



Add-in created in COMSOL Multiphysics 6.4

PCB to Material Data

Introduction

This add-in provides a streamlined way to prepare a printed circuit board (PCB) for simulation by replacing its detailed internal structure with a space-dependent interpolation function for the material properties. Instead of explicitly modeling every trace and via in the geometry, the interpolation function captures their effect across the board. The resulting simplified geometry is generated by extruding the PCB outline according to the layer stack defined in the input file.

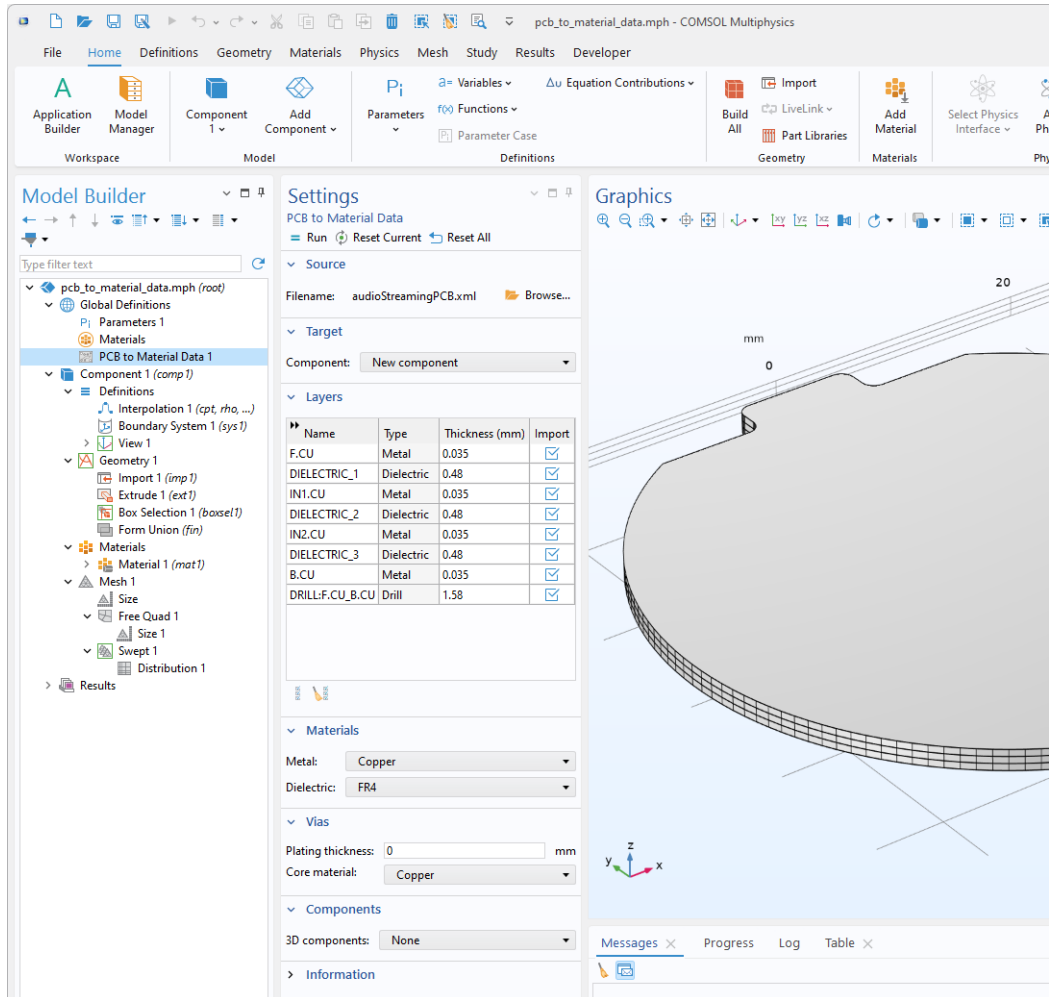
With this approach, you can assign materials for traces, dielectric regions, and via cores directly through the add-in. There is also flexibility to include domains for selected components, giving you control over how much detail to preserve in the model.

This simplification technique is particularly valuable for studies where full geometric detail is unnecessary but material behavior is critical. For example, when analyzing heat loads, evaluating temperature distributions, or investigating the effects of thermal expansion on the board. By reducing geometric detail while preserving the essential material behavior, the add-in speeds up geometry creation, meshing, and simulation, helping you explore results faster and with fewer computational resources.

