Created in COMSOL Multiphysics 5.6

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

$$\frac{\partial \varepsilon_c}{\partial t} = (A_1 \sigma_e^{n_1} + A_2 \sigma_e^{n_2} m t^{m-1}) f_2(T)$$

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

$$A_2 \sigma_e^{n_2} m t^{m-1} f_2(T)$$

Parameter $A_2 = 10\ \mathrm{h}^{-1}$ accounts for the stress normalization of the equivalent stress, σ_e, in MPa and time, t, in hours. The other parameters are $n_2 = 3.5$, $m = 0.5$, and

$f_2(T) = e^{-12500/T}$, where T defines the temperature in K .

Results and Discussion

Figure 1 shows the von Mises stress at 10^8 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.

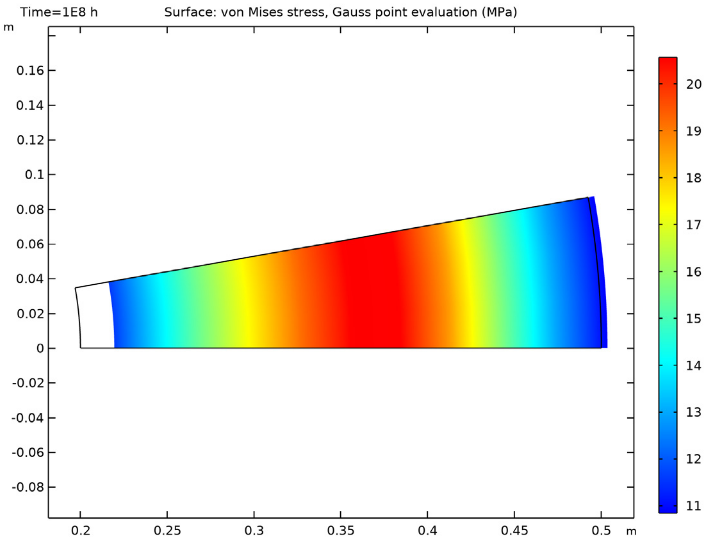

Figure 1: Distribution of von Mises stress at $t = 10^8$ h.

To compare the 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; compare the turquoise and blue lines. Over time, both creep models converge to the same result. This is expected because the influence of the primary creep fades away with time.

The creep strain can also be plotted for primary and secondary creep separately. The primary creep strain increases earlier than secondary creep, as shown in Figure 3. Lastly, the history of the stress profile is shown in Figure 4. Relaxation causes the propagation from the inner to the outer radius of the position of the peak stress.

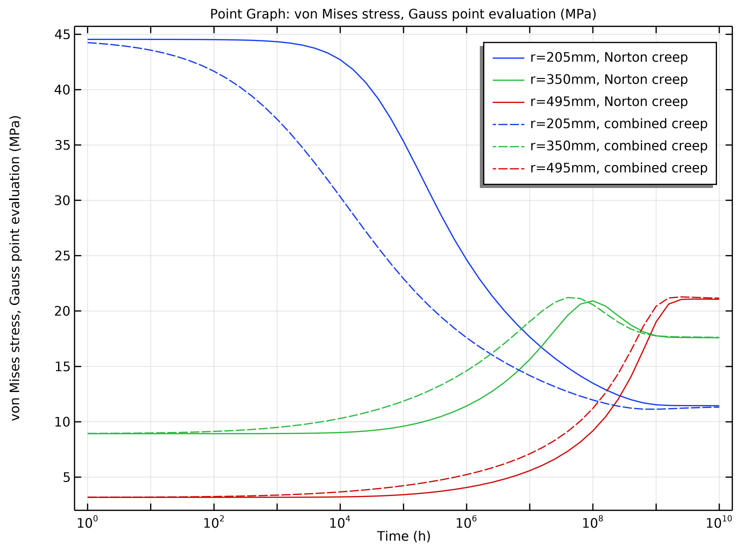

Figure 2: History of von Mises stress at radius 205 mm, 350 mm, and 495 mm, of the secondary creep only (solid lines), and with primary creep included (dashed lines).

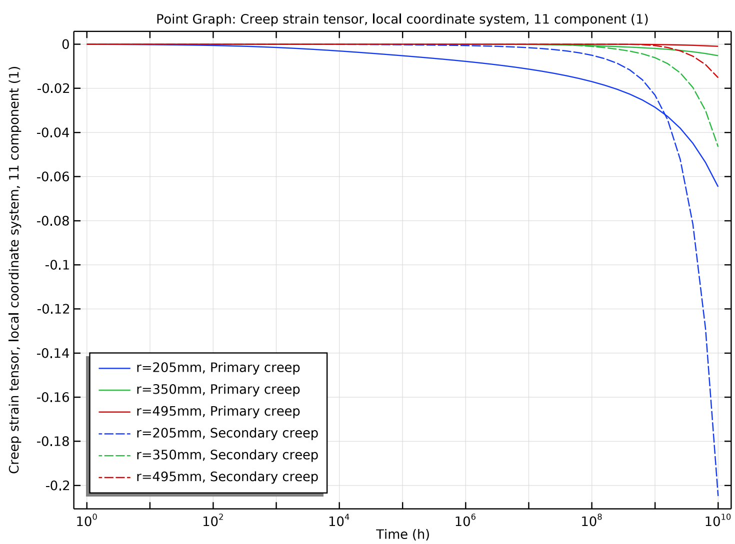

Figure 3: History of primary and secondary creep strain at radius 205 mm, 350 mm, and 495 mm.

