

# INTRODUCTION TO ECAD Import Module

# Introduction to the ECAD Import Module

© 1998–2025 COMSOL

Protected by patents listed on [www.comsol.com/patents](http://www.comsol.com/patents), or see Help > About COMSOL Multiphysics on the File menu in the COMSOL Desktop for less detailed lists of U.S. Patents that may apply. Patents pending.

This Documentation and the Programs described herein are furnished under the COMSOL Software License Agreement ([www.comsol.com/sla](http://www.comsol.com/sla)) and may be used or copied only under the terms of the license agreement.

Support for implementation of the ODB++ Format was provided by Mentor Graphics Corporation pursuant to the ODB++ Solutions Development Partnership General Terms and Conditions. ODB++ is a trademark of Mentor Graphics Corporation.

COMSOL, the COMSOL logo, COMSOL Multiphysics, COMSOL Desktop, COMSOL Compiler, COMSOL Server, and LiveLink are either registered trademarks or trademarks of COMSOL AB. All other trademarks are the property of their respective owners, and COMSOL AB and its subsidiaries and products are not affiliated with, endorsed by, sponsored by, or supported by those trademark owners. For a list of such trademark owners, see [www.comsol.com/trademarks](http://www.comsol.com/trademarks).

Version: COMSOL 6.4

## Contact Information

Visit the Contact COMSOL page at [www.comsol.com/contact](http://www.comsol.com/contact) to submit general inquiries or search for an address and phone number. You can also visit the Worldwide Sales Offices page at [www.comsol.com/contact/offices](http://www.comsol.com/contact/offices) for address and contact information.

If you need to contact Support, an online request form is located on the COMSOL Access page at [www.comsol.com/support/case](http://www.comsol.com/support/case). Useful links:

- Support Center: [www.comsol.com/support](http://www.comsol.com/support)
- Product Download: [www.comsol.com/product-download](http://www.comsol.com/product-download)
- Product Updates: [www.comsol.com/product-update](http://www.comsol.com/product-update)
- COMSOL Blog: [www.comsol.com/blogs](http://www.comsol.com/blogs)
- Discussion Forum: [www.comsol.com/forum](http://www.comsol.com/forum)
- Events: [www.comsol.com/events](http://www.comsol.com/events)
- COMSOL Video Gallery: [www.comsol.com/videos](http://www.comsol.com/videos)
- Support Knowledge Base: [www.comsol.com/support/knowledgebase](http://www.comsol.com/support/knowledgebase)
- Learning Center: [www.comsol.com/support/learning-center](http://www.comsol.com/support/learning-center)

Part number: CM023602

# Contents

---

Introduction .....	5
Importing an ODB++ Archive.....	6



## Introduction

---

The ECAD Import Module expands the capabilities of COMSOL Multiphysics with the construction of 3D geometry from popular ECAD layout formats, namely GDS II, IPC-2581, and ODB++. You can configure the provided import tool with selective import of layers and nets from the files. The module enables you to store and import layer information from text files, and allows for faster model set up by automatically generating selections that you can use during the modeling process.

Read the detailed step-by-step example that follows to learn how to use the provided functionality.

## Importing an ODB++ Archive


---

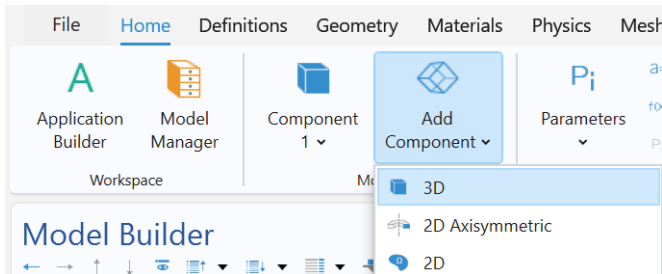
The ODB++ file format can contain most of the information needed to manufacture a PCB, including various types of layers such as footprint layers to draw outlines of components on the PCB board. You can configure the import to use selected layers for construction of 3D geometry objects. By adding multiple Import nodes to a geometry sequence of your model, where each node imports different layers from the same file, you can create a complete 3D model of a device. This process is illustrated in the following example where you can import the layout of a PCB contained in an ODB++ archive. The file contains layers for metal traces, for dielectric regions, and different drill holes.

