Model created in COMSOL Multiphysics 6.4

Indentation of a Cylindrical Battery Cell

Introduction

This study investigates the mechanical response of a cylindrical battery cell in the well-known 18650 format subjected to an indentation test. The test setup consists of a cylindrical indenter, whose axis is perpendicular to that of the battery cell, pressed into the cell. On the opposite side, the cell rests on a flat plate. The objective of the study is to demonstrate the use of an explicit dynamics analysis that accounts for contact mechanics and plastic deformation, and to highlight recommended settings for such simulations.

Model Definition

The battery cell is composed of a jelly roll wound around a central core and enclosed within a thin-walled cylindrical can with a cap. The can has an outer diameter of $D = 18$ mm, a height of $H = 65$ mm, and a wall thickness of $t = 0.6$ mm.

Due to the symmetry of both the geometry and the loading conditions, the problem can be simplified by modeling only half of the cell, along with the indenter and the support plate, using the xz-plane as the symmetry plane.

The jelly roll consists of multiple layers of different materials. For this example, it is treated as a homogenized **Hyperelastic Material** combined with a pressure-dependent plasticity model. The can, core, and cap are metallic components and are therefore modeled as a **Linear Elastic Material** with an additional plasticity model to capture permanent deformation.

Because the indenter is significantly stiffer than the battery cell, it is modeled as a rigid body. Its motion is prescribed by a **Prescribed Displacement**, with the indenter penetrating into the cell by 8 mm.

Accurate representation of the contact between the indenter, cell, and plate requires very small time steps. This motivates the use of explicit dynamics analysis, which is well-suited for such problems due to its ability to handle small, stable time steps with relatively low computational cost per step. Computational efficiency is further improved by employing reduced integration.

However, reduced integration can introduce spurious hourglass modes in the finite element solution. To suppress these unphysical deformations, a stabilization scheme is applied.

The maximum stable time step in explicit dynamics is proportional to the smallest element size for constant density and stiffness. Consequently, a few very small elements can significantly reduce the overall time step. To prevent excessive computational cost, artificial mass scaling is applied to these smallest elements, allowing the use of a target stable time step.

It is recommended to always assess the influence of a stabilization scheme and the artificially added mass on the computed solution.

Results and Discussion

Figure 1 visualizes the pressure p_{m} inside of the jelly roll at the end of the indentation process. The pressure is evaluated as

$$p_{\mathrm{m}} = -\frac{1}{3}\mathrm{tr}(\sigma)$$

with the Cauchy stress tensor σ.

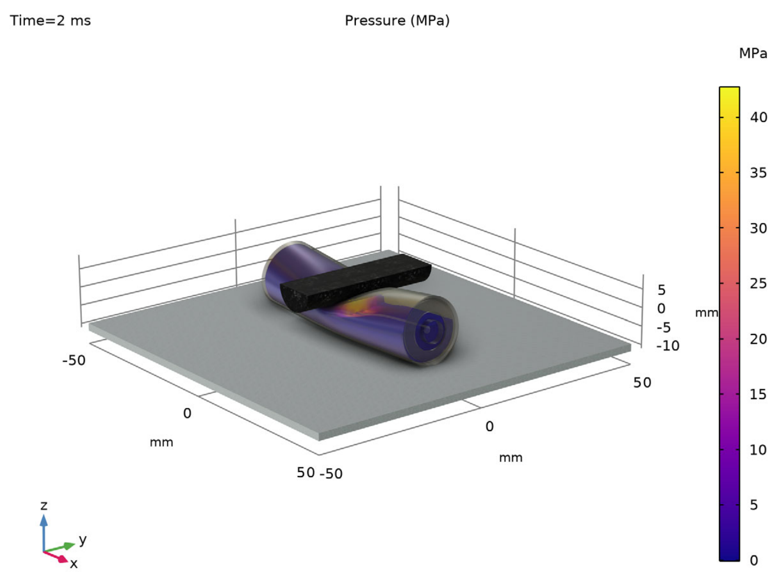

Figure 1: Pressure in the jelly roll at the end of the indentation process.

Figure 2 shows the equivalent plastic strain at the end of the indentation process. The plastic strain exceeds 70% at the center of the cylindrical battery cell.

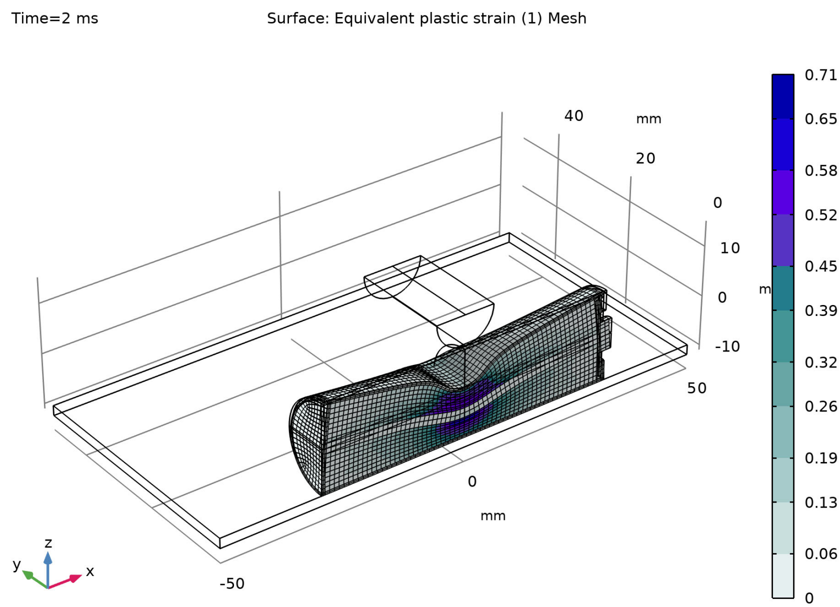

Figure 2: Equivalent plastic strain at the end of the indentation process.

