Created in COMSOL Multiphysics 5.6

Necking of an Elastoplastic Metal Bar

Introduction

A circular bar of metal is subjected to a uniaxial tensile test. The the material presents an elastoplastic behavior with nonlinear isotropic hardening. When subjected to large deformations, the specimen experiences a significant plastic deformation and necking in its central cross section. This example demonstrates the large strain plasticity option available in the Nonlinear Structural Materials Module. The simulation results are compared with results in the literature.

Model Definition

In this model, a cylindrical steel bar with a height, H_0, of 53.334 mm and a radius, R_0, of 6.413 mm, is subjected to a total elongation of 14 mm.

The problem exhibits 2D axial symmetry as well as a reflection symmetry in the mid cross section of the bar. It is therefore possible to reduce the model geometry to a rectangle with a width equal to the radius of the bar, and a height equal to half of the length of the bar, see Figure 1.

The boundary conditions for the displacements in the radial and axial directions, u and w, that follow from the symmetries are

$$\begin{aligned} w(r, 0) &= 0 \\ u(0, z) &= 0 \end{aligned}$$

where r and z are the radial and axial coordinates. The tensile loadingis imposed through a prescribed elongation of $\Delta = 7$ mm,

$$\begin{aligned} w(r, H_0/2) &= \Delta \\ u(r, H_0/2) &= 0 \end{aligned}$$

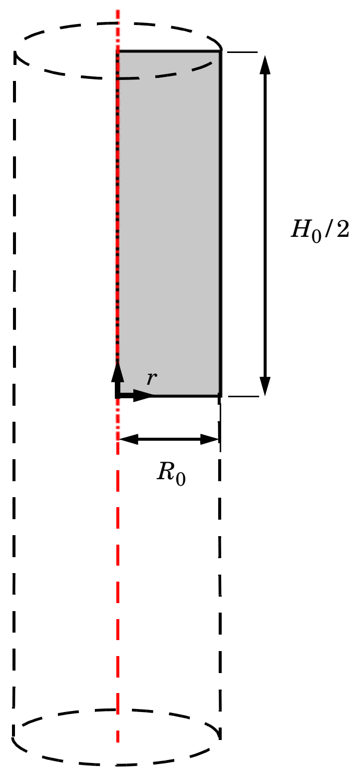

Figure 1: Schematic description of the numerical model. The red line shows the rotation symmetry axis.

MATERIAL MODEL

The elastic behavior of the material is characterized by a Young's modulus $E = 206.9$ GPa and a Poisson's ratio $\nu = 0.29$. The plastic response follows a nonlinear isotropic hardening model with a yield stress given by

$$\sigma_{\mathrm{ys}} = \sigma_{\mathrm{ys0}} + \sigma_{\mathrm{h}} \tag{1}$$

where σ_{ys0} is the initial yield stress and σ_{h} is the nonlinear hardening function. The latter is defined as

$$\sigma_{\mathrm{h}}(\varepsilon_{\mathrm{pe}}) = H\varepsilon_{\mathrm{pe}} + (\sigma_{\mathrm{ysf}} - \sigma_{\mathrm{ys0}})[1 - e^{-\zeta\varepsilon_{pe}}] \tag{2}$$

The hardening function σ_h depends nonlinearly on the equivalent plastic strain ε_{pe}. Here, H is the linear hardening coefficient, σ_{ysf} is the saturation flow stress or residual yield stress, and ζ is the saturation exponent.

The numerical values of the parameters for the hardening function are given in Table 1. Figure 2, shows this nonlinear hardening as a function of equivalent plastic strain.

TABLE 1: HARDENING FUNCTION CONSTANTS.