USING THE ADD-IN

Start by selecting a PCB file for analysis. The layers contained in the file are then read and displayed in a table where their names, types, and thicknesses can be reviewed and modified. Copper layers that are not selected for import are still accounted for and they are approximated with dielectric material properties, preserving the overall thickness of the

board. At least one metal or dielectric layer with a nonzero thickness must be included to ensure that the board is correctly represented in the analysis.



With the layout defined, the next step is to assign material properties. In the **Materials** section you decide how copper layers (including hole plating, if specified) and dielectric layers should be treated. While in the **Vias** section, you can choose which materials will be assigned to via cores. The add-in supports both built-in materials and user-defined materials added to **Global Materials**. Regardless of the choice, the assigned materials must contain a consistent set of six material properties: heat capacity, density, thermal conductivity, Young's modulus, Poisson's ratio, and coefficient of thermal expansion.

These properties are essential for simulations involving heat transfer, solid mechanics, and thermal expansion. Note that only constant values are supported.

Once the layers and materials are specified, the add-in can be run. A progress bar indicates the status while the add-in systematically processes each layer and generates a space-dependent interpolation function representing the interior structure of the board. A file with a resulting interpolation function is stored in the user files directory. By default, this directory is set under **Preferences > Files > Folder for user files**, but the location can be changed if desired. A working version of the selected PCB file is stored in the same folder.

In the last steps, add-in creates a simplified PCB geometry, applies the imported interpolation function for the material properties and generates a mesh using a Swept mesh generator. The mesh element size is determined based on the copper traces within the file and you can adjust it later to create finer or coarser mesh.

In addition to generating the interpolation-based board model, the add-in offers the option of creating simplified 3D components from the file's outlines. This step does not affect the interpolation function but adds geometric detail that can be useful when constructing the complete simulation model.

An interpolation function is defined by a file that provides function values at discrete grid points. In this workflow, an interpolation file is automatically generated by the add-in, with the naming convention

interpolationFunction_<PCB_filename>. In the settings window for **Interpolation** to the right you can see an example of an imported interpolation function file generated by the add-in. Six material properties (functions) are defined in the file: *cpt* (heat capacity at constant pressure), *rho* (density), *kmean* (thermal conductivity), *E* (Young's modulus), *nu* (Poisson's ratio), and *alpha* (coefficient of thermal expansion).

The **Interpolation** and **Extrapolation** settings determine how values are evaluated between grid points and outside the defined coordinate domain. Both are set by default to **Nearest neighbor**, meaning the value of the closest defined point is used. If required, these settings can be adjusted to other methods, depending on the needs of the analysis.

Finally, the **Units** section specifies the units of both the functions and their arguments. This table is generated automatically, with argument units derived from the geometry units defined in the PCB file.

To inspect the created interpolation function, you can click **Create Plot** in the settings toolbar. This generates a **3D Plot Group** with slice plots of the function, similar to the example shown below. By default, the plot displays a low-resolution version

