean material with C_{10}= 186 MPa and D=0.001 MPa^{-1}. It is meshed using first order plane strain quadrilateral elements with reduced integration (CPE4R).

Defining Step

The analysis will be performed in two steps. A strain is imposed using displacement boundary condition in the first step. This loading is performed using the Static, General procedure. The static procedure does not allow viscous material behavior so the response is purely elastic during first step. To simulate the viscoelastic response a second step using the quasi-static procedure (Visco procedure) has been defined. During this step, strain stays constant and the material relaxes for 2000 seconds. In this step the initial increment size is set to **0.01**, minimum to **1E-6** and maximum size to **200.** A viscoelastic strain error tolerance of **0.1** has been set.

Strain error tolerance is the tolerance value for the maximum difference in the creep strain increment over a time increment.

Strain error tolerance is usually the same order of magnitude as the maximum elastic strain. (Maximum elastic strain at the end of first step is 9.9 x 10^{-1} in this exercise.)

Boundary Conditions

The bushing is glued to a rigid fixed cylindrical part on the outside. This is modeled by fixing the displacements degrees of freedom of outer edge. The boundary condition "BC-OuterEdge", which has already been defined, constrains the degrees of freedom of the outer edge of the bushing.
The bushing is glued to a rigid shaft on the inside. The inner edges of the bushing are connected using a kinematic coupling constraint to a reference point located in the center. Therefor displacement of the reference point represents the displacement of shaft. Now we will create a boundary condition to impose the displacement to the reference point.

To define boundary condition, change to **Load** module.

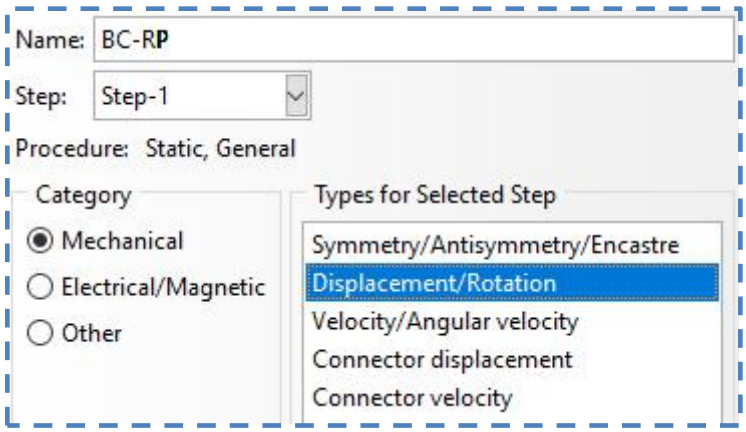
Pick  to open Boundary Condition Manager.

Pick **Create** to define a new boundary condition.

Enter **BC-RP** as name.

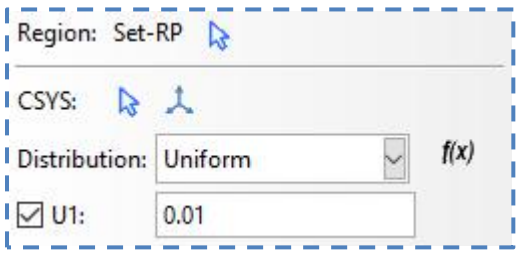
Pick **Step-1** as step. This is the step defined with static procedure.

Pick Displacement/Rotation as type.



Pick **Continue** and system will ask you to pick the region. As we have already defined a set, so pick **Sets** and select the Set-RP.

Pick **Continue** to proceed and enter ***0.01*** as magnitude of displacement in U1 direction as shown below.



Pick **OK** in the Edit Constraint dialog box to apply changes and exit.

Pick **Dismiss** to close the manager.

Material Model

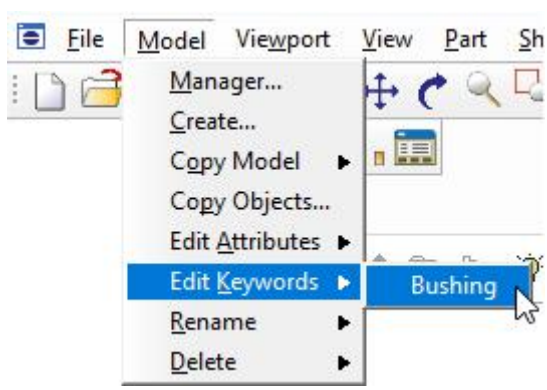
A material model has already been defined where the instantaneous behavior of the viscoelastic material is defined by hyperelastic properties. EPDM exhibits nonlinear viscoelastic properties. Now we will define the viscoelastic behavior by using Power-law strain hardening model. We will use the following values of constants in the law.

A	3.0E-39
n	6.0
m	-0.15
r	0.9

Here r is the stiffness ratio.

To define the nonlinear viscoelastic model, editing of the keywords file is required.

Pick **Model > Edit Keywords > Bushing** from the main menu.



Enter the following keywords after the '*Hyperelastic, new hooke, text.