Figure 3 shows the evolution of the reaction force experienced by the indenter as a function of its vertical displacement. As the indentation progresses, both the reaction force and particularly its slope increase. This behavior is attributed to the growing contact area between the cylindrical battery cell and the indenter, which leads to a greater mechanical resistance of the cell against further indentation.

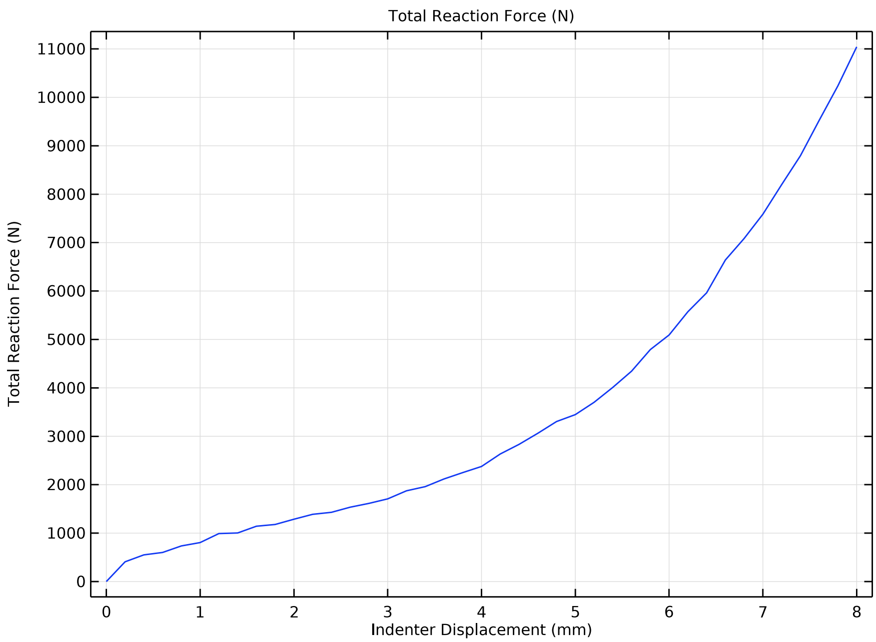

Figure 3: Development of the reaction force in the indenter over the displacement of the indenter.

For the used explicit dynamics analysis with stabilization against hourglass modes and artificially added mass, it is recommended to compare the energies introduced by these methods with physically meaningful energies. Figure 4 compares the different energies occurring in this study over time. Most prominently, it can be seen that most of the energy is dissipated due to the plastic deformations inside of the battery cell, while a comparably minor amount of the energy is stored elastically. The total kinetic energy is almost zero meaning that the given example behaves quasi-static. In this example, the total artificial kinetic energy and the total stabilization energy are of particular interest with the first being an indicator of the influence of the **Mass Scaling** on the computed result and the second of the stabilization scheme. First, it can be seen that the total artificial kinetic energy is almost zero, like the physically meaningful total kinetic energy. From this, it can be concluded that the **Mass Scaling** does not drastically influence the computed results. Second, it can be seen that also the total stabilization strain energy consumes only a minor amount of the energy compared to the physically meaningful dissipated energy and the elastic strain energy. This leads to the conclusion that the computed results are also not critically influenced by the stabilization scheme.

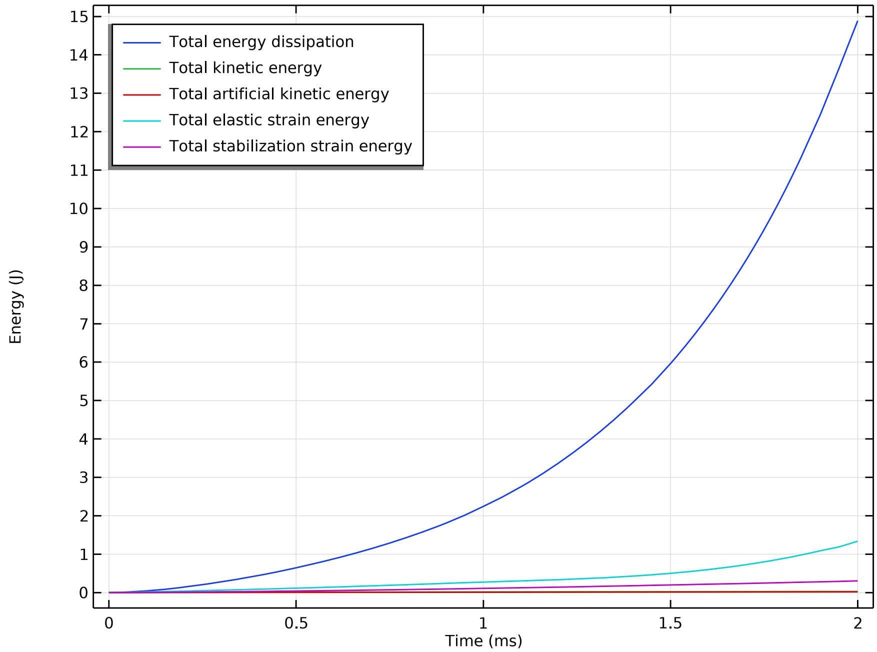

Figure 4: Development of the different energies over time.

