Created in COMSOL Multiphysics 6.2

Pharmaceutical Tableting Process

Introduction

Powder compaction is a widely adopted manufacturing process in the ceramic, automotive, and pharmaceutical industries due to its high flexibility, high material utilization, and better control over quality.

The Capped Drucker–Prager (DPC) model is popular for modeling the compaction processes of pharmaceutical powders since it is relatively easy to characterize the material parameters from experimental data.

The model is inspired by the example presented in Ref. 1, where microcrystalline cellulose (MCC) powder is compacted, and the constitutive material properties are obtained from experiments. Friction between the metal powder and the compaction tools is taken into account.

The material properties are considered density dependent, and since the formulation of the DPC model presented in Ref. 1 differs from the one in COMSOL Multiphysics, a material property mapping is applied before using these parameters directly.

Model Definition

The geometry of the workpiece (pharmaceutical powder), punches, and die are shown in Figure 1. The actual compaction process needs two punches: a fixed bottom punch and a moving top punch. Because the bottom punch and die are fixed and rigid, they are not explicitly modeled. The top punch is modeled with a moving rigid material. Due to axial symmetry, the size of the model can be reduced.

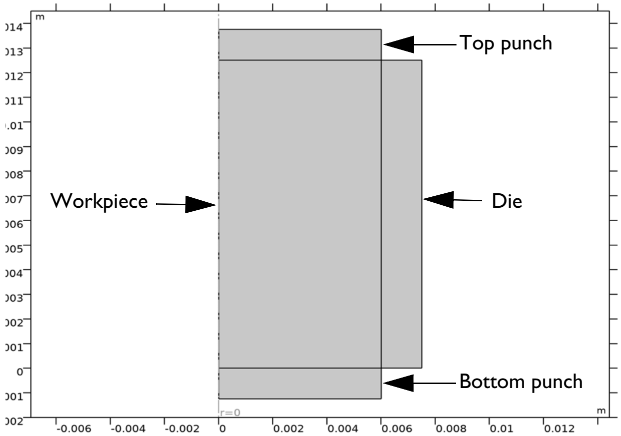

Figure 1: Geometry of the workpiece (pharmaceutical powder), punches, and die.

MATERIAL PROPERTY MAPPING

A series of experiments to calibrate elastic and plastic material properties were performed for several specimens (Ref. 1) so that the samples were compacted to different final densities. During the compaction process, the powder density changes, affecting the material properties. Therefore, all material properties are expressed in terms of the relative density of the powder.

Young's modulus and Poisson's ratio are given as functions of the relative density in Ref. 1. But since the variation of Poisson's ratio with respect to changes in relative density is small, a constant Poisson's ratio of 0.16 is used instead.

The DPC model formulation presented in Ref. 1 is different than the one in COMSOL Multiphysics. Hence, a material property mapping is needed before directly using these parameters in COMSOL Multiphysics.

Capped Drucker–Prager

In Ref. 1, the Drucker–Prager yield function F_c and the plastic potential Q_c are defined as

$$F_c = Q_c = q - p\tan\beta - d$$

where $q = \sqrt{3J_2}$ is the von Mises equivalent stress, $p = I_1/3$ is the hydrostatic pressure, β is the angle of internal friction, and d is the cohesion. Here, I_1 is the first stress invariant and J_2 is the second deviatoric stress invariant.

The elliptic cap function F_{cap} and plastic potential Q_{cap} are

$$F_{cap} = Q_{cap} = \sqrt{\left(\left(\frac{p-p_a}{p_b-p_a}\right)^2 + \left(\frac{q}{fq_a}\right)^2\right)} - 1$$

where $q_a = d + p_a \tan\beta$, and $f = l + \alpha - \alpha/\cos\beta$. The eccentricity of the ellipse, $\bar{R}$, is given by

$$\bar{R} = \frac{p_b - p_a}{q_a}$$

The Drucker–Prager cone is connected with the elliptic cap using a transition surface that serves as a smooth transition between the cone and cap surfaces. By using a transition surface, it is possible to control the variables p_a, p_b, and R independently.

The hardening law is given by

$$p_b = Ae^{B\varepsilon_{pvol}}$$

where $\varepsilon_{p\mathrm{vol}}$ is the volumetric plastic strain, and A and B are the material parameters calibrated from experimental data.

