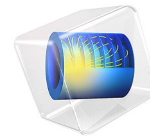
Created in COMSOL Multiphysics 6.1



Stress Relaxation of a Viscoelastic Tube

Introduction

This example studies the temperature effects on the viscoelastic stress relaxation in a generalized Maxwell material with four branches.

Assume that the viscous part of the deformation is incompressible, so that the volume change is purely elastic. The relaxation shear modulus function is approximated by a Prony series as

$$\Gamma(t) = G + \sum_{m=1}^{N} G_m \exp\left(-\frac{t}{\tau_m}\right) \tag{1}$$

where G_m represents the stiffness of the spring in the m^{th} Maxwell branch, and τ_m is the relaxation time of the spring-dashpot pair in the same branch.

The instantaneous shear modulus is defined as

$$G_0 = G + \sum_{m=1}^{N} G_m$$

The alternative form of Equation 1 is

$$\Gamma(t) = G_0\left[\mu_0 + \sum_{m=1}^{N} \mu_m \exp\left(-\frac{t}{\tau_m}\right)\right]$$

where the constants $\mu_m = G_m/G_0$ are such that

$$\sum_{m=0}^{N} \mu_m = 1$$

For many materials the viscoelastic properties have a strong dependence on the temperature. A common assumption is that the material is *thermorheologically simple* (TRS). In a material of this class, a change in temperature can be directly transformed into a change in the time scale. Thus, the relaxation time for a TRS material is modified to $a_T(T)\tau_m$, where $a_T(T)$ is a shift function. One of the most commonly used shift functions is defined by the Williams-Landel-Ferry (WLF) equation:

$$\log(a_T) = \frac{-C_1(T-T_0)}{C_2+(T-T_0)}$$

where a base-10 logarithm is assumed. Usually T_0 is the *glass transition temperature* of the material. Note that $a_T(T_0) = 1$. If the temperature drops below $T_0 - C_2$, the WLF equation is no longer valid. The constants C_1 and C_2 are material dependent.

Model Definition

A long thick-walled cylinder has an inner radius of 5 mm and an outer radius of 10 mm. The inner surface is subjected to a prescribed radial displacement of 1 µm. A four-branch generalized Maxwell model represents the viscoelastic material.

In this example, you study the decay of the stresses during a period of two hours under the influence of the temperature field. You model a quarter of the cylinder and use the Solid Mechanics interface to compute the displacements in the cylinder cross section. The geometry is shown in Figure 1.

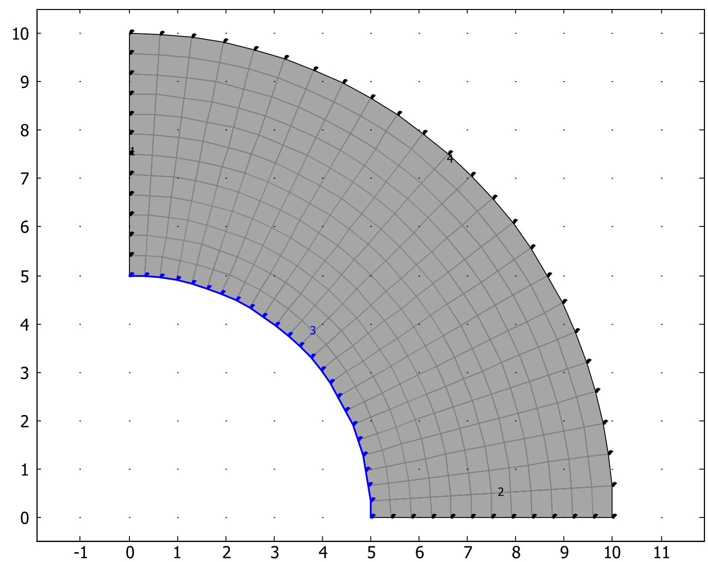


Figure 1: Geometry and mesh.

Study the following two cases:

- Apply a stationary temperature field, causing the problem to lose its axisymmetry.
- Make the temperature field transient.

