Created in COMSOL Multiphysics 6.1

Two-Stage Powder Compaction Process

Introduction

Powder compaction is a key process in powder metallurgy that allows the manufacturing of quality products of complex shape. The density of the compact is a fundamental factor that determines the overall quality of sintered products, as regions with lower density could reduce the mechanical strength. Multiple parameters and operation conditions are optimized to obtain a uniform density, among these, reducing the friction with the walls and using multistage compaction processes.

This example shows how to setup a two-stage compaction process of metal powder for a simple geometry, and compares the outcome with the results of a single-stage process. The Gurson-Tvergaard-Needleman (GTN) model is used as constitutive model for the porous powder. Friction between the metal powder and the die is taken into account, while perfect bonding between two powder molds is assumed.

Model Definition

The geometry of the workpiece (metal powder) and die are shown in Figure 1. The workpiece geometry is divided into two different domains, for a two-stage compaction process these two domains represent two powder molds at two distinct stages. For the single-stage compaction they represent one powder mold. The punch geometry is not included. Instead, a prescribed displacement at the top boundary is used to compact the powder. Due to the axial symmetry, the geometry can be reduced.

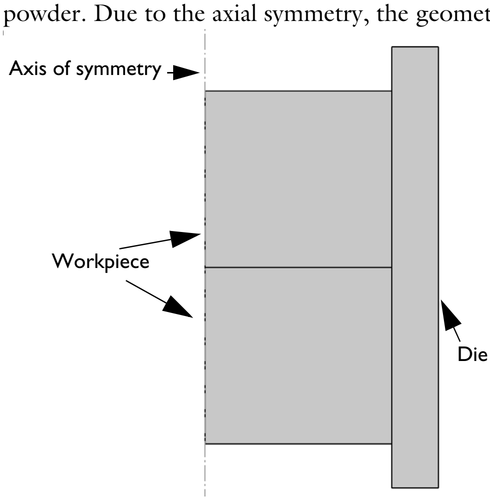

Figure 1: Geometry of the workpiece (metal powder) and die.

MATERIAL PROPERTIES

The Gurson–Tvergaard–Needleman (GTN) material model is used for the aluminum metal powder. The parameters for the GTN model are given below.

Material parameter	Value
Young's modulus	70 GPa
Poisson's ratio	0.33
Initial yield stress	200 MPa
Tvergaard correction coefficient q_1	1.5
Tvergaard correction coefficient q_2	1
Tvergaard correction coefficient q_3	2.25
Initial void volume fraction	0.28
Critical void volume fraction	0.36
Failure void volume fraction	0.4

The material of the die is not considered, since it is assumed to be rigid. Hence, the rigid domain material model is selected for the die.

BOUNDARY CONDITIONS

The applied boundary conditions are:

- The die is fixed.
- The axial displacement on the lower face of the metal powder is constrained.
- For the single-stage compaction process, the axial displacement of the upper face is controlled by a parameter called `disp`.
- For the two-stage compaction process, the first compaction stage applies an axial load on the upper boundary of the first powder mold, while in the second stage, the load is applied on the upper boundary of the second powder mold.

Results

Figure 2 shows the volumetric plastic strain at the end of the compaction for two-stage and single-stage processes. For the two-stage process there is layer-wise uniformity in the volumetric plastic strain distribution, while for the single-stage compaction there is a large variation in volumetric plastic strain between the upper and lower boundaries.

