Created in COMSOL Multiphysics 6.0

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.

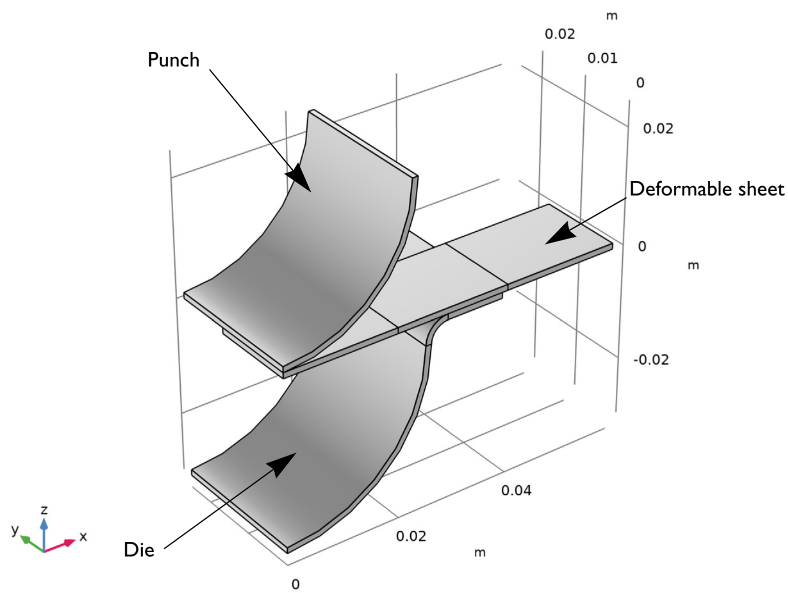

Figure 1: Model geometry: the punch at the top, the thin sheet in the middle, and the die at the bottom.

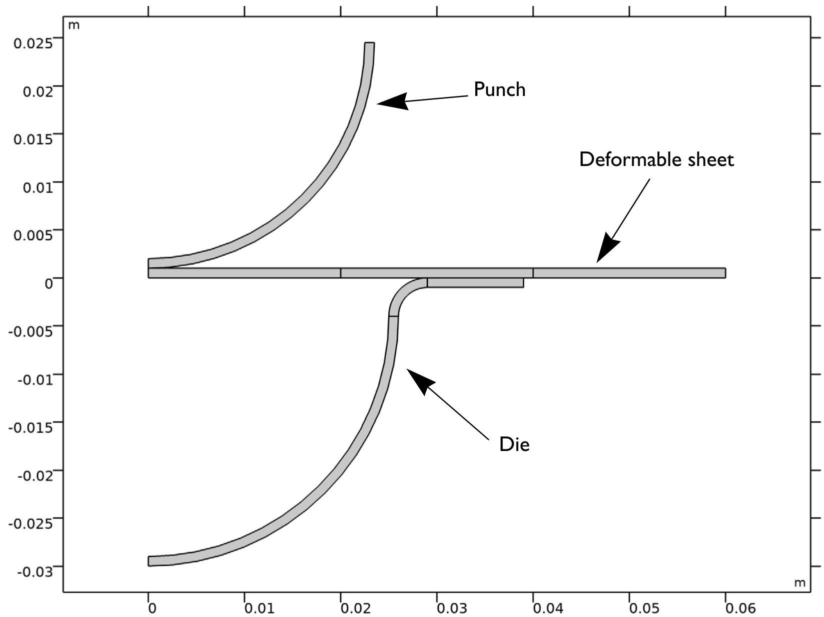

Figure 2: Section cut of the model geometry used for the 2D assumption.

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.

