Created in COMSOL Multiphysics 6.1

Lemaitre–Chaboche Viscoplastic Model

Introduction

Most metals and alloys undergo viscoplastic deformation at high temperatures. In case of cyclic loading, a constitutive law with both isotropic and kinematic hardening is necessary to describe effects such as ratcheting, cyclic softening/hardening, and stress relaxation. The Lemaitre–Chaboche viscoplastic model combines isotropic hardening with nonlinear kinematic hardening to model these effects.

This tutorial model demonstrates the uniaxial deformation of an indium IN 100 test specimen submitted to cyclic tension-compression loading at high temperature, as described in Ref. 1

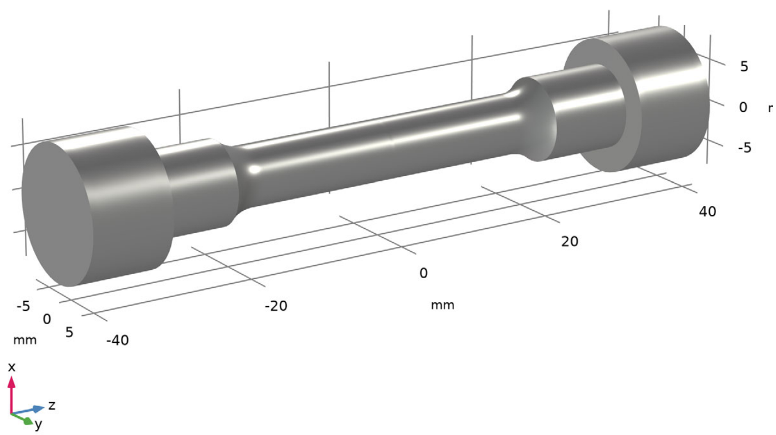


Figure 1: Specimen used for the uniaxial tension-compression tests.

Model Definition

The test specimen consists of a small cylinder with a thin central section to ensure uniform stress and strain distributions. Thick parts are added to the ends of the specimen to ease the mounting in a universal testing machine. The loading for the tension-compression cycle is directed in the axial direction. For shortening the computation time, half of the specimen is modeled in a 2D axisymmetric geometry.

GOVERNING EQUATIONS

The strain tensor consists of the sum of the elastic strain tensor $\varepsilon_{\mathrm{el}}$ and the viscoelastic strain tensor $\varepsilon_{\mathrm{vp}}$:

$$\varepsilon = \varepsilon_{\mathrm{el}} + \varepsilon_{\mathrm{vp}}$$

The constitutive law between stress and strain is given by Hooke's law

$$\sigma = \mathbf{C}:(\varepsilon - \varepsilon_{\mathrm{vp}})$$

The viscoplastic strain tensor is defined by Lemaitre–Chaboche viscoplastic rule

$$\dot{\varepsilon}_{\mathrm{vp}} = \frac{3}{2}\dot{\varepsilon}_{\mathrm{vpe}}\frac{\sigma' - \sigma_{\mathrm{b}}'}{J_2(\sigma - \sigma_{\mathrm{b}})}$$

where σ_{b} is the back-stress tensor, derived by the nonlinear kinematic hardening rule; and σ' and σ_{b}' are the deviatoric parts of the stress and back-stress tensors σ and σ_{b}.

The equivalent viscoplastic strain rate is defined by the expression

$$\dot{\varepsilon}_{\mathrm{vpe}} = \max\left(A\left(\frac{J_2(\sigma - \sigma_{\mathrm{b}}) - \sigma_{\mathrm{y}}}{\sigma_{\mathrm{ref}}}\right)^n, 0\right)$$

where A is the viscoplastic rate coefficient, σ_{ref} is a reference stress, n is the stress exponent, and σ_{y} is the yield stress given by the nonlinear isotropic hardening rule.

MIXED HARDENING

The isotropic hardening model represents the change in yield stress as a function of the equivalent viscoplastic strain. Lemaitre and Chaboche (Ref. 1) derived a nonlinear isotropic hardening relation of the type

$$\sigma_{\mathrm{y}} = \sigma_{\mathrm{ys0}} + \sigma_{\mathrm{sat}}\left(1 - e^{-\beta\varepsilon_{\mathrm{vpe}}}\right)$$

where σ_{ys0} is the initial yield stress, and σ_{sat} and β are material parameters.

The kinematic hardening rule represents the translation of the yield surface as a function of the back-stress σ_{b} and the equivalent viscoplastic strain $\varepsilon_{\mathrm{vp}}$. Lemaitre and Chaboche (Ref. 1) derived a nonlinear kinematic hardening relation where the back-stress tensor is calculated from the ordinary differential equation

$$\dot{\sigma}_{\mathrm{b}} = \frac{2}{3}C_{\mathrm{k}}\dot{\varepsilon}_{\mathrm{vp}} - \gamma_{\mathrm{k}}\dot{\varepsilon}_{\mathrm{vpe}}\sigma_{\mathrm{b}}$$

where C_k and γ_k are material parameters. The kinematic hardening parameter γ_k also depends exponentially on the equivalent viscoplastic strain:

$$\gamma_k = \gamma_s + (\gamma_0 - \gamma_s)e^{-\beta_k \varepsilon_{vpe}}$$

where γ_s, γ_0, and β_k are material parameters.

MATERIAL DATA

The material parameters of the indium alloy IN 100 at a temperature of 900 K are given in Ref. 1.

TABLE 1: MATERIAL PARAMETERS, IN 100 AT 900 K.

Parameter	**Value**
K	490 MPa.s$^{1/n}$
n	9
σ_{y0}	60 MPa
$\sigma_{y0}+\sigma_{sat}$	25 MPa
b	200
C_k	362,500 MPa
γ_0	1200
γ_s	1540
β_k	1000

In Ref. 1, the parameter K is given in units of MPa·s$^{1/n}$. The parameters K, A, and σ_{ref} are related by the expression

$$\frac{A}{(\sigma_{ref})^n} = \frac{1}{K^n}$$

Use the values $A = 1/s$ and $\sigma_{ref} = 490$ MPa to obtain similar results as in given in Ref. 1.

Results and Discussion

STRESS VS. TIME

After the first tensile cycle the axial stress (blue line in Figure 2) increases linearly, exceeding the yield strength (cyan line). Because of viscous effects, the onset of plasticity starts once the viscous stress (green line) reaches the yield strength. At this point, the relation between stress and strain is no longer linear, kinematic/isotropic hardening evolves, and the back-stress (red line) increases.

When the maximum allowed tensile strain is reached, the prescribed axial velocity turns negative to prescribe axial unloading. The axial stress then decreases linearly, and the back stress remains constant to account for kinematic hardening and the Bauschinger effect. The yield strength in compression after a tensile cycle is lower than the initial yield strength.

In the compressive cycle, the viscoplastic flow starts earlier and lasts for a longer period. When the maximum allowed compressive strain is reached, the prescribed axial velocity becomes positive to prescribe tensile unloading. This procedure is repeated for each load cycle.

.