MATERIAL PARAMETERS

- Elastic data: instantaneous shear modulus G_0 = 27.46 GPa and long-term shear modulus of $G = \mu_0 G_0$, bulk modulus K = 39.88 GPa.
- Viscoelastic data: four-branch Generalized Maxwell material with the shear modulus per branch defined from $G_m = \mu_m G_0$
  - $\mu_1 = 0.04$, τ_1 = 30 s
  - $\mu_2 = 0.08$, τ_2 = 300 s
  - $\mu_3 = 0.09$, τ_3 = 3000 s
  - $\mu_4 = 0.25$, τ_4 = 12000 s
- Thermal properties: A WLF shift function is used with C_1 = 17.44 and C_2 = 51.6. These values are reasonable approximations for many polymers.
- The reference temperature is 500 K.
- Heat transfer properties: ρ = 7850 kg/m^3, C_p = 2100 J/(kg·K), $k = 6\cdot10^{-2}$ W/(m·K).

CONSTRAINTS

- The circumferential displacements are constrained on the radial edges.
- The inner edge of the hole is constrained to a radial displacement of 1 μm.

HEAT TRANSFER BOUNDARY CONDITIONS

- Stationary analysis: The inner and outer circular edges both have a temperature distribution varying linearly with the y-coordinate from 500 K at the $y = 0$ symmetry section to 506 K at the $x = 0$ symmetry section.
- Transient analysis: The temperature distribution obtained from the stationary analysis is used as initial condition, while all edges are insulated.

Results and Discussion

In the first case, the temperature field is stationary and is shown in Figure 2.

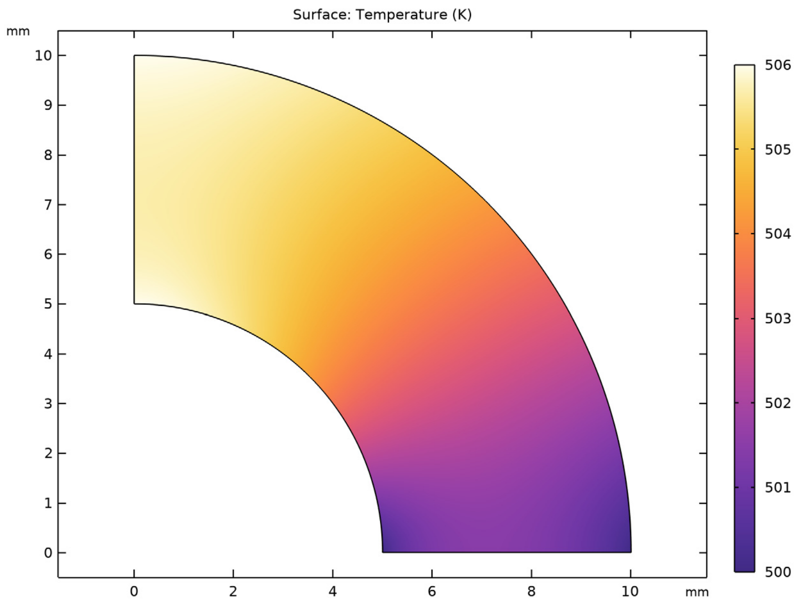


Figure 2: Stationary temperature distribution.

The stress relaxation is faster where the temperature is higher. In Figure 3, the plots of radial stresses at radius 7.5 mm display this effect. The solid line is taken at $y = 0$ (cold) and the dashed line at $x = 0$ (warm).