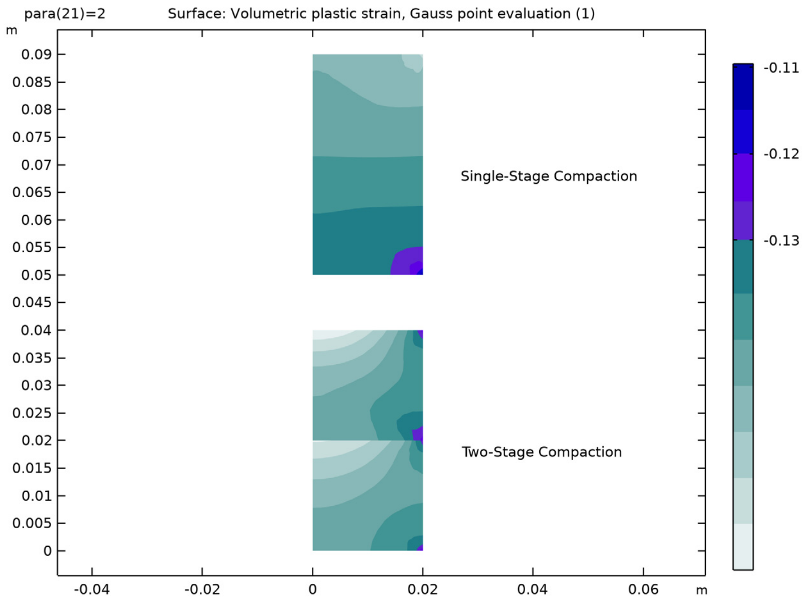

Figure 2: Volumetric plastic strain at the end of compaction.

The relative density distribution at the end of the two-stage and single-stage processes are shown in Figure 3 and Figure 4, respectively. The layer-wise uniformity is observed in the relative density for the two-stage process, while for single-stage process there is a clear variation in density from top to bottom. For the two-stage process, the variation in relative density is smaller compared to the single-stage process.

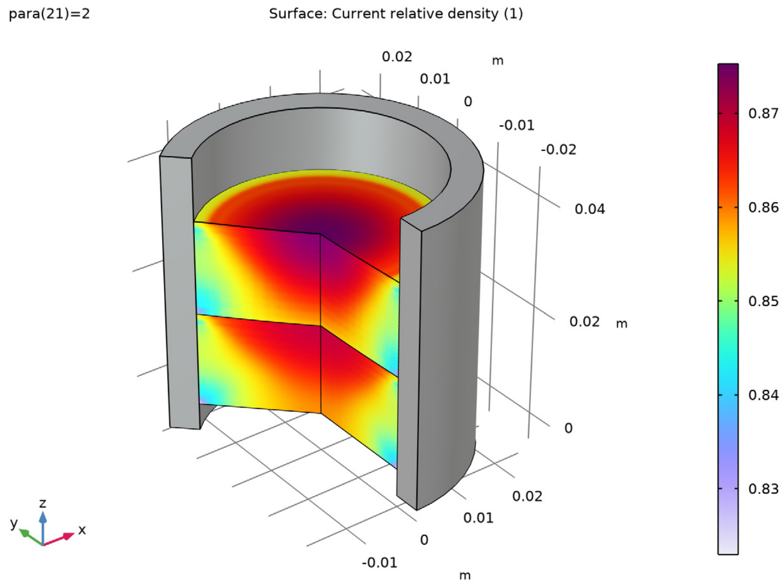

Figure 3: Relative density at the end of compaction for the two-stage process.

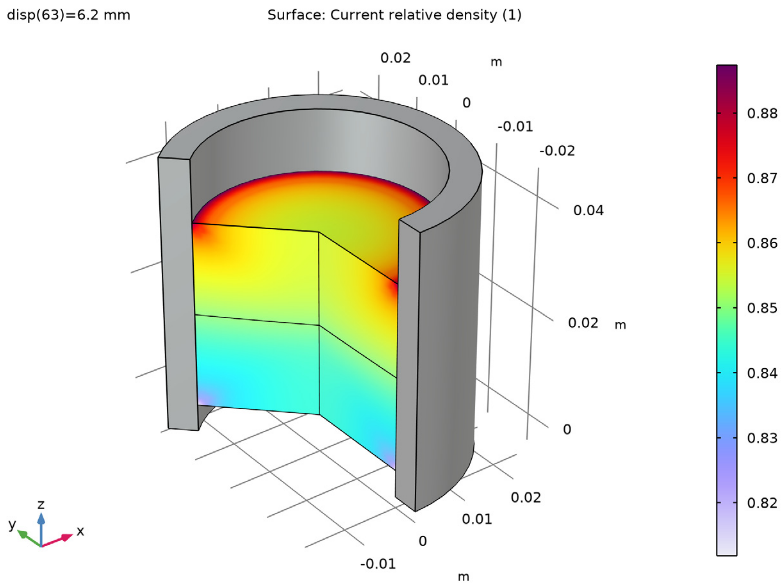

Figure 4: Relative density at the end of the compaction for the single-stage process.

