Created in COMSOL Multiphysics 5.6

Powder Compaction of a Rotational Flanged Component

Introduction

The powder compaction process is becoming common in the manufacturing industry, thanks to its potential to produce components of complex shape and high strength. The mechanical properties of the produced component depends on the final density, therefore the importance of uniform densification. 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.

In 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 large 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.

The same example is studied in Ref. 1 with isotropic multiplicative hyperelastoplasticity, and in Ref. 2 with isotropic additive nonlinear elastoplasticity. In COMSOL Multiphysics, a linear elastoplastic material model is used in the geometric nonlinear regime with a multiplicative finite plastic strain formulation. The elastoplastic material model used in COMSOL Multiphysics is different from that used in Ref. 1 and Ref. 2, so the material parameters are chosen such as to get results in a similar range.

Before analyzing the rotational flanged component, isotropic compression and triaxial tests are carried out with the chosen material model and properties. The simulation results are then compared with experimental results given in Ref. 3. A good agreement is obtained between results for the isotropic compression test, and an acceptable agreement is obtained for the triaxial test. In this model, the isotropic and triaxial tests are not presented.

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.

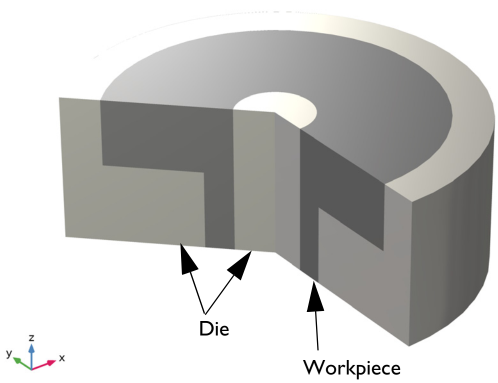

Figure 1: Geometry of the workpiece (metal powder) and the die.

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
Drucker-Prager alpha coefficient	0.25
Drucker-Prager k coefficient	25 MPa
Initial void volume fraction	0.4
Isotropic hardening modulus	100 MPa

MATERIAL PARAMETER	VALUE
Maximum plastic volumetric strain	0.525
Ellipse aspect ratio	1.5
Initial location of compressive cap	10 MPa

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

BOUNDARY CONDITIONS

A prescribed displacement boundary condition 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.
- Since no physics is defined on the die, it is in the **Contact** node considered as external to the current physics.

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

As discussed in the introduction section of this documentation, the isotropic compression and triaxial tests are carried out with the same material model and properties. However, these tests are not part of the example, and only the results are presented for validation. A sample with an initial height of 24 mm and a diameter of 20 mm was used, see Ref. 1. The triaxial test is carried out by compacting the sample isotropically up to 50 MPa, and then axially compressing it while keeping the radial pressure constant at 50 MPa.

The simulation results are then compared with experimental results given in Ref. 3. Figure 2 shows that a good agreement is obtained between the simulation andthe experimental results for the isotropic compression test. Results from the triaxial test is presented in Figure 3 and Figure 4 that shows the variation of the current density and axial Cauchy stress with axial logarithmic strain. Both are in acceptable agreement with the experimental results from the triaxial test. The large difference in the evolution of the density in the triaxial test is due to a large difference at the initial isotropic stage, see

Figure 2. Considering the fact that the elastoplastic material model used in COMSOL Multiphysics is different than that used in Ref. 1 and Ref. 2, differences in the results are expected. Nevertheless, the results show the same trend and are within an acceptable level. The results indicate that the chosen material model and properties can be used for analysis of the compaction of rotational flanged component, whose results are presented next.

.

