Model created in COMSOL Multiphysics 6.4

Combining Elastoplastic and Creep Material Models

Introduction

This example shows how to combine different types of material nonlinearity, such as creep and elastoplasticity. In this specific example, you perform a stress and nonlinear strain analysis on a thick cylinder under a nonproportional loading: an initial temperature increase followed by a fluctuating pressure applied to the internal surface of the cylinder.

This load case involves two different nonlinear material behaviors: elastoplasticity and creep. The plastic behavior is introduced during the temperature increase and the rapid change of the sign of the pressure. The creep behavior develops under the constant pressure load applied to the pipe for a sufficiently long period of time.

The original model is a NAFEMS benchmark model described in *Selected Benchmarks for Material Non-Linearity* (Ref. 1). The COMSOL Multiphysics solutions are compared with the reference data.

Model Definition

The physical geometry of the problem consists of a hollow cylinder with a 0.16 m inner radius and a 0.25 m outer radius. For simplicity, the study is performed on a 2D plane using an axial symmetry assumption, reducing the model geometry to a rectangle.

The benchmark setup takes the cylinder to be infinitely long so that it can be modeled using a plane strain assumption in the pipe axis plane. This assumption can easily be implemented by constraining the axial displacement in the whole geometry.

MATERIAL PROPERTIES

- Isotropic material with $E = 2.2 \cdot 10^7$ Pa, $\nu = 0.3$, $\alpha = 1.85 \cdot 10^{-5}$ K^{-1}
- Elastoplastic material with an initial yield stress, $\sigma_{y0} = 9900$ Pa,
- Nonlinear isotropic hardening with a stress–strain curve according to the table below:

Plastic strain	Stress (Pa)
3.9e-4	12500
9.5e-4	15200
2.95e-3	17500
6.15e-3	20000

  This nonlinear hardening function $\sigma_h(\varepsilon_{pe})$ is defined in the Materials node by an interpolation function.

- Creep constitutive law:

$$\frac{\partial \varepsilon_c}{\partial t} = 1e^{-26}\sigma^{5.25}t^{-0.1} \tag{1}$$

where ε_c is the equivalent creep strain, σ is the equivalent stress, and t is the time. In Equation 1, the stress is given in Pa and the time in seconds. To follow the benchmark example, the creep behavior starts after 2 s.

LOAD HISTORY

- The internal pressure is ramped up from 0 to 3600 Pa during 1 s at $t = 1$ s and change sign within 1 s at $t = 999$ s as illustrated in Figure 1.

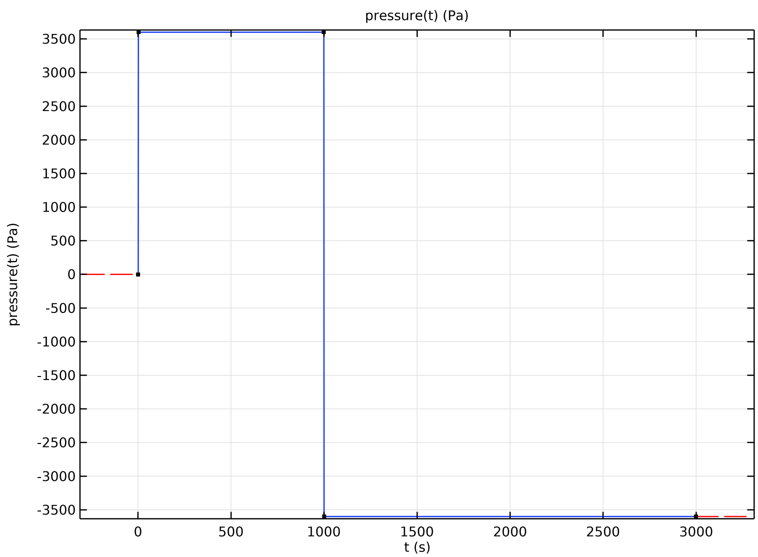

Figure 1: Internal pressure versus time. The red line is the default extrapolation of the pressure outside the defined domain.

- The temperature transient is defined by:

$$
T = \begin{cases} 0 & t = 0 \\ 500t^{\left(\frac{r-0.07}{0.09}\right)} & 0 \le t \le 0.1 \\ 50t^{\left(\frac{r-0.16}{0.09}\right)} & 0.1 \le t \le 1 \\ 50 & t > 1 \end{cases}
$$

where both time and radius, r, are given in default units.

