



Model created in COMSOL Multiphysics 6.4

Sheet Metal Forming

Introduction

In sheet metal forming processes, the springback phenomena caused by the material elastic recovery behavior makes the formed part partially return to its original shape during the release of the forming load.

It is important to predict the springback accurately as the final deformed shape differs slightly from the shape given by the die at full forming load.

This model is a NAFEMS validation problem of sheet metal forming based on experimental data, see [Ref. 1](#). The problem involves several nonlinearities such as boundary nonlinearity (contact), material nonlinearity (elastoplastic material) and geometric nonlinearity. Both 2D plane strain and 3D shell assumption are considered.

The target results consist in the forming angle, the angle after release, and the punch forces as function of the punch displacement.

Model Definition

The geometry consists of an assembly with three parts: the punch, the die, and a 1 mm metal sheet. Due to the symmetry, only one half of the geometry is represented for the 2D assumption and one quarter for the 3D one. [Figure 1](#) shows the geometry used for the 3D problem, while [Figure 2](#) shows the section cut used with the 2D plane strain problem.

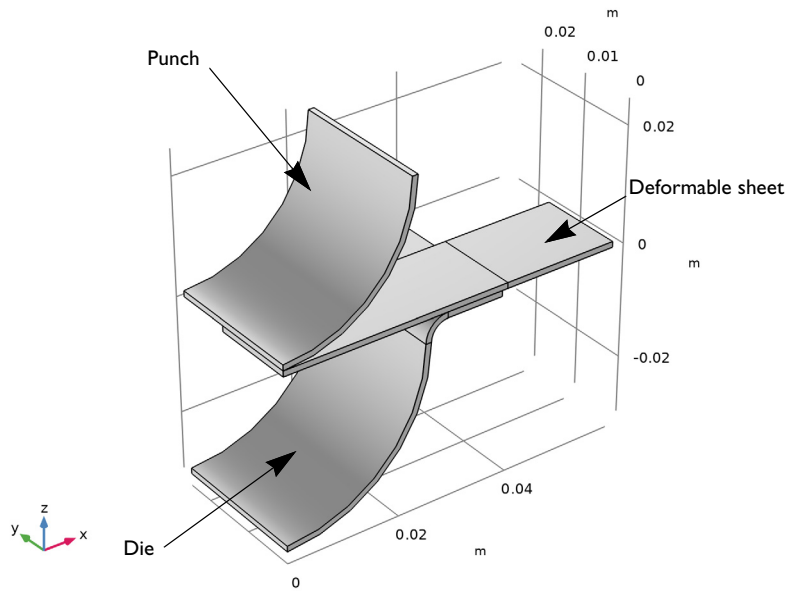


Figure 1: Model geometry: the punch at the top, the thin sheet in the middle, and the die at the bottom.

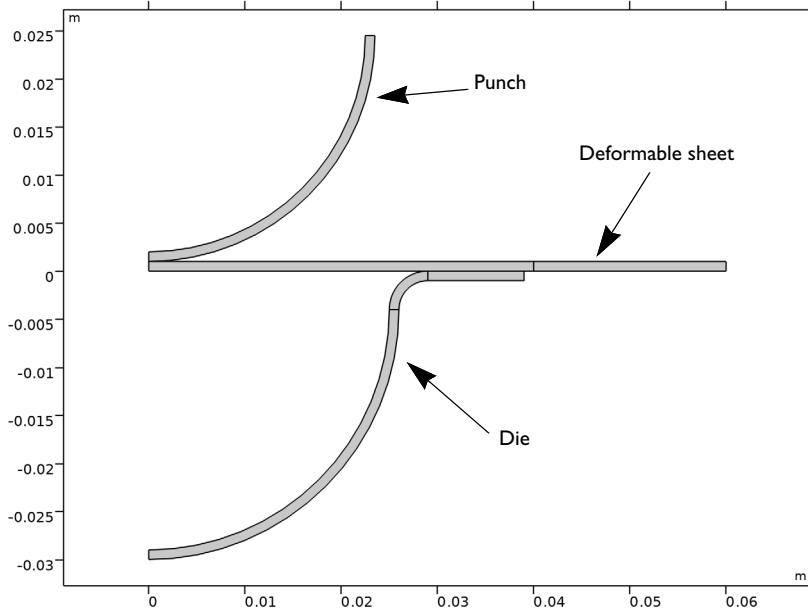


Figure 2: Section cut of the model geometry used for the 2D assumption.

The plane strain assumption is valid here, as the out-of-plane thickness of the sheet is large. Since the thickness of the plate is small compared to the curvature of the tool, moderate strains are expected even though the displacements and rotations are large.

The blank material used in the experiments is a 6111-T4 Aluminum alloy, having an isotropic Young's modulus of $70.5 \cdot 10^3$ MPa and a Poisson's ratio of 0.342. The yield stress of the material is 194 MPa and an Hollomon hardening function is used to represent the elastoplastic behavior. The Hollomon hardening function is a two-coefficient function described by

$$\sigma = K\varepsilon^n$$

Here K is 550.4 MPa and n is 0.223.

To improve the accuracy of the contact condition and forces, the augmented Lagrangian contact method is used. Friction between the thin sheet and the tools is defined using a Coulomb friction coefficient of 0.1342.

Experimental data for the forming angle and the angle after release are available, see [Table 1](#).

TABLE 1: ANGLE AT MAXIMUM PUNCH DISPLACEMENT AND AFTER RELEASE.

Forming angle (degrees)	Angle after release (degrees)
19.6–21.0	53.4–55.8

Data of the punch force are also available; these are stored in the text file `sheet_metal_forming.txt` available in the Nonlinear Structural Material Module's Application Library folder.

Results and Discussion

[Figure 3](#) shows the residual stress distribution and deformed shape after the release for the problem including friction.

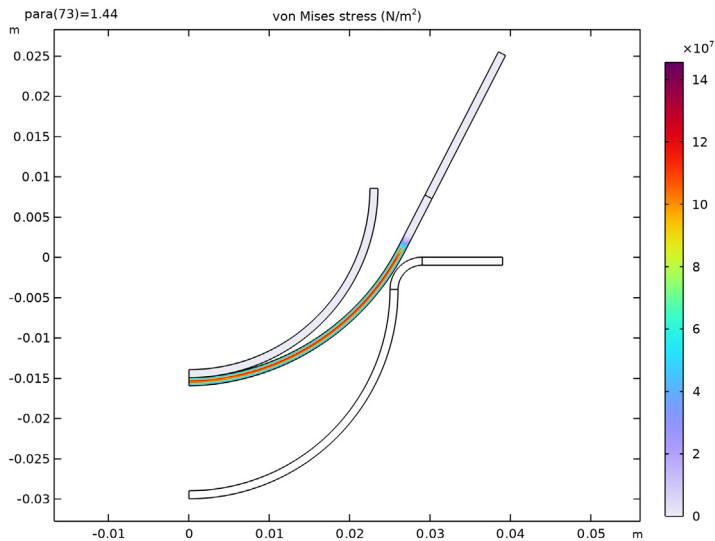


Figure 3: Von Mises stress and deformed shape after release for the 2D plane strain.

In [Figure 4](#) you can see that the maximum value of the plastic strain is about 2%, which validates the small strain assumption.

