Necking of an Elastoplastic Metal Bar

Introduction

A circular bar of metal is subjected to a uniaxial tensile test. 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 formulation of plasticity 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 20 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

$$w(r, 0) = 0$$
$$u(0, z) = 0$$

where r and z are the radial and axial coordinates. The tensile loading is imposed through a prescribed elongation of $\Delta = 10$ mm,

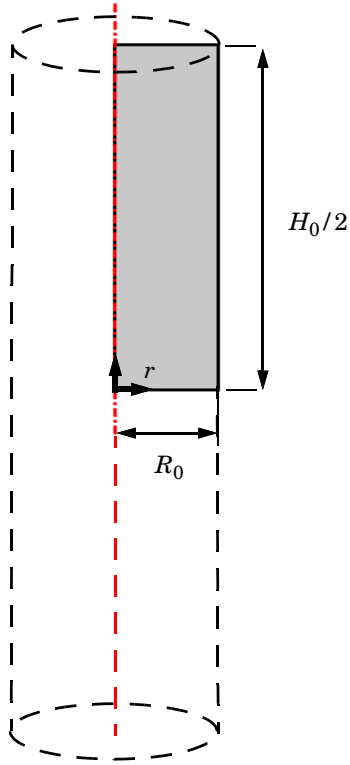$$w(r, H_0/2) = \Delta$$
$$u(r, H_0/2) = 0$$

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

$$\sigma_{ys} = \sigma_{ys0} + \sigma_h \tag{1}$$

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

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

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.

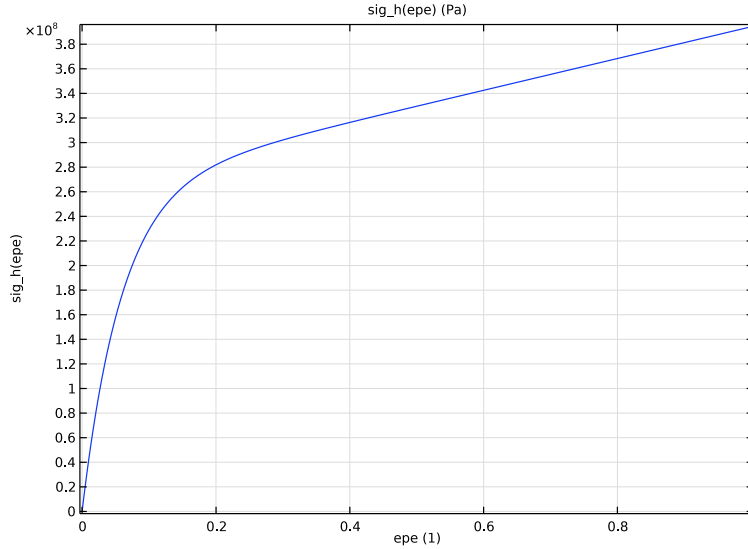| CONSTANT	VALUE
σ_{ys0}	450 MPa
H	129.24 MPa
σ_{ysf}	715 MPa
ζ	16.93

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

As plastic strain become large, the material becomes unstable which will result in localization of the deformation in to the smallest possible width; in the case of finite elements, rows of mesh element or rows of Gauss points. To keep the mesh objectivity of the solution, a length scale can be introduced to the material model to limit strain localization to a predefined width. Here the **Implicit gradient** nonlocal plasticity model is used with a length scale l_{int} = 0.15 mm.

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 10 mm elongation results in an equivalent plastic strain larger than 2.5 (this is, 250%) in the most affected region. At such large plastic strains, the specimen has likely already ruptured.
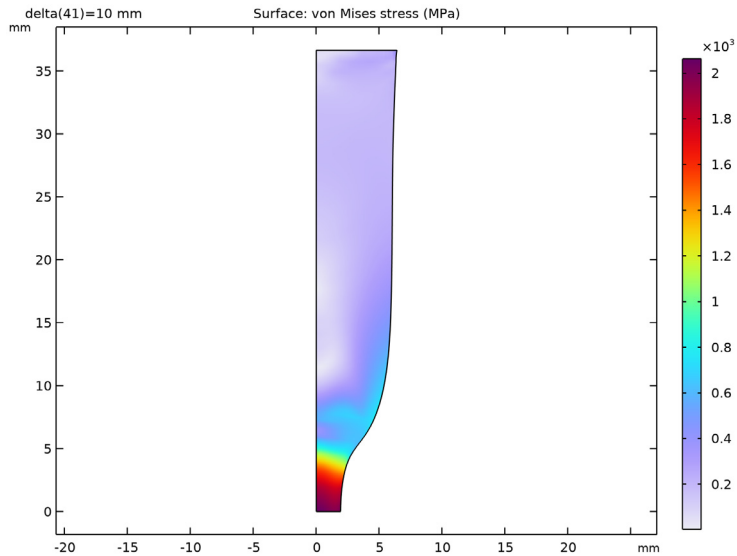