Results and Discussion

In Figure 2 you can see the increase of the equivalent plastic strain for $t < 2$ s.

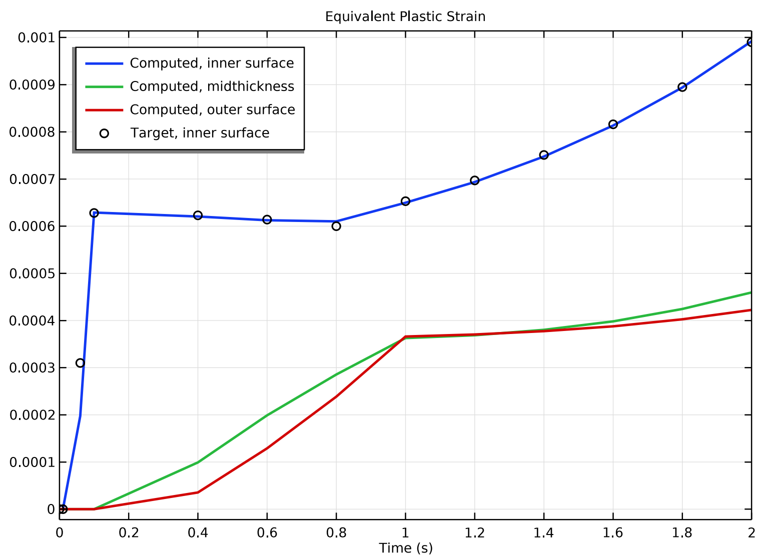

Figure 2: History of the equivalent plastic strain when the cylinder is affected by the temperature gradient.

The dotted curve corresponds to the target data from the benchmark. You can see that the computed solution is in fairly good agreement with the targeted solution.

The initial temperature gradient creates thermal stresses in the cylinder. The stress varies through the cross section and is high enough to develop plastic strains at the inner surface. In the considered time interval, stress is significantly higher at the inner surface than in the center, and at the outer radius. After 0.1 s, the thermal load at the inner surface is removed, but it is still present in the rest of the cylinder, where the material starts to deform plastically. After 1 s, the temperature is constant in the material and, in combination with the mechanical load, the plasticity develops, but at a lower rate. After 2 s, the material relaxes and creep takes over, see Figure 3.

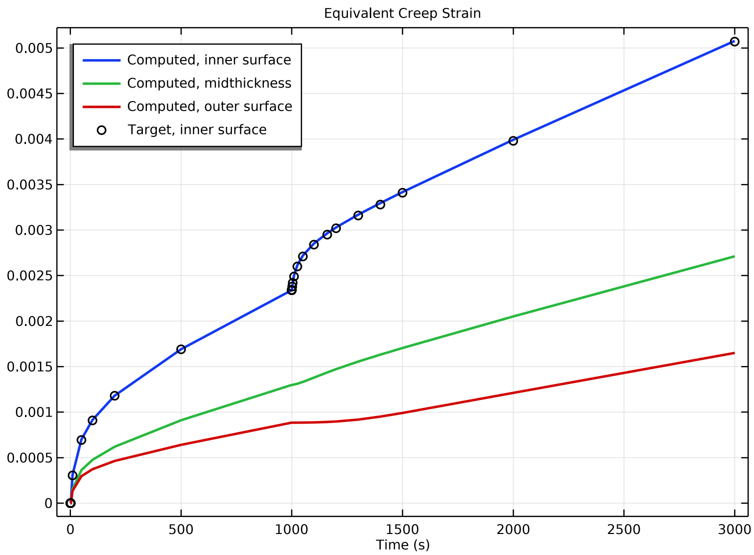

Figure 3: History of the equivalent creep strain.

The change in pressure at $t = 999$ s has the most significant effect on the creep strain at the inner wall.

Figure 4, Figure 5, and Figure 6 show, respectively, the axial, hoop, and von Mises stress at the inner wall. The axial and hoop stresses both change direction when the inner pressure is reversed. The asymptotic behavior is caused by the creep of the material. The equivalent stress, on the other hand, cannot be negative. When the pressure is changed within 1 s, the von Mises stress first decreases, pressure decreases to zero, and then increases, the pressure decreases further to a nonzero negative pressure. The peak is caused by the creep strains, which needs instantaneous reverse due to the pressure change.

