Created in COMSOL Multiphysics 6.0

Die Forming

Introduction

Die forming is a widespread manufacture process for sheet-metal forming. The workpiece, usually a metal sheet, is permanently reshaped around a die through plastic deformation by forming and drawing processes.

Simulations can be carried out in order to avoid cracks, tears, wrinkles, and too much thinning and stretching. They are also useful to estimate and overcome the springback phenomenon: once the forming process is done and the forming tools are removed, the workpiece attempts to partially recover its initial shape through relaxation of the elastic stresses. The springback can cause the formed blank to get an unexpected state of warping. To cope with this effect, it is possible to over-bend the sheet. The die, the punch, and the blank-holder must be manufactured to incorporate this effect.

In this model, a flat metal sheet made of aluminum is pressed onto a curved die by a similarly shaped punch. Both the forming and the springback phenomena are modeled. From a simulation point of view, the problem is severely nonlinear due to contact, large strain plasticity, and geometric nonlinearity. Figure 1shows the shape after forming.

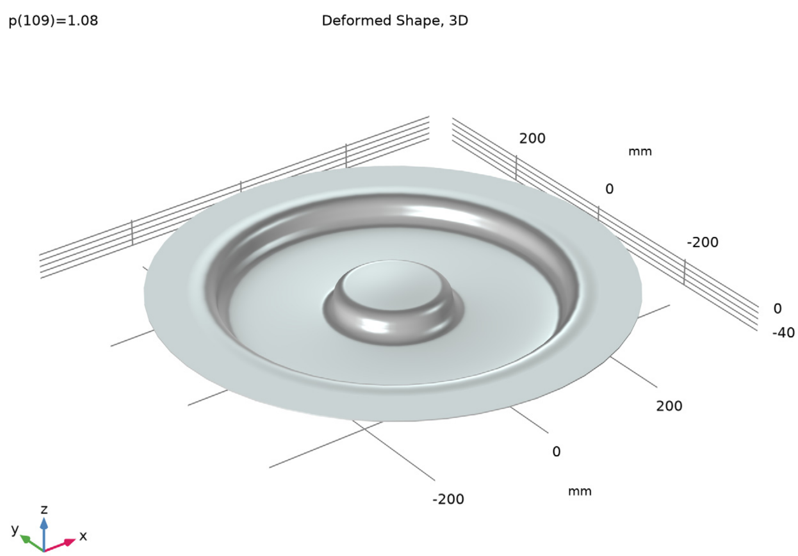


Figure 1: Deformed shape after the forming process.

Model Definition

The model geometry is shown in Figure 2. Due to the axial symmetry, a 2D axisymmetric formulation can be used. The die and the blank-holder together clamp the blank to be reshaped, while the punch performs the drawing, stretching, and bending.

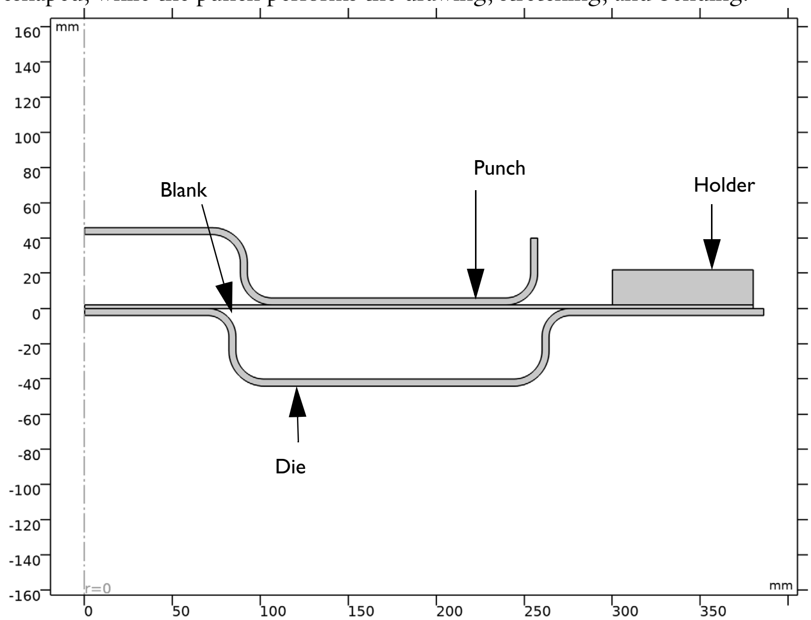


Figure 2: Forming tools setup.

The analysis is carried out in two steps. First, the punch is pushed towards the sheet through a displacement of 40 mm. This step aims to simulate the forming and drawing processes. A second step is used to perform the springback analysis. The punch is released progressively to model the springback phenomenon.

The sheet is made of aluminum. An isotropic elastoplastic material with user-defined isotropic hardening and large plastic strain formulation is used to characterize the plastic deformation.