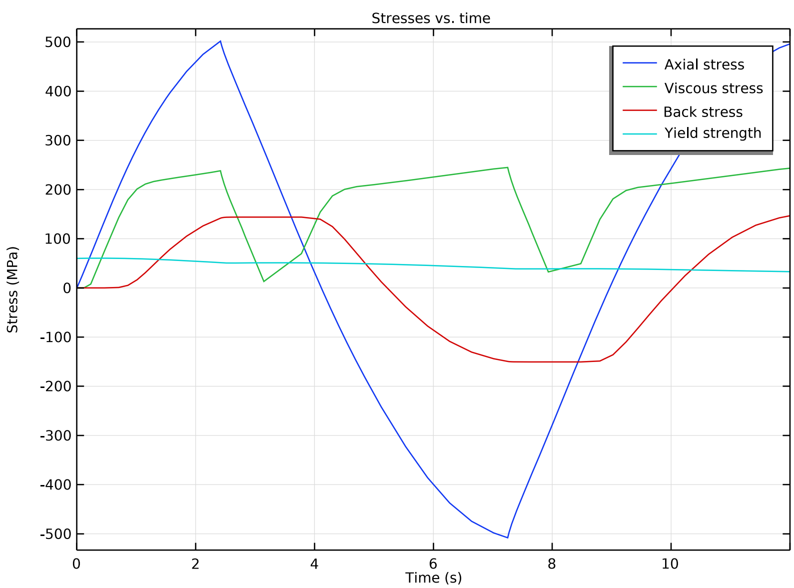


Figure 2: Evolution of the axial stress variables with time.

Four studies are performed to prescribe four different types of loading cycles: prescribed symmetric strain, prescribed symmetric stress, prescribed nonsymmetric strain, and prescribed nonsymmetric stress. The stress-strain graph for each type of loading cycle highlights material properties of the Lemaitre–Chaboche viscoplastic model.

SYMMETRIC CYCLES

Symmetric loading aims to show the stabilization effect of the hysteresis cycle. This emphasizes the softening or hardening effect during cyclic loadings for a given hardening rule.

For the symmetric prescribed axial strain cycle shown in Figure 3, there is a periodic response to the periodic load and a stabilized state is reached after few cycles. A closer view shows that the stress amplitude decreases during the first cycles due to material softening.

A similar effect can be seen in for the symmetric prescribed stress cycle in Figure 4, but more cycles are needed to reach a stabilized state when compared to the prescribed strain loading. Lastly, the axial strain amplitude increases with the number of cycles. This is an effect of the isotropic hardening rule. With kinematic hardening only, the stabilized state is reached after the first cycle of prescribed stress. Thus, a mixed hardening formulation is needed to model both cyclic softening/hardening and Bauschinger effects.

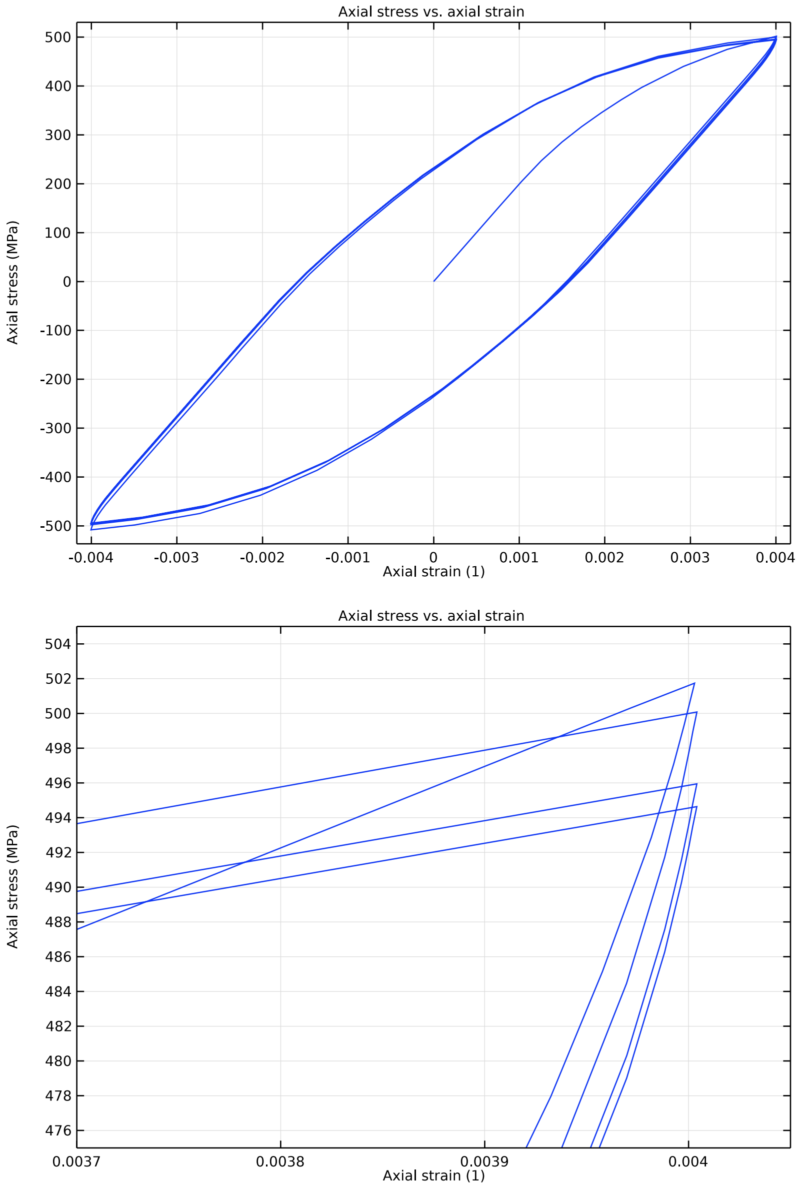
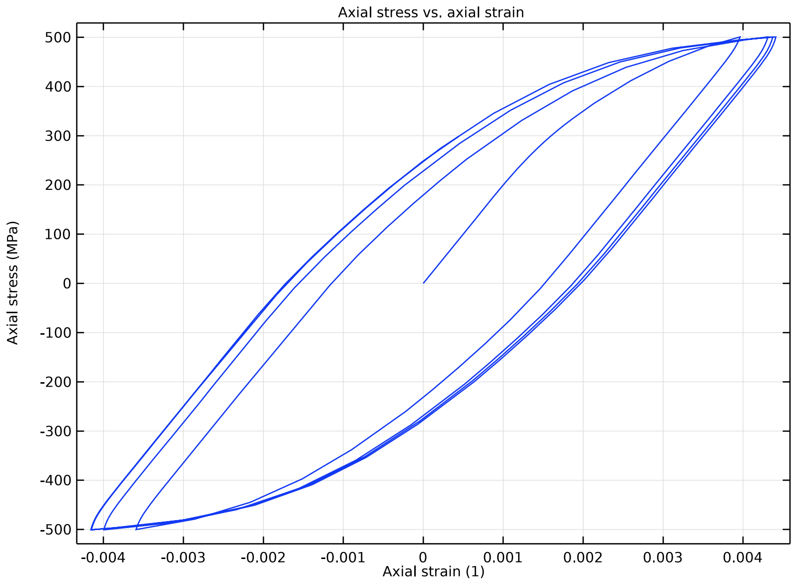


Figure 3: Axial stress versus axial strain for a symmetric prescribed strain loading.



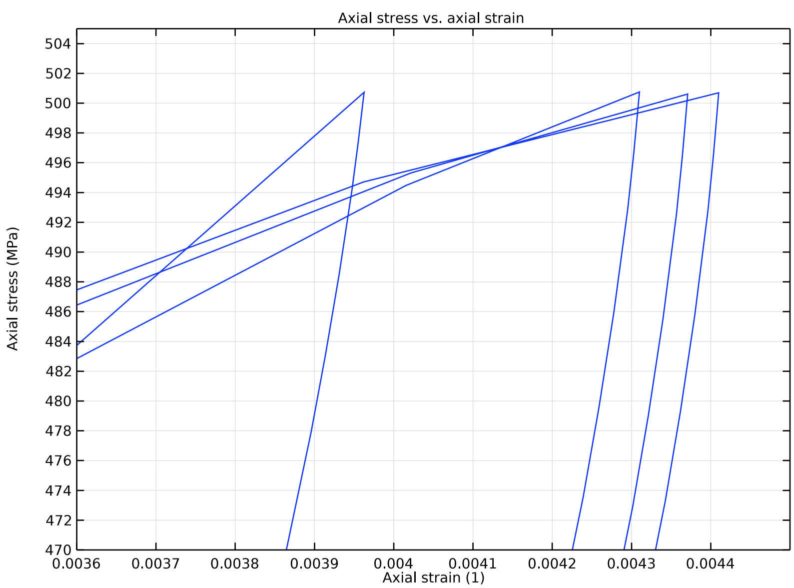


Figure 4: Axial stress versus axial strain for a symmetric prescribed stress loading.

