Created in COMSOL Multiphysics 5.6

Compression of an Elastoplastic Pipe

Introduction

In offshore applications, it is sometimes necessary to quickly seal a pipe as part of the prevention of a blowout. This example shows a simulation in which a circular pipe is squeezed between two flat stiff indenters.

The tutorial serves as an example of an analysis with very large plastic strains and contact.

Model Definition

The pipe has an external radius, R_0, of 200 mm and a wall thickness of 25 mm. The pipe is compressed between two flat indenters that can be considered as rigid. The geometry is shown in Figure 1. The original position of each indenter is 0.1 mm from the outer pipe wall. During the compression of the pipe, the distance between the indenters is decreased by 300 mm, and then they are retracted to their original positions. Due to the symmetries, only one quarter of the geometry needs to be modeled. The problem is considered as 2D with the plane strain assumption.

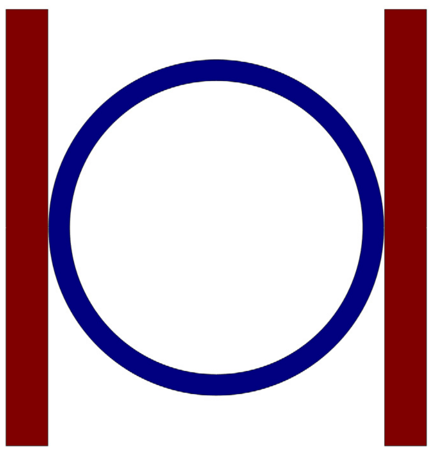

Figure 1: The geometry of the pipe and indenters.

The pipe is modeled using an elastoplastic material model and is assumed to be made of stainless steel with the following properties:

TABLE 1: MATERIAL DATA.

PROPERTY	VALUE
Young's modulus	195 GPa
Poisson's ratio	0.3
Yield stress	250 MPa
Ultimate tensile stress	616 MPa
Ultimate strain	0.52

The hardening curve is available as a text file containing pairs of data (plastic strain, stress) which can be imported as a function. The function is shown in Figure 2 below. This nonlinear hardening function $\sigma_h(\varepsilon_{pe})$ is defined in the **Materials** node by an interpolation function. The stress is measured as true (Cauchy) stress, and the strain is measured as true (logarithmic) strain. The data can thus be used directly as input to COMSOL when large strain plasticity is used. Note that if the strain is above the ultimate strain, then the curve is implicitly flat, so that deformation continues under constant stress.

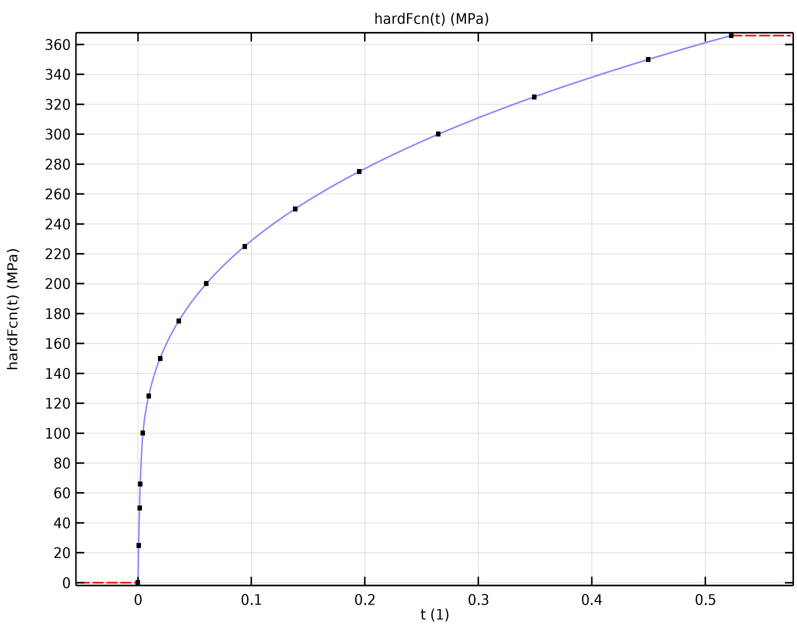

Figure 2: The hardening curve.