Figure 7 shows variation the total strain energy in the model, and the energy that is dissipated due to creep and plastic deformations.

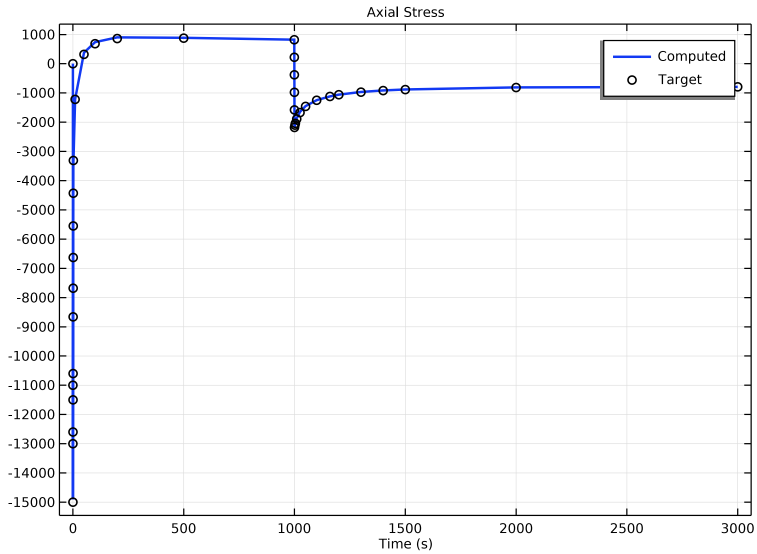

Figure 4: History of axial stress at the inner surface.

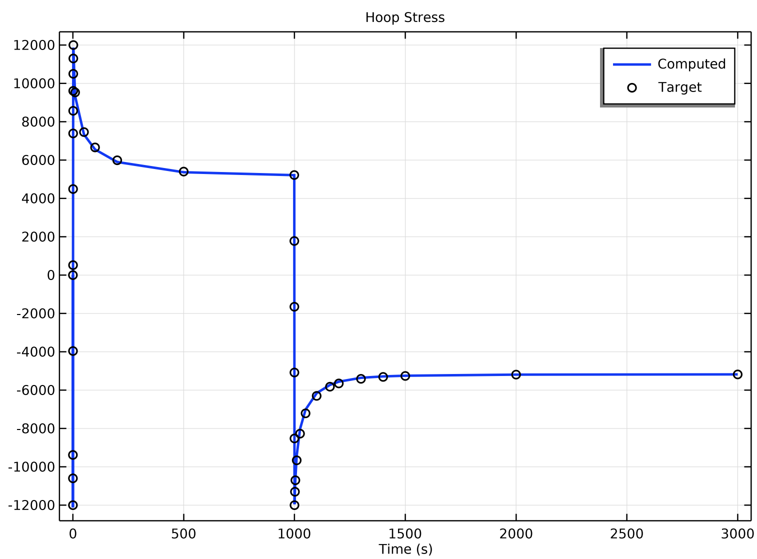

Figure 5: History of hoop stress at the inner surface.

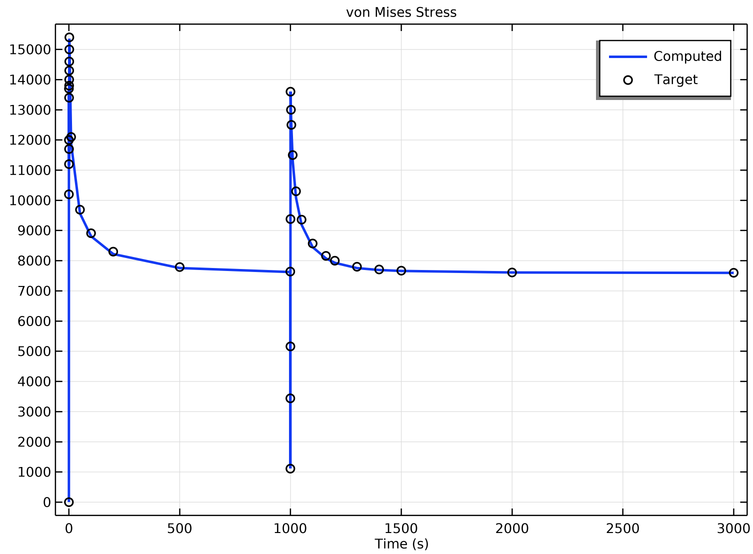

Figure 6: History of von Mises stress at the inner surface.