CONSTANT	VALUE
σ_{ys0}	450 MPa
H	129,24 MPa
σ_{ysf}	715 MPa
ζ	16.93

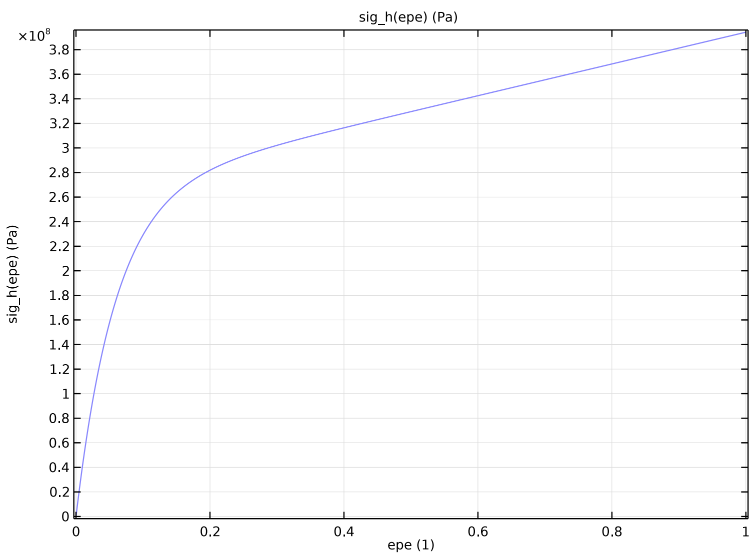

Figure 2: Nonlinear isotropic hardening as a function of equivalent plastic strain.

Results and Discussion

Figure 3 shows the distribution of the von Mises stress in the specimen. The figure also shows the deformation of the specimen, and how the nonlinear material behavior causes necking of its central cross section. The necking is caused by the very large plastic strains at the vertical symmetry plane as seen in Figure 4. A 7 mm elongation results in an equivalent plastic strain larger than 1.6 (this is, 160%) in the most affected region.

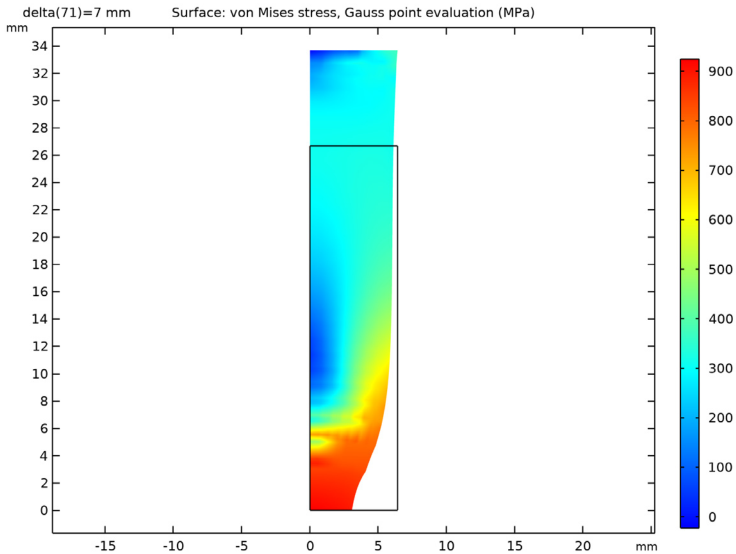

Figure 3: Distribution of the von Mises stress at 7 mm end displacement.

