Created in COMSOL Multiphysics 6.1

Using Meshing Sequences

Introduction

COMSOL Multiphysics provides an interactive meshing environment where, with a few mouse clicks, you can easily mesh individual faces or domains. Each meshing operation is added to the meshing sequence. The final mesh is the result of building all the operations in the meshing sequence.

This example demonstrates how to use the meshing sequence to create a mesh consisting of different element types. You learn how to add, move, disable and delete mesh operations, and how to control the mesh using size features in the meshing sequence.

Model Definition

The geometry shown in Figure 1 represents a small part of a circuit board with an electronic component (chip) mounted by means of several solder ball joints.

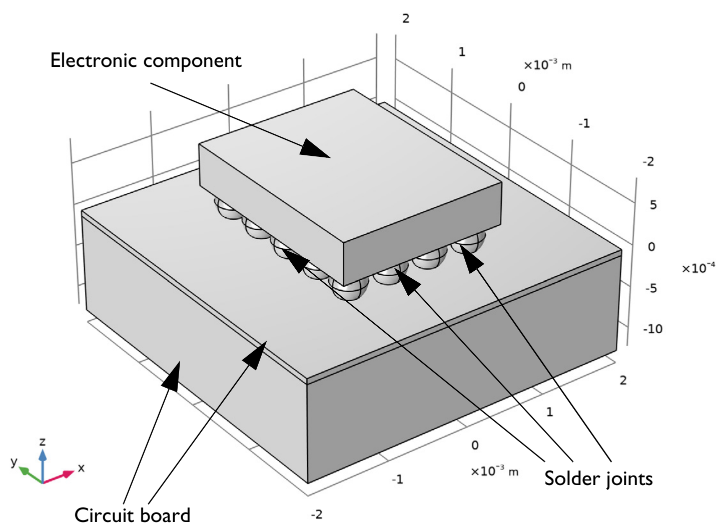


Figure 1: Geometry consisting of an electronic component mounted on a circuit board.

Electronic components can experience high temperatures during prolonged time periods, which can lead to permanent deformation and failure of the solder joints due to creep in the material.

For a larger geometry, this phenomenon is studied in the model *Viscoplastic Creep in Solder Joints* found under Nonlinear Structural Mechanics Module>Viscoplasticity. Note, however, that running this model requires additional licenses.

Application Library path: `COMSOL_Multiphysics/Meshing_Tutorials/meshing_sequence`

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

Insert the geometry sequence from the `meshing_sequence_geom_sequence.mph` file.

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 `meshing_sequence_geom_sequence.mph`.

Block 1 (blk1)

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

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

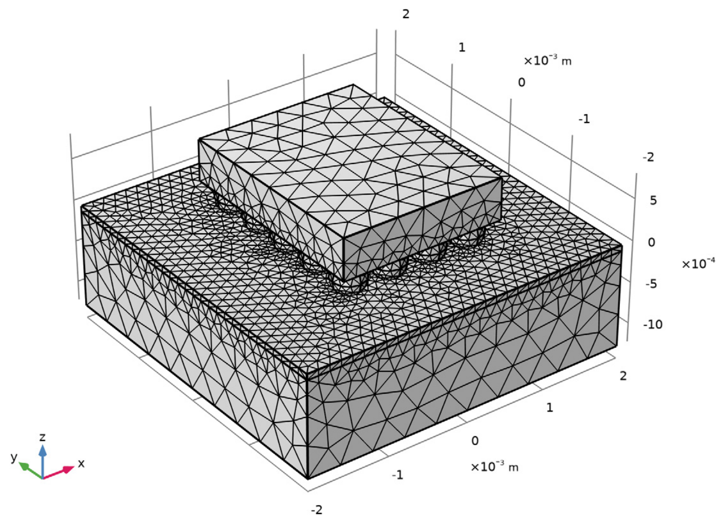
Continue by creating a simple unstructured tetrahedral mesh.

MESH 1

As you click the **Mesh 1** node, the **Form Union** node is automatically built, which means that the various objects are combined into one object with domains separated by inner boundaries. This assures that a continuous mesh can be created across touching geometry objects that make up the geometry in this case.

With the default **Physics-controlled mesh**, in the **Sequence type** list in the **Mesh** settings window, COMSOL automatically creates a mesh adapted for the physics settings in the model. A set of nine predefined element sizes, ranging from **Extremely fine** to **Extremely coarse** and the default **Normal** size, give you control of how well you would like to resolve the geometry. With the default physics-controlled option, the sequence of mesh operations is always hidden.

