Created in COMSOL Multiphysics 5.6

Importing and Meshing a PCB Geometry from an ODB++ Archive[1]

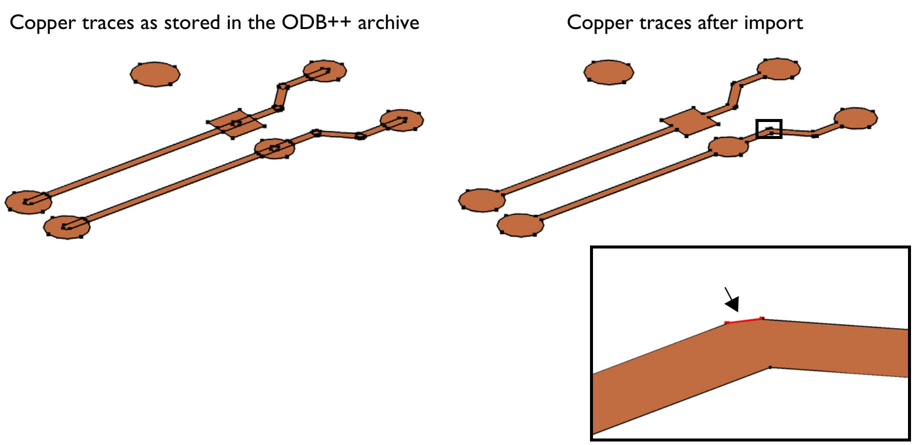
1. The file is provided courtesy by Hypertherm, Inc., Hanover, NH, USA.

Introduction

This tutorial demonstrates how to import data from an ODB++ archive to generate the geometry for a printed circuit board (PCB) that is suitable for simulation assuming that the copper traces can be modeled as shell geometry. Such a simulation is, for example, a heat transfer in solids analysis where the difference in thermal conductivity between the copper and the surrounding FR4 dielectric is so large that we can neglect the temperature difference and heat flux across the copper layer. The tutorial further demonstrates how to remove small details such as short edges from the generated geometry, and how to generate a mesh with adjusted element size parameters to resolve the small features of the copper traces.

COPPER LAYERS

The copper layers in an ODB++ archive contain geometric shapes that when united during import result in the final geometric objects for the copper traces. The image to the left in the figure below depicts shapes from the ODB++ archive used in this tutorial. The copper pads are represented by circles and rectangles and the lines by the elongated rectangles with fully rounded short edges.

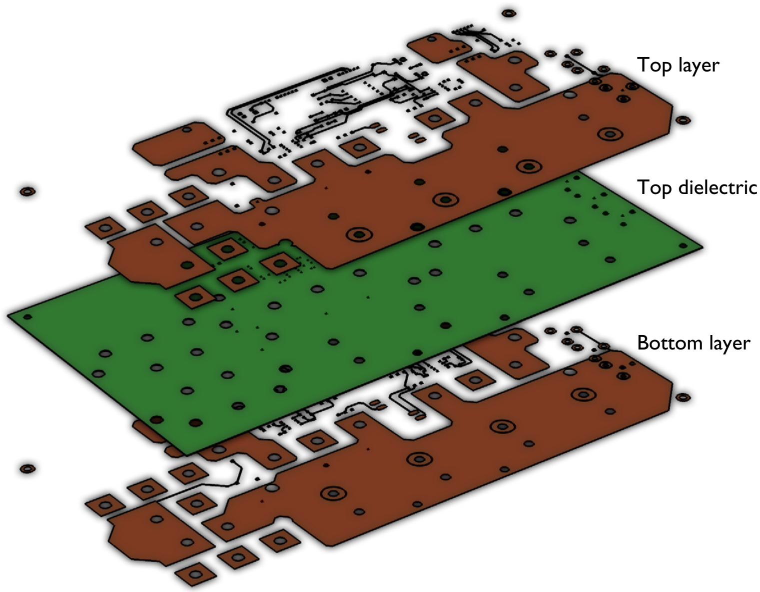


During import of the copper layers the individual shapes are united and the internal edges are removed to reduce the complexity of the geometry. The final geometry objects for the copper traces are shown to the right in the figure above. The short edge, which is highlighted above in the magnified section of the copper trace, is a result of two overlapping copper lines.