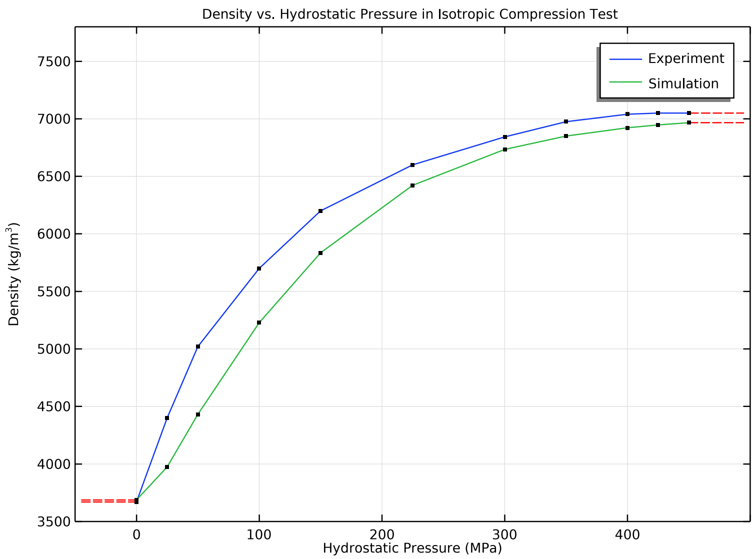

Figure 2: Current density versus hydrostatic pressure in the isotropic compression test.

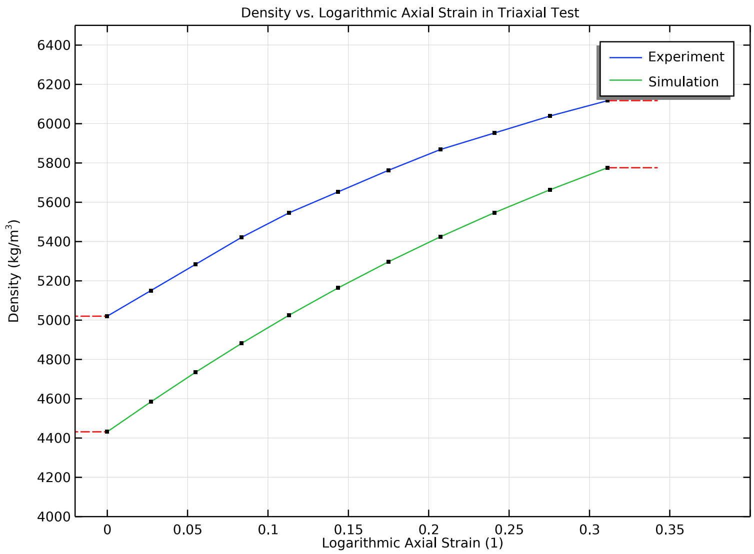

Figure 3: Current density versus axial logarithmic strain in the triaxial test.

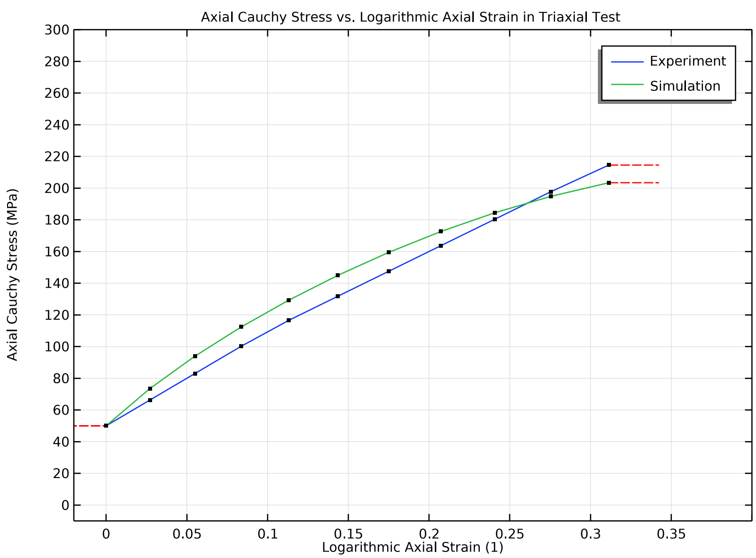

Figure 4: Axial Cauchy stress versus axial logarithmic strain in the triaxial strain.

