Created in COMSOL Multiphysics 6.0

Primary Creep Under Nonconstant Load

Introduction

In this model example, you study the creep behavior of material under nonconstant loading. You model the primary creep using a Norton-Bailey law and study the difference between the time hardening and the strain hardening methods available in COMSOL Multiphysics.

Time hardening assumes that the creep strain rate depends on the current stress and on the time passed from the start of the test. Strain hardening assumes that the creep strain rate depends on the current stress and the accumulated creep strain. Therefore, once the stress level changes, in time hardening the material follows the new behavior from the point where the times are equal, the strain-time relation shifts vertically. In strain hardening, the material follows the new behavior form the point where the strains are equal, the strain-time behavior shifts horizontally. This is illustrated in Figure 1 below.

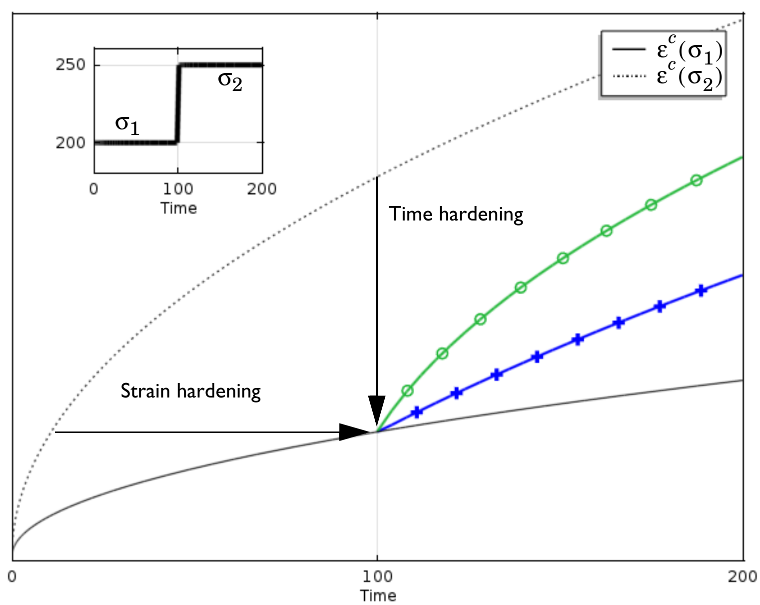

Figure 1: Strain versus time curve for different primary creep formulations: time hardening in blue(+) and strain hardening in green (o). The load case is represented in the thumbnail in the upper-left corner.

The time hardening formulation is easier to use, while the strain hardening is usually considered to be more accurate.

The model is taken from *NAFEMS Understanding Non-Linear Finite Analysis Through Illustrative Benchmarks* (Ref. 1). The load consist of a uniaxial and a biaxial stepped load. The step in the load occurs at after a half of the full study time. The value of interest is the creep strain variation along the time. The computed solutions are compared with analytical solution given in the reference.

Additionally, a short discussion describes how to avoid nonphysical creep strains that can appear at the initial time step when a singular load condition is applied at early time.

Model Definition

The problem consists of a 100 mm length square plate under uniaxial and biaxial load cases. Different boundary constraints for each load case ensure uniaxial stress. After 100 hours, the applied load jump from 200 MPa to 250 MPa.

The thickness of the plate is assumed to be small enough to use the 2D plane stress assumption.

The Norton-Bailey that model the creep behavior is represented with the following creep strain definition:

$$\frac{\partial \varepsilon_c}{\partial t} = A\sigma^n t^m$$

The material constants are listed in the table below:

E	2e5[MPa]
nu	0.3
A	3.125e-14[1/h]
n	5
m	0.5

Both the time hardening and the strain hardening are used to represent the step load response versus time.

Results and Discussion

The problem has analytical results for both uniaxial and biaxial load cases, and for both the time and strain hardening formulations.