1 In the **Model Builder** window, under **Component 1 (comp1)** right-click **Mesh 1** and choose **Build All**.



According to information displayed in the **Messages** window this mesh consists of approximately 45,000 elements. While the geometry is resolved quite well by this mesh, you may want to reduce the number of elements to reduce the memory required for solving the problem.

In the following instructions you will manually modify the meshing sequence.

2 In the **Settings** window for **Mesh**, locate the **Sequence Type** section.

3 From the list, choose **User-controlled mesh**.

A default meshing sequence consisting of the **Size** and **Free Tetrahedral 1** nodes appears under the **Mesh 1** node.

The first **Size** node is called a *global attribute*, since it influences all subsequent *operations* in the meshing sequence. This first **Size** node cannot be deleted from the meshing sequence. You can also add attributes as a subnode to an operation node, in which case it is called a *local attribute*.

Free Tetrahedral 1

Assume that you are investigating the solder joints and would therefore like to keep the detailed mesh in the spherical domains but create a mesh with fewer elements in the remaining domains.

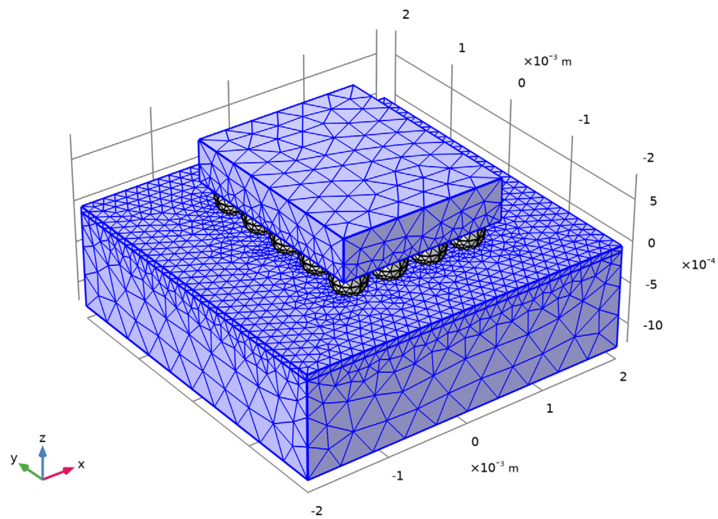
Size 1

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

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

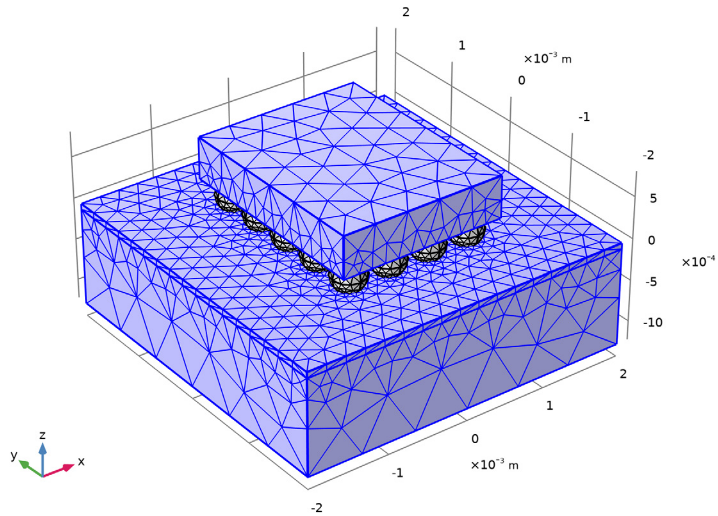
3 From the **Geometric entity level** list, choose **Domain**.

4 Select Domains 1–3, as shown below.



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

6 Click **Build All**.



According to the **Messages** window the mesh now consists of approximately 28,000 elements.

For even fewer elements in the circuit board and electronic component, you can create a swept mesh. This technique sweeps a boundary mesh through the domains to create a structured mesh in the sweep direction.

Free Tetrahedral 1

First you can modify the free tetrahedral mesh operation to apply on the solder joints, then disable the corresponding **Size 1** node, as it is no longer needed.