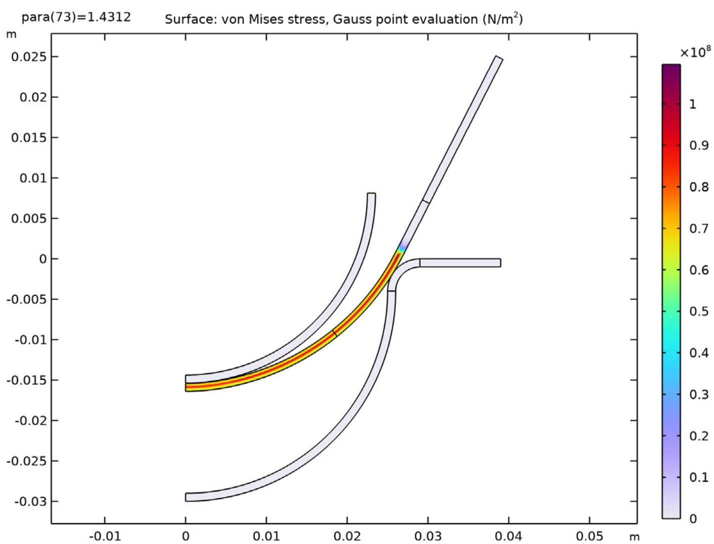

Figure 3: Von Mises stress and deformed shape after release for the 2D plane strain.

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

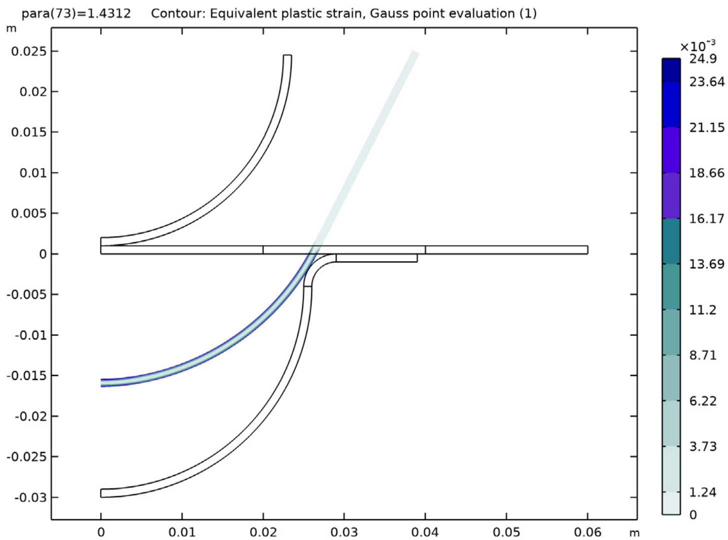

Figure 4: Equivalent plastic strain at maximum punch deflection.

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.

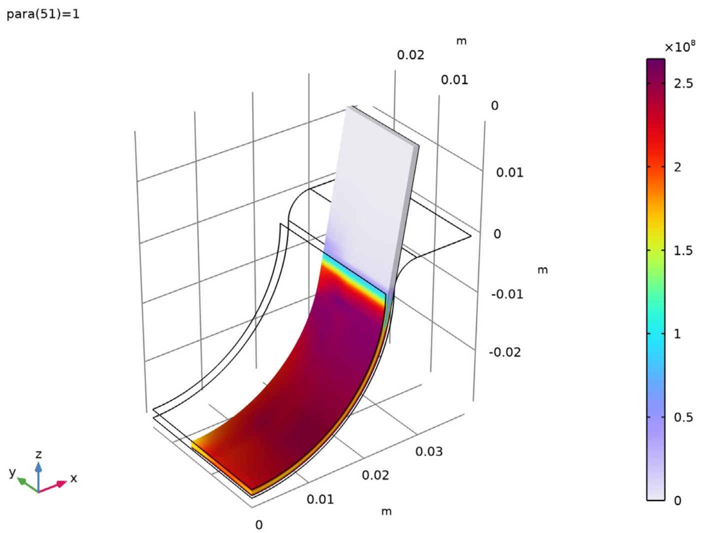

Figure 5: Von Mises stress and deformed shape at forming for the 3D shell problem.

