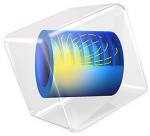

Pharmaceutical Tableting Process

Introduction

Powder compaction is a widely adopted manufacturing process not only in the ceramic and automotive industries, but also in the pharmaceutical industry due to its high flexibility, high material utilization, and better control over quality. The Capped Drucker–Prager (DPC) model is popular for modeling compaction processes of pharmaceutical powders, due to easy characterization and calibration of the material parameters with experimental data. For simulating pharmaceutical powder compaction using the DPC model, the material properties are considered as density dependent.

This model is inspired by the example presented in Ref. 1. In particular, the calibrated material properties for the DPC model presented in Ref. 1 are used. Note that because the formulation of the DPC model presented in Ref. 1 differs from the one in COMSOL Multiphysics, a material property mapping is needed before using these parameters directly.

A pharmaceutical powder known as microcrystalline cellulose (MCC) is compacted. The constitutive material properties are calibrated using experiments. Friction between the metal powder and the die is taken into account.

Model Definition

The geometry of the workpiece (pharmaceutical powder) and the die are shown in Figure 1. The actual compaction process needs two punches: a fixed bottom punch and a top moving punch. The punches are not modeled explicitly; the fixed bottom punch is modeled as a fixed axial displacement on the bottom boundary of the workpiece and the top moving punch is modeled by a prescribed displacement in the axial direction. Because the die is rigid compared to the powder, it is not modeled explicitly. Due to axial symmetry, the size of the model can be reduced.

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

MATERIAL PROPERTY MAPPING

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

Young's modulus and Poisson's ratio are given as functions of the relative density in Ref. 1 used in this model. The variation of Poisson's ratio with relative density is small. Hence, a constant Poisson's ratio of 0.16 is used in this model.

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

Formulation in the Reference

This section presents the formulation of the DPC model used in Ref. 1.

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

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

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 elliptical cap yield function F_{cap} and plastic potential Q_{cap} are

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

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

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

The Drucker–Prager cone is connected with the elliptical cap using a transition surface that does not have any physical meaning but 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 given is given by

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

where ε_{pvol} is the volumetric plastic strain, and A and B are the material constants calibrated after conducting experiments.

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 initial relative density.

Formulation in COMSOL Multiphysics

In this section, the formulation of the DPC model implemented in COMSOL Multiphysics is presented and a relation between the COMSOL material parameters with those used in Ref. 1 is established.

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 relationship between the Drucker–Prager parameters in COMSOL and Ref. 1 are

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

The elliptical cap yield function F_{cap} and plastic potential Q_{cap} are

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

where J_a is the ordinate on $\sqrt{J_2}$ axis at $I_1 = I_a$.

Hence, the eccentricity parameter R is given by

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

In COMSOL, no transition zone is required between the Drucker–Prager cone and the elliptical cap, as the cone always meets the cap at the point of tangency, so there is a unique and smooth transition between the two surfaces. Hence, the variables I_{a} and J_{a} are not free variables but determined based on α, k, R, and I_{b}.

The hardening law is,

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

$$I_b = -3p_b$$

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 laws in Ref. 1 and in COMSOL are different. The parameters of the hardening law K_{iso} and $\varepsilon_{p\text{vol,max}}$ are chosen in such way that it is similar to the hardening law curve given in Ref. 1. The initial location of the cap p_{b0} is found out using values of p_{a0}, J_{a0} and R.

For the *small plastic strains* plasticity model, the relation between current relative density R_{D} and volumetric plastic strain is given by

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

For the *large plastic strains* plasticity model, the relation between current relative density R_D and volumetric plastic strain is given by

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

where J_p is plastic volume ratio. In this example, the large plastic strains plasticity model is used.

All the calibrated elastic and plastic material properties given in Ref. 1 starts at relative density at 0.6, while in the powder compaction example presented in Ref. 1 the initial relative density is below 0.6 (exact value is not known). It is not mentioned how the material properties are extrapolated for the relative density below 0.6. To avoid this ambiguity for the compaction process presented in this model the initial relative density is assumed as 0.6, and the height of geometry is almost 33% smaller than in Ref. 1.

The penalty contact method with Coulomb friction (coefficient of friction to be 0.1) is used to model the contact interaction between workpiece and die, same as Ref. 1.