Settings

Interpolation

Plot Create Plot

Label: Interpolation 1

Definition

Data imported into model

Filename: interpolationFunction_audioStre

Data type: Grid

Dimension: 3D

Export... Discard

Functions

Function name	Position in file
cpt	1
rho	2
kmean	3
E	4

Use spatial coordinates as arguments

Frame: Spatial

Interpolation and Extrapolation

Interpolation: Nearest neighbor

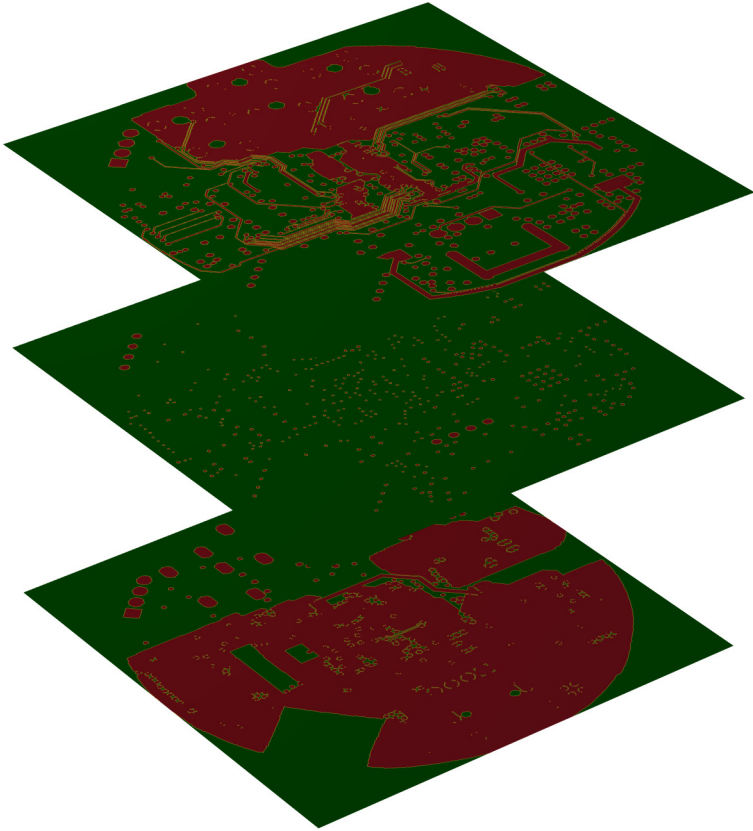
Extrapolation: Nearest function

Units

Function	Unit
cpt	J/(kg*K)
rho	kg/(m^3)
kmean	W/(m*K)
E	GPa
nu	1
alpha	1/K

Argument	Unit
Argument 1	mm
Argument 2	mm
Argument 3	mm

of the interpolation data, but you can increase the level of detail by adjusting the grid resolution in the **3D Grid** settings under **Datasets**.

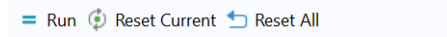


The PCB featured in the images originates from [OtterCastAudioV2](#), Copyright © 2024 Ottercast, Jana Marie Hensing, and it is made available under the [MIT license](#).

Add-in Library path: ECAD_Import_Module/pcb_to_material_data

TOOLBAR

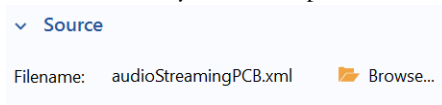
The toolbar at the top of the **Settings** window provides quick access to main controls of the add-in.



Click **Run** to execute the add-in and create an interpolation function representing the geometry of selected PCB. Click **Reset Current** to reset only the options related to the currently selected board. Click **Reset All** to restore all settings in the window to their defaults.

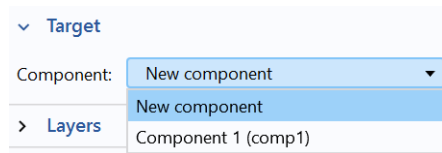
SOURCE

Use the **Source** section to select PCB. Click the **Browse** button to open a file browser and select a PCB file you want to process.



TARGET

The **Target** section defines where the interpolation function is created in the model. You can create a **New component** (default) or insert the function into an already defined 3D component in the current model. The list shows all valid 3D components available for selection.



LAYERS

The **Layers** section provides a table that lists all metal, dielectric, and drill layers in the selected PCB. Each row contains the layer name, which can be renamed if needed, along with the layer type, such as metal, dielectric, or drill. The table also includes the thickness of each layer, with values that can be modified to match the desired board thickness. A set of checkboxes makes it possible to control whether individual layers are included in the

analysis, with all layers selected for processing by default. For quick access, you can use buttons below table to **Clear all Layers** or **Select All Layers**.

Layers

Name	Type	Thickness (mm)	Import
F.CU	Metal	0.035	<input checked="" type="checkbox"/>
DIELECTRIC_1	Dielectric	0.48	<input checked="" type="checkbox"/>
IN1.CU	Metal	0.035	<input checked="" type="checkbox"/>
DIELECTRIC_2	Dielectric	0.48	<input checked="" type="checkbox"/>
IN2.CU	Metal	0.035	<input checked="" type="checkbox"/>
DIELECTRIC_3	Dielectric	0.48	<input checked="" type="checkbox"/>
B.CU	Metal	0.035	<input checked="" type="checkbox"/>
DRILL:F.CU_B.CU	Drill	1.58	<input checked="" type="checkbox"/>

MATERIALS

In the **Materials** section, you can specify material properties for **Metal** (including hole plating, if specified) and **Dielectric** layers. The default option for metal is set to **Copper**, while material for dielectric layers is set to **FR4**. An alternative option is to set the layer material properties to an already defined material in the **Materials** node under **Global Definitions**.