NONSYMMETRIC STRAIN CYCLES

When the average strain is not zero in a prescribed strain cycle as in Figure 5, the initial asymmetry in the axial stress gradually disappears over the cycles: This happens because of the stress relaxation effect, which is observed in many alloys.

Once a stabilized cycle is reached, the tensile and compressive stresses are equal in absolute value. This is an effect of the nonlinear kinematic hardening rule. Applying linear kinematic hardening only (or isotropic hardening) would not allow to observe the relaxation of the mean stress, so a nonlinear kinematic hardening rule is needed to model stress relaxation.

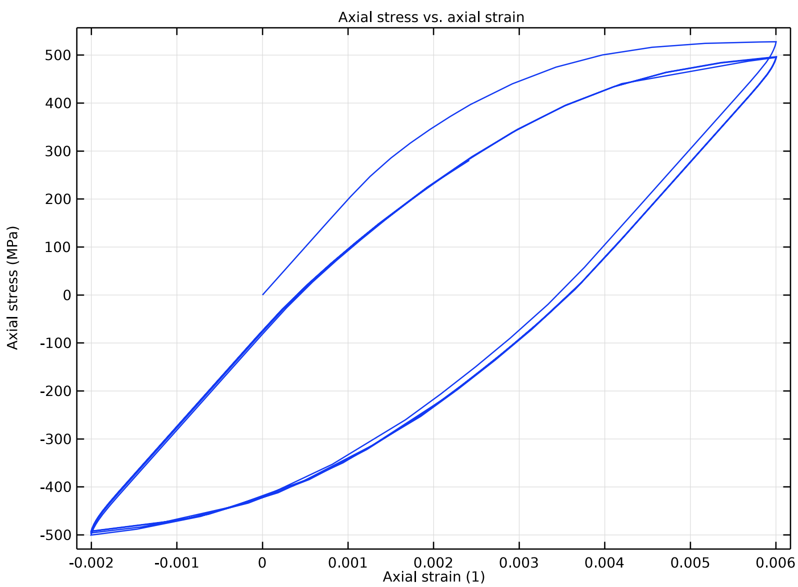


Figure 5: Axial stress versus axial strain for a nonsymmetric prescribed strain loading.

NONSYMMETRIC STRESS CYCLES

The gradual deformation that occurs in tension-compression cycles when the mean stress is nonzero is called the ratcheting effect. The ratcheting effect is most pronounced when the lower stress limit is $-\sigma_{ys0}$. This behavior is possible to observe thanks to the nonlinear kinematic hardening rule.

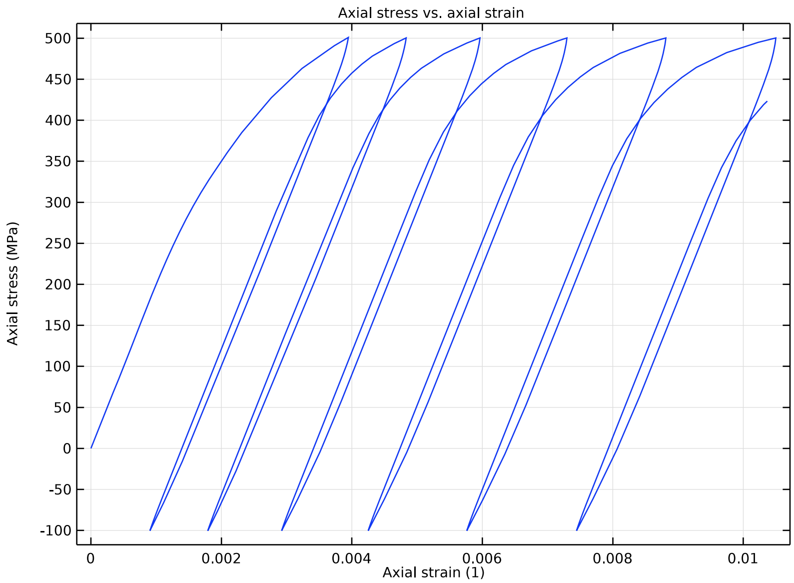


Figure 6: Axial stress versus axial strain for a nonsymmetric prescribed stress loading.

Notes About the COMSOL Implementation

Several yield functions and plastic potentials are available in the **Viscoplasticity** node. Use von Mises equivalent stress to reproduce the results described in Ref. 1.

Solve four studies to account for the different types of load cycles: the load cycle can be symmetric or nonsymmetric, and the control can be done by prescribing either the axial strain or the axial stress.

Apply a constant axial velocity on one end of the test specimen to avoid using complicate loading functions. Multiply this axial velocity by 1 to represent axial tension, or by −1 to represent axial compression.

To achieve this, define a discrete state called `LoadingType` in an **Events** interface. Add **Indicator States** in the Events interface to define tension and compression limits based on the stress or strain state. For instance, for a symmetric loading cycle with a prescribed strain of 0.4 % the following **Indicator States** are used:

Tension	`intop1(solid.el33)-0.004`
Compression	`intop1(solid.el33)+0.004`

Then add two **Implicit Event** nodes to define the discrete state `LoadingType`: when Tension > 0, then `LoadingType` is set to −1, and when Compression < 0, `LoadingType` is set to 1.

Reference

1. J. Lemaitre and J.-L. Chaboche, *Mécanique des matériaux solides,* 2nd ed., Dunod, 2001 (in French).

Application Library path: `Nonlinear_Structural_Materials_Module/Viscoplasticity/lemaitre_chaboche_viscoplastic_model`

Modeling Instructions

From the **File** menu, choose **New**.

NEW

In the **New** window, click **Model Wizard**.

MODEL WIZARD

1 In the **Model Wizard** window, click **2D Axisymmetric**.

2 In the **Select Physics** tree, select **Structural Mechanics>Solid Mechanics (solid)**.

3 Click **Add**.

4 In the **Select Physics** tree, select **Mathematics>ODE and DAE Interfaces>Events (ev)**.

5 Click **Add**.

6 Click **Study**.

7 In the **Select Study** tree, select **General Studies>Time Dependent**.

8 Click **Done**.

GLOBAL DEFINITIONS

Parameters 1

Define the parameters that will be needed in the model.

1 In the **Model Builder** window, under **Global Definitions** click **Parameters 1**.

2 In the **Settings** window for **Parameters**, locate the **Parameters** section.

3 In the table, enter the following settings:

Name	Expression	Value	Description
e0t	1e-3[1/s]	0.001 1/s	Prescribed strain rate
L0	(35/2+14+10)[mm]	0.0415 m	Half length
gamma0	1200	1200	Initial kinematic hardening parameter
gammas	1540	1540	Saturation kinematic hardening parameter
beta	1000	1000	Kinematic hardening parameter exponent

Add a step function to apply the load smoothly.

