Model created in COMSOL Multiphysics 6.4

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.

This model is inspired by the example presented in Ref. 1, where material properties for a microcrystalline cellulose (MCC) powder are obtained from experiments. A specimen of the powder is then compacted. Friction between the metal powder and the compaction tools is taken into account.

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.

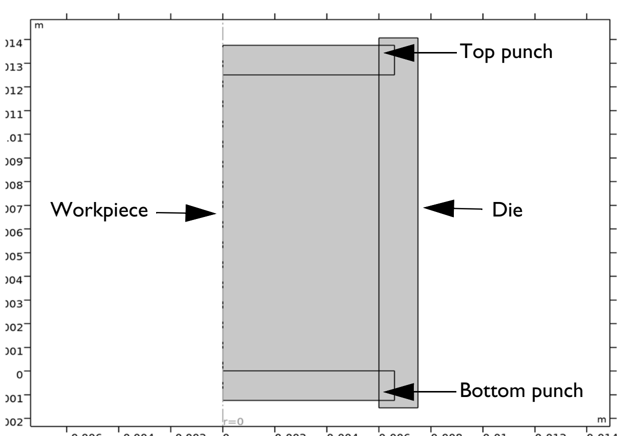

Figure 1: Geometry of the workpiece (pharmaceutical powder), punches, and die.

MATERIAL PROPERTIES

A series of experiments were performed in Ref. 1 for several specimens compacted to different final densities to calibrate elastic and plastic material properties of the MCC powder. During the compaction process, the powder density changes, affecting the material properties. Therefore, many 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.

It is clear that the hardening law given in Ref. 1 and the exponential law used in COMSOL Multiphysics are different. The parameters p_{c0}, K_c, and $\varepsilon_{pvol,max}$ are therefore chosen so to match the results given in Ref. 1. The initial location of the cap, p_{c0}, is set to zero since loose powder undergoes negligible initial elastic loading.

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.

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`.

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.

Results

Figure 2 to Figure 4 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 during decompression.

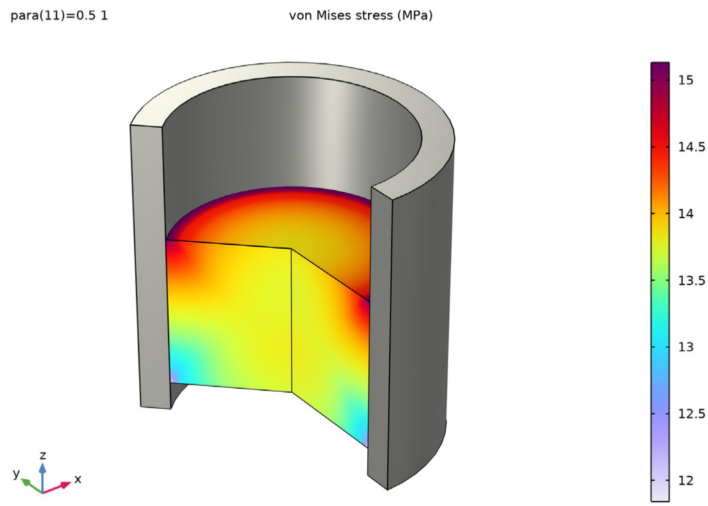

Figure 2: Distribution of von Mises stress at the middle of the compaction process.