The relation between the current relative density R_D and the volumetric plastic strain is given by

$$R_D = R_{D0}e^{\varepsilon_{pvol}}$$

where R_D0 is the initial relative density.

Formulation in COMSOL Multiphysics

The formulation of the DPC model implemented in COMSOL Multiphysics is related to the material parameters used in Ref. 1.

The Drucker–Prager yield function F_c and the plastic potential Q_c are

$$F_c = Q_c = \sqrt{J_2} + \alpha I_1 - k$$

where α and k are the Drucker–Prager parameters. The relation between the Drucker–Prager parameters and the parameters given in Ref. 1 is

$$\alpha = \frac{\tan\beta}{3\sqrt{3}} \text{ and } k = \frac{d}{\sqrt{3}}$$

The elliptic cap function F_cap and plastic potential Q_cap are then

$$F_{\text{cap}} = Q_{\text{cap}} = \left(\frac{I_1 - I_a}{I_b - I_a}\right)^2 + \left(\frac{\sqrt{J_2}}{J_a}\right)^2 - 1$$

where J_a is the ordinate in the $\sqrt{J_2}$ axis at $I_1 = I_a$, and the eccentricity of the ellipse, R, is related to the eccentricity given in Ref. 1 by

$$R = \frac{I_a - I_b}{J_a} = 3\sqrt{3}\bar{R}$$

In COMSOL Multiphysics, no transition zone is required between the Drucker–Prager cone and the elliptic cap, since there is a unique and smooth transition between the two surfaces. Hence, the variables I_a and J_a are determined from the values of the parameters α, k, R, and I_b.

The hardening law is given by

$$p_b = p_{b0} - K_{\text{iso}}\log\left(1 + \frac{\varepsilon_{p\text{vol}}}{\varepsilon_{p\text{vol, max}}}\right)$$

$$I_b = -3p_b$$

where p_{b0} is the initial location of the cap, K_{iso} is the isotropic hardening modulus, and $\varepsilon_{p\text{vol,max}}$ is the maximum volumetric plastic strain.

It is clear that the hardening law given in Ref. 1 and in COMSOL are different. The parameters p_{b0}, K_{iso}, and $\varepsilon_{p\text{vol,max}}$ are chosen so that they match the results given in Ref. 1. The initial location of the cap, p_{b0}, is defined as zero since loose powder undergoes negligible initial elastic loading.

For the large strain formulation used in this example the relation between the current relative density R_D and the volumetric plastic strain is given by

$$R_D = R_{D0}J_p^{-1}$$

where J_p is the plastic volume ratio.

The size of the die is the same as given in Ref. 1, which is 12.5 mm in height and 12 mm in diameter. The true density of the powder is taken as 1590 kg/m^3 (Ref. 1), while the loose bulk density is taken as 360 kg/m^3 in order to get a similar hardening and final relative density range as given in Ref. 1. These values give an initial relative density of 0.2264.

The penalty contact method with Coulomb friction (coefficient of friction equal to 0.1) is used to model the contact interaction between the powder and the die, as well as between the powder and punches (Ref. 1). Nonlocal plasticity is used to achieve mesh objectivity.

BOUNDARY CONDITIONS

The applied boundary conditions are:

- The die and bottom punch are fixed.
- The vertical displacement of the top punch is controlled by a parameter called `para`.

Results

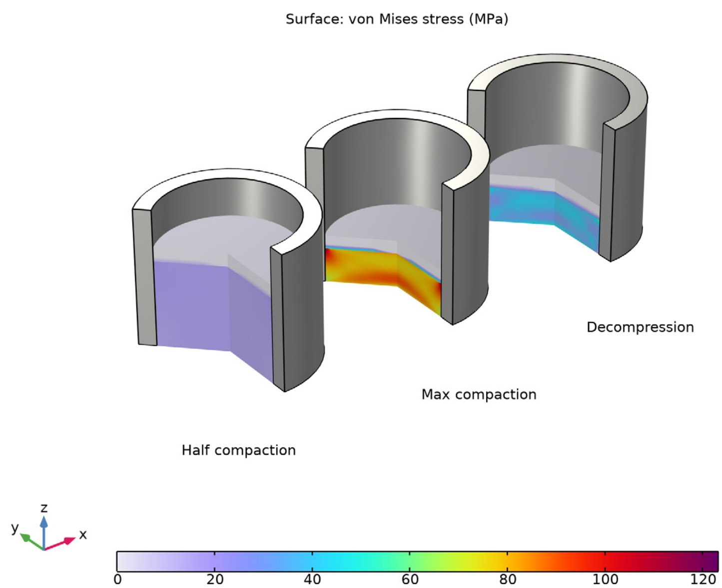

Figure 2: von Mises stress at different stages of compaction and decompression.