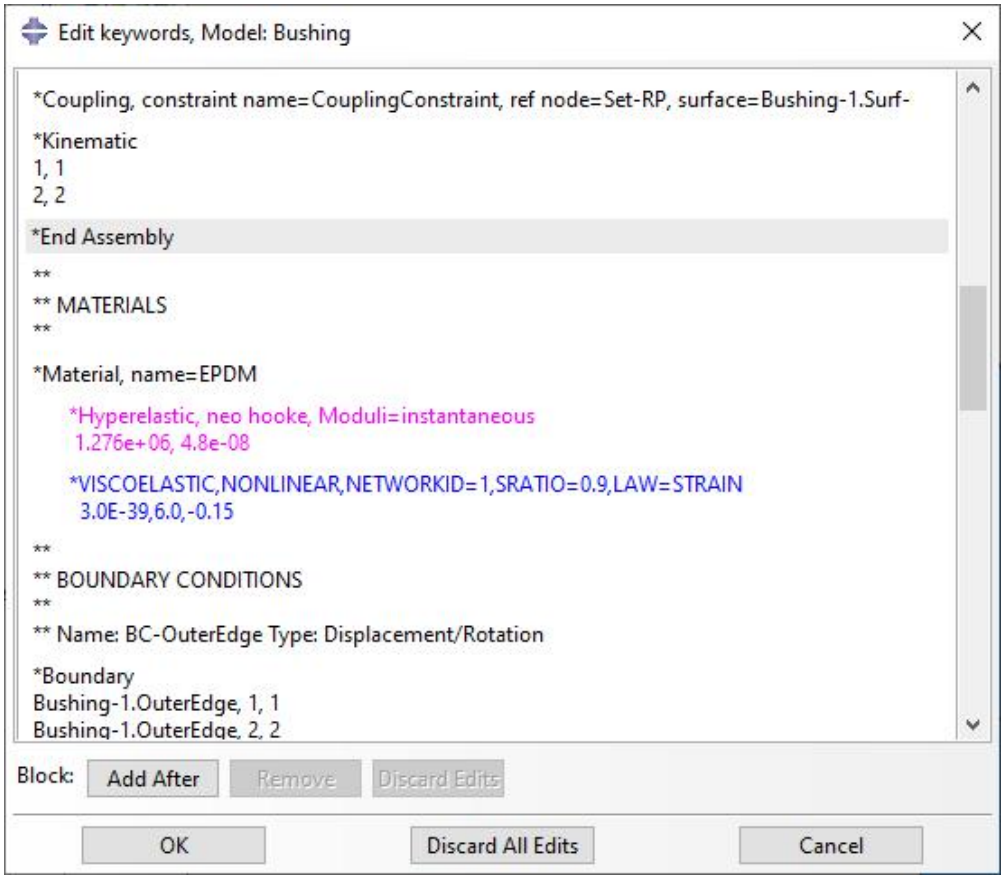
, Moduli=instantaneous

MODULI=INSTANTANEOUS is used to indicate that the hyperelastic material constants define the instantaneous behavior.

Use the **Add After** tab after placing the cursor at the end of data line of Hyperelastic keyword and enter following.

```
*VISCOELASTIC, NONLINEAR, NETWORKID=1, SRATIO=0.9, LAW=STRAIN
  3.0E-39, 6.0, -0.15
```

The keyword property editor after editing is shown below.



Pick **OK** to apply and exit.

⇨ Job Submission

All the information required for analysis has been set up. Now we can submit the job for analysis.

So change to **Job** module and pick  to open the job manager.

Pick **Create** and create a job named Relaxation or whatever name you would like.

Pick **Continue** and then **OK**.

Pick **Submit** to submit the job for analysis.

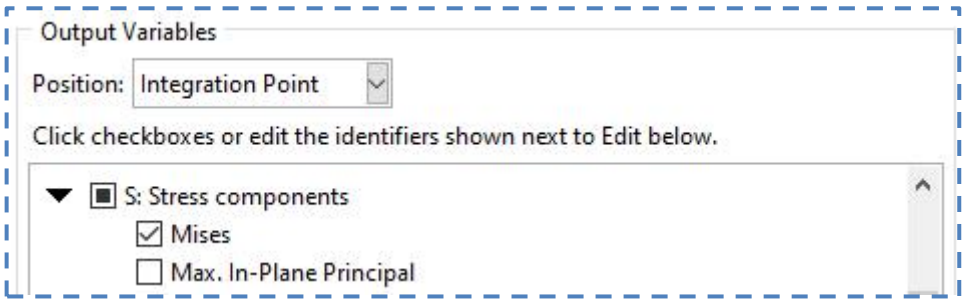
Pick **Results** to view the results in Visualization module.

⇨ Postprocessing

Pick  and Create XY Data dialog box will appear.

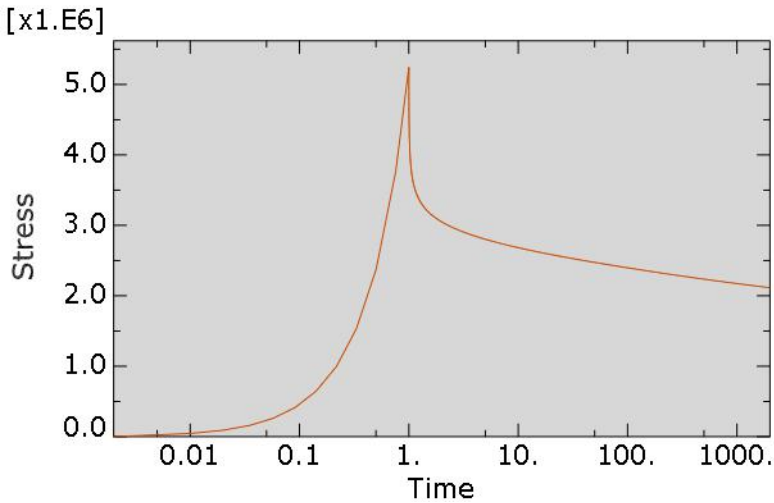
Pick **ODB field output > Continue**

In the Position field, select **Integration Point** and check the **Mises** checkbox.



Under the Elements/Nodes tab, pick the **Elements sets** as method and select the **BUSHING-1.ELEMTOP**

Pick **Plot** tab and graph will appear as shown below. The time axis is shown using log scale.



It can be seen that stress relaxes for 2000 seconds as the strain is kept constant during second step.

Select **File > Save** to save the changes we made so for.