Sheet Metal Forming

Introduction

This example is a NAFEMS validation model of a friction contact problem with an elastoplastic material model; see Ref. 1.

A thin metal sheet is being forced into a die by a punch. Both the compressing displacement and the release of the punch are modeled in order to compute the forming angle (at the maximum punch displacement) and the angle after the punch release.

Experimental data are available for the forming angle, the angle after release, and the punch forces with respect to the punch displacement.

Model Definition

The geometry of the problem consists of an assembly made with three parts: the punch, the die, and a 1 mm metal sheet.

A plane strain assumption is used. This will be a good approximation if the out-of-plane thickness of the sheet is large. Due to the symmetry, only one half of the geometry is represented. Figure 1 shows the geometry used in the model.

©2012 COMSOL

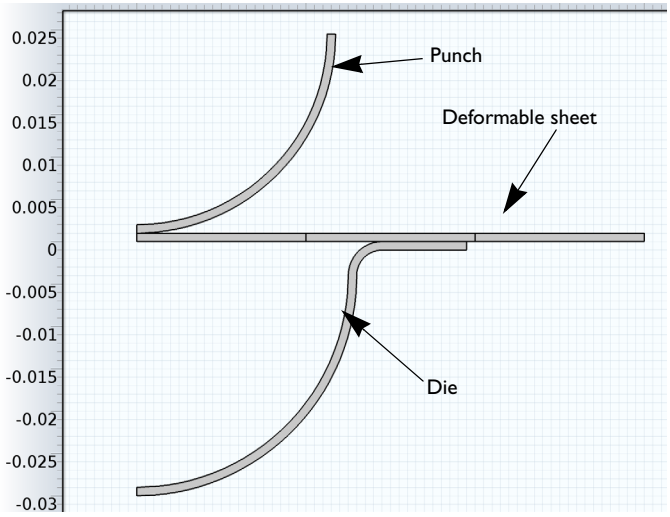


Figure 1: Model geometry: the punch at the top, the thin sheet in the middle, and the die at the bottom.

Because of the large difference in stiffness between the tools and the thin sheet, these two first parts can be considered being rigid. Only the thin sheet is considered as an deformable body.

The thin sheet has a 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.

The contact between the thin sheet and the tools is defined using a Mohr-Coulomb friction coefficient of 0.1342.

©2012 COMSOL

Experimental data of the punch forces, as well as the forming angle and the angle after release are available. These will be the target data of the analysis. Table 1 shows the experimental punch force values with as function of the punch displacement.

TABLE 1: PUNCH FORCE VERSUS PUNCH DISPLACEMENT.

PUNCH DISPLACEMENT (MM)	PUNCH FORCE (N)
1	42
3	108
6	133
9	150
12	170
15	188
18	208
21	227
24	236
27	218
28.5	187

Table 2 shows the value of the forming angle and angle after release.

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

FORMING ANGLE (DEGREES)	ANGLE AFTER RELEASE (DEGREES)
19.6-21	53.4-55.8

Because of the small thickness of the plate, limited strain values are expected even though the displacements and rotations are large. This satisfies the assumptions in the elastoplastic formulation used in COMSOL Multiphysics.

©2012 COMSOL

Results and Discussion

The default plot in Figure 2 shows the residual stress distribution and deformed shape after the release.

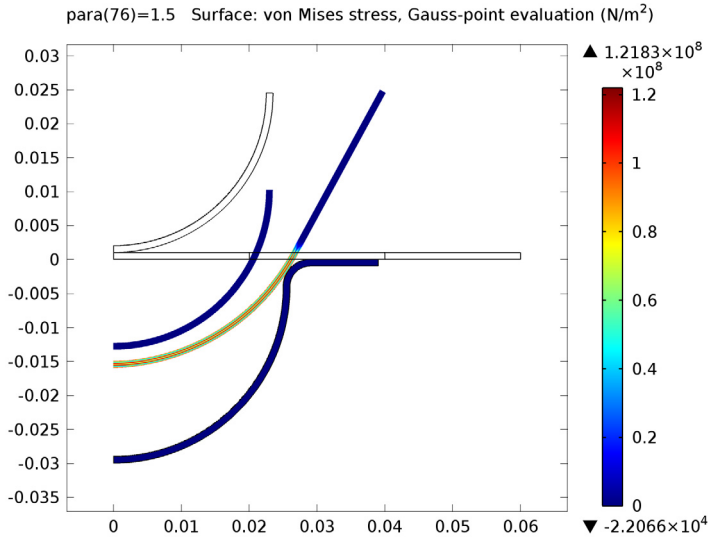


Figure 2: von Mises stress and deformed shape after release.