Edges corresponding to such overlaps can become significantly shorter than the typical line width of the copper, and can be automatically detected and removed before meshing the geometry.

Model Definition

The ODB++ archive imported in this tutorial contains a PCB with two copper layers connected by vias.



During import, geometry objects are generated for the copper layers (surface objects), the dielectric layer (solid object) and vias (solid objects). The solid object for the dielectric layer is created according to the shape of the board as defined in the ODB++ archive. The vias are generated as solid cylinders. To get the geometry for simulation, the objects for the dielectric material are united with the copper surface objects, and the cylinders for the vias are subtracted from the result.

The ODB++ archive also contains some disconnected copper surfaces that are located outside the PCB, and need to be deleted after the import.

Even though the PCB consists of many geometric entities, the geometry can be set up very quickly by leveraging the layer selections generated by the import and the available selection operations. In addition to generating the geometry, the step-by-step instructions also result in domain and boundary selections that are available as input for material and physics settings.

The last part of the instructions cover meshing of the PCB geometry to demonstrate how to find and remove details such as short edges before meshing, and how to adjust the maximum and minimum element size parameters to resolve the small details in the copper layers.

Application Library path: `ECAD_Import_Module/Tutorials/pcb_import`

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

In the Settings window for **Geometry**, locate the **Advanced** section. From the **Geometry representation** list, if it is visible, choose the **COMSOL kernel**.

Import 1 (imp1)

1. In the **Home** toolbar, click **Import**.
2. In the **Settings** window for **Import**, locate the **Import** section.
3. Click **Browse**.
4. Browse to the model's Application Libraries folder and double-click the file `printed_circuit_board_si_geom.zip`.

5 Find the **Layers to import** subsection. In the table, select to import only the following layers:

Name	Type	Thickness(in)	Import
`TOP`	Metal	`1.3403[mil]`	√
`TOP.DIEL`	Dielectric	`12.6[mil]`	√
`BOTTOM`	Metal	`1.3403[mil]`	√
`DRILL`	Drill	`12.6[mil]`	√

6 From the **Type of import** list, choose **Metal shell**.

Using this option the copper traces are generated as surface objects with zero thickness.

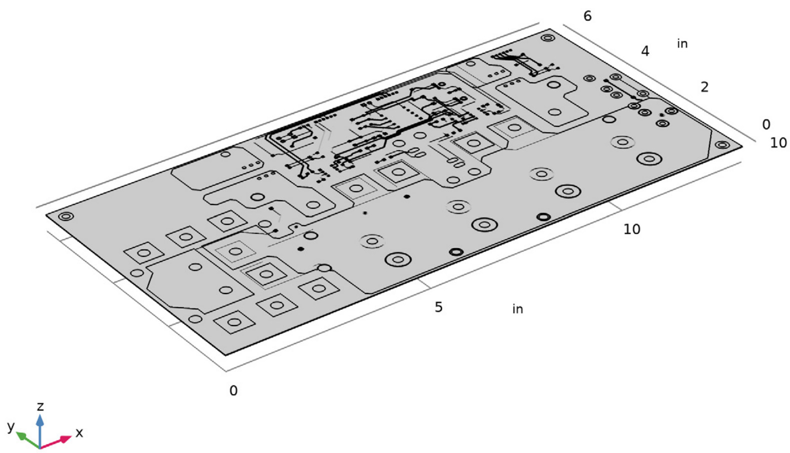
Copper layers in PCB files often include text and other shapes that are positioned outside the boundaries of the board. Select the **Ignore objects outside of board** check box to automatically exclude the text boxes that are located outside the outline of this PCB.

7 Select the **Ignore objects outside of board** check box.

Note that in the **Selections of Resulting Entities** section the **Layer selections** check box is already selected, and the **Show in physics** list is set to **All levels**. The selections generated for each imported layer are available for downstream geometry operations, and also for any applicable physics, mesh or other model setting. The following steps illustrate how you can use the layer selections to generate the final geometry in an efficient way.