Figure 5 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 in the middle and at the end of the compaction are shown in Figure 6 and Figure 7, 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, Ref. 2. Note that the relative densities presented in Ref. 1 and Ref. 2 do not match each other exactly due to the different elastoplastic material models and material properties, but they do show a similar trend. Along the same line, the results presented in the current example show the same trend as Ref. 1 and Ref. 2, and are also in close range with them.

Lastly, the von Mises stress along in the workpiece at the end of compaction is shown in Figure 8 in a 3D representation of the model.

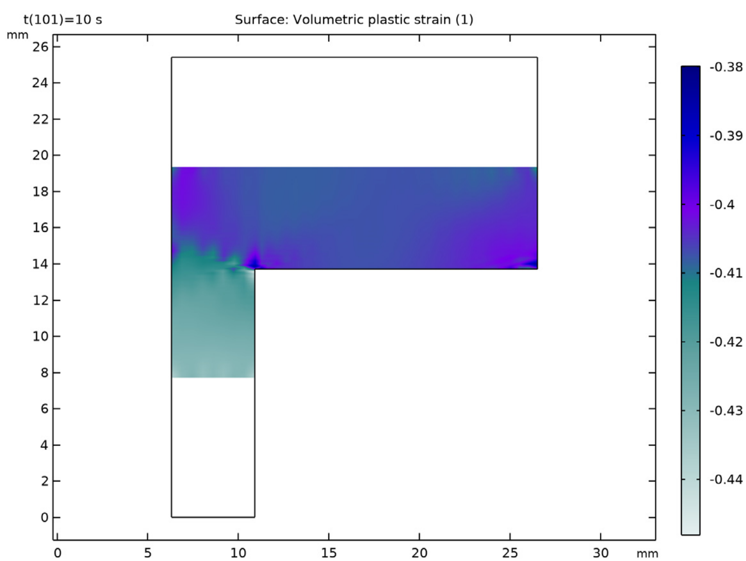

Figure 5: Volumetric plastic strain at the end of compaction.

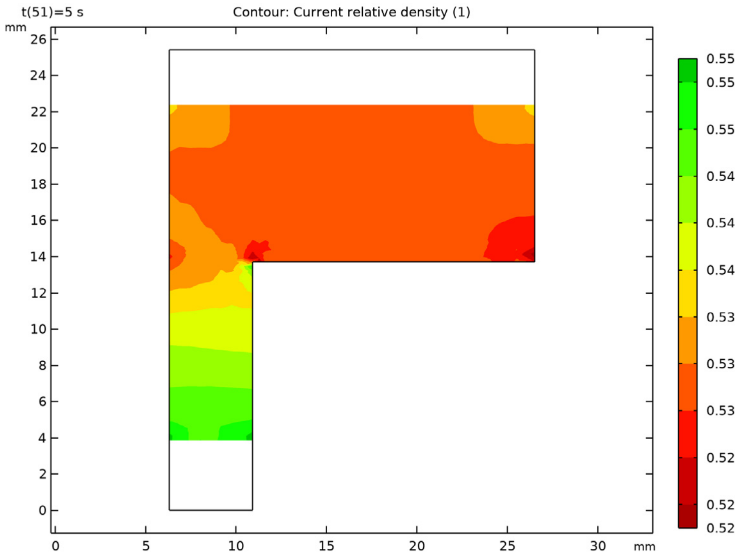

Figure 6: Current relative density in the middle of compaction.

