Created in COMSOL Multiphysics 5.6

Powder Compaction of a Cup

Introduction

In this example, the fabrication of a cup through powder compaction is simulated. The powder compaction process is becoming common in the manufacturing industry, thanks to its potential to produce components of complex shape and high strength.

Combining the Fleck-Kuhn-McMeeking (FKM) model with the Gurson-Tvergaard-Needleman (GTN) model for porous plasticity makes it possible to cover a wide range of porosity values. Friction between the metal powder and the die is taken in to account. From a simulation point of view, this is a highly nonlinear structural analysis because of the contact interaction between the moving parts, the elastoplastic constitutive law selected for the metal powder, and the geometrical nonlinearity caused by the large displacements.

Model Definition

The geometry of workpiece (metal powder) and die are shown in Figure 1. The punch to compact the workpiece is not modeled. Instead, a prescribed displacement in the normal direction is used to compact the powder. Due to the axial symmetry, the size of model can be reduced.

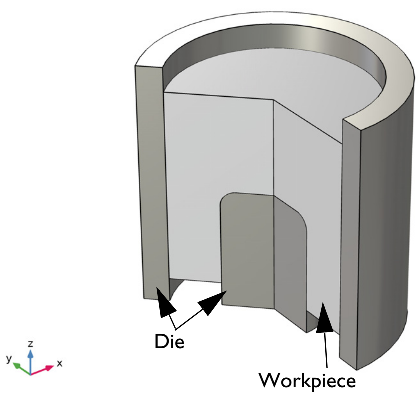


Figure 1: Geometry of the workpiece (metal powder) and die.

MATERIAL PROPERTIES

For the aluminum metal powder, an elastoplastic material model with a constitutive relation given by a combination of the Fleck–Kuhn–McMeeking (FKM) model and the

Gurson–Tvergaard–Needleman (GTN) is used. The parameters for the FKM-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
Maximum void volume fraction	0.36

The material of the die is irrelevant, 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 inner and outer dies are fixed.
- A prescribed displacement boundary condition for the upper and lower face of the metal powder. The displacement in the z direction is controlled by a parameter called `para`.

Results

Figure 2 shows the volumetric plastic strain at the end of compaction process. At the middle of the fillet, the volumetric plastic strain is at its minimum. At ends of the fillet, the volumetric plastic strain is high. The volumetric plastic strain at the corner points of the workpiece are about 12%, probably due to the friction.

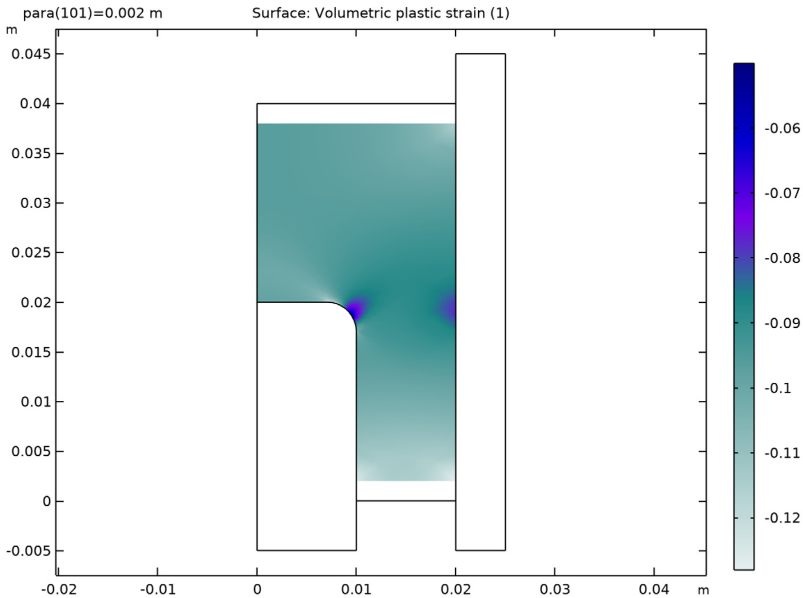


Figure 2: The volumetric plastic strain at the end of compaction.

The compaction process reduces the porosity of the aluminum powder and increases its density. This process also results in an increase in the strength of the component. Considering the type of geometry and loading, non-uniform changes in porosity are expected. Contours of the current void volume fraction or porosity are shown in Figure 3. The metal powder in the thin lower portion of the workpiece is more compacted than the material in the middle or top portion. At the central region near to the fillet, the metal powder is less compacted due to material sliding on the rounded corner.