For the uniaxial load case, the target solution for the x-component of the creep strain is represented by the following expression:

$$\varepsilon_{c,xx} = \begin{cases} 0.01t^{0.5} & t < 100\text{h} \\ 0.03052t^{0.5} - 0.2052 & t > 100\text{h, time hardening} \\ 0.03052(t - 89.262)^{0.5} & t > 100\text{h, strain hardening} \end{cases}$$

The target solution for the biaxial load case is represented by the following expression:

$$\varepsilon_{c,xx} = \begin{cases} 0.005t^{0.5} & t < 100\text{h} \\ 0.01529t^{0.5} - 0.1028 & t > 100\text{h, time hardening} \\ 0.01529(t - 89.262)^{0.5} & t > 100\text{h, strain hardening} \end{cases}$$

In Figure 2, you can see the results of the computed x-component of the creep strain for the uniaxial load case together with the target data (represented with markers).

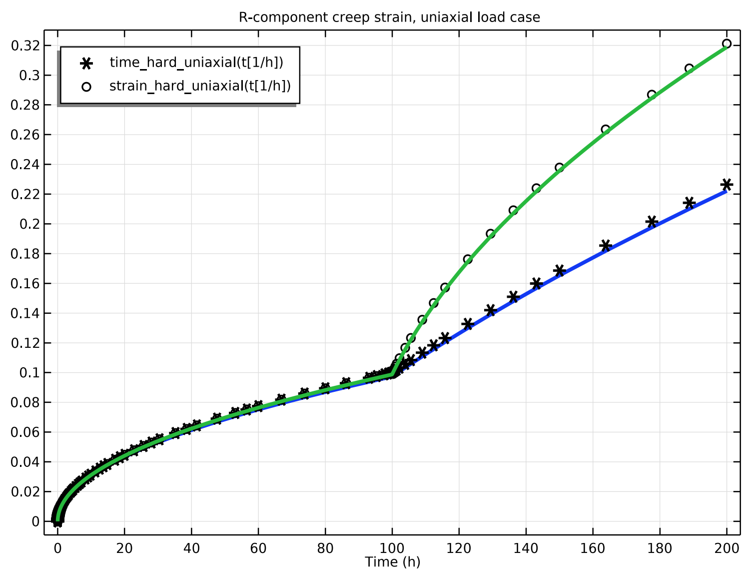

Figure 2: The creep strain for time hardening (blue line) and strain hardening (green line) for the uniaxial load case. The reference data is represented by markers.

In the Figure 3, you can see the results of the computed x-component of the creep strain for the biaxial load case together with the target data (represented with markers).

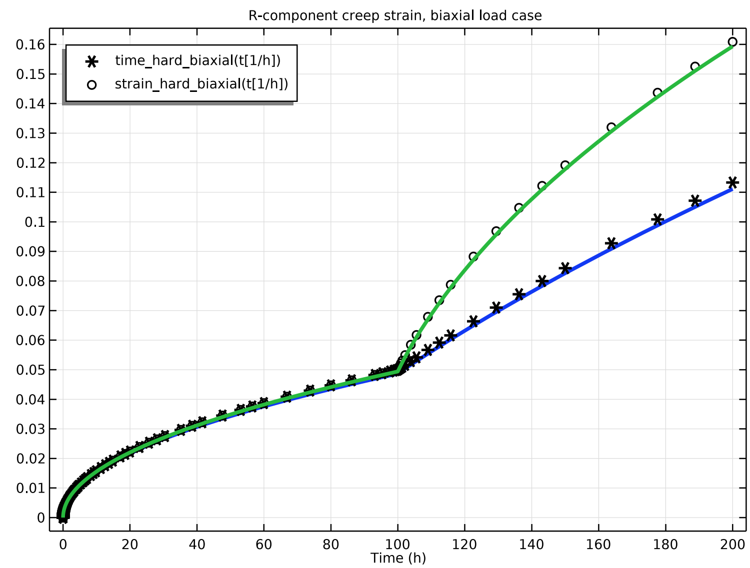

Figure 3: The creep strain for time hardening (blue line) and strain hardening (green line) for the biaxial load case. The reference data is represented by markers.

