Model created in COMSOL Multiphysics 6.4

Importing PCBs — Using the Add-In

This model is licensed under the COMSOL Software License Agreement 6.4.
All trademarks are the property of their respective owners. See www.comsol.com/trademarks.

Introduction

This tutorial series demonstrates how to import printed circuit board (PCB) data from the IPC-2581 file format and generate simulation-ready geometry. Key topics include selecting the import type, handling drill layers, creating simplified 3D components and pads, and previewing file content before importing. The tutorials also show how to remove small details, such as short edges, either through Automatic Geometry Cleanup or by simplifying and repairing geometry during import. Geometry finalization is done using Form Union or Form Assembly, followed by mesh generation with adjusted element sizes to resolve small copper trace features using the Free Tetrahedral and Swept mesh generators.

This tutorial, the fourth in the series, uses the PCB to Material data add-in to prepare a PCB for simulation by replacing its detailed internal structure with a space-dependent interpolation function for the material properties. The model also considers the influence of thermal expansion caused by the heat dissipation from several resistors placed on the board.

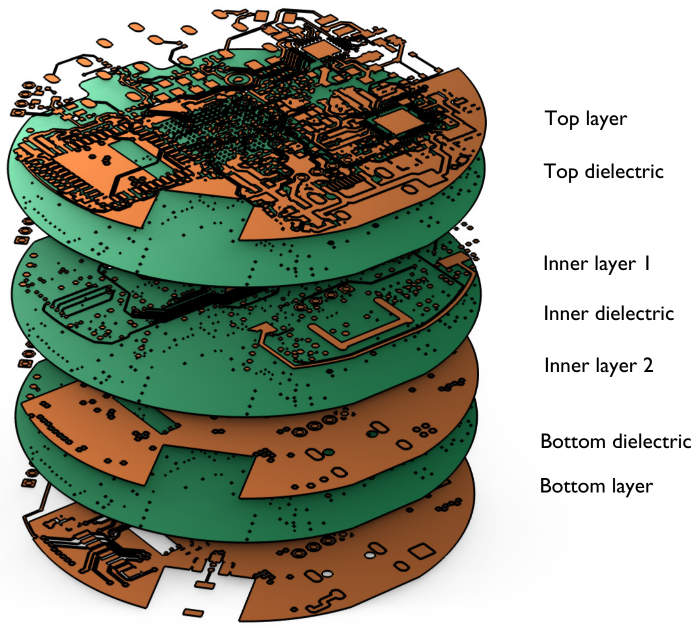
The first tutorial in the series, Importing PCBs — Generating Shell Traces, shows how to model copper traces as shell geometry, automatically remove small details with Automatic Geometry Cleanup, and use import-generated selections to assign materials.

The second tutorial, Importing PCBs — Creating Component Domains, demonstrates how to import PCBs with extruded copper traces, simplify and repair geometry during import, and create pads for simplified components. The geometry is finalized with Form Assembly and meshed using the Swept operation.

The third tutorial, Importing PCBs — Working with Nets, shows how to use the 2D Preview to selectively import nets and prepare a simplified geometry. It also demonstrates how to partition the board geometry with a bounding box for selected nets.

Model Definition

The IPC-2581 file imported in this tutorial contains a PCB with four copper layers connected by vias.



This model uses the PCB to Material Data add-in available from the Add-in Libraries with the ECAD Import module. Instead of explicitly modeling every trace and via in the geometry, the add-in creates an interpolation function that captures the different material properties across the board. A resulting simplified geometry is then generated by extruding the PCB outline according to the layer stackup defined in the input file. With the user interfaces provided by the add-in, you can choose the material properties for the traces, dielectric regions, and via cores, and select the components to include in the model. In this tutorial, the simplified domains of a few resistors are generated to study the heat dissipation across the board. For the physics setup, assume that a resistor dissipates 62.5 mW of power as heat. Conduction to the PCB and convection to the surrounding air provide cooling. All boundaries are cooled with a heat transfer coefficient of 5 $W/(m^2{\cdot}K)$ that corresponds to natural convection. The thermal conductivities are chosen to be similar to:

- Alumina, for the resistors

- FR4, for the dielectric layer of the PCB
- Copper, for the metal layer of the PCB

The displacements in the Solid Mechanics interface are constrained in such way that the PCB can freely deform while its rigid body motions are suppressed.

