Model created in COMSOL Multiphysics 6.4

Powder Compaction of a Rotational Flanged Component

Introduction

The powder compaction process is common in the manufacturing industry, thanks to its potential to produce components of complex shape and high strength. Because the mechanical properties of the produced component depend on the final density, uniform densification is important. Large variations in density can make a part weak, which affects the overall quality of the component. Finite element analysis with a properly chosen constitutive material model is a handy tool that can provide detailed information about densification, punch forces, friction phenomena, plastic deformation, and internal stresses, in order to better understand the process and to improve the quality of the manufacturing process.

The constitutive material models that are relevant for this type of simulations can broadly be classified in two types:

1. Porous material models — These can be used for the compaction of medium to low porosity powder. Two examples are the Shima–Oyane and Gurson material models.
2. Granular material models — These can be used for the compaction of high porosity powder. Examples of this class of material models include the Capped Drucker–Prager and Capped Mohr–Coulomb models.

This example studies the compaction of a rotational flanged component made of iron powder. As the porosity of the powder is large before compaction, a granular material model like the Capped Drucker–Prager (DPC) model is best suited as argued in Ref. 1 and Ref. 2. The elastic regime is represented by a linear elastic material, while a multiplicative plastic strain formulation together with a DPC yield function is used for the plastic regime. Additionally, friction between the powder and the die is taken into account. Simultaneous movement of the top and bottom punches is applied to avoid mesh distortion and other numerical problems. The simulation is performed using a 2D structured mesh with a linear displacement field as reported in Ref. 2.

Model Definition

The geometry of the workpiece (metal powder) and the die is shown in Figure 1. The punch that compacts the workpiece is not modeled. Instead, a prescribed displacement in the normal direction is used to compact the powder. Due to the axial symmetry, the size of model can be reduced.

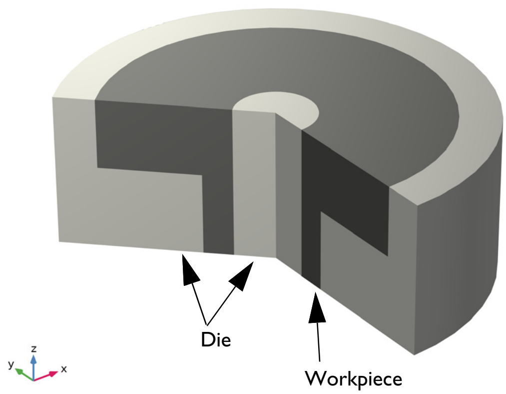

Figure 1: Geometry of the workpiece (metal powder) and the die.

MATERIAL PROPERTIES

For the iron metal powder, an elastoplastic material model with a constitutive relation given by a combination of the linear elastic material model and the Capped Drucker–Prager (DPC) model is used. The material parameters are listed below.

MATERIAL PARAMETER	VALUE
Young's modulus	2000 MPa
Poisson's ratio	0.37
Yield function parameter α_1	1.3
Initial yield stress σ_{ys0}	43.5 MPa
Initial void volume fraction	0.4
Hardening modulus	100 MPa
Maximum plastic volumetric strain	0.65
Initial ellipse centroid	1.25 MPa]
Initial pressure limit	10 MPa

The material of the die is assumed to be rigid. Therefore, it does no require any physics nor material properties.

BOUNDARY CONDITIONS

A prescribed displacement boundary condition is used for the upper and lower faces of the metal powder. The displacement in the z direction is controlled by an interpolation function.

CONTACT

- Contact pairs are defined with boundaries on the die selected as source, and boundaries on the workpiece selected as destination.
- Contact with Coulomb friction is considered using the Augmented Lagrangian method. The coefficient of friction is chosen as 0.08.

The mesh on the source only needs to resolve the geometry of the contact surface. Hence no mesh is needed for the domain, but in order to show the die domains in the postprocessing plots the die domains are coarsely meshed.

Results and Discussion

Figure 2 shows the volumetric plastic strain at the end of the compaction process. At the middle corner, the compressive volumetric plastic strain is at its minimum, while it at the top corner is at its maximum.

