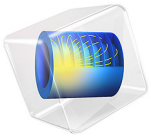

Model created in COMSOL Multiphysics 6.4

# Objects Falling in a Box

## *Introduction*

This conceptual example shows how to handle dynamic contact between multiple objects using the general contact pair functionality and explicit time-stepping. Contact interactions may occur arbitrarily between any of a set of deformable objects falling under the influence of gravity, and also with a rigid surrounding box. The contact model includes damping and friction.

## *Model Definition*

The geometry consists of several geometry objects built from basic shapes including spheres, cylinders, toruses, and dodecahedra. Each object is positioned randomly at the top of a rigid box with an obstacle to induce seemingly chaotic interactions as the objects fall under the influence of gravity. The objective is to demonstrate the automatic contact detection of a general contact pair. The geometry at the start of the analysis is shown in Figure 1.

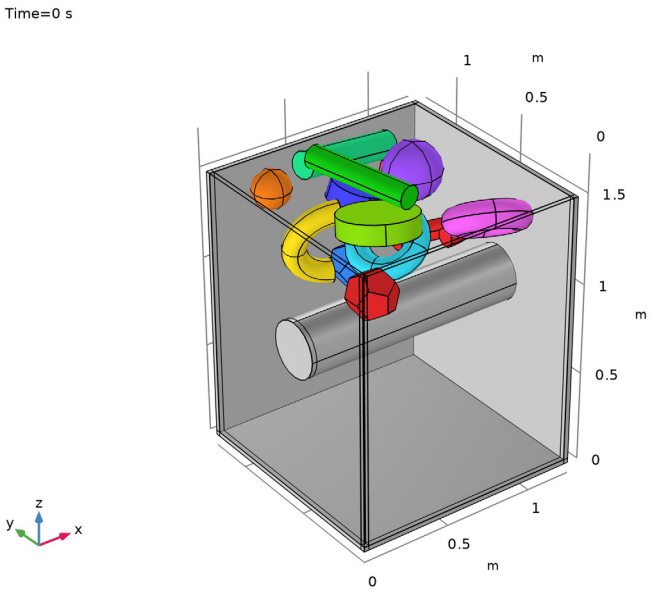


*Figure 1: Geometry. The ox*

The material properties of the deformable objects in Table 1 are fictitious but chosen to resemble a rubber-like material. For the created mesh, these properties result in a time step

for the explicit solver of the order of 1 microseconds. The box is assumed fixed and thus no material definition is required.

TABLE 1: MATERIAL PROPERTIES.

| **Property** | **Value** |
|---|---|
| Young's modulus | 30 MPa |
| Poisson's ratio | 0.3 |
| Density | 4000 $kg/m^3$ |

Contact interactions between all objects as well as with the box is set up by adding a single **General Contact Pair**. The contact properties defined in the default **General Contact** node include friction with an assumed static coefficient of friction equal to 1.1 when the deformable objects interact with each other, and 0.6 when they interact with the box. Contact damping is also included to stabilize the contact state and reduce high-frequency oscillations in the contact pressure.

## *Results and Discussion*

Five snapshots are shown in Figure 2 indicating how contact interactions evolve with time as the deformable objects fall. To verify that the solution using explicit time-stepping is sound, the most important energy contributions are shown in Figure 3.