1 In the **Model Builder** window, click **Free Tetrahedral 1**.

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

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

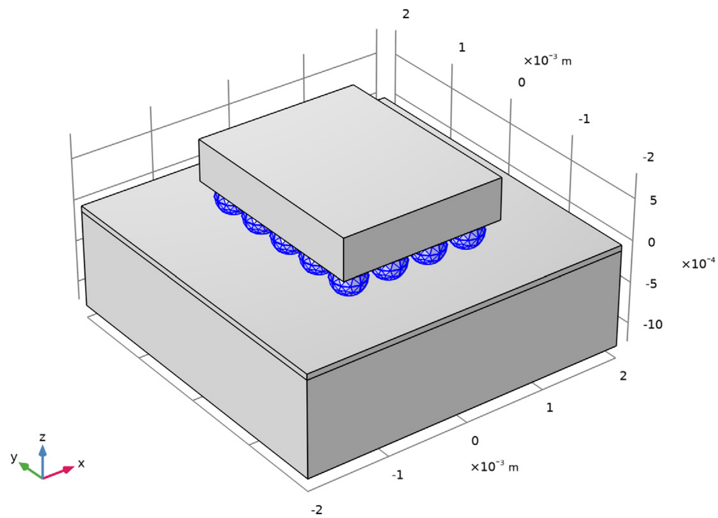
4 Select Domains 4–23, by removing Domains 1–3 from the selection list.

Size 1

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

Free Tetrahedral 1

1 In the **Model Builder** window, right-click **Free Tetrahedral 1** and choose **Build All**.



Only the mesh of the solder joints is built this time.

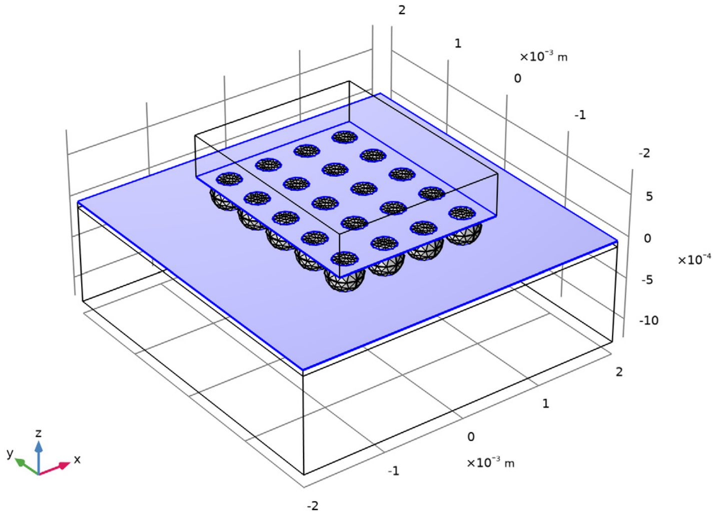
Free Triangular 1

The swept mesher operates on a 3D domain by first meshing a source face, and then sweeping the resulting face mesh through the domain to an opposite destination face. Both the source and destination can consist of several connected faces, as long as each destination face corresponds to at least one source face, and each source face corresponds to exactly one destination face or to a subset of it. All faces that encompass a domain are classified as either source faces, a destination face, or linking faces. The linking faces are the faces connecting the source and destination faces.

1 In the **Mesh** toolbar, click  **Boundary** and choose **Free Triangular**.

Here, the sources for the swept mesh on domains 2 and 3 (the upper part of the circuit board and the electronic component) consist of several faces. Some of these faces are already meshed, as they form the boundary to the solder joints. By meshing faces 7 and

12, highlighted in the figure below, you can complete the source mesh for the mesh sweep operation that will follow.



The **Free Triangular 1** node is added after the **Free Tetrahedral 1** node. COMSOL Multiphysics always inserts new nodes in the meshing sequence after the *current node*. To indicate the current node, it appears with a quadratic frame around its icon. As soon as it is inserted, the **Free Triangular 1** node becomes the current node.

