



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 by a tool. The main target of the analysis is to find the remaining cross-section area through which a fluid can pass.

The tutorial serves as an example of an analysis involving 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 the two sides of a tool that can be considered as rigid. The geometry is shown in [Figure 1](#). The original position of the tool is 0.1 mm from the outer pipe wall. During the compression of the pipe, the width of the tool is decreased until the maximum available force, 6 MN, is reached. The tool is then retracted, while the pipe expands as an effect of elastic springback. 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.

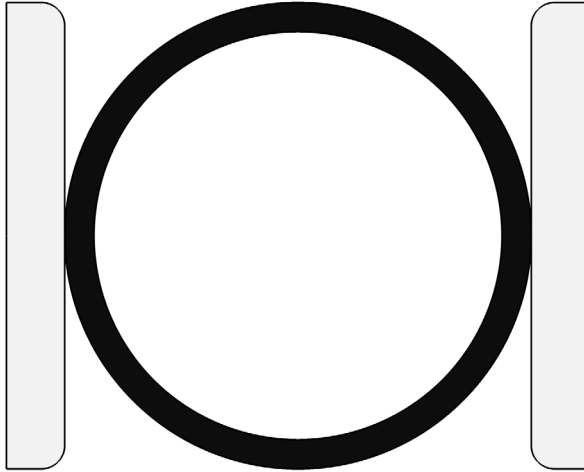


Figure 1: The geometry of the pipe and tool.

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

TABLE I: 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 $\sigma_h(\epsilon_{pe})$ 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, the curve is implicitly flat, so that deformation continues under constant stress.

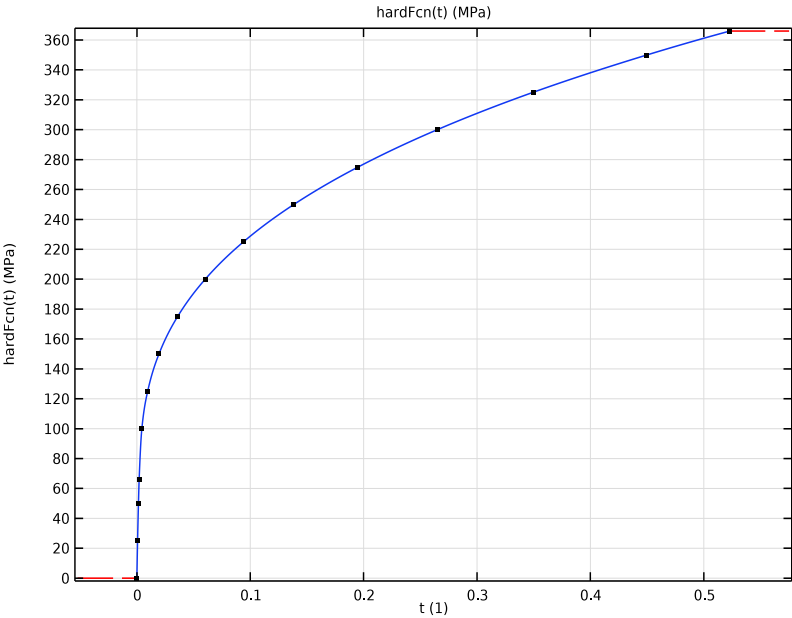


Figure 2: The hardening curve.

In order to be able to assess the sensitivity to the material data, a multiplier to the hardening curve is introduced in the model, although it has the value 1 throughout this analysis.

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 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.