Although it was found that the **Mass Scaling** did not deteriorate the computed results, it is still a good habit to check which elements contain artificially added mass and to which extent mass was added. Thus, in Figure 5, the ratio of the artificially added mass density and the physically meaningful mass density is shown. Note that the values are element specific. It can be seen that mass is added where elements are small, have a small aspect ratio, or where the material features a stiff behavior. In this example this is mainly in the can of the battery cell with a small wall thickness and a large Young's modulus.

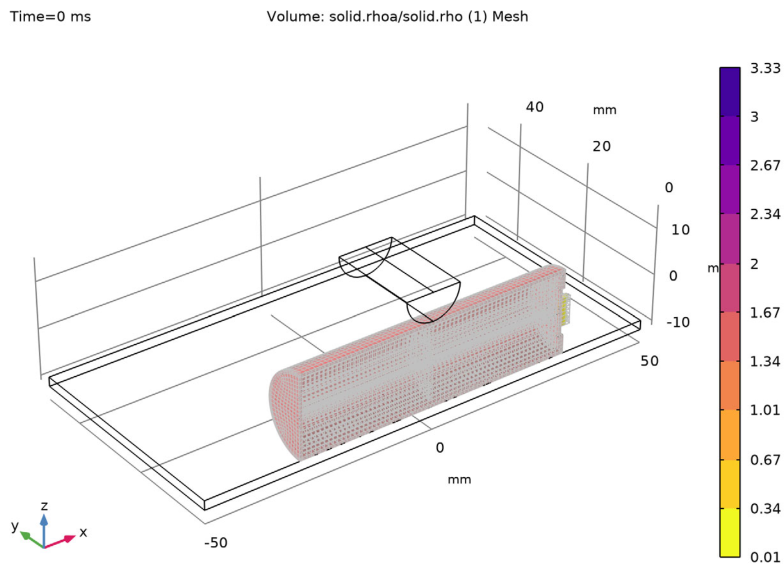

Figure 5: Ratio of artificial mass density and actual mass density.

Notes About the COMSOL Implementation

Consider the following when setting up the model:

- The vertical movement of the indenter is modeled by **Prescribed Displacement** node. This represents a perfectly stiff body.
- To circumvent the need to specify potential contact boundaries between the different components, a **General Contact Pair** is used in which contact mechanics is considered between all domains.
- Multiple ways of evaluating the contact force between the battery cell and the indenter are conceivable due to balance of momentum. In this example, the reaction forces in the indenter are evaluated.
- Particular attention should be paid to the meshing sequence to avoid small or distorted elements that would lead to small time steps in the explicit dynamics analysis.

Application Library path: `Nonlinear_Structural_Materials_Module/Plasticity/cylindrical_battery_indentation`

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 **3D**.

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

3 Click **Add**.

4 Click **Study**.

5 In the **Select Study** tree, select **Preset Studies for Selected Physics Interfaces > Explicit Dynamics**.

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 Click the **Load** button. From the menu, choose **Load from File**.

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

GEOMETRY 1

1 In the **Geometry** toolbar, click **Insert Sequence** and choose **Insert Sequence**.

2 Browse to the model's Application Libraries folder and double-click the file `cylindrical_battery_indentation_geom_sequence.mph`.

Complete geometry instructions can be found in the Appendix — Geometry Modeling Instructions section. Note that the geometric symmetry and the symmetry of the applied load is utilized and the geometry is cut at the zx-plane.

DEFINITIONS

Create a **General Contact Pair** node to include contact between all surfaces. The cap and the core of the battery cell are glued to the shell and the jelly roll, respectively. Model this through a continuity condition by adding an **Identity Boundary Pair**.

General Contact Pair 1 (p1)

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

2 Right-click **Definitions** and choose **Pairs** > **General Contact Pair**.

Identity Boundary Pair 2 (p2)

1 In the **Definitions** toolbar, click **Pairs** and choose **Identity Boundary Pair**.

2 In the **Settings** window for **Pair**, locate the **Source Boundaries** section.

3 Click to select the **Activate Selection** toggle button.

4 Select Boundaries 46, 49, 55, 58, 103, and 108 only.

5 Locate the **Destination Boundaries** section. Click to select the **Activate Selection** toggle button.

6 Select Boundaries 32, 34, 65, 69, 72, and 76 only.

7 Locate the **Frame** section. From the **Source frame** list, choose **Material (X, Y, Z)**.

8 From the **Destination frame** list, choose **Material (X, Y, Z)**.

SOLID MECHANICS, EXPLICIT DYNAMICS (SOLID)

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

2 Select Domains 2–17 only.

3 In the **Settings** window for **Solid Mechanics, Explicit Dynamics**, click to expand the **Energy Dissipation** section.

4 From the **Store dissipation** list, choose **Total**.

Linear Elastic Material 1

In the **Model Builder** window, under **Component 1 (comp1)** > **Solid Mechanics, Explicit Dynamics (solid)** click **Linear Elastic Material 1**.

Plasticity 1

1 In the **Physics** toolbar, click **Attributes** and choose **Plasticity**.
2 Select Domains 2–13 and 15–17 only.
3 In the **Settings** window for **Plasticity**, locate the **Plasticity Model** section.
4 Find the **Isotropic hardening model** subsection. From the list, choose **Swift**.

Hyperelastic Material 1

1 In the **Physics** toolbar, click **Domains** and choose **Hyperelastic Material**.
2 Select Domains 6–9 only.
3 In the **Settings** window for **Hyperelastic Material**, locate the **Hyperelastic Material** section.
4 From the **Specify** list, choose **Young's modulus and Poisson's ratio**.