THE PCB TO MATERIAL DATA ADD-IN

The settings window for the add-in is used to select the PCB file and to configure the import. First, you can review the layers to consider for the analysis, and if necessary change the layer names and thicknesses. The material properties for skipped copper layers will be replaced with dielectric material properties in the interpolation file. At least one copper or dielectric layer with nonzero thickness needs to be selected for the import.

Next, you can choose the materials for the metal and dielectric layers and the via cores. The default choices are copper for the metal domains and FR4 for the dielectric domains. You can also select custom materials defined under the global Materials node in the model. Make sure that the materials contain at least the following six material properties: heat capacity, density, thermal conductivity, Young's modulus, Poisson's ratio, and coefficient of thermal expansion. These properties are necessary for simulations involving heat transfer, solid mechanics, and thermal stress. Note that only constant values are supported.

Once run, the add-in starts with determining an appropriate interpolation grid size that is selected to correspond to the radius of the smallest via in the PCB. If no valid drill layers can be found in the file, a grid size will instead be estimated based on the bounding box of the PCB.

The add-in then processes metal and dielectric layers, evaluating for each layer the material properties on the generated grid, and storing the property values in the file that will be used as input for the interpolation function added later. The interpolation data is written using a grid file format, where the first section of the file defines a regular grid - points inside the PCB volume where the material data is evaluated. The second section contains the material property values. You can easily reuse the interpolation data in other models by creating a copy of the generated file, which you can find in the folder for user files specified under **Preferences** > **Files** > **Folder for user files**.

Interpolation Function

When the processing of the PCB file is complete, an Interpolation feature, inserted under the Definitions branch of the target Component, is used to load the generated file. Another possibility for creating a copy of the interpolation data is by clicking Export from the settings for the interpolation feature.

The interpolation and extrapolation settings determine how values are evaluated between grid points and outside the defined coordinate domain. By default, both use Nearest neighbor, which preserves the discrete layer transitions. You can select one of the other available alternatives to interpolate between points where material property values are defined.

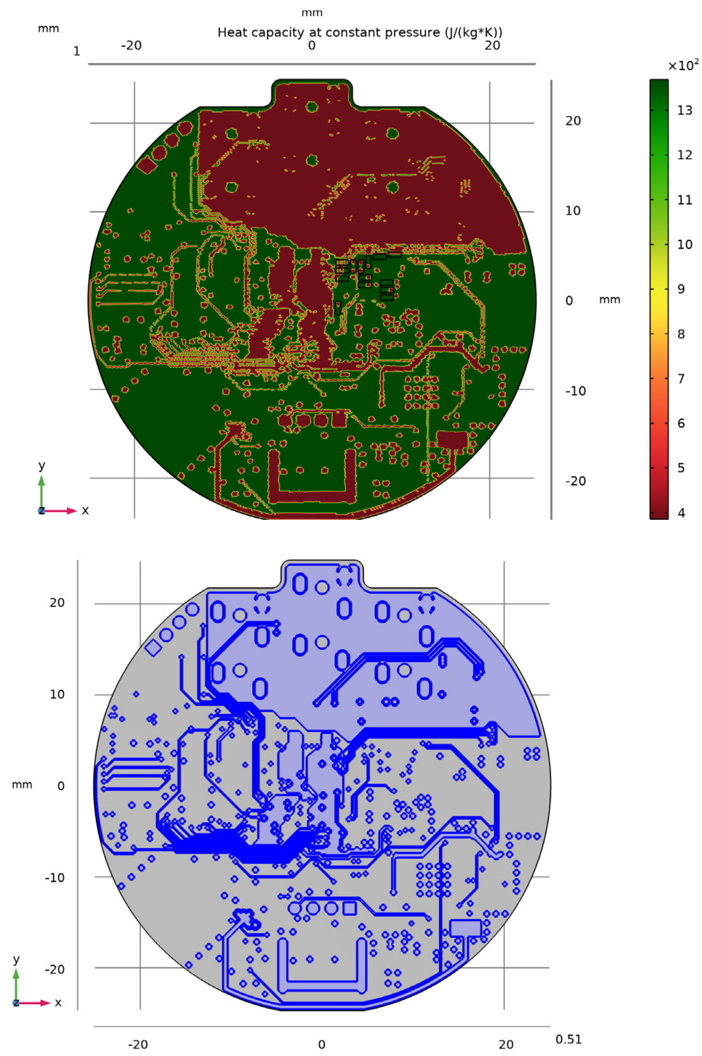


Figure 1: Material properties of the board (top), geometry (bottom) of the inner layer 1.

