INTRODUCTION TO

ECAD Import Module

COMSOL

Introduction to the ECAD Import Module

Version: COMSOL 6.0

Contact Information

Visit the Contact COMSOL page at 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 for address and contact information.

If you need to contact Support, an online request form is located at the COMSOL Access page at www.comsol.com/support/case. Other useful links include:

- Support Center: www.comsol.com/support
- Product Download: www.comsol.com/product-download
- Product Updates: www.comsol.com/support/updates
- COMSOL Blog: www.comsol.com/blogs
- Discussion Forum: www.comsol.com/forum
- Events: www.comsol.com/events
- COMSOL Video Gallery: www.comsol.com/videos
- Support Knowledge Base: www.comsol.com/support/knowledgebase

Contents

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 IPC-2581 File

The IPC-2581 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. The format contains the data in a single XML file. 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 planar transformer contained in an IPC-2581 file. The file contains layers for metal traces, for dielectric regions, and for the layout of the ferrite core of the transformer.

New Window

1 In the New window click Blank Model to skip the steps of selecting physics and study type.

2 On the Home toolbar, click Add Component ⬦ and choose 3D.
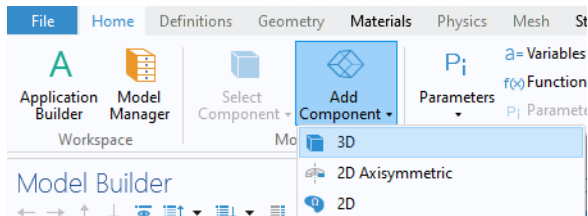
Import

1 On 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 `planar_transformer_layout.xml`.

The Settings window for Import will now show the appropriate settings for the file, including the layers in the Layers to import table. 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 (mm)	Import
UPPER.COMP	Other	0	☐
UPPER.DIST	Other	0	☐
TOP.LAYER	Metal	0.075088	☑
TOP.LAYER.DIEL	Dielectric	0.32499	☑
INNER.LAYER	Metal	0.075088	☑
INNER.LAYER.DIEL	Dielectric	0.25	☑
BOTTOM.LAYER	Metal	0.075088	☑

Note: For 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 Click Import.

The imported geometry appears in the Graphics window.
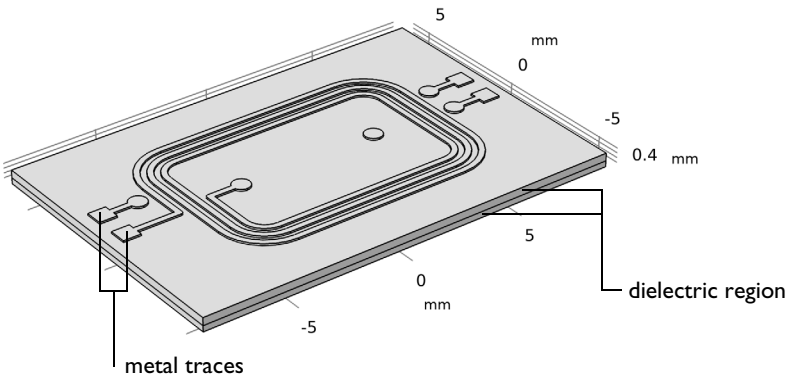
The import can be configured to use one of 3 methods to generate 3D solid objects from the 2D layouts and group these together:

• 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*, is also available that 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*.

5 To view a list of the extruded objects, on the Home toolbar click Windows and choose Selection List .

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.
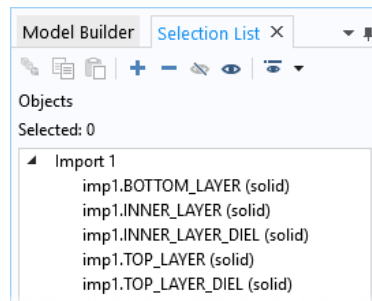
6 To get a better view of the interior of the geometry click the Wireframe Rendering button on the Graphics toolbar.

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

Drill Layers

1 Switch back from the Selection List window to the Model Builder window.

The file includes two drill layers that specify the vias and the ferrite core. These layers are called `THROUGH.HOLES` and `VIAS`.

2 In the Layers to import table, find the row for the `VIAS` and `THROUGH.HOLES` layers and turn on their import by clicking the corresponding cells in the Import column.