Pressure-Dependent Plasticity 1

1 In the **Physics** toolbar, click **Attributes** and choose **Pressure-Dependent Plasticity**.
2 In the **Settings** window for **Pressure-Dependent Plasticity**, locate the **Plasticity Model** section.
3 From the list, choose **Foam**.

Define a contact model including friction between all domains.

Contact Model 1

1 In the **Model Builder** window, expand the **Component 1 (comp1)** > **Solid Mechanics, Explicit Dynamics (solid)** > **General Contact 1** node, then click **Contact Model 1**.
2 In the **Settings** window for **Contact Model**, locate the **Contact Model** section.
3 From the **Penalty factor multiplier** list, choose **Manual tuning**.

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 `0.3`.

Add a **Mass Scaling** node to automatically add artificial localized mass to increase the size of the stable time step of the explicit dynamics analysis.

Mass Scaling 1

1 In the **Physics** toolbar, click **Domains** and choose **Mass Scaling**.
2 In the **Settings** window for **Mass Scaling**, locate the **Domain Selection** section.
3 From the **Selection** list, choose **All domains**.

4 Locate the **Mass Scaling** section. In the $\Delta t_{\mathrm{cell}}{}^0$ text field, type `3e-8`.

Add a symmetry condition at the *zx*-plane to respect the symmetry cut in the geometry.

Symmetry 1

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

2 Select Boundaries 8, 11, 14, 16, 20, 26, 30, 35, 44, 48, 53, 57, 64, 67, 71, 74, 86, 89, 92, 95, 97, and 100 only.

Prescribed Displacement 1

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

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

3 Specify the u_0 vector as

`-indenter_velocity*t`	z

4 Select Domain 14 only.

MATERIALS

Can

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 `Can` in the **Label** text field.

3 Locate the **Geometric Entity Selection** section. From the **Selection** list, choose **Manual**.

4 Select Domains 2–5 only.

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

Property	**Variable**	**Value**	**Unit**	**Property group**
Young's modulus	E	`210[GPa]`	Pa	Young's modulus and Poisson's ratio
Poisson's ratio	nu	`0.3`	1	Young's modulus and Poisson's ratio
Density	rho	`7850[kg/m^3]`	kg/m³	Basic
Initial yield stress	sigmags	`350[MPa]`	Pa	Elastoplastic material model

Property	Variable	Value	Unit	Property group
Reference strain	e0_swi	`0.008`	1	Swift
Hardening exponent	n_swi	`0.14`	1	Swift

Jelly Roll

1 Right-click **Materials** and choose **Blank Material**.

2 In the **Settings** window for **Material**, type `Jelly Roll` in the **Label** text field.

3 Select Domains 6–9 only.

4 Locate the **Material Contents** section. In the table, enter the following settings:

Property	Variable	Value	Unit	Property group
Young's modulus	E	`1.5[GPa]`	Pa	Young's modulus and Poisson's ratio
Poisson's ratio	nu	`0`	1	Young's modulus and Poisson's ratio
Density	rho	`2300[kg/m^3]`	kg/m³	Basic
Initial hydrostatic yield stress, compression	pc0	`2[MPa]`	Pa	Foam plasticity
Hydrostatic yield stress, tension	pt	`80[kPa]`	Pa	Foam plasticity
Initial uniaxial compressive yield stress	sigmauc0	`0.4[MPa]`	Pa	Foam plasticity
Hardening function, uniaxial data	sigmauch	`(0.7[MPa]-(0.7[MPa]-sigmauc0)*exp(-epea/0.01))*(1+1250*epea^2.7)-sigmauc0`	Pa	Foam plasticity

Cap

1 Right-click **Materials** and choose **Blank Material**.

2 In the **Settings** window for **Material**, type `Cap` in the **Label** text field.

3 Select Domains 15–17 only.

4 Locate the **Material Contents** section. In the table, enter the following settings:

Property	**Variable**	**Value**	**Unit**	**Property group**
Young's modulus	E	`210[GPa]`	Pa	Young's modulus and Poisson's ratio
Poisson's ratio	nu	`0.3`	1	Young's modulus and Poisson's ratio
Density	rho	`7850[kg/m^3]`	kg/m³	Basic
Initial yield stress	sigmags	`350[MPa]`	Pa	Elastoplastic material model
Reference strain	e0_swi	`0.008`	1	Swift
Hardening exponent	n_swi	`0.14`	1	Swift

Core

1 Right-click **Materials** and choose **Blank Material**.

2 In the **Settings** window for **Material**, type `Core` in the **Label** text field.

3 Select Domains 10–13 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	Young's modulus and Poisson's ratio
Poisson's ratio	nu	`0.35`	1	Young's modulus and Poisson's ratio
Density	rho	`2699[kg/m^3]`	kg/m³	Basic
Initial yield stress	sigmags	`100[MPa]`	Pa	Elastoplastic material model
Reference strain	e0_swi	`0.008`	1	Swift
Hardening exponent	n_swi	`0.14`	1	Swift

Create a dummy material for the rigid indenter.

