Model created in COMSOL Multiphysics 6.4

Tensile Test with Strain Rate Dependent Plasticity

Introduction

When a metal is deformed plastically at a high strain rate, the hardening function will exhibit higher values than at quasistatic conditions. That is, a higher stress is reached for a certain strain.

In this example, the Johnson–Cook material model is used to simulate this behavior for a tensile test run at different loading rates. The current yield stress σ_{ys} is in this model described by Equation 1.

$$\sigma_{ys}(\varepsilon_{pe}) = (\sigma_{ys0} + k\varepsilon_{pe}^{n})\left(1 + C\max\left(0, \log\left(\frac{\dot{\varepsilon}_{pe}}{\dot{\varepsilon}_{0}}\right)\right)\right)(1 - f(T_{h})) \tag{1}$$

Here, σ_{ys0}, k, n, C, and $\dot{\varepsilon}_0$ are material parameters. The function $f(T_h)$ describes the temperature softening, where T_h is a normalized temperature. It is often expressed as $f(T_h) = T_h^m$ where m is a material parameter.

The current yield stress is a product of three factors:

- A standard strain hardening of the Ludvik form, with two parameters (k and n)
- A strain rate dependent factor with two parameters (C and $\dot{\varepsilon}_0$). The reference strain rate $\dot{\varepsilon}_0$ is the one at which k and n are determined
- A temperature-dependent factor for softening. The reference temperature used to define T_h is typically the one at which k and n are determined

Two different studies are performed to quantify the influence of the three factors. In the first study, both strain rate hardening and temperature softening are included, while in the second study, only the strain rate hardening is considered. In the model, four different values of the strain rate are investigated.

The analysis is coupled to a heat-transfer analysis, in which the heating caused by the plastic deformation is included. Due to this, the temperature will increase during the process, which is the source of the thermal softening.

Model Definition

GEOMETRY

A 100 mm long cylindrical test specimen having a diameter of 10 mm in its central section is used. The detailed geometry is shown in Figure 1.

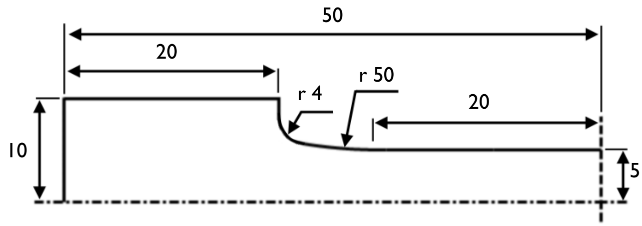

Figure 1: Geometry of the test specimen.

Axisymmetry is assumed, and only one half of the specimen is modeled due to the symmetry in the axial direction.

MATERIAL MODEL

The material is steel with properties as shown in Table 1.

TABLE 1: MATERIAL PROPERTIES.

PROPERTY	SYMBOL	VALUE
Young's Modulus	E	200 GPa
Poisson's ratio	ν	0.3
Coefficient of thermal expansion	α	$12.3\cdot10^{-6}$ 1/K
Initial yield strength	σ_{ys0}	400 MPa
Strength coefficient	k	200 MPa
Hardening exponent	n	0.5
Reference strain rate	$\dot{\varepsilon}_0$	1 1/s
Strain rate strength coefficient	C	0.12
Temperature exponent	m	0.6
Reference temperature	T_{ref}	293.15 K
Melting temperature	T_m	1700 K
Thermal conductivity	k	44.5 W/(m·K)
Mass density	ρ	7850 kg/m^3
Heat capacity at constant pressure	c_p	475 J/(kg·K)

BOUNDARY CONDITIONS

Mechanical

At the thick end of the specimen, the displacement is prescribed in the axial direction. The displacement varies linearly with time, and the maximum elongation of the specimen is 10 mm. This elongation corresponds to an average strain of 10%, but since the plastic deformation occurs only in the thinner part of the specimen, the actual plastic strains will be of the order 20%.

Symmetry conditions are applied in the axial direction at the symmetry plane.

Thermal

On all external boundaries, free convection to external room temperature (293.15 K) is assumed. The heat transfer coefficient is assumed to be 15 W/(m^2·K).

HEAT GENERATION

The inelastic deformation causes heat generation. The generated power per unit volume is the product of stress and the rate of plastic strain:

$$q = \sigma : \dot{\varepsilon}_{\mathrm{pl}} \tag{2}$$

This power is used as a source term in the heat transfer analysis through the **Thermal Expansion** multiphysics coupling.