Name	Type	Thickness (mm)	Import
INNER.LAYER.DIEL	Dielectric	0.25	☑
BOTTOM.LAYER	Metal	0.075088	☑
LOWER.DIST	Other	0	☐
LOWER.COMP	Other	0	☐
VIAS	Drill	0.57499	☑
THROUGH.HOLES	Drill	0.57499	☑

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

3 Click Import.

This time the resulting geometry also contains the extruded objects from the drill layers.
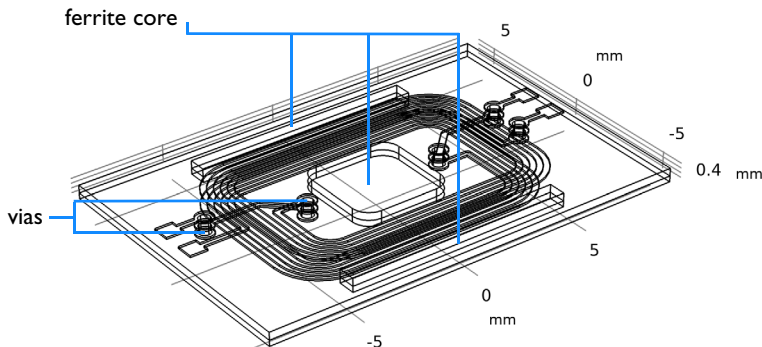
4 Switch to the Selection List window to view the generated objects. The import extrudes drill layers separately for each dielectric layer, and names these by combining the name of the drill and dielectric layers. For example the ferrite core, generated from the `THROUGH.HOLES` layer is extruded into two separate objects: `THROUGH_HOLES_INNER_LAYER_DIEL` and `THROUGH_HOLES_TOP_LAYER_DIEL`.

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.

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 Selection List 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 Automatic.
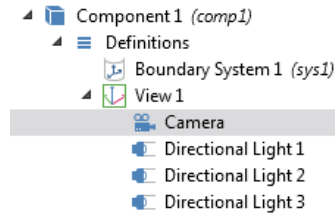
Using this setting the geometry fills the graphics window, while the default no scaling sets the increments equal in all dimensions preserving the proportions of the geometry.

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 on 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.
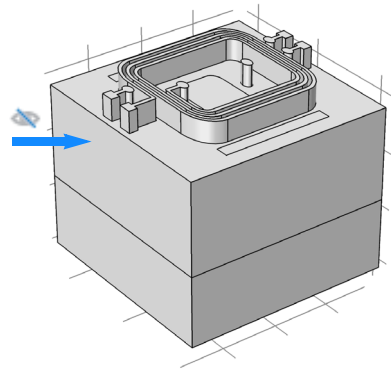
8 Click the Click and Hide button ⬟ from the Graphics toolbar, then in the Graphics window click the geometry object marked in the figure to the right.
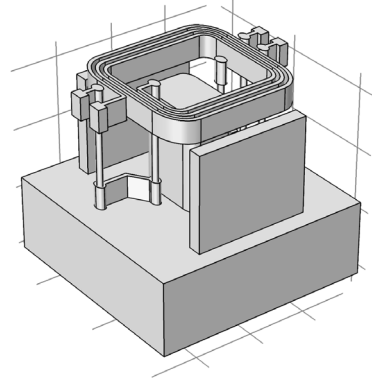
9 Click the Click and Hide button ⬟ again to deactivate hiding of selected objects. This is important in order to avoid hiding everything we click on in the Graphics window.

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

10 To bring back the hidden object click the Reset Hiding button ⤵ on the Graphics toolbar.

Layers of Type: Other

In addition to the copper, dielectric, and drill layers a PCB file may also contain other type of layers such as for example layers for component outlines, and solder mask. In the Layers to import table the type for such layers is set to *Other*. To import such a layer you may need to specify thickness and elevation, as this information is typically not included in the file.

For this planar transformer the file contains four such layers that outline the portions of the ferrite core outside the board. To import we will add a separate import node.

1 On the Home toolbar, click Import ⬛.

2 In the Settings window for Import click the Browse button, then select the same file(planar_transformer_layout.xml) as previously.
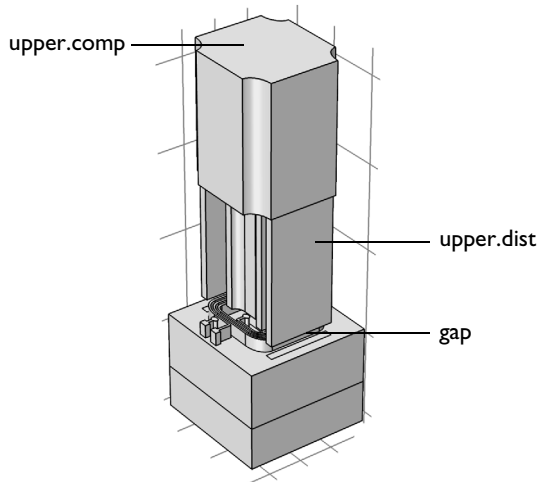
3 In the Layers to import table enable only the import of the layers UPPER.COMP and UPPER.DIST.

