Model created in COMSOL Multiphysics 6.4

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.

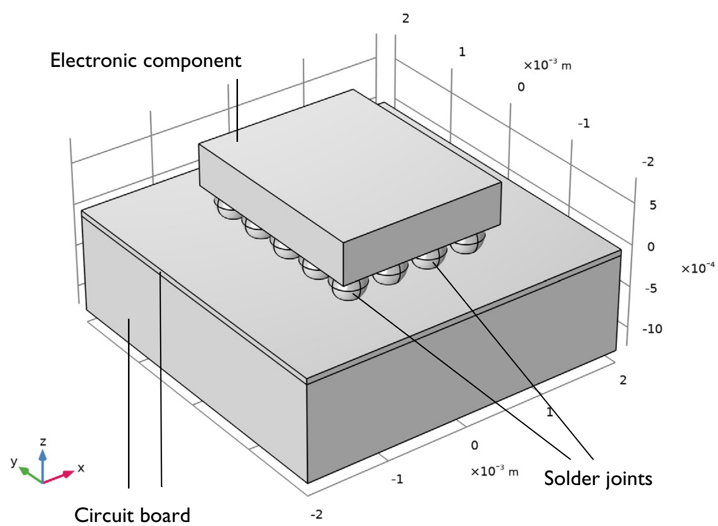


Figure 1: Geometry consisting of an electronic component mounted on a circuit board.

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 Viscoplasticity in the Nonlinear Structural Materials Module Application Library. 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.

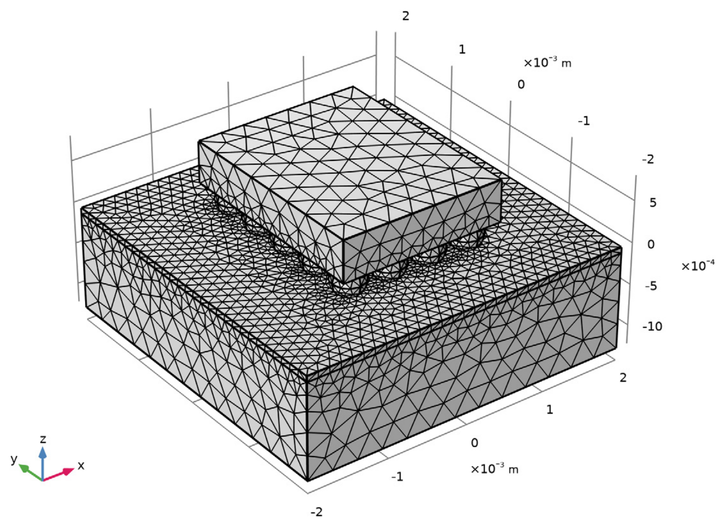
MESH 1

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.

In the table under the Mesh 1 settings window, Geometric Analysis, Detail Size appears in the Contributor column. The checkbox is selected by default, meaning the geometric analysis contributes to the physics-controlled mesh. The geometry is analyzed before meshing, and the mesh element size parameters take into account small geometric details such as short edges or small faces.

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



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. The Information section in the Settings window of the Mesh node as well as the Messages window display the total number of mesh elements after the last build.

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. Additionally, the Size 1 node is added to the sequence as the contributor Geometric Analysis, Detail size is turned on.

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

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.

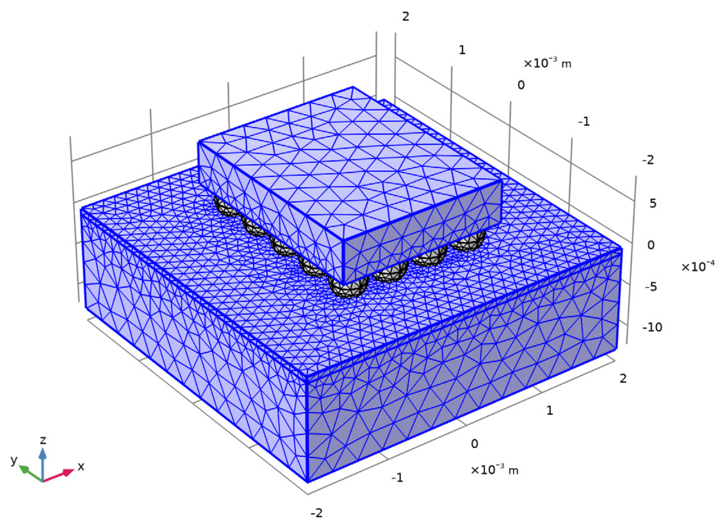
Size 1

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.