Indenter

1 Right-click **Materials** and choose **Blank Material**.

2 In the **Settings** window for **Material**, type `Indenter` in the **Label** text field.

3 Select Domain 14 only.

4 Locate the **Material Contents** section. In the table, enter the following settings:

Property	Variable	Value	Unit	Property group
Young's modulus	E	`1[Pa]`	Pa	Young's modulus and Poisson's ratio
Poisson's ratio	nu	`0.0`	1	Young's modulus and Poisson's ratio
Density	rho	`1[kg/m^3]`	kg/m³	Basic

MESH 1

Mapped 1

1 In the **Mesh** toolbar, click **More Generators** and choose **Mapped**.

2 Select Boundaries 2, 39–42, 88, and 96 only.

Distribution 1

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

2 Select Edges 3 and 9 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 Edges 56 and 71 only.

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

4 In the **Number of elements** text field, type `2`.

Free Quad 1

1 In the **Mesh** toolbar, click **More Generators** and choose **Free Quad**.

2 Select Boundaries 61, 62, 77, 78, 81, 91, and 94 only.

Swept 1

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

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

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

4 Select Domains 1, 2, and 4–17 only.

Distribution 1

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

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

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

4 Select Domains 1 and 14 only.

Distribution 2

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

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

3 In the **Number of elements** text field, type `2`.

4 Select Domains 2, 5, 15, and 16 only.

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 Click the **Custom** button.

4 Locate the **Element Size Parameters** section. In the **Maximum element size** text field, type `1`.

5 In the **Minimum element size** text field, type `0.75`.

Free Quad 2

1 In the **Mesh** toolbar, click **More Generators** and choose **Free Quad**.

2 Select Boundaries 10 and 13 only.

Swept 2

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

STUDY 1

Step 1: Explicit Dynamics

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

2 In the **Settings** window for **Explicit Dynamics**, locate the **Study Settings** section.

3 From the **Time unit** list, choose **ms**.

4 In the **Output times** text field, type `range(0,0.05,2)`.

Decrease the number of updates of data structures used by the contact search algorithm. This is justified for the small deformations expected in each time step.

5 Click to expand the **Results While Solving** section. Select the **Plot** checkbox.

6 Click to expand the **Explicit Dynamics Settings** section. Select the **Contact update frequency** checkbox.

7 In the **Hierarchical data** text field, type `100`.

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

Create a dataset that mirrors the solution at the symmetry plane for visualization.

RESULTS

Mirror 3D 1

1 In the **Results** toolbar, click **More Datasets** and choose **Mirror 3D**.

2 In the **Settings** window for **Mirror 3D**, locate the **Plane Data** section.

3 From the **Plane** list, choose **ZX-planes**.

4 Click to expand the **Advanced** section.

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, type `stress` in the text field.

5 In the tree, select **Solid Mechanics** > **Stress tensor (N/m^2)**.

6 Click **OK**.

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

8 In the table, enter the following settings:

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

9 Click **Apply**.

Stress (solid)

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

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

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

4 Select Domains 2–13 and 15–17 only.

Mesh 1

Right-click **Stress (solid)** and choose **Mesh**.

Deformation 1

1 In the **Model Builder** window, expand the **Results** > **Stress (solid)** > **Volume 1** node.

2 Right-click **Mesh 1** and choose **Deformation**.

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

4 Select the **Scale factor** checkbox. In the associated text field, type `1`.

Mesh 1

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

2 In the **Settings** window for **Mesh**, locate the **Coloring and Style** section.

3 From the **Element color** list, choose **None**.

Jelly Roll Pressure

1 In the **Results** toolbar, click **3D Plot Group**.

2 In the **Settings** window for **3D Plot Group**, type `Jelly Roll Pressure` in the **Label** text field.

3 Locate the **Data** section. From the **Dataset** list, choose **Mirror 3D 1**.

4 Locate the **Plot Settings** section. Clear the **Plot dataset edges** checkbox.

5 Locate the **Color Legend** section. Select the **Show units** checkbox.

Jelly Roll

1 Right-click **Jelly Roll Pressure** and choose **Volume**.

2 In the **Settings** window for **Volume**, type `Jelly Roll` in the **Label** text field.

3 Locate the **Expression** section. In the **Expression** text field, type `solid.pmGp`.

4 Locate the **Coloring and Style** section. From the **Color table** list, choose **Plasma**.

Deformation 1

1 Right-click **Jelly Roll** and choose **Deformation**.

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

3 Select the **Scale factor** checkbox. In the associated text field, type `1`.

Selection 1

1 In the **Model Builder** window, right-click **Jelly Roll** and choose **Selection**.

2 Select Domains 6–9 only.

Filter 1

1 Right-click **Jelly Roll** and choose **Filter**.

2 In the **Settings** window for **Filter**, locate the **Element Selection** section.

3 In the **Logical expression for inclusion** text field, type `X<0 || Z < 0`.

Transformation 1

1 Right-click **Jelly Roll** and choose **Transformation**.

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

3 Select the **Scale** checkbox.

4 In the **x** text field, type `0.995`.

5 In the **y** text field, type `0.995`.

6 In the **z** text field, type `0.995`.

Can

1 In the **Model Builder** window, right-click **Jelly Roll Pressure** and choose **Volume**.

2 In the **Settings** window for **Volume**, type `Can` in the **Label** text field.

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

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

Deformation 1

1 Right-click **Can** and choose **Deformation**.

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

3 Select the **Scale factor** checkbox. In the associated text field, type `1`.

Selection 1

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

2 Select Domains 2–5 only.

Transparency 1

1 Right-click **Can** and choose **Transparency**.

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

3 Find the **Fresnel transmittance** subsection. Set the **Fresnel transmittance** value to **0.5**.

RESULTS

Can

In the **Model Builder** window, collapse the **Results** > **Jelly Roll Pressure** > **Can** node.

Material Appearance 1

1 In the **Model Builder** window, expand the **Can** node.
2 Right-click **Can** and choose **Material Appearance**.
3 In the **Settings** window for **Material Appearance**, locate the **Appearance** section.
4 From the **Appearance** list, choose **Custom**.
5 From the **Material type** list, choose **Steel**.