©2012 COMSOL

Figure 3 below shows the von Mises stress at the maximum punch displacement. The maximum stress value is far above the yield stress (194 MPa) in large parts of the plate.

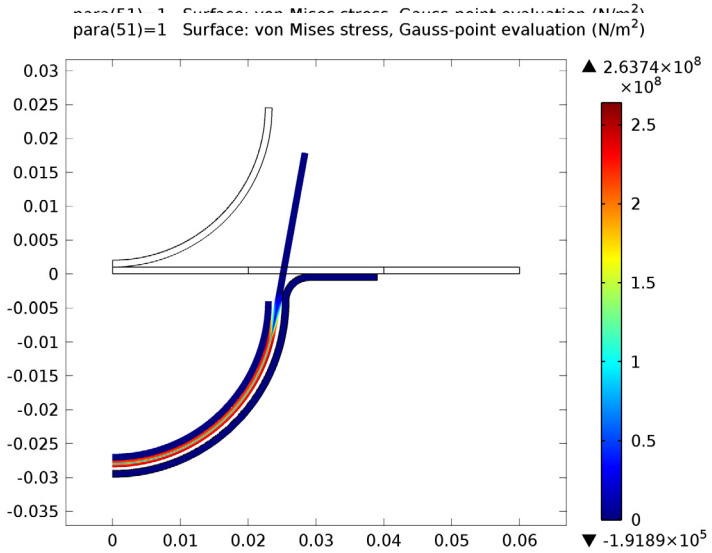


Figure 3: von Mises stress at maximum punch displacement.

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

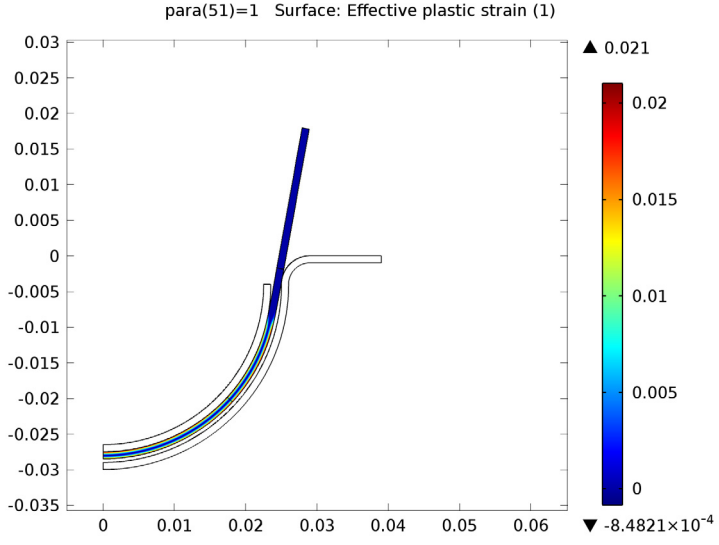


Figure 4: Effective plastic strain at maximum punch deflection.

The computed forming angle is 20.5 degrees and the angle after release is 57.7 degrees. These values are in good agreement with the experimental data. The deviation is about 6%.

©2012 COMSOL

Figure 5 shows the punch forces versus the displacement. Note that the maximum value occurs at 90% of the maximum punch displacement. The same behavior can be seen in the experimental results.