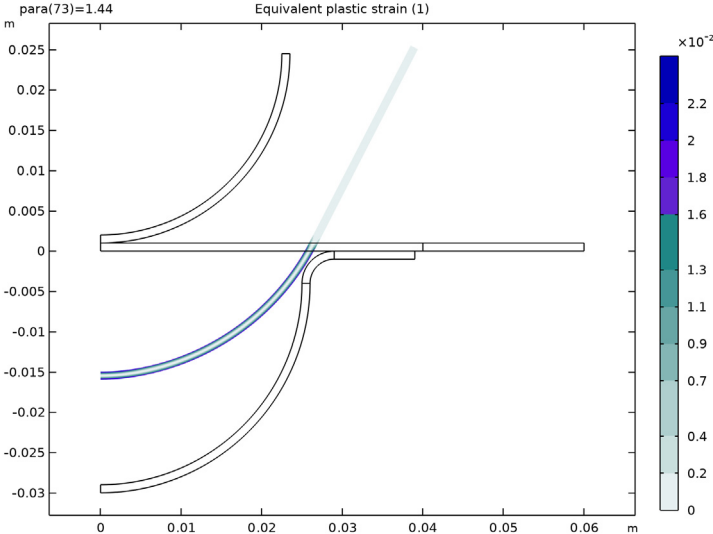


Figure 4: Equivalent plastic strain at maximum punch deflection.

[Figure 5](#) and [Figure 6](#) show the von Mises stress in the sheet computed with a 3D shell assumption. Notice the stress distribution along the y direction that cannot be evaluated with the 2D plane strain assumption.

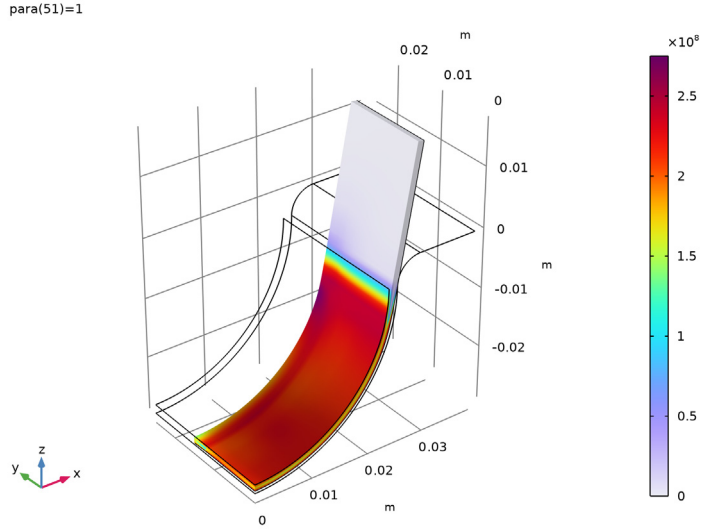


Figure 5: Von Mises stress and deformed shape at forming for the 3D shell problem.

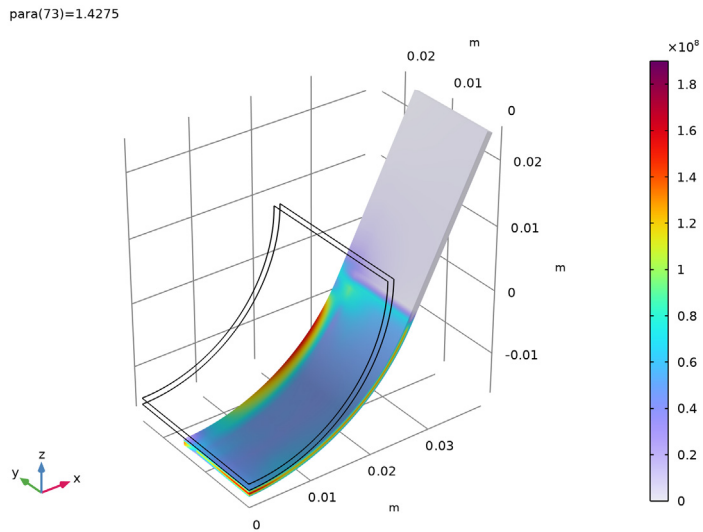


Figure 6: Von Mises stress and deformed shape after release for the 3D shell problem.

The computed angle at the maximum punch displacement and after release are listed in [Table 2](#). These values are in good agreement with the experimental data and the numerical results discussed in [Ref. 1](#).

TABLE 2: COMPUTED ANGLE AT MAXIMUM PUNCH DISPLACEMENT AND AFTER RELEASE.

	Forming angle (degrees)	Angle after release (degrees)
Without friction	20.6	46.0
With friction	20.5	53.9
With friction (shell)	21.6	52.8

[Figure 7](#) shows the punch forces versus the punch displacement. One can clearly notice the effect of friction in the applied load. Furthermore, the blank remains in contact with the tools much longer when friction forces are included.

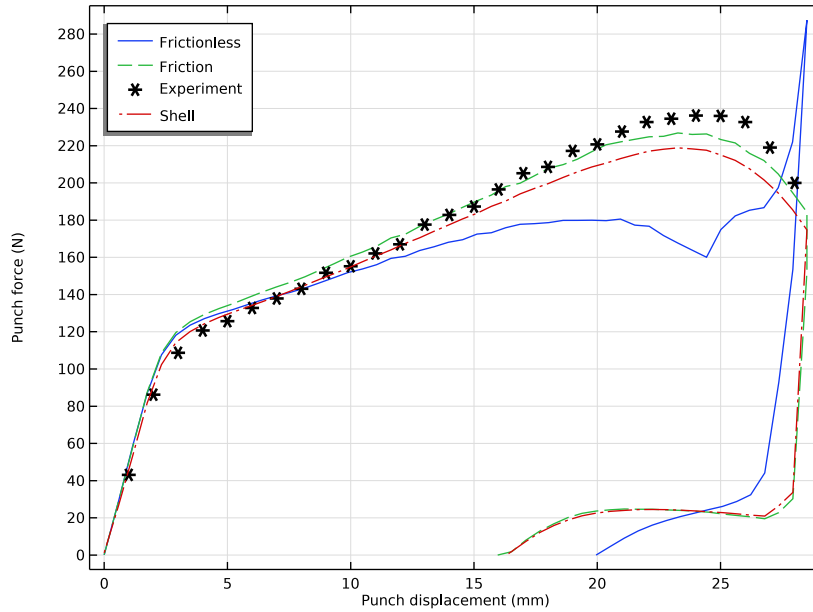


Figure 7: Punch force vs. punch displacement.

In the model without friction the computed forces in the forming process have two peaks. The forces keep on increasing to get the blank within the die; when the blank is sufficiently deformed, it requires a lower force to push down the blank in die. Just before the punch reaches the forming shape, the blank touches the bottom of the die and the force increases

significantly to finish the forming step. In the release step, the punch forces keep decreasing with the punch going back to its original position.

When friction is added, the applied load history differs significantly. First of all, it requires higher loads to get the blank into the die. Secondly, only one peak is observed during the forming step, since the blank never touches the bottom of the die. During the release step, a maximum is also observed, which is explained by the friction force.

Figure 8 shows the position of the blank in the die for the frictionless problem. On the left side, the punch goes into the die (from top to bottom), this is the forming stage. On the right side, you can see the release stage when the punch is removed from the die (from top to bottom).

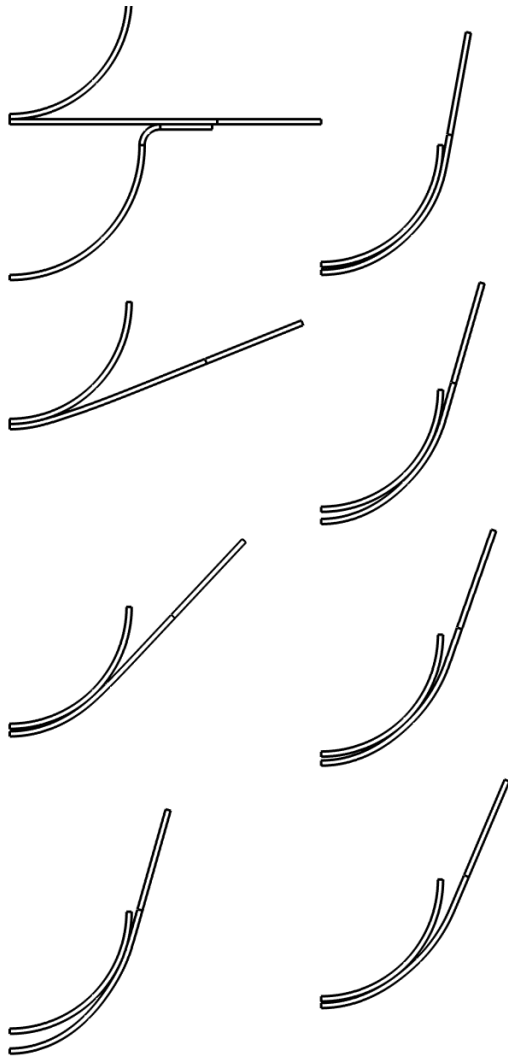


Figure 8: Deformed shape of the blank in the die (without friction).