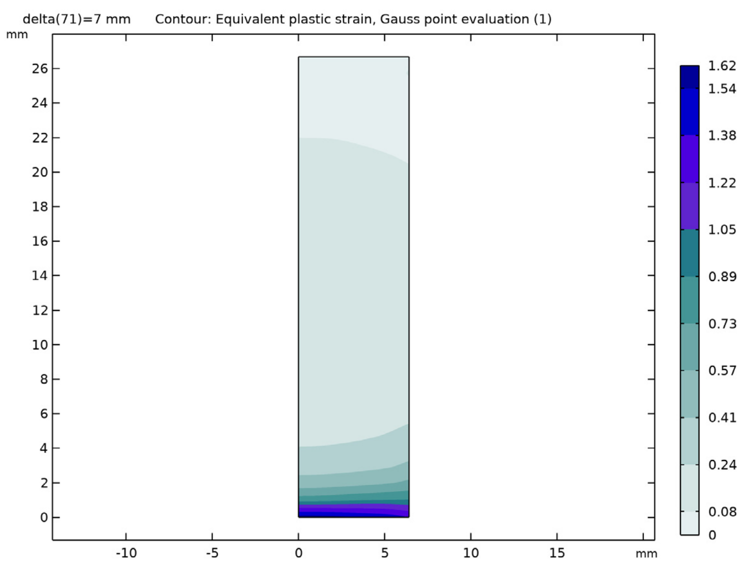

Figure 4: Distribution of the equivalent plastic strain at 7 mm end displacement. The results are shown in the undeformed configuration of the bar.

Figure 5 shows the change in radius as a function the elongation. Initially, the radius decreases linearly with the applied displacement. After an axial displacement of 3 mm, the radial reduction increases significantly and the specimen experiences necking. A similar case has been examined by Simo and Hughes (Ref. 1) as well as by Elguedj and Hughes (Ref. 2). Table 2 compares their results with the outcome of the current analysis. The calculated radial neck radius is in good agreement with results found in the literature.

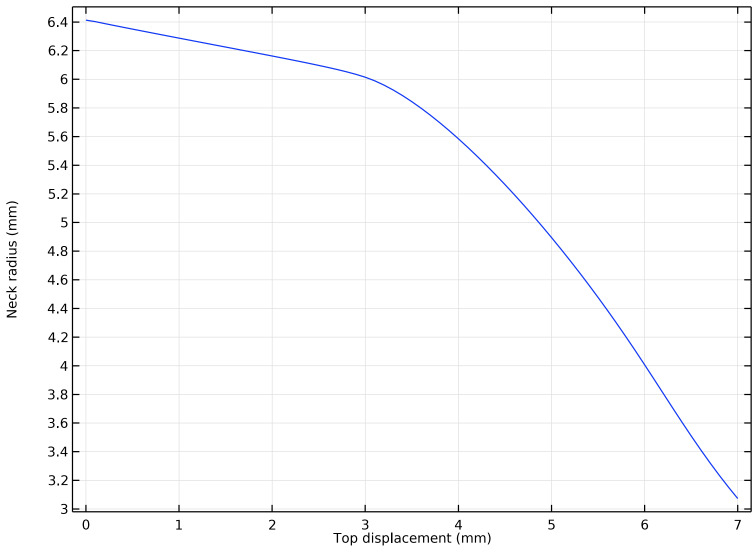

Figure 5: Necking development in the mid section of the bar.

TABLE 2: COMPARISON BETWEEN APPLIED DISPLACEMENT AND BAR RADIUS (MM) AT THE MID SECTION.

δ (mm)	COMSOL	Ref. 1	Ref. 2
1.0	6.3	6.3	6.3
2.0	6.2	6.1	6.1
3.0	6.0	5.9	5.9
4.0	5.6	5.3	5.4
5.0	4.9	4.6	4.6
6.0	4.0	3.7	3.7

Notes About the COMSOL Implementation

The nonlinear hardening behavior is implemented using an analytic function. With reference to equation Equation 2, the hardening function is defined as

$$\sigma_{\mathrm{h}}(\varepsilon_{\mathrm{pe}}) = H\varepsilon_{\mathrm{pe}} + (\sigma_{\mathrm{ysf}} - \sigma_{\mathrm{ys0}})[1 - e^{-\zeta\varepsilon_{pe}}]$$

The user-defined hardening $\sigma_{\mathrm{h}}(\varepsilon_{\mathrm{pe}})$, portrayed in Figure 2, assumes zero stress for no plastic strain. In Equation 1, the yield stress is defined as the sum of the initial yield stress σ_{ys0} and the hardening function $\sigma_{\mathrm{h}}(\varepsilon_{\mathrm{pe}})$. The hardening function is defined in the **Materials** node by an analytic function.

You find the **Large plastic strains** option in the **Plasticity** node. This option uses multiplicative decomposition between the elastic and plastic deformations, as opposed to additive decomposition which is used for the **Small plastic strains** option. The small plastic strain assumption is generally not valid for strains above 0.1 (this is, more than 10%).