8 Click **Import**.

As the geometry import completes note the information displayed in the **Messages** window. In total four objects are imported: one solid object each for the dielectric and the vias, and one surface object each for the copper layers.



Due to the high aspect ratio of the imported geometry it is not possible to see the vias that are inside the dielectric. To make this possible create a second view.

DEFINITIONS

View no scaling

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

2 In the **Settings** window for **View**, type `View no scaling` in the **Label** text field.

View automatic scaling

1 Right-click **View no scaling** and choose **Duplicate**.

2 In the **Model Builder** window, click **View no scaling 1**.

3 In the **Settings** window for **View**, type `View automatic scaling` in the **Label** text field.

Camera

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

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

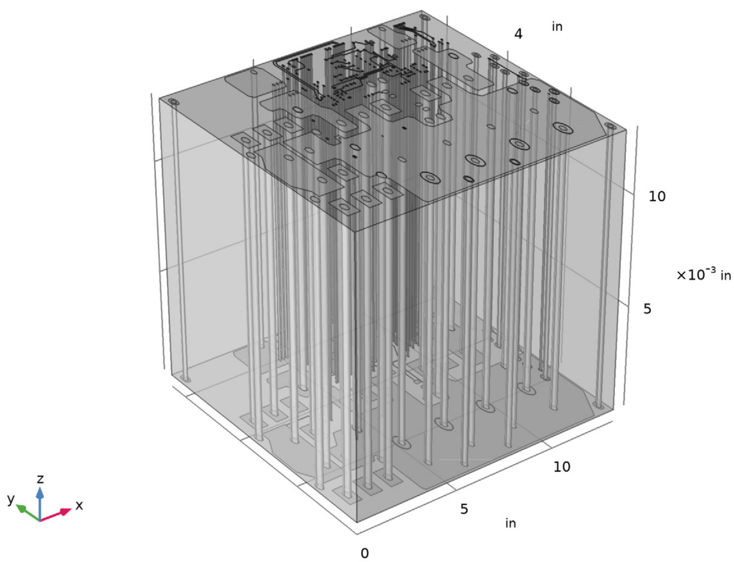
3 From the **View scale** list, choose **Automatic**.

4 Click **Update**.

5 Click the **Go to Default View** button in the **Graphics** toolbar.

6 Click the **Transparency** button in the **Graphics** toolbar.

The geometry should now fill the **Graphics** window, just as it is displayed in the figure below.



GEOMETRY 1

PCB Domain

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

You can now configure this union selection to contain the objects for the dielectric and copper layers, then set it to be visible for physics features as a domain selection.

2 In the **Settings** window for **Union Selection**, type `PCB Domain` in the **Label** text field.

3 Locate the **Geometric Entity Level** section. From the **Level** list, choose **Object**.

4 Locate the **Input Entities** section. Click **Add**.

5 In the **Add** dialog box, in the **Selections to add** list, choose **TOP (Import 1)**, **BOTTOM (Import 1)**, and **TOP.DIEL (Import 1)**.

It is important that this selection includes all dielectric and copper objects as you will use it further ahead as the input for the union operation that combines the objects of the geometry.

6 Click **OK**.

7 In the **Settings** window for **Union Selection**, locate the **Resulting Selection** section.

8 From the **Show in physics** list, choose **Domain selection**.

9 Click **Build Selected**.

Copper Boundaries

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

This second union selection gathers all copper surfaces into one selection that you can use when assigning the material properties.

2 In the **Settings** window for **Union Selection**, type `Copper Boundaries` in the **Label** text field.

3 Locate the **Geometric Entity Level** section. From the **Level** list, choose **Boundary**.

4 Locate the **Input Entities** section. Click **Add**.

5 In the **Add** dialog box, in the **Selections to add** list, choose **TOP (Import 1)**, **BOTTOM (Import 1)**, and **DRILL.TOP.DIEL (Import 1)**.

6 Click **OK**.

7 In the **Settings** window for **Union Selection**, click **Build Selected**.

Union 1 (uni1)

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

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

3 From the **Input objects** list, choose **PCB Domain**.

4 Click **Build Selected**.

Difference 1 (dif1)

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

Using this difference operation you can subtract the cylinders of the vias from the board geometry to get the final geometry where the vias are represented only by the vertical faces of the holes.