The heat capacity evaluated at the elevation of the inner layer 1 using the interpolation function set up by the add-in can be compared to the geometry generated for the layer in Figure 1.

Simplified Geometry and Meshing

To generate a simplified PCB geometry, the outline of the board is extruded to match the layer elevations. The geometric domains for the selected components are also created at this stage. Finally, a mesh is generated using the quad and swept mesh generators. The mesh element size is determined based on the interpolation grid size.

The PCB featured in this tutorial originates from OtterCastAudioV2, Copyright © 2024 Ottercast, Jana Marie Hemsing, and it is made available under the MIT license.

Results and Discussions

Figure 2 illustrates the temperature distribution inside the PCB.

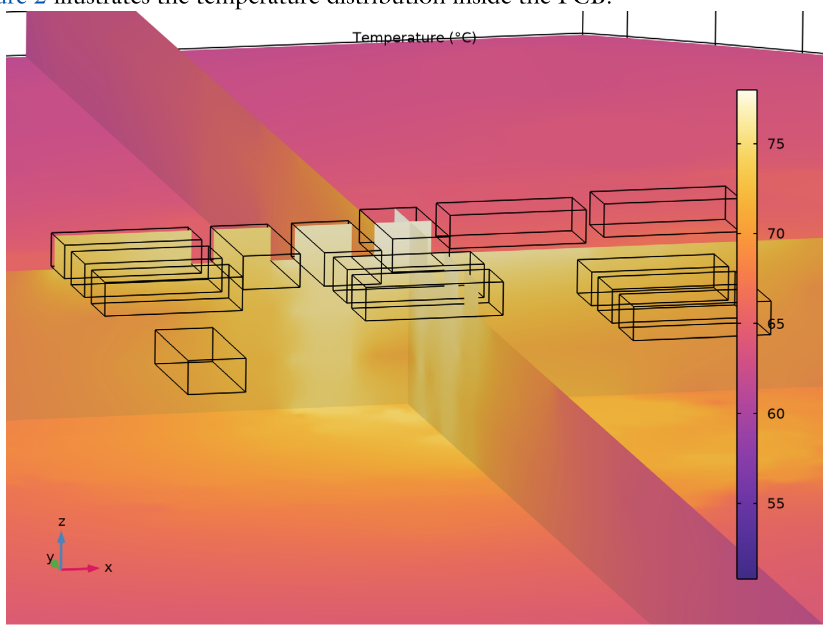


Figure 2: Multislice plot of the temperature distribution in the PCB close to heat dissipating components.

[Figure 3](#) shows the displacement magnitude due to thermal expansion.

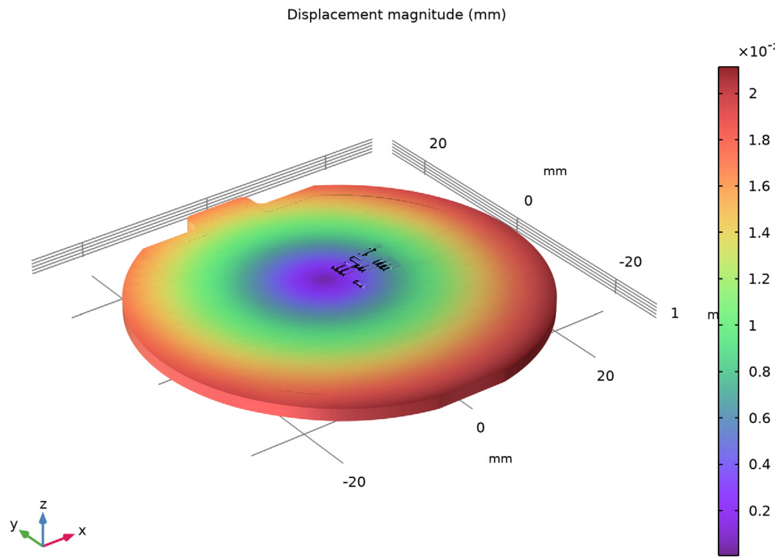


Figure 3: Magnitude of displacement in the PCB (exaggerated by a factor of 160).

Application Library path: `ECAD_Import_Module/Tutorials/pcb_import_addin`

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 In the **Select Physics** tree, select **Structural Mechanics** > **Thermal–Structure Interaction** > **Thermal Stress, Solid**.

3 Click **Add**.

4 Click **Study**.

5 In the **Select Study** tree, select **General Studies** > **Stationary**.

6 Click **Done**.

ADD MATERIAL

Start by adding materials to the **Global Materials** that will be used to define material properties of the PCB.