Figure 2 shows the von Mises stress at the middle of the compaction process, at the end of the compaction process, and after decompression. The stress is at its maximum on the top periphery, and at its minimum on the bottom periphery for all stages of compaction. The higher and lower stress rings are visible at the top and bottom surfaces of the compacted mold, which is consistent with the experimental observations; see Ref. 1. The stress relaxes once the top punch is moved upward from the mold (decompression stage).

Figure 3 shows the volumetric plastic strain at the end of compaction for the pharmaceutical powder mold. There is a large variation in volumetric plastic strain from the bottom face to the top face, with the maximum plastic strain occurring in the top region.

The relative density distribution at different stages of compaction processes is shown in Figure 4. During all stages of compaction, the high-density zone is formed at the top periphery while a low-density zone is formed at the bottom periphery. Due to friction, a nonuniform density is observed at the powder mold, which is consistent with the experimental observations reported in Ref. 1. The decompression stage has negligible impact on the relative density, unlike the stress plot, as relative density is dependent on plastic deformation which is irreversible.

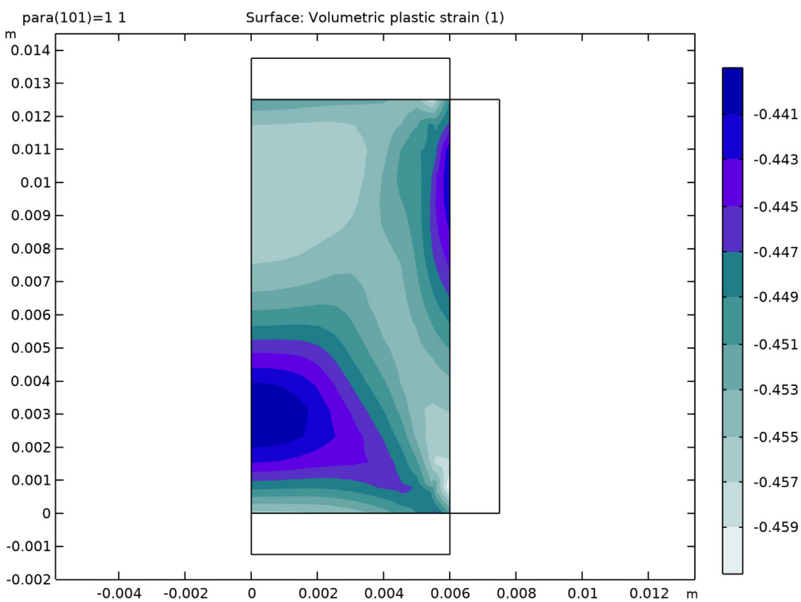

Figure 3: Volumetric plastic strain at the end of compaction.

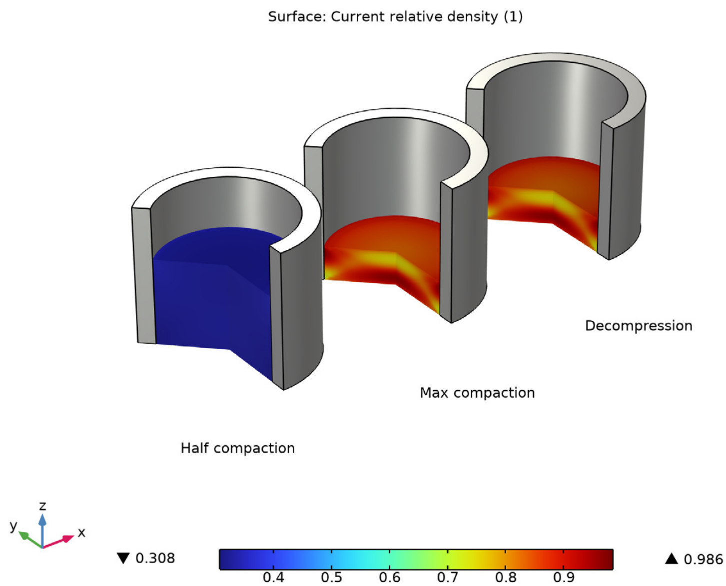

Figure 4: Relative density at different stages of compaction and decompression.