The compaction process reduces the porosity of the iron powder and increases its density. This process also results in an increase in the strength of the component. Considering the type of geometry and loading, nonuniform changes in porosity are expected. Contours of the current relative density at the middle and at the end of the compaction are shown in Figure 3 and Figure 4, respectively. The metal powder in the thin lower portion of the workpiece is more compacted than the material in the middle or top portion. At the corners, the metal powder is less compacted due to the friction effects

Lastly, the von Mises stress along in the workpiece at the end of compaction is shown in Figure 5 in a 3D representation of the model.

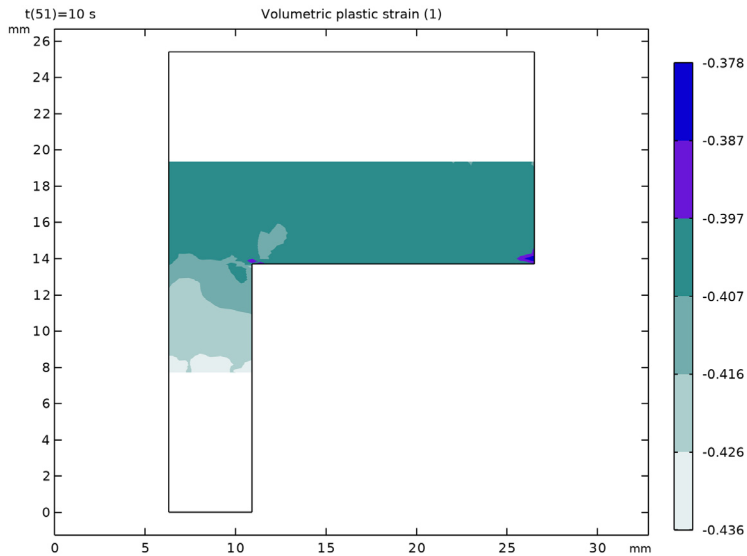

Figure 2: Volumetric plastic strain at the end of compaction.

.

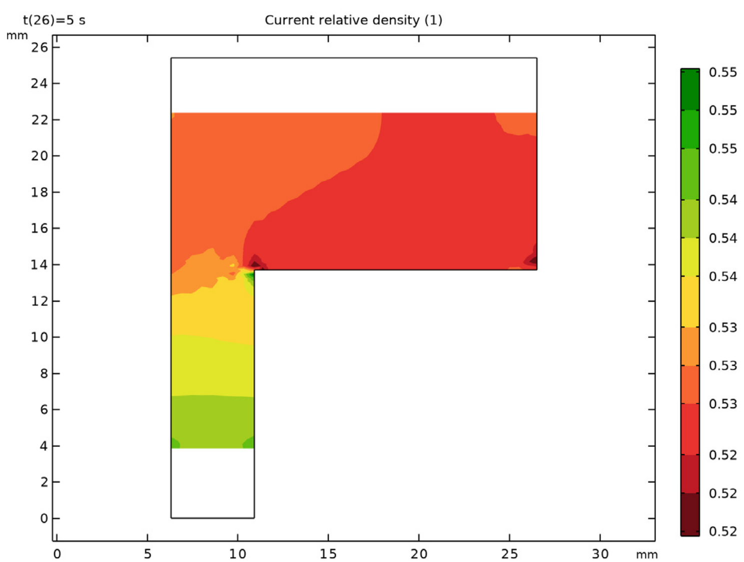

Figure 3: Current relative density in the middle of compaction.

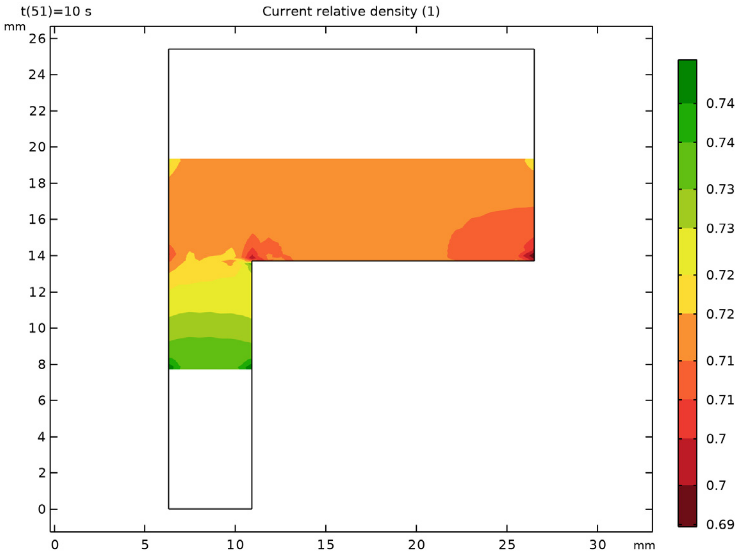

Figure 4: Current relative density at the end of compaction.