Step 1 (step1)

1 In the **Home** toolbar, click f(x) **Functions** and choose **Global>Step**.

2 In the **Settings** window for **Step**, locate the **Parameters** section.

3 In the **Location** text field, type `5[ms]`.

4 Click to expand the **Smoothing** section. In the **Size of transition zone** text field, type `0.01`.

5 Click **Plot**.

GEOMETRY 1

1 In the **Model Builder** window, under **Component 1 (comp1)** click **Geometry 1**.

2 In the **Settings** window for **Geometry**, locate the **Units** section.

3 From the **Length unit** list, choose **mm**.

Rectangle 1 (r1)

1 In the **Geometry** toolbar, click **Rectangle**.

2 In the **Settings** window for **Rectangle**, locate the **Size and Shape** section.

3 In the **Width** text field, type `7/2`.

4 In the **Height** text field, type `35/2`.

Rectangle 2 (r2)

1 In the **Geometry** toolbar, click **Rectangle**.

2 In the **Settings** window for **Rectangle**, locate the **Size and Shape** section.

3 In the **Width** text field, type `5`.

4 In the **Height** text field, type `10`.

5 Locate the **Position** section. In the **z** text field, type `21.5`.

Rectangle 3 (r3)

1 In the **Geometry** toolbar, click **Rectangle**.

2 In the **Settings** window for **Rectangle**, locate the **Size and Shape** section.

3 In the **Width** text field, type `8`.

4 In the **Height** text field, type `10`.

5 Locate the **Position** section. In the **z** text field, type `31.5`.

Circular Arc 1 (ca1)

1 In the **Geometry** toolbar, click **More Primitives** and choose **Circular Arc**.

2 In the **Settings** window for **Circular Arc**, locate the **Properties** section.

3 From the **Specify** list, choose **Endpoints and start angle**.

4 Locate the **Starting Point** section. In the **r** text field, type `3.5`.

5 In the **z** text field, type `17.5`.

6 Locate the **Endpoint** section. In the **r** text field, type `5`.

7 In the **z** text field, type `21.5`.

8 Locate the **Angles** section. In the **Start angle** text field, type `180`.

9 Select the **Clockwise** check box.

10 Click **Build Selected**.

Polygon 1 (pol1)

1 In the **Geometry** toolbar, click **Polygon**.

2 In the **Settings** window for **Polygon**, locate the **Object Type** section.

3 From the **Type** list, choose **Open curve**.

4 Locate the **Coordinates** section. From the **Data source** list, choose **Vectors**.

5 In the **r** text field, type `5 0 0 3.5` .

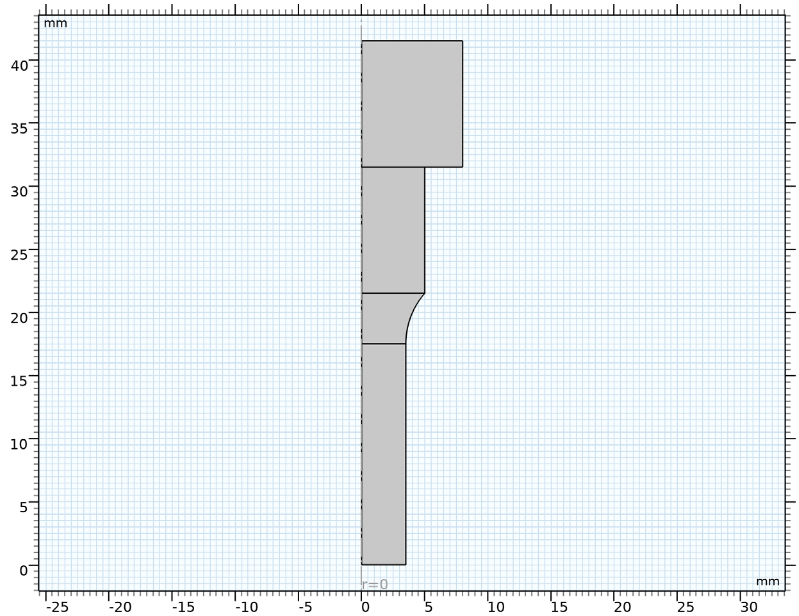
6 In the **z** text field, type `21.5 21.5 17.5 17.5` .

Convert to Solid 1 (csol1)

1 In the **Geometry** toolbar, click **Conversions** and choose **Convert to Solid**.

2 Select the objects **pol1** and **ca1** only.

3 In the **Settings** window for **Convert to Solid**, click **Build All Objects**.



Add conditions to toggle the boundary conditions between tension and compression. This is controlled by the **Events** interface. First create a nonlocal integration coupling to get variable from a point.

DEFINITIONS

Integration 1 (intop1)

1 In the **Definitions** toolbar, click **Nonlocal Couplings** and choose **Integration**.

2 In the **Settings** window for **Integration**, locate the **Source Selection** section.

3 From the **Geometric entity level** list, choose **Point**.

4 Select Point 6 only.

5 Locate the **Advanced** section. Clear the **Compute integral in revolved geometry** check box.

EVENTS (EV)

Discrete States 1

1 In the **Model Builder** window, under **Component 1 (comp1)** right-click **Events (ev)** and choose **Discrete States**.

2 In the **Settings** window for **Discrete States**, locate the **Discrete States** section.

3 In the table, enter the following settings:

Name	Initial value (u0)	Description
LoadingType	1	

The `LoadingType` variable is 1 when tension is applied, and -1 when compression is applied.

The first indicator state is used to control the loading with a strain measure. The tension and compression limits are symmetric.

Indicator States: Strain, Symmetric

1 In the **Physics** toolbar, click **Global** and choose **Indicator States**.

2 In the **Settings** window for **Indicator States**, type `Indicator States: Strain, Symmetric` in the **Label** text field.

3 Locate the **Indicator Variables** section. In the table, enter the following settings:

Name	g(v,vt,vtt,t)
Tension	intop1(solid.el33)-0.004
Compression	intop1(solid.el33)+0.004

The second indicator state is used to control the loading with a stress measure. The tension and compression limits are symmetric.

Indicator States: Stress, Symmetric

1 In the **Physics** toolbar, click **Global** and choose **Indicator States**.

2 In the **Settings** window for **Indicator States**, type `Indicator States: Stress, Symmetric` in the **Label** text field.

3 Locate the **Indicator Variables** section. In the table, enter the following settings:

Name	g(v,vt,vtt,t)
Tension	intop1(solid.sl33)-500[MPa]
Compression	intop1(solid.sl33)+500[MPa]

The third indicator state is used to control the loading with a strain measure. The tension and compression limits are not symmetric.

Indicator States: Strain, Nonsymmetric

1 In the **Physics** toolbar, click **Global** and choose **Indicator States**.

2 In the **Settings** window for **Indicator States**, type `Indicator States: Strain, Nonsymmetric` in the **Label** text field.

3 Locate the **Indicator Variables** section. In the table, enter the following settings:

Name	g(v,vt,vtt,t)
Tension	intop1(solid.el33)-0.006
Compression	intop1(solid.el33)+0.002

The fourth indicator state is used to control the loading with a stress measure. The tension and compression limits are not symmetric.

Indicator States: Stress, Nonsymmetric

1 In the **Physics** toolbar, click **Global** and choose **Indicator States**.

2 In the **Settings** window for **Indicator States**, type `Indicator States: Stress, Nonsymmetric` in the **Label** text field.