Figure 5 shows the punch pressure versus axial compaction in the compaction and decompression process. The yielding starts occurring at the beginning of the compaction process. The curve matches the numerical and experimental results presented in the Ref. 1.

The relative density and average relative density during the compaction process are shown in Figure 6. The difference between them can be better explained by the volume ratios presented in the same plot. The average relative density of the powder is related to the plastic volume ratio, while the tablet's relative density is related to the total volume ratio. The elastic deformation is small during the compaction process.

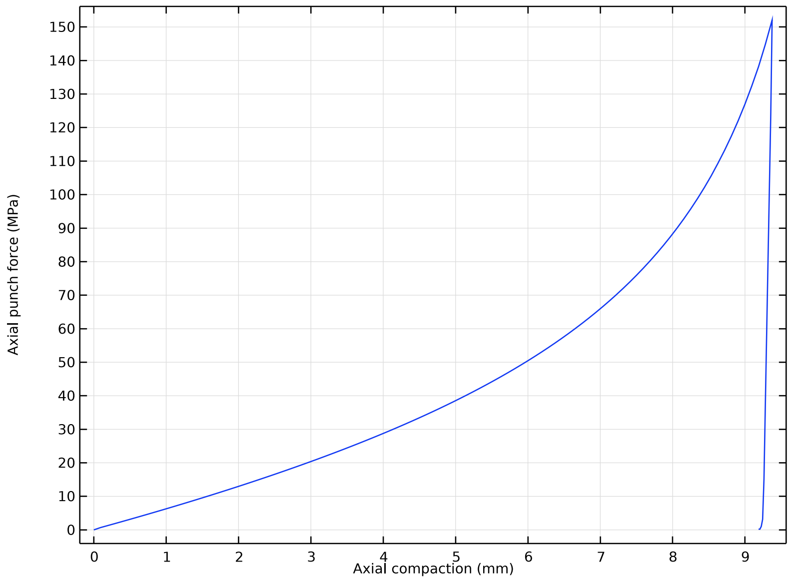

Figure 5: Axial punch pressure versus axial compaction.

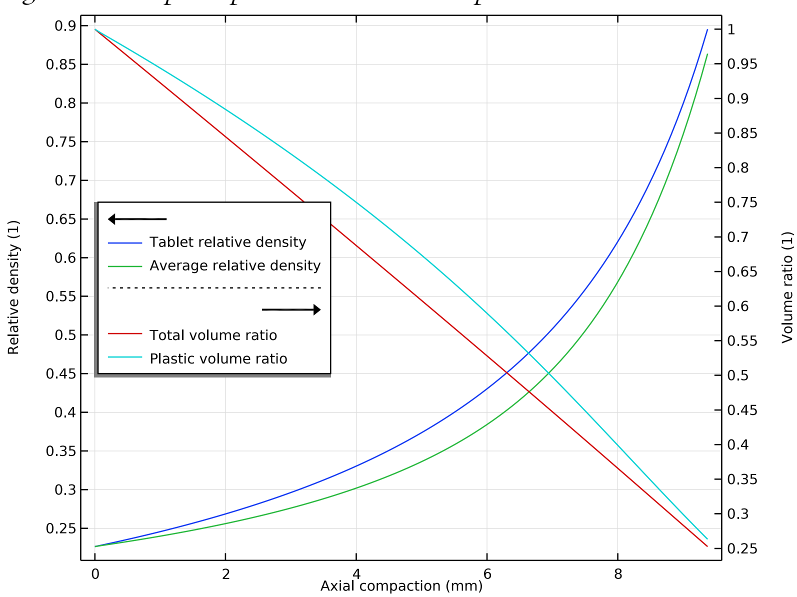

Figure 6: Relative density and volume ratio during axial compaction.

Notes About the COMSOL Implementation

In the compaction process, the interaction between the workpiece and the die as well as the workpiece and the punches is modeled using a contact node. The die and bottom punch are assumed to be rigid due to their high stiffness compared to the powder mold. As the bottom punch and die are rigid and fixed, they do not need to be modeled explicitly. In the contact node, the workpiece is taken as the destination boundary.