The die and the punch are made of structural steel, so they are much more rigid than the aluminum sheet. The die and holder are fixed, and the punch deforms the blank with a prescribed vertical displacement, which is ramped linearly.

Results and Discussion

Figure 3 shows the residual stress after the release of the punch, where large stresses are present in the deformed corners of the blank. These are a consequence of the plastic strains that are shown inFigure 4. As expected, large plastic strains occur, and at the most strained location, the plastic strain exceeds 60%.

Figure 5 shows the variation of the thickness of the deformed part obtained from the die forming process. The initial blank thickness is 0.2 mm. The maximum thinning is observed at the center of the part. A comparison of the maximum radial position between the forming and the release stage would give an idea of the springback.

Finally, the reaction force that is necessary to apply to the punch during the process is shown in Figure 6.

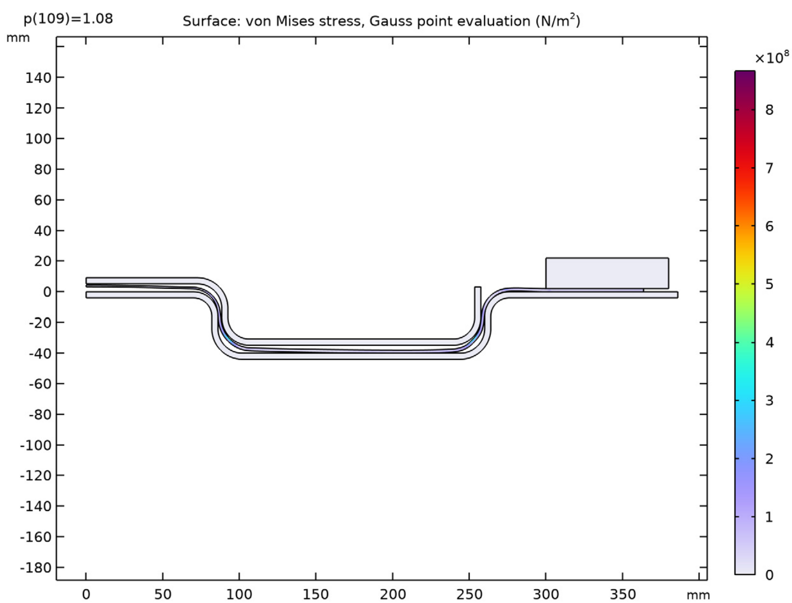


Figure 3: Residual stress after release of the punch.

.

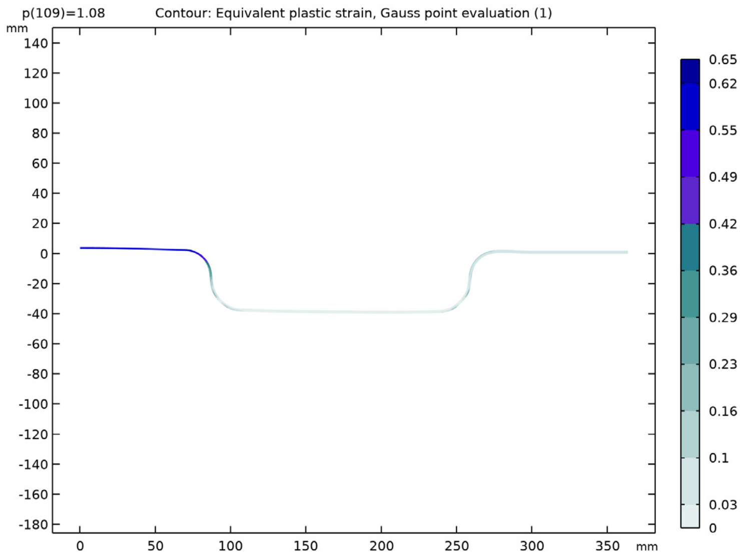


Figure 4: Plastic strains after release of the punch.

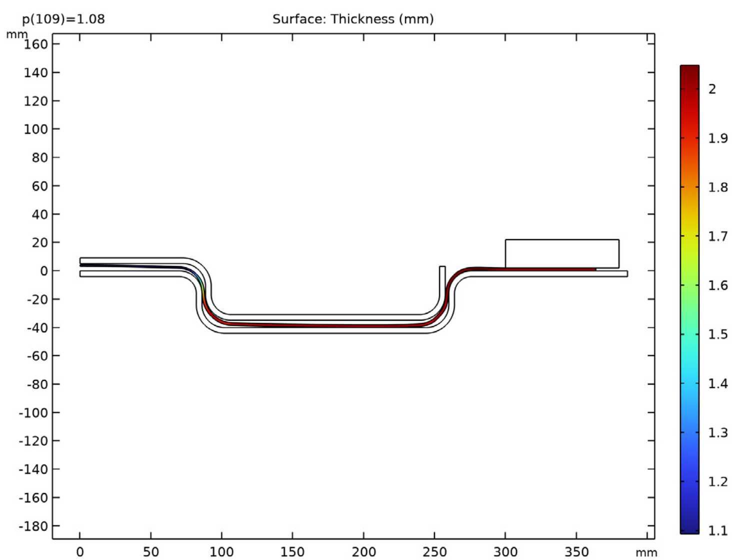


Figure 5: Thickness of the part after the die forming process.