The ODB++ file is provided courtesy of Hypertherm, Inc., Hanover, NH, USA.

## New Window


---

- 1 In the New window click Blank Model to skip the steps of selecting physics and study type.
- 2 In the Home toolbar, click Add Component  and choose 3D.



## Import

---


- 1 In the Home toolbar, click Import .
- 2 In the Settings window for Import click the Browse button.
- 3 In your COMSOL installation directory navigate to the folder `applications/ECAD_Import_Module/Tutorials` and double click the file `printed_circuit_board_si_geom.zip`.

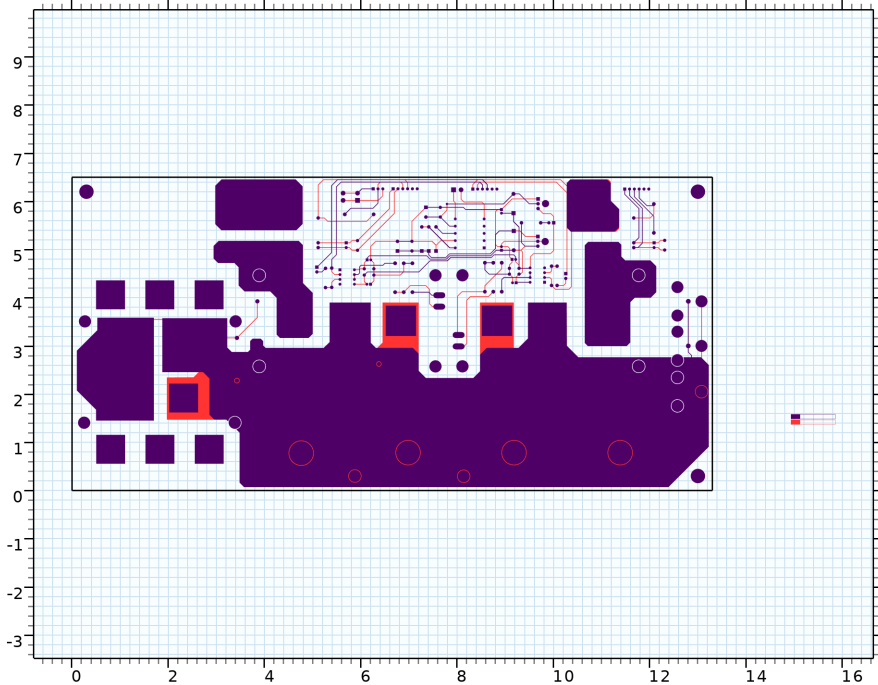
The Settings window for Import will now show the appropriate settings for the file, including the layers in the table under Layers section. As you can see from

the table only the metal and dielectric layers are selected by default, the import of all other types of layers is turned off.

Name	Type	Thickness (in)	Import
COMP+_TOP	Other	0	<input type="checkbox"/>
TOPOVERLAY	Other	0	<input type="checkbox"/>
TOPSOLDER	Other	0	<input type="checkbox"/>
TOP	Metal	1.3328092637424853[mil]	<input checked="" type="checkbox"/>
TOP.DIEL	Dielectric	12.6[mil]	<input checked="" type="checkbox"/>
BOTTOM	Metal	1.3328092637424853[mil]	<input checked="" type="checkbox"/>
BOTTOM.DIEL	Dielectric	0.4[mil]	<input checked="" type="checkbox"/>
BOTTOMSOLDER	Other	0	<input type="checkbox"/>
BOTTOMOVERLAY	Other	0	<input type="checkbox"/>
DRILL	Drill	15.265618527484971[mil]	<input type="checkbox"/>
MECHANICAL1	Other	0	<input type="checkbox"/>
MECHANICAL3	Other	0	<input type="checkbox"/>
MECHANICAL26	Other	0	<input type="checkbox"/>

**Note:** For very large board layouts it is recommended to import one metal layer at a time to check the geometry of each layout in the PCB file. To avoid reimporting layers (this may take time for large layout files) when activating new layers in the table, you can add several import nodes to the geometry sequence, with each import node configured for a different layer.

- 4 Before importing you can open a preview of the selected PCB file in the graphics window and inspect the board before importing it. In the Settings window toolbar, click Preview .