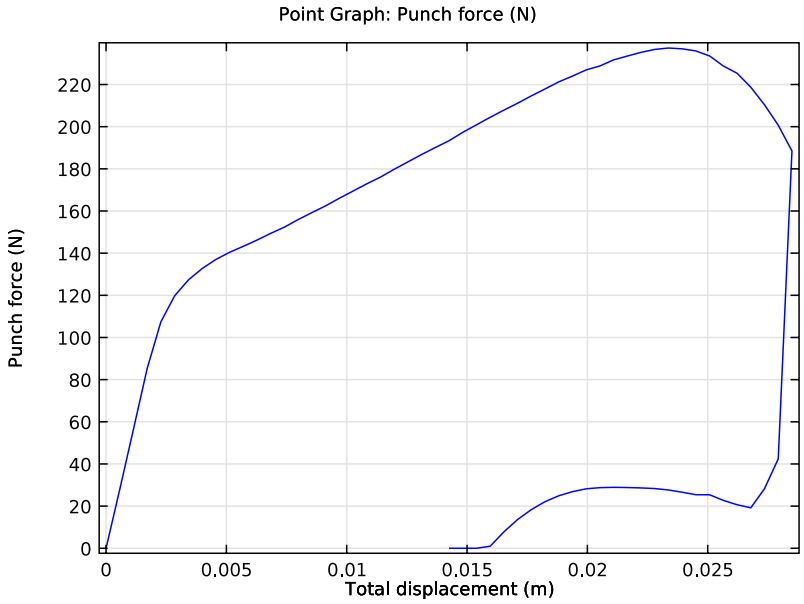


Figure 5: Punch force versus punch displacement

Notes About the COMSOL Implementation

The user-defined hardening function can be used to implement the Hollomon hardening function in COMSOL Multiphysics. The latter can be defined as

$$\sigma_{\text{hard}} = K \cdot \left(\varepsilon_{\text{plastic}} + \frac{\sigma_{\text{mises}}}{E} + \varepsilon_{\text{shift}} \right)^{n}$$

The first two terms represent the total strain, while $\varepsilon_{\text{shift}}$ is a parameter to adjust the hardening function so that it will exactly match the yield stress at initial yield.

$$\varepsilon_{\text{shift}} = \left(\frac{\sigma_{\text{yield}}}{K} \right)^{1/n} - \frac{\sigma_{\text{yield}}}{E}$$

Since the problem is severely nonlinear due to the combination of contact with friction, the elastoplastic material model, and geometric non-linearity, you need to set up the model carefully to avoid convergence problems.

- Use a reduced normal penalty factor for both the normal contact pressure and the friction forces. The convergence is improved with a normal penalty factor set to `min(1e-4*5^segiter, 1)*solid.Eequ/solid.hmin_dst` and a tangential penalty factor set to `min(1e-4*5^segiter, 1)*solid.Eequ/3/solid.hmin_dst`. Since the default penalty factors are based on the elastic stiffness, they must often be reduced if plasticity is present at the contact boundaries.
- Make sure that the parametric step is small enough so that the initial guess defined by the previous computed solution is close enough to the solution at the current parametric step. You can avoid too large steps by using manual tuning of the parametric solver.
- Use a constant predictor for the parametric solver. Because of the strong nonlinearities, the default linear predictor is not a good choice here.
- Use a manual scaling for each variable with a good estimate of the order of magnitude. For instance you can use 10^9 for the normal contact pressure, 10^8 for the tangential friction forces and 10^{-2} for the structural displacement variables.
- With a tight tolerance it may happen that no converged solution can be found at a certain parameter value. In that case you can try to release the tolerance somewhat, as shown in this example.

Because of small radius of curvature of the die it is important to refine the mesh along the curved edges. As these edges belongs to a domain assumed to be rigid, it is no problem to have the mesh on the contact pair source boundaries finer than on the destination boundaries.

Reference

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

Model Library path: `Nonlinear_Structural_Materials_Module/Plasticity/sheet_metal_forming`

 ©2012 COMSOL

Modeling Instructions

MODEL WIZARD

1 Go to the **Model Wizard** window.

2 Click the **2D** button.

3 Click **Next**.

4 In the **Add physics** tree, select **Structural Mechanics>Solid Mechanics (solid)**.

5 Click **Next**.

6 Find the **Studies** subsection. In the tree, select **Preset Studies>Stationary**.

7 Click **Finish**.

GLOBAL DEFINITIONS

Parameters

1 In the **Model Builder** window, right-click **Global Definitions** and choose **Parameters**.

2 In the **Parameters** settings window, locate the **Parameters** section.

3 In the table, enter the following settings:

NAME	EXPRESSION	DESCRIPTION
R1	23.5[mm]	Punch radius
R2	25[mm]	Die radius
R3	4[mm]	Die shoulder
width	30[mm]	Sheet width
L	60[mm]	Sheet length
th	1[mm]	Sheet thickness
E0	70.5e3[MPa]	Young's modulus
nu0	0.342	Poisson's ratio
K	550.4[MPa]	Hollomon hardening parameter
n	0.223	Hollomon hardening exponent
sigma_yield	194[MPa]	Initial yield stress
eps_shift	(sigma_yield/K)^(1/n)-sigma_yield/E0	Shift parameter
para	0	Punch displacement parameter