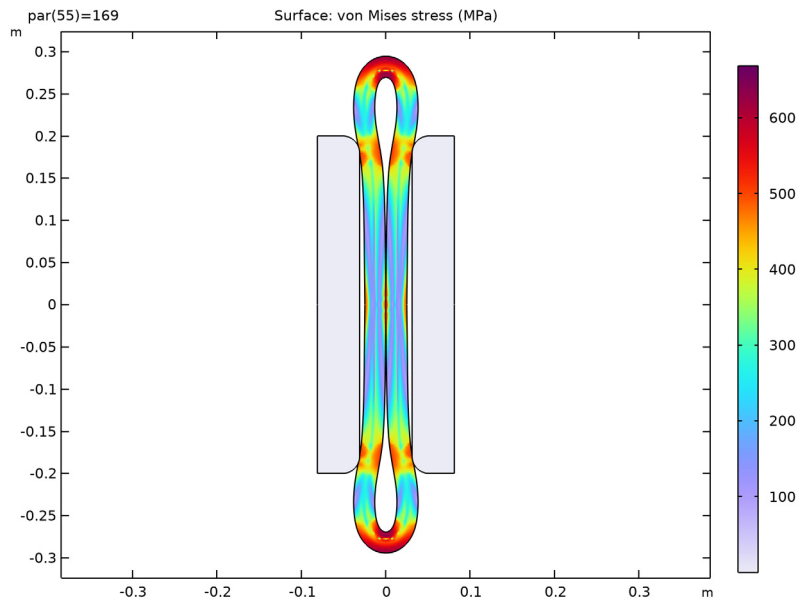


Figure 3: Distribution of von Mises stress at maximum compression.

The distribution of the equivalent plastic strain is shown in Figure 4 and Figure 5. As can be seen, the peak value above 1 is far above the ultimate strain (0.52). The majority of values above the ultimate strain are, however, located 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 also exceeds the ultimate strain, but not to the same extent. There is thus a 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.

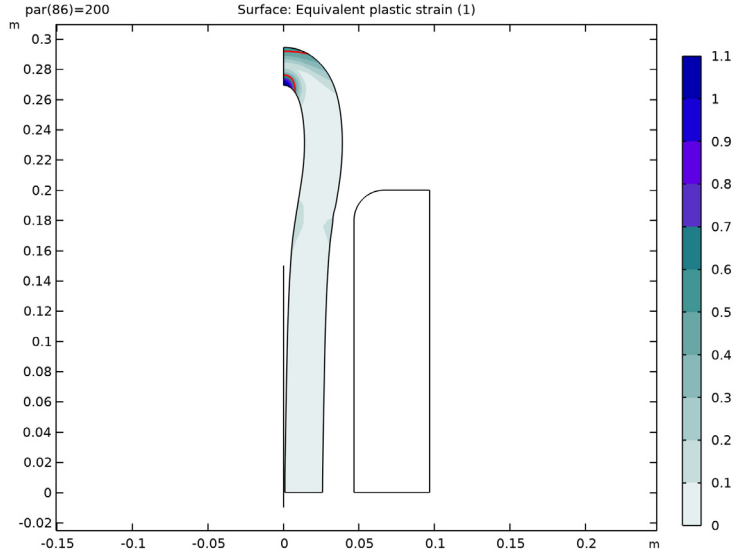


Figure 4: Equivalent plastic strain at the end of the process.

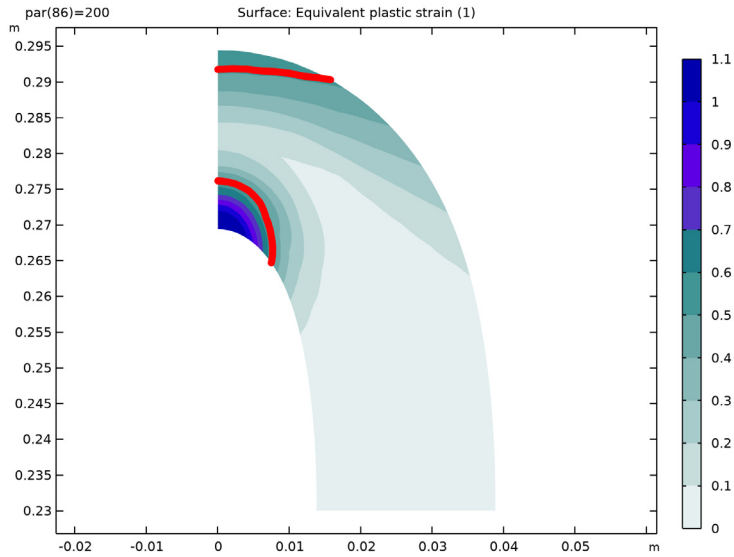


Figure 5: Equivalent plastic strain at the end of the process, detail. The red contour indicates the ultimate strain (0.52).

The final state after the retraction of the compression tool is shown in [Figure 6](#). There is some reversed yielding during the unloading process. The springback effect is very small.

