



# 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  $\sigma_{ys}$  is in this model described by [Equation 1](#).

$$\sigma_{ys}(\varepsilon_{pe}) = (\sigma_{ys0} + k\varepsilon_{pe}^n) \left( 1 + C \max \left( 0, \log \left( \frac{\dot{\varepsilon}_{pe}}{\dot{\varepsilon}_0} \right) \right) \right) (1 - f(T_h)) \quad (1)$$

Here,  $\sigma_{ys0}$ ,  $k$ ,  $n$ ,  $C$ , 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

In the model, four different strain rates are investigated. Two different studies are performed. 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.

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.

## 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](#).

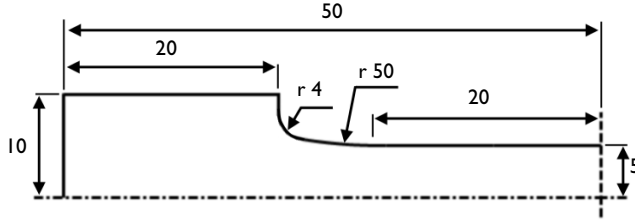


Figure 1: Geometry of the test specimen.

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	$\nu$	0.3
Coefficient of thermal expansion	$\alpha$	$12.3 \cdot 10^{-6} 1/K$
Initial yield strength	$\sigma_{ys0}$	400 MPa
Strength coefficient	$k$	200 MPa
Hardening exponent	$n$	0.5
Reference strain rate	$\dot{\epsilon}_0$	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	$\rho$	7850 kg/m <sup>3</sup>
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  $15 \text{ W}/(\text{m}^2 \cdot \text{K})$ .

## 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:

$$q = \sigma : \dot{\epsilon}_{pl} \quad (2)$$

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 the thermal strain and the plastic strain can be estimated if the heat produced by plastic dissipation is assumed not to be conducted away from where it is generated:

$$\frac{\epsilon_{th}}{\epsilon_{pl}} = \frac{\alpha \Delta T}{\epsilon_{pl}} = \frac{\alpha \sigma_y \epsilon_{pl}}{\epsilon_{pl} \rho c_p} = \frac{\alpha \sigma_y}{\rho c_p} = \frac{12.3 \cdot 10^{-6} \cdot 4 \cdot 10^8}{7850 \cdot 475} \approx 1.3 \cdot 10^{-3} \quad (3)$$

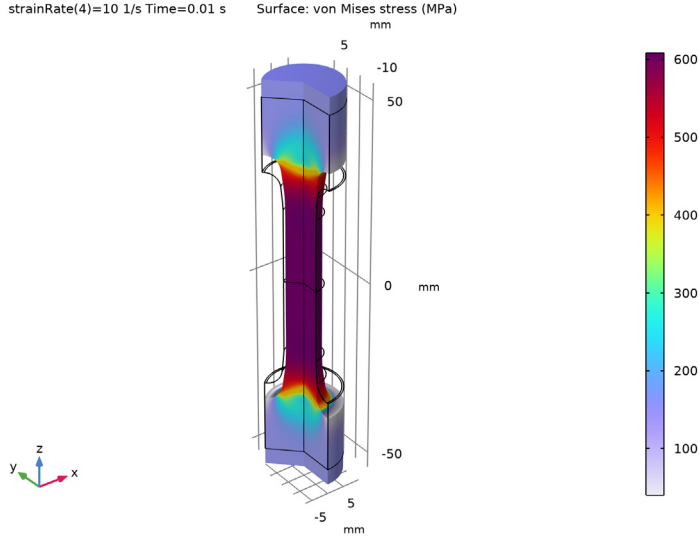
## *Results and Discussion*

---

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

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

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.