Reference

1. A. Baroutaji, S. Lenihan, and K. Bryan, "Combination of finite element method and Drucker-Prager Cap material model for simulation of pharmaceutical tableting process," *Material Science and Engineering Technology*, vol. 48, no. 11, 2017.

Application Library path: `Nonlinear_Structural_Materials_Module/Porous_Plasticity/pharmaceutical_tableting_process`

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**.

GEOMETRY 1

Model parameters are available in text file.

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 Click **Load from File**.

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

Young's Modulus

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

2 In the **Settings** window for **Analytic**, type `Young's Modulus` in the **Label** text field.

3 In the **Function name** text field, type `EE`.

4 Locate the **Definition** section. In the **Expression** text field, type `111.96*exp(4.395*x)`.

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

Argument	Unit
x	1

6 In the **Function** text field, type `MPa`.

7 Locate the **Plot Parameters** section. In the table, enter the following settings:

Plot	Argument	Lower limit	Upper limit	Fixed value	Unit
√	x	0.3	1	0	

Drucker Prager Parameter k

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

2 In the **Settings** window for **Analytic**, type `Drucker Prager Parameter k` in the **Label** text field.

3 In the **Function name** text field, type `Kd`.

4 Locate the **Definition** section. In the **Expression** text field, type `0.2955*exp(4.5642*x)/sqrt(3)`.

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

Argument	Unit
x	1

6 In the **Function** text field, type `MPa`.

7 Locate the **Plot Parameters** section. In the table, enter the following settings:

Plot	Argument	Lower limit	Upper limit	Fixed value	Unit
√	x	0.6	0.875	0	

Drucker Prager Parameter alpha

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

2 In the **Settings** window for **Analytic**, type `Drucker Prager Parameter alpha` in the **Label** text field.

3 In the **Function name** text field, type `Alpha`.

4 Locate the **Definition** section. In the **Expression** text field, type `tan((12.628*x+56.194)[deg])/(3*sqrt(3))`.

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

Argument	Unit
x	1

6 In the **Function** text field, type `1`.

7 Locate the **Plot Parameters** section. In the table, enter the following settings:

Plot	Argument	Lower limit	Upper limit	Fixed value	Unit
√	x	0.6	0.875	0	

Hardening Function

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

2 In the **Settings** window for **Analytic**, type `Hardening Function` in the **Label** text field.

3 In the **Function name** text field, type `Pbh`.

4 Locate the **Definition** section. In the **Expression** text field, type `-KIso*log(1+x/Epvolmax)`.

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

Argument	Unit
x	1

6 In the **Function** text field, type `Pa`.

7 Locate the **Plot Parameters** section. In the table, enter the following settings:

Plot	Argument	Lower limit	Upper limit	Fixed value	Unit
√	x	`-Epvolmax`	0	0	

GEOMETRY 1

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 `R0`.
4. In the **Height** text field, type `H0`.
5. Right-click **Rectangle 1 (r1)** and choose **Duplicate**.

Rectangle 2 (r2)

1. In the **Model Builder** window, click **Rectangle 2 (r2)**.
2. In the **Settings** window for **Rectangle**, locate the **Size and Shape** section.
3. In the **Height** text field, type `H0/10`.
4. Locate the **Position** section. In the **z** text field, type `H0`.
5. Right-click **Rectangle 2 (r2)** and choose **Duplicate**.

Rectangle 3 (r3)

1. In the **Model Builder** window, click **Rectangle 3 (r3)**.
2. In the **Settings** window for **Rectangle**, locate the **Position** section.
3. In the **z** text field, type `-H0/10`.
4. Right-click **Rectangle 3 (r3)** and choose **Duplicate**.

Rectangle 4 (r4)

1. In the **Model Builder** window, click **Rectangle 4 (r4)**.
2. In the **Settings** window for **Rectangle**, locate the **Size and Shape** section.
3. In the **Width** text field, type `R0/4`.
4. In the **Height** text field, type `H0`.
5. Locate the **Position** section. In the **r** text field, type `R0`.
6. In the **z** text field, type `0`.
7. Click **Build All Objects**.

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**.