Results and Discussion

This example exhibits extremely large plastic strains. The deformation and stress state at the maximum compression are shown in Figure 3. The maximum stress displayed is slightly above the ultimate tensile stress (616 MPa). This is caused by the extrapolation of the results from the integration points inside the elements, where the constitutive law is exactly fulfilled.

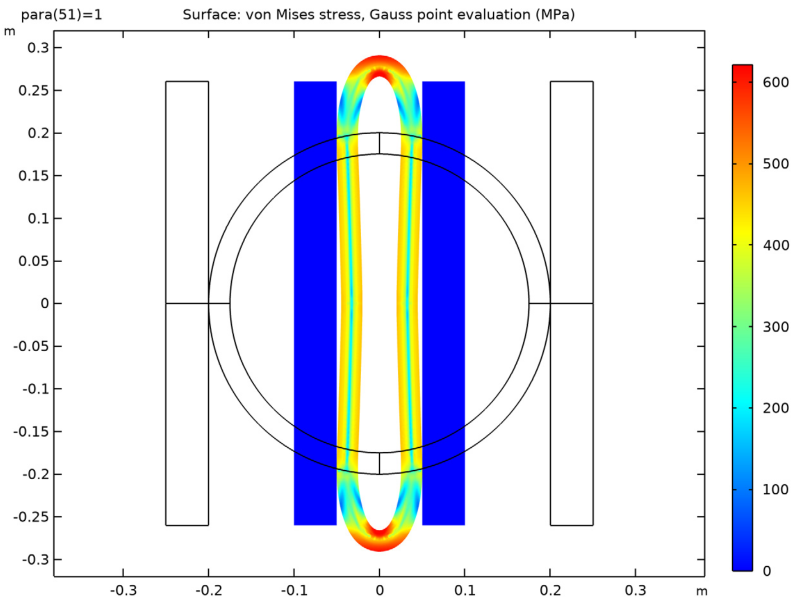

Figure 3: Distribution of von Mises stress at maximum compression.

The distribution of the equivalent plastic strain is shown in Figure 4 and Figure 5. As can be seen, the peak value close to 1 is far above the ultimate strain (0.52). All values above the ultimate strain are however on the inside of the pipe, where the strain state is mainly in compression. At the outer edge of the pipe, the plastic strain approximately reaches the ultimate strain. There is thus a certain risk that cracks could start forming. The values of ultimate stress and strain are related to the specimen used for the testing (usually a cylindrical bar) and cannot directly be transferred to general multiaxial conditions.

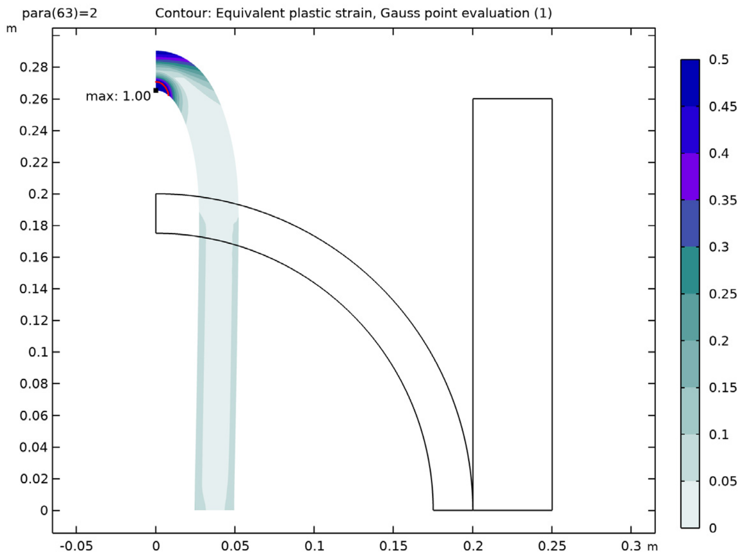

Figure 4: Equivalent plastic strain at maximum compression.