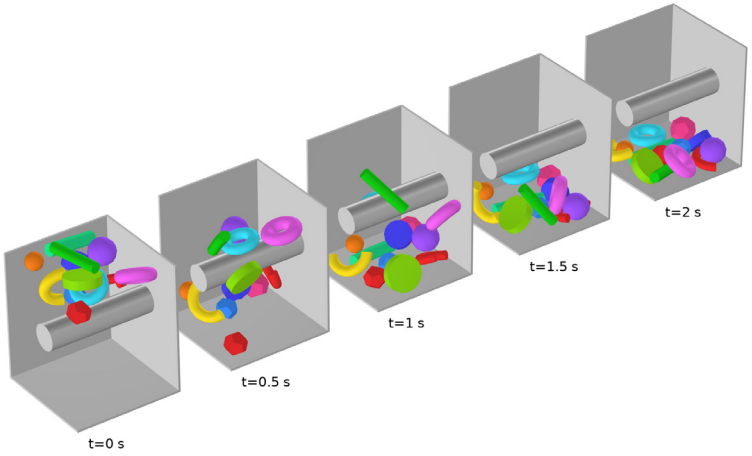


*Figure 2: Deformation at different times.*

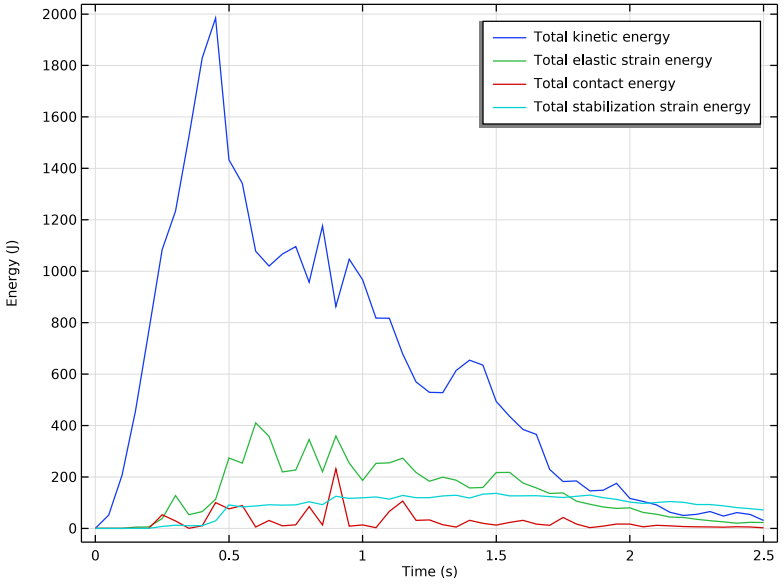


*Figure 3: Total energies.*

## *Notes About the COMSOL Implementation*

The **Solid Mechanics, Explicit Dynamics** interface only supports a linear discretization order and by default enables reduced integration for all material models. Thin and unconstrained objects, as in this model, are sensitive to hourglass deformations. The default hourglass stiffness is increased to further suppress such instabilities. Always make sure that there are no hourglass modes present in the model, and that the energy added by hourglass stabilization is small.

An **Artificial Viscosity** node is added to partially damp high-frequency noise in the deformable objects caused by impact.

The **General Contact Pair** automatically handles false contact mapping for self-intersection However, for this chaotic contact between many thin and coarsely meshed objects, the stability of the contact detection can be further improved by using a stricter search distance. Here, a search distance equal to the characteristic size of the local mesh element is used by setting **Tolerance** to 1.

**Application Library path:** `Structural_Mechanics_Module/Contact_and_Friction/falling_objects`

## *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**.

## GEOMETRY 1

Insert the geometry sequence from the `falling_objects_geom_sequence.mph` file. Complete instructions to create the geometry are available in the Appendix — Geometry Modeling Instructions section.

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 `falling_objects_geom_sequence.mph`.

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

Add a **General Contact Pair** to define contact interaction between all geometry objects in the assembly.

## DEFINITIONS

*General Contact Pair 1 (p1)*

1 In the **Definitions** toolbar, click **Pairs** and choose **General Contact Pair**.

The search distance can be set manually. Use a strict tolerance to avoid possible false contact mappings for this complex assembly of many thin and coarsely meshed objects.

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

3 From the **Search distance** list, choose **Tolerance**.

4 In the **Tolerance** text field, type `1`.

## SOLID MECHANICS, EXPLICIT DYNAMICS (SOLID)

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

2 In the **Settings** window for **Solid Mechanics, Explicit Dynamics**, locate the **Domain Selection** section.

3 From the **Selection** list, choose **Falling objects**.

*Linear Elastic Material 1*

Thin and unconstrained domains can be sensitive to hourglass instabilities. Use the Flanagan–Belytschko stabilization method with an increased stiffness to suppress such instabilities. The amount of stabilization added can be inspected a posteriori.

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

2 In the **Settings** window for **Linear Elastic Material**, locate the **Quadrature Settings** section.

3 Find the **Hexahedron** subsection. From the **Hourglass stabilization** list, choose **Flanagan–Belytschko**.

4 In the $f_{\text{stb}}$ text field, type `4`.

*Contact Model 1*

1 In the **Model Builder** window, expand the **General Contact 1** node, then click **Contact Model 1**.

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

3 Find the **Penalty function** subsection. From the list, choose **Smooth ramp**.

*Friction 1*

1 In the **Physics** toolbar, click **Attributes** and choose **Friction**.

The static friction coefficient can be defined conditionally, based on what type of boundary a contact mapping finds. Here, assume that the friction coefficient is 1.1 when the falling objects interact with each other, but 0.6 when they interact with the box.