In subsequent steps, the die and punch are rigid domains. 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, expand the **Component 1 (comp1)>Definitions** node, then click **Contact Pair 2 (ap2)**.
2 In the **Settings** window for **Pair**, click the **Swap Source and Destination** button.

Contact Pair 3 (ap3)

1 In the **Model Builder** window, click **Contact Pair 3 (ap3)**.
2 In the **Settings** window for **Pair**, click the **Swap Source and Destination** button.

Add a nonlocal integration coupling operator to compute the axial force and pressure.

Integration 1 (intop1)

1 In the **Definitions** toolbar, click **Nonlocal Couplings** and choose **Integration**.
2 In the **Settings** window for **Integration**, locate the **Source Selection** section.
3 From the **Geometric entity level** list, choose **Boundary**.
4 Select Boundary 7 only.
5 Right-click **Integration 1 (intop1)** and choose **Duplicate**.

Integration 2 (intop2)

Add a nonlocal integration coupling operator to compute the axial compaction.

1 In the **Model Builder** window, click **Integration 2 (intop2)**.
2 In the **Settings** window for **Integration**, locate the **Source Selection** section.
3 Click **Clear Selection**.
4 Select Boundary 8 only.

5 Locate the **Advanced** section. Clear the **Compute integral in revolved geometry** check box.

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
`Punchforce`	`intop1(-solid.sz)`	N	`Punch force`
`Punchpressure`	`Punchforce/A0`	N/m^2	`Punch force`
`Rho`	`PowderMass/(A0* intop2(1))`	kg/m^3	`Current powder density`

Piecewise 1 (pw1)

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

2 In the **Settings** window for **Piecewise**, type `punchDisp` in the **Function name** text field.

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

Start	End	Function
0	1	`0.75*H0*x`
1	2	`0.75*H0-0.15*H0*(x-1)`

4 Locate the **Units** section. In the **Arguments** text field, type `1`.

5 In the **Function** text field, type `m`.

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

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 Domains 2 and 3 only.

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 **Material model** list, choose **Capped Drucker–Prager**.

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

5 In the p_{b0} text field, type `Pb0`.

6 Click to expand the **Nonlocal Plasticity Model** section. From the list, choose **Implicit gradient**.

7 In the $l_{int,m}$ text field, type `1.6[mm]`.

Contact 1

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

Friction 1

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

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

3 In the μ text field, type `0.1`.

Rigid Material 1

1 In the **Physics** toolbar, click **Domains** and choose **Rigid Material**.

2 Select Domain 3 only.

Prescribed Displacement/Rotation 1

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

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

3 In the w_0 text field, type `-punchDisp(para)`.

MATERIALS

Microcrystalline Cellulose (MCC)

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 `Microcrystalline Cellulose (MCC)` 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(nojac(solid.lemm1.popl1.rhorel))`	Pa	Young's modulus and Poisson's ratio
Poisson's ratio	nu	`Nu`	1	Young's modulus and Poisson's ratio
Density	rho	`Rho`	kg/m³	Basic
Initial void volume fraction	f0	`F0`	1	Poroplastic material model
Drucker-Prager alpha coefficient	alphaDrucker	`Alpha(nojac(solid.lemm1.popl1.rhorel))`	1	Drucker-Prager
Drucker-Prager k coefficient	kDrucker	`Kd(nojac(solid.lemm1.popl1.rhorel))`	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

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

Distribution 1

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

2 Select Boundaries 2, 4, and 11–14 only.

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

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

Distribution 2

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

2 Select Boundary 6 only.

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

4 In the **Number of elements** text field, type `12`.

Distribution 3

1 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 `16`.

5 In the **Model Builder** window, right-click **Mesh 1** and choose **Build All**.

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 **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
para (Parameter)	`range(0,0.01,1.1)`	1

Use customized solver settings in order to get the faster convergence.

Solution 1 (sol1)

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

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

3 In the **Model Builder** window, expand the **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 **Initial step size** text field, type `1E-5`.

7 In the **Minimum step size** text field, type `1E-5`.

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

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 **Constant (Newton)**.

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

RESULTS

Stress (solid)

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

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

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

First create the revolution datasets needed to create the plots used in the documentation.

Study 1/Solution 1 (sol1)

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

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

Selection

1 In the **Model Builder** window, right-click **Study 1/Solution 1 (2) (sol1)** 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 4 only.