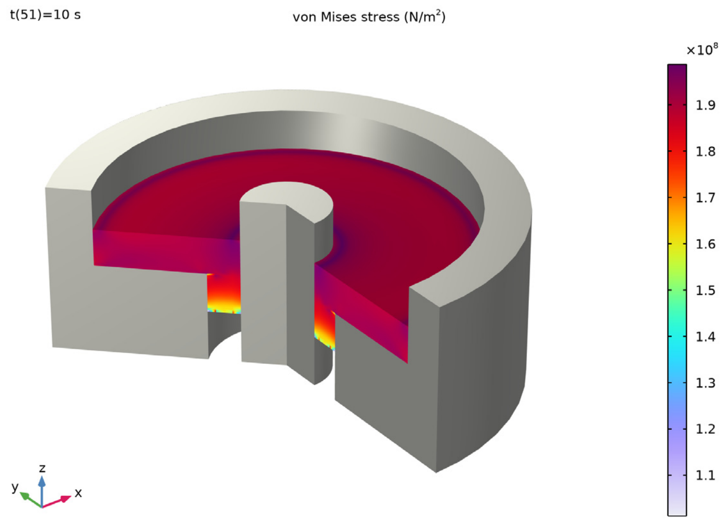

Figure 5: Distribution of the von Mises stress in the workpiece at the end of the compaction process.

Notes About the COMSOL Implementation

The density distribution in the compaction process is affected by the contact pressure and the friction forces, so the accuracy of the contact algorithm is important in the powder compaction process. In this example, the **Augmented Lagrangian** method with a **Fully Coupled** solver is used.

References

1. A. Perez-Foguet, A. Rodriguez-Ferran, and A. Huerta, "Consistent tangent matrices for density-dependent finite plasticity models," *Int. J. Numer. Anal. Meth. Geomech.*, vol. 25, pp. 1045–1075, 2001.

2. A.R. Khoei, A. Shamloo, and A.R. Azami, "Extended finite element method in plasticity forming of powder compaction with contact friction," *Int.J. Solids Struct.*, vol. 43, pp. 5421–5448, 2006..

Application Library path: `Nonlinear_Structural_Materials_Module/Porous_Plasticity/compaction_of_a_rotational_flange`

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. Click **Study**.
5. In the **Select Study** tree, select **General Studies** > **Stationary**.
6. 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
t	0[s]	0 s	Time parameter

DEFINITIONS

Top Punch Displacement

1. In the **Definitions** toolbar, click **Interpolation**.
2. In the **Settings** window for **Interpolation**, type `Top Punch Displacement` in the **Label** text field.
3. Locate the **Definition** section. In the **Function name** text field, type `disp1`.

4 In the table, enter the following settings:

t	f(t)
0	0
10	-6.06

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

Argument	Unit
t	s

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

Function	Unit
disp1	mm

Bottom Punch Displacement

1 Right-click **Top Punch Displacement** and choose **Duplicate**.

2 In the **Settings** window for **Interpolation**, type `Bottom Punch Displacement` in the **Label** text field.

3 Locate the **Definition** section. In the table, enter the following settings:

t	f(t)
10	7.7

GEOMETRY 1

Rectangle 1 (r1)

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

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

3 In the **Settings** window for **Geometry**, locate the **Units** section.

4 From the **Length unit** list, choose **mm**.

5 In the **Model Builder** window, click **Rectangle 1 (r1)**.

6 In the **Settings** window for **Rectangle**, locate the **Size and Shape** section.

7 In the **Width** text field, type `6.3`.

8 In the **Height** text field, type `25.4`.

9 Click **Build Selected**.

Rectangle 2 (r2)

1 Right-click **Rectangle 1 (r1)** and choose **Duplicate**.

2 In the **Settings** window for **Rectangle**, locate the **Size and Shape** section.

3 In the **Width** text field, type `4.6`.

4 In the **Height** text field, type `13.7`.

5 Locate the **Position** section. In the **r** text field, type `6.3`.

6 Click **Build Selected**.

Rectangle 3 (r3)