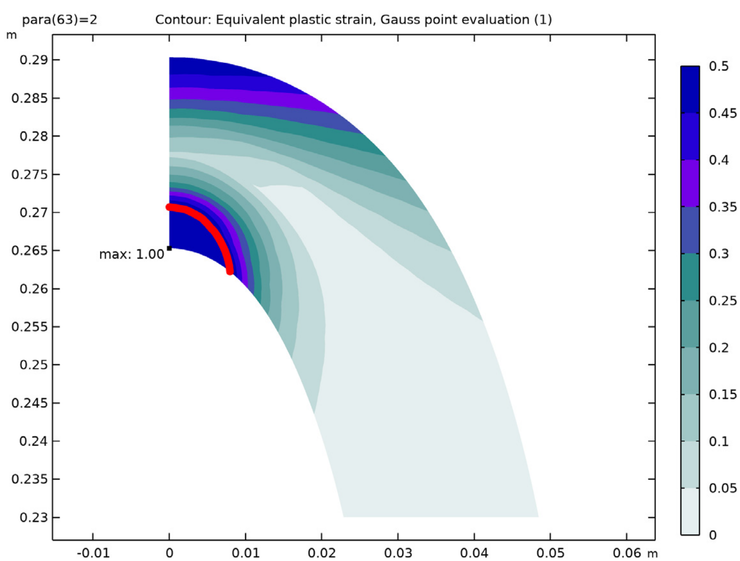

Figure 5: Equivalent plastic strain at maximum compression, detail. The red contour indicates the ultimate strain (0.52).

The final state after the retraction of the indenters is shown in Figure 6. There is some reversed yielding during the unloading process. The springback effect is very small.

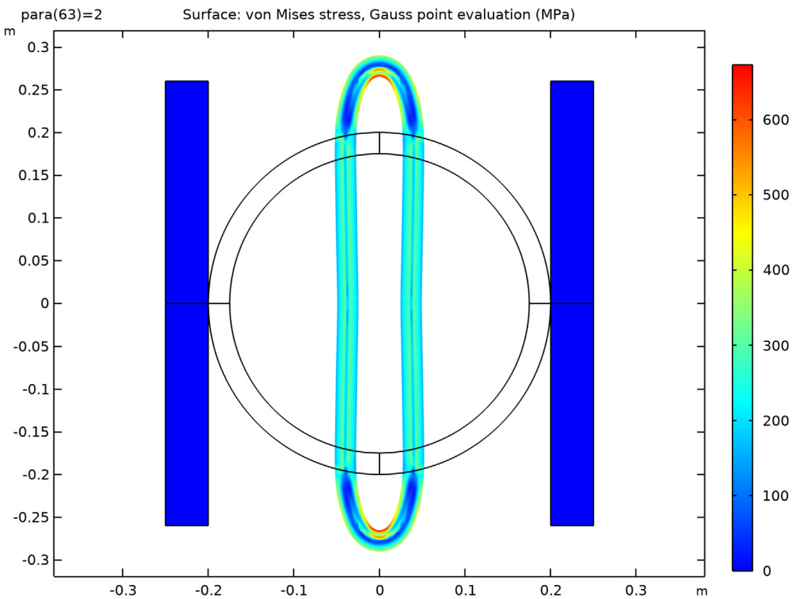

Figure 6: Deformed shape and residual stresses at the end of the process.

The load used to compress the pipe is computed from the reaction force in the indenter, and it is shown in Figure 7.