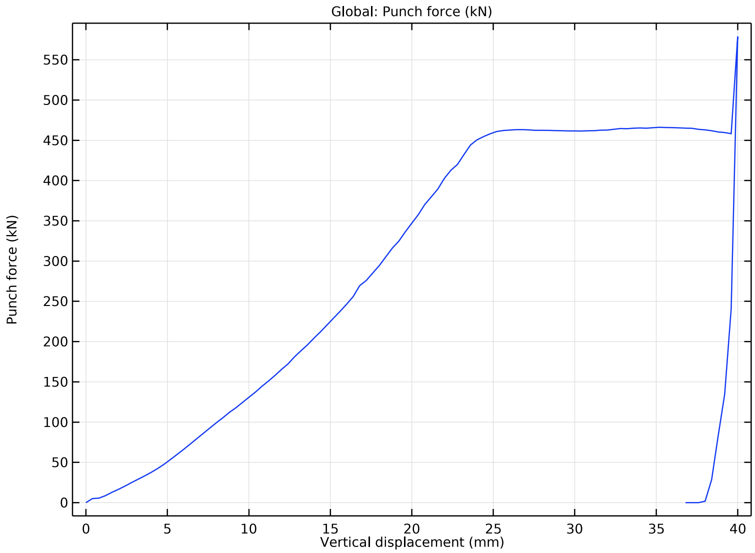


Figure 6: Force applied to the punch during the die forming.

Notes About the COMSOL Implementation

The strong nonlinearity of the problem due to contact, plasticity, and geometric nonlinearity may cause some convergence issues. Some useful tips for improving the convergence are given below.

- Choose the source and destination in contact pairs carefully. Based on stiffness of the metal parts, the sheet should be the destination. The blank/sheet is meshed finely compared to the source. The curved boundaries of the source are also finely meshed compared to the flat boundaries.
- The parametric sweep must use steps that are small enough to avoid divergence of the solution. If the parametric step chosen by the solver is too large, the initial guess of a solution based on previous computed solution could be too far from the solution in the current step. Use **Tuning of step size** to manually adjust the parametric steps.
- To compute the thickness of the deformed shape, a nonlocal projection coupling is used to define the variable `th`. Such a coupling can integrate any expression across a domain in the spatial frame. Here, integrating '1' across the blank returns its deformed thickness.

Contact is in the model described using the augmented Lagrangian method with a coupled solution method. This method is chosen over the default penalty method to improve the accuracy of the contact condition, that is, to avoid large overclosure between the parts.

Application Library path: `Nonlinear_Structural_Materials_Module/Plasticity/die_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 Axisymmetric**.

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

3 Click **Add**.

4 In the **Added physics interfaces** tree, select **Solid Mechanics (solid)**.

5 Click **Study**.

6 In the **Select Study** tree, select **General Studies>Stationary**.

7 Click **Done**.

GLOBAL DEFINITIONS

Parameters 1

1 In the **Model Builder** window, under **Global Definitions** click **Parameters 1**.

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

3 In the table, enter the following settings:

Name	Expression	Value	Description
sigma_y0	124[MPa]	1.24E8 Pa	Initial yield stress
U_p	40[mm]	0.04 m	Punch displacement

Name	Expression	Value	Description
R0	0.69	0.69	Lankford's coefficient r0
R45	0.69	0.69	Lankford's coefficient r45
R90	0.69	0.69	Lankford's coefficient 90
F	R0/(R90*(R0+1))/sigma_y0^2	3.8483E-17 $m^2 \cdot s^4/kg^2$	Hill's coefficient
G	1/(R0+1)/sigma_y0^2	3.8483E-17 $m^2 \cdot s^4/kg^2$	Hill's coefficient
H	R0/(R0+1)/sigma_y0^2	2.6553E-17 $m^2 \cdot s^4/kg^2$	Hill's coefficient
L	0	0	Hill's coefficient
M	0	0	Hill's coefficient
N	0	0	Hill's coefficient
p	0	0	Loading parameter

GEOMETRY 1

Set the unit of geometry to millimeters (mm).

1. In the **Model Builder** window, under **Component 1 (comp1)** click **Geometry 1**.
2. In the **Settings** window for **Geometry**, locate the **Units** section.
3. From the **Length unit** list, choose **mm**.