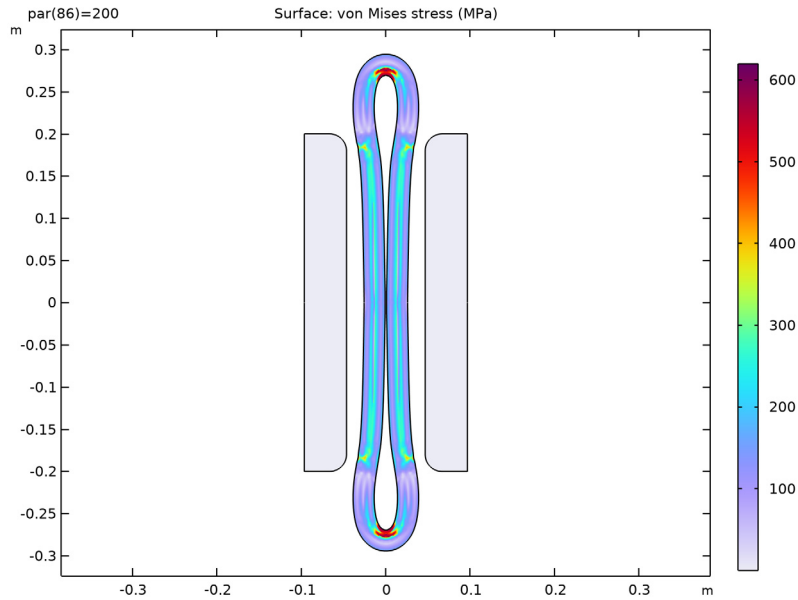


Figure 6: Deformed shape and residual stresses at the end of the process.

The load used to compress the pipe is computed from the reaction force on the tool, and it is shown in [Figure 7](#). When the inside of the pipe makes contact with itself, the force rises steeply.

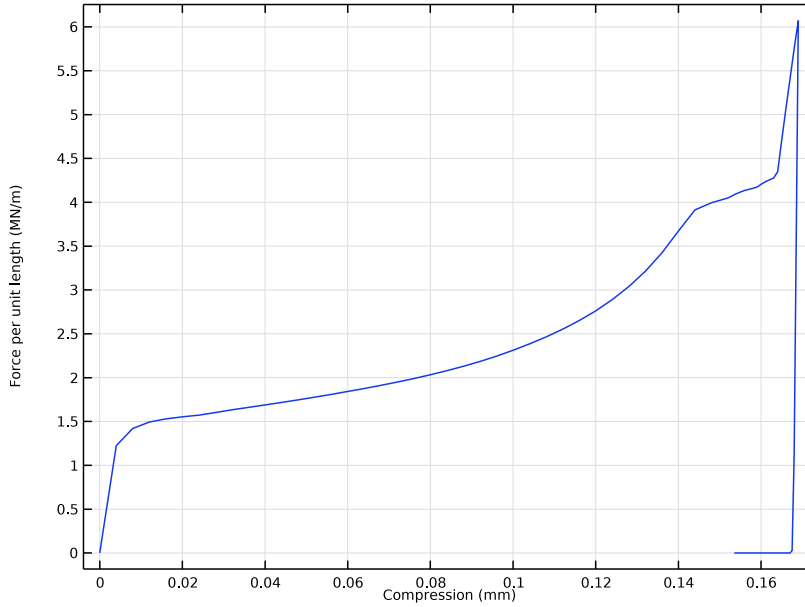


Figure 7: Applied force as function of the compression.

The change in the cross section area during the simulation is shown in [Figure 8](#). The original area is 96211 mm^2 . At the maximum compression, it is reduced to 4831 mm^2 , and in the final state it is 6031 mm^2 . Thus, the area increases by about 25% due to elastic springback. Still, the final cross section area is only 6% of the original. Due to the change in shape, the flow will be reduced even more, since the wall perimeter is essentially unchanged. The resistance to the flow is largest near the walls.