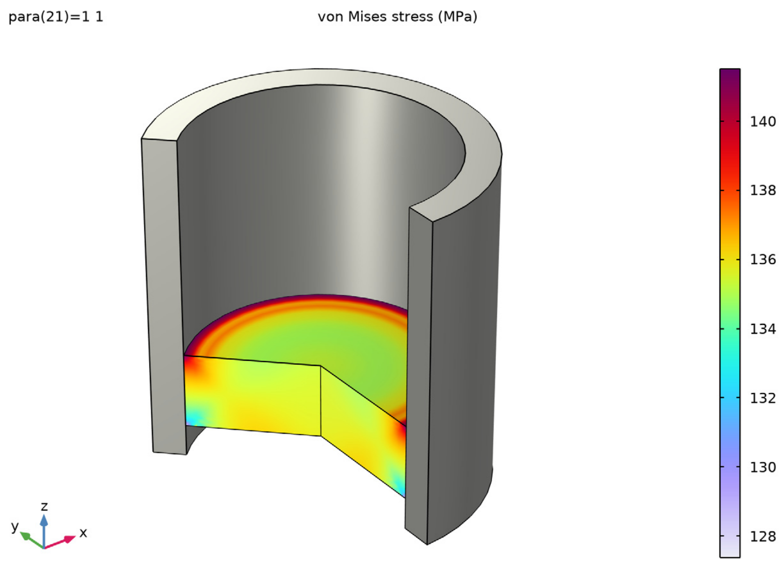

Figure 3: Distribution of von Mises stress at the end of the compaction process.

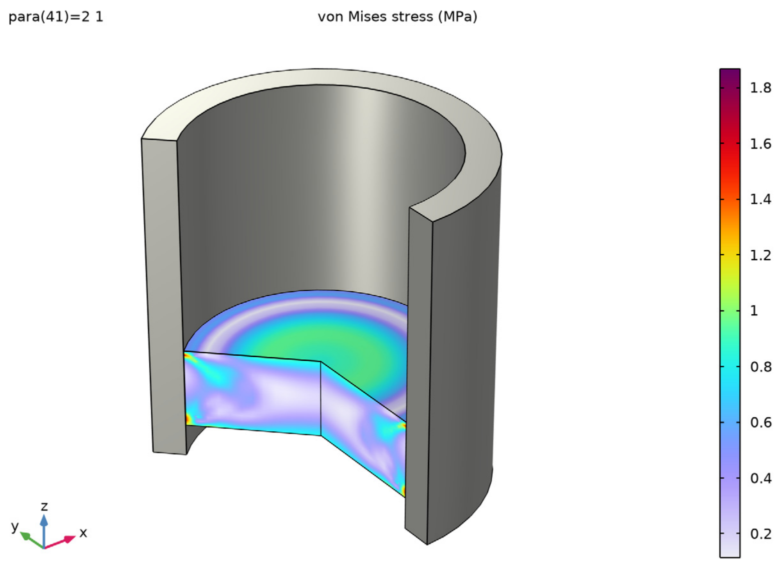

Figure 4: Distribution of von Mises stress after decompression.

Figure 5 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.

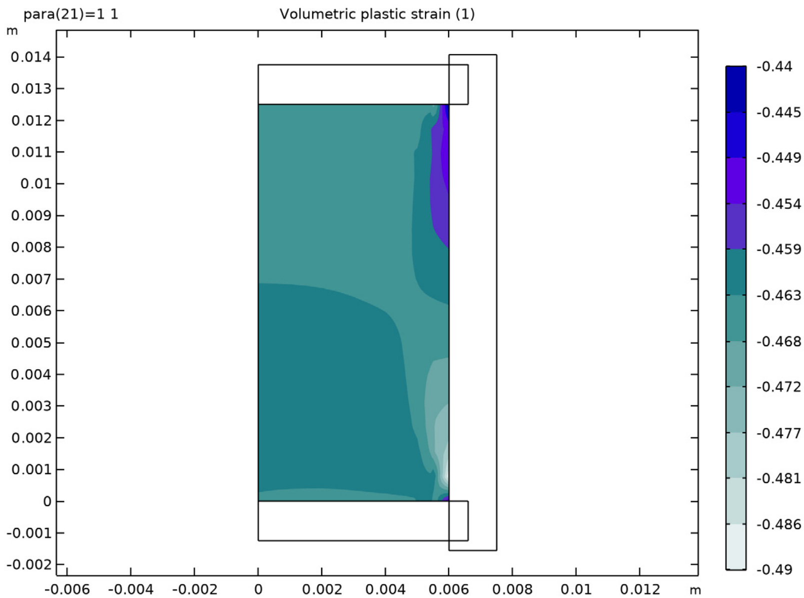

Figure 5: Volumetric plastic strain at the end of the compaction process.

The relative density distribution at different stages of compaction processes is shown in Figure 6 to Figure 8. 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.