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

3 From the **Objects to add** list, choose **PCB Domain**.

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

5 From the **Objects to subtract** list, choose **DRILL.TOP.DIEL (Import 1)**.

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

Now that the geometry is complete you can assign materials for the dielectric domain and the copper faces using the selections defined in the geometry sequence.

ADD MATERIAL

1 In the **Home** toolbar, click **Add Material** to open the **Add Material** window.

2 Go to the **Add Material** window.

3 In the tree, select **Built-in>FR4 (Circuit Board)**.

4 Click **Add to Component 1 (comp1)**.

5 In the tree, select **Built-in>Copper**.

6 Click **Add to Component 1 (comp1)**.

7 In the **Home** toolbar, click **Add Material** to close the **Add Material** window.

MATERIALS

FR4 (Circuit Board) (mat1)

1 In the **Settings** window for **Material**, locate the **Geometric Entity Selection** section.

2 From the **Selection** list, choose **PCB Domain**.

Copper (mat2)

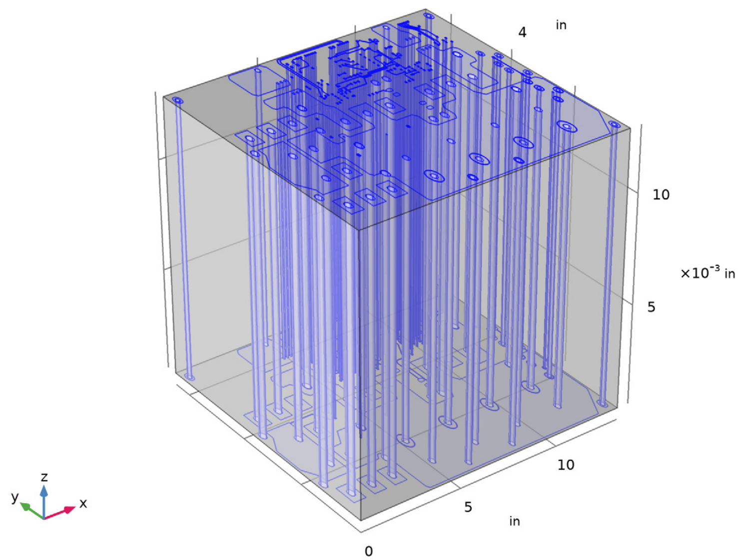
1 In the **Model Builder** window, click **Copper (mat2)**.

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

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

4 From the **Selection** list, choose **Copper Boundaries**.

As expected the material is assigned to the top and bottom copper traces and the vertical walls of the vias.

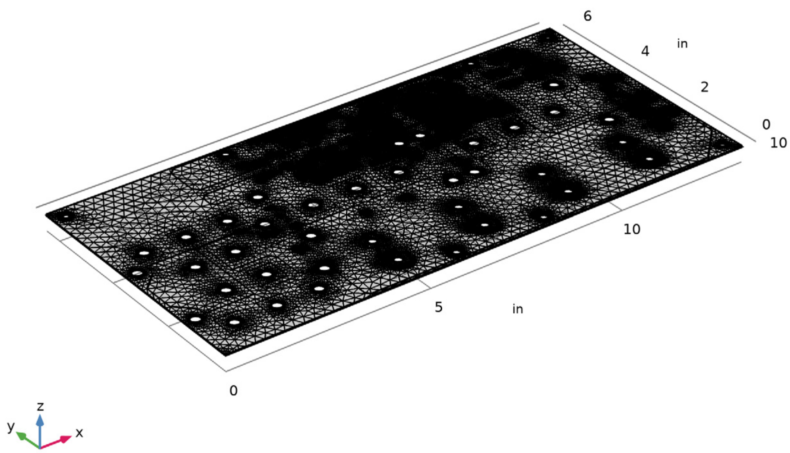


5 Click the **Go to View No Scaling** button in the **Graphics** toolbar.

MESH 1

1 In the **Home** toolbar, click **Build Mesh**.

Since no physics interface has been added in the model the mesh is generated with the default general settings. When physics interfaces are present the mesh settings can depend on the physics settings, and the resulting mesh can be different from the one generated here.