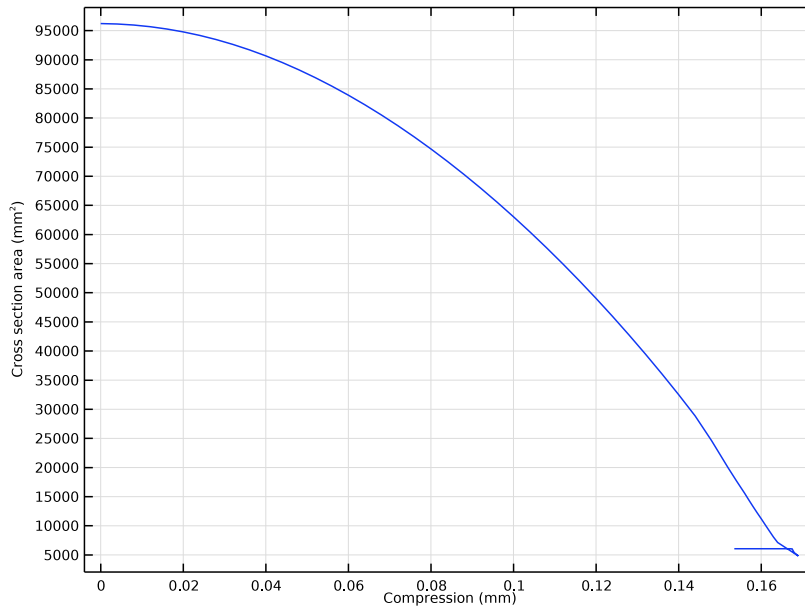


Figure 8: Development of the cross section area as function of compression.

Notes About the COMSOL Implementation

In order to capture when the maximum force from the indenter is reached, a state variable is used. The state variable is updated after each parameter step, and has an update expression that stores the parameter value when the maximum attainable force is reached.


Since symmetry is used, the possible self-contact inside the pipe is modeled as contact against the symmetry plane.

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 I

- 1 In the **Model Builder** window, under **Global Definitions** click **Parameters I**.
- 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
Sy	250[MPa]	2.5E8 Pa	Yield stress
hfact	1	1	Hardening curve multiplier
maxLoad	6[MN]	6E6 N	The maximum load from the tool
par	0	0	Solution parameter

GEOMETRY I


Circle: Pipe

- 1 In the **Geometry** toolbar, click  **Circle**.
- 2 In the **Settings** window for **Circle**, type Circle: Pipe in the **Label** text field.
- 3 Locate the **Size and Shape** section. In the **Radius** text field, type Ro.
- 4 In the **Sector angle** text field, type 90.


5 Click to expand the **Layers** section. In the table, enter the following settings:

Layer name	Thickness (m)
Layer 1	thic


Delete Entities I (delI)

- 1 In the **Model Builder** window, right-click **Geometry 1** and choose **Delete Entities**.
- 2 In the **Settings** window for **Delete Entities**, locate the **Entities or Objects to Delete** section.
- 3 From the **Geometric entity level** list, choose **Domain**.
- 4 On the object **c1**, select Domain 1 only.
- 5 Click  **Build Selected**.


Rectangle: Tool

- 1 In the **Geometry** toolbar, click  **Rectangle**.
- 2 In the **Settings** window for **Rectangle**, type Rectangle: Tool in the **Label** text field.
- 3 Locate the **Size and Shape** section. In the **Width** text field, type 0.05.
- 4 In the **Height** text field, type R_0 .
- 5 Locate the **Position** section. In the **x** text field, type $R_0 + 0.0001$.

Fillet I (filI)

- 1 In the **Geometry** toolbar, click  **Fillet**.
- 2 On the object **r1**, select Point 4 only.
- 3 In the **Settings** window for **Fillet**, locate the **Radius** section.
- 4 In the **Radius** text field, type $R_0/10$.



Polygon: Symmetry Contact Plane

- 1 In the **Geometry** toolbar, click  **Polygon**.
- 2 In the **Settings** window for **Polygon**, type Polygon: Symmetry Contact Plane in the **Label** text field.
- 3 Locate the **Object Type** section. From the **Type** list, choose **Open curve**.
- 4 Locate the **Coordinates** section. In the table, enter the following settings:

x (m)	y (m)
0	0.15
0	-0.01



- 5 Click  **Build Selected**.

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 Boundaries 6 and 10 only.
- 3 In the **Settings** window for **Pair**, locate the **Destination Boundaries** section.
- 4 Click to select the  **Activate Selection** toggle button.
- 5 Select Boundary 5 only.