THERMAL EXPANSION

Due to the heating, there will also be some thermal expansion of the specimen. This is included in the analysis through the **Thermal Expansion** multiphysics coupling, even though the effect is not large. The ratio between thermal and plastic strain can be used to quantify its influence. It can be estimated if the heat produced by plastic dissipation is assumed not to be conducted away from where it is generated:

$$\frac{\varepsilon_{\mathrm{th}}}{\varepsilon_{\mathrm{pl}}} = \frac{\alpha \Delta T}{\varepsilon_{\mathrm{pl}}} = \frac{\alpha \sigma_{\mathrm{y}} \varepsilon_{\mathrm{pl}}}{\varepsilon_{\mathrm{pl}} \rho c_{\mathrm{p}}} = \frac{\alpha \sigma_{\mathrm{y}}}{\rho c_{\mathrm{p}}} = \frac{12.3 \cdot 10^{-6} \cdot 4 \cdot 10^{8}}{7850 \cdot 475} \approx 1.3 \cdot 10^{-3} \tag{3}$$

Results and Discussion

In Figure 2, a general overview of the stress state is shown for the highest loading rate in the first study. The stress in the central part is about 600 MPa, whereas the initial yield stress is 400 MPa. It can be found that the plastic strain at this stage is approximately 20%, and the contributions to the current stress can be estimated from Equation 1 as

$(400 \text{ MPa} + 200 \text{ MPa}\cdot 0.2^{0.5})(1+0.12\cdot\log(29))(1\text{-}0.025^{0.6}) = 489 \text{ MPa} \cdot 1.36 \cdot 0.89$

Thus, there is about 22% strain hardening, 36% strain rate hardening, and 11% temperature softening.

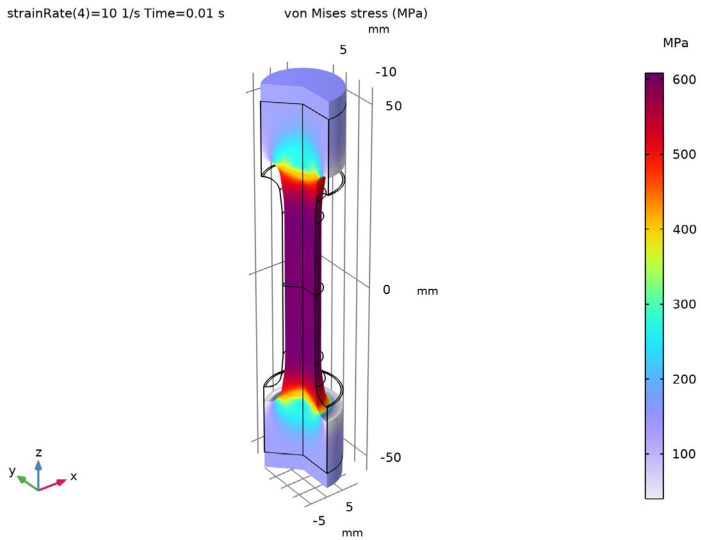

Figure 2: Distribution of von Mises stress at the end of the tensile test at the highest strain rate.

The influence of the strain rate is shown in Figure 3. The axial stress at the center of the bar is plotted as a function of the axial strain. Since the stress state is close to uniaxial, this graph essentially shows the constitutive law. For the two lower strain rates, the strain rate hardening effect is negligible since these are below the reference strain rate. It becomes significant when the average strain rate approaches 1/s. Figure 4 shows a corresponding graph of what would be measured in a testing machine, that is the total force versus the end point displacement.

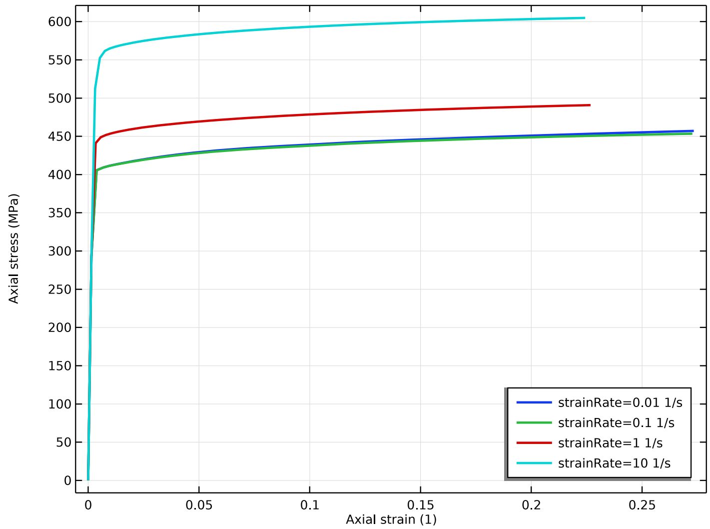

Figure 3: Axial stress and strain at the center of the test specimen for the four tensile tests at different strain rates.