Figure 9 shows the position of the blank in the die for the problem including friction. On the left side the punch goes into the die (from top to bottom). On the right side you can see the release stage when the punch is removed from the die (from top to bottom).

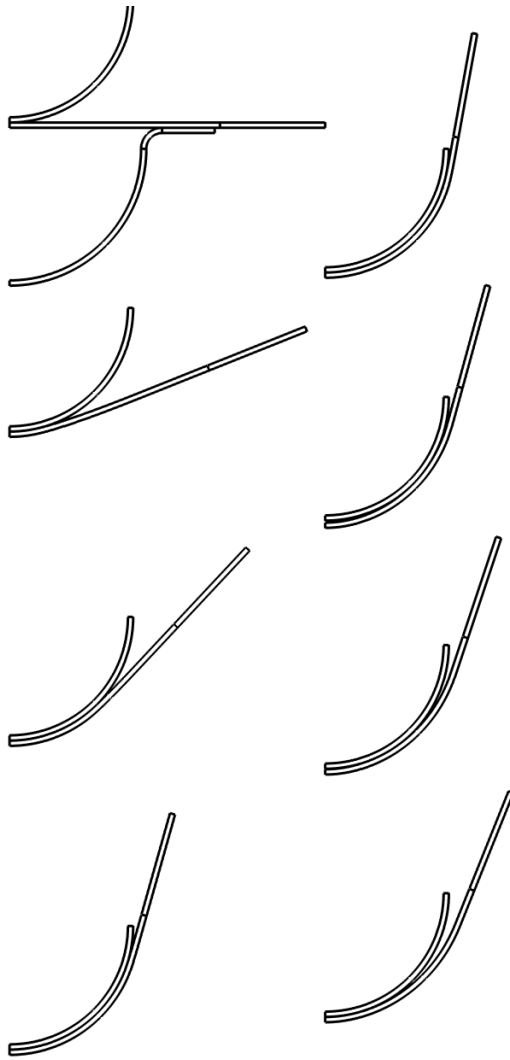


Figure 9: Deformed shape of the blank in the die (including friction).

Notes About the COMSOL Implementation

In the benchmark problem as described in [Ref. 1](#), the plastic hardening is represented with the Hollomon law. Hollomon hardening is a good representation of material hardening for large strains. In the small strain region, however, the computed stress is negative. To

ensure the continuity between the elastic and the hardening material behavior, you use the tangent of the Hollomon curve to match no stress hardening at zero equivalent plastic strain.

Figure 10 shows the difference between the Hollomon hardening with the parameters provided in the model description and the hardening function including the smoothed transition used to implement the numerical model.

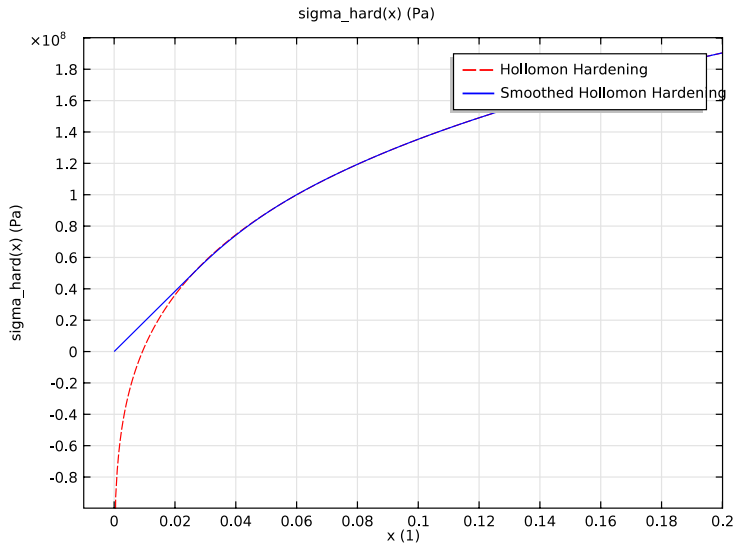


Figure 10: Hollomon hardening with smooth transition (blue) and without (dashed red).

Due to the combination of contact with friction, the elastoplastic material model, and geometric nonlinearity, the model requires manual contact and solver settings to obtain good convergence. Furthermore, the blank becomes unstable in the tool for certain deformation. To circumvent numerical problems, follow the suggestions below:

- Use manual tuning to define the penalty factor and disable relaxation to ensure a constant normal penalty factor. The default settings use softer penalty factor at every initial segregated iteration. Here use a constant value to keep the same stiffness between the initial guess and the computed solution. Using the penalty factor multiplier you can influence the convergence rate, a low value of the penalty factor multiplier improve the convergence at the cost of extra computation time, while a higher value speed up the computation. For this problem different values for the penalty factor multiplier are used: in the 2D case a value of 0.1 for the contact between the punch and the sheet and

0.02 for the contact between the die and the sheet. For the 3D case, the penalty factor for the contact between the punch and the sheet is set to 0.05.

Reference


1. A.W.A. Konter, *Advanced Finite Element Contact Benchmarks*, NAFEMS, 2006.

Application Library path: Nonlinear_Structural_Materials_Module/
Plasticity/sheet_metal_forming




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

Load all model parameters from a file containing parameters for the geometry and some material properties.

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 Click  **Load from File**.
- 4 Browse to the model's Application Libraries folder and double-click the file `sheet_metal_forming_parameters.txt`.

PART I

- 1 In the **Model Builder** window, right-click **Global Definitions** and choose **Geometry Parts > 2D Part**.

Now build a 2D section of the model geometry, starting with the blank geometry.


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 L.
- 4 In the **Height** text field, type th.

Add an extra domain to get better mesh control in the blank.


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

Layer name	Thickness (m)
Layer 1	40 [mm]

- 6 Select the **Layers to the left** checkbox.
- 7 Clear the **Layers on bottom** checkbox.
- 8 Click  **Build Selected**.

Continue with the punch geometry.

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 R1.
- 4 In the **Sector angle** text field, type 90.
- 5 Locate the **Position** section. In the **y** text field, type R1+th.
- 6 Locate the **Rotation Angle** section. In the **Rotation** text field, type -90.
- 7 Click to expand the **Layers** section. In the table, enter the following settings:

Layer name	Thickness (m)
Layer 1	th

- 8 Click  **Build Selected**.

Finally, draw the die geometry.


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 R2+th.
- 4 In the **Sector angle** text field, type 90.
- 5 Locate the **Position** section. In the **y** text field, type -R3.
- 6 Locate the **Rotation Angle** section. In the **Rotation** text field, type -90.
- 7 Locate the **Layers** section. In the table, enter the following settings:

Layer name	Thickness (m)
Layer 1	th

- 8 Click  **Build Selected**.