Contact Pair 2 (p2)


- 1 In the **Definitions** toolbar, click  **Pairs** and choose **Contact Pair**.
- 2 Select Boundary 1 only.
- 3 In the **Settings** window for **Pair**, locate the **Destination Boundaries** section.
- 4 Click to 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 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	Young's modulus and Poisson's ratio
Poisson's ratio	nu	0.3	1	Young's modulus and Poisson's ratio
Density	rho	8000	kg/m ³	Basic
Initial yield stress	sigmags	Sy	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 **Elastoplastic Material Model**, 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 **Component 1 (comp1)>Materials>Material 1 (mat1)>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 **Argument** table, enter the following settings:

Argument	Unit
t	1

8 In the **Function** table, enter the following settings:

Function	Unit
hardFcn	MPa

9 Click  **Plot**.

Material 1 (mat1)

1 In the **Model Builder** window, under **Component 1 (comp1)>Materials** 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)* hfact	Pa	Elastoplastic material model

Add a state variable to store the parameter value when the maximum load is reached.

GLOBAL DEFINITIONS

1 Click the  **Show More Options** button in the **Model Builder** toolbar.

2 In the **Show More Options** dialog box, select **General>Variable Utilities** in the tree.

3 In the tree, select the check box for the node **General>Variable Utilities**.

4 Click **OK**.

State Variables 1 (state1)

1 In the **Home** toolbar, click  **Variable Utilities** and choose **Global>State Variables**.

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

3 In the table, enter the following settings:


State	Initial value	Update expression	Description
peakParam	0	if(peakParam>0, peakParam, if(comp1.reacF> maxLoad,par,0))	Parameter value at max compression

- 4 From the **Update** list, choose **At end of step**.


Add variables computing the reaction force on the tool and the current pipe cross section.

DEFINITIONS

Integration 1 (intop1)

- 1 In the **Definitions** toolbar, click  **Nonlocal Couplings** and choose **Integration**.
- 2 In the **Settings** window for **Integration**, type ReacInt in the **Operator name** text field.
- 3 Select Domain 2 only.
- 4 Locate the **Advanced** section. From the **Method** list, choose **Summation over nodes**.

Integration 2 (intop2)

- 1 In the **Definitions** toolbar, click  **Nonlocal Couplings** and choose **Integration**.
- 2 In the **Settings** window for **Integration**, type AreaInt in the **Operator name** text field.
- 3 Locate the **Source Selection** section. From the **Geometric entity level** list, choose **Boundary**.
- 4 Select Boundary 4 only.

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
reacF	$-2 * \text{ReacInt}(\text{solid.RFx})$	N	Applied force from tool
area	$\text{AreaInt}(-x * \text{solid.nx}) * 4$	m ²	Current cross section area for flow
disp	$\text{if}(\text{peakParam}, \text{peakParam} - (\text{par} - \text{peakParam}) / 2, \text{par}) [\text{mm}]$	m	Compression function

SOLID MECHANICS (SOLID)

Symmetry 1

- 1 In the **Physics** toolbar, click  **Boundaries** and choose **Symmetry**.
- 2 Select Boundaries 2 and 3 only.

Prescribed Displacement: Compression

- 1 In the **Physics** toolbar, click  **Domains** and choose **Prescribed Displacement**.
- 2 In the **Settings** window for **Prescribed Displacement**, type Prescribed Displacement: Compression in the **Label** text field.
- 3 Select Domain 2 only.
- 4 Locate the **Prescribed Displacement** section. From the **Displacement in x direction** list, choose **Prescribed**.
- 5 From the **Displacement in y direction** list, choose **Prescribed**.
- 6 In the u_{0x} text field, type -disp.

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**.

Free Triangular 1

- 1 In the **Model Builder** window, under **Component 1 (comp1)**>**Mesh 1** click **Free Triangular 1**.
- 2 In the **Settings** window for **Free Triangular**, locate the **Domain Selection** section.
- 3 From the **Geometric entity level** list, choose **Domain**.
- 4 Select Domain 2 only.

Distribution 1


- 1 Right-click **Free Triangular 1** and choose **Distribution**.
- 2 In the **Settings** window for **Distribution**, locate the **Boundary Selection** section.
- 3 From the **Selection** list, choose **All boundaries**.

