Created in COMSOL Multiphysics 6.2

Primary Creep Under Nonconstant Load

Introduction

In this model example, you study the creep behavior of a 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.

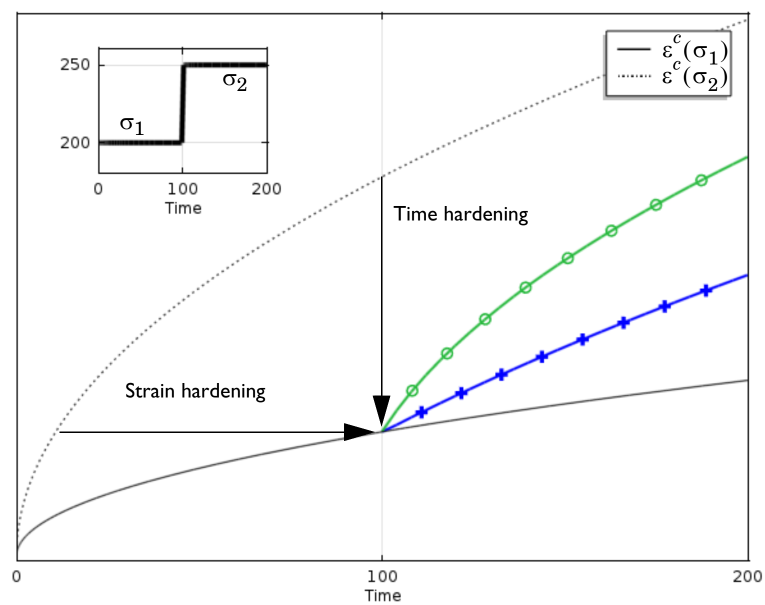

Figure 1: Strain versus time curve for different primary creep formulations: time hardening in blue(+) and strain hardening in green (o). The load case is represented in the thumbnail in the upper-left corner.

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 consists of a uniaxial and a biaxial stepped load. The step in the load occurs after half of the full study time. The value of interest is the creep strain variation versus time. The computed solutions are compared with an analytical solution given in the reference.

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

Model Definition

The problem consists of a square plate with the side length 100 mm under uniaxial and biaxial load cases. Different boundary constraints for each load case ensure uniaxial stress. After 100 hours, the applied load jumps 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 law used to model the creep behavior is represented by 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).

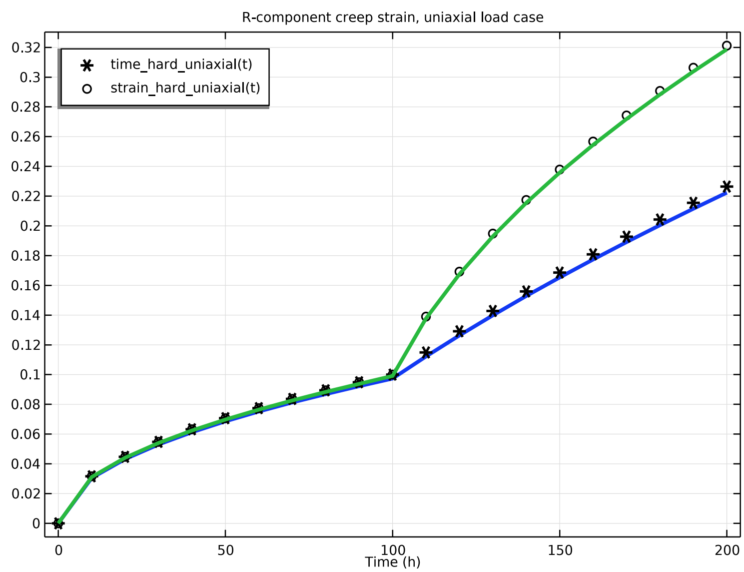

Figure 2: The creep strain for time hardening (blue line) and strain hardening (green line) for the uniaxial load case. The reference data is represented by markers.

In 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).