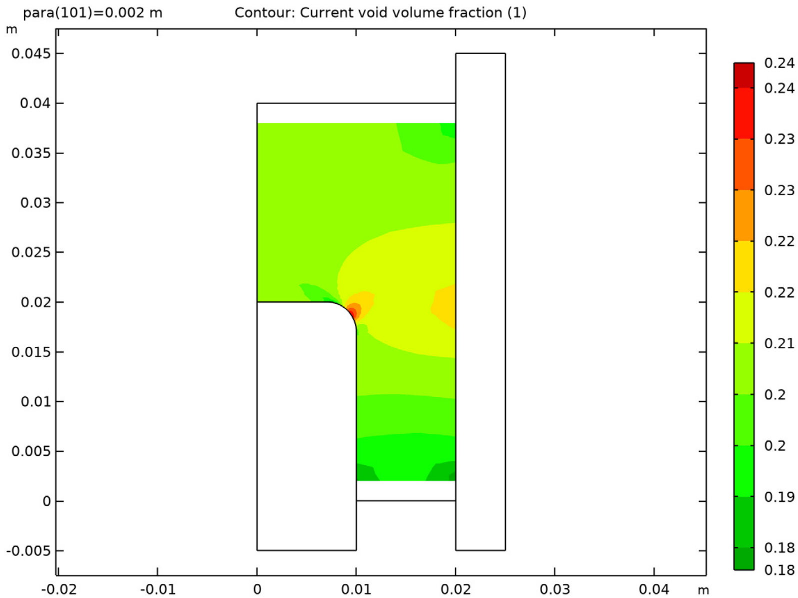


Figure 3: Current void volume fraction at the end of compaction.

The von Mises stress along with effective plastic strain in the workpiece at the end of compaction is shown in Figure 4

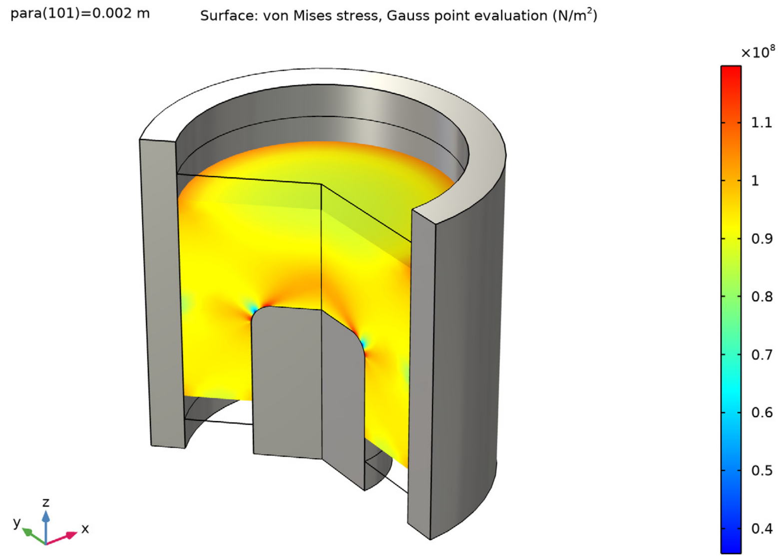


Figure 4: The von-Mises stress in the workpiece at the end of compaction.

Notes About the COMSOL Implementation

To improve the convergence and speed up the computations a customized mesh is used. The curved boundaries in a contact are well resolved using finer mesh, while the straight edges of the rigid domains are meshed with single elements, see Figure 5. The mesh on rigid parts only serve to describe the geometry accurately.

The parametric steps in the solver settings are tuned according to the used mesh. Changes in the mesh size may cause slower convergence and could thus require a modified step size to obtain the solution efficiently.