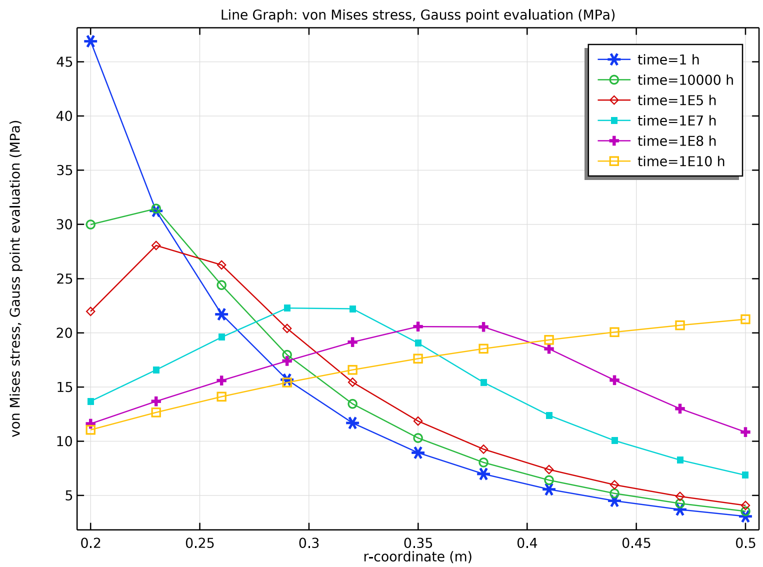

Figure 4: von Mises stress versus radius at t = 1, 10^4, 10^5, 10^7, 10^8, and 10^{10} hours with primary creep included.

Modeling in COMSOL Multiphysics

In COMSOL Multiphysics you can add several **Creep** nodes under the same Linear Elastic Material feature. Each creep node can define its unique creep model.

The Norton-Bailey's law is defied in Nonlinear Structural Materials Module with

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

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_{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

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

Browse to the model's Application Libraries folder and double-click the file `thermally_induced_creep.mph`.

COMPONENT 1 (COMP1)

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

SOLID MECHANICS (SOLID)

Linear Elastic Material 1

In the **Model Builder** window, expand the **Component 1 (comp1)>Solid Mechanics (solid)** node, then 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 Data** section.
3 From the **Material model** list, choose **Norton-Bailey**.
4 From the A list, choose **User defined**. In the associated text field, type `10[1/h]`.
5 From the σ_{ref} list, choose **User defined**. From the n list, choose **User defined**. In the associated text field, type `3.5`.
6 In the m text field, type `0.5`.
7 In the t_{shift} text field, type `1[min]`.
8 Select the **Include temperature dependency** check box.
9 In the Q text field, type `1.0393e5[J/mol]`.
10 Locate the **Model Input** section. From the T list, choose **User defined**. In the associated text field, type `T`.

Study 1 solves the model with only **Creep 1** enabled. Disable **Creep 2** 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** check box.

4 In the **Physics and variables selection** tree, select **Component 1 (comp1)> Solid Mechanics (solid)>Linear Elastic Material 1>Creep 2**.

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

3 In the **Output times** text field, type `0 10^{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** check box.

5 In the associated text field, type `1[min]`.

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

7 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 **1E8**.

Surface 1

1 In the **Model Builder** window, expand the **Stress with Combined Creep** node, then click **Surface 1**.

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

3 From the **Unit** list, choose **MPa**.

4 In the **Stress with Combined Creep** toolbar, click **Plot**.

5 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 **1E8**.

Surface 1

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

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

3 From the **Unit** list, choose **MPa**.

4 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 table, enter the following settings:

Legends
`r=205mm, Norton creep`
`r=350mm, Norton creep`
`r=495mm, 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 table, enter the following settings:

Legends
`r=205mm, combined creep`
`r=350mm, combined creep`
`r=495mm, 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 **1D Plot Group**.

2 In the **Settings** window for **1D 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 (Gauss points)>solid.misesGp - von Mises stress, Gauss point evaluation - N/m²**.

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

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

5 In the **Expression** text field, type `r`.

6 Click to expand the **Coloring and Style** section. 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 **Quality** section. From the **Resolution** list, choose **No refinement**.

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

10 Find the **Include** subsection. In the **Prefix** text field, type `time=`.

11 In the **von†Mises Stress, Profile** toolbar, click **Plot**.

Primary and secondary creep strain can be plotted separately, see Figure 3.

Creep Strains

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 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 (Gauss points)> Creep strain tensor, local coordinate system>solid.lemm1.cmm2.ecGp11 - Creep strain tensor, local coordinate system, 11 component**.

5 Locate the **Legends** section. Select the **Show legends** check box.

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

7 In the table, enter the following settings:

Legends
`r=205mm, Primary creep`
`r=350mm, Primary creep`
`r=495mm, Primary creep`

Point Graph 2

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

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 (Gauss points)>Creep strain tensor, local coordinate system> solid.lemm1.cmm1.ecGp11 - Creep strain tensor, local coordinate system, 11 component**.

3 Locate the **Coloring and Style** section. Find the **Line style** subsection. From the **Line** list, choose **Dashed**.

4 From the **Color** list, choose **Cycle (reset)**.

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

Legends
`r=205mm, Secondary creep`
`r=350mm, Secondary creep`
`r=495mm, Secondary creep`

6 Locate the **Title** section. From the **Title type** list, choose **None**.

7 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 **1D Plot Group**, locate the **Legend** section.

3 From the **Position** list, choose **Lower left**.

4 In the **Creep Strains** toolbar, click **Plot**.