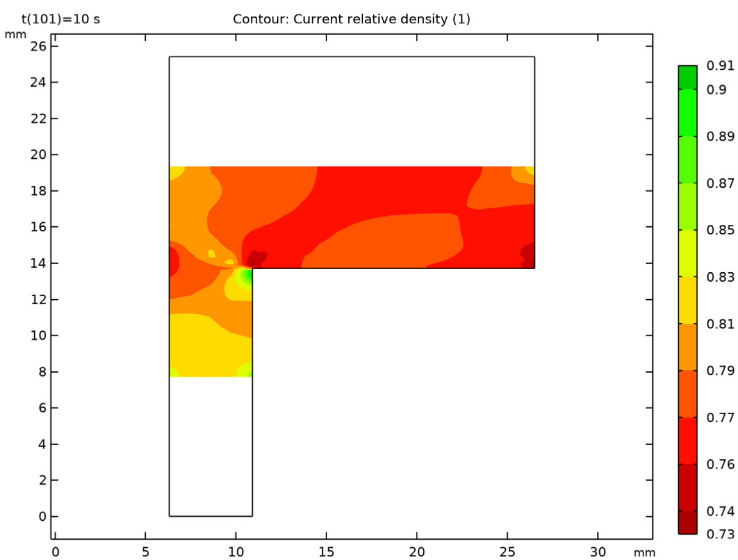

Figure 7: Current relative density at the end of compaction.

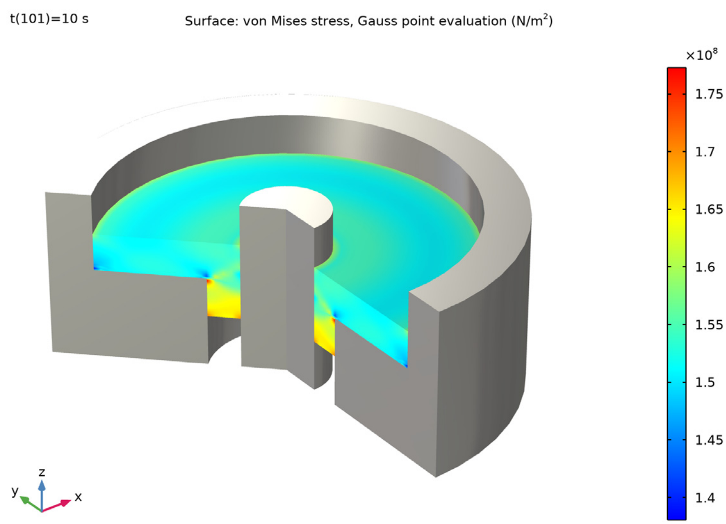

Figure 8: Distribution of the von Mises stress in the workpiece at the end of the compaction process.

Notes About the COMSOL Implementation

The default contact method in COMSOL Multiphysics is the *Penalty* method, while the *Augmented Lagrangian* method is a more accurate but can also be more computationally expensive. The *Augmented Lagrangian* method comes in two flavors based on solution strategy: *Fully Coupled* and *Segregated*. Each flavor has its own advantages, see the *COMSOL Multiphysics Reference Manual* for further details.

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. To get smooth convergence, parametric steps are adjusted and the *Constant Newton* method with a *Linear* predictor 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.

3. P. Doremus, C.Geindreau, A. Martin, L.Debove, R. Lecot, and M. Dao, “*High Pressure Triaxial Cell for Metal Powder*”, Powder Metallurgy, vol. 38, no. 4, pp. 284–287, 1995.

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
mu	0.08	0.08	Coefficient of friction

Name	Expression	Value	Description
EE	2000[MPa]	2E9 Pa	Young's modulus
Nu	0.37	0.37	Poisson's ratio
rhorel0	0.4	0.4	Initial relative density
F0	1-rhorel0	0.6	Initial void volume fraction
Alpha	0.25	0.25	Drucker-Prager parameter
Kd	25[MPa]	2.5E7 Pa	Drucker-Prager parameter
KIso	100[MPa]	1E8 Pa	Isotropic hardening modulus
Rc	1.5	1.5	Ellipse aspect ratio
Epvolmax	0.525	0.525	Maximum plastic volumetric strain
pc0	10[MPa]	1E7 Pa	Initial location of the hardening cap
Rho	7540[kg/m^3]	7540 kg/m³	Final density of the component

DEFINITIONS

Before creating the geometry and setting up the physics, import the results of the isotropic compression test and the triaxial test done on an iron powder specimen with the same material properties and constitutive model as used in this example. Import also the experimental results of the isotropic compression test and the triaxial test done in Ref. 3.