The variation of average volumetric elastic strain for the two-stage process is shown in Figure 5. During the first stage of compaction (parameter `para` between 0 and 1) there is a linear variation of volumetric elastic strain in the first powder mold and no strain in the second one. This is expected as the upper portion of powder is not present in the first stage of compaction. There is elastic rebounding in the first powder mold when the parameter `para` increases from 1 to 1.1 (punch retraction after first stage of compaction). The second stage of compaction starts when the parameter `para` reaches the value of 1.1, and ends at the value of 2. In this loading step there is a linear variation in volumetric elastic strains in both powder molds.

Figure 6 shows the punch force versus axial compaction for the two-stage and single-stage processes. The yield point and the end point are almost the same for both cases, but the intermediate states are different. For the two-stage process, there are three stages of punch movement: a first forward movement of the punch on the first powder mold, a retraction of the punch, and finally the forward movement of the punch on the second powder mold. For the single-stage process there is only a forward movement of the punch on the mold.

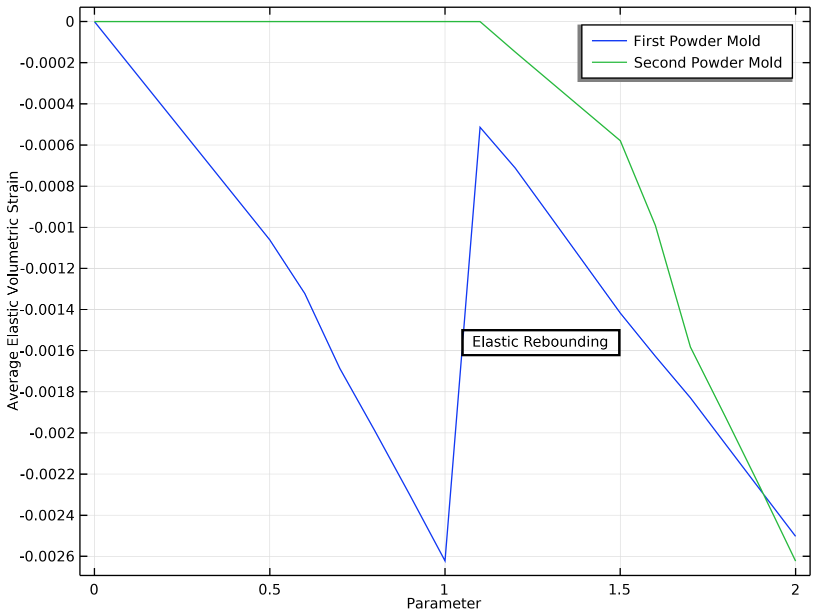

Figure 5: Variation of average volumetric elastic strain in the two-stage process.

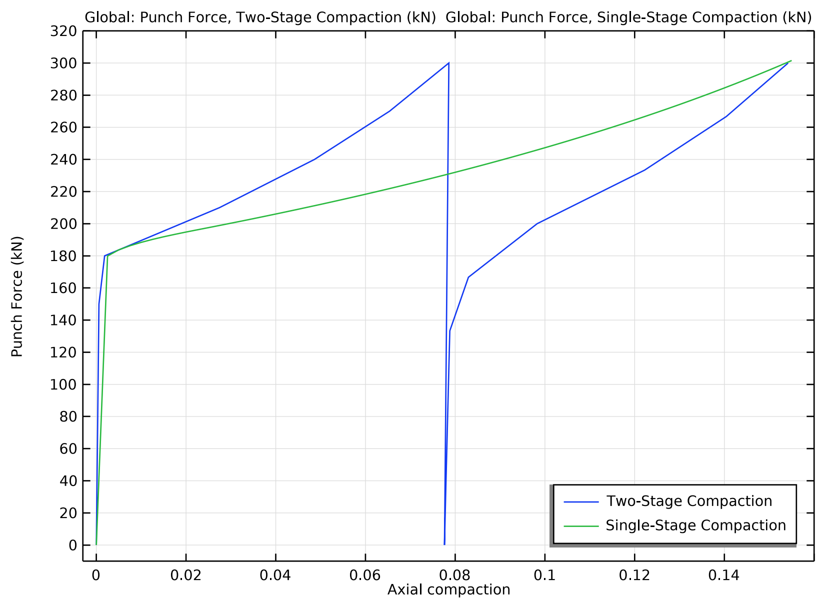

Figure 6: Punch force versus axial compaction.