3 Locate the **Indicator Variables** section. In the table, enter the following settings:

Name	g(v,vt,vtt,t)
Tension	intop1(solid.sl33)-500[MPa]
Compression	intop1(solid.sl33)+100[MPa]

Implicit Event 1

1 In the **Physics** toolbar, click **Global** and choose **Implicit Event**.

2 In the **Settings** window for **Implicit Event**, locate the **Event Conditions** section.

3 In the **Condition** text field, type `Tension>0`.

4 Locate the **Reinitialization** section. In the table, enter the following settings:

Variable	Expression
LoadingType	-1

Implicit Event 2

1 In the **Physics** toolbar, click **Global** and choose **Implicit Event**.

2 In the **Settings** window for **Implicit Event**, locate the **Event Conditions** section.

3 In the **Condition** text field, type `Compression<0`.

4 Locate the **Reinitialization** section. In the table, enter the following settings:

Variable	Expression
LoadingType	1

SOLID MECHANICS (SOLID)

1 In the **Model Builder** window, under **Component 1 (comp1)** click **Solid Mechanics (solid)**.

2 In the **Settings** window for **Solid Mechanics**, locate the **Structural Transient Behavior** section.

3 From the list, choose **Quasistatic**.

Symmetry Plane 1

1 In the **Physics** toolbar, click **Boundaries** and choose **Symmetry Plane**.

2 Select Boundary 2 only.

Prescribed Velocity 1

1 In the **Physics** toolbar, click **Boundaries** and choose **Prescribed Velocity**.

2 Select Boundary 9 only.

3 In the **Settings** window for **Prescribed Velocity**, locate the **Prescribed Velocity** section.

4 Select the **Prescribed in z direction** check box.

5 In the $\mathbf{v}_z$ text field, type `e0t*L0*step1(t)*LoadingType`.

Linear Elastic Material 1

Add viscoplasticity with combined isotropic and kinematic hardening.

1 In the **Model Builder** window, click **Linear Elastic Material 1**.

Viscoplasticity 1

1 In the **Physics** toolbar, click **Attributes** and choose **Viscoplasticity**.

2 Select Domains 1–3 only.

3 In the **Settings** window for **Viscoplasticity**, locate the **Viscoplasticity Model** section.

4 From the **Material model** list, choose **Chaboche**.

5 Find the **Isotropic hardening model** subsection. From the list, choose **Voce**.

6 Find the **Kinematic hardening model** subsection. From the list, choose **Armstrong-Frederick**.

Set the material properties.

MATERIALS

Material 1 (mat1)

1 In the **Model Builder** window, under **Component 1 (comp1)** right-click **Materials** and choose **Blank Material**.

2 In the **Settings** window for **Material**, locate the **Material Contents** section.

3 In the table, enter the following settings:

Property	Variable	Value	Unit	Property group
Young's modulus	E	`200[GPa]`	Pa	Young's modulus and Poisson's ratio
Poisson's ratio	nu	`0.3`	1	Young's modulus and Poisson's ratio
Density	rho	7500	kg/m³	Basic
Viscoplastic rate coefficient	A_cha	1	1/s	Chaboche viscoplasticity
Reference stress	sigRef_cha	`490[MPa]`	N/m²	Chaboche viscoplasticity
Stress exponent	n_cha	9	1	Chaboche viscoplasticity
Initial yield stress	sigmags	`60[MPa]`	Pa	Elastoplastic material model
Saturation flow stress	sigma_voc	`-35[MPa]`	Pa	Voce
Saturation exponent	beta_voc	200	1	Voce
Kinematic hardening modulus	Ck	`362.5[GPa]`	Pa	Armstrong-Frederick

The kinematic hardening parameter `gammak` is nonlinear and a function of the equivalent viscoplastic strain. Add the latter from the model input list to make it available.

4 In the **Model Builder** window, expand the **Material 1 (mat1)** node, then click **Armstrong-Frederick (ArmstrongFrederick)**.

5 In the **Settings** window for **Armstrong-Frederick**, locate the **Model Inputs** section.

6 Click **Select Quantity**.

7 In the **Physical Quantity** dialog box, type `viscoplastic` in the text field.

8 Click **Filter**.

9 In the tree, select **Solid Mechanics>Equivalent viscoplastic strain (1)**.

10 Click **OK**.

11 In the **Settings** window for **Armstrong-Frederick**, locate the **Output Properties** section.

12 In the table, enter the following settings:

Property	Variable	Expression	Unit	Size
Kinematic hardening parameter	gammak	`gammas+(gamma0-gammas)*exp(-beta*evpe)`	1	1x1

Create a mapped mesh.

MESH 1

Mapped 1

In the **Mesh** toolbar, click **Mapped**.

Distribution 1

1 Right-click **Mapped 1** and choose **Distribution**.

2 Select Boundary 8 only.

3 In the **Settings** window for **Distribution**, locate the **Distribution** section.

4 In the **Number of elements** text field, type `4`.

Distribution 2

1 In the **Model Builder** window, right-click **Mapped 1** and choose **Distribution**.

2 Select Boundary 12 only.

3 In the **Settings** window for **Distribution**, locate the **Distribution** section.

4 In the **Number of elements** text field, type `2`.

Distribution 3

1 Right-click **Mapped 1** and choose **Distribution**.

2 Select Boundary 10 only.

3 In the **Settings** window for **Distribution**, locate the **Distribution** section.

4 In the **Number of elements** text field, type `16`.

5 Select Boundary 1 only.

Distribution 4

1 Right-click **Mapped 1** and choose **Distribution**.

2 Select Boundary 3 only.

3 In the **Settings** window for **Distribution**, locate the **Distribution** section.

4 In the **Number of elements** text field, type 4.

Distribution 5

1 Right-click **Mapped 1** and choose **Distribution**.

2 Select Boundary 5 only.

Distribution 6

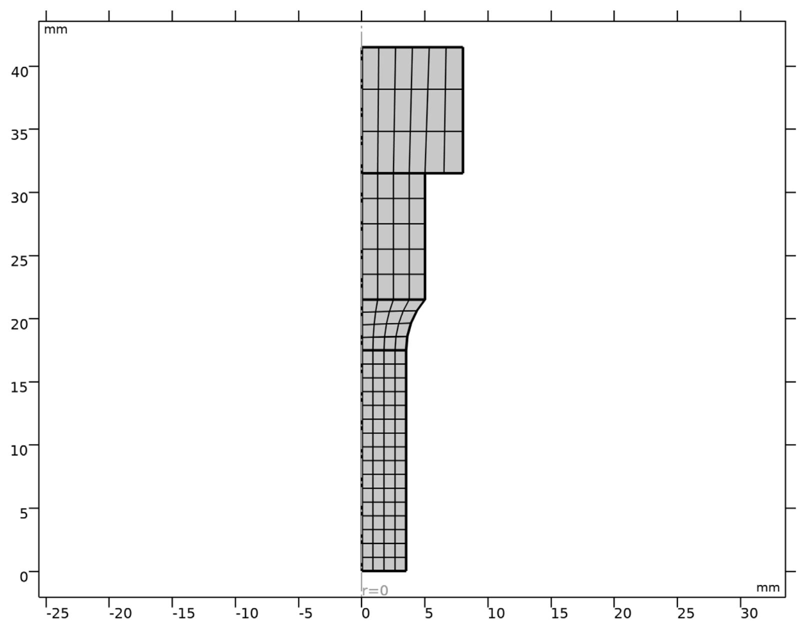
1 Right-click **Mapped 1** and choose **Distribution**.

2 Select Boundary 7 only.

3 In the **Settings** window for **Distribution**, locate the **Distribution** section.

4 In the **Number of elements** text field, type 3.

5 Click **Build All**.



STUDY 1

Step 1: Time Dependent

1 In the **Model Builder** window, under **Study 1** click **Step 1: Time Dependent**.

2 In the **Settings** window for **Time Dependent**, locate the **Study Settings** section.

3 In the **Output times** text field, type `0 40`.

For the first study, control the loading with symmetric strain cycles.

4 Locate the **Physics and Variables Selection** section. Select the **Modify model configuration for study step** check box.