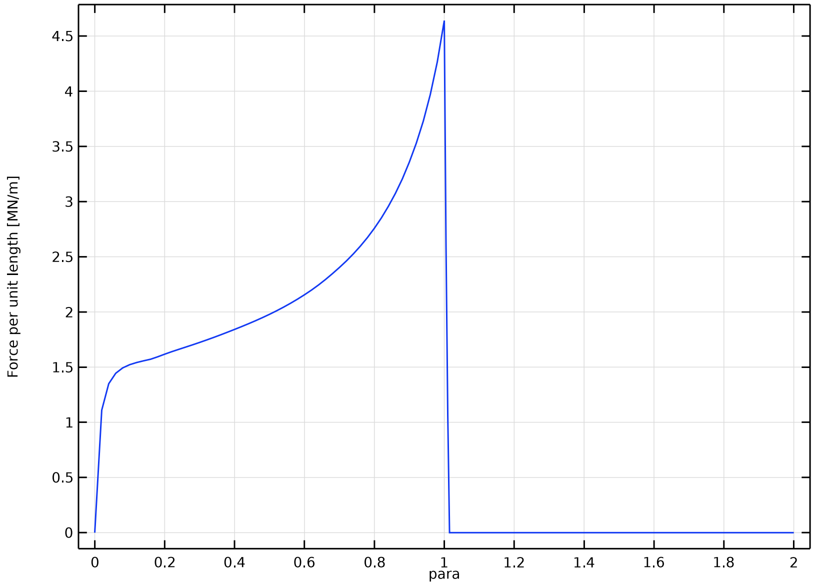

Figure 7: Applied force as function of the loading parameter.

Application Library path: `Nonlinear_Structural_Materials_Module/Plasticity/compressed_elastoplastic_pipe`

Modeling Instructions

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

NEW

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

MODEL WIZARD

1 In the **Model Wizard** window, click **2D**.

2 In the **Select Physics** tree, select **Structural Mechanics>Solid Mechanics (solid)**.

3 Click **Add**.

4 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
`Ro`	`200[mm]`	0.2 m	`Outer radius`
`thic`	`25[mm]`	0.025 m	`Pipe wall thickness`
`Ri`	`Ro-thic`	0.175 m	`Inner radius`
`para`	`0`	0	`Solution parameter`

Add a function for the displacement of the indenting part.

Interpolation 1 (int1)

1 In the **Home** toolbar, click f(X) **Functions** and choose **Global>Interpolation**.

2 In the **Settings** window for **Interpolation**, locate the **Definition** section.

3 In the **Function name** text field, type `compr`.

4 In the table, enter the following settings:

t	f(t)
0	0
1	0.15
2	0

5 Locate the **Units** section. In the **Arguments** text field, type `1`.

6 In the **Function** text field, type `m`.

7 Click **Plot**.

GEOMETRY 1

Circle 1 (c1)

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

2 In the **Settings** window for **Circle**, locate the **Size and Shape** section.

3 In the **Radius** text field, type `Ro`.

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

Circle 2 (c2)

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

2 In the **Settings** window for **Circle**, locate the **Size and Shape** section.

3 In the **Radius** text field, type `Ri`.

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

Difference 1 (dif1)

1 In the **Geometry** toolbar, click **Booleans and Partitions** and choose **Difference**.

2 Select the object **c1** only.

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

4 Find the **Objects to subtract** subsection. Select the **Activate Selection** toggle button.

5 Select the object **c2** only.

6 Click **Build All Objects**.

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 `0.05`.