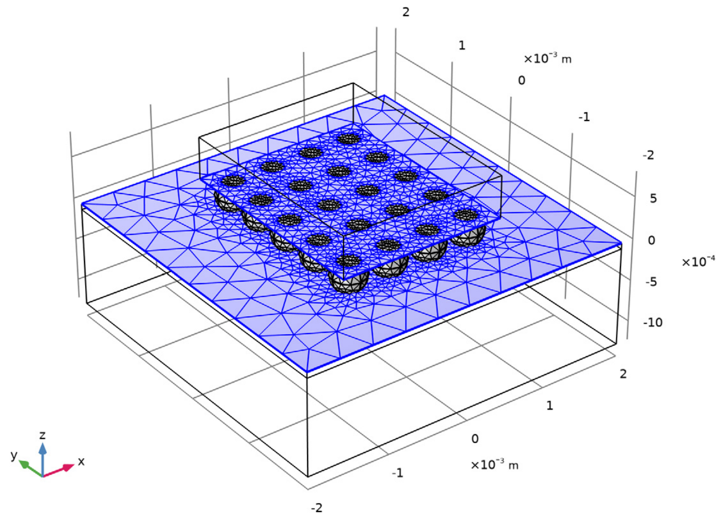
You can easily move nodes in the meshing sequence which will influence the mesh. It could even lead to build errors as an operation can depend on earlier operations in the sequence. Test to move the **Free Triangular 1** operation in the sequence to see how this influences the mesh. If you do, move it back to be after the **Free Tetrahedral 1** node before continuing.

To make selection of the faces easier activate wireframe rendering of the geometry.

2 Click the  **Wireframe Rendering** button in the **Graphics** toolbar.

3 Select Boundaries 7 and 12 only.

4 In the **Settings** window for **Free Triangular**, click **Build All**.



Note that the settings of the first global size attribute are applied by the mesher when creating the triangular mesh. For a coarser triangular mesh add a local size attribute to the free triangular mesh operation.

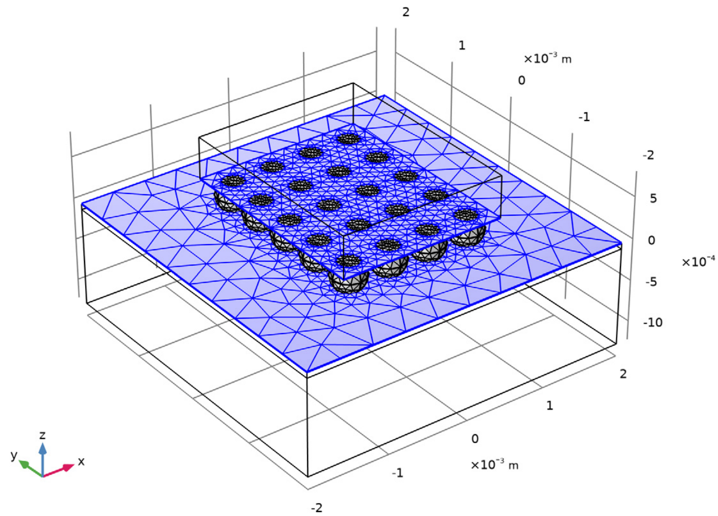
Size 1

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

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

3 From the **Predefined** list, choose **Coarser**.

4 Click **Build All**.



The new triangular mesh looks almost the same as the previous one. The reason is that the local coarser size attribute applies only on the interior of the meshed faces. On the edges the mesher has to respect the already existing mesh on the solder joints, and, on the outer edges the global size settings, which is set to normal mesh size.

To avoid this situation a good practice is to set the first global **Size** setting to the coarsest mesh that you plan to have in the geometry, then specify local size nodes for the mesh operations that need finer mesh.

Size

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

Size 1

Since the global mesh size is now too coarse for the solder joints, enable the **Size 1** attribute under the **Free Tetrahedral 1** node.

1. In the **Model Builder** window, under **Component 1 (comp1)>Mesh 1>Free Tetrahedral 1** right-click **Size 1** and choose **Enable**.
2. In the **Settings** window for **Size**, locate the **Geometric Entity Selection** section.
3. From the **Selection** list, choose **All domains**.