5 In the tree, select **Component 1 (comp1)>Events (ev)>Indicator States: Stress, Symmetric**, **Component 1 (comp1)>Events (ev)>Indicator States: Strain, Nonsymmetric**, and **Component 1 (comp1)>Events (ev)>Indicator States: Stress, Nonsymmetric**.

6 Click **Disable**.

Solution 1 (sol1)

1 In the **Study** toolbar, click **Show Default Solver**.

2 In the **Model Builder** window, expand the **Solution 1 (sol1)** node, then click **Time-Dependent Solver 1**.

3 In the **Settings** window for **Time-Dependent Solver**, click to expand the **Time Stepping** section.

4 In the **Event tolerance** text field, type `0.001`.

5 Click to expand the **Output** section. Locate the **General** section. From the **Times to store** list, choose **Steps taken by solver**.

Only the start and end times are defined in the time step node. The solver sets the time steps automatically. Store results for all solved time steps.

6 Locate the **Output** section. Clear the **Store time derivatives** check box.

7 In the **Model Builder** window, click **Study 1**.

8 In the **Settings** window for **Study**, type `Study 1, Prescribed Symmetric Strain` in the **Label** text field.

9 Locate the **Study Settings** section. Clear the **Generate default plots** check box.

10 In the **Study** toolbar, click **Compute**.

Create a first group of plot groups for the results of the first study.

RESULTS

Prescribed Symmetric Strain

1 In the **Model Builder** window, right-click **Results** and choose **Node Group**.

2 In the **Settings** window for **Group**, type `Prescribed Symmetric Strain` in the **Label** text field.

3 Right-click **Prescribed Symmetric Strain** and choose **Move to Plot Groups**.

Add a first plot group to reproduce Figure 3.

Stress vs. Strain 1

1 In the **Home** toolbar, click **Add Plot Group** and choose **1D Plot Group**.

Add a plot group to reproduce Figure 2.

2 In the **Settings** window for **1D Plot Group**, type `Stress vs. Strain 1` in the **Label** text field.

3 Locate the **Plot Settings** section.

4 Select the **x-axis label** check box. In the associated text field, type `Axial strain (1)`.

5 Select the **y-axis label** check box. In the associated text field, type `Axial stress (MPa)`.

6 Click to expand the **Title** section. From the **Title type** list, choose **Manual**.

7 In the **Title** text area, type `Axial stress vs. axial strain`.

Point Graph 1

1 In the **Stress vs. Strain 1** toolbar, click **Point Graph**.

2 Select Point 6 only.

3 In the **Settings** window for **Point Graph**, click **Replace Expression** in the upper-right corner of the **y-Axis Data** section. From the menu, choose **Component 1 (comp1)>Solid Mechanics>Stress>Stress tensor, local coordinate system - N/m²>solid.sl33 - Stress tensor, local coordinate system, 33-component**.

4 Locate the **y-Axis Data** section. From the **Unit** list, choose **MPa**.

5 Locate the **x-Axis Data** section. From the **Parameter** list, choose **Expression**.

6 Click **Replace Expression** in the upper-right corner of the **x-Axis Data** section. From the menu, choose **Component 1 (comp1)>Solid Mechanics>Strain>Strain tensor, local coordinate system>solid.el33 - Strain tensor, local coordinate system, 33-component**.

7 In the **Stress vs. Strain 1** toolbar, click **Plot**.

Stresses vs. Time 1

1 In the **Home** toolbar, click **Add Plot Group** and choose **1D Plot Group**.

2 In the **Settings** window for **1D Plot Group**, type `Stresses vs. Time 1` in the **Label** text field.

3 Locate the **Title** section. From the **Title type** list, choose **Manual**.

4 In the **Title** text area, type `Stresses vs. time`.

Point Graph 1

1 Right-click **Stresses vs. Time 1** and choose **Point Graph**.

2 Select Point 6 only.

3 In the **Settings** window for **Point Graph**, click **Replace Expression** in the upper-right corner of the **y-Axis Data** section. From the menu, choose **Component 1 (comp1)>Solid Mechanics>Stress>Stress tensor, local coordinate system - N/m²>solid.sl33 - Stress tensor, local coordinate system, 33-component**.

4 Locate the **y-Axis Data** section. From the **Unit** list, choose **MPa**.

5 Click to expand the **Title** section. From the **Title type** list, choose **None**.

6 Click to expand the **Legends** section. Select the **Show legends** check box.

7 From the **Legends** list, choose **Manual**.

8 In the table, enter the following settings:

Legends
`Axial stress`

Point Graph 2

1 Right-click **Point Graph 1** and choose **Duplicate**.

2 In the **Settings** window for **Point Graph**, locate the **y-Axis Data** section.

3 In the **Expression** text field, type `max(0,solid.lemm1.vpl1.Fyield)`.

4 Locate the **Legends** section. In the table, enter the following settings:

Legends
`Viscous stress`

Point Graph 3

1 Right-click **Point Graph 2** and choose **Duplicate**.

2 In the **Settings** window for **Point Graph**, locate the **y-Axis Data** section.

3 In the **Expression** text field, type `solid.lemm1.vpl1.Sl_back33`.

4 Locate the **Legends** section. In the table, enter the following settings:

Legends
`Back stress`

Point Graph 4

1 Right-click **Point Graph 3** and choose **Duplicate**.

2 In the **Settings** window for **Point Graph**, locate the **y-Axis Data** section.

3 In the **Expression** text field, type `solid.lemm1.vpl1.sY`.

4 Locate the **Legends** section. In the table, enter the following settings:

Legends
`Yield strength`

Stresses vs. Time 1

1 In the **Model Builder** window, click **Stresses vs. Time 1**.

2 In the **Settings** window for **1D Plot Group**, locate the **Plot Settings** section.

3 Select the **y-axis label** check box. In the associated text field, type `Stress (MPa)`.

4 In the **Stresses vs. Time 1** toolbar, click **Plot**.

Plot the equivalent viscoplastic strain on the whole geometry.

Revolution 2D 1

In the **Results** toolbar, click **More Datasets** and choose **Revolution 2D**.

Mirror 3D 1

1 In the **Results** toolbar, click **More Datasets** and choose **Mirror 3D**.

2 In the **Settings** window for **Mirror 3D**, locate the **Plane Data** section.

3 From the **Plane** list, choose **XY-planes**.

Equivalent Viscoplastic Strain 1

1 In the **Results** toolbar, click **3D Plot Group**.

2 In the **Settings** window for **3D Plot Group**, type `Equivalent Viscoplastic Strain 1` in the **Label** text field.

3 Locate the **Data** section. From the **Dataset** list, choose **Mirror 3D 1**.

Surface 1

1 Right-click **Equivalent Viscoplastic Strain 1** and choose **Surface**.

2 In the **Settings** window for **Surface**, click **Replace Expression** in the upper-right corner of the **Expression** section. From the menu, choose **Component 1 (comp1)>Solid Mechanics>Strain (Gauss points)>solid.evpeGp - Equivalent viscoplastic strain**.

3 Locate the **Coloring and Style** section. Click **Change Color Table**.

4 In the **Color Table** dialog box, select **Aurora>AuroraAustralisDark** in the tree.

5 Click **OK**.

Deformation 1

Right-click **Surface 1** and choose **Deformation**.

Surface 2

1 In the **Model Builder** window, right-click **Equivalent Viscoplastic Strain 1** and choose **Surface**.

2 In the **Settings** window for **Surface**, locate the **Expression** section.

3 In the **Expression** text field, type `0`.

4 Click to expand the **Title** section. From the **Title type** list, choose **None**.

5 Click to expand the **Inherit Style** section. From the **Plot** list, choose **Surface 1**.

Deformation 1

Right-click **Surface 2** and choose **Deformation**.

