Model created in COMSOL Multiphysics 6.4

Thermally Induced Creep

Introduction

This example computes the stress history over a very long time for a material that exhibits creep behavior. The model is taken from *NAFEMS Selected Benchmarks For Material Non-Linearity, Volume 2* (Ref. 1). The displacement and stress levels are compared with the values given in the reference.

Model Definition

The geometry is a hollow sphere with an inner radius of 200 mm and an outer radius of 500 mm. The problem has rotational symmetry where the solution depends only on the radial coordinate. You could therefore select any section having radial cuts as the computational domain. To follow the original example the sphere is modeled with a 2D axisymmetric 10° sector with symmetry constraint conditions applied on edges of the sector; see Figure 1.

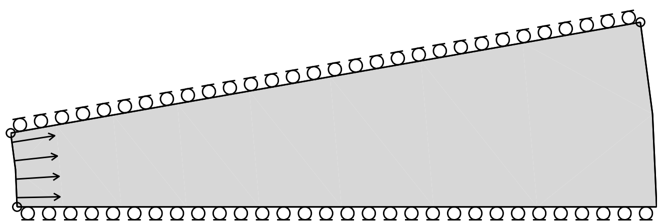

Figure 1: The model geometry, using a 10° sector of the original geometry.

MATERIAL PROPERTIES

- Isotropic with $E = 10$ GPa, $\nu = 0.25$.
- Creep data according to:

$$\frac{\partial \varepsilon_c}{\partial t} = A_1 \sigma_e^{n_1} g(T) \tag{1}$$

with $A_1 = 3.0 \cdot 10^{-6}\ \mathrm{h}^{-1}$ that accounts for the stress normalization of the equivalent stress, σ_e, in MPa, $n_1 = 5.5$, and $g(T) = e^{-12500/T}$, where T defines the temperature in K .

LOADS

- An internal pressure of 30 MPa.
- A temperature field with the distribution $T = 333(1 + 100/(\sqrt{R^2 + Z^2}))$ where R and Z are material coordinates in mm.

Results and Discussion

The evolution of the displacement with time is shown in Figure 2. The upper curve represents the inner radius, and the lower curve represents the outer radius.

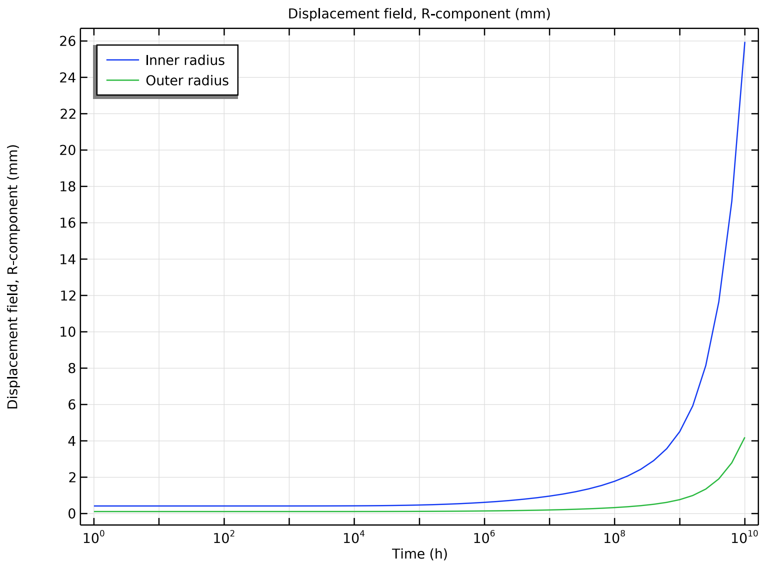

Figure 2: Radial displacement at the inner radius and the outer radius.

In the following table you can compare the values at time 10^{10} h with the reference values:

RADIUS	COMSOL MULTIPHYSICS	REFERENCE (Ref. 1)
200 mm	26.0 mm	26.1 mm
500 mm	4.20 mm	4.22 mm

Initially, the mechanical and thermal load have greater influence on the inner boundary of the sphere and results in larger creep strains. This with time causes relaxation that propagates from the inner radius toward the outer radius. This phenomena is visible in Figure 3 where the von Mises stress is shown at 10^8 h

.

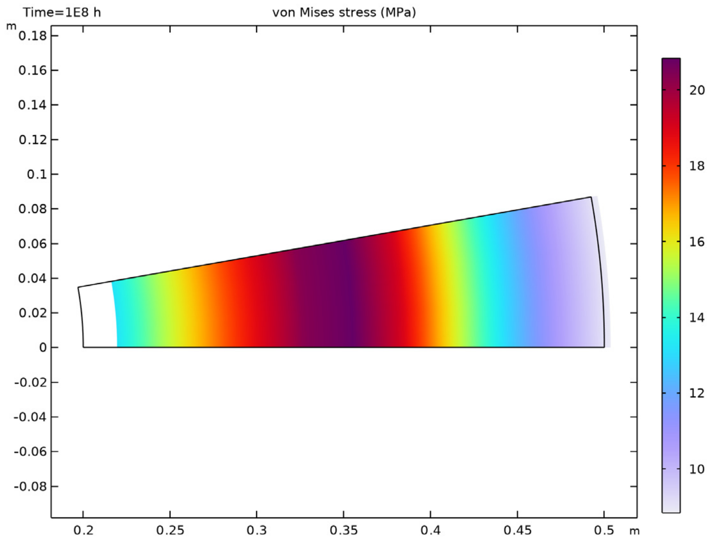

Figure 3: Distribution of von Mises stress at $t = 10^8$ h.