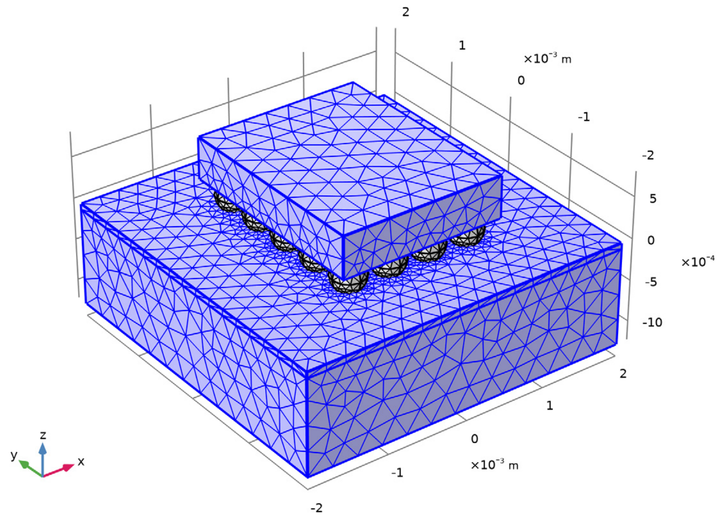
3 From the **Geometric entity level** list, choose **Domain**.

4 Select Domains 1–3, as shown below.



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

6 Click **Build All**.



This reduced the total number of 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**.

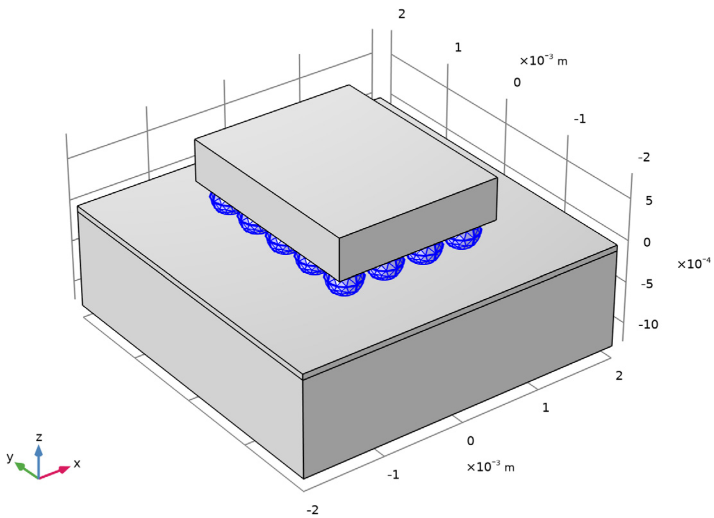
4 Select Domains 4–23, by removing Domains 1–3 from the selection list.

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.

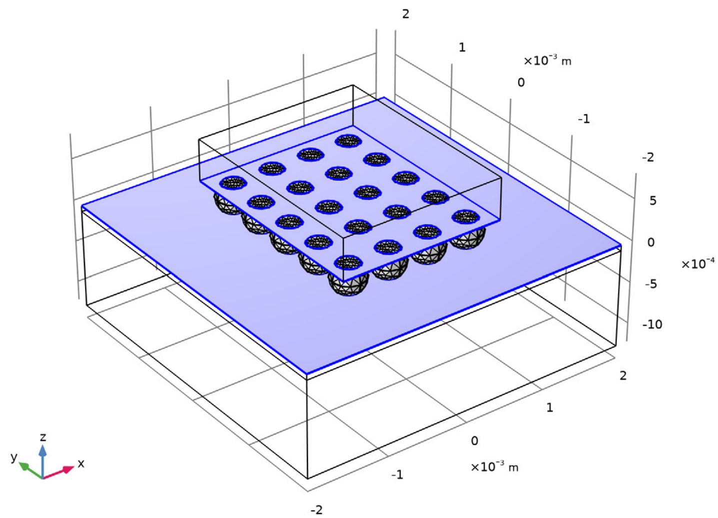
Free Triangular 1

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.