Polygon 1 (pol1)

1. In the **Geometry** toolbar, click **Polygon**.
2. In the **Settings** window for **Polygon**, locate the **Object Type** section.
3. From the **Type** list, choose **Open curve**.
4. Locate the **Coordinates** section. In the table, enter the following settings:

r (mm)	z (mm)
0	42
88.5	42
88.5	2
257.5	2
257.5	40

Fillet 1 (fil1)

1 In the **Geometry** toolbar, click **Fillet**.
2 On the object **pol1**, select Point 3 only.
3 In the **Settings** window for **Fillet**, locate the **Radius** section.
4 In the **Radius** text field, type 16.

Fillet 2 (fil2)

1 In the **Geometry** toolbar, click **Fillet**.
2 On the object **fil1**, select Points 3 and 5 only.
3 In the **Settings** window for **Fillet**, locate the **Radius** section.
4 In the **Radius** text field, type 18.

Thicken 1 (thi1)

1 In the **Geometry** toolbar, click **Conversions** and choose **Thicken**.
2 Select the object **fil2** only.
3 In the **Settings** window for **Thicken**, locate the **Options** section.
4 From the **Offset** list, choose **Asymmetric**.
5 In the **Upside thickness** text field, type 4.

Fillet 1 (fil1), Fillet 2 (fil2), Polygon 1 (pol1), Thicken 1 (thi1)

1 In the **Model Builder** window, under **Component 1 (comp1)>Geometry 1**, Ctrl-click to select **Polygon 1 (pol1)**, **Fillet 1 (fil1)**, **Fillet 2 (fil2)**, and **Thicken 1 (thi1)**.
2 Right-click and choose **Group**.

Punch

1 In the **Settings** window for **Group**, type Punch in the **Label** text field.
2 Click **Build Selected**.

Polygon 2 (pol2)

1 In the **Geometry** toolbar, click **Polygon**.
2 In the **Settings** window for **Polygon**, locate the **Object Type** section.
3 From the **Type** list, choose **Open curve**.
4 Locate the **Coordinates** section. In the table, enter the following settings:

r (mm)	z (mm)
0	0
86	0

r (mm)	z (mm)
86	-40
260	-40
260	0
386	0

Fillet 3 (fil3)

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

2 On the object **pol2**, select Points 3 and 5 only.

3 In the **Settings** window for **Fillet**, locate the **Radius** section.

4 In the **Radius** text field, type 16.

Fillet 4 (fil4)

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

2 On the object **fil3**, select Points 3 and 5 only.

3 In the **Settings** window for **Fillet**, locate the **Radius** section.

4 In the **Radius** text field, type 16.

Thicken 2 (thi2)

1 In the **Geometry** toolbar, click **Conversions** and choose **Thicken**.

2 Select the object **fil4** only.

3 In the **Settings** window for **Thicken**, locate the **Options** section.

4 From the **Offset** list, choose **Asymmetric**.

5 In the **Downside thickness** text field, type 4.

Fillet 3 (fil3), Fillet 4 (fil4), Polygon 2 (pol2), Thicken 2 (thi2)

1 In the **Model Builder** window, under **Component 1 (comp1)>Geometry 1**, Ctrl-click to select **Polygon 2 (pol2)**, **Fillet 3 (fil3)**, **Fillet 4 (fil4)**, and **Thicken 2 (thi2)**.

2 Right-click and choose **Group**.

Die

1 In the **Settings** window for **Group**, type `Die` in the **Label** text field.

2 Click **Build Selected**.

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 `80`.

4 In the **Height** text field, type `20`.

5 Locate the **Position** section. In the **r** text field, type `300`.

6 In the **z** text field, type `2`.

7 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 `380`.

4 In the **Height** text field, type `2`.

5 Click **Build All Objects**.

6 Click **Build Selected**.

Rectangle 1 (r1)

In the **Model Builder** window, under **Component 1 (comp1)>Geometry 1** right-click **Rectangle 1 (r1)** and choose **Group**.

Holder

In the **Settings** window for **Group**, type `Holder` in the **Label** text field.

Rectangle 2 (r2)

In the **Model Builder** window, under **Component 1 (comp1)>Geometry 1** right-click **Rectangle 2 (r2)** and choose **Group**.

Blank

In the **Settings** window for **Group**, type `Blank` in the **Label** text field.

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 From the **Pair type** list, choose **Contact pair**.

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

GLOBAL DEFINITIONS

Prescribed Punch Displacement

1 In the **Model Builder** window, expand the **Component 1 (comp1)>Definitions** node.
2 Right-click **Global Definitions** and choose **Functions>Interpolation**.
3 In the **Settings** window for **Interpolation**, type `Prescribed Punch Displacement` in the **Label** text field.
4 Locate the **Definition** section. In the **Function name** text field, type `U_punch`.
5 In the table, enter the following settings:

t	f(t)
0	0
1	-U_p
2	0

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

Argument	Unit
t	1

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

Function	Unit
U_punch	m

Hardening Function

1 In the **Home** toolbar, click f(X) **Functions** and choose **Global>Interpolation**.
2 In the **Settings** window for **Interpolation**, type `Hardening Function` in the **Label** text field.
3 Locate the **Definition** section. In the **Function name** text field, type `sigma_h`.
4 In the table, enter the following settings:

t	f(t)
0	0
0.02	43
0.05	76
0.1	103
0.15	115

t	f(t)
0.2	127
0.3	129
0.4	129.3
0.5	129.4

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

Argument	Unit
t	1

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

Function	Unit
sigma_h	MPa

DEFINITIONS

Integration 1 (intop1)

1 In the **Definitions** toolbar, click **Nonlocal Couplings** and choose **Integration**.

2 Select Domain 3 only.

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

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

Create explicit selections for the contact boundaries and various domains.