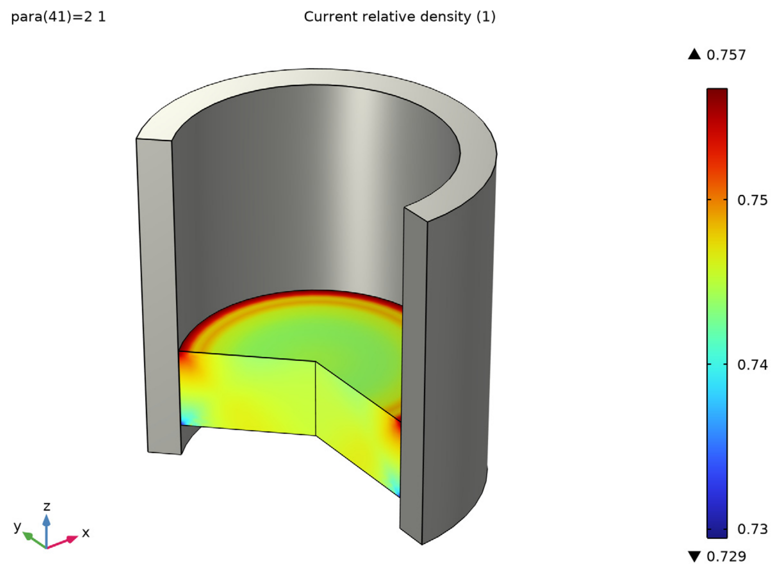

Figure 6: Relative density at the middle of the compaction process.

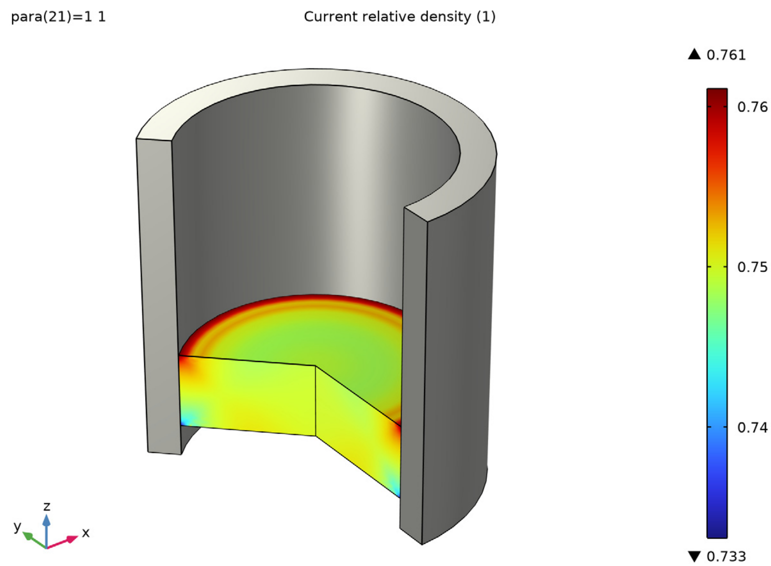

Figure 7: Relative density at the end of the compaction process.

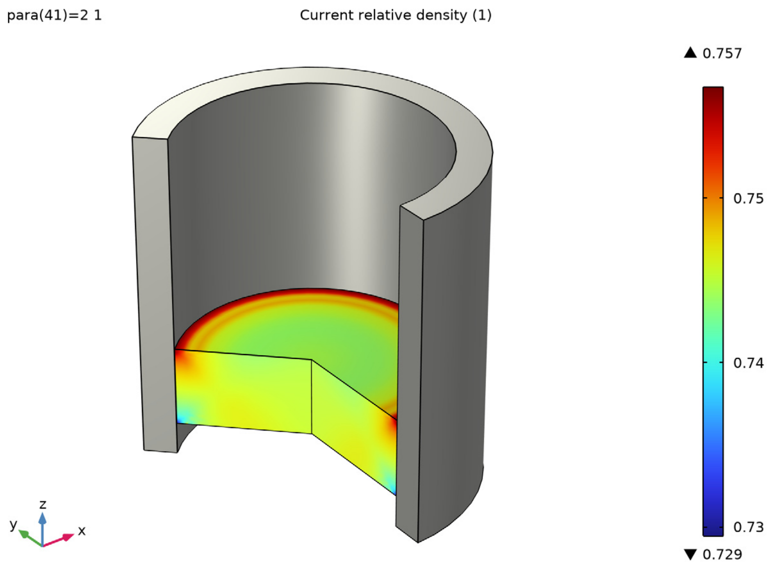

Figure 8: Relative density at different after decompression.