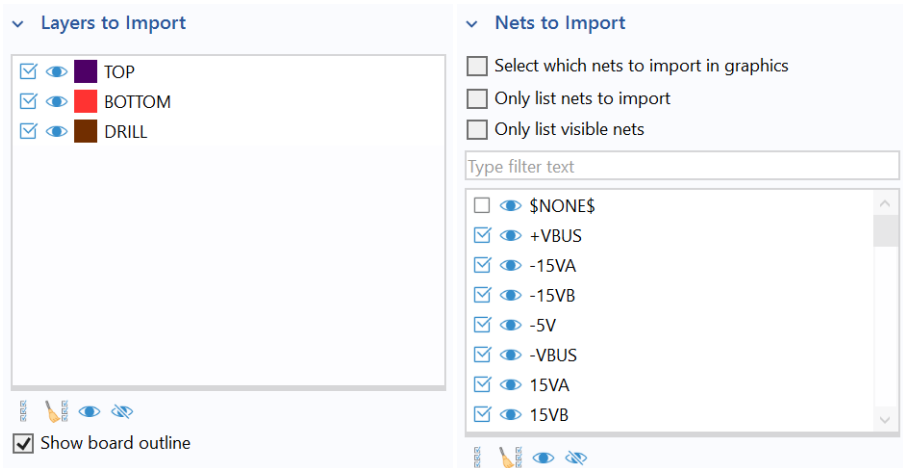


The preview rendering consists of 2D representation of metal and drill layers, organized by layer and net, and colored according to the layer.

The Preview settings opened on the right side of the Graphics window consists of two sections. In the section Layers to Import, you can choose which layers to import and to display. In the section Nets to Import, you control which nets to import and to display. A specific part of the PCB will only be imported if both its layer and net are selected for import, with the analogous conditions applying for it to be displayed in the preview.

- 5 Select the checkbox for DRILL in the Layers to Import table in the Preview settings window.

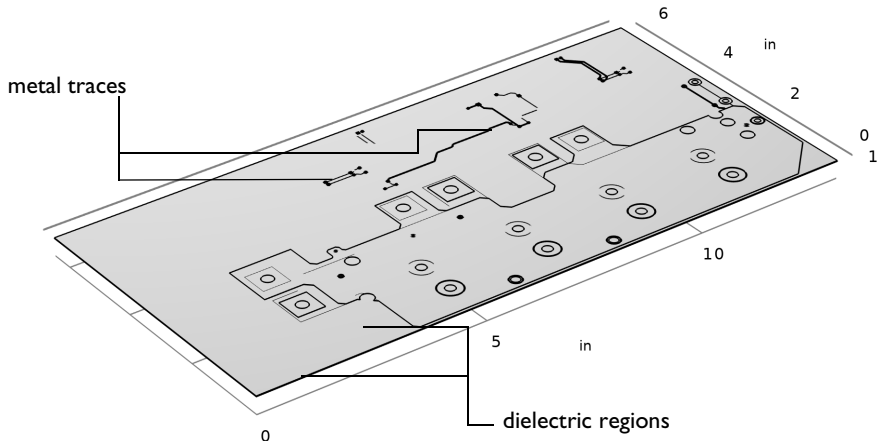
- 6 In the Preview settings window, under the table Nets to Import, click the Import None (🗑️) button, to deselect all the nets. Now, in the table select the checkboxes for +VBUS, -15VA, -15VB, -5V, -VBUS, 15VA, 15VB nets.



When changing the values of checkboxes in these tables, it automatically updates the corresponding checkboxes in the table under Layer section and table under Nets to Import section in the Import settings window.

- 7 Click Import (📁).

The imported geometry appears in the Graphics window.




The import can be configured to use one of 3 methods to generate 3D solid objects from the 2D layouts and group these together:

- By default the *Grouping of geometries* property of the import node is set to *By layer*, which results in separate geometry objects for each imported layer.

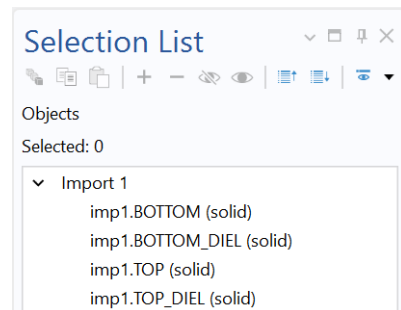
This is the most robust method for extruding the geometry from the 2D layouts, as each layer is extruded separately from the other layers.