The computed solutions agree very well with the analytical target for both the uniaxial and the biaxial load cases.

Notes About the COMSOL Implementation

A constant load at the initial time introduces a stress singularity. This can be a source of errors when the strain is defined via a strain rate formulation. At an infinitesimal time step, a nonphysical strain rate can be generated. You can avoid such a singularity by defining the load case using a smooth increase in time. Another solution is to enforce the initial computational time step to be large enough so that the creep strain is reduced. In COMSOL Multiphysics, the time hardening implementation makes it possible to define a time offset. Adjusting the time offset with the initial computational time step ensures to reduce the initial creep strain error. The time should be small compared to the computational study time. In the current example, a time of 1 second as been found sufficient.

For the strain hardening implementation, an initial strain value is requested to avoid an error related to computation a non-integer power of a negative number. Here, a value of 1×10^{-5} is found to be sufficient.

Reference

1. A.A. Becker, *Understanding Non-Linear Finite Element Analysis Through Illustrative Benchmarks,* NAFEMS R0080, 2001.

Application Library path: `Nonlinear_Structural_Materials_Module/Creep/variable_load_creep`

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**.
2 In the **Select Physics** tree, select **Structural Mechanics>Solid Mechanics (solid)**.
3 Click **Add**.
4 Click **Study**.
5 In the **Select Study** tree, select **General Studies>Time Dependent**.
6 Click **Done**.

ROOT

1 In the **Model Builder** window, click the root node.
2 In the root node's **Settings** window, locate the **Unit System** section.
3 From the **Unit system** list, choose **MPa**.

GLOBAL DEFINITIONS

Parameters 1

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
A	3.125e-14[1/h]	8.6806E-18 1/s	Creep rate coefficient
n	5	5	Stress exponent
m	0.5	0.5	Power law exponent
t0	1[s]	1 s	Time offset

DEFINITIONS

Step 1 (step1)

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

2 In the **Settings** window for **Step**, type `load` in the **Function name** text field.

3 Locate the **Parameters** section. In the **Location** text field, type `100`.

4 In the **From** text field, type `200`.

5 In the **To** text field, type `250`.

GLOBAL DEFINITIONS

Piecewise 1 (pw1)

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

2 In the **Settings** window for **Piecewise**, type `time_hard_uniaxial` in the **Function name** text field.

3 Locate the **Definition** section. In the **Argument** text field, type `t`.

4 Find the **Intervals** subsection. In the table, enter the following settings:

Start	End	Function
0	100	0.01*t^0.5
100	200	0.03052*t^0.5-0.2052

Piecewise 2 (pw2)

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

2 In the **Settings** window for **Piecewise**, type `strain_hard_uniaxial` in the **Function name** text field.

3 Locate the **Definition** section. In the **Argument** text field, type `t`.

4 Find the **Intervals** subsection. In the table, enter the following settings:

Start	End	Function
0	100	0.01*t^0.5
100	200	0.03052*(t-89.262)^0.5

Piecewise 3 (pw3)

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

2 In the **Settings** window for **Piecewise**, type `time_hard_biaxial` in the **Function name** text field.

3 Locate the **Definition** section. In the **Argument** text field, type `t`.

4 Find the **Intervals** subsection. In the table, enter the following settings:

Start	End	Function
0	100	0.005*t^0.5
100	200	0.01529*t^0.5-0.1029

Piecewise 4 (pw4)

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

2 In the **Settings** window for **Piecewise**, type `strain_hard_biaxial` in the **Function name** text field.

3 Locate the **Definition** section. In the **Argument** text field, type `t`.

4 Find the **Intervals** subsection. In the table, enter the following settings:

Start	End	Function
0	100	0.005*t^0.5
100	200	0.01529*(t-89.262)^0.5

GEOMETRY 1

Square 1 (sq1)

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

2 In the **Settings** window for **Square**, locate the **Size** section.

3 In the **Side length** text field, type `100[mm]`.

4 Click **Build Selected**.

5 Click the **Zoom Extents** button in the **Graphics** toolbar.

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 **2D Approximation** section.