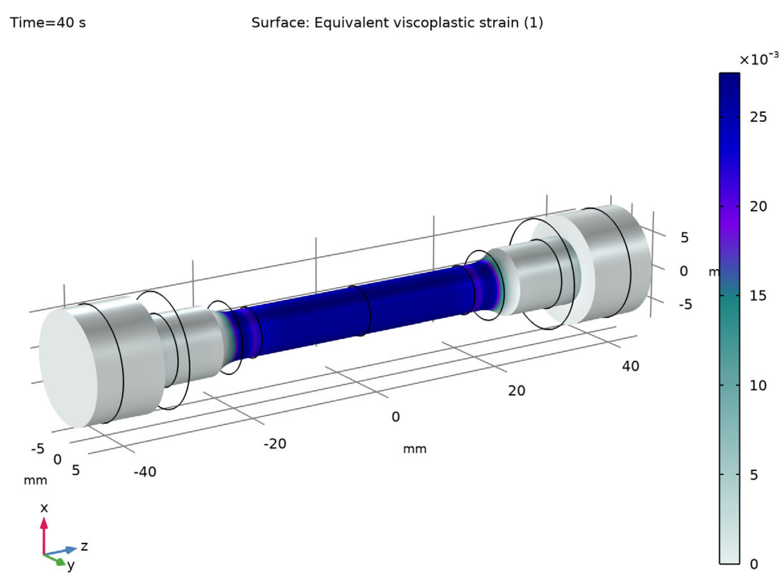
Filter 1

1 In the **Model Builder** window, right-click **Surface 2** and choose **Filter**.

2 In the **Settings** window for **Filter**, locate the **Element Selection** section.

3 In the **Logical expression for inclusion** text field, type `z>=31.5[mm]`.

4 In the **Equivalent Viscoplastic Strain 1** toolbar, click **Plot**.



Equivalent Viscoplastic Strain 1

Add a study to compute loading controlled by symmetric stress cycles.

ADD STUDY

1 In the **Home** toolbar, click **Add Study** to open the **Add Study** window.

2 Go to the **Add Study** window.

3 Find the **Studies** subsection. In the **Select Study** tree, select **General Studies> Time Dependent**.

4 Click **Add Study** in the window toolbar.

5 In the **Home** toolbar, click **Add Study** to close the **Add Study** window.

STUDY 2

Step 1: Time Dependent

1 In the **Settings** window for **Time Dependent**, locate the **Study Settings** section.

2 In the **Output times** text field, type `0 40`.

3 Locate the **Physics and Variables Selection** section. Select the **Modify model configuration for study step** check box.

4 In the tree, select **Component 1 (comp1)>Events (ev)>Indicator States: Strain, Symmetric**, **Component 1 (comp1)>Events (ev)>Indicator States: Strain, Nonsymmetric**, and **Component 1 (comp1)>Events (ev)>Indicator States: Stress, Nonsymmetric**.

5 Click **Disable**.

6 In the **Model Builder** window, click **Study 2**.

7 In the **Settings** window for **Study**, type `Study 2, Prescribed Symmetric Stress` in the **Label** text field.

8 Locate the **Study Settings** section. Clear the **Generate default plots** check box.

Solution 2 (sol2)

1 In the **Study** toolbar, click **Show Default Solver**.

2 In the **Model Builder** window, expand the **Solution 2 (sol2)** node, then click **Time-Dependent Solver 1**.

3 In the **Settings** window for **Time-Dependent Solver**, locate the **Time Stepping** section.

4 In the **Event tolerance** text field, type `0.001`.

5 Locate the **General** section. From the **Times to store** list, choose **Steps taken by solver**.

6 Locate the **Output** section. Clear the **Store time derivatives** check box.

7 In the **Study** toolbar, click **Compute**.

RESULTS

Revolution 2D 2

1 In the **Results** toolbar, click **More Datasets** and choose **Revolution 2D**.

2 In the **Settings** window for **Revolution 2D**, locate the **Data** section.

3 From the **Dataset** list, choose **Study 2, Prescribed Symmetric Stress/Solution 2 (sol2)**.

Mirror 3D 2

1 In the **Results** toolbar, click **More Datasets** and choose **Mirror 3D**.

2 In the **Settings** window for **Mirror 3D**, locate the **Data** section.

3 From the **Dataset** list, choose **Revolution 2D 2**.

4 Locate the **Plane Data** section. From the **Plane** list, choose **XY-planes**.

Prescribed Symmetric Stress

1 In the **Model Builder** window, right-click **Prescribed Symmetric Strain** and choose **Duplicate**.

2 In the **Settings** window for **Group**, type `Prescribed Symmetric Stress` in the **Label** text field.

Stress vs. Strain 2

1 In the **Model Builder** window, expand the **Prescribed Symmetric Stress** node, then click **Stress vs. Strain 1.1**.

2 In the **Settings** window for **1D Plot Group**, type `Stress vs. Strain 2` in the **Label** text field.

3 Locate the **Data** section. From the **Dataset** list, choose **Study 2, Prescribed Symmetric Stress/Solution 2 (sol2)**.

4 In the **Stress vs. Strain 2** toolbar, click **Plot**.

Stresses vs. Time 2

1 In the **Model Builder** window, under **Results>Prescribed Symmetric Stress** click **Stresses vs. Time 1.1**.

2 In the **Settings** window for **1D Plot Group**, type `Stresses vs. Time 2` in the **Label** text field.

3 Locate the **Data** section. From the **Dataset** list, choose **Study 2, Prescribed Symmetric Stress/Solution 2 (sol2)**.

Equivalent Viscoplastic Strain 2

1 In the **Model Builder** window, under **Results>Prescribed Symmetric Stress** click **Equivalent Viscoplastic Strain 1.1**.

2 In the **Settings** window for **3D Plot Group**, type `Equivalent Viscoplastic Strain 2` in the **Label** text field.

3 Locate the **Data** section. From the **Dataset** list, choose **Mirror 3D 2**.

4 Locate the **Plot Settings** section. From the **View** list, choose **View 3D 3**.

5 In the **Equivalent Viscoplastic Strain 2** toolbar, click **Plot**.

Add a study to compute loading controlled by nonsymmetric strain cycles.

ADD STUDY

1 In the **Home** toolbar, click **Add Study** to open the **Add Study** window.

2 Go to the **Add Study** window.

3 Find the **Studies** subsection. In the **Select Study** tree, select **General Studies> Time Dependent**.

4 Click **Add Study** in the window toolbar.

5 In the **Home** toolbar, click **Add Study** to close the **Add Study** window.

STUDY 3

Step 1: Time Dependent

1 In the **Settings** window for **Time Dependent**, locate the **Study Settings** section.

2 In the **Output times** text field, type `0 40`.

3 Locate the **Physics and Variables Selection** section. Select the **Modify model configuration for study step** check box.

4 In the tree, select **Component 1 (comp1)>Events (ev)>Indicator States: Strain, Symmetric**, **Component 1 (comp1)>Events (ev)>Indicator States: Stress, Symmetric**, and **Component 1 (comp1)>Events (ev)>Indicator States: Stress, Nonsymmetric**.