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

3 In the μ text field, type `if(src2dst_p1(p1.deformable), 1.1, 0.6)`.

Add **Contact Damping** and **Artificial Viscosity** to damp high-frequency noise and vibrations.

*Contact Model 1*

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

*Damping 1*

1 In the **Physics** toolbar, click **Attributes** and choose **Damping**.

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

3 In the ξ text field, type `0.05`.

*Artificial Viscosity 1*

1 In the **Physics** toolbar, click **Domains** and choose **Artificial Viscosity**.

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

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

*Gravity 1*

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

## MATERIALS

*Material 1 (mat1)*

1 In the **Model Builder** window, under **Component 1 (comp1)** right-click **Materials** and choose **Blank Material**.

A fictitious but rubber-like material is used for this example. Material properties are chosen to obtain a large time step for the explicit dynamics solution rather than being realistic.

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

3 In the table, enter the following settings:

| Property | Variable | Value | Unit | Property group |
|---|---|---|---|---|
| Young's modulus | E | 30e6 | Pa | Young's modulus and Poisson's ratio |
| Poisson's ratio | nu | 0.3 | 1 | Young's modulus and Poisson's ratio |
| Density | rho | 4000 | kg/m³ | Basic |

4 Click to expand the **Appearance** section. From the **Material type** list, choose **Rubber**.

For explicit dynamics, it is important to create a mesh with a uniform size distribution and avoid small mesh elements that restrict the time step size. Also use a swept mesh were possible to facilitate a hex dominant mesh.

## MESH 1

*Free Tetrahedral 1*

1 In the **Mesh** toolbar, click  **Free Tetrahedral**.

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

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

4 Select Domains 14, 15, 23, 26, and 29 only.

*Size 1*

1 Right-click **Free Tetrahedral 1** and choose **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.

5 Select the **Maximum element size** checkbox. In the associated text field, type `0.075`.

*Swept 1*

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

*Size 1*

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

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

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

4 Select Domains 5, 11, 16–22, 24, 25, 27, 28, 30–32, and 37 only.

5 Locate the **Element Size** section. Click the **Custom** button.

6 Locate the **Element Size Parameters** section.

7 Select the **Maximum element size** checkbox. In the associated text field, type `0.05`.

*Size 2*

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

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

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

4 Select Domains 1–4, 6–10, 12, 13, 33–36, 38, and 39 only.

5 Locate the **Element Size** section. From the **Predefined** list, choose **Extremely coarse**.

*Distribution 1*

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

2 Select Domains 17 and 24 only.

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

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

5 Click **Build All**.

## 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 In the **Output times** text field, type `range(0,0.05,2.5)`.

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

## RESULTS

*Deformation*

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

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

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

4 Locate the **Color Legend** section. Clear the **Show legends** checkbox.

*Surface 1*

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

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

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

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

5 Locate the **Coloring and Style** section. From the **Color table** list, choose **Cyclic**.

*Deformation 1*

1 Right-click **Surface 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`.

*Selection 1*

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

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

3 From the **Selection** list, choose **Falling objects**.

*Material Appearance 1*

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

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

3 Select the **Use the plot's color** checkbox.

*Surface 2*

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

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

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

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

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

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

*Selection 1*

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

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

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

4 Select Domains 1, 3, 5–8, 10–13, and 33–39 only.

5 In the **Deformation** toolbar, click **Plot**.

*Deformation 1*

1 In the **Model Builder** window, right-click **Deformation** and choose **Duplicate**.

2 Expand the **Deformation 1** node.

*Solution Array 1*

1 In the **Model Builder** window, expand the **Results** > **Deformation 1** > **Surface 1** node.

2 Right-click **Surface 1** and choose **Solution Array**.

*Multiselect Solution 1*

1 In the **Results** toolbar, click **Configurations** and choose **Multiselect Solution**.

2 In the **Settings** window for **Multiselect Solution**, locate the **Solution** section.

3 From the **Time selection** list, choose **Manual**.

4 In the **Time indices (1-51)** text field, type `1 11 21 31 41`.

*Solution Array 1*

1 In the **Model Builder** window, under **Results** > **Deformation 1** > **Surface 1** click **Solution Array 1**.

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

3 From the **Solution parameters** list, choose **From configuration**.

Press Ctrl, then drag and drop on **Surface 2**.