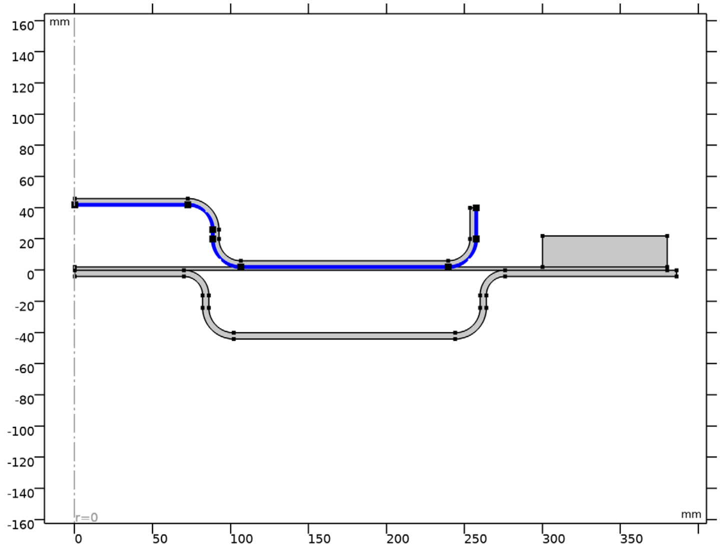
contact_punch

1 In the **Definitions** toolbar, click **Explicit**.

2 In the **Settings** window for **Explicit**, locate the **Input Entities** section.

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

4 Select Boundaries 26, 28, 30, 34, 35, 37, and 39 only.



5 In the **Label** text field, type `contact_punch`.

contact_die

1 In the **Definitions** toolbar, click **Explicit**.

2 In the **Settings** window for **Explicit**, type `contact_die` in the **Label** text field.

3 Locate the **Input Entities** section. From the **Geometric entity level** list, choose **Boundary**.

4 Select Boundaries 3, 5, 7, 8, 11, 14, 16, 18, and 19 only.

contact_holder

1 In the **Definitions** toolbar, click **Explicit**.

2 In the **Settings** window for **Explicit**, type `contact_holder` in the **Label** text field.

3 Locate the **Input Entities** section. From the **Geometric entity level** list, choose **Boundary**.

4 Select Boundary 42 only.

Select the boundary of the blank which is in contact with the punch and the holder.

contact_blank_up

1 In the **Definitions** toolbar, click **Explicit**.

2 In the **Settings** window for **Explicit**, type `contact_blank_up` in the **Label** text field.

3 Locate the **Input Entities** section. From the **Geometric entity level** list, choose **Boundary**.

4 Select Boundary 23 only.

Select the boundary of the blank which is in contact with the die.

contact_blank_down

1 In the **Definitions** toolbar, click **Explicit**.
2 In the **Settings** window for **Explicit**, type `contact_blank_down` in the **Label** text field.
3 Locate the **Input Entities** section. From the **Geometric entity level** list, choose **Boundary**.
4 Select Boundary 22 only.

die

1 In the **Definitions** toolbar, click **Explicit**.
2 Select Domain 1 only.
3 In the **Settings** window for **Explicit**, type `die` in the **Label** text field.

punch

1 In the **Definitions** toolbar, click **Explicit**.
2 Select Domain 3 only.
3 In the **Settings** window for **Explicit**, type `punch` in the **Label** text field.

holder

1 In the **Definitions** toolbar, click **Explicit**.
2 In the **Settings** window for **Explicit**, type `holder` in the **Label** text field.
3 Select Domain 4 only.

blank

1 In the **Definitions** toolbar, click **Explicit**.
2 Select Domain 2 only.
3 In the **Settings** window for **Explicit**, type `blank` in the **Label** text field.

Create a variable to evaluate the blank thickness after of the release of the punch.

General Projection 1 (genproj1)

1 In the **Definitions** toolbar, click **Nonlocal Couplings** and choose **General Projection**.
2 Select Domain 2 only.
3 In the **Settings** window for **General Projection**, locate the **Source Map** section.
4 In the **r-expression** text field, type `R`.
5 In the **z-expression** text field, type `Z`.
6 Locate the **Destination Map** section. In the **r-expression** text field, type `R`.