Notes About the COMSOL Implementation

In the two-stage compaction process analysis the second powder mold is only present during the second stage of compaction. The activation of this mold is performed using the **Activation** subnode under **Linear Elastic Material**. Note that the mold will be activated in a stress-free state. The stresses and strains are not considered when the domain is deactivated.

Application Library path: `Nonlinear_Structural_Materials_Module/Porous_Plasticity/two_stage_compaction`

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 In the **Select Physics** tree, select **Structural Mechanics>Solid Mechanics (solid)**.
5 Click **Add**.
6 Click **Study**.
7 In the **Select Study** tree, select **General Studies>Stationary**.
8 Click **Done**.

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 `two_stage_compaction_parameters.txt`.

Punch Force

1 In the **Home** toolbar, click **Functions** and choose **Global>Interpolation**.
2 In the **Settings** window for **Interpolation**, type `Punch Force` in the **Label** text field.
3 Locate the **Definition** section. In the **Function name** text field, type `PunchForce`.
4 In the table, enter the following settings:

t	f(t)
0	0
1	300
1.1	0
2	300

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

Argument	Unit
t	1

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

Function	Unit
PunchForce	kN

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 **Position** section.

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

Union 1 (uni1)

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

2 Click in the **Graphics** window and then press Ctrl+A to select both objects.

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

Rectangle 3 (r3)

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`.

4 In the **Height** text field, type `2.5*H0`.

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

6 In the **z** text field, type `-H0/4`.

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.

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 5 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 Boundaries 6 and 7 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**
Force1	PunchForce(para)	N	Punch force, two-stage compaction

Name	Expression	Unit	Description
Force2	intop1(-solid2.sz)	N	Punch force, single-stage compaction
delta	1-intop2(1)/(2*H0)		Axial compaction

Set up the physics interfaces for two-stage and single-stage compaction.

SOLID MECHANICS (SOLID)

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 **Formulation** list, choose **Large strains**.
4 From the **Material model** list, choose **Gurson-Tvergaard-Needleman**.

Linear Elastic Material 1

In the **Model Builder** window, click **Linear Elastic Material 1**.

Activation 1

1 In the **Physics** toolbar, click **Attributes** and choose **Activation**.
2 In the **Settings** window for **Activation**, locate the **Domain Selection** section.
3 Click **Clear Selection**.
4 Select Domain 2 only.
5 Locate the **Activation** section. In the **Activation expression** text field, type para>1.

Rigid Material 1

1 In the **Physics** toolbar, click **Domains** and choose **Rigid Material**.
2 Select Domain 3 only.

Fixed Constraint 1

In the **Physics** toolbar, click **Attributes** and choose **Fixed Constraint**.

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 `0.05`.

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.

Boundary Load 1

1 In the **Physics** toolbar, click **Boundaries** and choose **Boundary Load**.

2 Select Boundary 4 only.

3 In the **Settings** window for **Boundary Load**, locate the **Force** section.

4 From the **Load type** list, choose **Total force**.

5 Specify the $\mathbf{F}_{\mathrm{tot}}$ vector as

`0`	r
`-PunchForce(para)*(para<=1)`	z

Boundary Load 2

1 Right-click **Boundary Load 1** and choose **Duplicate**.

2 In the **Settings** window for **Boundary Load**, locate the **Boundary Selection** section.

3 Click **Clear Selection**.

4 Select Boundary 5 only.

5 Locate the **Force** section. Specify the $\mathbf{F}_{\mathrm{tot}}$ vector as

`0`	r
`-PunchForce(para)*(para>=1.1)`	z

SOLID MECHANICS 2 (SOLID2)

Linear Elastic Material 1

In the **Model Builder** window, under **Component 1 (comp1)>Solid Mechanics 2 (solid2)** 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 **Formulation** list, choose **Large strains**.