A minimal resolution is needed for the tool, since the whole domain is under displacement control. On the curved boundary, however, a fine resolution is needed.
- 4 Locate the **Distribution** section. In the **Number of elements** text field, type 1.

Distribution 2

- 1 Right-click **Free Triangular 1** and choose **Distribution**.
- 2 Select Boundary 10 only.
- 3 In the **Settings** window for **Distribution**, locate the **Distribution** section.
- 4 In the **Number of elements** text field, type 30.

Mapped 1

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

Distribution 1

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

Distribution 2


- 1 In the **Model Builder** window, right-click **Mapped 1** and choose **Distribution**.
- 2 Select Boundaries 2 and 3 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 8.
- 6 In the **Element ratio** text field, type 2.
- 7 Select the **Symmetric distribution** check box.

The symmetry boundary must also be meshed because of the contact, even though there is no physics attached to it.

Edge 1

- 1 In the **Mesh** toolbar, click  **More Generators** and choose **Edge**.
- 2 Select Boundary 1 only.

Distribution 1


- 1 Right-click **Edge 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 Click  **Build All**.

STUDY 1


Step 1: Stationary

Set up an auxiliary continuation sweep for the parameter **par**. We do not know exactly how many parameter steps that are needed, but the last ones will be cheap to compute, since the tool is no longer in contact with the pipe.



- 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
par (Solution parameter)	range(0,4,152) range(154,1,200)

- 6 Click to expand the **Results While Solving** section. Select the **Plot** check box.
- 7 In the **Study** toolbar, click  **Get Initial Value**.

ADD PREDEFINED PLOT

- 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 I/Solution I (sol1)>Solid Mechanics>Equivalent Plastic Strain (solid)** and **Study I/Solution I (sol1)>Solid Mechanics>Contact Forces (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.


STUDY I

Solution I (sol1)

- 1 In the **Model Builder** window, expand the **Study I>Solver Configurations** node.
 - 2 In the **Model Builder** window, expand the **Solution I (sol1)** node.
 - 3 In the **Model Builder** window, expand the **Study I>Solver Configurations>Solution I (sol1)>Stationary Solver I** node, then click **Parametric I**.
 - 4 In the **Settings** window for **Parametric**, click to expand the **Continuation** section.
 - 5 From the **Predictor** list, choose **Linear**.
 - 6 Select the **Tuning of step size** check box.
 - 7 In the **Initial step size** text field, type 1.
 - 8 In the **Maximum step size** text field, type 1.
- Mirror the solution twice to get a full view of the pipe.

RESULTS

Mirror 2D 1

- 1 In the **Results** toolbar, click  **More Datasets** and choose **Mirror 2D**.
- 2 In the **Settings** window for **Mirror 2D**, click to expand the **Advanced** section.
- 3 Select the **Remove elements on the symmetry axis** check box.


Mirror 2D 2

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

Mirror 2D 2

- 1 In the **Model Builder** window, click **Mirror 2D 2**.
- 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.
- 5 Locate the **Advanced** section. Select the **Remove elements on the symmetry axis** check box.


Stress (solid)

- 1 In the **Model Builder** window, under **Results** 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**.
- 4 Click the  **Zoom Extents** button in the **Graphics** toolbar.

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**.

STUDY 1

- 1 In the **Home** toolbar, click  **Compute**.
Check the remaining cross section area.

RESULTS

Remaining Cross-Section Area

- 1 In the **Results** toolbar, click  **Evaluation Group**.

- 2 In the **Settings** window for **Evaluation Group**, type Remaining Cross-Section Area in the **Label** text field.
- 3 Locate the **Data** section. From the **Parameter selection (par)** list, choose **Last**.



Global Evaluation I

- 1 Right-click **Remaining Cross-Section Area** and choose **Global Evaluation**.
- 2 In the **Settings** window for **Global Evaluation**, locate the **Expressions** section.
- 3 In the table, enter the following settings:

Expression	Unit	Description
area	mm^2	

- 4 In the **Remaining Cross-Section Area** toolbar, click  **Evaluate**.

Stress (solid)


- 1 In the **Model Builder** window, under **Results** click **Stress (solid)**.
- 2 In the **Stress (solid)** toolbar, click  **Plot**.
Plot the stress at the maximum compression. This gives [Figure 5](#).
- 3 In the **Settings** window for **2D Plot Group**, locate the **Data** section.
- 4 From the **Parameter value (par)** list, choose **I69**.
- 5 In the **Stress (solid)** toolbar, click  **Plot**.