*Surface 2*

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

2 In the **Settings** window for **Surface**, click to expand the **Plot Array** section.

3 Select the **Manual indexing** checkbox.

*Annotation 1*

1 In the **Model Builder** window, right-click **Deformation 1** and choose **Annotation**.

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

3 In the **Text** text field, type `t=eval(t) s`.

4 Click to expand the **Advanced** section. Clear the **Show trailing zeros** checkbox.

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

6 Click to expand the **Plot Array** section. Select the **Manual indexing** checkbox.

*Solution Array 1*

1 Right-click **Annotation 1** and choose **Solution Array**.

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

3 From the **Solution parameters** list, choose **From configuration**.

4 In the **Deformation 1** toolbar, click **Plot**.

*Deformation Array*

1 In the **Model Builder** window, under **Results** click **Deformation 1**.

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

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

4 Click to expand the **Plot Array** section. In the **Relative padding** text field, type `0.5`.

5 In the **Deformation Array** toolbar, click **Plot**.

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

Add a plot showing the most important energy contributions to verify that the solution is sound.

*Total Energies*

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

2 In the **Settings** window for **1D Plot Group**, type `Total 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 **Total Energies** and choose **Global**.

2 In the **Settings** window for **Global**, click **Replace Expression** in the upper-right corner of the **y-Axis Data** section. From the menu, choose **Component 1 (comp1)** > **Solid Mechanics, Explicit Dynamics** > **Global** > **solid.Wk_tot - Total kinetic energy - J**.

3 Click **Add Expression** in the upper-right corner of the **y-Axis Data** section. From the menu, choose **Component 1 (comp1)** > **Solid Mechanics, Explicit Dynamics** > **Global** > **solid.Ws_tot - Total elastic strain energy - J**.

4 Click **Add Expression** in the upper-right corner of the **y-Axis Data** section. From the menu, choose **Component 1 (comp1)** > **Solid Mechanics, Explicit Dynamics** > **Global** > **solid.Wcnt_tot - Total contact energy - J**.

5 Click **Add Expression** in the upper-right corner of the **y-Axis Data** section. From the menu, choose **Component 1 (comp1)** > **Solid Mechanics, Explicit Dynamics** > **Global** > **solid.Wstb_tot - Total stabilization strain energy - J**.

6 In the **Total Energies** toolbar, click **Plot**.

*Deformation*

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

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

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

4 From the **Time (s)** list, choose **1**.

5 In the **Deformation** toolbar, click **Plot**.

## *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**.

### GEOMETRY 1

*Block 1 (blk1)*

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

2 In the **Settings** window for **Block**, locate the **Size and Shape** section.

3 In the **Width** text field, type `1.25`.

4 In the **Depth** text field, type `1.25`.

5 In the **Height** text field, type `1.5`.

6 Click to expand the **Layers** section. In the table, enter the following settings:

| Layer name | Thickness (m) |
|---|---|
| Layer 1 | 0.025 |

7 Find the **Layer position** subsection. Select the **Left** checkbox.

8 Select the **Right** checkbox.

9 Select the **Front** checkbox.

10 Select the **Back** checkbox.

*Delete Entities 1 (del1)*

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

2 On the object **blk1**, select Boundary 37 only.

*Cylinder 1 (cyl1)*

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

2 In the **Settings** window for **Cylinder**, locate the **Size and Shape** section.

3 In the **Radius** text field, type `0.15`.

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

5 Locate the **Position** section. In the **y** text field, type `0.55`.

6 In the **z** text field, type `0.85`.

7 Locate the **Axis** section. From the **Axis type** list, choose **x-axis**.

*Sphere 1 (sph1)*

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

2 In the **Settings** window for **Sphere**, locate the **Size** section.

3 In the **Radius** text field, type `0.15`.

4 Locate the **Position** section. In the **x** text field, type `0.85`.

5 In the **y** text field, type `0.75`.

6 In the **z** text field, type `1.45`.

7 Locate the **Selections of Resulting Entities** section. Find the **Cumulative selection** subsection. Click **New**.

8 In the **New Cumulative Selection** dialog, type `Falling objects` in the **Name** text field.

9 Click **OK**.

*Sphere 2 (sph2)*

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

2 In the **Settings** window for **Sphere**, locate the **Size** section.

3 In the **Radius** text field, type `0.1`.

4 Locate the **Position** section. In the **x** text field, type `0.2`.

5 In the **y** text field, type `1`.

6 In the **z** text field, type `1.45`.

7 Locate the **Selections of Resulting Entities** section. Find the **Cumulative selection** subsection. From the **Contribute to** list, choose **Falling objects**.

