



Model created in COMSOL Multiphysics 6.4

# Combining Creep Material Models

## Introduction

---

This model illustrates how to combine different creep models to accurately represent the material behavior. An example for such a material model combination is to add a Norton-Bailey's law with a Norton's law as described in the equation

$$\frac{\partial \epsilon_c}{\partial t} = (A_1 \sigma_e^{n_1} + A_2 \sigma_e^{n_2} m t^{m-1}) g(T)$$

## Model Definition

---

This model is a modification of the application library model [Thermally Induced Creep](#). The same geometry, hollow sphere, load, inner pressure and thermal gradient, are used here. The parameters of Norton's law are also reused and govern secondary creep. The primary creep data, which differentiates the two models, is defined as

$$A_2 \sigma_e^{n_2} m t^{m-1} g(T)$$

Parameter  $A_2 = 10 \text{ h}^{-1}$  accounts for the stress normalization of the equivalent stress,  $\sigma_e$ , in MPa and time,  $t$ , in hours. The other parameters are  $n_2 = 3.5$ ,  $m = 0.5$ , and  $g(T) = e^{-12500/T}$ , where  $T$  defines the temperature in K.

## Results and Discussion

---

[Figure 1](#) shows the von Mises stress at  $10^8 \text{ h}$  when both primary creep and secondary creep are active. The largest equivalent stress, found in the center of the sphere, demonstrates that the relaxation take place starting from the inner radius and continuing toward the outer boundary. Without creep due to the internal pressure, the largest stresses are found on the inner radius and decreases toward the outer radius instead.

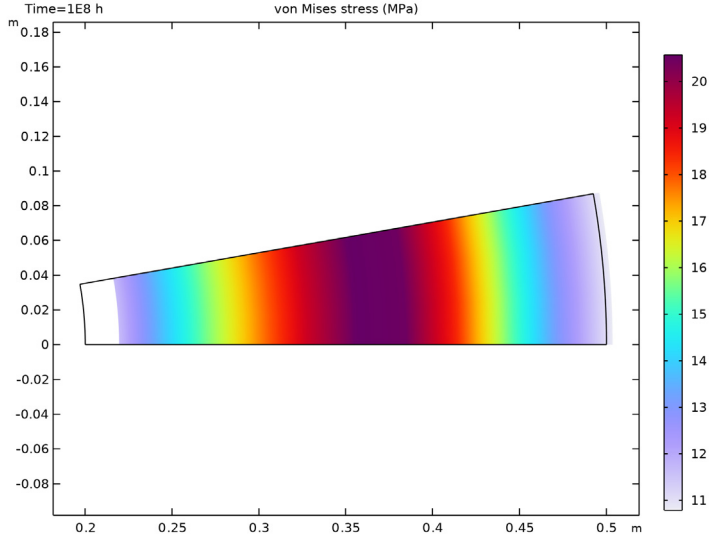


Figure 1: Distribution of von Mises stress at  $t = 10^8$  h.

To compare both creep models, secondary creep against combined creep, see [Figure 2](#) where the time history of the equivalent stress is shown at three different locations. It is clear that the primary creep relaxes the inner boundary earlier than the secondary creep. Over time, both creep models converge to the same result. This is expected because the influence of the primary creep fades away with time.

[Figure 3](#) shows the evolution of the creep strains using the two models at the same points as used in [Figure 2](#). The effect of including primary creep is clearly visible, and also how it fades away with time. Lastly, the history of the stress profile is shown in [Figure 4](#). Relaxation causes the location of peak stress to propagate from the inner to the outer surface.

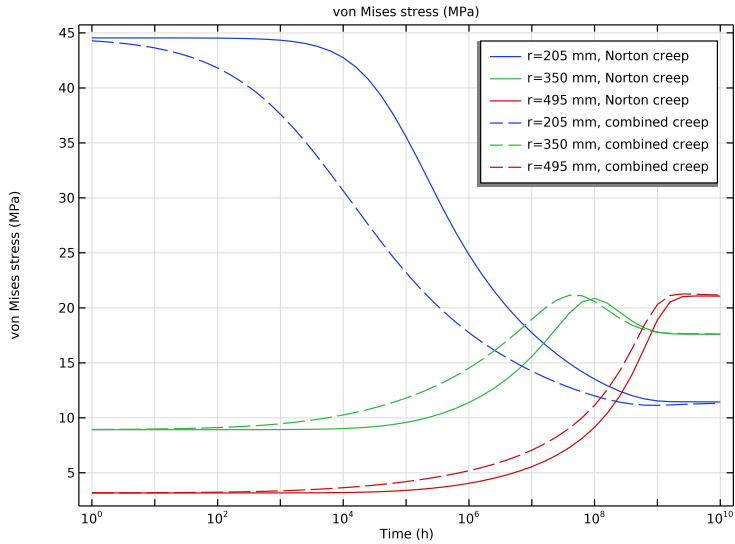


Figure 2: History of von Mises stress at radius 205 mm, 350 mm, and 495 mm, of the secondary creep only (solid lines), and with primary creep included (dashed lines).