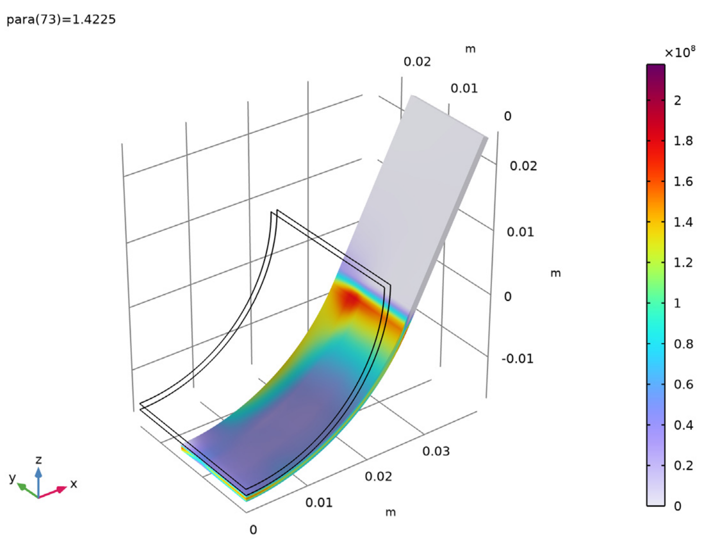

Figure 6: Von Mises stress and deformed shape after release for the 3D shell problem.

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	20.4	45.0
With friction	20.4	54.1
With friction (shell)	20.4	55.2

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.

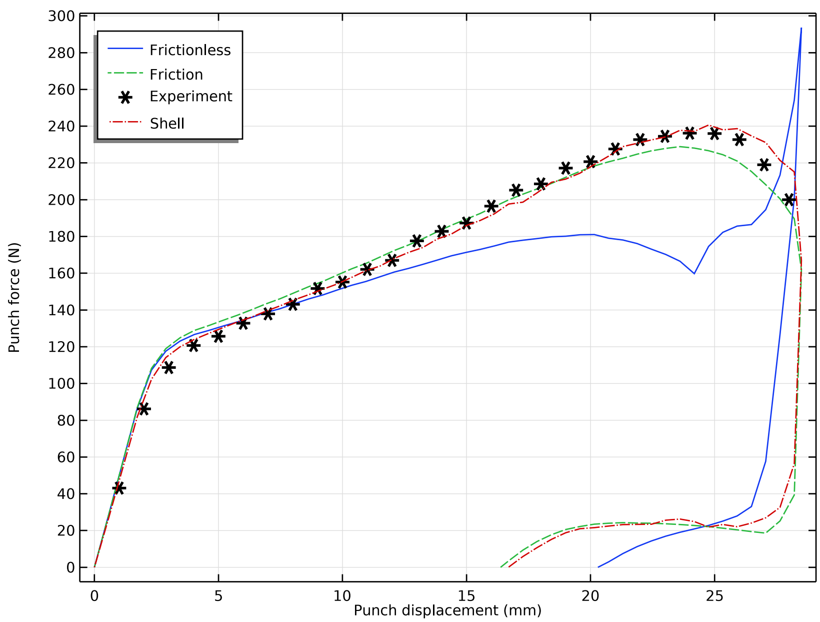

Figure 7: Punch force vs. punch displacement.

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 consists in 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).