*Cylinder 2 (cyl2)*

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

2 In the **Settings** window for **Cylinder**, locate the **Size and Shape** section.

3 In the **Radius** text field, type `0.06`.

4 In the **Height** text field, type `0.6`.

5 Locate the **Position** section. In the **x** text field, type `0.45`.

6 In the **y** text field, type `0.25`.

7 In the **z** text field, type `1.65`.

8 Locate the **Axis** section. From the **Axis type** list, choose **Cartesian**.

9 In the **x** text field, type `-0.2`.

10 In the **y** text field, type `1`.

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

12 Locate the **Selections of Resulting Entities** section. Find the **Cumulative selection** subsection. From the **Contribute to** list, choose **Falling objects**.

*Cylinder 3 (cyl3)*

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

2 In the **Settings** window for **Cylinder**, locate the **Size and Shape** section.

3 In the **Radius** text field, type `0.07`.

4 In the **Height** text field, type `0.6`.

5 Locate the **Position** section. In the **x** text field, type `0.35`.

6 In the **y** text field, type `0.95`.

7 In the **z** text field, type `1.55`.

8 Locate the **Axis** section. From the **Axis type** list, choose **Cartesian**.

9 In the **x** text field, type `1`.

10 In the **y** text field, type `0.2`.

11 In the **z** text field, type `-0.2`.

12 Locate the **Selections of Resulting Entities** section. Find the **Cumulative selection** subsection. From the **Contribute to** list, choose **Falling objects**.

*Cylinder 4 (cyl4)*

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

2 In the **Settings** window for **Cylinder**, locate the **Size and Shape** section.

3 In the **Radius** text field, type `0.2`.

4 In the **Height** text field, type `0.1`.

5 Locate the **Position** section. In the **x** text field, type `0.35`.

6 In the **y** text field, type `0.35`.

7 In the **z** text field, type `1.45`.

8 Locate the **Axis** section. From the **Axis type** list, choose **Cartesian**.

9 In the **x** text field, type `0.1`.

10 In the **y** text field, type `0.2`.

11 Locate the **Selections of Resulting Entities** section. Find the **Cumulative selection** subsection. From the **Contribute to** list, choose **Falling objects**.

*Cylinder 5 (cyl5)*

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

2 In the **Settings** window for **Cylinder**, locate the **Size and Shape** section.

3 In the **Radius** text field, type `0.15`.

4 In the **Height** text field, type `0.1`.

5 Locate the **Position** section. In the **x** text field, type `0.65`.

6 In the **y** text field, type `0.95`.

7 In the **z** text field, type `1.25`.

8 Locate the **Axis** section. From the **Axis type** list, choose **Cartesian**.

9 In the **x** text field, type `0.3`.

10 In the **y** text field, type `-0.1`.

11 Locate the **Selections of Resulting Entities** section. Find the **Cumulative selection** subsection. From the **Contribute to** list, choose **Falling objects**.

*Torus 1 (tor1)*

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

2 In the **Settings** window for **Torus**, locate the **Size and Shape** section.

3 In the **Major radius** text field, type `0.15`.

4 In the **Minor radius** text field, type `0.06`.

5 Select the **Interior faces** checkbox.

6 Locate the **Position** section. In the **x** text field, type `0.55`.

7 In the **y** text field, type `0.55`.

8 In the **z** text field, type `1.25`.

9 Locate the **Axis** section. From the **Axis type** list, choose **Cartesian**.

10 In the **x** text field, type `2`.

11 In the **y** text field, type `1`.

12 In the **z** text field, type `-3`.

13 Locate the **Selections of Resulting Entities** section. Find the **Cumulative selection** subsection. From the **Contribute to** list, choose **Falling objects**.

*Torus 2 (tor2)*

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

2 In the **Settings** window for **Torus**, locate the **Size and Shape** section.

3 In the **Major radius** text field, type `0.15`.

4 In the **Minor radius** text field, type `0.07`.

5 Select the **Interior faces** checkbox.

6 Locate the **Position** section. In the **x** text field, type `0.95`.

7 In the **y** text field, type `0.3`.

8 In the **z** text field, type `1.35`.

9 Locate the **Axis** section. From the **Axis type** list, choose **Cartesian**.

10 In the **x** text field, type `1`.

11 In the **y** text field, type `2`.

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

13 Locate the **Selections of Resulting Entities** section. Find the **Cumulative selection** subsection. From the **Contribute to** list, choose **Falling objects**.