4 In the Thickness (mm) column enter the settings from the following table:

NAME	THICKNESS (MM)
UPPER.COMP	0.75
UPPER.DIST	0.25

5 Click Import.

6 Click the Zoom Extents ⬌ button on the Graphics toolbar.

upper.comp ────────

──────── upper.dist

──────── gap

The two layers are extruded taking into account the order in which they are listed in the table. The elevation is determined automatically, and the layers are placed on top of the uppermost copper layer (`TOP.LAYER`). Due to this there is a gap that equals the thickness of the copper layer between the `UPPER.DIST` layer and the top dielectric layer. One way to avoid this gap is to switch to manual elevation control, or to change the import to generate all copper layers as shells. In the following we will use the latter option. Since we are importing these additional layers in a separate import node using the metal shell option will not influence the already imported copper traces, and the method is faster than calculating the correct elevation for each layer.

The Metal Shell Option

The setting for Type of import controls whether the metal layers are extruded (Full 3D) or imported as surface objects (Metal shell). In addition, the extrusion of layers of type Other is also influenced by this setting, since the elevation is determined according to the stack-up information in the Layers to import table.

1 In the Settings window for Import (make sure it is Import 2) set Type of import to Metal shell.

2 In the Layers to import table enable the import of the remaining layers of the type `Other`, namely `LOWER.DIST` and `LOWER.COMP`.

3 In the Thickness (mm) column enter the settings from the following table:

NAME	THICKNESS (MM)
LOWER.DIST	0.25
LOWER.COMP	0.75

4 Click Import.

The objects for the ferrite core are now positioned at the appropriate elevation.
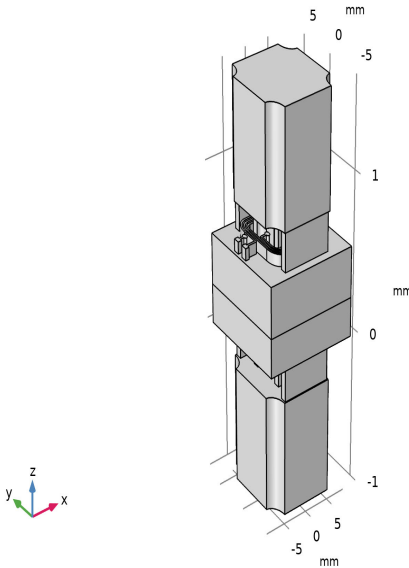
Generating Layer Selections

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 create one easily identifiable selection for all copper traces on the board, 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 the Selections for Resulting Entities section.

By default the Layer Selections check box 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 On the Geometry toolbar click Selections and choose Union Selection.

Even though the individual domain selections are available for each copper layer using the Union selection you can create a single selection to be visible for assigning materials or a meshing operation.

4 Under the Selections to add table click Add.

5 In the Add dialog box select the three copper layers highlighted in the figure. You can press Ctrl to select multiple items from the list. Click OK.

6 To assign a more descriptive label for the selection type `Copper Traces` in the Label text field.
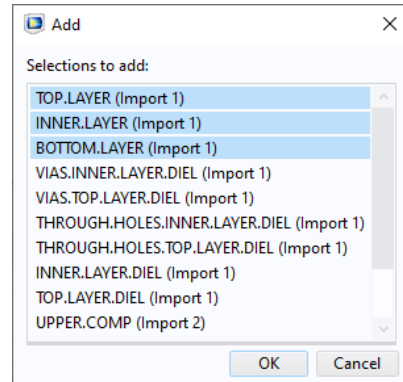
7 For a better view of the output selection click the Wireframe button on the Graphics toolbar.
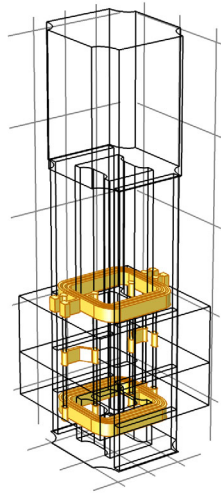

Meshing the Geometry

By using the selection that you have just created 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 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.
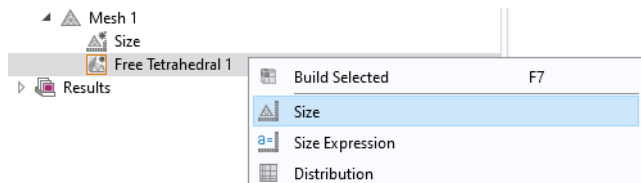
3 In the Settings window for Size 1 change the Geometric entity level to Domain.

4 From the Selection list box select Copper Traces.

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 we can change the setting on the Import 1 and Import 2 nodes.

| 15

5 Click the Import 1 node under Geometry 1.

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

7 Click the Import 2 node, then repeat the previous step.

8 Switch back to the Size 1 node in the Mesh 1 sequence.

9 In the Settings window for Size 1 the Selections list now only contains the Copper Traces alternative in addition to the Manual and All domains options. Leave the Copper Traces alternative selected.

10 From the Predefined list select Fine.

11 Finally, click the Build All button ⊞ to mesh the geometry.