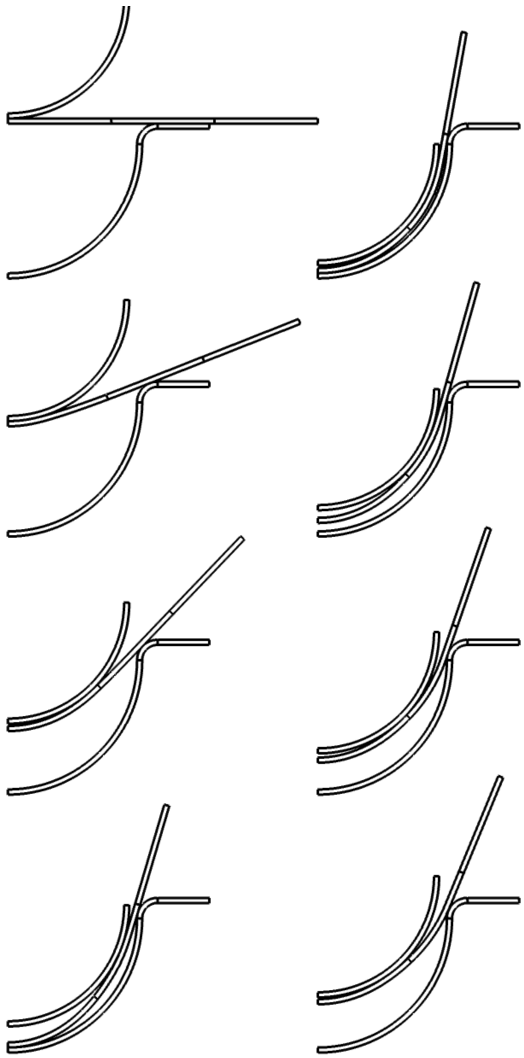

Figure 8: Deformed shape of the blank in the die (without friction)

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).

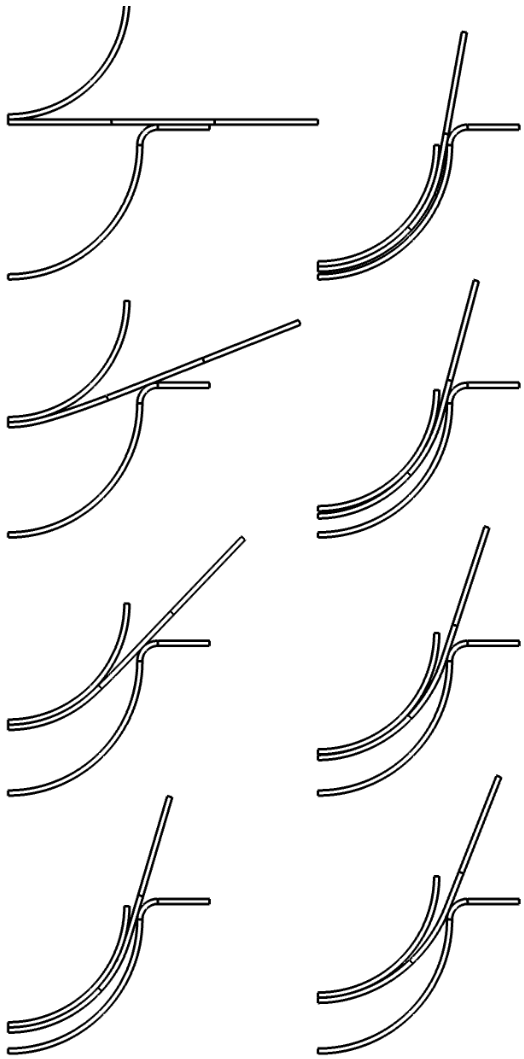

Figure 9: Deformed shape of the blank in the die (including friction)

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.

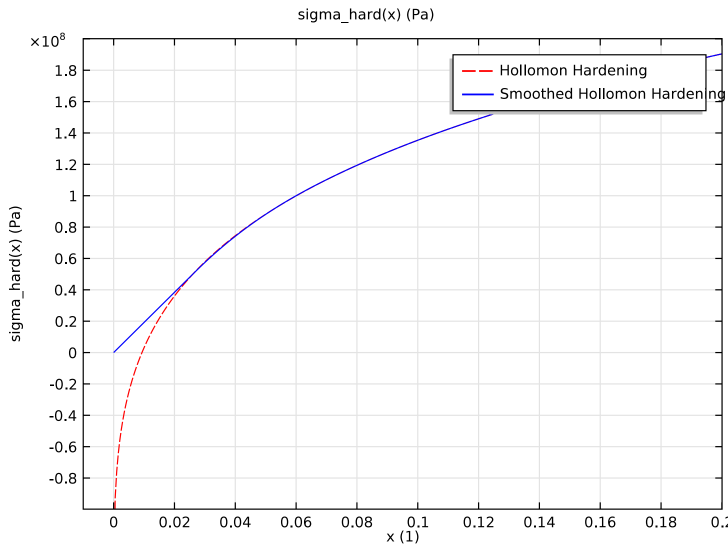

Figure 10: Hollomon hardening with smooth transition (blue) and without (dashed red).