Core

1 In the **Model Builder** window, right-click **Jelly Roll Pressure** and choose **Volume**.
2 In the **Settings** window for **Volume**, type `Core` in the **Label** text field.
3 Locate the **Expression** section. In the **Expression** text field, type `1`.
4 Locate the **Title** section. From the **Title type** list, choose **None**.

Deformation 1

1 Right-click **Core** and choose **Deformation**.
2 In the **Settings** window for **Deformation**, locate the **Scale** section.
3 Select the **Scale factor** checkbox. In the associated text field, type `1`.

Selection 1

1 In the **Model Builder** window, right-click **Core** and choose **Selection**.
2 Select Domains 10–13 only.

Transparency 1

1 Right-click **Core** and choose **Transparency**.
2 In the **Settings** window for **Transparency**, locate the **Transparency** section.
3 Find the **Fresnel transmittance** subsection. In the **Fresnel transmittance** text field, type `0.5`.

Material Appearance 1

1 Right-click **Core** 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 **Aluminum**.

Cap

1 In the **Model Builder** window, right-click **Jelly Roll Pressure** and choose **Volume**.

2 In the **Settings** window for **Volume**, type `Cap` in the **Label** text field.

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

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

Deformation 1

1 Right-click **Cap** and choose **Deformation**.

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

3 Select the **Scale factor** checkbox. In the associated text field, type `1`.

Selection 1

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

2 Select Domains 15–17 only.

Transparency 1

1 Right-click **Cap** and choose **Transparency**.

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

3 Find the **Fresnel transmittance** subsection. In the **Fresnel transmittance** text field, type `0.5`.

Material Appearance 1

1 Right-click **Cap** 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 **Iron**.

Indenter

1 In the **Model Builder** window, right-click **Jelly Roll Pressure** and choose **Volume**.

2 In the **Settings** window for **Volume**, type `Indenter` in the **Label** text field.

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

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

Deformation 1

1 Right-click **Indenter** and choose **Deformation**.

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

3 Select the **Scale factor** checkbox. In the associated text field, type `1`.

Selection 1

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

2 Select Domain 14 only.

Material Appearance 1

1 Right-click **Indenter** 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 **Carbon (forged)**.

Selection 1

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

2 Select Domain 14 only.

3 In the **Jelly Roll Pressure** toolbar, click **Plot**.

Plate

1 In the **Model Builder** window, right-click **Jelly Roll Pressure** and choose **Volume**.

2 In the **Settings** window for **Volume**, type `Plate` in the **Label** text field.

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

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

Selection 1

1 Right-click **Plate** and choose **Selection**.

2 Select Domain 1 only.

Material Appearance 1

1 In the **Model Builder** window, right-click **Plate** 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 **Rock**.

Reaction Force

1 In the **Results** toolbar, click **1D Plot Group**.

2 In the **Settings** window for **1D Plot Group**, type `Reaction Force` in the **Label** text field.

3 Locate the **Legend** section. Clear the **Show legends** checkbox.

Global 1

1 Right-click **Reaction Force** and choose **Global**.

Note, that the expression for the total reaction forces is multiplied by 2 to account for the applied symmetry condition.

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

3 In the table, enter the following settings:

Expression	Unit	Description
sqrt(solid.pdisp1.RFsumx^2+ solid.pdisp1.RFsumy^2+ solid.pdisp1.RFsumz^2)*2	N	Total Reaction Force

4 In the **Description** text field, type `Total Reaction Force`.

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

6 In the **Expression** text field, type `indenter_velocity*t`.

7 Select the **Description** checkbox. In the associated text field, type `Indenter Displacement`.

Energies

1 In the **Results** toolbar, click **1D Plot Group**.

2 In the **Settings** window for **1D Plot Group**, type `Energies` in the **Label** text field.

3 Locate the **Plot Settings** section.

4 Select the **y-axis label** checkbox. In the associated text field, type `Energy (J)`.

Global 1

1 Right-click **Energies** 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
solid.Wd_tot	J	Total energy dissipation
solid.Wk_tot	J	Total kinetic energy
solid.Wka_tot	J	Total artificial kinetic energy
solid.Ws_tot	J	Total elastic strain energy
solid.Wstb_tot	J	Total stabilization strain energy

Energies

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

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

3 From the **Position** list, choose **Upper left**.

Added Mass

1 In the **Results** toolbar, click **3D Plot Group**.

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

3 Locate the **Data** section. From the **Time (ms)** list, choose **0**.

4 Locate the **Selection** section. From the **Geometric entity level** list, choose **Domain**.

5 From the **Selection** list, choose **All domains**.

6 Select Domains 2–13 and 15–17 only.

Volume 1

1 Right-click **Added Mass** and choose **Volume**.

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

3 In the **Expression** text field, type `solid.rhoa/solid.rho`.

4 Locate the **Coloring and Style** section. From the **Color table transformation** list, choose **Reverse**.

5 From the **Color table type** list, choose **Discrete**.

6 From the **Color table** list, choose **Plasma**.

Transparency 1

Right-click **Volume 1** and choose **Transparency**.