©2012 COMSOL

GEOMETRY 1

Rectangle 1

1 In the **Model Builder** window, under **Model 1** right-click **Geometry 1** and choose **Rectangle**.

2 In the **Rectangle** settings window, locate the **Size** section.

3 In the **Width** edit field, type `L`.

4 In the **Height** edit field, type `th`.

Rectangle 2

1 In the **Model Builder** window, right-click **Geometry 1** and choose **Rectangle**.

2 In the **Rectangle** settings window, locate the **Size** section.

3 In the **Width** edit field, type `20[mm]`.

4 In the **Height** edit field, type `th`.

5 Locate the **Position** section. In the **x** edit field, type `20[mm]`.

Union 1

1 In the **Model Builder** window, right-click **Geometry 1** and choose **Boolean Operations>Union**.

2 Select the objects **r1** and **r2** only.

3 Click the **Build Selected** button.

Bézier Polygon 1

1 In the **Model Builder** window, right-click **Geometry 1** and choose **Bézier Polygon**.

2 In the **Bézier Polygon** settings window, locate the **Polygon Segments** section.

3 Find the **Added segments** subsection. Click the **Add Quadratic** button.

4 Find the **Control points** subsection. In row **1**, set **y** to `th`.

5 In row **2**, set **x** to `R1`.

6 In row **2**, set **y** to `th`.

7 In row **3**, set **x** to `R1`.

8 In row **3**, set **y** to `R1+th`.

9 Find the **Added segments** subsection. Click the **Add Linear** button.

10 Find the **Control points** subsection. In row **2**, set **x** to `R1-th`.

11 Find the **Added segments** subsection. Click the **Add Quadratic** button.

12 Find the **Control points** subsection. In row **2**, set **y** to `2*th`.

©2012 COMSOL

13 In row **3**, set **x** to `0`.

14 In row **3**, set **y** to `2*th`.

15 Click the **Build Selected** button.

Bézier Polygon 2

1 In the **Model Builder** window, right-click **Geometry 1** and choose **Bézier Polygon**.

2 In the **Bézier Polygon** settings window, locate the **Polygon Segments** section.

3 Find the **Added segments** subsection. Click the **Add Quadratic** button.

4 Find the **Control points** subsection. In row **1**, set **y** to `-R2-R3`.

5 In row **2**, set **x** to `R2`.

6 In row **2**, set **y** to `-R2-R3`.

7 In row **3**, set **x** to `R2`.

8 In row **3**, set **y** to `-R3`.

9 Find the **Added segments** subsection. Click the **Add Quadratic** button.

10 Find the **Control points** subsection. In row **2**, set **y** to `0`.

11 In row **3**, set **x** to `R2+R3`.

12 In row **3**, set **y** to `0`.

13 Find the **Added segments** subsection. Click the **Add Linear** button.

14 Find the **Control points** subsection. In row **2**, set **x** to `R2+R3+10[mm]`.

15 Find the **Added segments** subsection. Click the **Add Linear** button.

16 Find the **Control points** subsection. In row **2**, set **y** to `-th`.

17 Find the **Added segments** subsection. Click the **Add Linear** button.

18 Find the **Control points** subsection. In row **2**, set **x** to `R2+R3`.

19 Find the **Added segments** subsection. Click the **Add Quadratic** button.

20 Find the **Control points** subsection. In row **2**, set **x** to `R2+th`.

21 In row **3**, set **x** to `R2+th`.

22 In row **3**, set **y** to `-R3`.

23 Find the **Added segments** subsection. Click the **Add Quadratic** button.

24 Find the **Control points** subsection. In row **2**, set **y** to `-R3-R2-th`.

25 In row **3**, set **x** to `0`.

26 In row **3**, set **y** to `-R2-R3-th`.

27 Click the **Build Selected** button.

©2012 COMSOL

Form Union

1 In the **Model Builder** window, under **Model 1>Geometry 1** click **Form Union**.

2 In the **Finalize** settings window, locate the **Finalize** section.

3 From the **Finalization method** list, choose **Form an assembly**.

4 Clear the **Create pairs** check box.

5 Click the **Build Selected** button.

6 Click the **Zoom Extents** button on the Graphics toolbar.

[[Include here figure: sheet_metal_forming_geom.png]]

DEFINITIONS

1 In the **Model Builder** window, under **Model 1** right-click **Definitions** and choose **Pairs>Contact Pair**.

2 Select Boundary 21 only.

3 In the **Pair** settings window, click **Activate Selection** in the upper-right corner of the **Destination Boundaries** section. Select Boundaries 11 and 14 only.