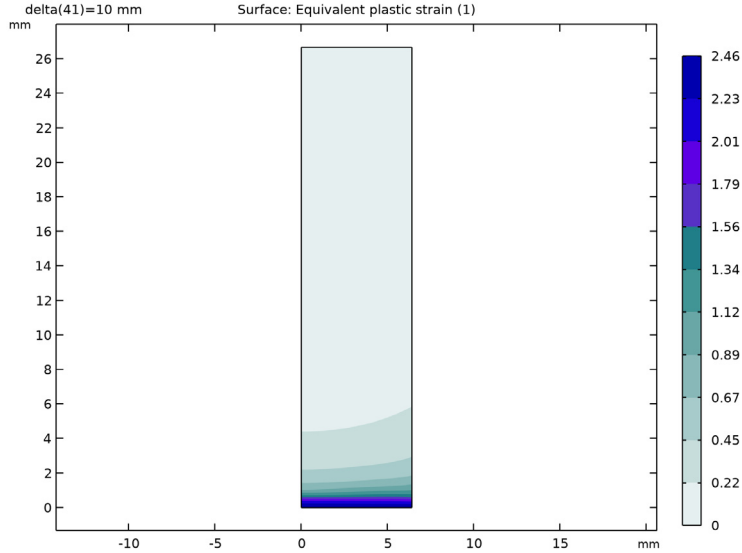Figure 3: Distribution of the von Mises stress at 10 mm end displacement.

Figure 4: Distribution of the equivalent plastic strain at 10 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.
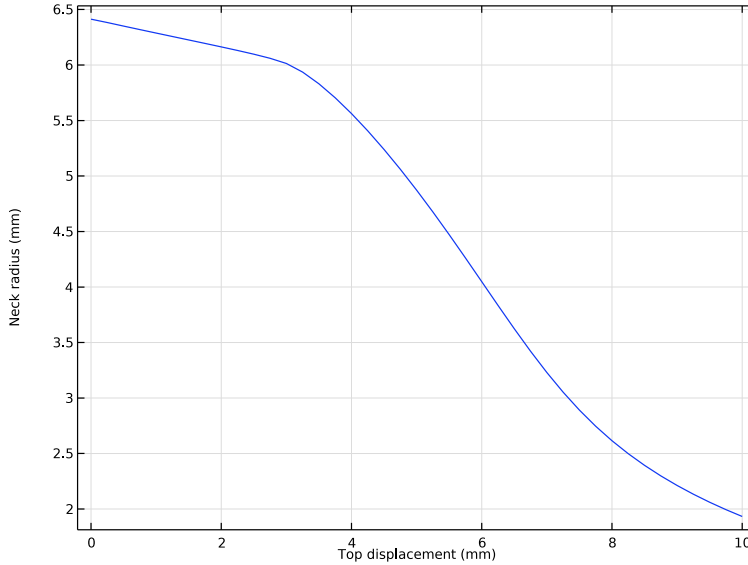
Figure 5: Necking development in the mid section of the bar.

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

Figure 6 shows the reaction force as function of the axial displacement. The peak load is recorded at 3 mm after which the load starts to decrease as the specimen experience necking as discussed above. At an axial displacement around 7 mm, the curve starts to flatten, this is an effect of the nonlocal plasticity model. However, at this point the specimen in reality would have ruptured, but to capture such behavior, the material model would have to be extended.
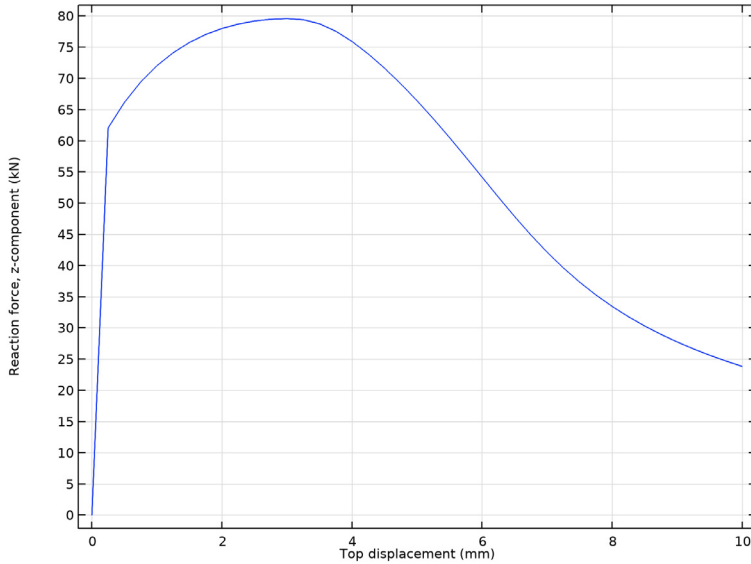
Figure 6: Reaction force of the bar.