Filter 1

1 In the **Model Builder** window, right-click **Volume 1** and choose **Filter**.

2 In the **Settings** window for **Filter**, locate the **Element Selection** section.

3 In the **Logical expression for inclusion** text field, type `solid.rhoa/solid.rho > 0.01`.

Mesh 1

1 In the **Model Builder** window, right-click **Added Mass** and choose **Mesh**.

2 In the **Settings** window for **Mesh**, locate the **Coloring and Style** section.

3 From the **Element color** list, choose **None**.

4 From the **Wireframe color** list, choose **Gray**.

Selection 1

1 Right-click **Mesh 1** and choose **Selection**.

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

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

4 From the **Selection** list, choose **All domains**.

5 Select Domains 2–13 and 15–17 only.

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, Explicit Dynamics** > **Equivalent 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

Equivalent Plastic Strain (solid)

1 In the **Settings** window for **3D Plot Group**, locate the **Selection** section.

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

3 Select Domains 2–13 and 15–17 only.

Deformation 1

1 In the **Model Builder** window, expand the **Equivalent Plastic Strain (solid)** node.

2 Right-click **Surface 1** and choose **Deformation**.

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

4 Select the **Scale factor** checkbox. In the associated text field, type 1.

Mesh 1

1 In the **Model Builder** window, right-click **Equivalent Plastic Strain (solid)** and choose **Mesh**.

2 In the **Settings** window for **Mesh**, locate the **Coloring and Style** section.

3 From the **Element color** list, choose **None**.

Deformation 1

1 Right-click **Mesh 1** and choose **Deformation**.

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

3 Select the **Scale factor** checkbox. In the associated text field, type 1.

Click the **Zoom Box** button on the Graphics toolbar and then use the mouse to zoom in and rotate the view.

Appendix — Geometry 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 **3D**.

2 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 Click the **Load** button. From the menu, choose **Load from File**.

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

GEOMETRY 1

1 In the **Model Builder** window, under **Component 1 (comp1)** click **Geometry 1**.

2 In the **Settings** window for **Geometry**, locate the **Reduction for Symmetry Boundaries** section.

3 Select the **zx-plane: remove y<0** checkbox.

4 Locate the **Units** section. From the **Length unit** list, choose **mm**.

5 Locate the **Cleanup** section. Clear the **Automatic detection of small details** checkbox.

Work Plane 1 (wp1)

1 In the **Geometry** toolbar, click **Work Plane**.

2 In the **Settings** window for **Work Plane**, locate the **Plane Definition** section.

3 From the **Plane** list, choose **yz-plane**.

Work Plane 1 (wp1) > Plane Geometry

In the **Model Builder** window, click **Plane Geometry**.

Work Plane 1 (wp1) > Circle 1 (c1)

1 In the **Work Plane** toolbar, click **Circle**.
2 In the **Settings** window for **Circle**, locate the **Size and Shape** section.
3 In the **Radius** text field, type `(can_outer_diameter)/2-can_wall_thickness`.
4 Click to expand the **Layers** section. In the table, enter the following settings:

Layer name	Thickness (mm)
Layer 1	(can_outer_diameter)/2-can_wall_thickness-core_diameter/2

Work Plane 1 (wp1) > Delete Entities 1 (del1)

1 Right-click **Plane Geometry** and choose **Delete Entities**.
2 In the **Settings** window for **Delete Entities**, locate the **Entities or Objects to Delete** section.
3 From the **Geometric entity level** list, choose **Domain**.
4 On the object **c1**, select Domain 5 only.

Extrude 1 (ext1)

1 In the **Model Builder** window, right-click **Geometry 1** and choose **Extrude**.
2 In the **Settings** window for **Extrude**, locate the **Distances** section.
3 In the table, enter the following settings:

Distances (mm)
jelly_roll_height/2
-jelly_roll_height/2

Work Plane 2 (wp2)

1 In the **Geometry** toolbar, click **Work Plane**.
2 In the **Settings** window for **Work Plane**, locate the **Plane Definition** section.
3 From the **Plane** list, choose **yz-plane**.

Work Plane 2 (wp2) > Plane Geometry

In the **Model Builder** window, click **Plane Geometry**.

Work Plane 2 (wp2) > Circle 1 (c1)

1 In the **Work Plane** toolbar, click **Circle**.
2 In the **Settings** window for **Circle**, locate the **Size and Shape** section.
3 In the **Radius** text field, type `core_diameter/2`.

Work Plane 2 (wp2) > Partition Domains 1 (pard1)

1 In the **Work Plane** toolbar, click **Booleans and Partitions** and choose **Partition Domains**.
2 On the object **c1**, select Domain 1 only.
3 In the **Settings** window for **Partition Domains**, locate the **Partition Domains** section.
4 Click to select the **Activate Selection** toggle button for **Vertices defining line segments**.
5 On the object **c1**, select Points 1 and 3 only.

Extrude 2 (ext2)

1 In the **Model Builder** window, right-click **Geometry 1** and choose **Extrude**.
2 In the **Settings** window for **Extrude**, locate the **Distances** section.
3 In the table, enter the following settings:

Distances (mm)
`jelly_roll_height/2`
`-jelly_roll_height/2`

Work Plane 3 (wp3)

1 In the **Geometry** toolbar, click **Work Plane**.
2 In the **Settings** window for **Work Plane**, locate the **Plane Definition** section.
3 From the **Plane** list, choose **zx-plane**.

Work Plane 3 (wp3) > Plane Geometry

In the **Model Builder** window, click **Plane Geometry**.

Work Plane 3 (wp3) > Circle 1 (c1)

1 In the **Work Plane** toolbar, click **Circle**.
2 In the **Settings** window for **Circle**, locate the **Size and Shape** section.
3 In the **Radius** text field, type `indenter_diameter / 2`.
4 In the **Sector angle** text field, type `180`.
5 Locate the **Rotation Angle** section. In the **Rotation** text field, type `90`.
6 Locate the **Position** section. In the **xw** text field, type `can_outer_diameter / 2 + indenter_diameter / 2 +initial_gap`.