5 Click **Disable**.

Solution 3 (sol3)

1 In the **Study** toolbar, click **Show Default Solver**.

2 In the **Model Builder** window, expand the **Solution 3 (sol3)** node, then click **Time-Dependent Solver 1**.

3 In the **Settings** window for **Time-Dependent Solver**, locate the **Time Stepping** section.

4 In the **Event tolerance** text field, type `0.001`.

5 Locate the **General** section. From the **Times to store** list, choose **Steps taken by solver**.

6 Locate the **Output** section. Clear the **Store time derivatives** check box.

7 In the **Model Builder** window, click **Study 3**.

8 In the **Settings** window for **Study**, type `Study 3, Prescribed Nonsymmetric Strain` in the **Label** text field.

9 Locate the **Study Settings** section. Clear the **Generate default plots** check box.

10 In the **Study** toolbar, click **Compute**.

RESULTS

Revolution 2D 3

1 In the **Results** toolbar, click **More Datasets** and choose **Revolution 2D**.

2 In the **Settings** window for **Revolution 2D**, locate the **Data** section.

3 From the **Dataset** list, choose **Study 3, Prescribed Nonsymmetric Strain/Solution 3 (sol3)**.

Mirror 3D 3

1 In the **Results** toolbar, click **More Datasets** and choose **Mirror 3D**.

2 In the **Settings** window for **Mirror 3D**, locate the **Data** section.

3 From the **Dataset** list, choose **Revolution 2D 3**.

4 Locate the **Plane Data** section. From the **Plane** list, choose **XY-planes**.

Prescribed Nonsymmetric Strain

1 In the **Model Builder** window, right-click **Prescribed Symmetric Strain** and choose **Duplicate**.

2 In the **Settings** window for **Group**, type `Prescribed Nonsymmetric Strain` in the **Label** text field.

Stress vs. Strain 3

1 In the **Model Builder** window, expand the **Prescribed Nonsymmetric Strain** node, then click **Stress vs. Strain 1.1**.

2 In the **Settings** window for **1D Plot Group**, type `Stress vs. Strain 3` in the **Label** text field.

3 Locate the **Data** section. From the **Dataset** list, choose **Study 3, Prescribed Nonsymmetric Strain/Solution 3 (sol3)**.

4 In the **Stress vs. Strain 3** toolbar, click **Plot**.

Stresses vs. Time 3

1 In the **Model Builder** window, under **Results>Prescribed Nonsymmetric Strain** click **Stresses vs. Time 1.1**.

2 In the **Settings** window for **1D Plot Group**, type `Stresses vs. Time 3` in the **Label** text field.

3 Locate the **Data** section. From the **Dataset** list, choose **Study 3, Prescribed Nonsymmetric Strain/Solution 3 (sol3)**.

Equivalent Viscoplastic Strain 3

1 In the **Model Builder** window, under **Results>Prescribed Nonsymmetric Strain** click **Equivalent Viscoplastic Strain 1.1**.

2 In the **Settings** window for **3D Plot Group**, type `Equivalent Viscoplastic Strain 3` in the **Label** text field.

3 Locate the **Data** section. From the **Dataset** list, choose **Mirror 3D 3**.

4 Locate the **Plot Settings** section. From the **View** list, choose **View 3D 3**.

5 In the **Equivalent Viscoplastic Strain 3** toolbar, click **Plot**.

Add a study to compute loading controlled by nonsymmetric stress cycles.

ADD STUDY

1 In the **Home** toolbar, click **Add Study** to open the **Add Study** window.

2 Go to the **Add Study** window.

3 Find the **Studies** subsection. In the **Select Study** tree, select **General Studies> Time Dependent**.

4 Click **Add Study** in the window toolbar.

5 In the **Home** toolbar, click **Add Study** to close the **Add Study** window.

STUDY 4

Step 1: Time Dependent

1 In the **Settings** window for **Time Dependent**, locate the **Study Settings** section.

2 In the **Output times** text field, type `0 30`.

3 Locate the **Physics and Variables Selection** section. Select the **Modify model configuration for study step** check box.

4 In the tree, select **Component 1 (comp1)>Events (ev)>Indicator States: Strain, Symmetric**, **Component 1 (comp1)>Events (ev)>Indicator States: Stress, Symmetric**, and **Component 1 (comp1)>Events (ev)>Indicator States: Strain, Nonsymmetric**.

5 Click **Disable**.

Solution 4 (sol4)

1 In the **Study** toolbar, click **Show Default Solver**.

2 In the **Model Builder** window, expand the **Solution 4 (sol4)** node, then click **Time-Dependent Solver 1**.

3 In the **Settings** window for **Time-Dependent Solver**, locate the **Time Stepping** section.

4 In the **Event tolerance** text field, type `0.001`.

5 Locate the **General** section. From the **Times to store** list, choose **Steps taken by solver**.

6 Locate the **Output** section. Clear the **Store time derivatives** check box.

7 In the **Model Builder** window, click **Study 4**.

8 In the **Settings** window for **Study**, type `Study 4, Prescribed Nonsymmetric Stress` in the **Label** text field.

9 Locate the **Study Settings** section. Clear the **Generate default plots** check box.

10 In the **Study** toolbar, click **Compute**.

RESULTS

Revolution 2D 4

1 In the **Results** toolbar, click **More Datasets** and choose **Revolution 2D**.

2 In the **Settings** window for **Revolution 2D**, locate the **Data** section.

3 From the **Dataset** list, choose **Study 4, Prescribed Nonsymmetric Stress/Solution 4 (sol4)**.

Mirror 3D 4

1 In the **Results** toolbar, click **More Datasets** and choose **Mirror 3D**.

2 In the **Settings** window for **Mirror 3D**, locate the **Data** section.

3 From the **Dataset** list, choose **Revolution 2D 4**.

4 Locate the **Plane Data** section. From the **Plane** list, choose **XY-planes**.

Prescribed Nonsymmetric Stress

1 In the **Model Builder** window, right-click **Prescribed Symmetric Strain** and choose **Duplicate**.

2 In the **Settings** window for **Group**, type `Prescribed Nonsymmetric Stress` in the **Label** text field.

Stress vs. Strain 4

1 In the **Model Builder** window, expand the **Prescribed Nonsymmetric Stress** node, then click **Stress vs. Strain 1.1**.

2 In the **Settings** window for **1D Plot Group**, type `Stress vs. Strain 4` in the **Label** text field.

3 Locate the **Data** section. From the **Dataset** list, choose **Study 4, Prescribed Nonsymmetric Stress/Solution 4 (sol4)**.

4 In the **Stress vs. Strain 4** toolbar, click **Plot**.

Stresses vs. Time 4

1 In the **Model Builder** window, under **Results>Prescribed Nonsymmetric Stress** click **Stresses vs. Time 1.1**.

2 In the **Settings** window for **1D Plot Group**, type `Stresses vs. Time 4` in the **Label** text field.

3 Locate the **Data** section. From the **Dataset** list, choose **Study 4, Prescribed Nonsymmetric Stress/Solution 4 (sol4)**.

Equivalent Viscoplastic Strain 4

1 In the **Model Builder** window, under **Results>Prescribed Nonsymmetric Stress** click **Equivalent Viscoplastic Strain 1.1**.

2 In the **Settings** window for **3D Plot Group**, type `Equivalent Viscoplastic Strain 4` in the **Label** text field.

3 Locate the **Data** section. From the **Dataset** list, choose **Mirror 3D 4**.

4 Locate the **Plot Settings** section. From the **View** list, choose **View 3D 3**.

5 In the **Equivalent Viscoplastic Strain 4** toolbar, click **Plot**.

Add an animation to show the increase of equivalent viscoplastic strain over time.

Equivalent Viscoplastic Strain

1 In the **Results** toolbar, click **Animation** and choose **Player**.

2 In the **Settings** window for **Animation**, type `Equivalent Viscoplastic Strain` in the **Label** text field.

3 Locate the **Scene** section. From the **Subject** list, choose **Equivalent Viscoplastic Strain 1**.

4 Locate the **Frames** section. In the **Number of frames** text field, type `50`.

5 Click the **Play** button in the **Graphics** toolbar.