To have more control over the mesh on the source faces, add a Free Triangular operation and premesh the boundaries that will be used as sources.

1 In the **Mesh** toolbar, click **More Generators** 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 boundaries 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.

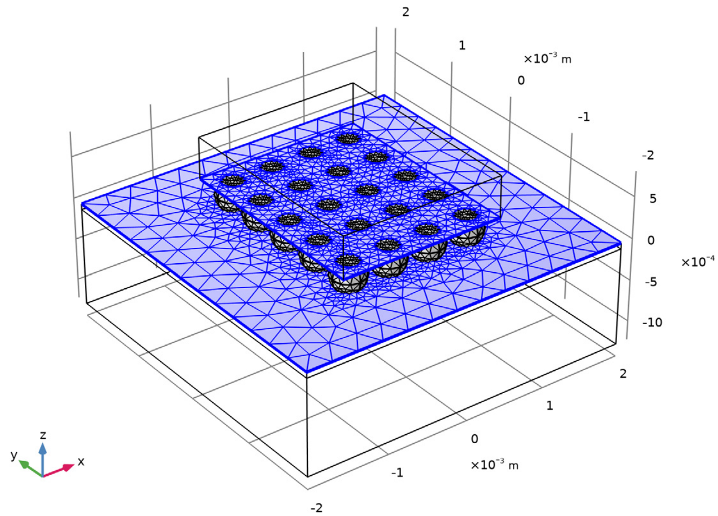
You can easily move nodes in the meshing sequence which will influence the mesh. It could even lead to build errors as an operation can depend on earlier operations in the sequence. Test to move the Free Triangular 1 operation in the sequence to see how this influences the mesh. If you do, move it back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

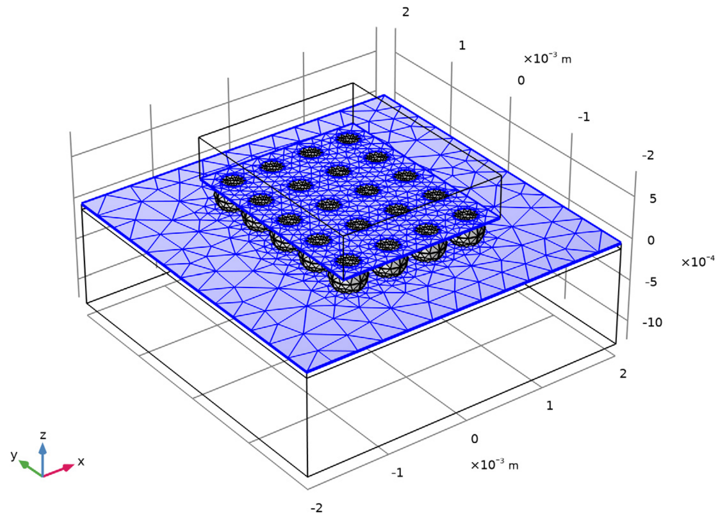
Size 1

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 Click the **Predefined** button.

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

However, the minimum element size is still constrained by the global Size 1 node, so create a boundary selection for the spheres and use this for the selection of Size 1.

GEOMETRY 1

Sphere 1 (sph1)

1. In the **Model Builder** window, under **Component 1 (comp1)** > **Geometry 1** click **Sphere 1 (sph1)**.
2. In the **Settings** window for **Sphere**, locate the **Selections of Resulting Entities** section.
3. Select the **Resulting objects selection** checkbox.
4. From the **Show in physics** list, choose **Boundary selection**.

MESH 1

Size 1

1. In the **Model Builder** window, under **Component 1 (comp1)** > **Mesh 1** click **Size 1**.
2. In the **Settings** window for **Size**, locate the **Geometric Entity Selection** section.
3. From the **Selection** list, choose **Sphere 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.