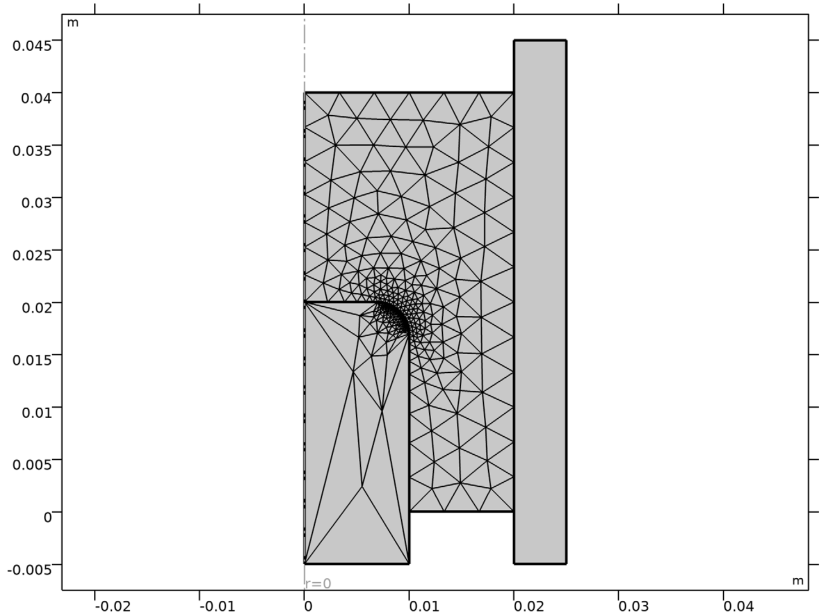


Figure 5: Customized mesh.

Application Library path: `Nonlinear_Structural_Materials_Module/Porous_Plasticity/powder_compaction_of_a_cup`

Modeling Instructions

From the **File** menu, choose **New**.

NEW

In the **New** window, click **Model Wizard**.

MODEL WIZARD

1 In the **Model Wizard** window, click **2D Axisymmetric**.

2 In the **Select Physics** tree, select **Structural Mechanics>Solid Mechanics (solid)**.

3 Click **Add**.

4 Click **Study**.

5 In the **Select Study** tree, select **General Studies>Stationary**.

6 Click **Done**.

GLOBAL DEFINITIONS

Parameters 1

1 In the **Model Builder** window, under **Global Definitions** click **Parameters 1**.

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

3 In the table, enter the following settings:

Name	Expression	Value	Description
para	0[mm]	0 m	Displacement parameter
sigmay0	200e6[Pa]	2E8 Pa	Initial yield stress
q1	1.5	1.5	Tvergaard correction coefficient
q2	1	1	Tvergaard correction coefficient
q3	2.25	2.25	Tvergaard correction coefficient
F0	0.28	0.28	Initial void volume fraction
Fmax	0.36	0.36	Maximum void volume fraction

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 `20[mm]`.

4 In the **Height** text field, type `40[mm]`.

Rectangle 2 (r2)

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 `10[mm]`.

4 In the **Height** text field, type `20[mm]`.

Difference 1 (dif1)

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

2 Select the object **r1** only.

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

4 Find the **Objects to subtract** subsection. Select the **Activate Selection** toggle button.

5 Select the object **r2** only.

6 Click **Build Selected**.

Fillet 1 (fil1)

1 In the **Geometry** toolbar, click **Fillet**.

2 On the object **dif1**, select Point 4 only.

It might be easier to select the correct point by using the **Selection List** window. To open this window, in the **Home** toolbar click **Windows** and choose **Selection List**. (If you are running the cross-platform desktop, you find **Windows** in the main menu.)

3 In the **Settings** window for **Fillet**, locate the **Radius** section.

4 In the **Radius** text field, type `3[mm]`.

5 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 `10[mm]`.

4 In the **Height** text field, type `25[mm]`.

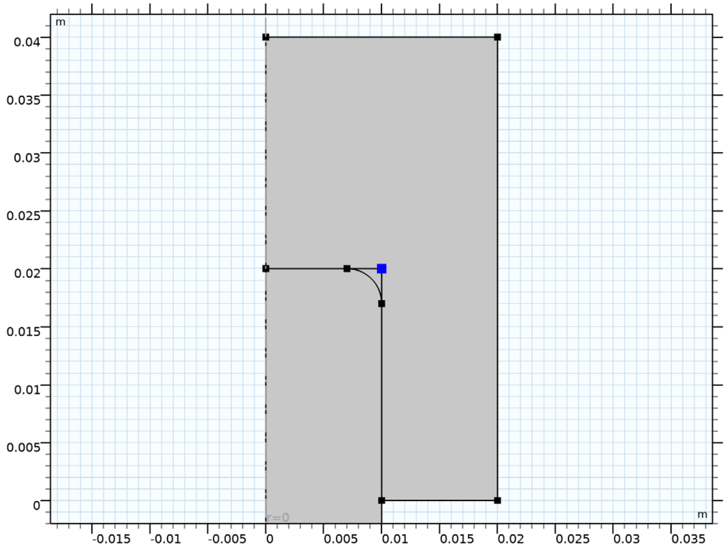
5 Locate the **Position** section. In the **z** text field, type `-5[mm]`.