In this example, you use the double dogleg solver. This solver often works better for this class of nonlinear problems. The default Newton solver could also be used, but would then require tuning of the default solver settings.

References

1. J.C. Simo and T.R.J. Hughes, *Computational Inelasticity*, Springer, 2000.

2. T. Elguedj and T.J.R. Hughes, *Isogeometric Analysis of Nearly Incompressible Large Strain Plasticity*, ICES REPORT 11-35, the Institute for Computational Engineering and Sciences, the University of Texas at Austin, 2011.

Application Library path: `Nonlinear_Structural_Materials_Module/Plasticity/bar_necking`

Modeling Instructions

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

NEW

In the **New** window, click **Model Wizard**.

MODEL WIZARD

1 In the **Model Wizard** window, click **2D Axisymmetric**.
2 In the **Select Physics** tree, select **Structural Mechanics>Solid Mechanics (solid)**.
3 Click **Add**.
4 Click **Study**.
5 In the **Select Study** tree, select **General Studies>Stationary**.
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
sigma0	450[MPa]	4.5E8 Pa	Initial yield stress
sigmaSF	715[MPa]	7.15E8 Pa	Saturation flow stress
H	129.24[MPa]	1.2924E8 Pa	Linear hardening coefficient
zeta	16.93	16.93	Saturation exponent
delta	0[m]	0 m	Top displacement
H0	53.334[mm]	0.053334 m	Bar length
R0	6.413[mm]	0.006413 m	Bar radius

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**.

Rectangle 1 (r1)

1 In the **Geometry** toolbar, click **Rectangle**.
2 In the **Settings** window for **Rectangle**, locate the **Size and Shape** section.
3 In the **Width** text field, type `R0`.
4 In the **Height** text field, type `H0/2`.
5 Click **Build All Objects**.

SOLID MECHANICS (SOLID)

Prescribed Displacement, Bottom

1 In the **Model Builder** window, under **Component 1 (comp1)** right-click **Solid Mechanics (solid)** and choose **Prescribed Displacement**.

2 In the **Settings** window for **Prescribed Displacement**, type `Prescribed Displacement, Bottom` in the **Label** text field.

3 Select Boundary 2 only.

4 Locate the **Prescribed Displacement** section. Select the **Prescribed in z direction** check box.

Prescribed Displacement, Top

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

2 In the **Settings** window for **Prescribed Displacement**, type `Prescribed Displacement, Top` in the **Label** text field.

3 Select Boundary 3 only.

4 Locate the **Prescribed Displacement** section. Select the **Prescribed in r direction** check box.

5 Select the **Prescribed in z direction** check box.

6 In the u_{0z} text field, type `delta`.

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 From the **Plasticity model** list, choose **Large plastic strains**.

4 Find the **Isotropic hardening model** subsection. From the list, choose **Hardening function**.

MATERIALS

Material 1 (mat1)

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

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

3 In the table, enter the following settings:

Property	Variable	Value	Unit	Property group
Young's modulus	E	206.9[GPa]	Pa	Basic
Poisson's ratio	nu	0.29	1	Basic
Density	rho	7850	kg/m³	Basic
Initial yield stress	sigmags	sigma0	Pa	Elastoplastic material model

Add a nonlinear hardening function.

Analytic 1 (an1)

1. In the **Model Builder** window, expand the **Material 1 (mat1)** node.
2. Right-click **Elastoplastic material model (ElastoplasticModel)** and choose **Functions>Analytic**.
3. In the **Settings** window for **Analytic**, type `sig_h` in the **Function name** text field.
4. Locate the **Definition** section. In the **Arguments** text field, type `epe`.
5. In the **Expression** text field, type `H*epe+(sigmaSF-sigma0)*(1-exp(-zeta*epe))`.
6. Locate the **Units** section. In the **Arguments** text field, type `1`.
7. In the **Function** text field, type `Pa`.
8. Locate the **Plot Parameters** section. In the table, enter the following settings:

Argument	Lower limit	Upper limit
epe	0	1

9. Click **Plot**.

Material 1 (mat1)

1. In the **Model Builder** window, click **Elastoplastic material model (ElastoplasticModel)**.
2. In the **Settings** window for **Property Group**, locate the **Model Inputs** section.
3. Click **Select Quantity**.
4. In the **Physical Quantity** dialog box, type `plastic strain` in the text field.
5. Click **Filter**.
6. In the tree, select **Solid Mechanics>Equivalent plastic strain (1)**.
7. Click **OK**.