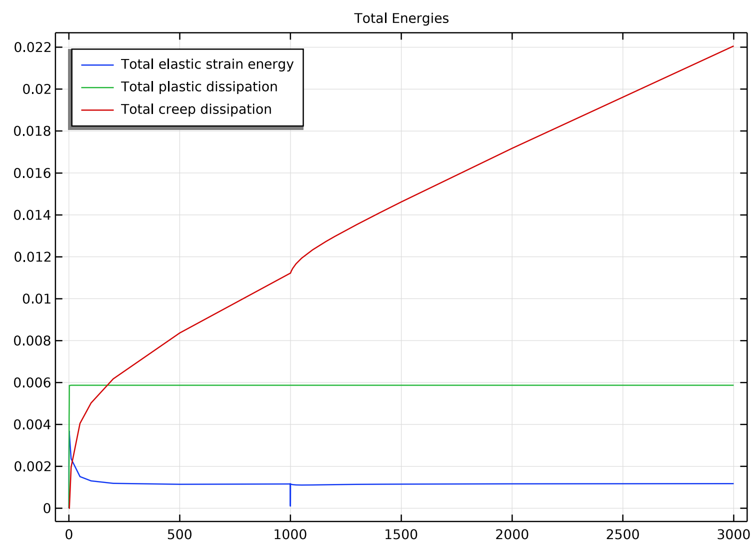

Figure 7: Elastic strain energy, plastic dissipation and creep dissipation.

Notes About the COMSOL Implementation

In COMSOL Multiphysics you can combine several nonlinear models for the same material. Different material models are available as subnodes to the **Linear Elastic Material** node. COMSOL Multiphysics adds the strain contribution of each node to the total strain.

The examined creep behavior can be modeled with the built-in Norton model combined with a time hardening model; this corresponds to a Norton–Bailey model. To normalize the equivalent stress in Pa and the time in hours, set the reference stress $\sigma_r = 1$ Pa and the reference time $t_r = 1$ s. Time offset, t_0, is here set to 1 ms to avoid the singularity that occurs at $t = 0$. To set the COMSOL model identical to the benchmark model, the creep strain is added only after $t= 2$ s. To do this multiply the creep rate coefficient, A, with the logical expression, $(t > 2)$.

The creep equations are here integrated using the explicit Forward Euler method. This is sometimes preferable for elastoplastic creep problems due to efficiency of the internal algorithms. The time duration of the benchmark problem is also so that the time step is below the stability limit of the Forward Euler method. Note that COMSOL Multiphysics automatically limits the time step so be below this stability limit when the default solver is generated.

When computing the creep material model, the time steps taken by the solver must be small when the creep rate is high and large when the creep rate is low. Because the time step can be difficult to predict, you can let the solver control what time steps to store.

Note that the benchmark (Ref. 1) utilizes a very coarse mesh. Due to the interpolation from Gauss points, this coarse mesh results in decreasing equivalent plastic strains, as shown around $0.1 < t < 1$ s in Figure 2.

Reference

1. D. Linkens, *Selected Benchmarks For Material Non-Linearity, volume 2,* NAFEMS, 1993.

Application Library path: `Nonlinear_Structural_Materials_Module/Creep/elastoplastic_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 Axisymmetric**.

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**.

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
m	0.9	0.9	Power law exponent

DEFINITIONS

Interpolation 1 (int1)

1 In the **Definitions** toolbar, click **Interpolation**.

2 In the **Settings** window for **Interpolation**, locate the **Definition** section.

3 In the **Function name** text field, type `pressure`.

4 In the table, enter the following settings:

t	f(t)
0	0
1	0
2	3600
999	3600

t	f(t)
1000	-3600
3000	-3600

5 Locate the **Units** section. In the **Argument** table, enter the following settings:

Argument	Unit
t	s

6 In the **Function** table, enter the following settings:

Function	Unit
pressure	Pa

7 Click **Plot**.

Analytic 1 (an1)

1 In the **Definitions** toolbar, click **Analytic**.

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

3 Locate the **Definition** section. In the **Expression** text field, type `500*t^((r-7e-2)/9e-2)*(t<=0.1)+50*t^((r-0.16)/9e-2)*(t>0.1)*(t<=1)+50*(t>1)`.