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 a value for the penalty factor multiplier of 0.1 ensure convergence all along the computation. For the 2D case the penalty factor for the contact between the die and the sheet is reduced to 0.05.

- Because of the small curvature in the die, the contact is likely to be on a single point. On the blank the contact position can vary significantly between two consecutive computational steps, on the die, however, the contact region remains around the same position. For this reason use the die boundaries as destination for the contact pair.

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 1

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	20[mm]
Layer 2	20[mm]

6 Select the **Layers to the left** check box.

7 Clear the **Layers on bottom** check box.

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 (del1)

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 (uni1)

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

2 Select the objects **del1(2)**, **del1(3)**, and **r2** only.

GEOMETRY 1

Part Instance 1 (pi1)

1 In the **Geometry** toolbar, click **Parts** 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** check box.

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 (p1)

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

2 Select Boundaries 13 and 16 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 23 only.

Contact Pair 2 (p2)

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

2 Select Boundaries 12 and 15 only.

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

4 Click to select the **Activate Selection** toggle button.

5 Select Boundaries 5, 8, and 9 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 **Definitions** toolbar, click **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
0.99	-punch_max
1.01	-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 **Home** 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 **Home** 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 4–6 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`	1	Young's modulus and Poisson's ratio
Density	rho	`2700[kg/m^3]`	kg/m³	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*eps0^n-sigma0)/eps0*x`
`eps0`	`0.5`	`K*x^n-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 box, type `plastic strain` in the text field.

5 Click **Filter**.

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

7 Click **OK**.

8 In the **Model Builder** window, click **Material 2 (mat2)**.

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

10 In the table, enter the following settings:

Property	**Variable**	**Value**	**Unit**	**Property group**
Hardening function	sigmagh	`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 In the **Settings** window for **Solid Mechanics**, locate the **Thickness** section.

3 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 4–6 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 21 only.

Fixed Constraint 1

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

2 Select Boundaries 4, 7, and 10 only.

Prescribed Displacement 1

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

2 Select Boundary 24 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_{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 **Formulation** 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**.

7 Locate the **Initial Value** section. In the T_n text field, type `1e4`.

Contact 1a

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 2 (p2)** in the **Pairs** list.

5 Click **OK**.

6 In the **Settings** window for **Contact**, locate the **Contact Method** section.

7 From the **Formulation** list, choose **Augmented Lagrangian**.

8 Locate the **Contact Pressure Penalty Factor** section. From the **Penalty factor control** list, choose **Manual tuning**.

9 In the f_p text field, type `5e-2`.

10 From the **Use relaxation** list, choose **Never**.

11 Locate the **Initial Value** section. In the T_n text field, type `1e4`.

MESH 1

Mapped 1

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

Distribution 1

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

2 Select Boundaries 8, 16, and 23 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, 21	1
Distribution 3	13	25
Distribution 4	11	3
Distribution 5	9	20

2 Click **Build All**.

DEFINITIONS

The blank deformation angle is the slope of the line between the points (4e-2, 0) and (5e-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 13 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 15 only.

Variables 1

1 In the **Definitions** toolbar, click a= **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*atan((intop2(x)-intop1(x))/(intop2(y)-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** check box.

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

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

5 Click + **Add**.

6 In the table, enter the following settings:

Parameter name	Parameter value list	Parameter unit
para (Punch displacement parameter)	`range(0,2e-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)>Dependent Variables 1** node, then click **Displacement field (comp1.u)**.