Variables 1

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

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

3 In the table, enter the following settings:

Name	Expression	Unit	Description
th	genproj1(1)	m	Thickness

4 Locate the **Geometric Entity Selection** section. From the **Geometric entity level** list, choose **Domain**.

5 Select Domain 2 only.

die_and_holder

1 In the **Definitions** toolbar, click **Union**.

2 In the **Settings** window for **Union**, type `die_and_holder` in the **Label** text field.

3 Locate the **Input Entities** section. Under **Selections to add**, click **Add**.

4 In the **Add** dialog box, in the **Selections to add** list, choose **die** and **holder**.

5 Click **OK**.

die_blank

Modify the contact pairs by activating **Manual control of selection** option. Make sure that the blank is destination in all contact pairs.

1 In the **Model Builder** window, under **Component 1 (comp1)>Definitions** click **Contact Pair 1 (ap1)**.

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

3 Select the **Manual control of selections and pair type** check box.

4 Locate the **Source Boundaries** section. From the **Selection** list, choose **contact_die**.

5 Locate the **Destination Boundaries** section. From the **Selection** list, choose **contact_blank_down**.

6 In the **Label** text field, type `die_blank`.

punch_blank

1 In the **Model Builder** window, click **Contact Pair 2 (ap2)**.

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

3 Select the **Manual control of selections and pair type** check box.

4 Locate the **Source Boundaries** section. From the **Selection** list, choose **contact_punch**.

5 Locate the **Destination Boundaries** section. From the **Selection** list, choose **contact_blank_up**.

6 In the **Label** text field, type `punch_blank`.

holder_blank

1 In the **Model Builder** window, click **Contact Pair 3 (ap3)**.

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

3 Select the **Manual control of selections and pair type** check box.

4 Locate the **Source Boundaries** section. From the **Selection** list, choose **contact_holder**.

5 Locate the **Destination Boundaries** section. From the **Selection** list, choose **contact_blank_up**.

6 In the **Label** text field, type `holder_blank`.

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 Click **Add to Component** in the window toolbar.

5 In the tree, select **Built-in>Aluminum**.

6 Click **Add to Component** in the window toolbar.

7 In the **Home** toolbar, click **Add Material** to close the **Add Material** window.

MATERIALS

Aluminum (mat2)

1 In the **Settings** window for **Material**, locate the **Geometric Entity Selection** section.

2 From the **Selection** list, choose **blank**.

SOLID MECHANICS (SOLID)

Linear Elastic Material 1

Set the blank domain with an elastoplatic material model.

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 In the **Settings** window for **Plasticity**, locate the **Domain Selection** section.

3 From the **Selection** list, choose **blank**.

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

5 From the **Yield function F** list, choose **Hill orthotropic plasticity**.

6 From the **Specify** list, choose **Hill's coefficients**.

7 Find the **Isotropic hardening model** subsection. From the list, choose **Hardening function**.

Contact 1

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

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

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

4 From the **Solution method** list, choose **Fully coupled**.

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 `1e-4`.

Increase the integration order of the contact equation to improve the numerical accuracy

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

8 In the **Show More Options** dialog box, in the tree, select the check box for the node **Physics>Advanced Physics Options**.

9 Click **OK**.

10 In the **Settings** window for **Contact**, click to expand the **Quadrature Settings** section.

11 Clear the **Use automatic quadrature settings** check box.

12 In the **Integration order** text field, type `6`.

Fix the die and the holder.

Fixed Constraint 1

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

2 In the **Settings** window for **Fixed Constraint**, locate the **Domain Selection** section.

3 From the **Selection** list, choose **die_and_holder**.

Set prescribed displacements for the punch.

Prescribed Displacement 1

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

2 In the **Settings** window for **Prescribed Displacement**, locate the **Domain Selection** section.

3 From the **Selection** list, choose **punch**.

4 Locate the **Prescribed Displacement** section. Select the **Prescribed in r direction** check box.

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

6 In the u_{0z} text field, type `U_punch(p)`.

MATERIALS

Aluminum (mat2)

1 In the **Model Builder** window, under **Component 1 (comp1)>Materials** click **Aluminum (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**
Hill's coefficients	{Hillcoefficients1, Hillcoefficients2, Hillcoefficients3, Hillcoefficients4, Hillcoefficients5, Hillcoefficients6}	`{F, G, H, L, M, N}`	$m^2 \cdot s^4/kg^2$	Elastoplastic material model
Hardening function	sigmagh	`sigma_h(solid.epe)`	Pa	Elastoplastic material model
Density	rho	`2700[kg/m^3]`	kg/m^3	Basic
Young's modulus	E	`70e9[Pa]`	Pa	Young's modulus and Poisson's ratio
Poisson's ratio	nu	`0.33`	1	Young's modulus and Poisson's ratio
Relative permeability	mur_iso ; murii = mur_iso, murij = 0	1	1	Basic
Heat capacity at constant pressure	Cp	`900[J/(kg*K)]`	J/(kg·K)	Basic
Thermal conductivity	k_iso ; kii = k_iso, kij = 0	`238[W/(m*K)]`	W/(m·K)	Basic