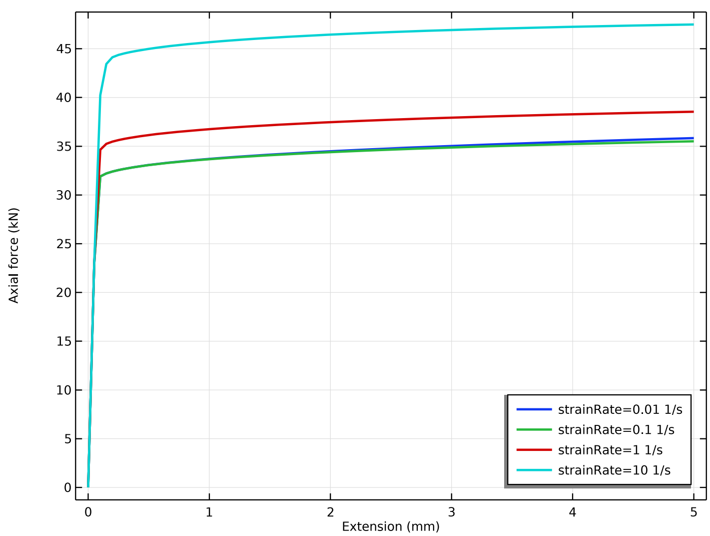

Figure 4: Force versus displacement for the four tensile tests at different strain rates.

In the second study, the temperature softening part of the constitutive law is switched off. The effect on the stress–strain relation is shown in Figure 5. Without the thermal softening effect, the hardening is much more pronounced. It can also be noted that the curves for the two lowest strain rates now completely coincide. The small difference that could be seen in Figure 3 was an effect of heating only.

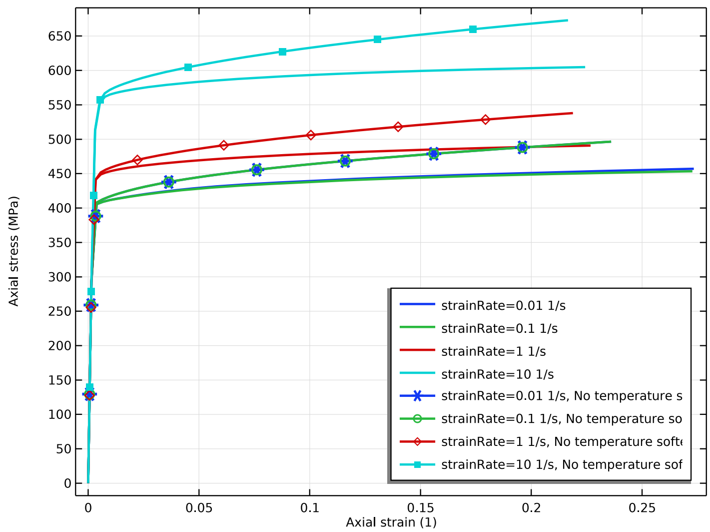

Figure 5: Comparison of stress vs. strain with and without thermal softening.

In Figure 6, the plastic strain distribution at the end of the experiment is shown for all four loading rates in the first study. At the lower strain rates, the maximum plastic deformation occurs in the central parts of the bar, whereas at the higher strain rates, the peak value actually occurs in a region closer to the loaded end. This redistribution is an effect of the thermal softening. The temperature will be higher at the center of the specimen at low loading rates.

In Figure 7, the final temperature is shown. At the lowest loading rate, the whole process takes 100 s. There is thus enough time for a substantial redistribution of the temperature field. At the two highest loading rates, the temperature field to a large extent matches the strain distribution, since the time is not sufficient for any substantial diffusion of heat. The temperature is however higher at the highest loading rate, since the strain rate hardening causes a higher stress and thus a larger heat production for the same strain.

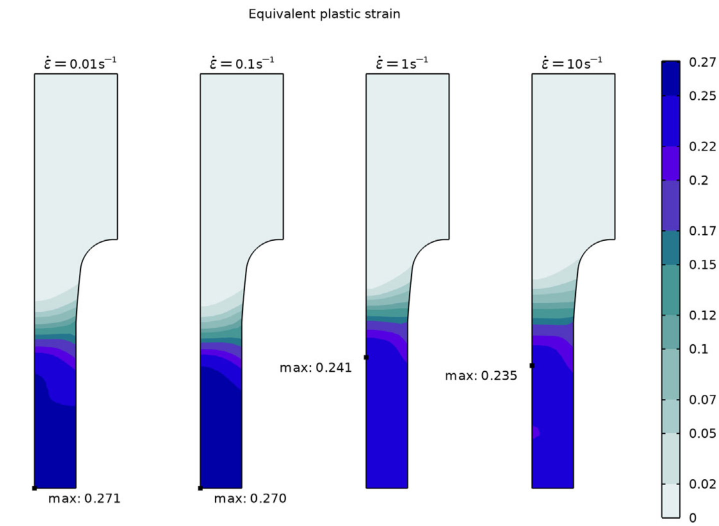

Figure 6: Distribution of plastic strain at the end of the process for all four strain rates.