- When using the import method corresponding to *Grouping of geometries* set to *All*, a special extrusion method generates a single geometry object that includes domains, or regions, corresponding to the selected layers. While using this method may sometimes provide a faster route to meshing by avoiding the need to combine objects using the Form Union operation, it is more sensitive to inaccuracies between layers.
- A third method, *No grouping*, generates separate objects for each symbol from the 2D layouts of the layers. This method can be useful for troubleshooting purposes, if the import of a layer fails with one of the other methods.

**Note:** Separate geometry objects are automatically combined into a composite object when the geometry is finalized by the Form Union operation, which appears at the end of the geometry sequence. An alternative that can lead to easier meshing for complex board geometries, and is suitable for most physics applications, is to switch the finalization method to Form Assembly. Read more about Form Union and Form Assembly in the *COMSOL Multiphysics Reference Manual*.

- 8 To view a list of the extruded objects, in the Geometry toolbar click Selection List .

The top part of the Selection List window lists all geometry objects in the geometry sequence. The bottom part of the window lists all named selections, in this case the selections created for the layers by the import.



- 9 To get a better view of the interior of the geometry click the Wireframe Rendering button  in the Graphics toolbar.

- 10 Click the objects in the Selection List window to identify them in the Graphics window.

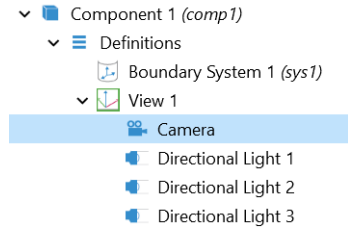
## View Settings

---

The extruded geometry of a circuit board can get quite complex, and due to the large thickness to width and length ratios of the layers it can be very difficult to see the structure after the import. Adjusting the view scaling in the Graphics window

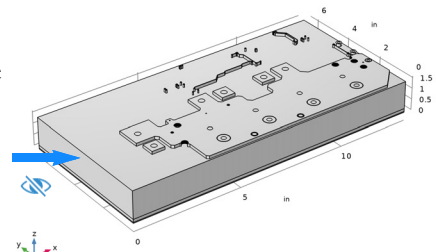
can help to get a better view, as well as hiding geometric entities, such as objects and faces, to be able to look inside the board structure.

- 1 Switch back from the Import settings window to the Model Builder window.
- 2 Under Component 1 expand first the Definitions and then the View 1 nodes.
- 3 Click the Camera node.
- 4 In the Settings window for Camera set the View scale to Manual. In the z scale text field, type 100.



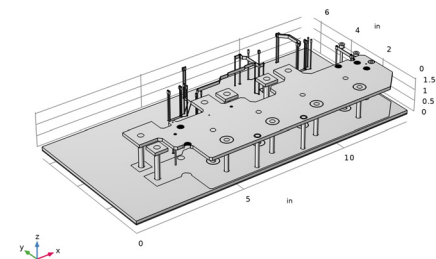
Using this setting you can change the proportion of the geometry in all directions while the default no scaling sets the increments equal in all dimensions preserving the proportions of the geometry. You can also use Automatic settings, where the geometry fills the graphics window,.

- 5 Click Update to refresh the view in the Graphics window.
- 6 To fit the entire board in the window click the Zoom Extents button in the Graphics toolbar.
- 7 For less clutter turn off wireframe rendering by clicking the Wireframe Rendering button .



You can now look inside the geometry for, example, by hiding the top dielectric layer.

- 8 Click next to Select Objects button from the Graphics toolbar, then choose Select Domains. Next, click on the Click and Hide button from the Graphics toolbar, then in the Graphics window click the domain marked in the figure to the right.
- 9 Click the Click and Hide button again to deactivate hiding of selected objects. This is important in order to avoid hiding everything we click on in the Graphics window.