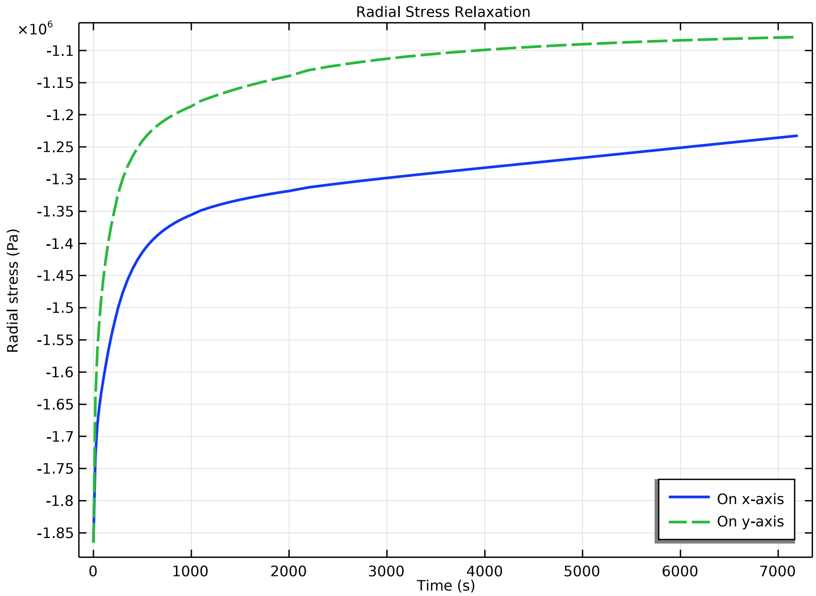


Figure 3: Stress relaxation under a stationary temperature distribution.

In the second case, the temperature initially has the same distribution as shown in Figure 2, but it is allowed to settle in time to a final homogeneous value. Again, compare the radial stresses at radius 7.5 mm. The initial behavior is similar to the previous case, but as the difference in material properties decreases, the curves approach each other (Figure 4). The strain rate in the initially warm point decreases, while it increases in the initially cold point.

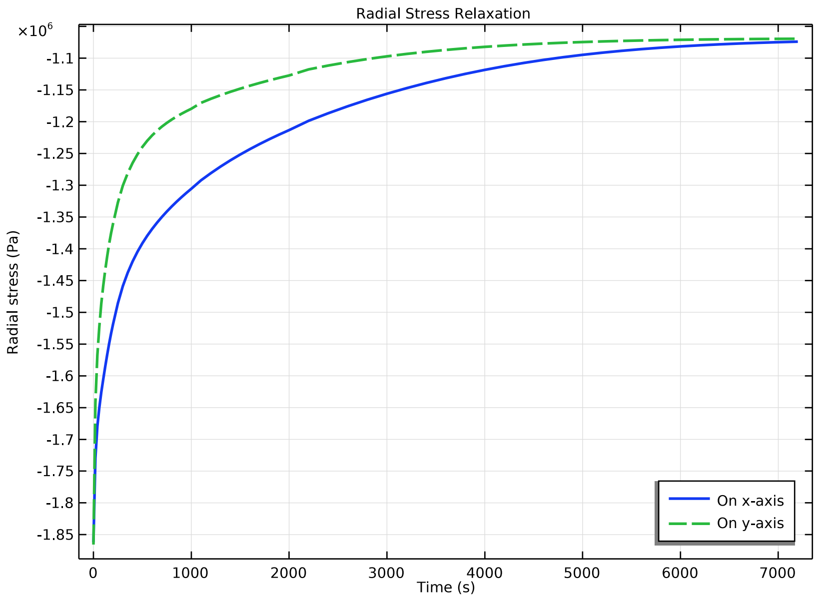


Figure 4: Stress relaxation under temperature settling conditions.

Notes About the COMSOL Implementation

When you solve a problem where a load is applied instantaneously at the beginning of a transient analysis, you can choose between two approaches to compute the initial stresses: either apply the load over a short period of time at the beginning of the time-dependent study. An alternative approach is to set the viscoelastic material to use an instantaneous static stiffness, and to add a separate stationary step before the time-dependent step. In the present example, you use the later method.

The results computed during the stationary step are stored and used as the initial value for the consequent transient analysis. COMSOL Multiphysics handles this automatically if you use a single study with one solver sequence. In this example, however, you study two different relaxation histories with different thermal boundary conditions. In the second case, the thermal boundary conditions differ between the computation of the initial state and the transient analysis. For this reason you use two separate studies for the stationary and the time-dependent studies.