BOUNDARY CONDITIONS

The applied boundary conditions are:

- The die is fixed.
- The axial displacement of the lower face of the workpiece is constrained.
- The axial displacement of the upper face of the pharmaceutical powder is controlled by a parameter called `disp`.

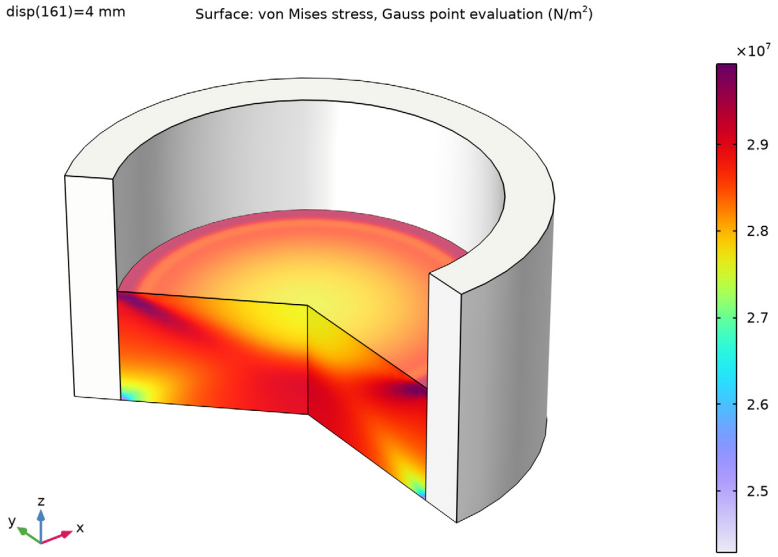

Figure 2: von Mises stress at the end of compaction.

Figure 2 shows the von Mises stress at the end of compaction for the pharmaceutical powder mold. The stress is maximum in top periphery while lower in bottom periphery. 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.

Figure 3 shows the volumetric plastic strain at the end of compaction for the pharmaceutical powder mold. There is large variation in volumetric plastic strain from the bottom face to the top face, with the maximum plastic strain occurring at the top. Due to friction, stress concentration at the corners gives the maximum volumetric plastic strain.

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 end while a low density zone is formed at the bottom end until the density is reaches the true density of the powder at the end of compaction. Due to friction, a nonuniform density is observed at the powder mold, which is consistent with the experimental observations reported in Ref. 1.

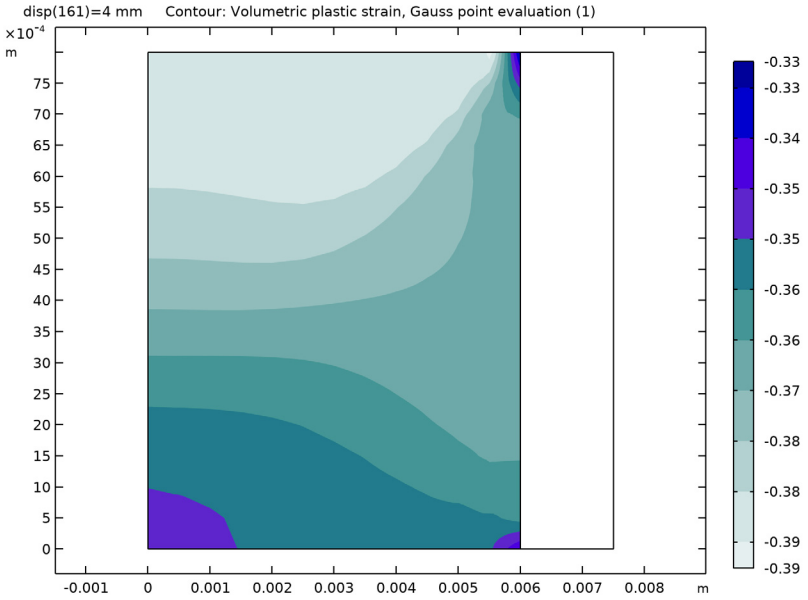

Figure 3: Volumetric plastic strain at the end of compaction.