Revolution 2D

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

Revolution 2D 1

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

Average 1

In the **Results** toolbar, click **More Datasets** and choose **Evaluation>Average**.

Selection

1 Right-click **Average 1** 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.

Stress, 3D (solid)

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

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

3 From the **Parameter value (para (1))** list, choose **0.5**.

4 Click to expand the **Title** section. From the **Title type** list, choose **Custom**.

5 Find the **Solution** subsection. Clear the **Solution** check box.

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

7 Click to expand the **Plot Array** section. Select the **Enable** check box.

8 Locate the **Color Legend** section. From the **Position** list, choose **Bottom**.

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

5 Locate the **Expression** section. In the **Expression** text field, type 1.

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

7 Locate the **Coloring and Style** section. From the **Coloring** list, choose **Uniform**.

8 From the **Color** list, choose **Gray**.

9 Click to expand the **Plot Array** section. Select the **Manual indexing** check box.

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**.

Line 1

1 In the **Model Builder** window, right-click **Stress, 3D (solid)** and choose **Line**.

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

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

4 Locate the **Expression** section. In the **Expression** text field, type 1.

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

6 Locate the **Coloring and Style** section. From the **Coloring** list, choose **Uniform**.

7 From the **Color** list, choose **Black**.

8 Click to expand the **Plot Array** section. Select the **Manual indexing** check box.

Surface 1

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

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

3 From the **Unit** list, choose **MPa**.

4 Right-click **Surface 1** and choose **Duplicate**.

Surface 3

1 In the **Model Builder** window, click **Surface 3**.

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

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

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

5 Locate the **Title** section. From the **Title type** list, choose **None**.

6 Click to expand the **Inherit Style** section. From the **Plot** list, choose **Surface 1**.

Surface 2

In the **Model Builder** window, right-click **Surface 2** and choose **Duplicate**.

Surface 4

1 In the **Model Builder** window, click **Surface 4**.

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

3 In the **Index** text field, type 1.

Line 1

In the **Model Builder** window, right-click **Line 1** and choose **Duplicate**.

Line 2

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

2 In the **Settings** window for **Line**, locate the **Plot Array** section.

3 In the **Index** text field, type 1.

Surface 3

In the **Model Builder** window, right-click **Surface 3** and choose **Duplicate**.

Surface 5

1 In the **Model Builder** window, click **Surface 5**.

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

3 From the **Parameter value (para (1))** list, choose **1.1**.

Surface 4

In the **Model Builder** window, right-click **Surface 4** and choose **Duplicate**.

Surface 6

1 In the **Model Builder** window, click **Surface 6**.

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

3 In the **Index** text field, type 2.

Line 2

In the **Model Builder** window, right-click **Line 2** and choose **Duplicate**.

Line 3

1 In the **Model Builder** window, click **Line 3**.

2 In the **Settings** window for **Line**, locate the **Plot Array** section.

3 In the **Index** text field, type 2.

Stress, 3D (solid)

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

Table Annotation 1

1 In the **Stress, 3D (solid)** toolbar, click **More Plots** and choose **Table Annotation**.

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

3 From the **Source** list, choose **Local table**.

4 In the table, enter the following settings:

x-coordinate	y-coordinate	z-coordinate	Annotation
-R0	0	-6[mm]	Half compaction
3*R0	0	-10[mm]	Max compaction
6*R0	0	-10[mm]	Decompression

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

6 Select the **LaTeX markup** check box.

7 Locate the **Coloring and Style** section. Clear the **Show point** check box.

Stress, 3D (solid)

1 Click the **Show Grid** button in the **Graphics** toolbar.

2 In the **Model Builder** window, click **Stress, 3D (solid)**.

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

4 Right-click **Stress, 3D (solid)** and choose **Duplicate**.

Relative Density

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

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

3 Locate the **Color Legend** section. Select the **Show maximum and minimum values** check box.

Surface 1

1 In the **Model Builder** window, expand the **Relative Density** 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 **Coloring and Style** section. Click **Change Color Table**.

4 In the **Color Table** dialog box, select **Rainbow>Rainbow** in the tree.

5 Click **OK**.

Surface 3

1 In the **Model Builder** window, click **Surface 3**.

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**.

Surface 5

1 In the **Model Builder** window, click **Surface 5**.

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**.