4 Select Domains 4–23, by removing Domains 1–3 from the selection list.

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

Size 1

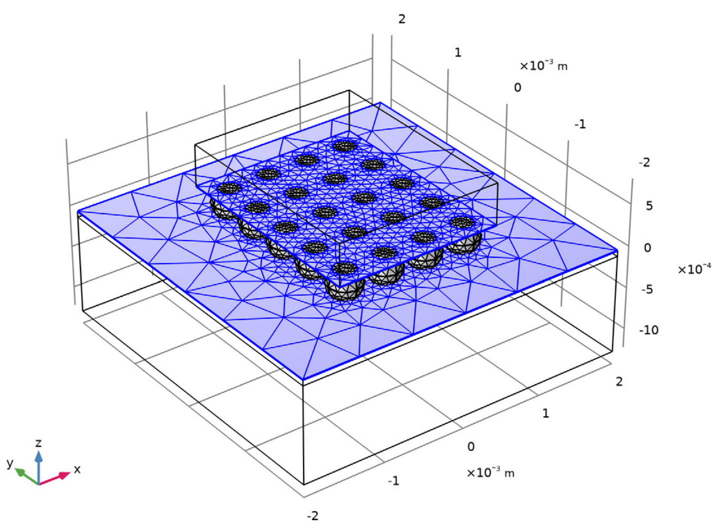
The size attribute for the triangular mesh is no longer needed and can be removed.

In the **Model Builder** window, under **Component 1 (comp1)>Mesh 1>Free Triangular 1** right-click **Size 1** and choose **Delete**.

Free Triangular 1

1 In the **Model Builder** window, under **Component 1 (comp1)>Mesh 1** click **Free Triangular 1**.

2 In the **Settings** window for **Free Triangular**, click **Build All**.



Notice that the triangular mesh is coarser this time.

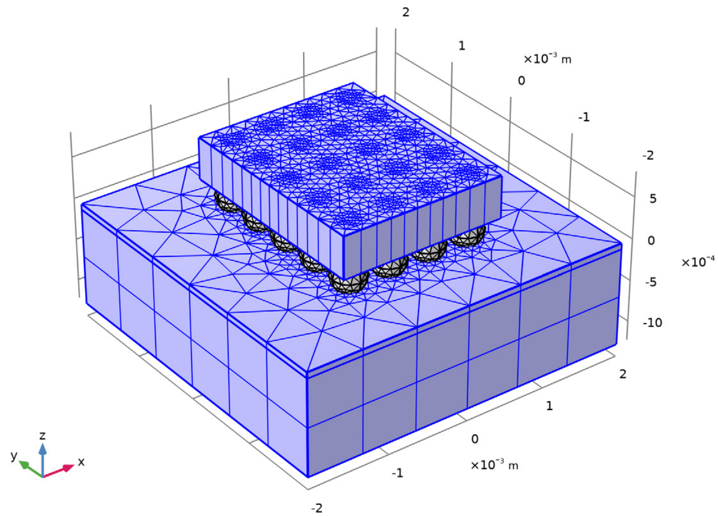
Continue by sweeping the source mesh through the remaining domains.

Swept 1

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

The **Geometric entity level** list is set to **Remaining** by default for new mesh operations. In this case the remaining domains correspond to the domains we would like to sweep mesh.

2 In the **Settings** window for **Swept**, click **Build All**.

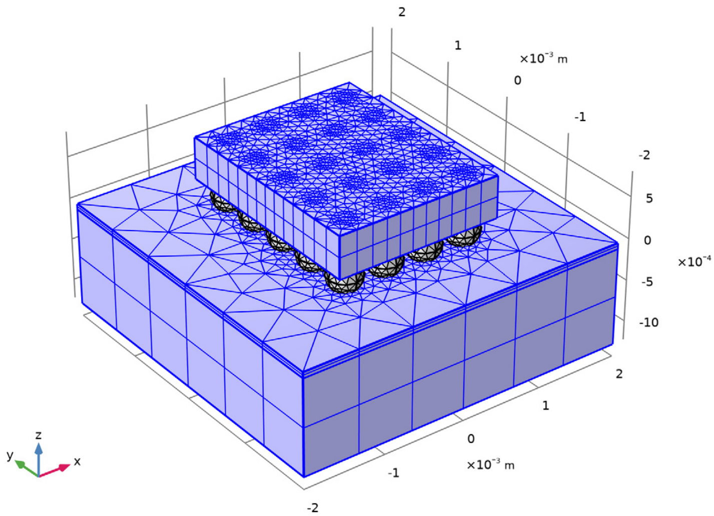


The mesh now consists of approximately 17,000 elements. The swept mesher used the Coarser predefined mesh size to determine the number of elements along the sweep direction. For precise control of the number of elements, specify a distribution for the swept mesher.

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 2.

4 Click **Build All**.



This latest mesh consists of approximately 21,000 elements, keeping the higher resolution for the domains that are important for the analysis, while providing a less dense structured mesh for the remaining domains.