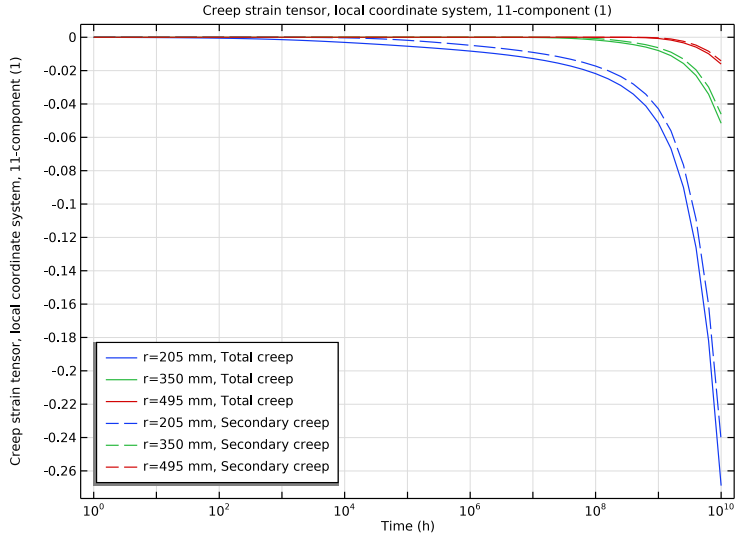


Figure 3: History of combined and secondary creep strain at radius 205 mm, 350 mm, and 495 mm.

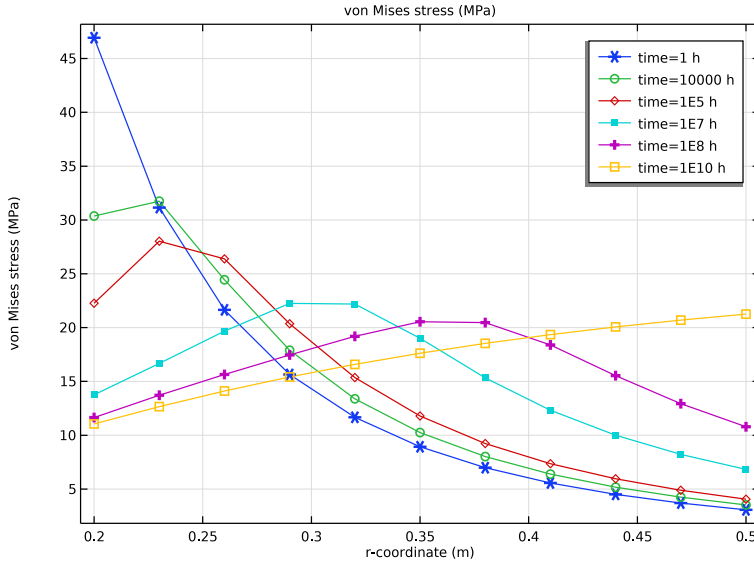


Figure 4: von Mises stress versus radius at  $t = 1, 10^4, 10^5, 10^7, 10^8,$  and  $10^{10}$  hours with primary creep included.

### Modeling in COMSOL Multiphysics

In COMSOL Multiphysics you can combine several creep models by adding **Additional Creep** subnodes to a **Creep** node.

Norton–Bailey’s creep law for primary creep can in COMSOL Multiphysics be defined by adding a hardening model to a Norton creep law. Using a time hardening model, gives

$$\frac{\partial \epsilon_c}{\partial t} = A \left( \frac{\sigma_{\text{eff}}}{\sigma_{\text{ref}}} \right)^n m \left( \frac{t + t_{\text{shift}}}{t_{\text{ref}}} \right)^{m-1} e^{-\frac{Q}{RT}}$$

In order to normalize the equivalent stress in MPa and the time in  $h$ , set the reference stress  $\sigma_{\text{ref}} = 1$  MPa and the reference time  $t_{\text{ref}} = 1$  h. Set the time shift  $t_{\text{shift}}$  to 1 min in order to avoid the singularity that occurs at  $t = 0$ . Considering the long time scale of the analysis this has a negligible effect on the results.