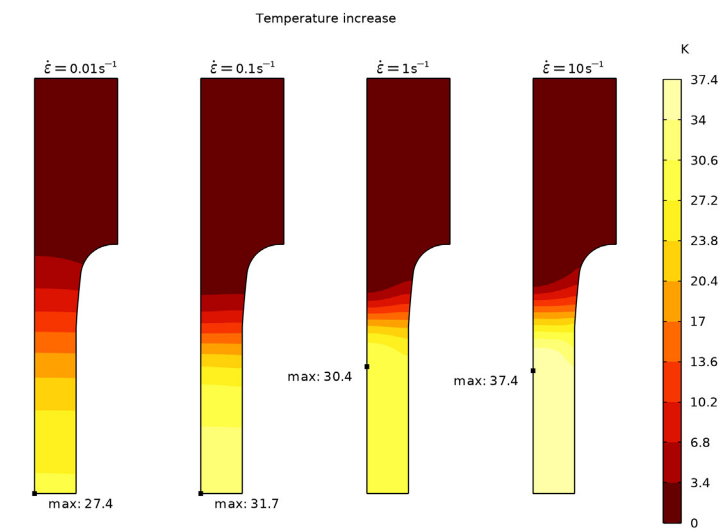

Figure 7: Increase in temperature at the end of the process for all four strain rates.

Note that all analyses were performed using an assumption of geometric linearity in order to speed up the analysis and simplify comparisons. In reality, geometric nonlinearity should be taken into account; without it, the strain localization ("necking") that may occur at the center of the bar cannot be predicted.

Application Library path: `Nonlinear_Structural_Materials_Module/Plasticity/strain_rate_dependent_plasticity`

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 > Thermal–Structure Interaction > Thermal Stress, 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
totL	100[mm]	0.1 m	Total length of the specimen
avgStrain	0.1	0.1	Average strain

Name	Expression	Value	Description
wMax	avgStrain*totL/2	0.005 m	Max displacement of the symmetric half
strainRate	0.1[1/s]	0.1 1/s	Strain rate
tFinal	avgStrain/strainRate	1 s	End time

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**.

Polygon 1 (pol1)

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

2 In the **Settings** window for **Polygon**, locate the **Object Type** section.

3 From the **Type** list, choose **Open curve**.

4 Locate the **Coordinates** section. From the **Data source** list, choose **Vectors**.

5 In the **r** text field, type `5 5 0  0 10 10 6.01`.

6 In the **z** text field, type `20 0 0 50 50 30 30`.

Quadratic Bézier 1 (qb1)

1 In the **Geometry** toolbar, click **More Primitives** and choose **Quadratic Bézier**.

2 In the **Settings** window for **Quadratic Bézier**, locate the **Control Points** section.

3 In row **1**, set **r** to `6.01`.

4 In row **1**, set **z** to `30`.

5 In row **2**, set **r** to `5.25`.

6 In row **2**, set **z** to `25`.

7 In row **3**, set **r** to `5`.

8 In row **3**, set **z** to `20`.

Convert to Solid 1 (csol1)

1 In the **Geometry** toolbar, click **Conversions** and choose **Convert to Solid**.

2 Click in the **Graphics** window and then press Ctrl+A to select both objects.

3 In the **Settings** window for **Convert to Solid**, click **Build Selected**.

Fillet 1 (fil1)

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

2 On the object **csol1**, select Point 5 only.

3 In the **Settings** window for **Fillet**, locate the **Radius** section.

4 In the **Radius** text field, type 4.

5 Click **Build Selected**.

SOLID MECHANICS (SOLID)

Symmetry Plane 1

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

2 Select Boundary 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 **Prescribed Displacement** section.

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

5 In the u_{0z} text field, type `wMax*t/tFinal`.

Linear Elastic Material 1

In the **Model Builder** window, 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 **Johnson–Cook**.