Extrude 3 (ext3)

1 In the **Model Builder** window, right-click **Geometry 1** and choose **Extrude**.

2 In the **Settings** window for **Extrude**, locate the **Distances** section.

3 In the table, enter the following settings:

Distances (mm)
indenter_height / 2
-indenter_height / 2

Work Plane 4 (wp4)

1 In the **Geometry** toolbar, click **Work Plane**.

2 In the **Settings** window for **Work Plane**, locate the **Plane Definition** section.

3 From the **Plane** list, choose **zx-plane**.

Work Plane 4 (wp4) > Plane Geometry

In the **Model Builder** window, click **Plane Geometry**.

Work Plane 4 (wp4) > Polygon 1 (pol1)

1 In the **Work Plane** toolbar, click **Polygon**.

2 In the **Settings** window for **Polygon**, locate the **Coordinates** section.

3 In the table, enter the following settings:

xw (mm)	yw (mm)
0	-jelly_roll_height/2
can_outer_diameter/2 - can_wall_thickness	-jelly_roll_height/2
can_outer_diameter/2 - can_wall_thickness	jelly_roll_height/2
can_outer_diameter/2 - can_wall_thickness	jelly_roll_height/2 + cap_gap+ cap_thickness
hole_diameter/2	jelly_roll_height/2 + cap_gap+ cap_thickness
hole_diameter/2	jelly_roll_height/2 + cap_gap+ cap_thickness + can_wall_thickness
can_outer_diameter/2- can_wall_thickness	jelly_roll_height/2 + cap_gap+ cap_thickness + can_wall_thickness
can_outer_diameter/2	jelly_roll_height/2 + cap_gap+ cap_thickness + can_wall_thickness
can_outer_diameter/2	jelly_roll_height/2 + cap_gap+ cap_thickness
can_outer_diameter/2	jelly_roll_height/2
can_outer_diameter/2	-jelly_roll_height/2

xw (mm)	yw (mm)
can_outer_diameter/2	-jelly_roll_height/2 - can_wall_thickness
can_outer_diameter/2-can_wall_thickness	-jelly_roll_height/2 - can_wall_thickness
0	-jelly_roll_height/2 - can_wall_thickness
0	-jelly_roll_height/2

Work Plane 4 (wp4) > Partition Domains 1 (pard1)

1 In the **Work Plane** toolbar, click **Booleans and Partitions** and choose **Partition Domains**.

2 On the object **pol1**, select Domain 1 only.

3 In the **Settings** window for **Partition Domains**, locate the **Partition Domains** section.

4 Click to select the **Activate Selection** toggle button for **Vertices defining line segments**.

5 On the object **pol1**, select Points 5–8 and 11–13 only.

Revolve 1 (rev1)

In the **Model Builder** window, right-click **Geometry 1** and choose **Revolve**.

Work Plane 5 (wp5)

1 In the **Geometry** toolbar, click **Work Plane**.

2 In the **Settings** window for **Work Plane**, locate the **Plane Definition** section.

3 From the **Plane** list, choose **zx-plane**.

Work Plane 5 (wp5) > Plane Geometry

In the **Model Builder** window, click **Plane Geometry**.

Work Plane 5 (wp5) > Polygon 1 (pol1)

1 In the **Work Plane** toolbar, click **Polygon**.

2 In the **Settings** window for **Polygon**, locate the **Coordinates** section.

3 In the table, enter the following settings:

xw (mm)	yw (mm)
can_outer_diameter / 2-can_wall_thickness*2	jelly_roll_height / 2+ cap_gap + cap_thickness
hole_diameter/2	jelly_roll_height / 2+ cap_gap + cap_thickness
terminal_diameter/2	jelly_roll_height / 2+ cap_gap + cap_thickness

xw (mm)	yw (mm)
terminal_diameter/2	jelly_roll_height / 2 + cap_gap + cap_thickness+terminal_height
0	jelly_roll_height / 2 + cap_gap + cap_thickness+terminal_height
0	jelly_roll_height / 2 + cap_gap
terminal_diameter/2	jelly_roll_height / 2 + cap_gap
hole_diameter/2	jelly_roll_height / 2 + cap_gap
can_outer_diameter / 2-can_wall_thickness*2	jelly_roll_height / 2 + cap_gap

Work Plane 5 (wp5) > Partition Domains 1 (pard1)

1 In the **Work Plane** toolbar, click **Booleans and Partitions** and choose **Partition Domains**.

2 On the object **pol1**, select Domain 1 only.

3 In the **Settings** window for **Partition Domains**, locate the **Partition Domains** section.

4 Click to select the **Activate Selection** toggle button for **Vertices defining line segments**.

5 On the object **pol1**, select Points 3, 4, 6, and 7 only.

Revolve 2 (rev2)

In the **Model Builder** window, right-click **Geometry 1** and choose **Revolve**.

Block 1 (blk1)

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

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

3 From the **Base** list, choose **Center**.

4 Locate the **Size and Shape** section. In the **Width** text field, type `plate_side_length`.

5 In the **Depth** text field, type `plate_side_length`.

6 In the **Height** text field, type `plate_height`.

7 Locate the **Position** section. In the **z** text field, type `-can_outer_diameter / 2 - plate_height / 2`.

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 Clear the **Create pairs** checkbox.

Form Composite Domains 1 (cmd1)

1 In the **Geometry** toolbar, click **Virtual Operations** and choose **Form Composite Domains**.

2 On the object **fin**, select Domains 4 and 5 only.

3 In the **Geometry** toolbar, click **Build All**.