4 From the **Material model** list, choose **Gurson-Tvergaard-Needleman**.

Rigid Material 1

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

2 Select Domain 3 only.

Fixed Constraint 1

In the **Physics** toolbar, click **Attributes** and choose **Fixed Constraint**.

Contact 1

In the **Model Builder** window, under **Component 1 (comp1)>Solid Mechanics 2 (solid2)** 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 `0.05`.

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 Right-click **Prescribed Displacement 1** and choose **Duplicate**.

2 Select Boundary 5 only.

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

4 In the u_{0z} text field, type `-disp`.

ADD MATERIAL

1 In the **Home** toolbar, click **Add Material** to open the **Add Material** window.

2 Go to the **Add Material** window.

3 In the tree, select **Built-in>Aluminum**.

4 Right-click and choose **Add to Component 1 (comp1)**.

5 In the **Home** toolbar, click **Add Material** to close the **Add Material** window.

MATERIALS

Aluminum (mat1)

1 In the **Settings** window for **Material**, locate the **Material Contents** section.

2 In the table, enter the following settings:

Property	Variable	Value	Unit	Property group
Initial yield stress	sigmags	`sigmay0`	Pa	Poroplastic material model
Initial void volume fraction	f0	`F0`	1	Poroplastic material model
Critical void volume fraction	fc	`Fc`	1	Poroplastic material model
Failure void volume fraction	ff	`Ff`	1	Poroplastic material model
Tvergaard correction coefficient q1	q1GTN	`q1`	1	Poroplastic material model
Tvergaard correction coefficient q2	q2GTN	`q2`	1	Poroplastic material model
Tvergaard correction coefficient q3	q3GTN	`q3`	1	Poroplastic material model

MESH 1

Mapped 1

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

Distribution 1

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

2 Select Boundaries 10 and 11 only.

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

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

5 Click **Build All**.

Size

1 In the **Model Builder** window, under **Component 1 (comp1)>Mesh 1** click **Size**.
2 In the **Settings** window for **Size**, locate the **Element Size** section.
3 From the **Predefined** list, choose **Finer**.
4 Click **Build All**.

STUDY: TWO-STAGE COMPACTION

1 In the **Model Builder** window, click **Study 1**.
2 In the **Settings** window for **Study**, type `Study: Two-Stage Compaction` in the **Label** text field.

Step 1: Stationary

1 In the **Model Builder** window, under **Study: Two-Stage Compaction** click **Step 1: Stationary**.
2 In the **Settings** window for **Stationary**, locate the **Physics and Variables Selection** section.
3 Select the **Modify model configuration for study step** check box.
4 In the tree, select **Component 1 (comp1)>Solid Mechanics 2 (solid2), Controls spatial frame**.
5 Click **Disable in Model**.
6 Click to expand the **Study Extensions** section. Select the **Auxiliary sweep** check box.
7 Click **Add**.
8 In the table, enter the following settings:

Parameter name	Parameter value list	Parameter unit
para (Parameter)	range(0,1e-1,2)	

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: Two-Stage Compaction> Solver Configurations>Solution 1 (sol1)>Stationary Solver 1** node, then click **Fully Coupled 1**.
4 In the **Settings** window for **Fully Coupled**, click to expand the **Method and Termination** section.
5 In the **Maximum number of iterations** text field, type 50.
6 In the **Study** toolbar, click **Compute**.

7 In the **Home** toolbar, click **Add Predefined Plot**.

ADD PREDEFINED PLOT

1 Go to the **Add Predefined Plot** window.

2 In the tree, select **Study: Two-Stage Compaction/Solution 1 (sol1)>Solid Mechanics>Volumetric Plastic Strain (solid)** and **Study: Two-Stage Compaction/Solution 1 (sol1)>Solid Mechanics>Current Void Volume Fraction (solid)**.