4 In the **Settings** window for **Field**, locate the **Scaling** section.

5 In the **Scale** text field, type `1e-2`.

6 In the **Model Builder** window, expand the **Frictionless>Solver Configurations>Solution 1 (sol1)>Stationary Solver 1** node, then click **Parametric 1**.

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

8 Select the **Tuning of step size** check box.

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

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

11 In the **Maximum step size** text field, type `5e-3`.

Add a nonlocal integration coupling to evaluate the punch force.

DEFINITIONS

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 **Boundary**.

4 Select Boundary 24 only.

5 Locate the **Advanced** section. From the **Method** list, choose **Summation over nodes**.

FRICTIONLESS

Solution 1 (sol1)

1 Right-click **Parametric 1** and choose **Stop Condition**.

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

3 Click **Add**.

4 In the table, enter the following settings:

Stop expression	Stop if	Active	Description
`comp1.solid.Tnmax_p 1-(para>1)`	Negative (<0)	√	`Stop expression 1`

5 Click **Add**.

6 In the table, enter the following settings:

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

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

8 Clear the **Add warning** check box.

9 Click **Compute**.

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 Locate the **Plot Settings** section. From the **Frame** list, choose **Spatial (x, y, z)**.

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

Deformation 1

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

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

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

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

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

6 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 **1** and **1.295**.

6 Click **Replace Expression** in the upper-right corner of the **Expressions** section. From the menu, choose **Component 1 (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 1**.

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 **1D Plot Group**.

2 In the **Settings** window for **1D 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. Select the **x-axis label** check box.

6 In the associated text field, type `Punch displacement (mm)`.

7 Select the **y-axis label** check box.

8 In the associated text field, type `Punch force (N)`.

9 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*intop3(solid.RFy)`	`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 `0.1348`.

4 Locate the **Initial Value** section. From the **Previous contact state** list, choose **In contact**.

Contact 1a

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

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 `0.1348`.

4 Locate the **Initial Value** section. From the **Previous contact state** list, choose **In contact**.

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 **Add Study** in the window toolbar.

5 In the **Home** toolbar, click **Add Study** to close the **Add Study** window.

FRICTION

1 In the **Model Builder** window, click **Study 2**.
2 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** check box.
4 From the **Plot group** list, choose **Default**.
5 Locate the **Study Extensions** section. Select the **Auxiliary sweep** check box.
6 Click **Add**.
7 In the table, enter the following settings:

Parameter name	Parameter value list	Parameter unit
para (Punch displacement parameter)	`range(0,2e-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)>Dependent Variables 1** node, then click **Displacement field (comp1.u)**.
4 In the **Settings** window for **Field**, locate the **Scaling** section.
5 In the **Scale** text field, type `1e-2`.
6 In the **Model Builder** window, expand the **Friction>Solver Configurations>Solution 2 (sol2)>Stationary Solver 1** node, then click **Parametric 1**.
7 In the **Settings** window for **Parametric**, locate the **Continuation** section.
8 Select the **Tuning of step size** check box.
9 In the **Initial step size** text field, type `1e-4`.
10 In the **Minimum step size** text field, type `1e-4`.
11 In the **Maximum step size** text field, type `5e-3`.
12 Right-click **Friction>Solver Configurations>Solution 2 (sol2)>Stationary Solver 1>Parametric 1** and choose **Stop Condition**.
13 In the **Settings** window for **Stop Condition**, locate the **Stop Expressions** section.
14 Click **Add**.

15 In the table, enter the following settings:

Stop expression	Stop if	Active	Description
comp1.solid.Tnmax_p1-(para>1)	Negative (<0)	√	Stop expression 1

16 Click Add.

17 In the table, enter the following settings:

Stop expression	Stop if	Active	Description
comp1.solid.Tnmax_p2-(para>1)	Negative (<0)	√	Stop expression 2

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

19 Clear the **Add warning** check box.

20 In the **Model Builder** window, expand the **Friction>Solver Configurations>Solution 2 (sol2)>Stationary Solver 1>Segregated 1** node, then click **Solid Mechanics**.