3 From the list, choose **Plane stress**.

4 Locate the **Structural Transient Behavior** section. From the list, choose **Quasistatic**.

Linear Elastic Material 1

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

Creep 1

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

2 In the **Settings** window for **Creep**, locate the **Creep Model** section.

3 Find the **Isotropic hardening model** subsection. From the $h(\varepsilon_{ce}, t)$ list, choose **Time hardening**.

4 In the m text field, type `m`.

5 In the t_{shift} text field, type `t0`.

Linear Elastic Material 1

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

Creep 2

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

2 In the **Settings** window for **Creep**, locate the **Creep Model** section.

3 Find the **Isotropic hardening model** subsection. From the $h(\varepsilon_{ce}, t)$ list, choose **Strain hardening**.

4 In the m text field, type `m`.

Roller 1

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

2 Select Boundaries 1 and 2 only.

Uniaxial Boundary Load

1 In the **Physics** toolbar, click **Boundaries** and choose **Boundary Load**.

2 In the **Settings** window for **Boundary Load**, type `Uniaxial Boundary Load` in the **Label** text field.

3 Locate the **Coordinate System Selection** section. From the **Coordinate system** list, choose **Boundary System 1 (sys1)**.

4 Select Boundary 4 only.

5 Locate the **Force** section. Specify the $\mathbf{F}_A$ vector as

0	t1
load(t[1/h])	n

Biaxial Boundary Load

1 In the **Physics** toolbar, click **Boundaries** and choose **Boundary Load**.

2 In the **Settings** window for **Boundary Load**, type `Biaxial Boundary Load` in the **Label** text field.

3 Select Boundaries 3 and 4 only.

4 Locate the **Coordinate System Selection** section. From the **Coordinate system** list, choose **Boundary System 1 (sys1)**.

5 Locate the **Force** section. Specify the $\mathbf{F}_A$ vector as

0	t1
load(t[1/h])	n

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	200e3[MPa]	Pa	Young's modulus and Poisson's ratio
Poisson's ratio	nu	0.3	1	Young's modulus and Poisson's ratio
Density	rho	0	t/mm³	Basic
Creep rate coefficient	A_nor	A	1/s	Norton

Property	Variable	Value	Unit	Property group
Reference stress	sigRef_nor	`1[MPa]`	N/m²	Norton
Stress exponent	n_nor	`n`	1	Norton

MESH 1

Mapped 1

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

Distribution 1

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

2 Select Boundaries 1 and 2 only.

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

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

5 Click **Build All**.

TIME HARDENING, UNIAXIAL

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

2 In the **Settings** window for **Study**, type `Time Hardening, Uniaxial` in the **Label** text field.

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

Step 1: Time Dependent

Store output data at the time instances where target values are specified in the benchmark.

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

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

3 From the **Time unit** list, choose **h**.

4 In the **Output times** text field, type `0 20 60 100 150 200`.

Disable unneeded features from the solver node.

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

2 In the tree, select **Component 1 (Comp1)>Solid Mechanics (Solid)>Linear Elastic Material 1>Creep 2**.

3 Click **Disable**.

4 In the tree, select **Component 1 (Comp1)>Solid Mechanics (Solid)>Biaxial Boundary Load**.

5 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 **Absolute Tolerance** section.