1 Right-click **Rectangle 2 (r2)** and choose **Duplicate**.

2 In the **Settings** window for **Rectangle**, locate the **Size and Shape** section.

3 In the **Width** text field, type `20.2`.

4 In the **Height** text field, type `11.7`.

5 Locate the **Position** section. In the **z** text field, type `13.7`.

6 Click **Build Selected**.

Union 1 (uni1)

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

2 Select the objects **r2** and **r3** only.

3 In the **Settings** window for **Union**, click **Build Selected**.

Rectangle 4 (r4)

1 In the **Model Builder** window, under **Component 1 (comp1)** > **Geometry 1** right-click **Rectangle 3 (r3)** and choose **Duplicate**.

2 In the **Settings** window for **Rectangle**, locate the **Size and Shape** section.

3 In the **Width** text field, type `15.6`.

4 In the **Height** text field, type `13.7`.

5 Locate the **Position** section. In the **r** text field, type `10.9`.

6 In the **z** text field, type `0`.

7 Click **Build Selected**.

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

Rectangle 5 (r5)

1 Right-click **Rectangle 4 (r4)** and choose **Duplicate**.

2 In the **Settings** window for **Rectangle**, locate the **Size and Shape** section.

3 In the **Width** text field, type `6.3`.

4 In the **Height** text field, type `25.4`.

5 Locate the **Position** section. In the **r** text field, type `26.5`.

6 Click **Build Selected**.

Union 2 (uni2)

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

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

3 Select the objects **r4** and **r5** only.

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

5 Clear the **Keep interior boundaries** checkbox.

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**.

Internal Die

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

Rectangle 2 (r2), Rectangle 3 (r3), Union 1 (uni1)

1 In the **Model Builder** window, under **Component 1 (comp1)** > **Geometry 1**, Ctrl-click to select **Rectangle 2 (r2)**, **Rectangle 3 (r3)**, and **Union 1 (uni1)**.

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

Compact

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

Rectangle 4 (r4), Rectangle 5 (r5), Union 2 (uni2)

1 In the **Model Builder** window, under **Component 1 (comp1)** > **Geometry 1**, Ctrl-click to select **Rectangle 4 (r4)**, **Rectangle 5 (r5)**, and **Union 2 (uni2)**.

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

External Die

In the **Settings** window for **Group**, type `External Die` 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 Click **Build Selected**.

Add the interior edge in the workpiece geometry to **Mesh Control Edges** in order to generate a structured mesh.

Mesh Control Edges 1 (mce1)

1 In the **Geometry** toolbar, click **Virtual Operations** and choose **Mesh Control Edges**.

2 On the object **fin**, select Boundary 8 only.

3 In the **Settings** window for **Mesh Control Edges**, click **Build Selected**.

In subsequent steps, the side domain will be not be part of the physics. Hence use the toggle button in **Contact Pair** to switch the boundaries, so that the workpiece boundaries are chosen as destination boundaries.

DEFINITIONS

Contact Pair 2 (ap2)

1 In the **Model Builder** window, under **Component 1 (comp1)** > **Definitions** click **Contact Pair 2 (ap2)**.

2 In the **Settings** window for **Pair**, click the **Swap Source and Destination** button.

3 Locate the **Source Boundaries** section. Click to select the **Activate Selection** toggle button.

4 Locate the **Destination Boundaries** section. Click to select the **Activate Selection** toggle button.

Domains 1 and 3 (die) are considered as rigid and fixed, hence there is no need to consider them in physics, only a mesh is required.

Change the discretization to **Linear**.

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 **Domain Selection** section.

3 Click **Clear Selection**.

4 Select Domain 2 only.

5 Click to expand the **Discretization** section. From the **Displacement field** list, choose **Linear**.

For the elastoplastic analysis of the workpiece, choose a **Capped Drucker–Prager** model by adding a **Porous Plasticity** subnode to the **Linear Elastic Material**. Set the formulation to **Total Lagrangian** to force large strains to be used.

Linear Elastic Material 1

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

2 In the **Settings** window for **Linear Elastic Material**, locate the **Geometric Nonlinearity** section.

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

4 From the **Strain decomposition** list, choose **Multiplicative**.

Porous Plasticity 1

1 In the **Physics** toolbar, click **Attributes** and choose **Porous Plasticity**.

2 In the **Settings** window for **Porous Plasticity**, locate the **Porous Plasticity Model** section.

3 From the **Material model** list, choose **Capped Drucker–Prager**.

4 Locate the **Cap Model** section. From the **Hardening model** list, choose **Exponential**.

For better accuracy, select the **Augmented Lagrangian** with a **Fully Coupled** solution method.