According to the information in the **Messages** window, the mesher generated a mesh of approximately 130000 domain elements, and also resulted in four warning nodes. Proceed with reviewing these warnings to get an insight into the various issues associated with meshing this PCB geometry.

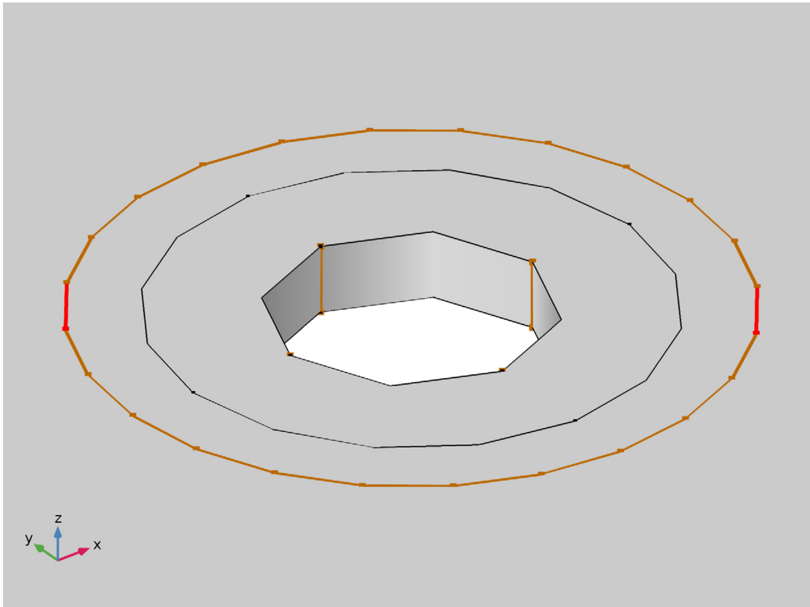
The first warning node lists those edges that are much shorter than the minimum element size used by the mesher.

Warning 1

1 In the **Model Builder** window, expand the **Mesh 1** node, then click **Warning 1**.
2 In the **Settings** window for **Warning**, locate the **Geometric Entity Selection** section.
3 In the list, select **317**.
4 Click  **Zoom to Selection**.

To get a better view of the selected edge first turn off mesh rendering, then use the mouse to zoom out until you get the view displayed in the figure below.

5 Click the **Mesh Rendering** button in the **Graphics** toolbar.



The highlighted edge is one of the segments of a polygon that represents a circular edge in the top copper layer. Such polygons can occur often in PCB data, and are one of the sources for short edges in the generated geometry.

6 In the **Mesh** toolbar, click **Measure**.

The length of the edge, 0.015 inches, is displayed in the **Messages** window. This length is similar to the width of the copper traces.

It is usually good practice to measure the length of various short edges to get an understanding of the feature size in the geometry.

7 Click the **Go to Default View** button in the **Graphics** toolbar.

Warning 2

This warning node contains a list of faces that are identified as much smaller, or as having narrow regions that are much smaller, than the minimum element size parameter for the mesh.

Warning 3

The message in this warning node warns that the domain is too thin compared to the minimum element size parameter.

Warning 4

In the last warning node you can see that the mesh contains some very low quality elements. This is due to the issues listed in the previous warning nodes.

While all these warnings can be overwhelming, they can be expected when meshing with default mesh settings for general physics a PCB geometry such as the one in this tutorial. The minimum and maximum element sizes for the mesh are determined based on the largest dimensions of the geometry, and produce too large elements to resolve the small details in the geometry. The PCB is a few inches in the x- and y-directions, but has a thickness of only a few thousands of an inch, which is similar to the size of the small details found on the copper layers.

To get rid of the warnings and generate a mesh with higher quality you can do two things: remove small details that are not needed, and allow the mesher to generate smaller elements to better resolve the small details that are needed for the geometry.

GEOMETRY 1

Remove Details 1 (rmd1)

1 In the **Geometry** toolbar, click **Remove Details**.

   Using this operation you can automatically find and remove small details from the geometry.