6 Click **Build Selected**.

Fillet 2 (fil2)

1 In the **Geometry** toolbar, click **Fillet**.

2 On the object **r3**, select Point 3 only.



3 In the **Settings** window for **Fillet**, locate the **Radius** section.

4 In the **Radius** text field, type `3[mm]`.

Rectangle 4 (r4)

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 `5[mm]`.

4 In the **Height** text field, type `50[mm]`.

5 Locate the **Position** section. In the **r** text field, type `20[mm]`.

6 In the **z** text field, type `-5[mm]`.

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**.

6 Click **Build Selected**.

DEFINITIONS

Add a **Rigid Domain** material model for domains 1 and 3 (die), and fix the domains. Set the density to zero as it does not affect the analysis.

SOLID MECHANICS (SOLID)

Rigid Domain 1

1 In the **Model Builder** window, expand the **Component 1 (comp1)>Definitions** node.

2 Right-click **Component 1 (comp1)>Solid Mechanics (solid)** and choose **Material Models> Rigid Domain**.

3 Select Domains 1 and 3 only.

4 In the **Settings** window for **Rigid Domain**, locate the **Density** section.

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

Fixed Constraint 1

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

For the elastoplastic analysis of the workpiece, choose the **FKM-GTN** porous plasticity model by adding a **Porous Plasticity** subnode to the **Linear Elastic Material**.

Linear Elastic Material 1

In the **Model Builder** window, 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 **Yield function F** list, choose **FKM-GTN**.

Assign aluminum material properties to domain 2 (workpiece).

MATERIALS

Aluminum

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 `Aluminum` 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	`70[GPa]`	Pa	Basic
Poisson's ratio	nu	`0.33`	1	Basic
Density	rho	`2700`	kg/m³	Basic
Initial yield stress	sigmags	`sigmay0`	Pa	Poroplastic material model
Initial void volume fraction	f0	`F0`	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
Maximum void volume fraction	fmax	`Fmax`	1	Poroplastic material model

SOLID MECHANICS (SOLID)

Contact 1

1 In the **Physics** toolbar, click **Pairs** and choose **Contact**.

2 In the **Settings** window for **Contact**, locate the **Pair Selection** section.

3 Under **Pairs**, click **Add**.

4 In the **Add** dialog box, in the **Pairs** list, choose **Contact Pair 1 (ap1)** and **Contact Pair 2 (ap2)**.

5 Click **OK**.

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 8 only.

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

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

5 In the u_{0z} text field, type `-para`.

Prescribed Displacement 2

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

2 Select Boundary 10 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 `para`.

Now create the mesh. Start by defining a refined mesh in the contact region between the inner die and the workpiece, specifically at the fillet. Use a single mesh element on the straight edges of the inner and outer die.

MESH 1

Mapped 1

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

2 In the **Settings** window for **Mapped**, locate the **Domain Selection** section.

3 From the **Geometric entity level** list, choose **Domain**.

4 Select Domain 3 only.

Distribution 1

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

2 Select Boundaries 15 and 16 only.

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

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

Free Triangular 1

In the **Mesh** toolbar, click **Free Triangular**.

Distribution 1

1 Right-click **Free Triangular 1** and choose **Distribution**.

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

3 From the **Selection** list, choose **All boundaries**.

4 Click **Clear Selection**.

5 Click **Paste Selection**.

6 In the **Paste Selection** dialog box, type 12 in the **Selection** text field.

7 Click **OK**.

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

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

Distribution 2

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

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

3 Click **Paste Selection**.

4 In the **Paste Selection** dialog box, type 5 in the **Selection** text field.

5 Click **OK**.

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

7 In the **Number of elements** text field, type 27.

Distribution 3

1 Right-click **Free Triangular 1** and choose **Distribution**.

2 Select Boundaries 1–4 only.

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

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

Free Triangular 1

Right-click **Free Triangular 1** and choose **Build All**.

STUDY 1

Step 1: Stationary

Set up an auxiliary continuation sweep for the `para` parameter.