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 list, choose **Augmented Lagrangian**.

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

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 `0.08`.

Prescribed Displacement 1

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

2 Select Boundary 7 only.

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

4 From the **Displacement in z direction** list, choose **Prescribed**.

5 In the u_{0z} text field, type `disp1(t)`.

Prescribed Displacement 2

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

2 Select Boundary 6 only.

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

4 From the **Displacement in z direction** list, choose **Prescribed**.

5 In the u_{0z} text field, type `disp2(t)`.

MATERIALS

Iron Powder

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

2 In the **Settings** window for **Material**, type `Iron Powder` in the **Label** text field.

3 Select Domain 2 only.

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

Property	Variable	Value	Unit	Property group
Young's modulus	E	2[GPa]	Pa	Young's modulus and Poisson's ratio
Poisson's ratio	nu	0.37	1	Young's modulus and Poisson's ratio
Density	rho	7540	kg/m³	Basic
Initial yield stress	sigmags	43.5[MPa]	Pa	Poroplastic material model
Initial void volume fraction	f0	0.6	1	Poroplastic material model
Yield function parameter	a1yield	1.3	1	Pressure-dependent plasticity
Initial pressure limit	pc0	10[MPa]	Pa	Cap and cutoff

Property	Variable	Value	Unit	Property group
Initial ellipse centroid	pcc0	1.25[MPa]	Pa	Cap and cutoff
Hardening modulus	Kc	100[MPa]	Pa	Cap and cutoff
Maximum volumetric plastic strain	epvolmax	0.65	1	Cap and cutoff

MESH 1

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 Select Domain 2 only.
5 Click to expand the **Control Entities** section. From the **Smooth across removed control entities** list, choose **Off**.
6 Click to expand the **Reduce Element Skewness** section. Select the **Adjust edge mesh** checkbox.

Distribution 1

1 Right-click **Mapped 1** and choose **Distribution**.
2 Select Boundaries 6 and 19 only.
3 In the **Settings** window for **Distribution**, locate the **Distribution** section.
4 In the **Number of elements** text field, type 4.

Distribution 2

1 In the **Model Builder** window, right-click **Mapped 1** and choose **Distribution**.
2 Select Boundary 5 only.
3 In the **Settings** window for **Distribution**, locate the **Distribution** section.
4 In the **Number of elements** text field, type 20.

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.

5 Locate the **Control Entities** section. From the **Smooth across removed control entities** list, choose **Off**.

6 Locate the **Reduce Element Skewness** section. Select the **Adjust edge mesh** checkbox.

Distribution 1

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

2 Select Boundaries 7 and 18 only.

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

4 In the **Number of elements** text field, type `20`.

Distribution 2

1 In the **Model Builder** window, right-click **Mapped 2** and choose **Distribution**.

2 Select Boundary 13 only.

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

4 In the **Number of elements** text field, type `16`.

Size

1 In the **Model Builder** window, under **Component 1 (comp1)** > **Mesh 1** click **Size**.

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

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

Free Quad 1

1 In the **Mesh** toolbar, click **Free Quad**.

2 In the **Settings** window for **Free Quad**, click **Build All**.

Augmented Lagrangian contact in addition to the material and geometric nonlinearity in the model demands special solver settings to achieve smooth convergence.

STUDY 1

Step 1: Stationary

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

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** checkbox.

4 Click **Add**.

5 In the table, enter the following settings:

Parameter name	Parameter value list	Parameter unit
t (Time parameter)	range(0,0.2,10)	s

Solution 1 (sol1)

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

2 In the **Model Builder** window, expand the **Solution 1 (sol1)** node.

Set up the solver in order to improve the convergence.

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

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

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

6 In the **Minimum step size** text field, type 0.0005.

7 In the **Maximum step size** text field, type 0.2.

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

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

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

11 From the **Nonlinear method** list, choose **Automatic (Newton)**.

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

RESULT TEMPLATES

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

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

3 In the tree, select **Study 1/Solution 1 (sol1)** > **Solid Mechanics** > **Volumetric Plastic Strain (solid)** and **Study 1/Solution 1 (sol1)** > **Solid Mechanics** > **Current Void Volume Fraction (solid)**.