Figure 4 shows the variation of von Mises stress with time at the inner, middle, and outer radii. Notice that significant changes in the stress state occur already at the time 10^4 h — that is, after one millionth of the total analysis time. In the final state, the stresses are completely redistributed. The mechanical load is then larger on the outer exterior than the inner exterior.

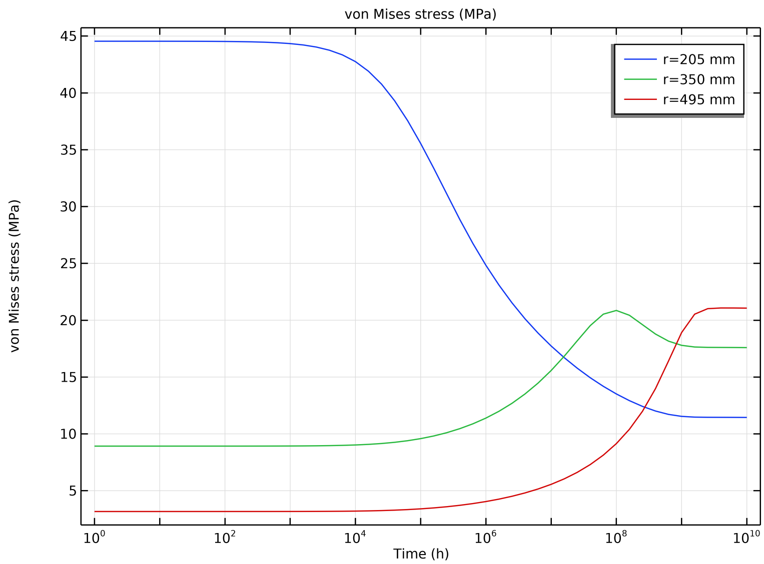

Figure 4: History of the von Mises stress at r = 205 mm, 350 mm, and 495 mm.

The following table shows the values of the von Mises stress at $t = 10^{10}$ h and the reference values for comparison:

RADIUS	COMSOL MULTIPHYSICS	REFERENCE (Ref. 1)
205 mm	11.4 MPa	11.5 MPa
350 mm	17.6 MPa	17.6 MPa
495 mm	21.1 MPa	21.1 MPa

Notes About the COMSOL Implementation

An interesting feature of creep problems is the extreme variation in the time scales over which different phenomena occur. Figure 4 shows that a significant change in stress starts after about 1000 h. It is therefore wise to solve for time steps before and after any significant change in the response. To capture this onset of the stress change, a strict time stepping is used, which forces the solver to provide a solution for all specified time steps. Alternative ways is to either provide an analytical solution for the inner pressure as initial conditions, or to first solve a stationary problem with the inner pressure followed by a time-dependent study.

The creep law defined in Equation 1 follows Norton law, which is available in the Nonlinear Structural Materials Module. In COMSOL Multiphysics it is defined as

$$\frac{\partial \varepsilon_c}{\partial t} = A\left(\frac{\sigma_e}{\sigma_r}\right)^n e^{-\frac{Q}{RT}}$$

In order to normalize the equivalent stress in MPa, set the reference stress $\sigma_r = 1$ MPa. In the exponential temperature function $R = 8.314$ J/(mol·K) and therefore the creep activation energy $Q = 1.039 \cdot 10^5$ J/mol.

Reference

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

Application Library path: `Nonlinear_Structural_Materials_Module/Creep/thermally_induced_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**.

DEFINITIONS

Variables 1

1 In the **Definitions** toolbar, click **Local Variables**.

2 In the **Settings** window for **Variables**, locate the **Variables** section.

3 In the table, enter the following settings:

Name	Expression	Unit	Description
T	333[K]*(1+0.1[m]/sqrt(R^2+Z^2))	K	Prescribed temperature field

GEOMETRY 1

Circle 1 (c1)

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

2 In the **Settings** window for **Circle**, locate the **Size and Shape** section.

3 In the **Radius** text field, type 0.5.

4 In the **Sector angle** text field, type 10.

5 Click to expand the **Layers** section. In the table, enter the following settings:

Layer name	Thickness (m)
Layer 1	0.3

6 Click **Build All Objects**.

Delete Entities 1 (del1)

1 In the **Model Builder** window, right-click **Geometry 1** and choose **Delete Entities**.

2 In the **Settings** window for **Delete Entities**, locate the **Entities or Objects to Delete** section.

3 From the **Geometric entity level** list, choose **Domain**.

4 On the object **c1**, select Domain 1 only.

5 Click **Build All Objects**.

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

SOLID MECHANICS (SOLID)

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 **Model Input** section.