Relative Density

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

2 In the **Model Builder** window, click **Relative Density**.

3 In the **Relative Density** toolbar, click **Plot**.

ADD PREDEFINED PLOT

1 In the **Home** toolbar, click **Add Predefined Plot** to open the **Add Predefined Plot** window.
2 Go to the **Add Predefined Plot** window.
3 In the tree, select **Study 1/Solution 1 (1) (sol1)>Solid Mechanics> Volumetric Plastic Strain (solid)**.
4 Click **Add Plot** in the window toolbar.
5 In the **Home** toolbar, click **Add Predefined Plot** to close the **Add Predefined Plot** window.

RESULTS

Volumetric Plastic Strain (solid)

1 In the **Settings** window for **2D Plot Group**, locate the **Data** section.
2 From the **Parameter value (para (1))** list, choose **1**.
3 Click to expand the **Number Format** section. Select the **Manual color legend settings** check box.
4 In the **Precision** text field, type `4`.
5 In the **Volumetric Plastic Strain (solid)** toolbar, click **Plot**.

Create a 1D plot of punch pressure for tableting processes.

Axial Punch Pressure Vs. Axial Compaction

1 In the **Home** toolbar, click **Add Plot Group** and choose **1D Plot Group**.
2 In the **Settings** window for **1D Plot Group**, type `Axial Punch Pressure Vs. Axial Compaction` in the **Label** text field.
3 Click to expand the **Title** section. From the **Title type** list, choose **None**.
4 Locate the **Plot Settings** section.
5 Select the **x-axis label** check box. In the associated text field, type `Axial compaction (mm)`.
6 Select the **y-axis label** check box. In the associated text field, type `Axial punch force (MPa)`.

Global 1

1 Right-click **Axial Punch Pressure Vs. Axial Compaction** 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
`Punchpressure`	`MPa`	`Punch Pressure`

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

5 In the **Expression** text field, type `punchDisp(para)`.

6 From the **Unit** list, choose **mm**.

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

8 In the **Axial Punch Pressure Vs. Axial Compaction** toolbar, click **Plot**.

Create a 1D plot of relative densities and volume ratios.

Relative Density and Volume Ratio

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

2 In the **Settings** window for **1D Plot Group**, type `Relative Density and Volume Ratio` in the **Label** text field.

3 Locate the **Data** section. From the **Parameter selection (para)** list, choose **Manual**.

4 In the **Parameter indices (1-111)** text field, type `range(1,1,101)`.

5 Locate the **Title** section. From the **Title type** list, choose **None**.

6 Locate the **Plot Settings** section.

7 Select the **x-axis label** check box. In the associated text field, type `Axial compaction (mm)`.

8 Select the **y-axis label** check box. In the associated text field, type `Relative density (1)`.

9 Select the **Two y-axes** check box.

10 Select the **Secondary y-axis label** check box. In the associated text field, type `Volume ratio (1)`.

Global 1

1 Right-click **Relative Density and Volume Ratio** 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
`((PowderMass/(A0*intop2(1)))/Rhof)`	`1`	`Tablet relative density`

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

5 In the **Expression** text field, type `punchDisp(para)`.

6 From the **Unit** list, choose **mm**.

7 Locate the **Legends** section. Find the **Include** subsection. Clear the **Solution** check box.

8 Right-click **Global 1** and choose **Duplicate**.

Global 2

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

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

3 From the **Dataset** list, choose **Average 1**.

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

5 In the **Parameter indices (1-111)** text field, type `range(1,1,101)`.

6 Locate the **y-Axis Data** section. In the table, enter the following settings:

Expression	Unit	Description
`solid.lemm1.popl1.rhorel`	`1`	`Average relative density`

7 Right-click **Global 2** and choose **Duplicate**.

Global 3

1 In the **Model Builder** window, click **Global 3**.

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

3 In the table, enter the following settings:

Expression	Unit	Description
`solid.J`	`1`	`Total volume ratio`
`solid.Jp`	`1`	`Plastic volume ratio`

Relative Density and Volume Ratio

1 In the **Model Builder** window, click **Relative Density and Volume Ratio**.

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

3 In the table, select the **Plot on secondary y-axis** check box for **Global 3**.

4 Locate the **Legend** section. From the **Position** list, choose **Middle left**.

5 In the **Relative Density and Volume Ratio** toolbar, click **Plot**.