Density in Isotropic Compression Test - Experiment

1. In the **Model Builder** window, expand the **Component 1 (comp1)>Definitions** node.
2. Right-click **Definitions** and choose **Functions>Interpolation**.
3. In the **Settings** window for **Interpolation**, type `Density in Isotropic Compression Test - Experiment` in the **Label** text field.
4. Locate the **Definition** section. Click **Load from File**.
5. Browse to the model's Application Libraries folder and double-click the file `compaction_of_a_rotational_flange_isotropiccompression_experiment.txt`.

Density in Isotropic Compression Test - Simulation

1. In the **Home** toolbar, click f(X) **Functions** and choose **Local>Interpolation**.
2. In the **Settings** window for **Interpolation**, type `Density in Isotropic Compression Test - Simulation` in the **Label** text field.

3 Locate the **Definition** section. Click **Load from File**.

4 Browse to the model's Application Libraries folder and double-click the file `compaction_of_a_rotational_flange_isotropiccompression_simulation.txt`.

Density in Triaxial Test - Experiment

1 In the **Home** toolbar, click f(X) **Functions** and choose **Local>Interpolation**.

2 In the **Settings** window for **Interpolation**, type `Density in Triaxial Test - Experiment` in the **Label** text field.

3 Locate the **Definition** section. Click **Load from File**.

4 Browse to the model's Application Libraries folder and double-click the file `compaction_of_a_rotational_flange_triaxial_experiment1.txt`.

Density in Triaxial Test - Simulation

1 In the **Home** toolbar, click f(X) **Functions** and choose **Local>Interpolation**.

2 In the **Settings** window for **Interpolation**, type `Density in Triaxial Test - Simulation` in the **Label** text field.

3 Locate the **Definition** section. Click **Load from File**.

4 Browse to the model's Application Libraries folder and double-click the file `compaction_of_a_rotational_flange_triaxial_simulation1.txt`.

Axial Stress in Triaxial Test - Experiment

1 In the **Home** toolbar, click f(X) **Functions** and choose **Local>Interpolation**.

2 In the **Settings** window for **Interpolation**, type `Axial Stress in Triaxial Test - Experiment` in the **Label** text field.

3 Locate the **Definition** section. Click **Load from File**.

4 Browse to the model's Application Libraries folder and double-click the file `compaction_of_a_rotational_flange_triaxial_experiment2.txt`.

Axial Stress in Triaxial Test - Simulation

1 In the **Home** toolbar, click f(X) **Functions** and choose **Local>Interpolation**.

2 In the **Settings** window for **Interpolation**, type `Axial Stress in Triaxial Test - Simulation` in the **Label** text field.

3 Locate the **Definition** section. Click **Load from File**.

4 Browse to the model's Application Libraries folder and double-click the file `compaction_of_a_rotational_flange_triaxial_simulation2.txt`.

Create a plot from these interpolation functions in order to compare the experimental results with the results of the simulation.

Density in Isotropic Compression Test - Experiment (int1)

1 In the **Model Builder** window, under **Component 1 (comp1)>Definitions** click **Density in Isotropic Compression Test - Experiment (int1)**.
2 In the **Settings** window for **Interpolation**, click **Create Plot**.

Density in Isotropic Compression Test - Simulation (int2)

1 In the **Model Builder** window, under **Component 1 (comp1)>Definitions** click **Density in Isotropic Compression Test - Simulation (int2)**.
2 In the **Settings** window for **Interpolation**, click **Create Plot**.

RESULTS

1D Plot Group 2

Combine the plots of the experimental and the simulation results of same type in one plot group, delete the unnecessary plots.