3 From the T list, choose **User defined**. In the associated text field, type `T`.

4 Locate the **Creep Model** section. Find the **Thermal effects** subsection. From the $g\,(T)$ list, choose **Arrhenius**.

5 In the Q text field, type `1.0393e5[J/mol]`.

To enforce a symmetry constraint, add a **Roller** node.

Roller 1

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

2 Select Boundaries 1 and 2 only.

Boundary Load 1

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

2 Select Boundary 3 only.

3 In the **Settings** window for **Boundary Load**, locate the **Coordinate System Selection** section.

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

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

0	t1
-30[MPa]	n

6 In the **Model Builder** window, click **Solid Mechanics (solid)**.

7 In the **Settings** window for **Solid Mechanics**, locate the **Structural Transient Behavior** section.

8 From the list, choose **Quasistatic**.

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	`10[GPa]`	Pa	Young's modulus and Poisson's ratio
Poisson's ratio	nu	`0.25`	1	Young's modulus and Poisson's ratio
Density	rho	`1000`	kg/m³	Basic
Creep rate coefficient	A_nor	`3e-6[1/h]`	1/s	Norton
Reference stress	sigRef_nor	`1[MPa]`	N/m²	Norton
Stress exponent	n_nor	`5.5`	1	Norton

MESH 1

Mapped 1

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

Size

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

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

3 From the **Predefined** list, choose **Coarse**.

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

4 In the **Output times** text field, type `0`.

5 Click **Range**.

6 In the **Range** dialog, type `0` in the **Start** text field.

7 In the **Stop** text field, type `10`.

8 In the **Step** text field, type `0.2`.

9 From the **Function to apply to all values** list, choose **exp10(x) – Exponential function (base 10)**.

10 Click **Add**.

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

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

13 In the **Relative tolerance** text field, type `1e-4`.

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.

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

5 Select the **Initial step** checkbox. In the associated text field, type `1[min]`.

Setting the initial step ensures an accurate calculation of the creep strain at $t = 0$.

6 Click **Run**.

Set default units for result presentation.

RESULTS

Preferred Units 1

1 In the **Results** toolbar, click **Configurations** and choose **Preferred Units**.

2 In the **Settings** window for **Preferred Units**, locate the **Units** section.

3 Click **Add Physical Quantity**.

4 In the **Physical Quantity** dialog, select **Solid Mechanics** > **Stress tensor (N/m^2)** in the tree.

5 Click **OK**.

6 In the **Settings** window for **Preferred Units**, locate the **Units** section.

7 In the table, enter the following settings:

Quantity	Unit	Preferred unit
Stress tensor	N/m^2	MPa

8 Click **Apply**.

Stress (solid)

Select the solution at 10^8 hours to reproduce Figure 3.

1. In the **Model Builder** window, under **Results** click **Stress (solid)**.
2. In the **Settings** window for **2D Plot Group**, locate the **Data** section.
3. From the **Time (h)** list, choose **1E8**.

Surface 1

1. In the **Model Builder** window, expand the **Stress (solid)** node, then click **Surface 1**.
2. In the **Stress (solid)** toolbar, click **Plot**.
3. Click the **Zoom Extents** button in the **Graphics** toolbar.

Stress, 3D (solid)

Follow the commands below to generate Figure 2.

Radial Displacement

1. In the **Model Builder** window, expand the **Results** > **Stress, 3D (solid)** node.
2. Right-click **Results** and choose **1D Plot Group**.
3. In the **Settings** window for **1D Plot Group**, type `Radial Displacement` in the **Label** text field.
4. Locate the **Axis** section. Select the **x-axis log scale** checkbox.

Point Graph 1

1. Right-click **Radial Displacement** and choose **Point Graph**.
2. Select Points 2 and 4 only.
3. 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** > **Displacement** > **Displacement field - m** > **u - Displacement field, R-component**.
4. Locate the **y-Axis Data** section. From the **Unit** list, choose **mm**.
5. Click to expand the **Legends** section. Select the **Show legends** checkbox.
6. From the **Legends** list, choose **Manual**.
7. In the table, enter the following settings:

Legends
`Inner radius`
`Outer radius`

Radial Displacement

1 In the **Model Builder** window, click **Radial Displacement**.

2 In the **Settings** window for **1D Plot Group**, locate the **Legend** section.

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

4 In the **Radial Displacement** toolbar, click **Plot**.

The commands below generate Figure 4.

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 `205[mm] 350[mm] 495[mm]`.

4 In the **Z** text field, type `0`.

von Mises Stress

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

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

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

4 Locate the **Axis** section. Select the **x-axis log scale** checkbox.

Point Graph 1

1 Right-click **von Mises Stress** 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** > **Stress** > **solid.misesGp - von Mises stress - N/m²**.

3 Locate the **Legends** section. Select the **Show legends** checkbox.

4 From the **Legends** list, choose **Evaluated**.

5 In the **Legend** text field, type `r=eval(r,mm) mm`.

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