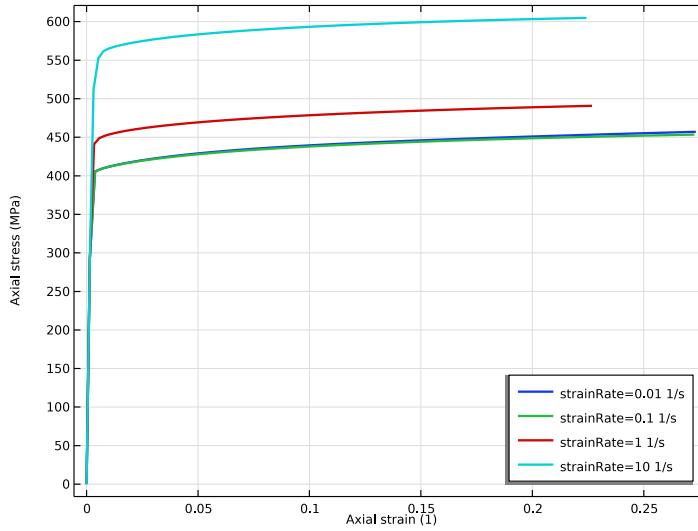


Figure 3: Axial stress and strain at the center of the test specimen for the four tensile tests at different strain rates.

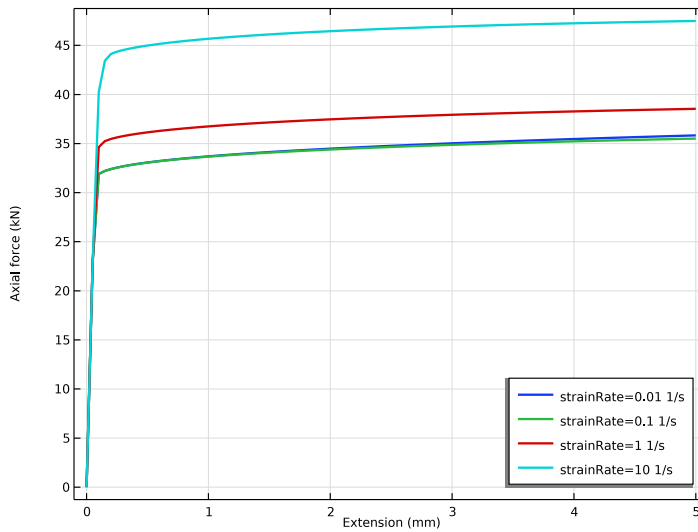


Figure 4: Force versus displacement for the four tensile tests at different strain rates.

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.

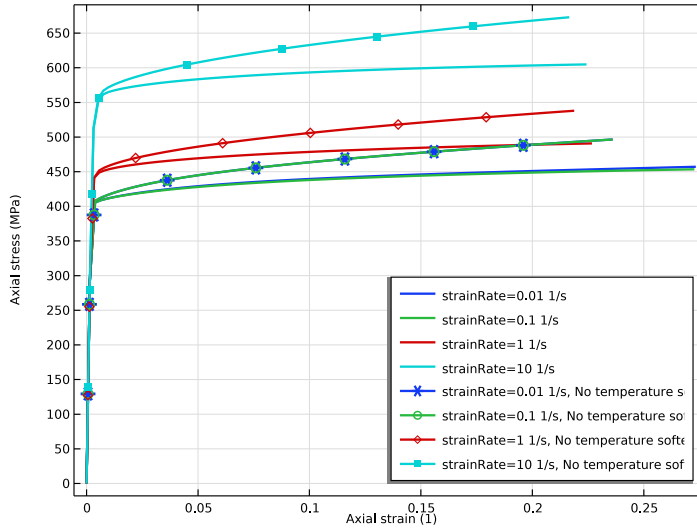


Figure 5: Comparison of stress vs. strain with and without thermal softening.

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.