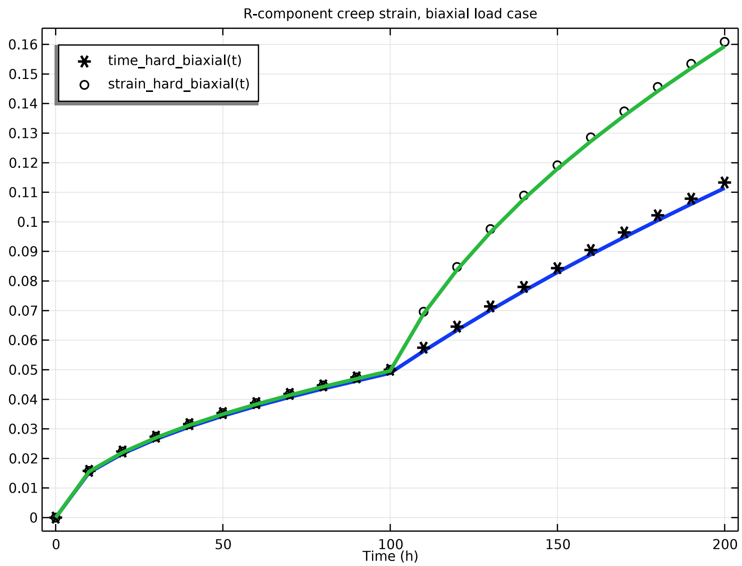

Figure 3: The creep strain for time hardening (blue line) and strain hardening (green line) for the biaxial load case. The reference data is represented by markers.

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, an unphysical 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 noninteger power of a negative number. Here, a value of 1×10^{-5} is found to be sufficient.

In this model, the **Test material** feature is used to automatize the setup of uniaxial and biaxial tests for the two different creep models.

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 **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

Step 1 (step1)

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

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

5 Locate the **Units** section. In the **Arguments** text field, type `h`.

6 In the **Function** text field, type `1`.

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`

5 Locate the **Units** section. In the **Arguments** text field, type `h`.

6 In the **Function** text field, type `1`.

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`

5 Locate the **Units** section. In the **Arguments** text field, type `h`.

6 In the **Function** text field, type `1`.

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`

5 Locate the **Units** section. In the **Arguments** text field, type `h`.

6 In the **Function** text field, type `1`.

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**.

Array 1 (arr1)

1 In the **Geometry** toolbar, click **Transforms** and choose **Array**.

2 Select the object **sq1** only.

3 In the **Settings** window for **Array**, locate the **Size** section.

4 In the **x size** text field, type `2`.

5 Locate the **Displacement** section. In the **x** text field, type `200[mm]`.

6 Click **Build Selected**.

7 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 [with Time Hardening Creep]

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

2 In the **Settings** window for **Linear Elastic Material**, type `Linear Elastic Material [with Time Hardening Creep]` in the **Label** text field.

3 Right-click **Linear Elastic Material [with Time Hardening Creep]** and choose **Duplicate**.

Linear Elastic Material [with Strain Hardening Creep]

1 In the **Model Builder** window, under **Component 1 (comp1)>Solid Mechanics (solid)** click **Linear Elastic Material [with Time Hardening Creep] 1**.

2 In the **Settings** window for **Linear Elastic Material**, type `Linear Elastic Material [with Strain Hardening Creep]` in the **Label** text field.

3 Locate the **Domain Selection** section. Click **Clear Selection**.

4 Select Domain 2 only.

Creep 1

1 In the **Model Builder** window, expand the **Linear Elastic Material [with Strain Hardening Creep]** node, then click **Creep 1**.

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`.

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/mm^3	Basic
Creep rate coefficient	A_nor	A	1/s	Norton
Reference stress	sigRef_nor	`1[MPa]`	N/m^2	Norton
Stress exponent	n_nor	n	1	Norton

Use the **Test Material** feature to carry out uniaxial and biaxial tests with time hardening and strain hardening creep.

SOLID MECHANICS (SOLID)

Test Material [Time Hardening Creep]

1 In the **Physics** toolbar, click **Global** and choose **Test Material**.

2 In the **Settings** window for **Test Material**, type `Test Material [Time Hardening Creep]` in the **Label** text field.

3 Select Domain 1 only.

4 Locate the **Material Tests** section. From the **Study setup** list, choose **Time dependent**.

5 In the N_p text field, type `20`.

6 In the t_f text field, type `200[h]`.

7 From the **Test setup** list, choose **User defined**.

8 From the **Test control** list, choose **Force driven**.

9 Find the **Tests** subsection. In the f text field, type `load(t[1/h])`.

10 Select the **Biaxial test** check box.

11 In the f text field, type `load(t[1/h])`.

12 Click **Automated Model Setup** in the upper-right corner of the **Material Tests** section. From the menu, choose **Set up Tests**.