**Note:** You can also view the hidden canvas by clicking the View Hidden Only button .

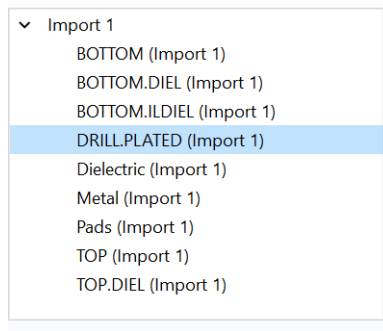
## Drill Layers

---

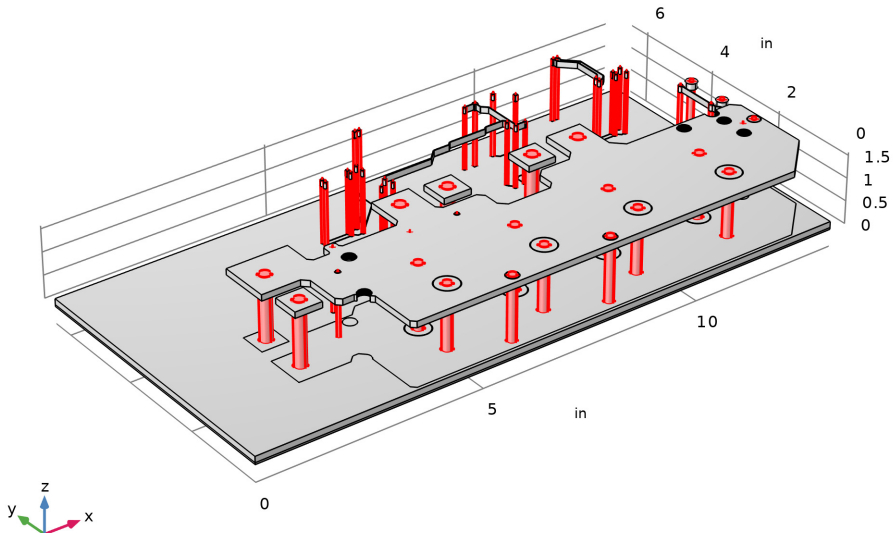
1 Switch back to the Selection List window from the View Settings.

In the lower part of the selection window notice the named domain selections available. There is one drill layer selection that specifies the imported plated holes, DRILL.PLATED.

2 Click DRILL.PLATED domain selection.



The holes are highlighted as in the figure below.




**Note:** When the file is scanned, the thickness and elevation of drill layers is automatically detected. In this case, the drill layers span both dielectric layers.

**3** In the Settings window for Import, you can find the Drill Holes section that lets you choose different options for handling drill layers.

- By default, the drill objects handling method is set to *Modify layer objects*, meaning no new objects are created for drill layers. However, the existing metal and dielectric layers have been modified to incorporate drill layers.
- The second method *Create separate objects*, extrudes drill layers separately for each layer.

**Note:** Objects for drill layers are generated only if the import of at least one dielectric layer spanned by the drill layer is turned on.


**4** To bring back the hidden object click the Reset Hiding (  ) button in the Graphics toolbar.

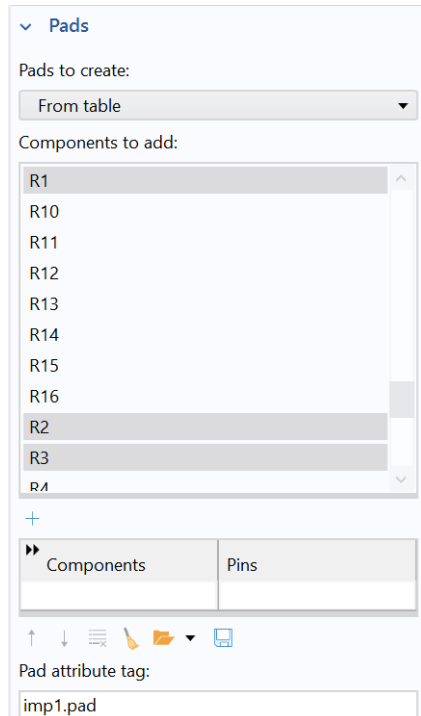
## Pads and Components

---

In the addition to the copper, dielectric and drill layers, a PCB file also contains the component outlines.