1 In the **Model Builder** window, under **Results** right-click **1D Plot Group 2** and choose **Delete**.

Density vs. Hydrostatic Pressure in Isotropic Compression Test

1 In the **Model Builder** window, under **Results** click **1D Plot Group 1**.
2 In the **Settings** window for **1D Plot Group**, type Density vs. Hydrostatic Pressure in Isotropic Compression Test 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. Select the **x-axis label** check box.
5 In the associated text field, type Hydrostatic Pressure (MPa).
6 In the **y-axis label** text field, type Density (kg/m³).
7 Locate the **Axis** section. Select the **Manual axis limits** check box.
8 In the **y minimum** text field, type 3500.
9 In the **y maximum** text field, type 7800.
10 In the **x maximum** text field, type 500.
11 In the **x minimum** text field, type -50.

Function 1

1 In the **Model Builder** window, expand the **Density vs. Hydrostatic Pressure in Isotropic Compression Test** node, then click **Function 1**.

2 In the **Settings** window for **Function**, click to expand the **Legends** section.

3 Select the **Show legends** check box.

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

5 In the table, enter the following settings:

Legends
`Experiment`

Function 2

1 Right-click **Results>Density vs. Hydrostatic Pressure in Isotropic Compression Test>Function 1** and choose **Duplicate**.

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

3 From the **Dataset** list, choose **Grid 1D 1a**.

4 Locate the **y-Axis Data** section. In the **Expression** text field, type `comp1.int2(t)`.

5 Locate the **Output** section. From the **Point definition** list, choose **Cut Point 1D 2**.

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

Legends
`Simulation`

7 In the **Density vs. Hydrostatic Pressure in Isotropic Compression Test** toolbar, click **Plot**.

DEFINITIONS

Density in Triaxial Test - Experiment (int3)

1 In the **Model Builder** window, click **Density in Triaxial Test - Experiment (int3)**.

2 In the **Settings** window for **Interpolation**, click **Create Plot**.

Density in Triaxial Test - Simulation (int4)

1 In the **Model Builder** window, under **Component 1 (comp1)>Definitions** click **Density in Triaxial Test - Simulation (int4)**.

2 In the **Settings** window for **Interpolation**, click **Create Plot**.

RESULTS

1D Plot Group 3

In the **Model Builder** window, under **Results** right-click **1D Plot Group 3** and choose **Delete**.

Density vs. Logarithmic Axial Strain in Triaxial Test

1 In the **Model Builder** window, under **Results** click **1D Plot Group 2**.
2 In the **Settings** window for **1D Plot Group**, type `Density vs. Logarithmic Axial Strain in Triaxial Test` in the **Label** text field.
3 Locate the **Title** section. From the **Title type** list, choose **Label**.
4 Locate the **Plot Settings** section. Select the **x-axis label** check box.
5 In the associated text field, type `Logarithmic Axial Strain (1)`.
6 In the **y-axis label** text field, type `Density (kg/m<sup>3</sup>)`.
7 Locate the **Axis** section. Select the **Manual axis limits** check box.
8 In the **y minimum** text field, type `4000`.
9 In the **y maximum** text field, type `6500`.
10 In the **x maximum** text field, type `0.4`.
11 In the **x minimum** text field, type `-0.02`.

Function 1

1 In the **Model Builder** window, expand the **Density vs. Logarithmic Axial Strain in Triaxial Test** node, then click **Function 1**.
2 In the **Settings** window for **Function**, locate the **Legends** section.
3 Select the **Show legends** check box.
4 From the **Legends** list, choose **Manual**.
5 In the table, enter the following settings:

Legends
Experiment

Function 2

1 Right-click **Results>Density vs. Logarithmic Axial Strain in Triaxial Test>Function 1** and choose **Duplicate**.
2 In the **Settings** window for **Function**, locate the **Data** section.
3 From the **Dataset** list, choose **Grid 1D 1c**.
4 Locate the **y-Axis Data** section. In the **Expression** text field, type `comp1.int4(t)`.