3 Click **Add Plot** in the window toolbar.

4 In the **Home** toolbar, click **Add Predefined Plot**.

ADD STUDY

1 In the **Home** toolbar, click **Add Study** to open the **Add Study** window.

2 Go to the **Add Study** window.

3 Find the **Studies** subsection. In the **Select Study** tree, select **General Studies>Stationary**.

4 Right-click and choose **Add Study**.

5 In the **Home** toolbar, click **Add Study** to close the **Add Study** window.

STUDY: SINGLE-STAGE COMPACTION

In the **Settings** window for **Study**, type `Study: Single-Stage Compaction` in the **Label** text field.

Step 1: Stationary

1 In the **Model Builder** window, under **Study: Single-Stage Compaction** click **Step 1: Stationary**.

2 In the **Settings** window for **Stationary**, locate the **Physics and Variables Selection** section.

3 Select the **Modify model configuration for study step** check box.

4 In the tree, select **Component 1 (comp1)>Solid Mechanics (solid), Controls spatial frame**.

5 Click **Disable in Model**.

6 Locate the **Study Extensions** section. Select the **Auxiliary sweep** check box.

7 Click **Add**.

8 In the table, enter the following settings:

Parameter name	Parameter value list	Parameter unit
disp (Displacement parameter)	`range(0,0.1,6.2)`	`mm`

Solution 2 (sol2)

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

2 In the **Model Builder** window, expand the **Solution 2 (sol2)** node.

3 In the **Model Builder** window, expand the **Study: Single-Stage Compaction>Solver Configurations>Solution 2 (sol2)>Stationary Solver 1** node, then click **Fully Coupled 1**.

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

5 In the **Maximum number of iterations** text field, type 50.

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

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

RESULTS

Revolution 2D 2

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

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

3 From the **Dataset** list, choose **Study: Single-Stage Compaction/Solution 2 (sol2)**.

Study: Two-Stage Compaction/Solution 1 (3) (sol1)

In the **Model Builder** window, under **Results>Datasets** right-click **Study: Two-Stage Compaction/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 3 only.

Revolution 2D 1

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

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

3 From the **Dataset** list, choose **Study: Two-Stage Compaction/Solution 1 (3) (sol1)**.

Study: Single-Stage Compaction/Solution 2 (4) (sol2)

In the **Model Builder** window, under **Results>Datasets** right-click **Study: Single-Stage Compaction/Solution 2 (sol2)** 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 3 only.

Revolution 2D 2a

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: Single-Stage Compaction/Solution 2 (4) (sol2)**.

Revolution 2D 1, Revolution 2D 2a, Study: Single-Stage Compaction/Solution 2 (4) (sol2), Study: Two-Stage Compaction/Solution 1 (3) (sol1)

1 In the **Model Builder** window, under **Results>Datasets**, Ctrl-click to select **Study: Two-Stage Compaction/Solution 1 (3) (sol1)**, **Revolution 2D 1**, **Study: Single-Stage Compaction/Solution 2 (4) (sol2)**, and **Revolution 2D 2a**.

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

Datasets for Die

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

Stress, Two-Stage Compaction

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

2 In the **Settings** window for **2D Plot Group**, type `Stress, Two-Stage Compaction` in the **Label** text field.

3 In the **Stress, Two-Stage Compaction** toolbar, click **Plot**.

Stress, Single-Stage Compaction

1 In the **Model Builder** window, click **Stress (solid2)**.

2 Drag and drop below **Stress, Two-Stage Compaction**.

3 In the **Settings** window for **2D Plot Group**, type `Stress, Single-Stage Compaction` in the **Label** text field.

4 In the **Stress, Single-Stage Compaction** toolbar, click **Plot**.

Relative Density, Two-Stage Compaction

1. In the **Model Builder** window, under **Results** click **Stress, 3D (solid)**.
2. In the **Settings** window for **3D Plot Group**, type `Relative Density, Two-Stage Compaction` in the **Label** text field.

Surface 1

1. In the **Model Builder** window, expand the **Relative Density, Two-Stage Compaction** node, then click **Surface 1**.
2. In the **Settings** window for **Surface**, click **Replace Expression** in the upper-right corner of the **Expression** section. From the menu, choose **Component 1 (comp1)>Solid Mechanics>Porous plasticity>solid.lemm1.popl1.rhorel - Current relative density**.