Circle 3 (c3)

- 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 R3.
- 4 In the **Sector angle** text field, type 90.
- 5 Locate the **Position** section. In the **x** text field, type R2+R3.
- 6 In the **y** text field, type -R3.
- 7 Locate the **Rotation Angle** section. In the **Rotation** text field, type 90.
- 8 Locate the **Layers** section. In the table, enter the following settings:

Layer name	Thickness (m)
Layer 1	th


- 9 Click  **Build Selected**.

Rectangle 2 (r2)


- 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 1[cm].
- 4 In the **Height** text field, type th.
- 5 Locate the **Position** section. In the **x** text field, type R2+R3.
- 6 In the **y** text field, type -th.

7 Click  **Build Selected**.

Delete Entities 1 (dell)

- 1 In the **Model Builder** window, right-click **Part 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 On the object **c2**, select Domain 1 only.
- 6 On the object **c3**, select Domain 1 only.
- 7 Click  **Build Selected**.

Union 1 (un1)


- 1 In the **Geometry** toolbar, click  **Booleans and Partitions** and choose **Union**.
- 2 Select the objects **dell(2)**, **dell(3)**, and **r2** only.

GEOMETRY 1

Part Instance 1 (pi1)

- 1 In the **Geometry** toolbar, click  **Part Instance** and choose **Part 1**.
- 2 In the **Settings** window for **Part Instance**, 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 Clear the **Create pairs** checkbox.
- 5 Click  **Build Selected**.

DEFINITIONS



You can now add contact pairs to define on which boundary you expect the contact to happen.

Contact Pair 1 (cp1)

- 1 In the **Definitions** toolbar, click  **Pairs** and choose **Contact Pair**.
- 2 Select Boundary 20 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 13 only.


Contact Pair 2 (p2)

- 1 In the **Definitions** toolbar, click  **Pairs** and choose **Contact Pair**.
- 2 Select Boundaries 5, 8, and 9 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 12 only.

GLOBAL DEFINITIONS

To define the punch displacement for the loading and the unloading stages, use an interpolation function that makes the displacement a function of a monotonic parameter.

Interpolation 1 (int1)

- 1 In the **Home** toolbar, click  **Functions** and choose **Global > Interpolation**.
- 2 In the **Settings** window for **Interpolation**, locate the **Definition** section.
- 3 In the **Function name** text field, type punch.
- 4 In the table, enter the following settings:

t	f(t)
0	0
0.98	-punch_max
1	-punch_max
2	0

- 5 Locate the **Units** section. In the **Function** table, enter the following settings:


Function	Unit
punch	mm

- 6 In the **Argument** table, enter the following settings:

Argument	Unit
t	1

ADD MATERIAL

- 1 In the **Materials** toolbar, click  **Add Material** to open the **Add Material** window.

- 2 Go to the **Add Material** window.
- 3 In the tree, select **Built-in > Structural steel**.
- 4 Right-click and choose **Add to Global Materials**.
- 5 In the **Materials** toolbar, click  **Add Material** to close the **Add Material** window.

GLOBAL DEFINITIONS

Material 2 (mat2)

In the **Model Builder** window, under **Global Definitions** right-click **Materials** and choose **Blank Material**.

MATERIALS

Material Link 1 (matLnk1)

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

Material Link 2 (matLnk2)

- 1 Right-click **Materials** and choose **More Materials > Material Link**.
- 2 Select Domains 4 and 5 only.
- 3 In the **Settings** window for **Material Link**, locate the **Link Settings** section.
- 4 From the **Material** list, choose **Material 2 (mat2)**.

GLOBAL DEFINITIONS

Material 2 (mat2)

- 1 In the **Model Builder** window, under **Global Definitions > Materials** click **Material 2 (mat2)**.
- 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	70.5 [kN/mm ²]	Pa	Young's modulus and Poisson's ratio
Poisson's ratio	nu	0.342	l	Young's modulus and Poisson's ratio
Density	rho	2700 [kg/m ³]	kg/m ³	Basic

4 Click to expand the **Material Properties** section. In the **Material properties** tree, select **Solid Mechanics > Elastoplastic Material > Elastoplastic Material Model > Initial yield stress (sigmags)**.

5 Click **+ Add to Material**.

6 Locate the **Material Contents** section. In the table, enter the following settings:

Property	Variable	Value	Unit	Property group
Initial yield stress	sigmags	sigma0	Pa	Elastoplastic material model

Use a piecewise constant function to define the hardening function with a smooth transition. At small equivalent plastic strain (lower than 3%) use a linear function; at higher values, use the Hollomon hardening function.

Piecewise 1 (pw1)

1 In the **Model Builder** window, expand the **Material 2 (mat2)** node.

2 Right-click **Global Definitions > Materials > Material 2 (mat2) > Elastoplastic material model (ElastoplasticModel)** and choose **Functions > Piecewise**.

3 In the **Settings** window for **Piecewise**, type `sigma_hard` in the **Function name** text field.

4 Locate the **Definition** section. From the **Extrapolation** list, choose **None**.

5 From the **Smoothing** list, choose **Continuous first derivative**.

6 Find the **Intervals** subsection. In the table, enter the following settings:

Start	End	Function
0	eps0	$(K \cdot \text{eps0}^n - \text{sigma0}) / \text{eps0} \cdot x$
eps0	0.5	$K \cdot x^n - \text{sigma0}$

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

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

9 Locate the **Definition** section. From the **Extrapolation** list, choose **Nearest function**.

Material 2 (mat2)

1 In the **Model Builder** window, under **Global Definitions > Materials > Material 2 (mat2)** 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, type `plastic strain` in the text field.

- 5 In the tree, select **Solid Mechanics > Equivalent plastic strain (1)**.
- 6 Click **OK**.
- 7 In the **Model Builder** window, click **Material 2 (mat2)**.
- 8 In the **Settings** window for **Material**, locate the **Material Contents** section.
- 9 In the table, enter the following settings:

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


SOLID MECHANICS (SOLID)

- 1 In the **Model Builder** window, under **Component 1 (comp1)** click **Solid Mechanics (solid)**.
- 2 Select Domains 4–6 only.
- 3 In the **Settings** window for **Solid Mechanics**, locate the **Thickness** section.
- 4 In the d text field, type w_{sheet} .


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 Domains 4 and 5 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**.

Symmetry 1

- 1 In the **Physics** toolbar, click  **Boundaries** and choose **Symmetry**.
- 2 Select Boundaries 1, 11, and 18 only.

Prescribed Displacement 1

- 1 In the **Physics** toolbar, click  **Domains** and choose **Prescribed Displacement**.
- 2 Select Domain 6 only.
- 3 In the **Settings** window for **Prescribed Displacement**, locate the **Prescribed Displacement** section.
- 4 From the **Displacement in x direction** list, choose **Prescribed**.
- 5 From the **Displacement in y direction** list, choose **Prescribed**.

6 In the u_{0y} text field, type punch(para).

For such a problem (using continuation parameter), a constant penalty factor is preferred. Set it with a lower value than the default to improve the stability.

Contact 1


- 1 In the **Model Builder** window, click **Contact 1**.
- 2 In the **Settings** window for **Contact**, locate the **Contact Method** section.
- 3 From the list, choose **Augmented Lagrangian**.
- 4 Locate the **Contact Pressure Penalty Factor** section. From the **Penalty factor control** list, choose **Manual tuning**.
- 5 In the f_p text field, type 0.1.
- 6 From the **Use relaxation** list, choose **Never**.

Contact 2

- 1 Right-click **Component 1 (comp1) > Solid Mechanics (solid) > Contact 1** and choose **Duplicate**.
- 2 In the **Settings** window for **Contact**, locate the **Pair Selection** section.
- 3 From the **Pairs** list, choose **From list**.
- 4 Click **+ Add**.
- 5 In the **Add** dialog, select **Contact Pair 2 (p2)** in the **Pairs** list.
- 6 Click **OK**.
- 7 In the **Settings** window for **Contact**, locate the **Contact Pressure Penalty Factor** section.
- 8 In the f_p text field, type 2e-2.