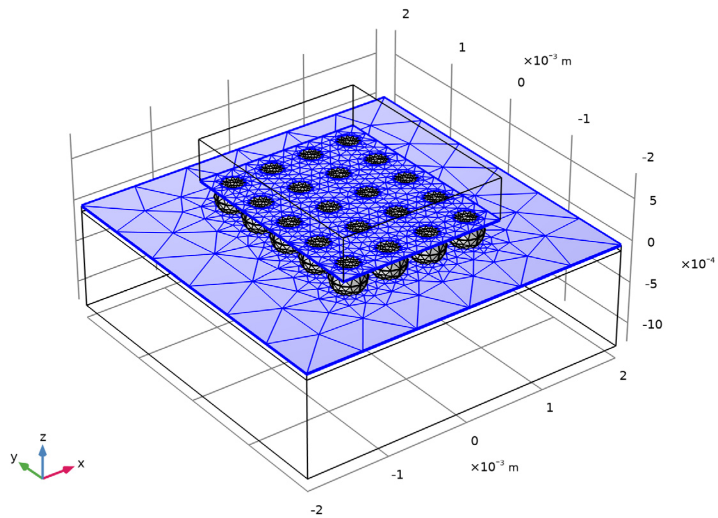
1. In the **Model Builder** window, under **Component 1 (comp1)** > **Mesh 1** > **Free Tetrahedral 1** right-click **Size 1** and choose **Enable**.
2. In the **Settings** window for **Size**, locate the **Geometric Entity Selection** section.
3. From the **Selection** list, choose **All domains**.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

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 **Settings** window for **Free Triangular**, click **Build All**.



Notice that the triangular mesh is coarser this time.

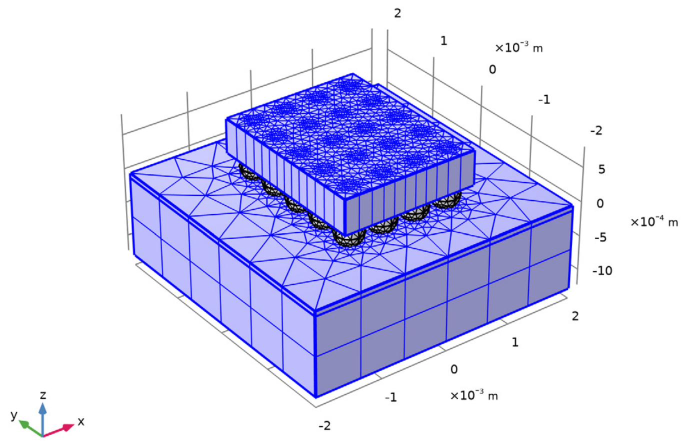
Continue by sweeping the source mesh through the remaining domains.

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. The premeshed faces will be selected as source faces automatically.

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



By looking in the Messages window, we can conclude that this has reduced the number of elements a lot. 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.

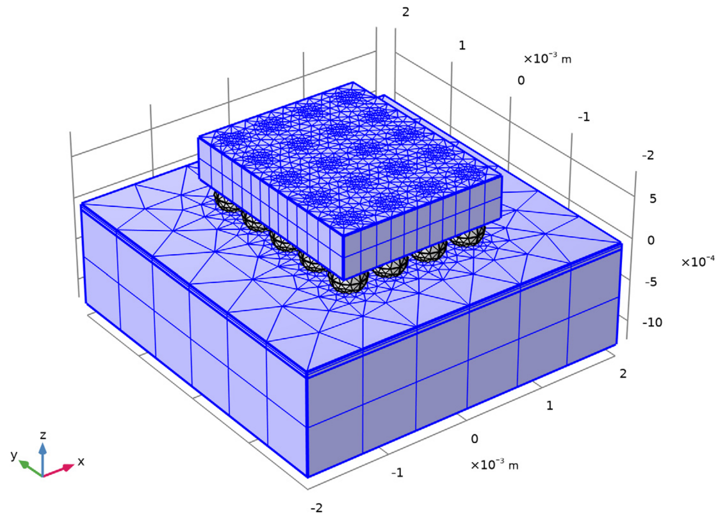
Distribution 1

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 keeps the higher resolution for the domains that are important for the analysis, while providing a mesh that is structured in one direction and that is less dense for the remaining domains.