4 In the **Model Builder** window, right-click **Definitions** and choose **Pairs>Contact Pair**.

5 Select Boundaries 3, 6, and 7 only.

6 In the **Pair** settings window, click **Activate Selection** in the upper-right corner of the **Destination Boundaries** section. Select Boundaries 10 and 13 only.

Integration 1

1 In the **Model Builder** window, right-click **Definitions** and choose **Model Couplings>Integration**.

2 In the **Integration** settings window, locate the **Source Selection** section.

3 From the **Geometric entity level** list, choose **Point**.

4 Select Point 14 only.

Integration 2

1 In the **Model Builder** window, right-click **Definitions** and choose **Model Couplings>Integration**.

2 In the **Integration** settings window, locate the **Source Selection** section.

3 From the **Geometric entity level** list, choose **Point**.

4 Select Point 16 only.

©2012 COMSOL

Integration 3

1 In the **Model Builder** window, right-click **Definitions** and choose **Model Couplings>Integration**.

2 Select Domain 5 only.

3 In the **Integration** settings window, locate the **Advanced** section.

4 From the **Method** list, choose **Summation over nodes**.

Variables 1

1 In the **Model Builder** window, right-click **Definitions** and choose **Variables**.

2 In the **Variables** settings window, locate the **Variables** section.

3 In the table, enter the following settings:

NAME	EXPRESSION	DESCRIPTION
sigma_hard	K*(solid.epe+solid.mises/ E0+eps_shift)^n-sigma_yield	Hollomon hardening function
angle	2*(pi/ 2-atan((intop2(y)-intop1(y))/ (intop2(x)-intop1(x))))	Plate deformation angle
force	intop3(-2*solid.RFy)	Punch force

Interpolation 1

1 In the **Model Builder** window, right-click **Definitions** and choose **Functions>Interpolation**.

2 In the **Interpolation** settings window, locate the **Parameters** section.

3 In the **Function name** edit field, type punch.

4 In the table, enter the following settings:

T	F(T)
0	0
1	-28.5
2	0

MATERIALS

1 In the **Model Builder** window, under **Model 1** right-click **Materials** and choose **Open Material Browser**.

2 In the **Material Browser** window, locate the **Materials** section.

3 In the tree, select **Built-In>Structural steel**.

4 Right-click and choose **Add Material to Model** from the menu.

Material 2

1 In the **Model Builder** window, right-click **Materials** and choose **Material**.

2 Select Domains 2–4 only.

3 In the **Material** settings window, locate the **Material Contents** section.

4 In the table, enter the following settings:

PROPERTY	NAME	VALUE
Young's modulus	E	E0
Poisson's ratio	nu	nu0
Density	rho	2700[kg/m^3]

SOLID MECHANICS

1 In the **Model Builder** window, under **Model 1** click **Solid Mechanics**.

2 In the **Solid Mechanics** settings window, locate the **Thickness** section.

3 In the *d* edit field, type `width`.

Linear Elastic Material 1

In the **Model Builder** window, expand the **Solid Mechanics** node.

Plasticity 1

1 Right-click **Linear Elastic Material 1** and choose the domain setting **Plasticity**.

2 Select Domains 2–4 only.

3 In the **Plasticity** settings window, locate the **Plasticity Model** section.

4 From the σ_{ys0} list, choose **User defined**. In the associated edit field, type `sigma_yield`.

5 From the **Isotropic hardening** list, choose **Use hardening function data**.

6 In the σ_hεpe) edit field, type `sigma_hard`.

Symmetry 1

1 In the **Model Builder** window, right-click **Solid Mechanics** and choose the boundary condition **Symmetry**.

2 Select Boundaries 1, 9, and 19 only.

©2012 COMSOL

Fixed Constraint 1

1 In the **Model Builder** window, right-click **Solid Mechanics** and choose the domain setting **More>Fixed Constraint**.

2 Select Domain 1 only.

Prescribed Displacement 1

1 In the **Model Builder** window, right-click **Solid Mechanics** and choose the domain setting **More>Prescribed Displacement**.

2 Select Domain 5 only.

3 In the **Prescribed Displacement** settings window, 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 v_0 edit field, type `punch(para)[mm]`.