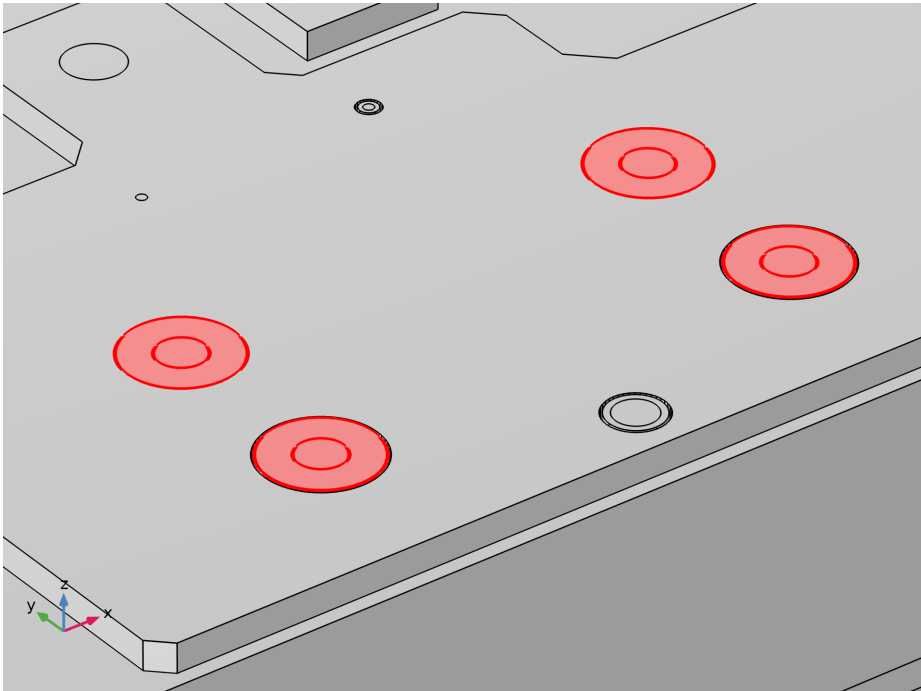
Here, we will import some components and create pads on the copper traces that they connect to.

- 1 Click Import 1 node and expand the Pads section.
- 2 In the Components to add table hold Ctrl and select components C2, C4, C5, C6, R1, R2, R3.
- 3 Click Add Selected Components to Table ( + ) button.  
This will add the components from the list to the table below.
- 4 Click Import .  
This will create pads for the selected components.






- 5 Go to the Selection list and click on the Pads domain selection created after importing.

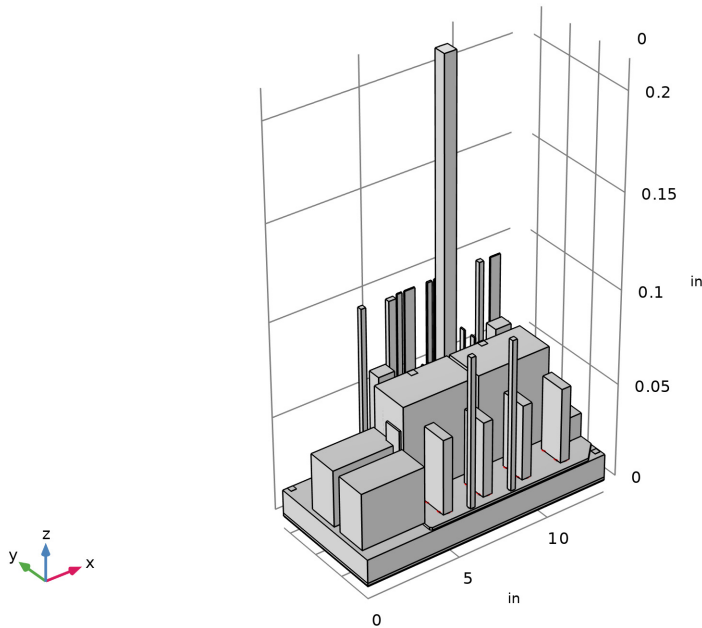
Notice that additional faces have been created.



To create a simplified 3D representation these pads' corresponding components, we will add a separate import node.

- 6 In the Home toolbar, click Import .
- 7 In the Settings window for Import click the Browse button, then select the same file (`printed_circuit_board_si_geom.zip`) as previously.
- 8 Under the Layers table, click the Clear All Imports ( button). This will not import any layers of the board.
- 9 Expand the Components section. Select the checkbox Import component outlines and click Import .