21 In the **Settings** window for **Segregated Step**, click to expand the **Method and Termination** section.

22 In the **Initial damping factor** text field, type `0.1`.

23 Click **Compute**.

RESULTS

Stress (Friction)

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

2 Locate the **Plot Settings** section. From the **Frame** list, choose **Spatial (x, y, z)**.

3 In the **Stress (Friction)** toolbar, click **Plot**.

Deformation 1

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

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

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

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

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

Equivalent Plastic Strain (solid) 1

1 In the **Model Builder** window, under **Results** click **Equivalent Plastic Strain (solid) 1**.

2 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 **1.4312**.

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 From the **Positioning** list, choose **In data points**.

8 Click to expand the **Legends** section. Select the **Show legends** check box.

9 From the **Legends** list, choose **Manual**.

10 In the table, enter the following settings:

Legends
`Experiment`

11 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.

FRICTIONLESS

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** check box.

4 In the tree, select **Component 1 (Comp1)>Solid Mechanics (Solid)>Contact 1>Friction 1**.

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

6 In the tree, select **Component 1 (Comp1)>Solid Mechanics (Solid)>Contact 1a>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** check box.

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 **Parts** 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** check box.
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** check box.

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** check box.

DEFINITIONS (COMP2)

Contact Pair 3 (p3)

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

2 Select Boundary 35 only.

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

4 Click to select the **Activate Selection** toggle button.

5 Select Boundaries 19 and 24 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 Boundaries 19 and 24 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**, click to expand 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/m³	Basic
Young's modulus	E	`70.5[kN/mm^2]`	Pa	Young's modulus and Poisson's ratio
Poisson's ratio	nu	`0.342`	1	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}`	$m^2 \cdot s^4/kg^2$	Elastoplastic material model
Initial tensile and shear yield stresses	{ys1, ys2, ys3, ys4, ys5, ys6}	`{0, 0, 0, 0, 0, 0}`	N/m²	Elastoplastic material model
Rotation	lrot	`0.0`	deg	Shell
Mesh elements	lne	2	1	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 35 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, 24, and 29 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** check boxes for **Frictionless** and **Friction**.
5 Click **Add to Component 2** 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, 29, and 35 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* text field, type `th`.
4 From the **Position** list, choose **Top surface on boundary**.

Layered Linear Elastic Material 1

1 In the **Physics** toolbar, click **Boundaries** and choose **Layered Linear Elastic Material**.

2 Select Boundaries 19, 24, and 29 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)

Layered Linear Elastic Material 1

In the **Model Builder** window, under **Component 2 (comp2)>Shell (shell)** click **Layered Linear Elastic Material 1**.

Plasticity 1

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 1

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

2 Select Boundary 35 only.

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

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

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

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

7 In the u_{0z} text field, type `punch(para)`.

Symmetry 1

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

2 Select Edges 30, 31, 39, and 47 only.

Contact 1

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

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 Method** section. From the **Formulation** list, choose **Augmented Lagrangian**.

5 Locate the **Contact Pressure Penalty Factor** section. From the **Penalty factor control** list, choose **Manual tuning**.

6 In the f_p text field, type `0.1`.

7 From the **Use relaxation** list, choose **Never**.

8 Locate the **Initial Value** section. In the T_n text field, type `1e4`.

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 **Initial Value** section. From the **Previous contact state** list, choose **In contact**.

Contact 1a

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 4 (p4)** in the **Pairs** list.

5 Click **OK**.

6 In the **Settings** window for **Contact**, locate the **Contact Method** section.

7 From the **Formulation** list, choose **Augmented Lagrangian**.

8 Locate the **Contact Pressure Penalty Factor** section. From the **Penalty factor control** list, choose **Manual tuning**.