4 From the **Tolerance method** list, choose **Manual**.

5 In the **Absolute tolerance** text field, type `1e-4`.

6 Click to expand the **Time Stepping** section. Select the **Initial step** check box.

7 From the **Steps taken by solver** list, choose **Strict**.

8 In the **Initial step** text field, type `t0`.

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

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

11 In the **Results** toolbar, click **Cut Point 2D**.

RESULTS

Cut Point 2D 1

1 In the **Settings** window for **Cut Point 2D**, locate the **Point Data** section.

2 In the **X** text field, type `50[mm]`.

3 In the **Y** text field, type `50[mm]`.

Creep Strain, Uniaxial

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

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

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

4 In the **Title** text area, type `R-component creep strain, uniaxial load case`.

5 Locate the **Legend** section. From the **Position** list, choose **Upper left**.

Point Graph 1

1 Right-click **Creep Strain, Uniaxial** and choose **Point Graph**.

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

3 From the **Dataset** list, choose **Cut Point 2D 1**.

4 Click **Replace Expression** in the upper-right corner of the **y-Axis Data** section. From the menu, choose **Component 1 (comp1)>Solid Mechanics>Strain (Gauss points)> Creep strain tensor, local coordinate system>solid.eclGp11 - Creep strain tensor, local coordinate system, 11 component**.

5 Click to expand the **Coloring and Style** section. In the **Width** text field, type 3.

Global 1

1 In the **Model Builder** window, right-click **Creep Strain, Uniaxial** and choose **Global**.

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

3 In the table, enter the following settings:

Expression	Unit	Description
time_hard_uniaxial(t[1/h])		

4 Click to expand the **Coloring and Style** section. Find the **Line style** subsection. From the **Line** list, choose **None**.

5 From the **Color** list, choose **From theme**.

6 Find the **Line markers** subsection. From the **Marker** list, choose **Cycle**.

7 From the **Positioning** list, choose **In data points**.

8 Click to expand the **Legends** section. Find the **Include** subsection. Select the **Expression** check box.

9 In the **Creep Strain, Uniaxial** toolbar, click **Plot**.

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.

STRAIN HARDENING, UNIAXIAL

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

2 In the **Settings** window for **Study**, type `Strain Hardening, Uniaxial` in the **Label** text field.

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

Step 1: Time Dependent

1 In the **Model Builder** window, under **Strain Hardening, Uniaxial** click **Step 1: Time Dependent**.

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

3 From the **Time unit** list, choose **h**.

4 In the **Output times** text field, type `0 20 60 100 150 200`.

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

6 In the tree, select **Component 1 (Comp1)>Solid Mechanics (Solid)>Biaxial Boundary Load**.

7 Click **Disable**.

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 **Absolute Tolerance** section.

4 From the **Tolerance method** list, choose **Manual**.

5 In the **Absolute tolerance** text field, type `1e-4`.

6 Locate the **Time Stepping** section. From the **Steps taken by solver** list, choose **Strict**.

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

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

RESULTS

Cut Point 2D 2

1 In the **Model Builder** window, under **Results>Datasets** right-click **Cut Point 2D 1** and choose **Duplicate**.

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

3 From the **Dataset** list, choose **Strain Hardening, Uniaxial/Solution 2 (sol2)**.

Point Graph 2

1 In the **Model Builder** window, under **Results>Creep Strain, Uniaxial** right-click **Point Graph 1** and choose **Duplicate**.

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

3 From the **Dataset** list, choose **Cut Point 2D 2**.

Global 1

1 In the **Model Builder** window, click **Global 1**.

2 In the **Creep Strain, Uniaxial** toolbar, click **Plot**.

3 In the **Settings** window for **Global**, locate the **y-Axis Data** section.

4 In the table, enter the following settings:

Expression	Unit	Description
time_hard_uniaxial(t[1/h])		
strain_hard_uniaxial(t[1/h])		

5 In the **Creep Strain, Uniaxial** toolbar, click **Plot**.

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.

TIME HARDENING, BIAXIAL

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

2 In the **Settings** window for **Study**, type `Time Hardening, Biaxial` in the **Label** text field.

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

Step 1: Time Dependent

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

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

3 From the **Time unit** list, choose **h**.

4 In the **Output times** text field, type `0 20 60 100 150 200`.

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

6 In the tree, select **Component 1 (Comp1)>Solid Mechanics (Solid)>Linear Elastic Material 1>Creep 2**.

7 Click **Disable**.

8 In the tree, select **Component 1 (Comp1)>Solid Mechanics (Solid)>Uniaxial Boundary Load**.

9 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 **Absolute Tolerance** section.