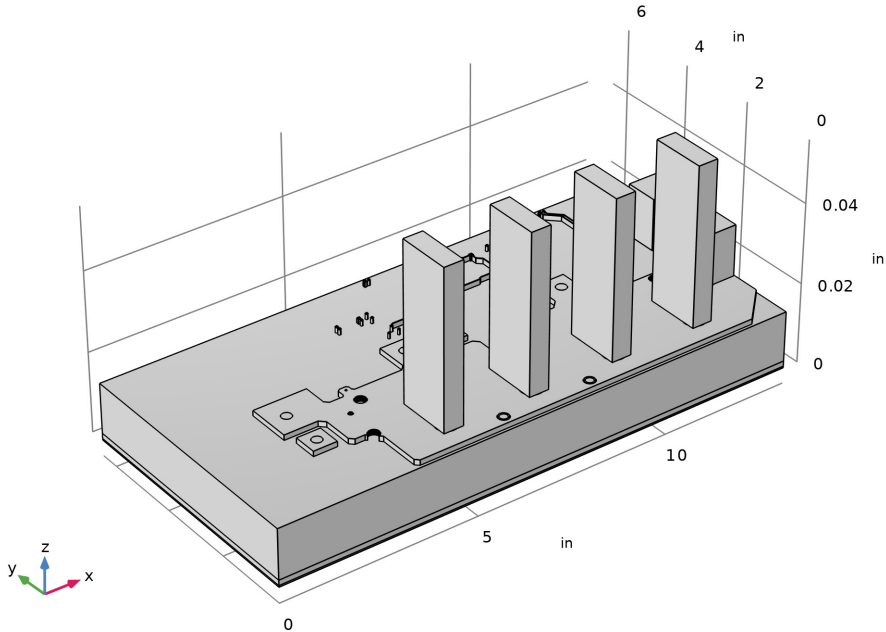
**10** Click the Zoom Extents  button in the Graphics toolbar.



Simplified 3D geometries were generated to represent the PCB's components based on the component outlines available in the file. Since we are only interested in a few components, we can extract them as follows.

- 11** In the Geometry toolbar, click Selections and choose Logical Expression Selection.
- 12** In the Logical expression text area, type `imp2.component('C2', 'C4', 'C5', 'C6', 'R1', 'R2', 'R3')`.
- 13** In the Geometry toolbar, click Extract.
- 14** In the Extract settings window, choose the Domain as Geometric entity level.
- 15** From the Selection list, choose Logical Expression Selection 1.
- 16** From the Input object handling list, choose Remove. Click Build Selected.

17 Click the Zoom Extents  button in the Graphics toolbar.



Only the desired components have been left and all the others have been deleted.

## Layer Selections

---



The *By layer* import method is an efficient way of creating selections when used together with the *Layer selections* option for the import. Easily identifiable selections with layer names are automatically created and can be used for assigning mesh size or physics settings, for example material properties.

In the following you will use these selections to assign appropriate materials, but first let's review the selection settings for the import.

- 1 In the Model Builder window click the Import 1 node.
- 2 In the Settings window for Import, locate and expand the Selections of Resulting Entities section.