MESH 1

Mapped 1

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

Distribution 1

- 1 Right-click **Mapped 1** and choose **Distribution**.
- 2 Select Boundary 13 only.
- 3 In the **Settings** window for **Distribution**, locate the **Distribution** section.
- 4 In the **Number of elements** text field, type 50.

Distribution 2-5

1 Proceed to add four additional distributions with the following settings:

Name	Boundary	Number of elements
Distribution 2	1, 18	1
Distribution 3	8 20	25
Distribution 4	11	5
Distribution 5	9	10

2 Click  **Build All**.

DEFINITIONS

The blank deformation angle is the slope of the line between the points $(4e-2, 0)$ and $(6e-2, 0)$. Use integration coupling variables to evaluate the spatial coordinates at these points.

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 **Point**.

4 Select Point 11 only.

Integration 2 (intop2)


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 **Point**.

4 Select Point 13 only.

Variables 1

1 In the **Definitions** toolbar, click  **Local Variables**.

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

3 In the table, enter the following settings:

Name	Expression	Unit	Description
angle	$2 * \text{atan}((\text{intop2}(x) - \text{intop1}(x)) / (\text{intop2}(y) - \text{intop1}(y)))$	rad	Blank deformation angle

FRICTIONLESS

1 In the **Model Builder** window, click **Study 1**.

2 In the **Settings** window for **Study**, type Frictionless in the **Label** text field.

Step 1: Stationary

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

2 In the **Settings** window for **Stationary**, click to expand the **Results While Solving** section.

3 Select the **Plot** checkbox.

Set up an auxiliary continuation sweep for the para parameter.


4 Click to expand the **Study Extensions** section. Select the **Auxiliary sweep** checkbox.

5 Click **+ Add**.

6 In the table, enter the following settings:

Parameter name	Parameter value list	Parameter unit
para (Punch displacement parameter)	1e-4 range (2e-2, 2e-2, 1.5)	

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 **Frictionless > Solver Configurations > Solution 1 (sol1) > Stationary Solver 1** node, then click **Parametric 1**.

4 In the **Settings** window for **Parametric**, click to expand the **Continuation** section.

5 Select the **Tuning of step size** checkbox.

6 In the **Initial step size** text field, type 1e-4.

7 In the **Minimum step size** text field, type 1e-4.

8 In the **Maximum step size** text field, type 2e-2.

9 From the **Predictor** list, choose **Linear**.

10 Right-click **Frictionless > Solver Configurations > Solution 1 (sol1) > Stationary Solver 1 > Parametric 1** and choose **Stop Condition**.

11 In the **Settings** window for **Stop Condition**, locate the **Stop Expressions** section.

12 Click **+ Add**.

B In the table, enter the following settings:

Stop expression	Stop if	Active	Description
min(comp1.solid.Tnm ax_p1, comp1.solid.Tnmax_p 2) - (para>1)	Negative (<0)	√	Stop expression 1

14 Locate the **Output at Stop** section. From the **Add solution** list, choose **Step after stop**.

15 Clear the **Add information** checkbox.


16 Click  **Run**.

RESULTS

Stress (Frictionless)

The default plot shows the von Mises stress, equivalent plastic strain, and contact pressure after the release. For better clarity, disable the equivalent plastic strain, which will be plotted separately.

1 In the **Settings** window for **2D Plot Group**, type **Stress (Frictionless)** in the **Label** text field.

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

RESULT TEMPLATES

1 In the **Results** toolbar, click  **Result Templates** to open the **Result Templates** window.

2 Go to the **Result Templates** window.

3 In the tree, select **Frictionless/Solution 1 (sol1) > Solid Mechanics > Equivalent Plastic Strain (solid)**.

4 Click the **Add Result Template** button in the window toolbar.

RESULTS

Deformation I

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

2 Right-click **Surface 1** and choose **Deformation**.

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

4 Select the **Scale factor** checkbox. In the associated text field, type 1.

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

Evaluate the forming angle and the angle after release.

The instruction below shows how to evaluate the forming angle and the angle after release.

Deformation Angle (Frictionless)

- 1 In the **Model Builder** window, expand the **Results > Datasets** node.
- 2 Right-click **Results > Derived Values** and choose **Global Evaluation**.
- 3 In the **Settings** window for **Global Evaluation**, type **Deformation Angle (Frictionless)** in the **Label** text field.
- 4 Locate the **Data** section. From the **Parameter selection (para)** list, choose **From list**.
- 5 In the **Parameter values (para)** list, choose **I** and **I.3**.
- 6 Click **Replace Expression** in the upper-right corner of the **Expressions** section. From the menu, choose **Component I (comp1) > Definitions > Variables > angle - Blank deformation angle - rad**.
- 7 Locate the **Expressions** section. In the table, enter the following settings:

Expression	Unit	Description
angle	deg	Blank deformation angle


- 8 Click  **Evaluate**.

Deformation Angle (Frictionless)

- 1 In the **Model Builder** window, expand the **Results > Tables** node, then click **Table I**.
- 2 In the **Settings** window for **Table**, type **Deformation Angle (Frictionless)** in the **Label** text field.

Now plot the punch forces versus the punch displacement as in [Figure 7](#).

Punch Force vs. Displacement

- 1 In the **Results** toolbar, click  **ID Plot Group**.
- 2 In the **Settings** window for **ID Plot Group**, type **Punch Force vs. Displacement** in the **Label** text field.
- 3 Locate the **Data** section. From the **Dataset** list, choose **None**.
- 4 Click to expand the **Title** section. From the **Title type** list, choose **None**.
- 5 Locate the **Plot Settings** section.
- 6 Select the **x-axis label** checkbox. In the associated text field, type **Punch displacement (mm)**.
- 7 Select the **y-axis label** checkbox. In the associated text field, type **Punch force (N)**.
- 8 Locate the **Legend** section. From the **Position** list, choose **Upper left**.

Global 1

- 1 Right-click **Punch Force vs. Displacement** and choose **Global**.
- 2 In the **Settings** window for **Global**, locate the **Data** section.
- 3 From the **Dataset** list, choose **Frictionless/Solution 1 (sol1)**.
- 4 Locate the **y-Axis Data** section. In the table, enter the following settings:

Expression	Unit	Description
2*solid.dcnt2.T_toty	N	

- 5 Locate the **x-Axis Data** section. From the **Parameter** list, choose **Expression**.
- 6 In the **Expression** text field, type -punch(para).
- 7 From the **Unit** list, choose **mm**.
- 8 Click to expand the **Legends** section. From the **Legends** list, choose **Manual**.
- 9 In the table, enter the following settings:

Legends
Frictionless

- 10 In the **Punch Force vs. Displacement** toolbar, click  **Plot**.


Now include friction in the problem.

SOLID MECHANICS (SOLID)

Contact 1

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

Friction 1

- 1 In the **Physics** toolbar, click  **Attributes** and choose **Friction**.
- 2 In the **Settings** window for **Friction**, locate the **Friction Parameters** section.
- 3 In the μ text field, type 0.1342.

Contact 2



In the **Model Builder** window, under **Component 1 (comp1) > Solid Mechanics (solid)** click **Contact 2**.

Friction 1

- 1 In the **Physics** toolbar, click  **Attributes** and choose **Friction**.
- 2 In the **Settings** window for **Friction**, locate the **Friction Parameters** section.

3 In the μ text field, type 0.1342.

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 > Stationary**.
- 4 Click the **Add Study** button in the window toolbar.
- 5 In the **Home** toolbar, click  **Add Study** to close the **Add Study** window.


FRICTION

In the **Settings** window for **Study**, type Friction in the **Label** text field.