Application Library path: `Structural_Mechanics_Module/Material_Models/viscoelastic_tube`

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 In the **Select Physics** tree, select **Heat Transfer>Heat Transfer in Solids (ht)**.

5 Click **Add**.

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
G_inst	27.46[GPa]	2.746E10 Pa	Instantaneous shear modulus

GEOMETRY 1

1 In the **Model Builder** window, under **Component 1 (comp1)** click **Geometry 1**.

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

3 From the **Length unit** list, choose **mm**.

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 `10`.

4 In the **Sector angle** text field, type `90`.

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

Layer name	Thickness (mm)
`Layer 1`	`5`

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**.

The equations only need to include first-order time derivatives since the inertia effects can be neglected for this class of problems.

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

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**, locate the **Linear Elastic Material** section.

3 From the **Specify** list, choose **Bulk modulus and shear modulus**.

Viscoelasticity 1

1 In the **Physics** toolbar, click **Attributes** and choose **Viscoelasticity**.

2 In the **Settings** window for **Viscoelasticity**, locate the **Viscoelasticity Model** section.

3 In the table, enter the following settings:

Branch	**Shear modulus (Pa)**	**Relaxation time (s)**
1	0.04*G_inst	30

4 Click **Add**.

5 In the table, enter the following settings:

Branch	**Shear modulus (Pa)**	**Relaxation time (s)**
2	0.08*G_inst	300

6 Click **Add**.

7 In the table, enter the following settings:

Branch	**Shear modulus (Pa)**	**Relaxation time (s)**
3	0.09*G_inst	3000

8 Click **Add**.

9 In the table, enter the following settings:

Branch	**Shear modulus (Pa)**	**Relaxation time (s)**
4	0.25*G_inst	12000

10 Locate the **Thermal Effects** section. From the **Shift function** list, choose **Williams-Landel-Ferry**.

11 In the T_{ref} text field, type 500[K].

12 Locate the **Viscoelasticity Model** section. From the **Stiffness used in stationary studies** list, choose **Instantaneous**.

Symmetry 1

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

2 Select Boundaries 1 and 2 only.

Prescribed Displacement 1

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

2 Select Boundary 3 only.

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

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

5 Locate the **Prescribed Displacement** section. Select the **Prescribed in n direction** check box.

6 In the u_{0n} text field, type `-1[um]`.

HEAT TRANSFER IN SOLIDS (HT)

In the **Model Builder** window, under **Component 1 (comp1)** click **Heat Transfer in Solids (ht)**.

Temperature 1

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

2 Select Boundaries 3 and 4 only.

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

4 In the T_0 text field, type `(500+6*y/sqrt(x^2+y^2))[K]`.

MATERIALS

Material 1 (mat1)

1 In the **Model Builder** window, under **Component 1 (comp1)** right-click **Materials** and choose **Blank Material**.

2 Select Domain 1 only.

3 In the **Settings** window for **Material**, locate the **Material Contents** section.

4 In the table, enter the following settings:

Property	Variable	Value	Unit	Property group
Bulk modulus	K	39.88[GPa]	N/m²	Bulk modulus and shear modulus
Shear modulus	G	0.54*G_inst	N/m²	Bulk modulus and shear modulus
Density	rho	7850[kg/m^3]	kg/m³	Basic
Thermal conductivity	k_iso ; kii = k_iso, kij = 0	0.06[N/(s*K)]	W/(m·K)	Basic
Heat capacity at constant pressure	Cp	2100[J/(kg*K)]	J/(kg·K)	Basic

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 12.

Distribution 2

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

2 Select Boundary 3 only.

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

4 In the **Number of elements** text field, type 24.

5 Click **Build All**.

ROOT

Next, add a stationary study to model the instantaneous preloading step.

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>Stationary**.

4 Click **Add Study** in the window toolbar.

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

STUDY: STATIONARY

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

2 In the **Settings** window for **Study**, type `Study: Stationary` in the **Label** text field.

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

RESULTS

Visualize the stationary temperature.