4 In the **Height** text field, type `1.3*Ro`.

5 Locate the **Position** section. In the **x** text field, type `Ro+0.0001`.

Form Union (fin)

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

2 In the **Settings** window for **Form Union/Assembly**, locate the **Form Union/Assembly** section.

3 From the **Action** list, choose **Form an assembly**.

4 In the **Geometry** toolbar, click **Build All**.

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

DEFINITIONS

Contact Pair 1 (p1)

1 In the **Definitions** toolbar, click **Pairs** and choose **Contact Pair**.

2 Select Boundary 5 only.

3 In the **Settings** window for **Pair**, locate the **Destination Boundaries** section.

4 Select the **Activate Selection** toggle button.

5 Select Boundary 4 only.

SOLID MECHANICS (SOLID)

Linear Elastic Material 1

In the **Model Builder** window, under **Component 1 (comp1)>Solid Mechanics (solid)** click **Linear Elastic Material 1**.

Plasticity 1

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

2 Select Domain 1 only.

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

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

5 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	195[GPa]	Pa	Basic
Poisson's ratio	nu	0.3	1	Basic
Density	rho	8000	kg/m³	Basic
Initial yield stress	sigmags	250[MPa]	Pa	Elastoplastic material model

Add the hardening curve for the elastoplastic material.

4 In the **Model Builder** window, expand the **Material 1 (mat1)** node, then click **Elastoplastic material model (ElastoplasticModel)**.

5 In the **Settings** window for **Property Group**, locate the **Model Inputs** section.

6 Click **Select Quantity**.

7 In the **Physical Quantity** dialog box, type `plastic strain` in the text field.

8 Click **Filter**.

9 In the tree, select **Solid Mechanics>Equivalent plastic strain (1)**.

10 Click **OK**.

Interpolation 1 (int1)

1 Right-click **Elastoplastic material model (ElastoplasticModel)** and choose **Functions>Interpolation**.

2 In the **Settings** window for **Interpolation**, locate the **Definition** section.

3 Click **Load from File**.

4 Browse to the model's Application Libraries folder and double-click the file `compressed_elastoplastic_pipe_stress_strain.txt`.

5 In the **Function name** text field, type `hardFcn`.

6 Locate the **Interpolation and Extrapolation** section. From the **Interpolation** list, choose **Piecewise cubic**.

7 Locate the **Units** section. In the **Arguments** text field, type `1`.

8 In the **Function** text field, type `MPa`.

9 Click **Plot**.

Material 1 (mat1)

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

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
Hardening function	sigmagh	`hardFcn(epe)`	Pa	Elastoplastic material model

SOLID MECHANICS (SOLID)

Contact 1

1 In the **Physics** toolbar, click **Pairs** and choose **Contact**.

2 In the **Settings** window for **Contact**, locate the **Pair Selection** section.

3 Under **Pairs**, click **Add**.

4 In the **Add** dialog box, select **Contact Pair 1 (p1)** in the **Pairs** list.

5 Click **OK**.

Symmetry 1

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

2 Select Boundaries 1 and 2 only.

Prescribed Displacement 1

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

2 Select Domain 2 only.

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

4 Select the **Prescribed in x direction** check box.

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

6 In the u_{0x} text field, type `-compr(para)`.

MESH 1

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

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

3 From the **Sequence type** list, choose **User-controlled mesh**.

Free Triangular 1

1 In the **Model Builder** window, under **Component 1 (comp1)>Mesh 1** right-click **Free Triangular 1** and choose **Delete**.

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

3 Click **Yes** to confirm.

Mapped 1

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

Distribution 1

1 Right-click **Mapped 1** and choose **Distribution**.

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

3 In the **Number of elements** text field, type `1`.

4 Select Boundaries 5 and 6 only.

A single element is enough for the indenter, since the whole domain is under displacement control.

Distribution 2

1. In the **Model Builder** window, right-click **Mapped 1** and choose **Distribution**.
2. In the **Settings** window for **Distribution**, locate the **Distribution** section.
3. From the **Distribution type** list, choose **Predefined**.
4. Select Boundaries 1 and 2 only.
5. In the **Number of elements** text field, type `8`.
6. In the **Element ratio** text field, type `2`.
7. Select the **Symmetric distribution** check box.