---


**Application Library path:** Nonlinear\_Structural\_Materials\_Module/Creep/  
combined\_creep

---

### *Modeling Instructions*

---

#### **APPLICATION LIBRARIES**

- 1 From the **File** menu, choose **Application Libraries**.
- 2 In the **Application Libraries** window, select **Nonlinear Structural Materials Module > Creep > thermally\_induced\_creep** in the tree.
- 3 Click  **Open**.

#### **COMPONENT 1 (COMP1)**

In the **Model Builder** window, expand the **Component 1 (comp1)** node.


#### **SOLID MECHANICS (SOLID)**

In the **Model Builder** window, expand the **Component 1 (comp1) > Solid Mechanics (solid)** node.

##### *Creep (Secondary)*

- 1 In the **Model Builder** window, expand the **Component 1 (comp1) > Solid Mechanics (solid) > Linear Elastic Material 1** node, then click **Creep 1**.
- 2 In the **Settings** window for **Creep**, type **Creep (Secondary)** in the **Label** text field.

##### *Additional Creep (Primary)*

- 1 In the **Physics** toolbar, click  **Attributes** and choose **Additional Creep**.
- 2 In the **Settings** window for **Additional Creep**, type **Additional Creep (Primary)** in the **Label** text field.
- 3 Locate the **Creep Model** section. From the *A* list, choose **User defined**. In the associated text field, type  $10[1/h]$ .
- 4 From the  $\sigma_{ref}$  list, choose **User defined**. From the *n* list, choose **User defined**. In the associated text field, type **3.5**.
- 5 Find the **Isotropic hardening model** subsection. From the  $h(\epsilon_{ce}, t)$  list, choose **Time hardening**.
- 6 In the *m* text field, type **0.5**.

7 In the  $t_{\text{shift}}$  text field, type 1 [min].

8 Find the **Thermal effects** subsection. From the  $g(T)$  list, choose **Arrhenius**.

9 In the  $Q$  text field, type  $1.0393e5$  [J/mol].

Study 1 solves the model with only **Creep I** enabled. Disable **Additional Creep I** for this study.

## STUDY 1

*Step 1: Time Dependent*

1 In the **Model Builder** window, expand the **Study 1** node, then click

**Step 1: Time Dependent**.

2 In the **Settings** window for **Time Dependent**, locate the **Physics and Variables Selection** section.

3 Select the **Modify model configuration for study step** checkbox.

4 In the tree, select **Component 1 (comp1) > Solid Mechanics (solid) > Linear Elastic Material 1 > Creep (Secondary) > Additional Creep (Primary)**.

5 Click  **Disable**.

Add a new study to solve for the combined creep.

## 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 the **Add Study** button in the window toolbar.

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

## STUDY 2

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



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

3 In the **Output times** text field, type  $0 \cdot 10^{\{\text{range}(0, 0.2, 10)\}}$ .

4 From the **Tolerance** list, choose **User controlled**.

5 In the **Relative tolerance** text field, type  $1e-4$ .

### *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**, click to expand the **Time Stepping** section.
- 4 Select the **Initial step** checkbox. In the associated text field, type 1 [min].
- 5 From the **Steps taken by solver** list, choose **Strict**.
- 6 In the **Study** toolbar, click  **Compute**.



## **RESULTS**

### *Stress with Combined Creep*

Select the solution at  $10^8$  hours to reproduce [Figure 1](#).

- 1 In the **Settings** window for **2D Plot Group**, type Stress with Combined Creep in the **Label** text field.
- 2 Locate the **Data** section. From the **Time (h)** list, choose **IE8**.

### *Surface 1*


- 1 In the **Model Builder** window, expand the **Stress with Combined Creep** node, then click **Surface 1**.
- 2 In the **Stress with Combined Creep** toolbar, click  **Plot**.
- 3 Click the  **Zoom Extents** button in the **Graphics** toolbar.

The commands below illustrate how to generate [Figure 2](#).

### *Stress with Combined Creep, 3D*

- 1 In the **Model Builder** window, under **Results** click **Stress, 3D (solid) 1**.
- 2 In the **Settings** window for **3D Plot Group**, type Stress with Combined Creep, 3D in the **Label** text field.
- 3 Locate the **Data** section. From the **Time (h)** list, choose **IE8**.

### *Surface 1*

- 1 In the **Model Builder** window, expand the **Stress with Combined Creep, 3D** node, then click **Surface 1**.
- 2 In the **Stress with Combined Creep, 3D** toolbar, click  **Plot**.


### *Cut Point 2D 2*

- 1 In the **Model Builder** window, expand the **Results > Datasets** node.
- 2 Right-click **Results > Datasets > Cut Point 2D 1** and choose **Duplicate**.
- 3 In the **Settings** window for **Cut Point 2D**, locate the **Data** section.
- 4 From the **Dataset** list, choose **Study 2/Solution 2 (sol2)**.