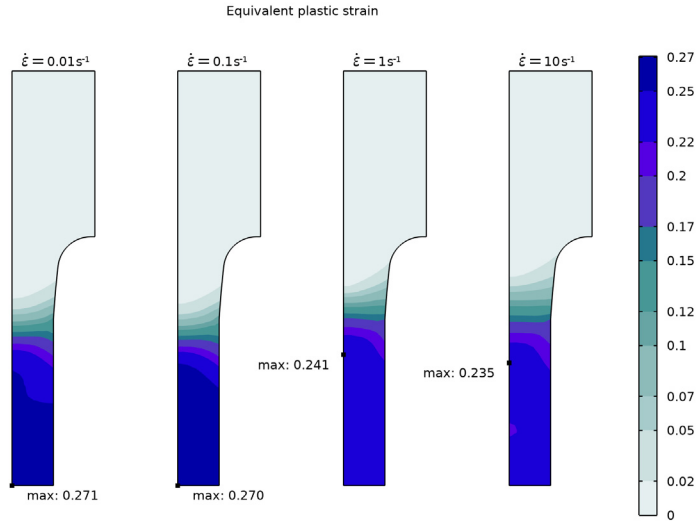


Figure 6: Distribution of plastic strain at the end of the process for all four strain rates.

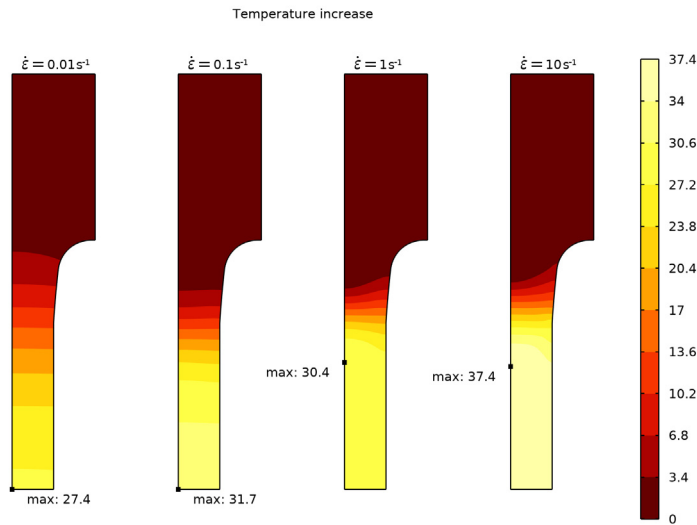


Figure 7: Temperature distribution at the end of the process for all four strain rates.



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 I

- 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 **csoll**, 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 I*

- 1 In the **Model Builder** window, under **Component 1 (comp1)** right-click **Solid Mechanics (solid)** and choose **More Constraints>Symmetry Plane**.
- 2 Select Boundary 2 only.


### *Prescribed Displacement I*

- 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  $w_{\text{Max}}*t/t_{\text{Final}}$ .

### *Linear Elastic Material I*

In the **Model Builder** window, click **Linear Elastic Material 1**.

### *Plasticity I*

- 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	I	Young's modulus and Poisson's ratio
Density	rho	7850	kg/m <sup>3</sup>	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	I	Johnson-Cook
Reference strain rate	epet0_jcook	1	I/s	Johnson-Cook
Strain rate strength coefficient	C_jcook	0.12	I	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 ; alphasii = alpha_iso, alphasij = 0	12.3e-6 [ 1 / K ]	I/K	Basic

## HEAT TRANSFER IN SOLIDS (HT)

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

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

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 box, in the tree, select the check box 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 Select the **Calculate dissipated energy** check box.

## 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  $\text{if}(z < 30[\text{mm}], 1[\text{mm}], 3[\text{mm}])$ .
- 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

#### 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 I (sol1)

- 1 In the **Study** toolbar, click  **Show Default Solver**.
- 2 In the **Model Builder** window, expand the **Solution I (sol1)** node, then click **Time-Dependent Solver I**.
- 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**.

#### *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 vs. strain at the center of the specimen for all strain rates.


- 1 In the **Home** toolbar, click  **Add Plot Group** and choose **ID Plot Group**.
- 2 In the **Settings** window for **ID 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** check box.
- 11 Find the **Include** subsection. Clear the **Point** check box.