Surface 2

1. Right-click **Results>Relative Density, Two-Stage Compaction>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 **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

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

Relative Density, Two-Stage Compaction

In the **Relative Density, Two-Stage Compaction** toolbar, click **Plot**.

Relative Density, Single-Stage Compaction

1. In the **Model Builder** window, click **Stress, 3D (solid2)**.
2. Drag and drop below **Relative Density, Two-Stage Compaction**.
3. In the **Settings** window for **3D Plot Group**, type `Relative Density, Single-Stage Compaction` in the **Label** text field.

Surface 1

1. In the **Model Builder** window, expand the **Relative Density, Single-Stage Compaction** node, then click **Surface 1**.

2 In the **Settings** window for **Surface**, click **Replace Expression** in the upper-right corner of the **Expression** section. From the menu, choose **Component 1 (comp1)> Solid Mechanics 2>Porous plasticity>solid2.lemm1.popl1.rhorel - Current relative density**.

Surface 2

1 Right-click **Results>Relative Density, Single-Stage Compaction>Surface 1** and choose **Duplicate**.

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

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

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

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

Relative Density, Single-Stage Compaction

In the **Relative Density, Single-Stage Compaction** toolbar, click **Plot**.

Volumetric Plastic Strain

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

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

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

Surface 2

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

2 Right-click **Results>Volumetric Plastic Strain>Surface 1** and choose **Duplicate**.

Filter 1

1 In the **Model Builder** window, expand the **Surface 2** node.

2 Right-click **Results>Volumetric Plastic Strain>Surface 2>Filter 1** and choose **Delete**.

Surface 2

1 In the **Model Builder** window, under **Results>Volumetric Plastic Strain** click **Surface 2**.

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

3 From the **Dataset** list, choose **Study: Single-Stage Compaction/Solution 2 (2) (sol2)**.

4 Locate the **Expression** section. In the **Expression** text field, type `if(isnan(solid2.epvol),NaN,solid2.epvol)`.

5 Click to expand 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**.

Translation 1

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

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

3 In the **y** text field, type `50[mm]`.

Volumetric Plastic Strain

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

Table Annotation 1

1 In the **Volumetric Plastic Strain** 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	Annotation
`0.025`	`0.02`	`Two-Stage Compaction`
`0.025`	`0.07`	`Single-Stage Compaction`

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

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

Void Volume Fraction

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

2 In the **Settings** window for **2D Plot Group**, type `Void Volume Fraction` in the **Label** text field.

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

Surface 2

1 In the **Model Builder** window, expand the **Void Volume Fraction** node.

2 Right-click **Results>Void Volume Fraction>Surface 1** and choose **Duplicate**.

Deformation

1 In the **Model Builder** window, expand the **Results>Void Volume Fraction>Surface 1** node.

2 Right-click **Results>Void Volume Fraction>Surface 1>Deformation** and choose **Delete**.

Surface 2

In the **Model Builder** window, expand the **Surface 2** node.

Deformation, Filter 1

1 In the **Model Builder** window, under **Results>Void Volume Fraction>Surface 2**, Ctrl-click to select **Filter 1** and **Deformation**.

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

Surface 2

1 In the **Model Builder** window, under **Results>Void Volume Fraction** click **Surface 2**.

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

3 From the **Dataset** list, choose **Study: Single-Stage Compaction/Solution 2 (2) (sol2)**.

4 Locate the **Expression** section. In the **Expression** text field, type `if(isnan(solid2.f),NaN,solid2.f)`.

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

6 Locate the **Inherit Style** section. From the **Plot** list, choose **Surface 1**.

Translation 1

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

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

3 In the **y** text field, type `50[mm]`.

Void Volume Fraction

In the **Model Builder** window, under **Results** click **Void Volume Fraction**.

Table Annotation 1

1 In the **Void Volume Fraction** 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	Annotation
0.025	0.02	Two-Stage Compaction
0.025	0.07	Single-Stage Compaction

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

6 In the **Void Volume Fraction** toolbar, click **Plot**.

Create 1D plot of average elastic volumetric strain in the workpiece to understand the elastic rebounding phenomenon in two-stage compaction.