Figure 9 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 10. 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.

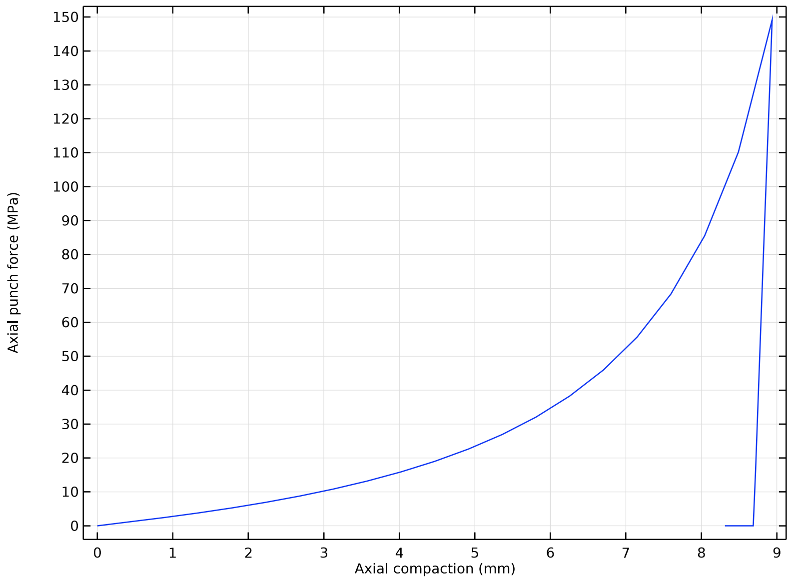

Figure 9: Axial punch pressure versus axial compaction.

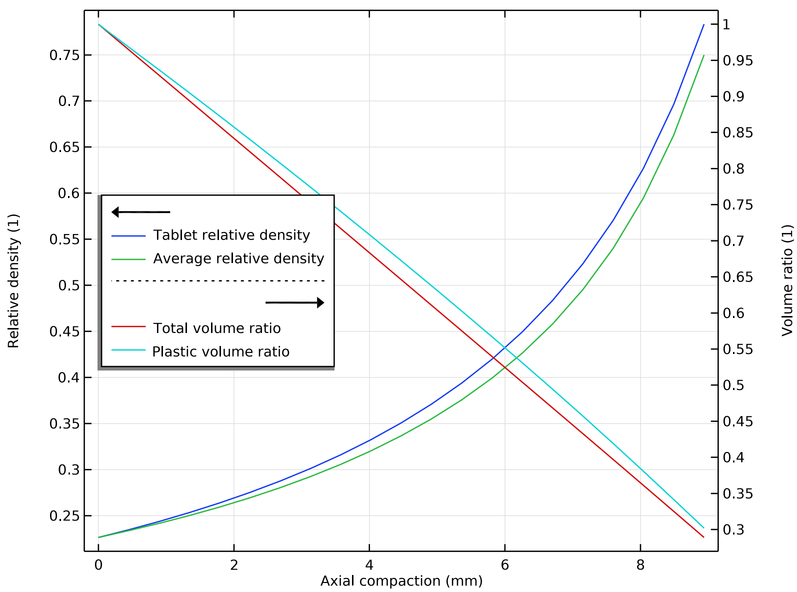

Figure 10: Relative density and volume ratio during axial compaction.

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	

Yield Stress

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

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

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

4 Locate the **Definition** section. In the **Expression** text field, type `0.2955*exp(4.5642*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	0.6	0.875	0	

Yield Function Parameter

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

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

3 In the **Function name** text field, type `a1`.

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

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	

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*1.1`.

4 In the **Height** text field, type `H0/10`.

5 Locate the **Position** section. In the **z** text field, type `H0`.

Rectangle 3 (r3)

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

2 In the **Settings** window for **Rectangle**, locate the **Position** section.

3 In the **z** text field, type `-H0/10`.

Rectangle 4 (r4)

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 `R0/4`.