Create an animation of the compression process.

Animation I

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

Equivalent Plastic Strain (solid)

- 1 In the **Model Builder** window, under **Results** click **Equivalent Plastic Strain (solid)**.
- 2 In the **Settings** window for **2D Plot Group**, locate the **Plot Settings** section.
- 3 From the **Frame** list, choose **Spatial (x, y, z)**.
- 4 In the **Equivalent Plastic Strain (solid)** toolbar, click  **Plot**.

Surface I

- 1 In the **Model Builder** window, expand the **Equivalent Plastic Strain (solid)** node, then click **Surface I**.
- 2 In the **Settings** window for **Surface**, click to expand the **Range** section.
- 3 Select the **Manual color range** check box.
- 4 In the **Maximum** text field, type 1.1.



Deformation I

- 1 Right-click **Surface I** and choose **Deformation**.
- 2 In the **Settings** window for **Deformation**, locate the **Scale** section.
- 3 Select the **Scale factor** check box. In the associated text field, type 1.

Contour I


- 1 In the **Model Builder** window, right-click **Equivalent Plastic Strain (solid)** and choose **Contour**.
- 2 In the **Settings** window for **Contour**, locate the **Expression** section.
- 3 In the **Expression** text field, type `solid.epεGp`.
- 4 Click to expand the **Title** section. From the **Title type** list, choose **None**.
- 5 Locate the **Levels** section. In the **Total levels** text field, type 1.
- 6 From the **Entry method** list, choose **Levels**.
- 7 In the **Levels** text field, type 0.52.
- 8 Locate the **Coloring and Style** section. From the **Contour type** list, choose **Tube**.
- 9 Select the **Radius scale factor** check box.
- 10 In the **Tube radius expression** text field, type $5e-4$.
- 11 From the **Coloring** list, choose **Uniform**.
- 12 Clear the **Color legend** check box.
- 13 Click to expand the **Inherit Style** section. From the **Plot** list, choose **Surface I**.
- 14 Clear the **Color** check box.

Deformation I

- 1 Right-click **Contour I** 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.

Compression Force


- 1 In the **Home** toolbar, click  **Add Plot Group** and choose **ID Plot Group**.
- 2 In the **Settings** window for **ID Plot Group**, type **Compression Force** in the **Label** text field.
- 3 Click to expand the **Title** section. From the **Title type** list, choose **None**.
- 4 Locate the **Plot Settings** section.

- 5 Select the **y-axis label** check box. In the associated text field, type Force per unit length (MN/m).
- 6 Select the **x-axis label** check box. In the associated text field, type Compression (mm).
- 7 Locate the **Legend** section. Clear the **Show legends** check box.

Global I

- 1 Right-click **Compression Force** and choose **Global**.
- 2 In the **Settings** window for **Global**, click **Replace Expression** in the upper-right corner of the **y-Axis Data** section. From the menu, choose **Component I (compI)>Definitions>Variables>reactF - Applied force from tool - N**.
- 3 Locate the **y-Axis Data** section. In the table, enter the following settings:

Expression	Unit	Description
reactF	MN	Applied force from tool

- 4 Locate the **x-Axis Data** section. From the **Parameter** list, choose **Expression**.
- 5 In the **Expression** text field, type disp.
- 6 In the **Compression Force** toolbar, click  **Plot**.

Compression Force

In the **Model Builder** window, right-click **Compression Force** and choose **Duplicate**.

Flow Area

- 1 In the **Model Builder** window, under **Results** click **Compression Force I**.
- 2 In the **Settings** window for **ID Plot Group**, type Flow Area in the **Label** text field.
- 3 Locate the **Plot Settings** section. In the **y-axis label** text field, type Cross section area (mm²).

Global I

- 1 In the **Model Builder** window, expand the **Flow Area** node, then click **Global I**.
- 2 In the **Settings** window for **Global**, locate the **y-Axis Data** section.
- 3 In the table, enter the following settings:

Expression	Unit	Description
area	mm ²	Current cross section area for flow

- 4 In the **Flow Area** toolbar, click  **Plot**.