#### *Stress vs. Strain*


- 1 In the **Model Builder** window, click **Stress vs. Strain**.
- 2 In the **Settings** window for **ID 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** check box. In the associated text field, type **Axial strain (1)**.
- 6 Select the **y-axis label** check box. 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 **Home** toolbar, click  **Add Plot Group** and choose **ID Plot Group**.
- 2 In the **Settings** window for **ID 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 I*

- 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  $w_{Max} * t / t_{Final}$ .
- 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** check box. In the associated text field, type **Extension**.
- 10 Click to expand the **Legends** section. Find the **Include** subsection. Clear the **Description** check box.
- 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 **ID 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

*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 1*

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

*Plasticity 2*

- 1 In the **Model Builder** window, click **Plasticity 2**.
- 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** check box.

- 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** check box.
- 8 In the **Study** toolbar, click  **Compute**.

## RESULTS

### *Point Graph 1*



In the **Model Builder** window, under **Results>Stress vs. Strain** right-click **Point Graph 1** and choose **Duplicate**.

### *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 **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**.

## ADD PREDEFINED PLOT

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


- 1 In the **Home** toolbar, click  **Add Predefined Plot** to open the **Add Predefined Plot** window.
- 2 Go to the **Add Predefined Plot** window.
- 3 In the tree, select **Study 1: With Thermal Softening/Parametric Solutions 1 (sol2)>Solid Mechanics>Equivalent Plastic Strain (solid)**.
- 4 Click **Add Plot** in the window toolbar.
- 5 In the **Home** toolbar, click  **Add Predefined Plot** to close the **Add Predefined Plot** 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. Select the **Enable** check box.
- 7 In the **Relative padding** text field, type 1.
- 8 In the **Model Builder** window, click **Equivalent Plastic Strain (solid)**.

### *Max/Min Surface 1*

- 1 In the **Equivalent Plastic Strain (solid)** toolbar, click  **More Plots** and choose **Max/Min Surface**.
- 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 **Display precision** text field, type 3.
- 6 Click to expand the **Plot Array** section. Select the **Manual indexing** check box.

#### *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  $\text{totL}/2$ .
- 4 Locate the **Annotation** section. From the **Geometry level** list, choose **Global**.
- 5 In the **Text** text field, type  $\dot{\epsilon} = \text{eval}(\text{strainRate}, 1/\text{s}, 1) \backslash; \backslash \mathrm{s}^{-1} \backslash$ .
- 6 Select the **LaTeX markup** check box.
- 7 Locate the **Coloring and Style** section. Clear the **Show point** check box.
- 8 Click to expand the **Advanced** section. In the **Expression 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** check box.
- 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 **Model Builder** window, click **Surface 2**.
- 2 In the **Settings** window for **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 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 **Model Builder** window, click **Surface 3**.
- 2 In the **Settings** window for **Surface**, locate the **Data** section.
- 3 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 **Model Builder** window, under **Results>Equivalent Plastic Strain (solid)**, Ctrl-click to select **Surface 4** and **Annotation 4**.
- 2 In the **Settings** window for **Surface**, locate the **Data** section.
- 3 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  $\dot{\epsilon} = \text{eval}(\text{strainRate}) \cdot \text{strainRate}^{-1}$ .

#### *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.

Now, compare also the final temperature increase.


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

#### *Temperature Increase*

- 1 In the **Model Builder** window, expand the **Results>Equivalent Plastic Strain (solid) I** node, then click **Equivalent Plastic Strain (solid) I**.
- 2 In the **Settings** window for **2D Plot Group**, type Temperature Increase in the **Label** text field.

- 3 Locate the **Title** section. In the **Title** text area, type Temperature increase.

#### *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. Click  **Change Color Table**.
- 5 In the **Color Table** dialog box, select **Thermal>Thermal** in the tree.
- 6 Click **OK**.

#### *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  $T - \text{minput} . \text{Tempref}$ .

#### *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  $T - \text{minput} . \text{Tempref}$ .

#### *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  $T - \text{minput} . \text{Tempref}$ .

#### *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  $T - \text{minput} . \text{Tempref}$ .

#### *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  $T - \text{minput} . \text{Tempref}$ .

#### *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**.