MATERIALS

Structural steel

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

2 In the **Settings** window for **Material**, type `Structural steel` in the **Label** text field.

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

Property	Variable	Value	Unit	Property group
Young's modulus	E	200[GPa]	Pa	Young's modulus and Poisson's ratio
Poisson's ratio	nu	0.3	1	Young's modulus and Poisson's ratio
Density	rho	7850[kg/m^3]	kg/m³	Basic
Initial yield stress	sigmags	400[MPa]	Pa	Elastoplastic material model
Strength coefficient	k_jcook	200[MPa]	Pa	Johnson-Cook
Hardening exponent	n_jcook	0.5	1	Johnson-Cook
Reference strain rate	epet0_jcook	1	1/s	Johnson-Cook
Strain rate strength coefficient	C_jcook	0.12	1	Johnson-Cook
Temperature exponent	m_jcook	0.6		Johnson-Cook
Thermal conductivity	k_iso ; kii = k_iso, kij = 0	44.5[W/(m*K)]	W/(m·K)	Basic
Heat capacity at constant pressure	Cp	475[J/(kg*K)]	J/(kg·K)	Basic
Coefficient of thermal expansion	alpha_iso ; alphaii = alpha_iso, alphaij = 0	12.3e-6[1/K]	1/K	Basic

HEAT TRANSFER IN SOLIDS (HT)

Heat Flux 1

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

2 In the **Settings** window for **Heat Flux**, locate the **Heat Flux** section.

3 From the **Flux type** list, choose **Convective heat flux**.

4 In the h text field, type 15.

5 Select Boundaries 4–8 only.

MULTIPHYSICS

Thermal Expansion 1 (te1)

1 In the **Model Builder** window, under **Component 1 (comp1)** > **Multiphysics** click **Thermal Expansion 1 (te1)**.
2 In the **Settings** window for **Thermal Expansion**, locate the **Heat Sources** section.
3 Clear the **Thermoelastic damping** checkbox.
4 Select the **Mechanical losses** checkbox.

   As a default, the losses due to material nonlinearities are not computed. You have to explicitly enable that to get access to the heat source.
5 Click the **Show More Options** button in the **Model Builder** toolbar.
6 In the **Show More Options** dialog, in the tree, select the checkbox for the node **Physics** > **Advanced Physics Options**.
7 Click **OK**.

SOLID MECHANICS (SOLID)

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**, click to expand the **Energy Dissipation** section.
3 From the **Store dissipation** list, choose **Individual contributions**.

MESH 1

1 In the **Model Builder** window, under **Component 1 (comp1)** click **Mesh 1**.
2 In the **Settings** window for **Mesh**, locate the **Sequence Type** section.
3 From the list, choose **User-controlled mesh**.

Size Expression 1

1 In the **Model Builder** window, right-click **Free Triangular 1** and choose **Size Expression**.
2 In the **Settings** window for **Size Expression**, locate the **Element Size Expression** section.
3 In the **Size expression** text field, type `if(z<30[mm],1[mm],3[mm])`.
4 Click **Build All**.

STUDY 1: WITH THERMAL SOFTENING

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

2 In the **Settings** window for **Study**, type `Study 1: With Thermal Softening` in the **Label** text field.

Parametric Sweep

1 In the **Study** toolbar, click **Parametric Sweep**.

2 In the **Settings** window for **Parametric Sweep**, locate the **Study Settings** section.

3 Click **Add**.

4 In the table, enter the following settings:

Parameter name	Parameter value list	Parameter unit
strainRate (Strain rate)	`0.01 0.1 1 10`	`1/s`

5 In the table, click to select the cell at row number 1 and column number 3.

Step 1: Time Dependent

1 In the **Model Builder** window, 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(0,0.01*tFinal,0.1*tFinal) range(0.12*tFinal,0.02*tFinal,tFinal)`.

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

RESULTS

Examine the distribution of stress after the fastest deformation.

Mirror 3D 1

1 In the **Results** toolbar, click **More Datasets** and choose **Mirror 3D**.

2 In the **Settings** window for **Mirror 3D**, locate the **Plane Data** section.

3 From the **Plane** list, choose **XY-planes**.

Stress, 3D (solid)

1 In the **Model Builder** window, under **Results** click **Stress, 3D (solid)**.

2 In the **Settings** window for **3D Plot Group**, locate the **Data** section.

3 From the **Dataset** list, choose **Mirror 3D 1**.

4 Locate the **Color Legend** section. Select the **Show units** checkbox.

Surface 1

1 In the **Model Builder** window, expand the **Stress, 3D (solid)** 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, 3D (solid)** toolbar, click **Plot**.

Stress vs. Strain

Add a graph comparing axial stress versus strain at the center of the specimen for all strain rates.

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

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

3 Locate the **Data** section. From the **Dataset** list, choose **Study 1: With Thermal Softening/Parametric Solutions 1 (sol2)**.