### *Point Graph 1*


- 1 In the **Model Builder** window, expand the **Results > von Mises Stress** node, then click **Point Graph 1**.
- 2 In the **Settings** window for **Point Graph**, click to expand the **Legends** section.
- 3 In the **Legend** text field, type  $r=\text{eval}(r,\text{mm})$  mm, Norton creep.

### *Point Graph 2*


- 1 Right-click **Results > von Mises Stress > 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**.
- 4 Click to expand the **Title** section. From the **Title type** list, choose **None**.
- 5 Click to expand the **Coloring and Style** section. Find the **Line style** subsection. From the **Line** list, choose **Dashed**.
- 6 From the **Color** list, choose **Cycle (reset)**.
- 7 Locate the **Legends** section. In the **Legend** text field, type  $r=\text{eval}(r,\text{mm})$  mm, combined creep.
- 8 In the **von Mises Stress** toolbar, click  **Plot**.

To get [Figure 4](#), continue with the steps below:

### *Cut Line 2D 1*


- 1 In the **Results** toolbar, click  **Cut Line 2D**.
- 2 In the **Settings** window for **Cut Line 2D**, locate the **Data** section.
- 3 From the **Dataset** list, choose **Study 2/Solution 2 (sol2)**.
- 4 Locate the **Line Data** section. In row **Point 1**, set **R** to 0.2.
- 5 In row **Point 2**, set **R** to 0.5.

### *von Mises Stress, Profile*

- 1 In the **Results** toolbar, click  **ID Plot Group**.
- 2 In the **Settings** window for **ID Plot Group**, type von Mises Stress, Profile in the **Label** text field.


- 3 Locate the **Data** section. From the **Dataset** list, choose **Cut Line 2D 1**.
- 4 From the **Time selection** list, choose **From list**.
- 5 In the **Times (h)** list, choose **1**, **10000**, **1E5**, **1E7**, **1E8**, and **1E10**.

#### *Line Graph 1*


- 1 Right-click **von Mises Stress, Profile** and choose **Line Graph**.
- 2 In the **Settings** window for **Line 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/m<sup>2</sup>**.
- 3 Locate the **x-Axis Data** section. From the **Parameter** list, choose **Expression**.
- 4 In the **Expression** text field, type **r**.
- 5 Click to expand the **Coloring and Style** section. Find the **Line markers** subsection. From the **Marker** list, choose **Cycle**.
- 6 Click to expand the **Quality** section. From the **Evaluation settings** list, choose **Manual**.
- 7 From the **Resolution** list, choose **No refinement**.
- 8 Click to expand the **Legends** section. Select the **Show legends** checkbox.
- 9 Find the **Prefix and suffix** subsection. In the **Prefix** text field, type **time=**.
- 10 In the **von Mises Stress, Profile** toolbar, click  **Plot**.

Plot the creep strains with and without primary creep, see [Figure 3](#).


#### *Creep Strains*

- 1 In the **Results** toolbar, click  **ID Plot Group**.
- 2 In the **Settings** window for **ID Plot Group**, type **Creep Strains** in the **Label** text field.


#### *Point Graph 1*

- 1 In the **Creep Strains** toolbar, click  **Point Graph**.
- 2 In the **Settings** window for **Point Graph**, locate the **Data** section.
- 3 From the **Dataset** list, choose **Cut Point 2D 2**.
- 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 > Creep strain tensor, local coordinate system > solid.eclGp11 - Creep strain tensor, local coordinate system, 11-component**.
- 5 Locate the **Legends** section. Select the **Show legends** checkbox.
- 6 From the **Legends** list, choose **Evaluated**.
- 7 In the **Legend** text field, type **r=eval(r,mm) mm, Total creep**.

### *Point Graph 2*

- 1 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 1**.
- 4 Locate the **Coloring and Style** section. Find the **Line style** subsection. From the **Line** list, choose **Dashed**.
- 5 From the **Color** list, choose **Cycle (reset)**.
- 6 Locate the **Legends** section. In the **Legend** text field, type `r=eval(r,mm) mm,`  
Secondary creep.
- 7 Locate the **Title** section. From the **Title type** list, choose **None**.
- 8 Click the  **x-Axis Log Scale** button in the **Graphics** toolbar.

### *Creep Strains*

- 1 In the **Model Builder** window, click **Creep Strains**.
- 2 In the **Settings** window for **ID Plot Group**, locate the **Legend** section.
- 3 From the **Position** list, choose **Lower left**.
- 4 In the **Creep Strains** toolbar, click  **Plot**.