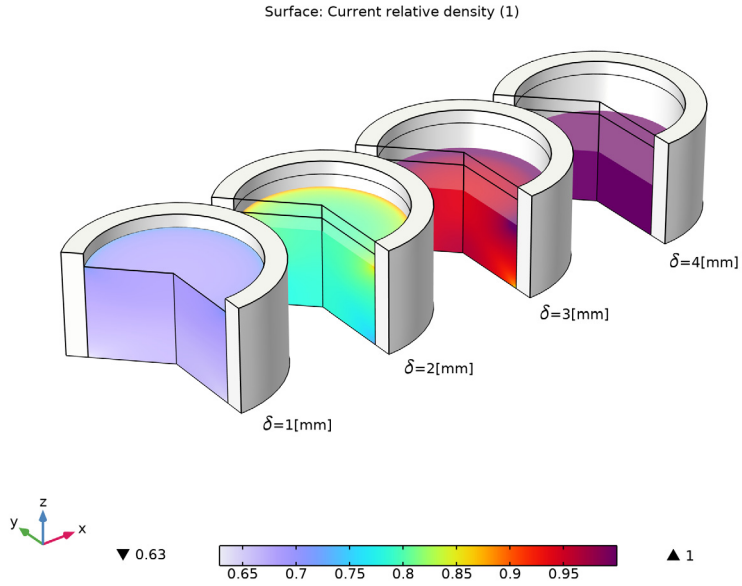

Figure 4: Relative density at various stages of compaction process.

Figure 5 shows the punch forces versus axial compaction in the compaction process. The yielding starts occurring at the very early stages of the compaction process.

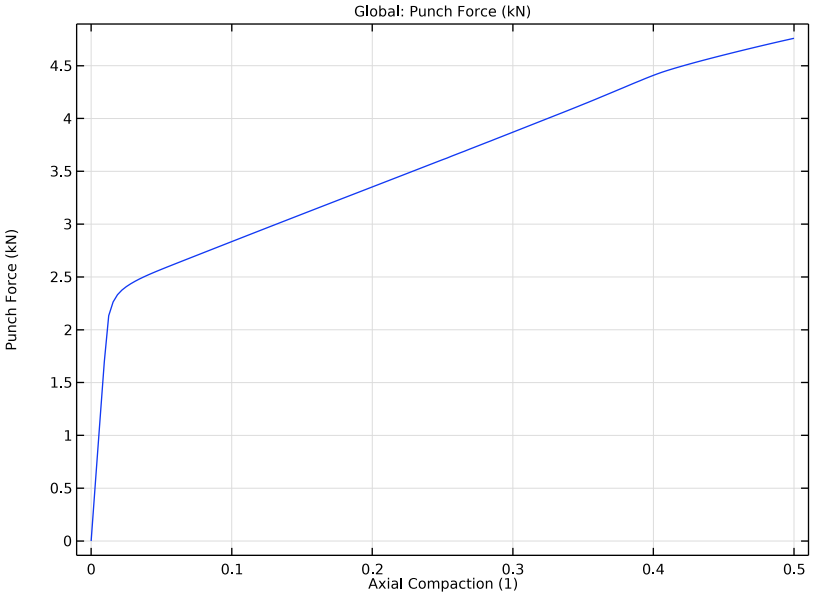

Figure 5: Punch force versus axial compaction.

Notes About the COMSOL Implementation

In the compaction process, the interaction between the workpiece and the die is modeled using a contact node. The die is assumed to be rigid due to its high stiffness compared to the powder mold. As the die is rigid and fixed, it does not need to be modeled explicitly. In the contact node, the die is considered as the destination. To model this situation, an option called **Source external to current physics** is selected.

References

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`

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`.

Youngs Modulus

1. In the **Home** toolbar, click **Functions** and choose **Global>Interpolation**.
2. In the **Settings** window for **Interpolation**, type `Youngs Modulus` in the **Label** text field.
3. Locate the **Definition** section. In the **Function name** text field, type `EE`.
4. Click **Load from File**.
5. Browse to the model's Application Libraries folder and double-click the file `pharmaceutical_tableting_process_Youngs_modulus.txt`.

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

Argument	Unit
t	1

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

Function	Unit
EE	MPa

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:

Argument	Lower limit	Upper limit	Unit
x	0.6	0.875	l

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:

Argument	Lower limit	Upper limit	Unit
x	0.6	0.875	1

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:

Argument	Lower limit	Upper limit	Unit
x	-1.2	-0.6	1

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`.