8 In the **Model Builder** window, click **Material 1 (mat1)**.

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

10 In the table, enter the following settings:

Property	Variable	Value	Unit	Property group
Hardening function	sigmagh	`sig_h(epe)`	Pa	Elastoplastic material model

MESH 1

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

2 In the **Settings** window for **Mesh**, locate the **Physics-Controlled Mesh** section.

3 From the **Element size** list, choose **Finer**.

Mapped 1

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

2 In the **Settings** window for **Mapped**, click **Build All**.

The mesh should consist of about 196 elements.

STUDY 1

Solution 1 (sol1)

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

2 In the **Model Builder** window, expand the **Solution 1 (sol1)** node.

3 In the **Model Builder** window, expand the **Study 1>Solver Configurations>Solution 1 (sol1)>Stationary Solver 1** node, then click **Fully Coupled 1**.

4 In the **Settings** window for **Fully Coupled**, click to expand the **Method and Termination** section.

5 From the **Nonlinear method** list, choose **Double dogleg**.

The Double dogleg solver is suitable for highly nonlinear problems.

Step 1: Stationary

Set up an auxiliary continuation sweep for the `delta` parameter.

1 In the **Model Builder** window, click **Step 1: Stationary**.

2 In the **Settings** window for **Stationary**, click to expand the **Study Extensions** section.

3 Select the **Auxiliary sweep** check box.

4 Click **Add**.

5 In the table, enter the following settings:

Parameter name	Parameter value list	Parameter unit
delta (Top displacement)	`range(0.0, 0.1, 7)`	`mm`

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

RESULTS

Surface 1

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

Neck Radius

1 In the **Home** toolbar, click **Add Plot Group** and choose **1D Plot Group**.

2 In the **Settings** window for **1D Plot Group**, type `Neck Radius` in the **Label** text field.

Point Graph 1

1 Right-click **Neck Radius** and choose **Point Graph**.

2 Select Point 3 only.

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

4 In the **Expression** text field, type `u+R0`.

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

6 In the **Expression** text field, type `delta`.

7 In the **Neck Radius** toolbar, click **Plot**.

8 Locate the **y-Axis Data** section. Select the **Description** check box.

9 In the associated text field, type `Neck radius`.

10 Click to expand the **Title** section. From the **Title type** list, choose **None**.

11 In the **Neck Radius** toolbar, click **Plot**.

Calculate the maximum equivalent plastic strain.

Surface Maximum 1

1 In the **Results** toolbar, click **More Derived Values** and choose **Maximum> Surface Maximum**.

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

3 From the **Parameter selection (delta)** list, choose **Last**.

4 Locate the **Selection** section. From the **Selection** list, choose **All domains**.

5 Click **Replace Expression** in the upper-right corner of the **Expressions** section. From the menu, choose **Component 1 (comp1)>Solid Mechanics>Strain (Gauss points)>solid.epeGp - Equivalent plastic strain, Gauss point evaluation**.

6 Click **Evaluate**.

Create an illustrative 3D plot.

Mirror 3D 1

1 In the **Model Builder** window, expand the **Results>Datasets** node.

2 Right-click **Datasets** and choose **More 3D Datasets>Mirror 3D**.

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

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

Stress, 3D (solid)

1 In the **Model Builder** window, 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 **Plot Settings** section. Clear the **Plot dataset edges** check box.

5 In the **Stress, 3D (solid)** toolbar, click **Plot**.