5 Locate the **Output** section. From the **Point definition** list, choose **Cut Point 1D 4**.

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

Legends
`Simulation`

7 In the **Density vs. Logarithmic Axial Strain in Triaxial Test** toolbar, click **Plot**.

DEFINITIONS

Axial Stress in Triaxial Test - Experiment (int5)

1 In the **Model Builder** window, click **Axial Stress in Triaxial Test - Experiment (int5)**.

2 In the **Settings** window for **Interpolation**, click **Create Plot**.

Axial Stress in Triaxial Test - Simulation (int6)

1 In the **Model Builder** window, under **Component 1 (comp1)>Definitions** click **Axial Stress in Triaxial Test - Simulation (int6)**.

2 In the **Settings** window for **Interpolation**, click **Create Plot**.

RESULTS

1D Plot Group 4

In the **Model Builder** window, under **Results** right-click **1D Plot Group 4** and choose **Delete**.

Axial Cauchy Stress vs. Logarithmic Axial Strain in Triaxial Test

1 In the **Model Builder** window, under **Results** click **1D Plot Group 3**.

2 In the **Settings** window for **1D Plot Group**, type `Axial Cauchy Stress vs. Logarithmic Axial Strain in Triaxial Test` in the **Label** text field.

3 Locate the **Title** section. From the **Title type** list, choose **Label**.

4 Locate the **Plot Settings** section. Select the **x-axis label** check box.

5 In the associated text field, type `Logarithmic Axial Strain (1)`.

6 In the **y-axis label** text field, type `Axial Cauchy Stress (MPa)`.

7 Locate the **Axis** section. Select the **Manual axis limits** check box.

8 In the **y minimum** text field, type `-10`.

9 In the **y maximum** text field, type `300`.

10 In the **x maximum** text field, type `0.4`.

11 In the **x minimum** text field, type `-0.02`.

Function 1

1 In the **Model Builder** window, expand the **Axial Cauchy Stress vs. Logarithmic Axial Strain in Triaxial Test** node, then click **Function 1**.

2 In the **Settings** window for **Function**, locate the **Legends** section.

3 Select the **Show legends** check box.

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

5 In the table, enter the following settings:

Legends
Experiment

Function 2

1 Right-click **Results>Axial Cauchy Stress vs. Logarithmic Axial Strain in Triaxial Test>Function 1** and choose **Duplicate**.

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

3 From the **Dataset** list, choose **Grid 1D 1e**.

4 Locate the **y-Axis Data** section. In the **Expression** text field, type `comp1.int6(t)`.

5 Locate the **Output** section. From the **Point definition** list, choose **Cut Point 1D 6**.

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

Legends
Simulation

7 In the **Axial Cauchy Stress vs. Logarithmic Axial Strain in Triaxial Test** toolbar, click **Plot**.

Now set up the model for compaction of a rotational flanged component.

DEFINITIONS

Top Punch Displacement

1 In the **Home** toolbar, click f(x) **Functions** and choose **Local>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 **Arguments** text field, type `s`.

6 In the **Function** text field, type `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 **Function name** text field, type `disp2`.

4 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** check box.
6 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 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**, locate the **Input** section.
4 Clear the **Include adjacent vertices** check box.
5 Click **Build Selected**.

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

DEFINITIONS

Contact Pair 2a (ap2)

1 In the **Model Builder** window, under **Component 1 (comp1)>Definitions** click **Contact Pair 2a (ap2)**.
2 In the **Settings** window for **Pair**, click the **Swap Source and Destination** button.
3 Locate the **Source Boundaries** section. Select the **Activate Selection** toggle button.

4 Locate the **Destination Boundaries** section. 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 **Capped Drucker-Prager** yield function with **Large Plastic Strain** porous plasticity model by adding a **Porous Plasticity** subnode to the **Linear Elastic Material**.