Property	Variable	Value	Unit	Property group
Electrical conductivity	sigma_iso ; sigmaii = sigma_iso, sigmaij = 0	3.774e7[S/m]	S/m	Basic
Relative permittivity	epsilonr_iso ; epsilonrii = epsilonr_iso, epsilonrij = 0	1	1	Basic
Coefficient of thermal expansion	alpha_iso ; alphaii = alpha_iso, alphaij = 0	23e-6[1/K]	1/K	Basic
Murnaghan third-order elastic moduli	l	-2.5e11[Pa]	N/m²	Murnaghan
Murnaghan third-order elastic moduli	m	-3.3e11[Pa]	N/m²	Murnaghan
Murnaghan third-order elastic moduli	n	-3.5e11[Pa]	N/m²	Murnaghan
Lamé parameter λ	lambLame	5.1e10[Pa]	N/m²	Lamé parameters
Lamé parameter μ	muLame	2.6e10[Pa]	N/m²	Lamé parameters

MESH 1

Mesh the curved boundaries more densely compared to the flat boundaries. Use a fine mesh for the blank as it is **destination** in all contact pairs.

Mapped 1

1 In the **Mesh** toolbar, click **Mapped**.
2 In the **Settings** window for **Mapped**, locate the **Domain Selection** section.
3 From the **Geometric entity level** list, choose **Domain**.
4 From the **Selection** list, choose **blank**.

Distribution 1

1 Right-click **Mapped 1** and choose **Distribution**.
2 Select Boundary 21 only.
3 Click the **Zoom Extents** button in the **Graphics** toolbar.

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

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

Distribution 2

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

2 Select Boundary 23 only.

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

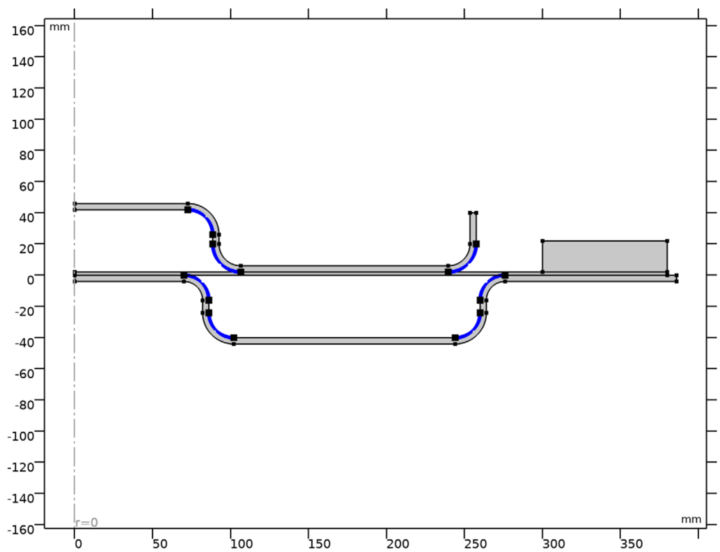
4 In the **Number of elements** text field, type 400.

Edge 1

1 In the **Mesh** toolbar, click **Edge**.

Zoom in and out to select the following boundaries.

2 Select Boundaries 14, 16, 18, 19, 35, 37, and 39 only.



Size 1

1 Right-click **Edge 1** and choose **Size**.

2 In the **Settings** window for **Size**, locate the **Element Size** section.

3 From the **Predefined** list, choose **Extremely fine**.

4 Click the **Custom** button.

5 Locate the **Element Size Parameters** section. Select the **Maximum element size** check box.

6 In the associated text field, type 1.

7 Select the **Minimum element size** check box.

8 In the associated text field, type `0.001`.

Mapped 2

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

2 In the **Settings** window for **Mapped**, locate the **Domain Selection** section.

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

4 Select Domain 4 only.

Distribution 1

1 Right-click **Mapped 2** and choose **Distribution**.

2 Select Boundaries 41 and 43 only.

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

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

Free Triangular 1

In the **Mesh** toolbar, click **Free Triangular**.

Size 1

1 Right-click **Free Triangular 1** and choose **Size**.

2 In the **Settings** window for **Size**, locate the **Element Size** section.

3 From the **Predefined** list, choose **Coarse**.

4 Click **Build All**.

STUDY 1

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

2 Click on **Show Default Solver** in order to customize the solver settings. Use a **Constant (Newton)** method as the nonlinear method in the **Fully Coupled** node. Use a **Linear** predictor is chosen in the **Parametric** node.

Solution 1 (sol1)

In the **Study** toolbar, click **Show Default Solver**.