4 In the **Arguments** text field, type `t, r`.

5 Locate the **Units** section. In the table, enter the following settings:

Argument	Unit
t	s
r	m

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

Interpolation 2 (int2)

1 In the **Definitions** toolbar, click **Interpolation**.

2 In the **Settings** window for **Interpolation**, locate the **Definition** section.

3 From the **Data source** list, choose **File**.

4 Click **Browse**.

5 Browse to the model's Application Libraries folder and double-click the file `elastoplastic_creep_target.txt`.

6 Click **Import**.

7 Locate the **Data Column Settings** section. In the table, enter the following settings:

Columns	Type	Settings
sz	Function values	Function name=int2a
sphi	Function values	Function name=int2b
epe	Function values	Function name=int2c
ece	Function values	Function name=int2d

8 In the table, click to select the cell at row number 2 and column number 2.

9 In the **Name** text field, type `sz_target`.

10 In the table, click to select the cell at row number 3 and column number 2.

11 In the **Name** text field, type `sphi_target`.

12 In the table, click to select the cell at row number 4 and column number 2.

13 In the **Name** text field, type `mises_target`.

14 In the table, click to select the cell at row number 5 and column number 2.

15 In the **Name** text field, type `epe_target`.

16 In the table, click to select the cell at row number 6 and column number 2.

17 In the **Name** text field, type `ece_target`.

GEOMETRY 1

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 `9e-2`.

4 In the **Height** text field, type `9e-3`.

5 Locate the **Position** section. In the **r** text field, type `0.16`.

6 Click **Build All Objects**.

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**.

Linear Elastic Material 1

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

Thermal Expansion 1

1 In the **Physics** toolbar, click **Attributes** and choose **Thermal Expansion**.
2 In the **Settings** window for **Thermal Expansion**, locate the **Model Input** section.
3 From the T list, choose **User defined**. In the associated text field, type `T(t,R)`.
4 Click **Go to Source** for **Volume reference temperature**.

GLOBAL DEFINITIONS

Default Model Inputs

1 In the **Model Builder** window, under **Global Definitions** click **Default Model Inputs**.
2 In the **Settings** window for **Default Model Inputs**, locate the **Browse Model Inputs** section.
3 Find the **Expression for remaining selection** subsection. In the **Volume reference temperature** text field, type `0`.

SOLID MECHANICS (SOLID)

Linear Elastic Material 1

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

Plasticity 1

1 In the **Physics** toolbar, click **Attributes** and choose **Plasticity**.
2 In the **Settings** window for **Plasticity**, locate the **Plasticity Model** section.
3 Find the **Isotropic hardening model** subsection. From the list, choose **Hardening function**.

Linear Elastic Material 1

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

Creep 1

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

In order to fulfill the benchmark requirements, the creep material model is activated only after t = 2 s.

1 In the **Settings** window for **Creep**, locate the **Creep Model** section.
2 From the A list, choose **User defined**. In the associated text field, type `1e-26/m*(t>2)`.

3 From the σ_{ref} list, choose **User defined**. In the associated text field, type `1`.

4 From the n list, choose **User defined**. In the associated text field, type `5.25`.

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

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

7 In the t_{shift} text field, type `1e-3`.

8 In the t_{ref} text field, type `1`.

Use the forward Euler method when combining creep and plasticity.

9 Locate the **Time Stepping** section. From the **Method** list, choose **Forward Euler**.

Linear Elastic Material 1

1 Click the **Show More Options** button in the **Model Builder** toolbar.

2 In the **Show More Options** dialog, in the tree, select the checkbox for the node **Physics** > **Advanced Physics Options**.

3 Click **OK**.

Enable the computation of the dissipated energy.