Contact 1

1 In the **Model Builder** window, right-click **Solid Mechanics** and choose the boundary condition **Pairs>Contact**.

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

3 In the **Pairs** list, select **Contact Pair 1**.

4 Locate the **Normal Contact** section. In the p_n edit field, type `min(1e-4*5^segiter, 1)*solid.Eequ/solid.hmin_dst`.

5 Right-click **Model 1>Solid Mechanics>Contact 1** and choose the boundary condition **Friction**.

Friction 1

1 In the **Friction** settings window, locate the **Friction** section.

2 In the μ_{stat} edit field, type `0.1342`.

3 In the p_t edit field, type `min(1e-4*5^segiter, 1)*solid.Eequ/3/solid.hmin_dst`.

Contact 2

1 In the **Model Builder** window, right-click **Solid Mechanics** and choose the boundary condition **Pairs>Contact**.

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

3 In the **Pairs** list, select **Contact Pair 2**.

4 Locate the **Normal Contact** section. In the p_n edit field, type `min(1e-4*5^segiter, 1)*solid.Eequ/solid.hmin_dst`.

5 Right-click **Model 1>Solid Mechanics>Contact 2** and choose the boundary condition **Friction**.

Friction 1

1 In the **Friction** settings window, locate the **Friction** section.

2 In the μ_{stat} edit field, type `0.1342`.

3 In the p_t edit field, type `min(1e-4*5^segiter, 1)*solid.Eequ/3/solid.hmin_dst`.

MESH 1

Mapped 1

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

Distribution 1

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

2 Select Boundaries 5, 6, and 21 only.

3 In the **Distribution** settings window, locate the **Distribution** section.

4 In the **Number of elements** edit field, type `300`.

Distribution 2

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

2 Select Boundaries 2, 3, 7, 8, and 14 only.

3 In the **Distribution** settings window, locate the **Distribution** section.

4 In the **Number of elements** edit field, type `100`.

Distribution 3

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

2 Select Boundaries 1 and 19 only.

3 In the **Distribution** settings window, locate the **Distribution** section.

4 In the **Number of elements** edit field, type `1`.

 ©2012 COMSOL

Distribution 4

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

2 Select Boundary 11 only.

3 In the **Distribution** settings window, locate the **Distribution** section.

4 In the **Number of elements** edit field, type 30.

Distribution 5

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

2 Select Boundary 17 only.

Distribution 6

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

2 Select Boundary 18 only.

3 In the **Distribution** settings window, locate the **Distribution** section.

4 In the **Number of elements** edit field, type 6.

5 Click the **Build All** button.

STUDY 1

Step 1: Stationary

1 In the **Model Builder** window, expand the **Study 1** node, then click **Step 1: Stationary**.

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

3 Select the **Plot** check box.

4 From the **Plot group** list, choose **Default**.

5 Click to expand the **Study Extensions** section. Select the **Continuation** check box.

6 Click **Add**.

7 In the table, enter the following settings:

CONTINUATION PARAMETER	PARAMETER VALUE LIST
para	range(0,2e-2,1.5)

Solver 1

1 In the **Model Builder** window, right-click **Study 1** and choose **Show Default Solver**.

2 Expand the **Solver 1** node.

3 In the **Model Builder** window, expand the **Study 1>Solver Configurations>Solver 1>Dependent Variables 1** node, then click **mod1.solid.Tn_p1**.

4 In the **Field** settings window, locate the **Scaling** section.

5 In the **Scale** edit field, type `1e9`.

6 In the **Model Builder** window, under **Study 1>Solver Configurations>Solver 1>Dependent Variables 1** click **mod1.solid.Tn_p2**.

7 In the **Field** settings window, locate the **Scaling** section.

8 In the **Scale** edit field, type `1e9`.

9 In the **Model Builder** window, under **Study 1>Solver Configurations>Solver 1>Dependent Variables 1** click **mod1.u**.

10 In the **Field** settings window, locate the **Scaling** section.

11 In the **Scale** edit field, type `1e-2`.

12 In the **Model Builder** window, under **Study 1>Solver Configurations>Solver 1>Dependent Variables 1** click **mod1.solid.Tt_p1**.

13 In the **Field** settings window, locate the **Scaling** section.

14 In the **Scale** edit field, type `1e8`.

15 In the **Model Builder** window, under **Study 1>Solver Configurations>Solver 1>Dependent Variables 1** click **mod1.solid.Tt_p2**.