   Configure the operation to remove short edges and points that are adjacent to continuous edges only. Other details, such as thin domains, small and sliver faces could be actual features of the copper layers and should not be removed.

2 In the **Settings** window for **Remove Details**, locate the **Details to Remove** section.

3 Clear the **Small faces** check box.

4 Clear the **Sliver faces** check box.

5 Clear the **Narrow face regions** check box.

6 Clear the **Thin domains** check box.

7 Locate the **Parameters** section. From the **Detail size** list, choose **Absolute**.

8 In the **Maximum absolute size** text field, type `0.008[in]`.

   Edges that are shorter than this length are going to be collapsed into a point by the remove details operation. The number you enter here should be smaller than the size of any details that you would like to keep in the geometry.

9 In the **Continuous tangent tolerance** text field, type `16[deg]`.

By increasing the default angular tolerance you ensure that the vertices of the polygons illustrated earlier are going to be removed.

10 Click **Build Selected**.

After the operation is completed you can see under the **Information** section of the **Settings** window that 2299 vertices and 3 edges were removed. To do this the automatic removal tool used two types of operations, one that ignores vertices and another for collapsing the edges. These operations are added under the **Remove Details 1** node in the geometry sequence, and they are hidden in automatic mode.

11 Right-click **Remove Details 1 (rmd1)** and choose **Edit Generated Sequence**.

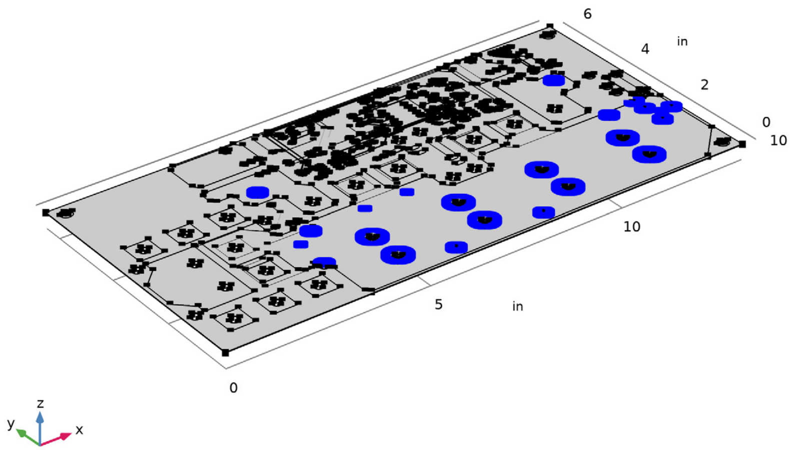
As the individual operations become visible note that the sequence contains six ignore vertices operations. These correspond to the number of passes made by the automatic removal when increasing the angular tolerance for removing vertices, and then making a final pass to remove vertices after collapsing edges. In manual mode you can modify each operation, for example to exclude entities that you would like to be left in the geometry.

Ignore Vertices 4 (aigv4)

1 In the **Model Builder** window, click **Ignore Vertices 4 (aigv4)**.

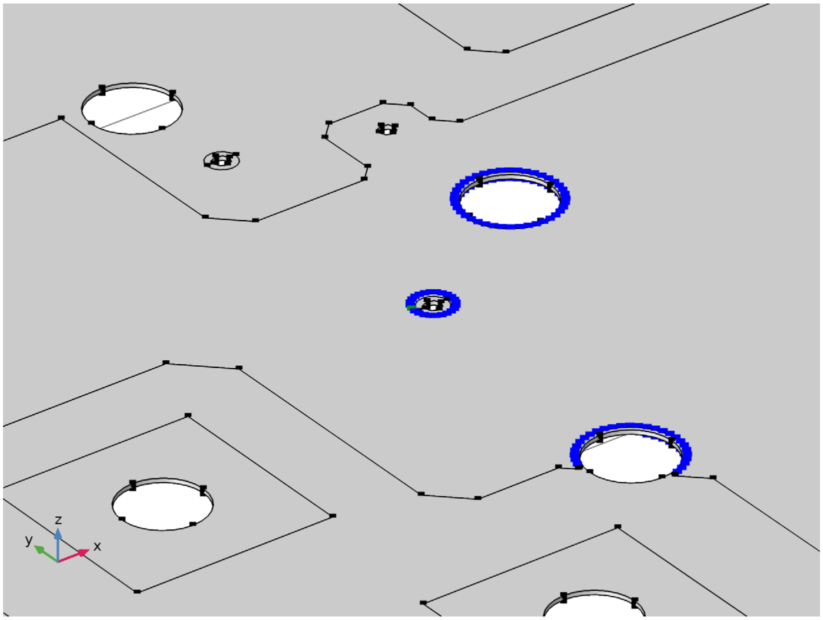
2 In the **Settings** window for **Ignore Vertices**, locate the **Input** section.