1 In the **Model Builder** window, under **Study 1** click **Step 1: Stationary**.

2 In the **Settings** window for **Stationary**, click to expand the **Study Extensions** section.

3 Select the **Auxiliary sweep** check box.

4 Click **Add**.

5 In the table, enter the following settings:

Parameter name	**Parameter value list**	**Parameter unit**
para (Displacement parameter)	`range(0,2e-5,2e-3)`	`m`

Solution 1 (sol1)

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

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

The step size is tuned in order to improve the convergence.

3 In the **Model Builder** window, expand the **Study 1>Solver Configurations>Solution 1 (sol1)>Stationary Solver 1** node, then click **Parametric 1**.

4 In the **Settings** window for **Parametric**, click to expand the **Continuation** section.

5 Select the **Tuning of step size** check box.

6 In the **Initial step size** text field, type `2e-7`.

7 In the **Minimum step size** text field, type `2e-7`.

8 In the **Maximum step size** text field, type `2e-5`.

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

RESULTS

Volumetric Plastic Strain

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

Surface 1

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

2 In the **Settings** window for **Surface**, click **Replace Expression** in the upper-right corner of the **Expression** section. From the menu, choose **Component 1 (comp1)>Solid Mechanics>Strain>Strain invariants>solid.epvol - Volumetric plastic strain**.

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

Volumetric Plastic Strain

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

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

In order to visualize the von Mises stress in the deformed workpiece along with undeformed dies, duplicate the **Study 1/Solution 1** dataset, and set the selection to the domains 1 and 3. Set up a new **Revolution 2D** dataset based on **Study 1/Solution 1 (2)**.

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**.

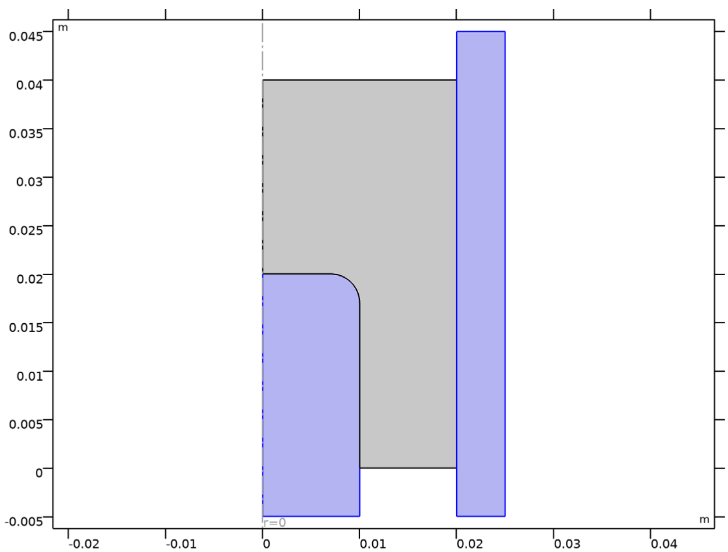
Selection

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 Domains 1 and 3 only.



Revolution 2D 2

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

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

3 From the **Dataset** list, choose **Study 1/Solution 1 (2) (sol1)**.

von Mises Stress

1 In the **Model Builder** window, expand the **Results>Stress, 3D (solid)** node, then click **Stress, 3D (solid)**.

2 In the **Settings** window for **3D Plot Group**, type `von Mises Stress` in the **Label** text field.

Surface 1

The **Scale** in **Deformation** node set to 2 in order to visualize the compaction.

Deformation

1 In the **Model Builder** window, expand the **Results>von Mises Stress>Surface 1** node, then click **Deformation**.

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

3 In the **Scale factor** text field, type `2`.

von Mises Stress

In order to visualize undeformed dies, set up **Surface 2** node by duplicating the **Surface 1** node. Select the **Revolution 2D 2** dataset, and type zero in the expression field. Add a **Material Appearance** node for the visualization.

Surface 2

1 In the **Model Builder** window, under **Results>von Mises Stress** 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 2**.

4 Locate the **Expression** section. In the **Expression** text field, type `0`.

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

Material Appearance 1

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

2 In the **Settings** window for **Material Appearance**, locate the **Appearance** section.

3 From the **Appearance** list, choose **Custom**.

4 From the **Material type** list, choose **Steel**.

5 In the **von Mises Stress** toolbar, click **Plot**.

Current Void Volume Fraction (solid)

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

2 In the **Current Void Volume Fraction (solid)** toolbar, click **Plot**.