Point Graph 1

1 Right-click **Stress vs. Strain** and choose **Point Graph**.

2 Select Point 1 only.

3 In the **Settings** window for **Point Graph**, locate the **y-Axis Data** section.

4 In the **Expression** text field, type `solid.sz`.

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

6 Locate the **x-Axis Data** section. From the **Axis source data** list, choose **Inner solutions**.

7 From the **Parameter** list, choose **Expression**.

8 In the **Expression** text field, type `solid.eZZ`.

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

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

11 Find the **Include** subsection. Clear the **Point** checkbox.

Stress vs. Strain

1 In the **Model Builder** window, click **Stress vs. Strain**.

2 In the **Settings** window for **1D Plot Group**, click to expand the **Title** section.

3 From the **Title type** list, choose **None**.

4 Locate the **Plot Settings** section.

5 Select the **x-axis label** checkbox. In the associated text field, type `Axial strain (1)`.

6 Select the **y-axis label** checkbox. In the associated text field, type `Axial stress (MPa)`.

7 Locate the **Legend** section. From the **Position** list, choose **Lower right**.

8 In the **Stress vs. Strain** toolbar, click **Plot**.

DEFINITIONS

Add also a graph of the force as function of displacement. To get the force, you need to sum the reaction forces over the boundary having the prescribed displacement. After defining a new variable, you must update the solution to make that variable accessible for postprocessing.

Integration 1 (intop1)

1 In the **Definitions** toolbar, click **Nonlocal Couplings** and choose **Integration**.

2 In the **Settings** window for **Integration**, locate the **Source Selection** section.

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

4 Select Boundary 3 only.

5 Locate the **Advanced** section. From the **Method** list, choose **Summation over nodes**.

Variables 1

1 In the **Model Builder** window, right-click **Definitions** and choose **Variables**.

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

3 In the table, enter the following settings:

Name	Expression	Unit	Description
`Fz`	`intop1(solid.RFz)`	N	`Axial force`

STUDY 1: WITH THERMAL SOFTENING

In the **Study** toolbar, click **Update Solution**.

RESULTS

Force vs. Displacement

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

2 In the **Settings** window for **1D Plot Group**, type `Force vs. Displacement` in the **Label** text field.

3 Locate the **Data** section. From the **Dataset** list, choose **Study 1: With Thermal Softening/ Parametric Solutions 1 (sol2)**.

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

Global 1

1 Right-click **Force vs. Displacement** 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
Fz	kN	Axial force

4 Locate the **x-Axis Data** section. From the **Axis source data** list, choose **Inner solutions**.

5 From the **Parameter** list, choose **Expression**.

6 In the **Expression** text field, type `wMax*t/tFinal`.

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

8 Locate the **x-Axis Data** section.

9 Select the **Description** checkbox. In the associated text field, type `Extension`.

10 Click to expand the **Legends** section. Find the **Include** subsection. Clear the **Description** checkbox.

11 In the **Force vs. Displacement** toolbar, click **Plot**.

Force vs. Displacement

1 In the **Model Builder** window, click **Force vs. Displacement**.

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

3 From the **Title type** list, choose **None**.

4 In the **Force vs. Displacement** toolbar, click **Plot**.

ROOT

Add a second study in which the temperature softening is ignored.

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 Right-click and choose **Add Study**.

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

STUDY 2: WITHOUT THERMAL SOFTENING

In the **Settings** window for **Study**, type `Study 2: Without Thermal Softening` in the **Label** text field.

Parametric Sweep

1 In the **Study** toolbar, click **Parametric Sweep**.

2 In the **Settings** window for **Parametric Sweep**, locate the **Study Settings** section.

3 Click **Add**.

4 In the table, enter the following settings:

Parameter name	Parameter value list	Parameter unit
strainRate (Strain rate)	`0.01 0.1 1 10`	`1/s`

5 In the table, click to select the cell at row number 1 and column number 3.

Step 1: Time Dependent

1 In the **Model Builder** window, 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(0,0.01*tFinal,0.1*tFinal) range(0.12*tFinal,0.02*tFinal,tFinal)`.

SOLID MECHANICS (SOLID)

Plasticity 2

1 In the **Model Builder** window, under **Component 1 (comp1)** > **Solid Mechanics (solid)** > **Linear Elastic Material 1** right-click **Plasticity 1** and choose **Duplicate**.

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

3 Find the **Thermal softening model** subsection. From the list, choose **No thermal softening**.

STUDY 1: WITH THERMAL SOFTENING

Step 1: Time Dependent

1 In the **Model Builder** window, under **Study 1: With Thermal Softening** 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** > **Plasticity 2**.

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

STUDY 2: WITHOUT THERMAL SOFTENING

Solution 7 (sol7)

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

2 In the **Model Builder** window, expand the **Solution 7 (sol7)** node, then click **Time-Dependent Solver 1**.

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

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

5 In the **Model Builder** window, click **Study 2: Without Thermal Softening**.

6 In the **Settings** window for **Study**, locate the **Study Settings** section.

7 Clear the **Generate default plots** checkbox.

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

RESULTS

Point Graph 2

1 In the **Model Builder** window, under **Results** > **Stress vs. Strain** 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 **Study 2: Without Thermal Softening/ Parametric Solutions 2 (sol8)**.