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

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

The user-defined hardening $\sigma_h(\varepsilon_{pe})$, 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 $\sigma_h(\varepsilon_{pe})$. The hardening function is defined in the **Materials** node by an analytic function.

The large strain formulation of plasticity is used when the parent material model to he **Plasticity** node requests a multiplicative decomposition of deformation gradients. Selecting the **Total Lagrangian** formulation in the **Geometric Nonlinearity** section of **Linear Elastic Material** will typically enable the multiplicative decomposition. By also setting **Strain decomposition** to **Multiplicative**, one can force the multiplicative decomposition to be used always. As a rule-of-thumb, the multiplicative formulation is recommended when elastic and/or plastic strains exceed 0.1 (this is, more than 10%).

Including plasticity in the material model requires local computations at each Gauss point during the assembly process which is expensive. By using **Reduced Integration**, the number of Gauss points are reduced by a factor two for the given displacement shape function order and mesh element type. This will speed up the computation significantly.

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

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. From the **Displacement in r direction** list, choose **Prescribed**.

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

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

Linear Elastic Material 1

Use a large strain formulation with a multiplicative decomposition of deformation gradients.

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

2 In the **Settings** window for **Linear Elastic Material**, locate the **Geometric Nonlinearity** section.

3 From the **Formulation** list, choose **Total Lagrangian**.

4 From the **Strain decomposition** list, choose **Multiplicative**.

The local computations at Gauss points during assembly are expensive when plasticity is added to the material model. Using a reduced integration scheme will reduce the overall simulation time.

5 Locate the **Quadrature Settings** section. Select the **Reduced integration** check box.

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 **Hardening function**.

4 Click to expand the **Nonlocal Plasticity Model** section. From the list, choose **Implicit gradient**.

5 In the l_{int} text field, type 0.15[mm].

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	Young's modulus and Poisson's ratio
Poisson's ratio	nu	0.29	1	Young's modulus and Poisson's ratio
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 **Component 1 (comp1)>Materials>Material 1 (mat1)> 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 table, enter the following settings:

Argument	Unit
epe	1

7 In the **Function** text field, type Pa.

8 Locate the **Plot Parameters** section. In the table, enter the following settings:

Plot	Argument	Lower limit	Upper limit	Fixed value	Unit
√	epe	0	1	0	

9 Click ⊙ **Plot**.

Material 1 (mat1)

1 In the **Model Builder** window, under **Component 1 (comp1)>Materials>Material 1 (mat1)** click **Elastoplastic material model (ElastoplasticModel)**.

2 In the **Settings** window for **Elastoplastic Material Model**, 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

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 From the **Distribution type** list, choose **Predefined**.

5 In the **Number of elements** text field, type 50.

6 In the **Element ratio** text field, type 5.

7 Select Boundaries 1 and 4 only.

Distribution 2

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

2 Select Boundary 2 only.

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

4 In the **Number of elements** text field, type 20.

5 Click **Build All**.

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, under **Study 1** 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, 0.25, 10)	mm

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

RESULTS

Stress (solid)

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

2 From the **Frame** list, choose **Spatial (r, phi, z)**.

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**.

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

2 In the **Settings** window for **ID 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.

9 Select the **Description** check box. 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 reaction force.

Line Integration 1

1 In the **Results** toolbar, click ⁸·⁸⁵ₑ₋₁₂ **More Derived Values** and choose **Integration> Line Integration**.

2 Select Boundary 3 only.

3 In the **Settings** window for **Line Integration**, click **Replace Expression** in the upper-right corner of the **Expressions** section. From the menu, choose **Component 1 (comp1)> Solid Mechanics>Reactions>Reaction force (spatial frame) - N>solid.RFz - Reaction force, z-component**.

4 Locate the **Expressions** section. In the table, enter the following settings:

Expression	Unit	Description
solid.RFz	kN	Reaction force, z-component

5 Locate the **Integration Settings** section. Clear the **Compute surface integral** check box.

6 Click ▬ **Evaluate**.

TABLE I

1 Go to the **Table 1** window.

2 Click **Table Graph** in the window toolbar.

RESULTS

Reaction Force

1 In the **Model Builder** window, under **Results** click **1D Plot Group 5**.

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

3 Locate the **Plot Settings** section.

4 Select the **x-axis label** check box. In the associated text field, type Top displacement (mm).

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>solid.epeGp - Equivalent plastic strain - 1**.

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 **Results>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, 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 From the **Parameter value (delta (mm))** list, choose **8**.

5 Locate the **Plot Settings** section. Clear the **Plot dataset edges** check box.

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