Temperature (ht)

1 In the **Model Builder** window, under **Results** click **Temperature (ht)**.

2 In the **Temperature (ht)** toolbar, click **Plot**.

Isothermal Contours (ht), Stress (solid), Temperature (ht)

1 In the **Model Builder** window, under **Results**, Ctrl-click to select **Stress (solid)**, **Temperature (ht)**, and **Isothermal Contours (ht)**.

2 Right-click and choose **Group**.

Stationary Results

In the **Settings** window for **Group**, type `Stationary Results` in the **Label** text field.

Proceed to add a transient study for analyzing the stress relaxation process.

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: TRANSIENT (CONSTANT TEMPERATURE)

1 In the **Model Builder** window, click **Study 2**.

2 In the **Settings** window for **Study**, type `Study: Transient (Constant Temperature)` in the **Label** text field.

Step 1: Time Dependent

1 In the **Model Builder** window, under **Study: Transient (Constant Temperature)** 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 `range(0,20,200) range(250,50,1000) range(1100,100,2000) range(2200,200,7200)`.

Solution 2 (sol2)

1 In the **Study** toolbar, click **Show Default Solver**.

Prescribe the solution of the stationary study as the initial step in the transient analysis.

2 In the **Model Builder** window, expand the **Solution 2 (sol2)** node, then click **Dependent Variables 1**.

3 In the **Settings** window for **Dependent Variables**, locate the **General** section.

4 From the **Defined by study step** list, choose **User defined**.

5 Locate the **Initial Values of Variables Solved For** section. From the **Method** list, choose **Solution**.

6 From the **Solution** list, choose **Solution 1 (sol1)**.

7 Locate the **Scaling** section. From the **Method** list, choose **Initial value based**.

Force the solver to evaluate results in between the specified time steps.

8 In the **Model Builder** window, under **Study: Transient (Constant Temperature)>Solver Configurations>Solution 2 (sol2)** click **Time-Dependent Solver 1**.

9 In the **Settings** window for **Time-Dependent Solver**, click to expand the **Time Stepping** section.

10 From the **Steps taken by solver** list, choose **Intermediate**.

11 In the **Study** toolbar, click **= Compute**.

RESULTS

Temperature (ht) 1

Display the stress relaxation.

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 **X** text field, type `7.5`.

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

5 Locate the **Data** section. From the **Dataset** list, choose **Study: Transient (Constant Temperature)/Solution 2 (sol2)**.

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 **X** text field, type `0`.

4 In the **Y** text field, type `7.5`.

5 Locate the **Data** section. From the **Dataset** list, choose **Study: Transient (Constant Temperature)/Solution 2 (sol2)**.

Stress Relaxation (Constant Temperature)

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

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

3 From the **Dataset** list, choose **Study: Transient (Constant Temperature)/Solution 2 (sol2)**.

4 In the **Label** text field, type `Stress Relaxation (Constant Temperature)`.

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

6 In the **Title** text area, type `Radial Stress Relaxation`.

7 Locate the **Plot Settings** section.

8 Select the **y-axis label** check box. In the associated text field, type `Radial stress (Pa)`.

Point Graph 1

1 Right-click **Stress Relaxation (Constant Temperature)** and choose **Point Graph**.

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

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

4 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.sxx - Stress tensor, xx-component**.

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

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

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

8 In the table, enter the following settings:

Legends
`On x-axis`

Point Graph 2

1 In the **Model Builder** window, right-click **Stress Relaxation (Constant Temperature)** and choose **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>Stress> Stress tensor (spatial frame) - N/m²>solid.syy - Stress tensor, yy-component**.

5 Locate the **Coloring and Style** section. From the **Width** list, choose **2**.

6 Find the **Line style** subsection. From the **Line** list, choose **Dashed**.

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

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

9 In the table, enter the following settings:

Legends
`On y-axis`

Stress Relaxation (Constant Temperature)

1 In the **Model Builder** window, click **Stress Relaxation (Constant Temperature)**.

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

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

Isothermal Contours (ht) 1, Stress (solid) 1, Stress Relaxation (Constant Temperature), Temperature (ht) 1

1 In the **Model Builder** window, under **Results**, Ctrl-click to select **Stress (solid) 1**, **Temperature (ht) 1**, **Isothermal Contours (ht) 1**, and **Stress Relaxation (Constant Temperature)**.