Distribution 3

1. Right-click **Mapped 1** and choose **Distribution**.
2. Select Boundary 3 only.
3. In the **Settings** window for **Distribution**, locate the **Distribution** section.
4. In the **Number of elements** text field, type `80`.
5. Click **Build All**.

STUDY 1

Step 1: Stationary

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

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
para (Solution parameter)	`range(0,0.02,1) range(1.005, 0.005, 1.05) 1.1 2`

6. Click to expand the **Results While Solving** section. Select the **Plot** check box.
7. In the **Home** toolbar, click **Compute**.

RESULTS

Stress (solid)

Mirror the solution twice to get a full view of the pipe.

Mirror 2D 1

In the **Results** toolbar, click **More Datasets** and choose **Mirror 2D**.

Mirror 2D 2

1 In the **Results** toolbar, click **More Datasets** and choose **Mirror 2D**.

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

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

4 Locate the **Axis Data** section. In row **Point 2**, set **x** to `1` and **y** to `0`.

Stress (solid)

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

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

3 From the **Dataset** list, choose **Mirror 2D 2**.

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**.

Stress (solid)

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

2 In the **Stress (solid)** toolbar, click **Plot**.

Plot the stresses at the maximum compression. This gives .

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

4 From the **Parameter value (para)** list, choose **1**.

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

Create an animation of the compression process.

Animation 1

In the **Stress (solid)** toolbar, click **Animation** and choose **Player**.

Contour 1

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

2 In the **Settings** window for **Contour**, locate the **Levels** section.

3 From the **Entry method** list, choose **Levels**.

4 In the **Levels** text field, type `range(0,0.05,0.45)`.

Deformation 1

1 Right-click **Contour 1** and choose **Deformation**.

2 In the **Settings** window for **Deformation**, locate the **Scale** section.

3 Select the **Scale factor** check box.

4 In the associated text field, type `1`.

Contour 2

1 Right-click **Contour 1** and choose **Duplicate**.

2 In the **Settings** window for **Contour**, click to expand the **Title** section.

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

4 Locate the **Levels** section. From the **Entry method** list, choose **Levels**.

5 In the **Levels** text field, type `0.52`.

6 Locate the **Coloring and Style** section. From the **Contour type** list, choose **Tube**.

7 Select the **Radius scale factor** check box.

8 In the **Tube radius expression** text field, type `5e-4`.

9 From the **Coloring** list, choose **Uniform**.

10 Clear the **Color legend** check box.

11 Click to expand the **Inherit Style** section. From the **Plot** list, choose **Contour 1**.

12 Clear the **Color** check box.

13 Clear the **Color and data range** check box.

Equivalent Plastic Strain (solid)

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**, click to expand the **Title** section.

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

4 Locate the **Expression** section. In the **Expression** text field, type `solid.epeGp`.

5 Locate the **Display** section. From the **Display** list, choose **Max**.

6 Locate the **Text Format** section. In the **Display precision** text field, type `3`.

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

8 Click to expand the **Inherit Style** section. From the **Plot** list, choose **Contour 1**.

Deformation 1

1 Right-click **Max/Min Surface 1** and choose **Deformation**.

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

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

Create a graph of the applied force as function of the compression.

Surface Integration 1

1 In the **Results** toolbar, click 8.85 e-12 **More Derived Values** and choose **Integration>Surface Integration**.

2 Select Domain 2 only.

3 In the **Settings** window for **Surface Integration**, locate the **Expressions** section.

4 In the table, enter the following settings:

Expression	Unit
`-solid.RFx*2/1E6`	N

5 Click **Evaluate**.

TABLE

1 Go to the **Table** window.

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

RESULTS

Compression Force

1 In the **Model Builder** window, under **Results** click **1D Plot Group 4**.

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

3 Locate the **Plot Settings** section. Select the **y-axis label** check box.

4 In the associated text field, type `Force per unit length [MN/m]`.

5 In the **Compression Force** toolbar, click **Plot**.