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

5 In the **Settings** window for **Linear Elastic Material**, click to expand the **Energy Dissipation** section.

6 From the **Store dissipation** list, choose **Individual contributions**.

Boundary Load 1

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

2 Select Boundary 1 only.

3 In the **Settings** window for **Boundary Load**, locate the **Force** section.

4 Specify the $\mathbf{f}_A$ vector as

pressure(t)	r
0	z

Prescribed Displacement 1

1 In the **Physics** toolbar, click **Domains** and choose **Prescribed Displacement**.

2 Select Domain 1 only.

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

4 From the **Displacement in z direction** list, choose **Prescribed**.

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	`22[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	kg/m³	Basic
Coefficient of thermal expansion	alpha_iso ; alphaii = alpha_iso, alphaij = 0	`18.5e-6`	1/K	Basic
Initial yield stress	sigmags	`sigma_yield(0)`	Pa	Elastoplastic material model
Hardening function	sigmagh	`sigma_yield(epe)-sigma_yield(0)`	Pa	Elastoplastic material model

4 In the **Model Builder** window, expand the **Material 1 (mat1)** node, then click **Elastoplastic material model (ElastoplasticModel)**.

5 In the **Settings** window for **Elastoplastic Material Model**, locate the **Model Inputs** section.

6 Click **Select Quantity**.

7 In the **Physical Quantity** dialog, type `plastic strain` in the text field.

8 In the tree, select **Solid Mechanics** > **Equivalent plastic strain (1)**.

9 Click **OK**.

Interpolation 1 (int1)

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

2 In the **Settings** window for **Interpolation**, locate the **Definition** section.

3 In the **Function name** text field, type `sigma_yield`.

4 In the table, enter the following settings:

t	f(t)
0	9900
3.9e-4	12500
9.5e-4	15200
2.95e-3	17500
6.15e-3	20000

5 Locate the **Units** section. In the **Argument** table, enter the following settings:

Argument	Unit
t	1

6 In the **Function** table, enter the following settings:

Function	Unit
sigma_yield	Pa

MESH 1

Mapped 1

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

Distribution 1

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

2 Select Boundary 1 only.

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

4 In the **Number of elements** text field, type 1.

Distribution 2

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

2 Select Boundary 2 only.

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

4 In the **Number of elements** text field, type 10.

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 0.01 0.06 0.1 range(0.4, 0.2, 2) 10 50 100 200 500 range(999, 0.2, 1000) 1002 1004 1010 1025 1050 1100 1160 range(1200, 100, 1500) 2000 3000`.

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.

Ensure that results are given at exactly the same times as in the benchmark reference.

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

2 Select the **Initial step** checkbox. In the associated text field, type `0.01`.

Setting the initial step ensures a good calculation of the creep strain at $t = 0$.

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

RESULTS

Stress (solid)

Follow the steps below to obtain Figure 2.

Cut Point 2D 1

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

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

3 In the **R** text field, type `0.16 0.205 0.25`.

4 In the **Z** text field, type `5e-3`.

Equivalent Plastic Strain

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

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

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

4 Locate the **Data** section. From the **Dataset** list, choose **Cut Point 2D 1**.

5 Click to expand the **Title** section. From the **Title type** list, choose **Label**.

Point Graph 1

1 Right-click **Equivalent Plastic Strain** and choose **Point Graph**.

2 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** > **Strain** > **solid.epeGp - Equivalent plastic strain - 1**.

3 Click to expand the **Coloring and Style** section. From the **Width** list, choose **2**.

4 Click to expand the **Legends** section. Select the **Show legends** checkbox.

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

6 In the table, enter the following settings:

Legends
`Computed, inner surface`
`Computed, midthickness`
`Computed, outer surface`

Global 1

1 In the **Model Builder** window, right-click **Equivalent Plastic Strain** 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
`epe_target(t)`		

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

5 Find the **Line markers** subsection. From the **Marker** list, choose **Circle**.

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

7 Click to expand the **Legends** section. From the **Legends** list, choose **Manual**.

8 In the table, enter the following settings:

Legends
`Target, inner surface`

9 In the **Equivalent Plastic Strain** toolbar, click **Plot**.

Display development of plastic strains in the first two seconds, as in the benchmark example.