9 In the f_p text field, type `0.1`.

10 From the **Use relaxation** list, choose **Never**.

11 Locate the **Initial Value** section. In the T_n text field, type `1e4`.

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 **Initial Value** section. From the **Previous contact state** list, choose **In contact**.

MESH 2

Mapped 1

1 In the **Mesh** toolbar, click **Boundary** and choose **Mapped**.

2 Select Boundaries 4, 7, 14, 19, 24, 29, and 35 only.

Distribution 1

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

2 Select Edges 1, 4, 21, 57, and 58 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 59 only.

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

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

Distribution 3

1 Right-click **Mapped 1** 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 `20`.

Distribution 4

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

2 Select Edges 31 and 39 only.

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

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

Distribution 5

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

2 Select Edges 30, 38, 46, and 54 only.

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

4 From the **Distribution type** list, choose **Fixed number of elements**.

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

6 Click **Build Selected**.

DEFINITIONS (COMP2)

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.

Integration 5 (intop5)

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 29 only.

Integration 6 (intop6)

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 **Boundary**.

4 Select Boundary 35 only.

5 Locate the **Advanced** section. From the **Method** list, choose **Summation over nodes**.

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*atan((intop5(x)-intop4(x))/(intop5(z)-intop4(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** check box for **Solid Mechanics (solid)**.

5 Click **Add Study** 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** check box.

3 Click **Add**.

4 In the table, enter the following settings:

Parameter name	Parameter value list	Parameter unit
para (Punch displacement parameter)	`range(0,2e-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)>Dependent Variables 1** node, then click **Displacement field (comp2.u2)**.

4 In the **Settings** window for **Field**, locate the **Scaling** section.

5 In the **Scale** text field, type `1e-2`.

6 In the **Model Builder** window, expand the **Friction (shell)>Solver Configurations>Solution 3 (sol3)>Stationary Solver 1** node, then click **Parametric 1**.

7 In the **Settings** window for **Parametric**, locate the **Continuation** section.

8 Select the **Tuning of step size** check box.

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

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

11 In the **Maximum step size** text field, type `5e-3`.

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

13 Right-click **Friction (shell)>Solver Configurations>Solution 3 (sol3)>Stationary Solver 1>Parametric 1** and choose **Stop Condition**.

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

15 Click **Add**.

16 In the table, enter the following settings:

Stop expression	Stop if	Active	Description
`comp2.shell.Tnmax_p3-(para>1)`	Negative (<0)	√	`Stop expression 1`

17 Click **Add**.

18 In the table, enter the following settings:

Stop expression	Stop if	Active	Description
`comp2.shell.Tnmax_p4-(para>1)`	Negative (<0)	√	`Stop expression 2`

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

20 Clear the **Add warning** check box.

21 In the **Model Builder** window, expand the **Friction (shell)>Solver Configurations>Solution 3 (sol3)>Stationary Solver 1>Segregated 1** node, then click **Shell**.

22 In the **Settings** window for **Segregated Step**, locate the **Method and Termination** section.

23 From the **Nonlinear method** list, choose **Constant (Newton)**.

24 From the **Termination technique** list, choose **Iterations or tolerance**.

25 Click **Compute**.

RESULTS

Stress (shell)

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

2 From the **Frame** list, choose **Spatial (x, y, z)**.

Surface 1

1 In the **Model Builder** window, expand the **Stress (shell)** node.

2 Right-click **Results>Stress (shell)>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 **Model Builder** window, click **Stress (shell)**.

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

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

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

Surface 2

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

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

3 From the **Solution parameters** list, choose **From parent**.

4 In the **Stress (shell)** toolbar, click **Plot**.

Deformation 1

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

2 Right-click **Contour 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. Select the **Scale factor** check box.

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

Equivalent Plastic Strain (shell)

1 In the **Model Builder** window, under **Results** click **Equivalent Plastic Strain (shell)**.

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

Expression	Unit	Description
angle_shell	deg	Blank deformation angle (shell)

5 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*intop6(shell.RFz)	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**.