4 In the **Height** text field, type `H0*1.25`.

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

6 In the **z** text field, type `-H0/8`.

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**.

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

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 7 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 8 only.

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

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/m²	Punch force
Rho	PowderMass/(A0*intop2(1))	kg/m³	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	0.715*H0*x
1	2	0.715*H0-0.05*H0*(x-1)

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

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

The die is considered as rigid and fixed, hence there is no to include them in the 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 Locate the **Cap Model** section. From the **Hardening model** list, choose **Exponential**.

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.

3 In the **Settings** window for **Rigid Material**, locate the **Density** section.

4 From the ρ list, choose **User defined**.

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)`.

Gravity 1

In the **Physics** toolbar, click **Global** and choose **Gravity**.

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(solid.lemm1.popl1.rhorel)`	Pa	Young's modulus and Poisson's ratio
Poisson's ratio	nu	`0.16`	1	Young's modulus and Poisson's ratio
Density	rho	`Rhobulk`	kg/m^3	Basic
Initial yield stress	sigmags	`sigmay(solid.lemm1.popl1.rhorel)`	Pa	Poroplastic material model
Initial void volume fraction	f0	`1-Rhorel0`	1	Poroplastic material model
Yield function parameter	a1yield	`a1(solid.lemm1.popl1.rhorel)`	1	Pressure-dependent plasticity
Initial pressure limit	pc0	`0`	Pa	Cap and cutoff
Initial ellipse centroid	pcc0	`-0.25[MPa]`	Pa	Cap and cutoff
Hardening modulus	Kc	`37.5[MPa]`	Pa	Cap and cutoff
Maximum plastic volumetric strain	epvolmax	`-log(Hf/H0)`	1	Cap and cutoff

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** checkbox.

4 Click + **Add**.

5 In the table, enter the following settings:

Parameter name	Parameter value list	Parameter unit
para (Parameter)	range(0,0.05,2)	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** checkbox.

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 **Maximum number of iterations** text field, type `8`.

13 In the **Tolerance factor** text field, type `0.1`.

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

Set default units for result presentation.

RESULTS

Preferred Units 1

1 In the **Results** toolbar, click **Configurations** and choose **Preferred Units**.

2 In the **Settings** window for **Preferred Units**, locate the **Units** section.

3 Click **Add Physical Quantity**.

4 In the **Physical Quantity** dialog, select **Solid Mechanics > Stress tensor (N/m^2)** in the tree.

5 Click **OK**.

6 In the **Settings** window for **Preferred Units**, locate the **Units** section.

7 In the table, enter the following settings:

Quantity	**Unit**	**Preferred unit**
Stress tensor	N/m^2	MPa

8 Select the **Apply conversions to expressions with the same dimensions** checkbox.

9 Click **Apply**.

First modify the revolution dataset to not include the punches.

Revolution 2D

In the **Model Builder** window, expand the **Results > Datasets** node, then click **Revolution 2D**.

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 Domains 2 and 4 only.

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 **Expression** section.

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

4 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**.

Selection 1

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

2 Select Domain 4 only.

Stress, 3D (solid)

Create a plot to show the relative density.

Relative Density

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

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** checkbox.

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. From the **Color table** list, choose **Rainbow**.

RESULT TEMPLATES

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

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

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

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

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

RESULTS

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** checkbox.

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 the tableting processes.

Axial Punch Pressure vs. Axial Compaction

1 In the **Results** toolbar, click **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** checkbox. In the associated text field, type `Axial compaction (mm)`.

6 Select the **y-axis label** checkbox. 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** checkbox.

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

Create a 1D plot of relative density and volume ratio.

Average 1

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

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.

Relative Density and Volume Ratio

1 In the **Results** toolbar, click **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 (1-41)** text field, type `range(1,1,21)`.

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

6 Locate the **Plot Settings** section.

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

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

9 Select the **Two y-axes** checkbox.

10 Select the **Secondary y-axis label** checkbox. 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** checkbox.

Global 2

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

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 (1-41)** text field, type `range(1,1,21)`.

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

Global 3

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

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** checkbox 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**.