Equivalent Plastic Strain

1 In the **Model Builder** window, click **Equivalent Plastic Strain**.
2 In the **Settings** window for **1D Plot Group**, locate the **Axis** section.
3 Select the **Manual axis limits** checkbox.
4 In the **x minimum** text field, type `0`.
5 In the **x maximum** text field, type `2`.
6 In the **Equivalent Plastic Strain** toolbar, click **Plot**.

You can obtain Figure 3 by running the following instructions.

Equivalent Creep Strain

1 Right-click **Equivalent Plastic Strain** and choose **Duplicate**.
2 In the **Settings** window for **1D Plot Group**, type `Equivalent Creep Strain` in the **Label** text field.
3 Locate the **Axis** section. Clear the **Manual axis limits** checkbox.

Point Graph 1

1 In the **Model Builder** window, expand the **Equivalent Creep Strain** node, then click **Point Graph 1**.
2 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** > **Strain** > **solid.eceGp - Equivalent creep strain - 1**.

Global 1

1 In the **Model Builder** window, click **Global 1**.
2 In the **Settings** window for **Global**, locate the **y-Axis Data** section.
3 In the table, enter the following settings:

Expression	Unit	Description
`ece_target(t)`		

4 In the **Equivalent Creep Strain** toolbar, click **Plot**.

To get Figure 4, follow the instruction below.

Cut Point 2D 2

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

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

3 In the **R** text field, type `0.16`.

4 In the **Z** text field, type `5e-3`.

Axial Stress

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

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

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

4 Locate the **Legend** section. From the **Position** list, choose **Upper right**.

Point Graph 1

1 In the **Model Builder** window, expand the **Axial Stress** node, then click **Point Graph 1**.

2 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 (spatial frame) - N/m²** > **solid.sGpzz - Stress tensor, zz-component**.

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

Legends
Computed

Global 1

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

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

3 In the table, enter the following settings:

Expression	Unit	Description
sz_target(t)		

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

Legends
Target

5 In the **Axial Stress** toolbar, click **Plot**.

Follow the instructions below to get Figure 5.

Hoop Stress

1 In the **Model Builder** window, right-click **Axial Stress** and choose **Duplicate**.

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

Point Graph 1

1 In the **Model Builder** window, expand the **Hoop Stress** node, then click **Point Graph 1**.

2 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 (spatial frame) - N/m²** > **solid.sGpphiphi - Stress tensor, phiphi-component**.

Global 1

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

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

3 In the table, enter the following settings:

Expression	Unit	Description
`sphi_target(t)`		

4 In the **Hoop Stress** toolbar, click **Plot**.

Follow the instructions below to get Figure 6.

von Mises Stress

1 In the **Model Builder** window, right-click **Hoop Stress** and choose **Duplicate**.

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

Point Graph 1

1 In the **Model Builder** window, expand the **von Mises Stress** node, then click **Point Graph 1**.

2 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** > **solid.misesGp - von Mises stress - N/m²**.

Global 1

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

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

3 In the table, enter the following settings:

Expression	Unit	Description
`mises_target(t)`		

4 In the **von Mises Stress** toolbar, click **Plot**.

Total Energies

1. In the **Results** toolbar, click **1D Plot Group**.
2. In the **Settings** window for **1D Plot Group**, type `Total Energies` in the **Label** text field.
3. Locate the **Title** section. From the **Title type** list, choose **Label**.

Global 1

1. Right-click **Total Energies** and choose **Global**.
2. In the **Settings** window for **Global**, click **Add Expression** in the upper-right corner of the **y-Axis Data** section. From the menu, choose **Component 1 (comp1)** > **Solid Mechanics** > **Global** > **solid.Ws_tot - Total elastic strain energy - J**.
3. Click **Add Expression** in the upper-right corner of the **y-Axis Data** section. From the menu, choose **Component 1 (comp1)** > **Solid Mechanics** > **Global** > **solid.Wp_tot - Total plastic dissipation - J**.
4. Click **Add Expression** in the upper-right corner of the **y-Axis Data** section. From the menu, choose **Component 1 (comp1)** > **Solid Mechanics** > **Global** > **solid.Wc_tot - Total creep dissipation - J**.

Total Energies

1. In the **Model Builder** window, click **Total Energies**.
2. In the **Settings** window for **1D Plot Group**, locate the **Legend** section.
3. From the **Position** list, choose **Upper left**.
4. In the **Total Energies** toolbar, click **Plot**.