13 In the **Model Builder** window, right-click **Test Material [Time Hardening Creep]** and choose **Duplicate**.

Test Material [Strain Hardening Creep]

1 In the **Model Builder** window, under **Component 1 (comp1)>Solid Mechanics (solid)** click **Test Material [Time Hardening Creep] 1**.

2 In the **Settings** window for **Test Material**, type `Test Material [Strain Hardening Creep]` in the **Label** text field.

3 Locate the **Domain Selection** section. Click **Clear Selection**.

4 Select Domain 2 only.

5 Click **Automated Model Setup** in the upper-right corner of the **Material Tests** section. From the menu, choose **Set up Tests**.

RESULTS

Creep Strain, Uniaxial

1 In the **Model Builder** window, expand the **Results** node.

2 Right-click **Results** and choose **1D Plot Group**.

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

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

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

6 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 **Study: Test Material [Time Hardening Creep]/ Solution 1 (4) (solidtm1sol1)**.

4 From the **Parameter selection (testPara_solidtm1)** list, choose **First**.

5 Select Point 8 only.

6 Click **Replace Expression** in the upper-right corner of the **y-Axis Data** section. From the menu, choose **Component: Test Material [Time Hardening Creep] (solidtm1comp)> Solid Mechanics>Strain>Creep strain tensor, local coordinate system>solid1.eclGp11 - Creep strain tensor, local coordinate system, 11-component**.

7 Locate the **x-Axis Data** section. From the **Unit** list, choose **h**.

8 Click to expand the **Coloring and Style** section. From the **Width** list, choose **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	

4 Locate the **x-Axis Data** section. From the **Unit** list, choose **h**.

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

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

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

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

Point Graph 1

In the **Model Builder** window, right-click **Point Graph 1** and choose **Duplicate**.

Point Graph 2

1 In the **Model Builder** window, click **Point Graph 2**.

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

3 From the **Dataset** list, choose **Study: Test Material [Strain Hardening Creep]/ Solution 1a (10) (solidtm2sol1)**.

4 Locate the **Selection** section. Click to select the **Activate Selection** toggle button.

5 Select Point 8 only.

6 Click **Replace Expression** in the upper-right corner of the **y-Axis Data** section. From the menu, choose **Component: Test Material [Strain Hardening Creep] (solidtm2comp)> Solid Mechanics>Strain>Creep strain tensor, local coordinate system>solid2.eclGp11 - Creep strain tensor, local coordinate system, 11-component**.

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)		
strain_hard_uniaxial(t)	1	

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

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 **Study: Test Material [Time Hardening Creep]/ Solution 1 (4) (solidtm1sol1)**.

4 From the **Parameter selection (testPara_solidtm1)** list, choose **Last**.

5 Select Point 8 only.

6 Click **Replace Expression** in the upper-right corner of the **y-Axis Data** section. From the menu, choose **Component: Test Material [Time Hardening Creep] (solidtm1comp)> Solid Mechanics>Strain>Creep strain tensor, local coordinate system>solid1.eclGp11 - Creep strain tensor, local coordinate system, 11-component**.

7 Locate the **x-Axis Data** section. From the **Unit** list, choose **h**.

8 Locate the **Coloring and Style** section. From the **Width** list, choose **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	

4 Locate the **x-Axis Data** section. From the **Unit** list, choose **h**.

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

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

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

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

Point Graph 1

In the **Model Builder** window, right-click **Point Graph 1** and choose **Duplicate**.

Point Graph 2

1 In the **Model Builder** window, click **Point Graph 2**.

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

3 From the **Dataset** list, choose **Study: Test Material [Strain Hardening Creep]/ Solution 1a (10) (solidtm2sol1)**.

4 Locate the **Selection** section. Click to select the **Activate Selection** toggle button.

5 Select Point 8 only.

6 Click **Replace Expression** in the upper-right corner of the **y-Axis Data** section. From the menu, choose **Component: Test Material [Strain Hardening Creep] (solidtm2comp)> Solid Mechanics>Strain>Creep strain tensor, local coordinate system>solid2.eclGp11 - Creep strain tensor, local coordinate system, 11-component**.

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)`		
`strain_hard_biaxial(t)`	1	

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