4 From the **Tolerance method** list, choose **Manual**.

5 In the **Absolute tolerance** text field, type `1e-4`.

6 Locate the **Time Stepping** section. From the **Steps taken by solver** list, choose **Strict**.

7 Select the **Initial step** check box.

8 In the associated text field, type `t0`.

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

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

RESULTS

Cut Point 2D 3

1 In the **Model Builder** window, under **Results>Datasets** right-click **Cut Point 2D 2** and choose **Duplicate**.

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

3 From the **Dataset** list, choose **Time Hardening, Biaxial/Solution 3 (sol3)**.

Creep Strain, Biaxial

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

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

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

4 In the **Title** text area, type `R-component creep strain, biaxial load case`.

5 Locate the **Legend** section. From the **Position** list, choose **Upper left**.

Point Graph 1

1 Right-click **Creep Strain, Biaxial** and choose **Point Graph**.

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

3 From the **Dataset** list, choose **Cut Point 2D 3**.

4 Click **Replace Expression** in the upper-right corner of the **y-Axis Data** section. From the menu, choose **Component 1 (comp1)>Solid Mechanics>Strain (Gauss points)> Creep strain tensor, local coordinate system>solid.eclGp11 - Creep strain tensor, local coordinate system, 11 component**.

5 Locate the **Coloring and Style** section. In the **Width** text field, type 3.

Global 1

1 In the **Model Builder** window, right-click **Creep Strain, Biaxial** and choose **Global**.

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

3 In the table, enter the following settings:

Expression	Unit	Description
time_hard_biaxial(t[1/h])		

4 Locate the **Coloring and Style** section. Find the **Line style** subsection. From the **Line** list, choose **None**.

5 From the **Color** list, choose **From theme**.

6 Find the **Line markers** subsection. From the **Marker** list, choose **Cycle**.

7 From the **Positioning** list, choose **In data points**.

8 Locate the **Legends** section. Find the **Include** subsection. Select the **Expression** check box.

9 In the **Creep Strain, Biaxial** toolbar, click **Plot**.

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.

STRAIN HARDENING, BIAXIAL

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

2 In the **Settings** window for **Study**, type `Strain Hardening, Biaxial` in the **Label** text field.

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

Step 1: Time Dependent

1 In the **Model Builder** window, under **Strain Hardening, Biaxial** click **Step 1: Time Dependent**.

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

3 From the **Time unit** list, choose **h**.

4 In the **Output times** text field, type `0 20 60 100 150 200`.

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

6 In the tree, select **Component 1 (Comp1)>Solid Mechanics (Solid)>Uniaxial Boundary Load**.

7 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 **Absolute Tolerance** section.

4 From the **Tolerance method** list, choose **Manual**.

5 In the **Absolute tolerance** text field, type `1e-4`.

6 Locate the **Time Stepping** section. From the **Steps taken by solver** list, choose **Strict**.

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

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

RESULTS

Cut Point 2D 4

1 In the **Model Builder** window, under **Results>Datasets** right-click **Cut Point 2D 3** and choose **Duplicate**.

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

3 From the **Dataset** list, choose **Strain Hardening, Biaxial/Solution 4 (sol4)**.

Point Graph 2

1 In the **Model Builder** window, under **Results>Creep Strain, Biaxial** right-click **Point Graph 1** and choose **Duplicate**.

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

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

Global 1

1 In the **Model Builder** window, click **Global 1**.

2 In the **Creep Strain, Biaxial** toolbar, click **Plot**.

3 In the **Settings** window for **Global**, locate the **y-Axis Data** section.

4 In the table, enter the following settings:

Expression	Unit	Description
`time_hard_biaxial(t[1/h])`		
`strain_hard_biaxial(t[1/h])`		

5 In the **Creep Strain, Biaxial** toolbar, click **Plot**.