4 Locate the **Coloring and Style** section. From the **Color** list, choose **Cycle (reset)**.

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

6 From the **Positioning** list, choose **Interpolated**.

7 Locate the **Legends** section. Find the **Prefix and suffix** subsection. In the **Suffix** text field, type `, No temperature softening`.

8 In the **Stress vs. Strain** toolbar, click **Plot**.

RESULT TEMPLATES

Create a plot comparing the distribution of plastic strains at the end of the process for all four strain rates.

1 In the **Results** toolbar, click **Result Templates** to open the **Result Templates** window.

2 Go to the **Result Templates** window.

3 In the tree, select **Study 1: With Thermal Softening/Parametric Solutions 1 (sol2)** > **Solid Mechanics** > **Equivalent Plastic Strain (solid)**.

4 Click the **Add Result Template** button in the window toolbar.

5 In the **Results** toolbar, click **Result Templates** to close the **Result Templates** window.

RESULTS

Equivalent Plastic Strain (solid)

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

2 From the **Parameter value (strainRate (1/s))** list, choose **0.01**.

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

4 In the **Title** text area, type `Equivalent plastic strain`.

5 Clear the **Parameter indicator** text field.

6 Click to expand the **Plot Array** section. From the **Array type** list, choose **Linear**.

7 In the **Relative padding** text field, type `1`.

Max/Min Surface 1

In the **Equivalent Plastic Strain (solid)** toolbar, click **More Plots** and choose **Max/Min Surface**.

Max/Min Surface 1

1 In the **Model Builder** window, expand the **Results** > **Equivalent Plastic Strain (solid)** node, then click **Max/Min Surface 1**.

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

3 In the **Expression** text field, type `solid.epeGp`.

4 Click to expand the **Advanced** section. Locate the **Display** section. From the **Display** list, choose **Max**.

5 Locate the **Text Format** section. In the **Precision** text field, type `3`.

6 Click to expand the **Plot Array** section. Select the **Manual indexing** checkbox.

Annotation 1

1 In the **Model Builder** window, right-click **Equivalent Plastic Strain (solid)** and choose **Annotation**.

2 In the **Settings** window for **Annotation**, locate the **Position** section.

3 In the **Z** text field, type `totL/2`.

4 Locate the **Annotation** section. From the **Geometry level** list, choose **Global**.

5 In the **Text** text field, type `$\dot \varepsilon = eval(strainRate,1/s,1) \; \mathrm s^{-1}$`.

6 Select the **LaTeX markup** checkbox.

7 Locate the **Coloring and Style** section. Clear the **Show point** checkbox.

8 Click to expand the **Advanced** section. In the **Precision** text field, type `2`.

9 Locate the **Coloring and Style** section. From the **Anchor point** list, choose **Lower left**.

10 Click to expand the **Plot Array** section. Select the **Manual indexing** checkbox.

11 In the **Equivalent Plastic Strain (solid)** toolbar, click **Plot**.

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

Annotation 1, Max/Min Surface 1, Surface 1

1 In the **Model Builder** window, under **Results** > **Equivalent Plastic Strain (solid)**, Ctrl-click to select **Surface 1**, **Max/Min Surface 1**, and **Annotation 1**.

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

Surface 2

1 In the **Settings** window for **Surface**, locate the **Data** section.

2 From the **Dataset** list, choose **Study 1: With Thermal Softening/ Parametric Solutions 1 (sol2)**.

3 From the **Parameter value (strainRate (1/s))** list, choose **0.1**.

4 Click to expand the **Inherit Style** section. From the **Plot** list, choose **Surface 1**.

Max/Min Surface 2

1 In the **Model Builder** window, click **Max/Min Surface 2**.

2 In the **Settings** window for **Max/Min Surface**, locate the **Data** section.

3 From the **Dataset** list, choose **Study 1: With Thermal Softening/ Parametric Solutions 1 (sol2)**.

4 From the **Parameter value (strainRate (1/s))** list, choose **0.1**.

5 Locate the **Plot Array** section. In the **Index** text field, type `1`.

Annotation 2

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

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

3 From the **Dataset** list, choose **Study 1: With Thermal Softening/ Parametric Solutions 1 (sol2)**.

4 From the **Parameter value (strainRate (1/s))** list, choose **0.1**.

5 Locate the **Plot Array** section. In the **Index** text field, type 1.

Annotation 2, Max/Min Surface 2, Surface 2

1 In the **Model Builder** window, under **Results** > **Equivalent Plastic Strain (solid)**, Ctrl-click to select **Surface 2**, **Max/Min Surface 2**, and **Annotation 2**.