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 `R0/4`.

4 Locate the **Position** section. In the **r** text field, type `R0`.

5 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**.

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

DEFINITIONS

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 3 only.

Integration 2 (intop2)

1 Right-click **Integration 1 (intop1)** and choose **Duplicate**.

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

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

3 Click **Clear Selection**.

4 Select Boundary 4 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
Force	intop1(-solid.sz)	N	Punch force
delta	1-intop2(1)/H0		Axial compaction
Rho	Rhof* solid.lemm1.popl1.rhorel		Current powder density

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.

Contact Pair 1 (ap1)

1 In the **Model Builder** window, click **Contact Pair 1 (ap1)**.

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

Domains 2 (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 Domain 1 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 **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 Pb0.

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`.

Prescribed Displacement 1

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

2 Select Boundary 2 only.

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

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

Prescribed Displacement 2

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

2 Select Boundary 3 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 `-disp`.

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 1 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 7 and 8 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 2 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 1 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
disp (Displacement parameter)	range(0,0.025,4)	mm

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 **Study** toolbar, click = **Compute**.

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

RESULTS

Study 1/Solution 1 (2) (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 **Results** toolbar, click **Attributes** and choose **Selection**.
2. In the **Settings** window for **Selection**, locate the **Geometric Entity Selection** section.
3. From the **Geometric entity level** list, choose **Domain**.
4. Select Domain 2 only.

Revolution 2D 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)**.

Surface 2

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

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**.

Stress, 3D (solid)

1. Click the **Show Grid** button in the **Graphics** toolbar.
2. In the **Model Builder** window, under **Results** click **Stress, 3D (solid)**.

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

Relative Density

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

2 Drag and drop **Stress, 3D (solid) 1** below **Stress, 3D (solid)**.

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

4 Locate the **Data** section. From the **Parameter value (disp (mm))** list, choose **1**.

5 Click to expand the **Title** section. From the **Title type** list, choose **Manual**.

6 In the **Title** text area, type `Surface: Current relative density (1)` .

7 Clear the **Parameter indicator** text field.

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

9 From the **Position** list, choose **Bottom**.

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.lemm1.popl1.rhorel - Current relative density**.

Deformation

1 In the **Model Builder** window, expand the **Surface 1** node, then click **Deformation**.

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

3 From the **Coordinate system** list, choose **Global Cartesian**.

4 In the **X component** text field, type `0`.

Surface 3

1 In the **Model Builder** window, under **Results>Relative Density** right-click **Surface 1** and choose **Duplicate**.

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

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

4 From the **Parameter value (disp (mm))** list, choose **2**.

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

Deformation

1 In the **Model Builder** window, expand the **Surface 3** node, then click **Deformation**.

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

3 In the **X component** text field, type `2*R0`.

Surface 4

In the **Model Builder** window, under **Results>Relative Density** right-click **Surface 3** and choose **Duplicate**.

Deformation

1 In the **Model Builder** window, expand the **Surface 4** node.

2 Right-click **Results>Relative Density>Surface 4>Deformation** and choose **Delete**.

Translation 1

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

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

3 In the **x** text field, type `2*R0`.

Surface 4

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

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

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

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

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

6 Locate the **Inherit Style** section. Clear the **Color** check box.

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

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

Material Appearance 1

1 Right-click **Surface 4** 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 **Relative Density** and choose **Line**.

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

3 In the **Expression** text field, type `1`.

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

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

Translation 1

1 Right-click **Line 1** and choose **Translation**.

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

3 In the **x** text field, type `2*R0`.

4 In the **y** text field, type `0`.

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

Duplicate the above **Surface** plots and change settings in the respective **Deformation** and **Translation** nodes to show the different stages of compaction.

Relative Density

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

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

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

Volumetric Plastic Strain (solid)

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

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

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

Punch Force 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 `Punch Force Vs. Axial Compaction` in the **Label** text field.

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

4 In the associated text field, type `Axial Compaction (1)`.

5 Select the **y-axis label** check box.

6 In the associated text field, type `Punch Force (kN)`.

Global 1

1 Right-click **Punch Force 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
`Force`	`kN`	`Punch Force`

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

5 In the **Expression** text field, type `delta`.

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

7 In the **Punch Force Vs. Axial Compaction** toolbar, click  **Plot**.