2 Right-click and choose **Group**.

Transient Results (Constant Temperature)

In the **Settings** window for **Group**, type `Transient Results (Constant Temperature)` in the **Label** text field.

Now, simulate the stress relaxation at a constant temperature.

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 3

Step 1: Time Dependent

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

2 In the **Output times** text field, type `range(0,20,200) range(250,50,1000) range(1100,100,2000) range(2200,200,7200)`.

3 Locate the **Physics and Variables Selection** section. Select the **Modify model configuration for study step** check box.

4 In the tree, select **Component 1 (comp1)>Heat Transfer in Solids (ht)>Temperature 1**.

5 Right-click and choose **Disable**.

6 In the **Model Builder** window, click **Study 3**.

7 In the **Settings** window for **Study**, type `Study: Transient (Variable Temperature)` in the **Label** text field.

Solution 3 (sol3)

1 In the **Study** toolbar, click **Show Default Solver**.

Prescribe the solution of the stationary study as the initial step in the transient analysis.

2 In the **Model Builder** window, expand the **Solution 3 (sol3)** node, then click **Dependent Variables 1**.

3 In the **Settings** window for **Dependent Variables**, locate the **General** section.

4 From the **Defined by study step** list, choose **User defined**.

5 Locate the **Initial Values of Variables Solved For** section. From the **Method** list, choose **Solution**.

6 From the **Solution** list, choose **Solution 1 (sol1)**.

7 Locate the **Scaling** section. From the **Method** list, choose **Initial value based**.

Force solver to evaluate results in between the specified time steps.

8 In the **Model Builder** window, under **Study: Transient (Variable Temperature)> Solver Configurations>Solution 3 (sol3)** click **Time-Dependent Solver 1**.

9 In the **Settings** window for **Time-Dependent Solver**, locate the **Time Stepping** section.

10 From the **Steps taken by solver** list, choose **Intermediate**.

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

RESULTS

Stress (solid) 2

Plot the results of the stress relaxation.

Cut Point 2D 3

1 In the **Model Builder** window, under **Results>Datasets** right-click **Cut Point 2D 1** and choose **Duplicate**.

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

3 From the **Dataset** list, choose **Study: Transient (Variable Temperature)/Solution 3 (sol3)**.

Cut Point 2D 4

1 In the **Model Builder** window, under **Results>Datasets** right-click **Cut Point 2D 2** and choose **Duplicate**.

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

3 From the **Dataset** list, choose **Study: Transient (Variable Temperature)/Solution 3 (sol3)**.

Stress Relaxation (Variable Temperature)

1 In the **Model Builder** window, right-click **Stress Relaxation (Constant Temperature)** and choose **Duplicate**.

2 Right-click **Stress Relaxation (Constant Temperature) 1** and choose **Move Out**.

3 In the **Model Builder** window, click **Stress Relaxation (Constant Temperature) 1**.

4 Drag and drop below **Isothermal Contours (ht) 2**.

5 In the **Settings** window for **1D Plot Group**, type `Stress Relaxation (Variable Temperature)` in the **Label** text field.

Point Graph 1

1 In the **Model Builder** window, expand the **Stress Relaxation (Variable Temperature)** node, then click **Point Graph 1**.

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

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

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 **Cut Point 2D 4**.

4 In the **Stress Relaxation (Variable Temperature)** toolbar, click  **Plot**.

Isothermal Contours (ht) 2, Stress (solid) 2, Stress Relaxation (Variable Temperature), Temperature (ht) 2

1 In the **Model Builder** window, under **Results**, Ctrl-click to select **Stress (solid) 2**, **Temperature (ht) 2**, **Isothermal Contours (ht) 2**, and **Stress Relaxation (Variable Temperature)**.

2 Right-click and choose **Group**.

Transient Results (Variable Temperature)

In the **Settings** window for **Group**, type `Transient Results (Variable Temperature)` in the **Label** text field.