Step 1: Stationary


- 1 In the **Model Builder** window, under **Friction** click **Step 1: Stationary**.
- 2 In the **Settings** window for **Stationary**, locate the **Results While Solving** section.
- 3 Select the **Plot** checkbox.
- 4 In the table, enter the following settings:


Plot group	Plot window
Default	Graphics

- 5 Locate the **Study Extensions** section. Select the **Auxiliary sweep** checkbox.
- 6 Click  **Add**.
- 7 In the table, enter the following settings:


Parameter name	Parameter value list	Parameter unit
para (Punch displacement parameter)	1e-4 range (2e-2, 2e-2, 1.5)	

Solution 2 (sol2)

- 1 In the **Study** toolbar, click  **Show Default Solver**.
- 2 In the **Model Builder** window, expand the **Solution 2 (sol2)** node.
- 3 In the **Model Builder** window, expand the **Friction > Solver Configurations > Solution 2 (sol2) > Stationary Solver 1** node, then click **Parametric 1**.
- 4 In the **Settings** window for **Parametric**, locate the **Continuation** section.
- 5 Select the **Tuning of step size** checkbox.
- 6 In the **Initial step size** text field, type 2e-2.

- 7 In the **Minimum step size** text field, type 1e-5.
- 8 In the **Maximum step size** text field, type 2e-2.
- 9 From the **Predictor** list, choose **Linear**.
- 10 Right-click **Friction** > **Solver Configurations** > **Solution 2 (sol2)** > **Stationary Solver 1** > **Parametric 1** and choose **Stop Condition**.
- 11 In the **Settings** window for **Stop Condition**, locate the **Stop Expressions** section.
- 12 Click  **Add**.
- 13 In the table, enter the following settings:

Stop expression	Stop if	Active	Description
$\min(\text{comp1.solid.Tnm}_{ax_p1}, \text{comp1.solid.Tnmax_p2}) - (\text{para} > 1)$	Negative (<0)	<input checked="" type="checkbox"/>	Stop expression 1


- 14 Locate the **Output at Stop** section. From the **Add solution** list, choose **Step after stop**.
- 15 Clear the **Add information** checkbox.
- 16 In the **Study** toolbar, click  **Show Default Plots**.

Step 1: Stationary

Click  **Compute**.

RESULTS

Stress (Friction)

- 1 In the **Settings** window for **2D Plot Group**, type **Stress (Friction)** in the **Label** text field.
- 2 In the **Stress (Friction)** toolbar, click  **Plot**.


RESULT TEMPLATES

- 1 Go to the **Result Templates** window.
- 2 In the tree, select **Friction/Solution 2 (sol2)** > **Solid Mechanics** > **Equivalent Plastic Strain (solid)**.
- 3 Click the **Add Result Template** button in the window toolbar.

RESULTS



Deformation 1

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

- 2 Right-click **Surface 1** and choose **Deformation**.
- 3 In the **Settings** window for **Deformation**, locate the **Scale** section.
- 4 Select the **Scale factor** checkbox. In the associated text field, type 1.
- 5 In the **Equivalent Plastic Strain (solid) 1** toolbar, click  **Plot**.

You can now evaluate the angle at the forming and after-release stages.

Deformation Angle (Friction)

- 1 In the **Model Builder** window, right-click **Deformation Angle (Frictionless)** and choose **Duplicate**.
- 2 In the **Settings** window for **Global Evaluation**, type Deformation Angle (Friction) in the **Label** text field.
- 3 Locate the **Data** section. From the **Dataset** list, choose **Friction/Solution 2 (sol2)**.
- 4 In the **Parameter values (para)** list, choose **1** and **1.42**.
- 5 Click  next to  **Evaluate**, then choose **New Table**.

Deformation Angle (Friction)

- 1 In the **Model Builder** window, under **Results > Tables** click **Table 2**.
- 2 In the **Settings** window for **Table**, type Deformation Angle (Friction) in the **Label** text field.

Global 2

- 1 In the **Model Builder** window, under **Results > Punch Force vs. Displacement** right-click **Global 1** and choose **Duplicate**.
- 2 In the **Settings** window for **Global**, locate the **Data** section.
- 3 From the **Dataset** list, choose **Friction/Solution 2 (sol2)**.
- 4 Click to expand the **Coloring and Style** section. Find the **Line style** subsection. From the **Line** list, choose **Dashed**.
- 5 Locate the **Legends** section. In the table, enter the following settings:


Legends


Friction

- 6 In the **Punch Force vs. Displacement** toolbar, click  **Plot**.

Add a table in which to store the benchmark experimental data.

Punch Force (Experiment)

- 1 In the **Results** toolbar, click  **Table**.

- 2 In the **Settings** window for **Table**, type Punch Force (Experiment) in the **Label** text field.
- 3 Locate the **Data** section. Click  **Import**.
- 4 Browse to the model's Application Libraries folder and double-click the file sheet_metal_forming.txt.

Experiment

- 1 In the **Model Builder** window, right-click **Punch Force vs. Displacement** and choose **Table Graph**.
- 2 In the **Settings** window for **Table Graph**, type Experiment in the **Label** text field.
- 3 Locate the **Data** section. From the **Table** list, choose **Punch Force (Experiment)**.
- 4 Locate the **Coloring and Style** section. Find the **Line style** subsection. From the **Line** list, choose **None**.
- 5 From the **Color** list, choose **From theme**.
- 6 Find the **Line markers** subsection. From the **Marker** list, choose **Cycle**.
- 7 Click to expand the **Legends** section. Select the **Show legends** checkbox.
- 8 From the **Legends** list, choose **Manual**.
- 9 In the table, enter the following settings:

Legends
Experiment

- 10 In the **Punch Force vs. Displacement** toolbar, click  **Plot**.

Friction is now included in the model. To compute the solution without friction in the first study again, you need to make sure that the friction nodes are disabled in the model for this specific study. This is convenient if you need to close the model and reopen it later.

FRICITIONLESS

Step 1: Stationary

- 1 In the **Model Builder** window, under **Frictionless** click **Step 1: Stationary**.
- 2 In the **Settings** window for **Stationary**, locate the **Physics and Variables Selection** section.
- 3 Select the **Modify model configuration for study step** checkbox.
- 4 In the tree, select **Component 1 (comp1) > Solid Mechanics (solid), Controls spatial frame > Contact 1 > Friction 1**.
- 5 Right-click and choose **Disable**.

6 In the tree, select **Component 1 (comp1) > Solid Mechanics (solid), Controls spatial frame > Contact 2 > Friction 1**.


7 Right-click and choose **Disable**.

ADD COMPONENT

In the **Model Builder** window, right-click the root node and choose **Add Component > 3D**.

GEOMETRY 2

Work Plane 1 (wp1)

1 In the **Geometry** toolbar, click  **Work Plane**.

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

3 From the **Plane** list, choose **xz-plane**.

4 Locate the **Unite Objects** section. Clear the **Unite objects** checkbox.


Work Plane 1 (wp1) > Plane Geometry

In the **Model Builder** window, click **Plane Geometry**.

Work Plane 1 (wp1) > Part Instance 1 (pi1)

1 In the **Work Plane** toolbar, click  **Part Instance** and choose **Part 1**.

2 In the **Settings** window for **Part Instance**, click  **Build Selected**.

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

Extrude 1 (ext1)

1 In the **Model Builder** window, under **Component 2 (comp2) > Geometry 2** right-click **Work Plane 1 (wp1)** and choose **Extrude**.

2 Select the object **wp1.pi1(1)** only.

3 In the **Settings** window for **Extrude**, locate the **Distances** section.

4 In the table, enter the following settings:


Distances (m)

w_sheet/2

5 Select the **Reverse direction** checkbox.

6 Click  **Build Selected**.

Extrude 2 (ext2)

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

2 In the **Settings** window for **Extrude**, locate the **Distances** section.

3 In the table, enter the following settings:

Distances (m)
w_tools/2

4 Select the **Reverse direction** checkbox.

5 Click  **Build Selected**.

Form Union (fin)

1 In the **Model Builder** window, under **Component 2 (comp2) > Geometry 2** 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 Clear the **Create pairs** checkbox.