Linear Elastic Material 1

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

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 **Plasticity model** list, choose **Large plastic strains**.

4 From the **Yield function F** list, choose **Capped Drucker-Prager**.

5 Find the **Isotropic hardening model** subsection. From the list, choose **Exponential**.

6 In the p_{b0} text field, type `pc0`.

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

Contact 1

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, in the **Pairs** list, choose **Contact Pair 1a (ap1)** and **Contact Pair 2a (ap2)**.
5 Click **OK**.
6 In the **Settings** window for **Contact**, locate the **Contact Surface** section.
7 Select the **Source external to current physics** check box.
8 Locate the **Contact Method** section. From the **Formulation** list, choose **Augmented Lagrangian**.
9 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 `mu`.

Prescribed Displacement 1

1 In the **Physics** toolbar, click **Boundaries** and choose **Prescribed Displacement**.
2 Select Boundary 8 only.
3 In the **Settings** window for **Prescribed Displacement**, locate the **Prescribed Displacement** section.
4 Select the **Prescribed in z direction** check box.
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 Select the **Prescribed in z direction** check box.
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	EE	Pa	Basic
Poisson's ratio	nu	Nu	1	Basic
Density	rho	Rho	kg/m³	Basic
Initial void volume fraction	f0	F0	1	Poroplastic material model
Drucker-Prager alpha coefficient	alphaDrucker	Alpha	1	Drucker-Prager
Drucker-Prager k coefficient	kDrucker	Kd	Pa	Drucker-Prager
Isotropic hardening modulus	Kiso	KIso	N/m²	Mohr-Coulomb
Maximum plastic volumetric strain	epvolmax	Epvolmax	1	Mohr-Coulomb
Ellipse aspect ratio	Rcap	Rc	1	Mohr-Coulomb

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. Clear the **Smooth across removed control entities** check box.

6 Click to expand the **Reduce Element Skewness** section. Select the **Adjust edge mesh** check box.

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 Click to expand the **Control Entities** section. Clear the **Smooth across removed control entities** check box.

6 Click to expand the **Reduce Element Skewness** section. Select the **Adjust edge mesh** check box.

Distribution 1

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

2 Select Boundaries 7 and 8 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 14 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, 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** check box.

4 Click **Add**.

5 In the table, enter the following settings:

Parameter name	Parameter value list	Parameter unit
t (Time parameter)	range(0,0.1,10)	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** check box.

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

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

8 In the **Model Builder** window, click **Fully Coupled 1**.

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

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

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

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

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.

Volumetric Plastic Strain

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

2 In the **Settings** window for **2D Plot Group**, type `Volumetric Plastic Strain` in the **Label** text field.

Surface 1

1 In the **Model Builder** window, expand the **Volumetric Plastic Strain** 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> Strain>Strain invariants>solid.epvol - Volumetric plastic strain**.

3 Locate the **Coloring and Style** section. From the **Color table** list, choose **AuroraAustralisDark**.

Volumetric Plastic Strain

1 In the **Model Builder** window, click **Volumetric Plastic Strain**.

2 In the **Volumetric Plastic Strain** toolbar, click **Plot**.

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)

In the **Model Builder** window, under **Results>Datasets** 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 2

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

von Mises Stress

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**, type `von Mises Stress` in the **Label** text field.

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

Surface 2

1 Right-click **von Mises Stress** and choose **Surface**.

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

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

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 **von Mises Stress** 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**.

Contour 1

1. In the **Model Builder** window, expand the **Current Relative Density at Middle of Compaction** node, then click **Contour 1**.
2. In the **Settings** window for **Contour**, click **Replace Expression** in the upper-right corner of the **Expression** section. From the menu, choose **Component 1 (comp1)>Solid Mechanics>Porous plasticity>solid.lemm1.popl1.rhorel - Current relative density**.
3. Locate the **Expression** section. Clear the **Description** check box.
4. Locate the **Coloring and Style** section. Select the **Reverse color table** check box.
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**.