3 Find the **Vertices to ignore** subsection. Select the  **Activate Selection** toggle button.



4 In the list, select **237**.

5 Click **Zoom to Selection**.



The selected vertex is between two segments of the polygon that you have examined earlier in one of the warnings from the mesher.

6 Click the **Go to Default View** button in the **Graphics** toolbar.

Switch back to automatic mode for the remove details operation, and then continue with generating a new mesh.

Remove Details 1 (rmd1)

1. In the **Model Builder** window, click **Remove Details 1 (rmd1)**.
2. In the **Settings** window for **Remove Details**, locate the **Automation** section.
3. From the **Mode of operation** list, choose **Automatic**.
4. Click **Build Selected**.

MESH 1

To manually edit the element size settings for the mesh switch to user-controlled mesh.

1. In the **Model Builder** window, under **Component 1 (comp1)** click **Mesh 1**.
2. In the **Settings** window for **Mesh**, locate the **Mesh Settings** section.
3. From the **Sequence type** list, choose **User-controlled 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 **Custom** button.

4 Locate the **Element Size Parameters** section. In the **Maximum element size** text field, type `0.7[in]`.

5 In the **Minimum element size** text field, type `0.07[in]`.

The first **Size** node in the meshing sequence is a global size node that applies to all downstream mesh operations. The maximum and minimum element sizes you specify here are much closer than the default parameters to the size of small details in the PCB geometry.

Size 1

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

By adding a local size node that applies only to the faces on the top and bottom of the PCB you ensure that the copper traces and surrounding faces are meshed with the appropriate detail.

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

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

4 From the **Selection** list, choose **TOP.DIEL (Import 1)**.

This automatically generated selection contains all horizontal boundaries adjacent to the dielectric.

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

6 Locate the **Element Size Parameters** section. Select the **Maximum element size** check box.

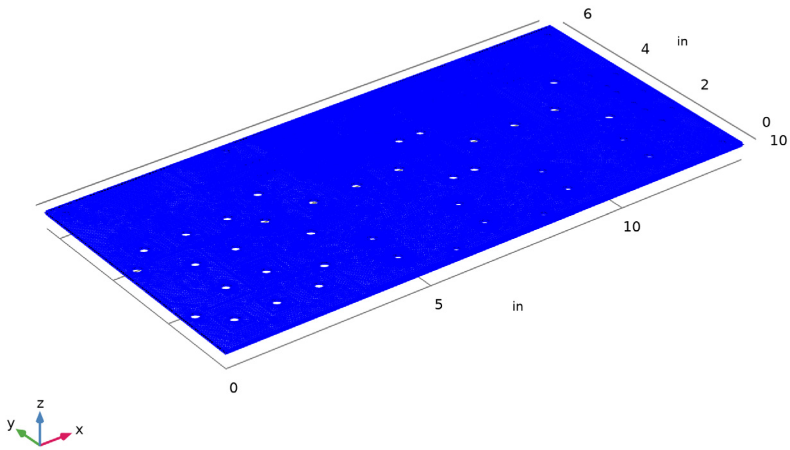
7 In the associated text field, type `0.1[in]`.

8 Select the **Minimum element size** check box.

9 In the associated text field, type `0.01[in]`.

10 Click **Build All**.

11 Click the **Mesh Rendering** button in the **Graphics** toolbar.



This final mesh has approximately 265,000 domain elements. Warnings are no longer displayed by the mesher as the applied maximum and minimum element size parameters are now much closer to the size of the small details of the geometry.