DEFINITIONS (COMP2)

Contact Pair 3 (p3)

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

2 Select Boundary 30 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 19 only.

Contact Pair 4 (p4)

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

2 Select Boundaries 4, 7, and 14 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 19 only.

GLOBAL DEFINITIONS

Material 3 (mat3)

1 In the **Model Builder** window, under **Global Definitions > Materials** right-click **Material 2 (mat2)** and choose **Duplicate**.

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

3 In the **Material properties** tree, select **Geometric Properties > Shell**.

4 Click **+** **Add to Material**.

5 Locate the **Material Contents** section. In the table, enter the following settings:

Property	Variable	Value	Unit	Property group
Isotropic tangent modulus	Et		Pa	Elastoplastic material model
Kinematic tangent modulus	Ek		Pa	Elastoplastic material model
Thickness	lth	th	m	Shell
Density	rho	2700 [kg/m ³]	kg/m ³	Basic
Young's modulus	E	70.5 [kN/mm ²]	Pa	Young's modulus and Poisson's ratio
Poisson's ratio	nu	0.342	l	Young's modulus and Poisson's ratio
Initial yield stress	sigmags	sigma0	Pa	Elastoplastic material model
Hardening function	sigmagh	sigma_hard (epe)	Pa	Elastoplastic material model
Hill's coefficients	{Hillcoefficients1, Hillcoefficients2, Hillcoefficients3, Hillcoefficients4, Hillcoefficients5, Hillcoefficients6}	{0, 0, 0, 0, 0, 0}	m ² ·s ⁴ /kg ²	Elastoplastic material model
Initial tensile and shear yield stresses	{ys1, ys2, ys3, ys4, ys5, ys6}	{0, 0, 0, 0, 0, 0}	N/m ²	Elastoplastic material model

Property	Variable	Value	Unit	Property group
Hill's coefficients, dimensionless	{HillcoefficientsDimensionless1, HillcoefficientsDimensionless2, HillcoefficientsDimensionless3, HillcoefficientsDimensionless4, HillcoefficientsDimensionless5, HillcoefficientsDimensionless6}	{0, 0, 0, 0, 0, 0}	l	Elastoplastic material model
Rotation	lrot	0.0	deg	Shell
Mesh elements	lne	2	l	Shell

MATERIALS

Material Link 3 (matlnk3)

- 1 In the **Model Builder** window, under **Component 2 (comp2)** right-click **Materials** and choose **More Materials > Material Link**.
- 2 In the **Settings** window for **Material Link**, locate the **Geometric Entity Selection** section.
- 3 From the **Geometric entity level** list, choose **Boundary**.
- 4 Select Boundary 30 only.

GLOBAL DEFINITIONS

Material 3 (mat3)



In the **Model Builder** window, under **Global Definitions > Materials** right-click **Material 3 (mat3)** and choose **Copy**.

MATERIALS


Material 3 (mat4)

- 1 In the **Model Builder** window, under **Component 2 (comp2)** right-click **Materials** and choose **Paste Material**.
- 2 In the **Settings** window for **Material**, locate the **Geometric Entity Selection** section.
- 3 From the **Geometric entity level** list, choose **Boundary**.
- 4 Select Boundaries 19 and 24 only.

ADD PHYSICS

- 1 In the **Home** toolbar, click  **Add Physics** to open the **Add Physics** window.
- 2 Go to the **Add Physics** window.
- 3 In the tree, select **Structural Mechanics > Shell (shell)**.
- 4 Find the **Physics interfaces in study** subsection. In the table, clear the **Solve** checkboxes for **Frictionless** and **Friction**.
- 5 Click the **Add to Component 2** button in the window toolbar.
- 6 In the **Home** toolbar, click  **Add Physics** to close the **Add Physics** window.


SHELL (SHELL)

- 1 In the **Settings** window for **Shell**, locate the **Boundary Selection** section.
- 2 Click  **Clear Selection**.
- 3 Select Boundaries 19, 24, and 30 only.

Thickness and Offset 1

- 1 In the **Model Builder** window, under **Component 2 (comp2) > Shell (shell)** click **Thickness and Offset 1**.
- 2 In the **Settings** window for **Thickness and Offset**, locate the **Thickness and Offset** section.
- 3 In the d_0 text field, type th.
- 4 From the **Position** list, choose **Top surface on boundary**.

Linear Elastic Material, Layered 1

- 1 In the **Physics** toolbar, click  **Boundaries** and choose **Linear Elastic Material, Layered**.
- 2 Select Boundaries 19 and 24 only.


MATERIALS

Material 3 (mat4)


- 1 In the **Model Builder** window, under **Component 2 (comp2) > Materials** click **Material 3 (mat4)**.
- 2 In the **Settings** window for **Material**, locate the **Orientation and Position** section.
- 3 From the **Coordinate system** list, choose **Boundary System 2 (sys2)**.
- 4 From the **Position** list, choose **Top side on boundary**.

SHELL (SHELL)


Linear Elastic Material, Layered I

- 1 Click the  **Show More Options** button in the **Model Builder** toolbar.
- 2 In the **Show More Options** dialog, in the tree, select the checkbox for the node **Physics > Advanced Physics Options**.
- 3 Click **OK**.
- 4 In the **Model Builder** window, under **Component 2 (comp2) > Shell (shell)** click **Linear Elastic Material, Layered I**.
- 5 In the **Settings** window for **Linear Elastic Material, Layered**, click to expand the **Out-of-Plane Strain** section.
- 6 Select the **Solve for out-of-plane strain components** checkbox.


Plasticity I

- 1 In the **Physics** toolbar, click  **Attributes** and choose **Plasticity**.
- 2 Select Boundaries 19 and 24 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**.

Prescribed Displacement/Rotation I

- 1 In the **Physics** toolbar, click  **Boundaries** and choose **Prescribed Displacement/Rotation**.
- 2 Select Boundary 30 only.
- 3 In the **Settings** window for **Prescribed Displacement/Rotation**, locate the **Prescribed Displacement** section.
- 4 From the **Displacement in x direction** list, choose **Prescribed**.
- 5 From the **Displacement in y direction** list, choose **Prescribed**.
- 6 From the **Displacement in z direction** list, choose **Prescribed**.
- 7 In the u_{0z} text field, type punch(para).

Symmetry I


- 1 In the **Physics** toolbar, click  **Edges** and choose **Symmetry**.
- 2 Select Edges 30, 31, and 39 only.

Contact I


- 1 In the **Model Builder** window, click **Contact I**.
- 2 In the **Settings** window for **Contact**, locate the **Contact Surface** section.

- 3 From the **Contact surface, destination** list, choose **Bottom**.
- 4 Locate the **Contact Pressure Penalty Factor** section. From the **Penalty factor control** list, choose **Automatic, soft**.

Friction 1


- 1 In the **Physics** toolbar, click  **Attributes** and choose **Friction**.
- 2 In the **Settings** window for **Friction**, locate the **Friction Parameters** section.
- 3 In the μ text field, type 0.1342.
- 4 Locate the **Friction Force Penalty Factor** section. From the **Penalty factor control** list, choose **From parent**.

Contact 2

- 1 Right-click **Contact 1** and choose **Duplicate**.
- 2 In the **Settings** window for **Contact**, locate the **Pair Selection** section.
- 3 From the **Pairs** list, choose **From list**.
- 4 Click  **Add**.
- 5 In the **Add** dialog, select **Contact Pair 4 (p4)** in the **Pairs** list.
- 6 Click **OK**.
- 7 In the **Settings** window for **Contact**, locate the **Contact Surface** section.
- 8 From the **Contact surface, destination** list, choose **Top**.