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

Surface 3

1 In the **Settings** window for **Surface**, locate the **Data** section.

2 From the **Parameter value (strainRate (1/s))** list, choose **1**.

Max/Min Surface 3

1 In the **Model Builder** window, expand the **Surface 3** node, then click **Results** > **Equivalent Plastic Strain (solid)** > **Max/Min Surface 3**.

2 In the **Settings** window for **Max/Min Surface**, locate the **Data** section.

3 From the **Parameter value (strainRate (1/s))** list, choose **1**.

4 Locate the **Coloring and Style** section. From the **Anchor point** list, choose **Upper right**.

5 Locate the **Plot Array** section. In the **Index** text field, type 2.

Annotation 3

1 In the **Model Builder** window, expand the **Max/Min Surface 3** node, then click **Results** > **Equivalent Plastic Strain (solid)** > **Annotation 3**.

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

3 From the **Parameter value (strainRate (1/s))** list, choose **1**.

4 Locate the **Plot Array** section. In the **Index** text field, type 2.

Annotation 3, Max/Min Surface 3, Surface 3

1 In the **Model Builder** window, under **Results** > **Equivalent Plastic Strain (solid)**, Ctrl-click to select **Surface 3**, **Max/Min Surface 3**, and **Annotation 3**.

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

Annotation 4, Surface 4

1 In the **Settings** window for **Surface**, locate the **Data** section.

2 From the **Parameter value (strainRate (1/s))** list, choose **10**.

Max/Min Surface 4

1 In the **Model Builder** window, expand the **Results** > **Equivalent Plastic Strain (solid)** > **Surface 4** node, then click **Results** > **Equivalent Plastic Strain (solid)** > **Max/Min Surface 4**.

2 In the **Settings** window for **Max/Min Surface**, locate the **Data** section.

3 From the **Parameter value (strainRate (1/s))** list, choose **10**.

4 Locate the **Plot Array** section. In the **Index** text field, type `3`.

Annotation 4

1 In the **Model Builder** window, expand the **Max/Min Surface 4** node, then click **Results** > **Equivalent Plastic Strain (solid)** > **Annotation 4**.

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

3 From the **Parameter value (strainRate (1/s))** list, choose **10**.

4 Locate the **Plot Array** section. In the **Index** text field, type `3`.

5 Locate the **Annotation** section. In the **Text** text field, type `$\dot \varepsilon = eval(strainRate) \;  \mathrm s^{-1}$`.

Equivalent Plastic Strain (solid)

1 In the **Model Builder** window, click **Equivalent Plastic Strain (solid)**.

2 In the **Equivalent Plastic Strain (solid)** toolbar, click **Plot**.

3 Click the **Show Grid** button in the **Graphics** toolbar.

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

Temperature Increase

Now, compare also the final temperature increase.

1 Right-click **Equivalent Plastic Strain (solid)** and choose **Duplicate**.

2 In the **Model Builder** window, click **Equivalent Plastic Strain (solid) 1**.

3 In the **Settings** window for **2D Plot Group**, type `Temperature Increase` in the **Label** text field.

4 Locate the **Title** section. In the **Title** text area, type `Temperature increase`.

5 Locate the **Color Legend** section. Select the **Show units** checkbox.

Surface 1

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

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

3 In the **Expression** text field, type `T-293.15`.

4 Locate the **Coloring and Style** section. From the **Color table** list, choose **Thermal**.

Surface 2

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

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

3 In the **Expression** text field, type `T-minput.Tempref`.

Surface 3

1 In the **Model Builder** window, click **Surface 3**.

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

3 In the **Expression** text field, type `T-minput.Tempref`.

Surface 4

1 In the **Model Builder** window, click **Surface 4**.

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

3 In the **Expression** text field, type `T-minput.Tempref`.

Max/Min Surface 1

1 In the **Model Builder** window, click **Max/Min Surface 1**.

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

3 In the **Expression** text field, type `T-minput.Tempref`.

Max/Min Surface 2

1 In the **Model Builder** window, click **Max/Min Surface 2**.

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

3 In the **Expression** text field, type `T-minput.Tempref`.

Max/Min Surface 3

1 In the **Model Builder** window, click **Max/Min Surface 3**.

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

3 In the **Expression** text field, type `T-minput.Tempref`.

Max/Min Surface 4

1 In the **Model Builder** window, click **Max/Min Surface 4**.

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

3 In the **Expression** text field, type `T-minput.Tempref`.

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

5 In the **Temperature Increase** toolbar, click **Plot**.

MAXIMUM AND MINIMUM VALUES 7

Go to the **Maximum and Minimum Values 7** window.