*Torus 3 (tor3)*

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

2 In the **Settings** window for **Torus**, locate the **Size and Shape** section.

3 In the **Major radius** text field, type `0.2`.

4 In the **Minor radius** text field, type `0.06`.

5 In the **Revolution angle** text field, type `180`.

6 Locate the **Position** section. In the **x** text field, type `0.35`.

7 In the **y** text field, type `0.7`.

8 In the **z** text field, type `1.25`.

9 Locate the **Axis** section. From the **Axis type** list, choose **Cartesian**.

10 In the **x** text field, type `1`.

11 In the **y** text field, type `3`.

12 Locate the **Selections of Resulting Entities** section. Find the **Cumulative selection** subsection. From the **Contribute to** list, choose **Falling objects**.

*Torus 4 (tor4)*

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

2 In the **Settings** window for **Torus**, locate the **Size and Shape** section.

3 In the **Major radius** text field, type `0.22`.

4 In the **Minor radius** text field, type `0.05`.

5 In the **Revolution angle** text field, type `180`.

6 Locate the **Position** section. In the **x** text field, type `0.9`.

7 In the **y** text field, type `0.75`.

8 In the **z** text field, type `1.05`.

9 Locate the **Axis** section. From the **Axis type** list, choose **Cartesian**.

10 In the **y** text field, type `-1`.

11 In the **z** text field, type `-2`.

12 Locate the **Selections of Resulting Entities** section. Find the **Cumulative selection** subsection. From the **Contribute to** list, choose **Falling objects**.

## PART LIBRARIES

1 In the **Geometry** toolbar, click  **Part Libraries**.

2 In the **Part Libraries** window, select **COMSOL Multiphysics** > **Platonic Solids** > **dodecahedron** in the tree.

3 Click  **Add to Geometry**.

4 In the **Select Part Variant** dialog, select **Specify circumradius** in the **Select part variant** list.

5 Click **OK**.

### GEOMETRY 1

*Dodecahedron 1 (pi1)*

1 In the **Model Builder** window, under **Component 1 (comp1)** > **Geometry 1** click **Dodecahedron 1 (pi1)**.

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

3 In the table, enter the following settings:

| Name | Expression | Value | Description |
|---|---|---|---|
| r_c | 0.15 | 0.15 | Circumradius |

4 Locate the **Position and Orientation of Output** section. Find the **Displacement** subsection. In the **xwi** text field, type 1.

5 In the **ywi** text field, type 1.

6 In the **zwi** text field, type 1.25.

*Dodecahedron 2 (pi2)*

1 In the **Geometry** toolbar, click  **Part Instance** and choose **Dodecahedron**.

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

3 In the table, enter the following settings:

| Name | Expression | Value | Description |
|---|---|---|---|
| r_c | 0.13 | 0.13 | Circumradius |

4 Locate the **Position and Orientation of Output** section. Find the **Displacement** subsection. In the **xwi** text field, type 0.2.

5 In the **ywi** text field, type 0.2.

6 In the **zwi** text field, type 1.25.

*Dodecahedron 3 (pi3)*

1 In the **Geometry** toolbar, click  **Part Instance** and choose **Dodecahedron**.

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

3 In the table, enter the following settings:

| Name | Expression | Value | Description |
|---|---|---|---|
| r_c | 0.12 | 0.12 | Circumradius |

4 Locate the **Position and Orientation of Output** section. Find the **Displacement** subsection. In the **xwi** text field, type `0.6`.

5 In the **ywi** text field, type `0.9`.

6 In the **zwi** text field, type `0.95`.

*Explicit Selection 1 (sel1)*

1 In the **Geometry** toolbar, click  **Selections** and choose **Explicit Selection**.

2 In the **Settings** window for **Explicit Selection**, locate the **Entities to Select** section.

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

4 Select the objects **pi1**, **pi2**, and **pi3** only.

5 Locate the **Resulting Selection** section. Find the **Cumulative selection** subsection. From the **Contribute to** list, choose **Falling objects**.

6 Clear the **Keep selection** checkbox.