16 In the **Field** settings window, locate the **Scaling** section.

17 In the **Scale** edit field, type `1e8`.

18 In the **Model Builder** window, under **Study 1>Solver Configurations>Solver 1** click **Stationary Solver 1**.

19 In the **Stationary Solver** settings window, locate the **General** section.

20 In the **Relative tolerance** edit field, type `1e-2`.

21 In the **Model Builder** window, expand the **Study 1>Solver Configurations>Solver 1>Stationary Solver 1** node, then click **Parametric 1**.

22 In the **Parametric** settings window, locate the **General** section.

23 From the **Predictor** list, choose **Constant**.

24 Click to expand the **Tuning** section. Select the **Tuning of step size** check box.

25 In the **Initial step size** edit field, type `1e-2`.

26 In the **Minimum step size** edit field, type `5e-5`.

27 In the **Maximum step size** edit field, type `2e-4`.

 ©2012 COMSOL

28 In the **Model Builder** window, right-click **Study 1** and choose **Compute**.

[[The generated default plot is shown in [[Figure 2]]. You can now proceed to display the same results at maximum punch deformation as shown in [[Figure 3]].]]

RESULTS

Stress (solid)

1 In the **2D Plot Group** settings window, locate the **Data** section.

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

3 Click the **Plot** button.

[[Create the effective strain plot in [[Figure 4]]:]]

4 In the **Model Builder** window, under **Results>Stress (solid)** click **Surface 1**.

5 In the **Surface** settings window, click **Replace Expression** in the upper-right corner of the **Expression** section. From the menu, choose **Solid Mechanics>Strain>Effective plastic strain (solid.epe)**.

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

7 In the **2D Plot Group** settings window, locate the **Plot Settings** section.

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

9 Click the **Plot** button.

10 Click the **Zoom Extents** button on the Graphics toolbar.

[[Evaluate the forming angle and the angle after release.]]

Derived Values

1 In the **Model Builder** window, under **Results** right-click **Derived Values** and choose **Global Evaluation**.

2 In the **Global Evaluation** settings window, locate the **Data** section.

3 From the **Parameter selection (para)** list, choose **From list**.

4 In the **Parameter values (para)** list, choose **1** and **1.5**.

5 Click **Replace Expression** in the upper-right corner of the **Expression** section. From the menu, choose **Definitions>Plate deformation angle (angle)**.

6 Locate the **Expression** section. From the **Unit** list, choose **degree**.

7 Click the **Evaluate** button.

[[At last, generate a graph over the force as function of the punch displacement.]]

©2012 COMSOL

1D Plot Group 2

1 In the **Model Builder** window, right-click **Results** and choose **1D Plot Group**.

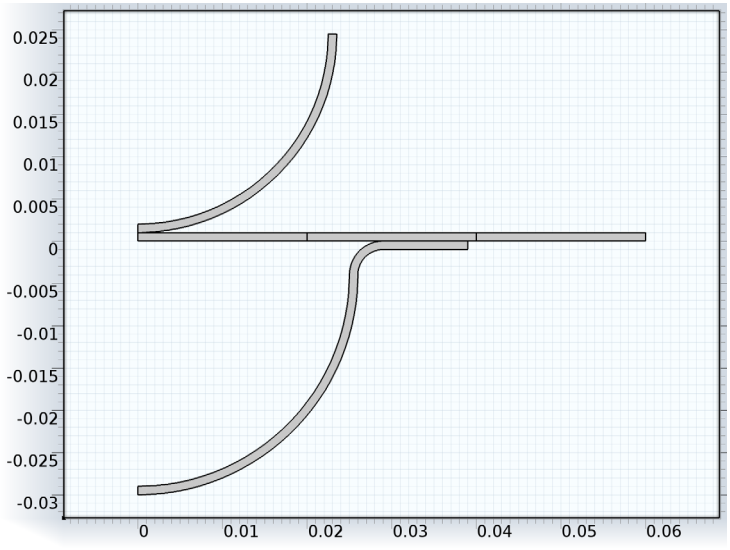
2 Right-click **1D Plot Group 2** and choose **Point Graph**.

3 Select Point 19 only.

4 In the **Point Graph** settings window, click **Replace Expression** in the upper-right corner of the **y-Axis Data** section. From the menu, choose **Definitions>Punch force (force)**.

5 Locate the **x-Axis Data** section. From the **Parameter** list, choose **Expression**.

6 Click the **Plot** button.



 ©2012 COMSOL