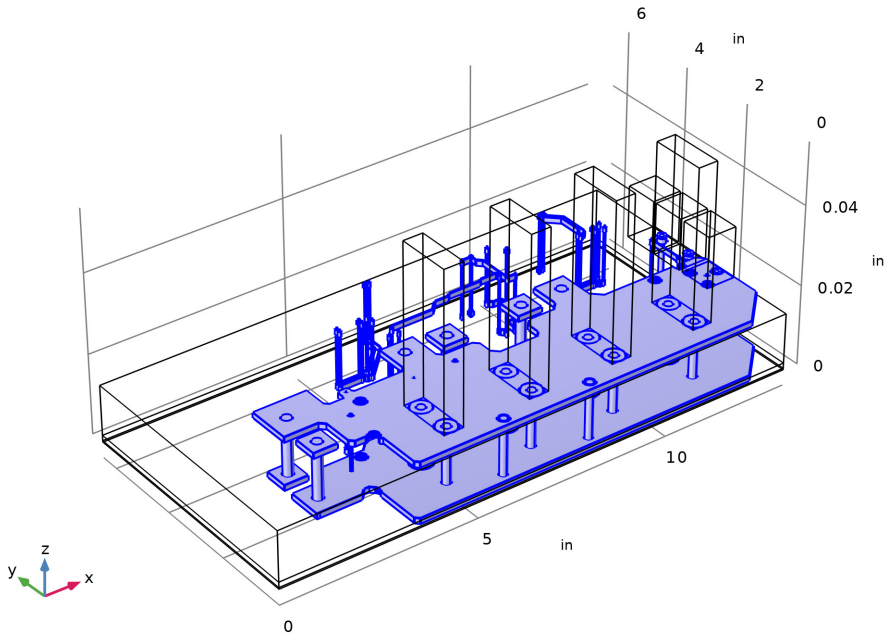
By default the Layer Selections checkbox is selected, and the Show in physics list is set to All levels so that the layer selections for domains and boundaries will be available in the physics and mesh settings. You may also choose Domain selection or Boundary selection, or turn off the display of these selections in

physics altogether. For this latter case the selections will still be available in the geometry sequence.

- 3 In the Model Builder, click on Geometry 1 node. Make sure that the checkbox Automatic detection of small details is selected under Cleanup section.
- 4 In the Home toolbar click  Add Material to open the Add Material window. After import, selection called Metal was created and it includes everything in the files specified as metal. This single selection can be used for assigning materials or a meshing operation.
- 5 In the tree, select Built-in > Copper.
- 6 Right click and choose Add to Component 1. If the Automatic detection of small details checkbox is selected under Geometry 1 node, this action will bring up a dialog from Geometry Cleanup. Some small details have been detected that could make the meshing difficult.
- 7 Click Clean up Automatically.
- 8 After the cleanup is finished click  Add Material to close the Add Material window.
- 9 Click the Copper node and locate the Geometric Entity Selection section.
- 10 From the selection list, choose Metal.

Notice that we have two more selection available apart from the individual selection. Metal is a selection that includes all the metal specified in the file, and Dielectric is a selection of everything else.

- || For a better view of the selection click the Wireframe button in the Graphics toolbar.

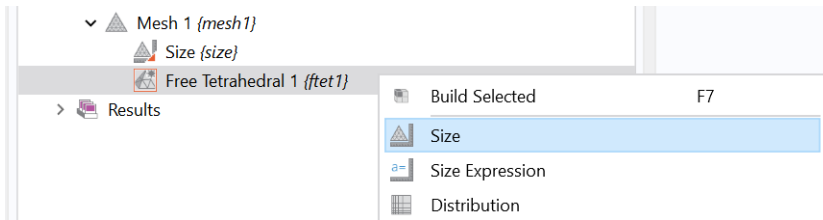


## Meshing the Geometry

---

By using the Metal selection, as in the previous section, it is quite easy to generate a finer mesh only for the copper traces. Note that a mesh suitable for a simulation may be different from the one generated by following the steps below.

- 1 Click the Mesh 1 node, and in the corresponding Settings window change the Element size to Extra coarse and clear the checkbox Geometric Analysis, Details size in the Contributor table. Select User-controlled mesh from the Sequence type list box.
- 2 Right-click the Free Tetrahedral 1 node and select Size. This adds a Size 1 node to the meshing sequence.
- 3 In the Settings window for Size 1 under Free Tetrahedral 1, change the Geometric entity level to Domain.



4 From the Selection list box select Metal.

As you can see the list contains selections corresponding to the layers resulting from the Import nodes. Since we do not need all these selections Import 2 node we can change the settings of Import 2.

5 Click the Import 2 node under Geometry 1.

6 In the Settings window for Import, select Off from the Show in physics list under Layer selections.

7 Switch back to the Size 1 node under Free Tetrahedral 1 in the Mesh 1 sequence.

8 In the Settings window for Size 1 the Selections list now only contains the selections from Import 1 in addition to the Manual and All domains options. Leave the Metal alternative selected.

9 From the Predefined list select Normal.

10 Finally, click the Build All  button to mesh the geometry.

|| Click the Mesh 1 node and zoom in the Graphics window.