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

5 In the **Results** toolbar, click **Result Templates** to close the **Result Templates** window.

RESULTS

Study 1/Solution 1 (sol1)

In the **Model Builder** window, expand the **Results** > **Datasets** node, then click **Study 1/ Solution 1 (sol1)**.

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.

In order to visualize the von Mises stress in the deformed workpiece along with the undeformed dies, duplicate the **Study 1/Solution 1** dataset, and set the selection to domains 1 and 3. Set up a new **Revolution 2D** dataset based on **Study 1/Solution 1 (2)**.

Study 1/Solution 1 (2) (sol1)

Right-click **Study 1/Solution 1 (sol1)** and choose **Duplicate**.

Selection

1 In the **Model Builder** window, expand the **Study 1/Solution 1 (2) (sol1)** node, then click **Selection**.

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

3 Click **Clear Selection**.

4 Select Domains 1 and 3 only.

Revolution 2D 1

1 In the **Model Builder** window, right-click **Revolution 2D** and choose **Duplicate**.

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

3 From the **Dataset** list, choose **Study 1/Solution 1 (2) (sol1)**.

In order to visualize the undeformed dies, set up **Surface 2** node in the next 3D plot with a zero expression. Select the **Revolution 2D 2** dataset, and add a **Material Appearance** node for the visualization.

Stress, 3D (solid)

1 In the **Model Builder** window, expand the **Results** > **Stress, 3D (solid)** node, then click **Stress, 3D (solid)**.

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

3 Clear the **Plot dataset edges** checkbox.

Surface 2

1 Right-click **Stress, 3D (solid)** and choose **Surface**.

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

3 From the **Dataset** list, choose **Revolution 2D 1**.

4 Locate the **Expression** section. In the **Expression** text field, type `0`.

5 Click to expand the **Title** section. From the **Title type** list, choose **None**.

Material Appearance 1

1 Right-click **Surface 2** and choose **Material Appearance**.

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

3 From the **Appearance** list, choose **Custom**.

4 From the **Material type** list, choose **Steel**.

5 In the **Stress, 3D (solid)** toolbar, click **Plot**.

Volumetric Plastic Strain (solid)

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

2 In the **Settings** window for **2D Plot Group**, click to expand the **Number Format** section.

3 Select the **Manual color legend settings** checkbox.

4 In the **Precision** text field, type `4`.

Surface 1

1 In the **Model Builder** window, expand the **Volumetric Plastic Strain (solid)** node, then click **Surface 1**.

2 In the **Settings** window for **Surface**, locate the **Coloring and Style** section.

3 In the **Number of bands** text field, type `6`.

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** checkbox. In the associated text field, type `1`.

4 In the **Volumetric Plastic Strain (solid)** toolbar, click **Plot**.

Current Relative Density at Middle of Compaction

1 In the **Model Builder** window, under **Results** click **Current Void Volume Fraction (solid)**.

2 In the **Settings** window for **2D Plot Group**, type `Current Relative Density at Middle of Compaction` in the **Label** text field.

3 Locate the **Data** section. From the **Parameter value (t (s))** list, choose **5**.

Surface 1

1 In the **Model Builder** window, expand the **Current Relative Density at Middle of Compaction** node, then click **Surface 1**.

2 In the **Settings** window for **Surface**, click **Replace Expression** in the upper-right corner of the **Expression** section. From the menu, choose **Component 1 (comp1)** > **Solid Mechanics** > **Porous plasticity** > **solid.rhorelGp - Current relative density - 1**.

3 Locate the **Expression** section. Clear the **Description** checkbox.

4 Locate the **Coloring and Style** section. From the **Color table transformation** list, choose **Reverse**.

5 In the **Current Relative Density at Middle of Compaction** toolbar, click **Plot**.

Current Relative Density at End of Compaction

1 In the **Model Builder** window, right-click **Current Relative Density at Middle of Compaction** and choose **Duplicate**.

2 In the **Settings** window for **2D Plot Group**, type `Current Relative Density at End of Compaction` in the **Label** text field.

3 Locate the **Data** section. From the **Parameter value (t (s))** list, choose **10**.

4 In the **Current Relative Density at End of Compaction** toolbar, click **Plot**.