Materials

Metal:

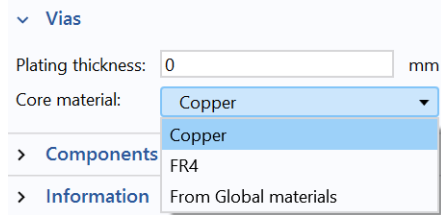
Dielectric:

> Vias

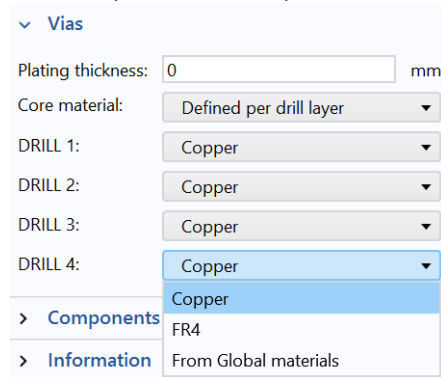
VIAS

In the **Vias** section, you can assign material properties to the via cores. The core material can be set to **Copper** or **FR4**, or you can choose **From Global materials** to assign material

properties of an existing material in the **Materials** node under **Global Definitions**. It is also possible to add a plating to all holes by specifying the plating thickness.

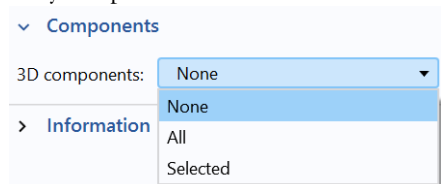


When the selected file contains multiple drill layers, an additional **Defined per drill layer** option becomes available. This makes it possible to define core material properties individually for each drill layer.

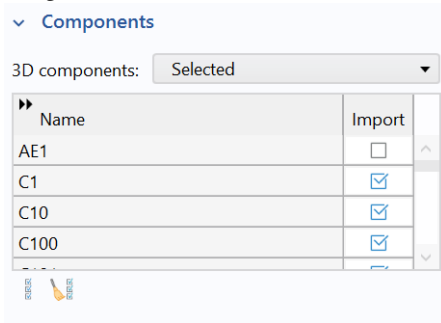


COMPONENTS

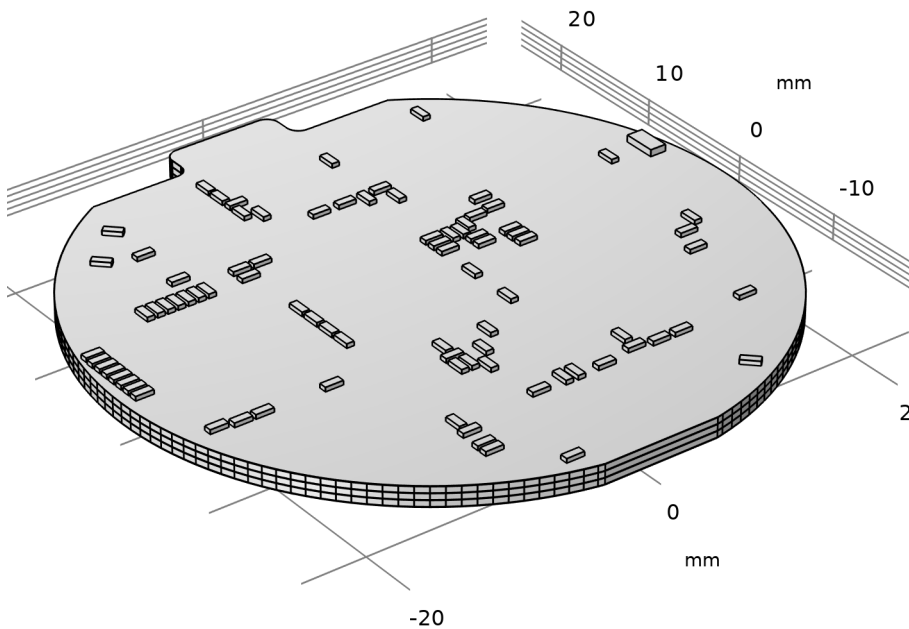
The **Components** section controls how 3D PCB components are created in the target component. If **None** is selected, no components are generated, while choosing **All** creates every component included in the file.



Choosing **Selected** displays a table with all available components. By default, component import is turned off, but you can enable specific items by checking them in the table or by using the buttons below the table to select or clear all entries.



This flexibility makes it possible to import only the components relevant to your analysis, as illustrated in the example where only a few selected components are included in the geometry.



INFORMATION

In the **Information** section, you will find the last build time, as well as the COMSOL Multiphysics version used to run the add-in.