Step 1: Stationary

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

2 In the **Settings** window for **Stationary**, click to expand the **Study Extensions** section.

3 Select the **Auxiliary sweep** check box.

4 Click **Add**.

5 In the table, enter the following settings:

Parameter name	Parameter value list	Parameter unit
p (Loading parameter)	`range(0,0.01,2)`	

Solution 1 (sol1)

1 In the **Model Builder** window, expand the **Study 1>Solver Configurations>Solution 1 (sol1)>Stationary Solver 1** node, then click **Fully Coupled 1**.

2 In the **Settings** window for **Fully Coupled**, click to expand the **Method and Termination** section.

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

4 From the **Stabilization and acceleration** list, choose **Anderson acceleration**.

5 In the **Model Builder** window, under **Study 1>Solver Configurations>Solution 1 (sol1)>Stationary Solver 1** click **Parametric**.

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

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

8 In the **Minimum step size** text field, type `1e-5`.

9 In the **Maximum step size** text field, type `0.01`.

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

Add a stop condition to prevent computation after full release. The stop condition is controlled by the gap distance between punch and blank.

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

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

13 Click + **Add**.

14 In the table, enter the following settings:

Stop expression	Stop if	Active	Description
`comp1.solid.gapmin_ap2>1e-4`	True (>=1)	√	`Stop expression 1`

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

Click on **Get Initial Value** in order to generate the default plots, so that they can be modified, and can be used for visualization while solving.

16 In the **Study** toolbar, click **Get Initial Value**.

RESULTS

Stress (solid)

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

2 From the **Frame** list, choose **Spatial (r, phi, z)**.

STUDY 1

Step 1: Stationary

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

4 From the **Update at** list, choose **Steps taken by solver**.

Now compute the solution.

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

RESULTS

Residual Stress

The default plot shows the von Mises stress after release of the punch as in Figure 3:

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

2 In the **Residual Stress** toolbar, click **Plot**.

The following steps create the plot in Figure 1:

Study 1/Solution 1, Blank

1 In the **Model Builder** window, expand the **Results>Datasets** node.

2 Right-click **Results>Datasets>Study 1/Solution 1 (sol1)** and choose **Duplicate**.

3 In the **Settings** window for **Solution**, type `Study 1/Solution 1, Blank` in the **Label** text field.

Selection

1 In the **Results** toolbar, click **Attributes** and choose **Selection**.

2 In the **Settings** window for **Selection**, locate the **Geometric Entity Selection** section.

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

4 Select Domain 2 only.

Revolution 2D 1

1 In the **Model Builder** window, under **Results>Datasets** click **Revolution 2D 1**.

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

3 From the **Dataset** list, choose **Study 1/Solution 1, Blank (sol1)**.

4 Click to expand the **Revolution Layers** section. In the **Start angle** text field, type `0`.

5 In the **Revolution angle** text field, type `360`.

Deformed Shape, 3D

1 In the **Model Builder** window, under **Results** click **Stress, 3D (solid)**.

2 In the **Settings** window for **3D Plot Group**, type `Deformed Shape, 3D` in the **Label** text field.

3 Click to expand the **Title** section. From the **Title type** list, choose **Label**.

4 Locate the **Plot Settings** section. Clear the **Plot dataset edges** check box.

Surface 1

1 In the **Model Builder** window, expand the **Deformed Shape, 3D** node, then click **Surface 1**.

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

3 Select the **Description** check box.

4 In the associated text field, type `Deformed shape, 3D`.

Material Appearance 1

1 Right-click **Surface 1** and choose **Material Appearance**.

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

3 From the **Material** list, choose **Aluminum (mat2)**.

4 In the **Deformed Shape, 3D** toolbar, click **Plot**.

The following steps create the plot in Figure 4:

Equivalent Plastic Strain (solid)

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

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

3 Clear the **Plot dataset edges** check box.

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**.

The following steps create the plot in Figure 5:

Blank Thickness

1 In the **Home** toolbar, click **Add Plot Group** and choose **2D Plot Group**.

2 In the **Settings** window for **2D Plot Group**, type `Blank Thickness` in the **Label** text field.

3 Locate the **Plot Settings** section. From the **Frame** list, choose **Spatial (r, phi, z)**.

Surface 1

1 Right-click **Blank Thickness** and choose **Surface**.

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

3 In the **Expression** text field, type `th`.

Deformation 1

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

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

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

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

5 In the **Blank Thickness** toolbar, click **Plot**.

The following steps create the plot in Figure 6:

Punch Force

1 In the **Home** toolbar, click **Add Plot Group** and choose **1D Plot Group**.

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

Global 1

1 Right-click **Punch Force** and choose **Global**.

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

3 In the table, enter the following settings:

Expression	Unit	Description
`-intop1(solid.RFz)`	kN	Punch force

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

5 In the **Expression** text field, type `-U_punch(p)`.

6 Select the **Description** check box.

7 In the associated text field, type `Vertical displacement`.

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

9 In the **Punch Force** toolbar, click **Plot**.