MESH 2

Mapped 1

- 1 In the **Mesh** toolbar, click  **More Generators** and choose **Mapped**.
- 2 Select Boundaries 4, 7, 14, 19, 24, and 30 only.

Distribution 1

- 1 Right-click **Mapped 1** and choose **Distribution**.
- 2 Select Edges 1, 4, 21, 49, and 50 only.
- 3 In the **Settings** window for **Distribution**, locate the **Distribution** section.
- 4 In the **Number of elements** text field, type 1.

Distribution 2

- 1 In the **Model Builder** window, right-click **Mapped 1** and choose **Distribution**.
- 2 Select Edges 5 and 51 only.
- 3 In the **Settings** window for **Distribution**, locate the **Distribution** section.

4 In the **Number of elements** text field, type 25.


Distribution 3

- 1 Right-click **Mapped I** and choose **Distribution**.
- 2 Select Edge 11 only.
- 3 In the **Settings** window for **Distribution**, locate the **Distribution** section.
- 4 In the **Number of elements** text field, type 10.

Distribution 4


- 1 Right-click **Mapped I** and choose **Distribution**.
- 2 Select Edge 31 only.
- 3 In the **Settings** window for **Distribution**, locate the **Distribution** section.
- 4 In the **Number of elements** text field, type 50.

Distribution 5


- 1 Right-click **Mapped I** and choose **Distribution**.
- 2 Select Edge 30 only.
- 3 In the **Settings** window for **Distribution**, click  **Build Selected**.

DEFINITIONS (COMP2)

Integration 3 (intop3)

- 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 **Point**.
- 4 Select Point 21 only.

Integration 4 (intop4)

- 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 **Point**.
- 4 Select Point 25 only.



Variables 2

- 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
angle_shell	$2 * \text{atan}((\text{intop4}(x) - \text{intop3}(x)) / (\text{intop4}(z) - \text{intop3}(z)))$	rad	Blank deformation angle (shell)

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 > Stationary**.
- 4 Find the **Physics interfaces in study** subsection. In the table, clear the **Solve** checkbox for **Solid Mechanics (solid)**.
- 5 Click the **Add Study** button in the window toolbar.
- 6 In the **Home** toolbar, click  **Add Study** to close the **Add Study** window.

STUDY 3


Step 1: Stationary


- 1 In the **Settings** window for **Stationary**, locate the **Study Extensions** section.
- 2 Select the **Auxiliary sweep** checkbox.
- 3 Click  **Add**.
- 4 In the table, enter the following settings:

Parameter name	Parameter value list	Parameter unit
para (Punch displacement parameter)	1e-4 range (2e-2, 2e-2, 1.5)	


- 5 In the **Model Builder** window, click **Study 3**.
- 6 In the **Settings** window for **Study**, type Friction (shell) in the **Label** text field.

Solution 3 (sol3)

- 1 In the **Study** toolbar, click  **Show Default Solver**.
- 2 In the **Model Builder** window, expand the **Solution 3 (sol3)** node.
- 3 In the **Model Builder** window, expand the **Friction (shell) > Solver Configurations > Solution 3 (sol3) > Stationary Solver 1** node, then click **Parametric 1**.
- 4 In the **Settings** window for **Parametric**, locate the **Continuation** section.

- 5 Select the **Tuning of step size** checkbox.
- 6 In the **Initial step size** text field, type 2e-2.
- 7 In the **Minimum step size** text field, type 1e-5.
- 8 In the **Maximum step size** text field, type 2e-2.
- 9 From the **Predictor** list, choose **Linear**.
- 10 Right-click **Friction (shell)** > **Solver Configurations** > **Solution 3 (sol3)** > **Stationary Solver 1** > **Parametric 1** and choose **Stop Condition**.
- 11 In the **Settings** window for **Stop Condition**, locate the **Stop Expressions** section.
- 12 Click  **Add**.
- 13 In the table, enter the following settings:

Stop expression	Stop if	Active	Description
min(comp2.shell.Tnm ax_p3, comp2.shell.Tnmax_p 4) - (para>1)	Negative (<0)	√	Stop expression 1

- 14 Locate the **Output at Stop** section. From the **Add solution** list, choose **Step after stop**.
- 15 Clear the **Add information** checkbox.
- 16 In the **Study** toolbar, click  **Show Default Plots**.

Step 1: Stationary

- 1 In the **Model Builder** window, under **Friction (shell)** click **Step 1: Stationary**.
- 2 In the **Settings** window for **Stationary**, locate the **Results While Solving** section.
- 3 Select the **Plot** checkbox.
- 4 In the table, enter the following settings:

Plot group	Plot window
Stress (shell)	Graphics


- 5 In the **Study** toolbar, click  **Compute**.

RESULTS


Surface 1

- 1 In the **Model Builder** window, expand the **Stress (shell)** node.
- 2 Right-click **Surface 1** and choose **Delete**.


Stress (shell)

- 1 In the **Model Builder** window, under **Results** click **Stress (shell)**.
- 2 In the **Stress (shell)** toolbar, click  **Plot**.
- 3 In the **Settings** window for **3D Plot Group**, locate the **Data** section.
- 4 From the **Dataset** list, choose **Friction (shell)/Solution 3 (4) (sol3)**.
- 5 From the **Parameter value (para)** list, choose 1.

Surface 2


- 1 In the **Model Builder** window, click **Surface 2**.
- 2 In the **Stress (shell)** toolbar, click  **Plot**.

RESULT TEMPLATES

- 1 Go to the **Result Templates** window.
- 2 In the tree, select **Friction (shell)/Solution 3 (4) (sol3) > Shell > Equivalent Plastic Strain (shell)**.
- 3 Click the **Add Result Template** button in the window toolbar.
- 4 In the **Results** toolbar, click  **Result Templates** to close the **Result Templates** window.

RESULTS

Deformation 1

- 1 In the **Model Builder** window, expand the **Equivalent Plastic Strain (shell)** node.
- 2 Right-click **Surface 1** and choose **Deformation**.
- 3 In the **Settings** window for **Deformation**, locate the **Expression** section.
- 4 In the **x-component** text field, type shell.u.
- 5 In the **y-component** text field, type shell.v.
- 6 In the **z-component** text field, type shell.w.
- 7 Locate the **Scale** section.
- 8 Select the **Scale factor** checkbox. In the associated text field, type 1.
- 9 In the **Equivalent Plastic Strain (shell)** toolbar, click  **Plot**.

Deformation angle (Shell)

- 1 In the **Model Builder** window, right-click **Deformation Angle (Friction)** and choose **Duplicate**.
- 2 In the **Settings** window for **Global Evaluation**, type Deformation angle (Shell) in the **Label** text field.

3 Locate the **Data** section. From the **Dataset** list, choose **Friction (shell)/Solution 3 (4) (sol3)**.

4 In the **Parameter values (para)** list, choose **I** and **I.4**.

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

Expression	Unit	Description
angle_shell	deg	Blank deformation angle (shell)

6 Click  next to  **Evaluate**, then choose **New Table**.

Global 3

1 In the **Model Builder** window, under **Results > Punch Force vs. Displacement** right-click **Global 2** and choose **Duplicate**.

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

3 From the **Dataset** list, choose **Friction (shell)/Solution 3 (4) (sol3)**.

4 Locate the **y-Axis Data** section. In the table, enter the following settings:

Expression	Unit	Description
4*shell.dcmt2.T_totz	N	

5 Locate the **Coloring and Style** section. Find the **Line style** subsection. From the **Line** list, choose **Dash-dot**.

6 Locate the **Legends** section. In the table, enter the following settings:

Legends
Shell

7 In the **Punch Force vs. Displacement** toolbar, click  **Plot**.