Average Elastic Volumetric Strain, Two-Stage Compaction

1 In the **Results** toolbar, click **Evaluation Group**.
2 In the **Settings** window for **Evaluation Group**, type `Average Elastic Volumetric Strain, Two-Stage Compaction` in the **Label** text field.

Surface Average 1

1 Right-click **Average Elastic Volumetric Strain, Two-Stage Compaction** and choose **Average>Surface Average**.
2 Select Domain 1 only.
3 In the **Settings** window for **Surface Average**, locate the **Expressions** section.
4 In the table, enter the following settings:

Expression	Unit	Description
`para`	1	`Parameter`
`solid.eelvol*solid.isactive`	1	`Elastic Volumetric strain`

Surface Average 2

1 In the **Model Builder** window, right-click **Average Elastic Volumetric Strain, Two-Stage Compaction** and choose **Average>Surface Average**.
2 Select Domain 2 only.
3 In the **Settings** window for **Surface Average**, locate the **Expressions** section.
4 In the table, enter the following settings:

Expression	Unit	Description
`solid.eelvol*solid.isactive`	1	`Elastic Volumetric strain`

Average Elastic Volumetric Strain, Two-Stage Compaction

1 In the **Model Builder** window, click **Average Elastic Volumetric Strain, Two-Stage Compaction**.
2 In the **Settings** window for **Evaluation Group**, click to expand the **Format** section.
3 From the **Include parameters** list, choose **Off**.
4 In the **Average Elastic Volumetric Strain, Two-Stage Compaction** toolbar, click **Evaluate**.

Average Elastic Volumetric Strain, Two-Stage 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 `Average Elastic Volumetric Strain, Two-Stage Compaction` in the **Label** text field.

3 Locate the **Plot Settings** section.

4 Select the **y-axis label** check box. In the associated text field, type `Average Elastic Volumetric Strain`.

5 Drag and drop below **Void Volume Fraction**.

Table Graph 1

1 Right-click **Average Elastic Volumetric Strain, Two-Stage Compaction** and choose **Table Graph**.

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

3 From the **Source** list, choose **Evaluation group**.

4 From the **x-axis data** list, choose **Parameter**.

5 Click to expand the **Legends** section. Select the **Show legends** check box.

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

7 In the table, enter the following settings:

Legends
`First Powder Mold`
`Second Powder Mold`

Annotation 1

1 In the **Model Builder** window, right-click **Average Elastic Volumetric Strain, Two-Stage Compaction** and choose **Annotation**.

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

3 In the **Text** text field, type `Elastic Rebounding`.

4 Locate the **Position** section. In the **R** text field, type `1.05`.

5 In the **Z** text field, type `-0.0015`.

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

7 Select the **Show frame** check box.

8 In the **Average Elastic Volumetric Strain, Two-Stage Compaction** toolbar, click **Plot**.

Create a 1D plot of punch forces for both processes.

Punch Force Vs. Axial Compaction

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

Error occurred when generating instruction for method 'X.doDragAndDrop(([Ljava.lang.String;), /result/feature/pg12, below, false, false). Cause: Cannot obtain a feature data provider for a null object.

2 In the **Model Builder** window, click **1D Plot Group 8**.

3 In the **Settings** window for **1D Plot Group**, type `Punch Force Vs. Axial Compaction` in the **Label** text field.

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

5 Select the **y-axis label** check box. In the associated text field, type `Punch Force (kN)`.

6 In the **x-axis label** text field, type `Axial compaction`.

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

8 In the **x minimum** text field, type `-0.003`.

9 In the **x maximum** text field, type `0.16`.

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

11 In the **y maximum** text field, type `320`.

12 Locate the **Legend** section. From the **Position** list, choose **Lower right**.

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
Force1	kN	Punch Force, Two-Stage Compaction

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

7 In the table, enter the following settings:

Legends
Two-Stage Compaction

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 **Study: Single-Stage Compaction/Solution 2 (2) (sol2)**.

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

Expression	Unit	Description
Force2	kN	Punch Force, Single-Stage Compaction

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

Legends
Single-Stage Compaction

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