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** > **Copper** and **Built-in** > **FR4 (Circuit Board)**.

4 Click the **Add to Global Materials** button in the window toolbar.

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

GLOBAL DEFINITIONS

FR4 (Circuit Board) (mat2)

In the **Home** toolbar, click **Windows** and choose **Add-in Libraries**.

ADD-IN LIBRARIES

1 In the **Add-in Libraries** window, in the tree, select the checkbox for the node **ECAD Import Module** > **pcb_to_material_data**.

2 Click **Done**.

3 In the **Developer** toolbar, click **Add-ins** and choose **PCB to Material Data** > **PCB to Material Data**.

PCB to Material Data

1 In the **Model Builder** window, under **Global Definitions** click **PCB to Material Data**.

2 In the **Settings** window for **PCB to Material Data**, locate the **Source** section.

3 Click **Browse**.

4 Browse to the model's Application Libraries folder and double-click the file `audioStreamingPCB.xml`.

5 Locate the **Layers** section.

The table contains the layer stackup for the PCB, including only the metal, dielectric and drill layers. By default, all layers are selected to be processed by the add-in.

6 Locate the **Target** section.

Here you can choose where the interpolation function and the simplified geometry will be created. By default, the add-in inserts them in a new 3D component. Alternatively, you can select an already existing 3D component.

7 From the **Component** list, choose **Component 1 (comp1)**.

8 Locate the **Materials** section.

You can assign different materials for metal and dielectric layers of the PCB. By default, copper is assigned for metal and FR4 for dielectric material. You can use other materials, like composite epoxy materials, metal-core substrates or specialized copper composites defined under **Global Materials**. Note that each material must provide six constant values, see The PCB to Material Data Add-In.

9 From the **Metal** list, choose **From Global materials**.

10 From the list, choose **Copper**.

11 From the **Dielectric** list, choose **From Global materials**.

12 From the list, choose **FR4 (Circuit Board)**.

13 Locate the **Vias** section.

In this section you can choose the material properties for the via cores and optionally their plating. By default, via cores are defined by copper material properties, but you can also choose between FR4 or a user-defined material.

14 From the **Core material** list, choose **From Global materials**. From the list, choose **Copper**.

15 Locate the **Components** section.

Here you can select the components to import from the PCB file. While these components do not affect the interpolation function, their outline will be extruded to generate domains that can become useful for setting up a simulation, for example when defining heat sources. After the add-in finishes, you could also create component domains manually using the geometry functionality or import from other sources.

16 From the **3D components** list, choose **Selected**. In the table, select the **Import** checkboxes for **R39, R48-R51, R78, R81, R87-R93**.

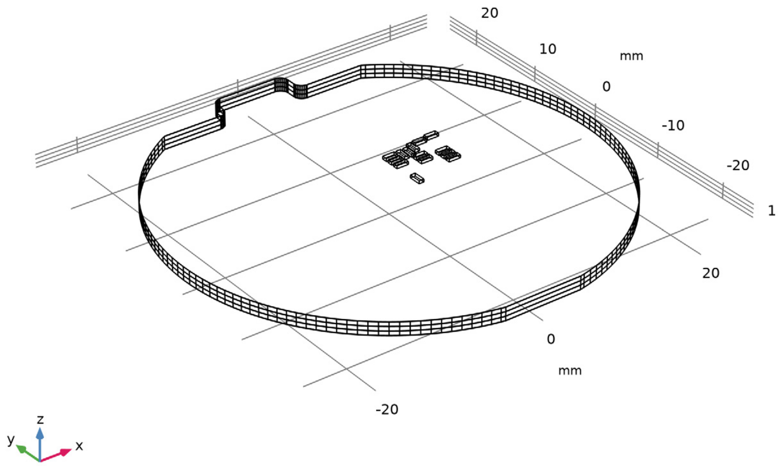
17 Click **Run** button in the window toolbar to create material data.

While the add-in is running, the progress window is displayed. The top progress bar shows the total progress of the add-in, while bottom bar shows the progress for the current step that the add-in is working on, for example `Processing layer: F.Cu`.

Component 1 (comp1)

1 In the **Model Builder** window, click **Component 1 (comp1)**.

2 Click the **Wireframe Rendering** button in the **Graphics** toolbar.



The simplified PCB geometry does not contain any copper traces and vias, as these are instead captured by the interpolation data with the material properties.

3 Click the **Wireframe Rendering** button in the **Graphics** toolbar.

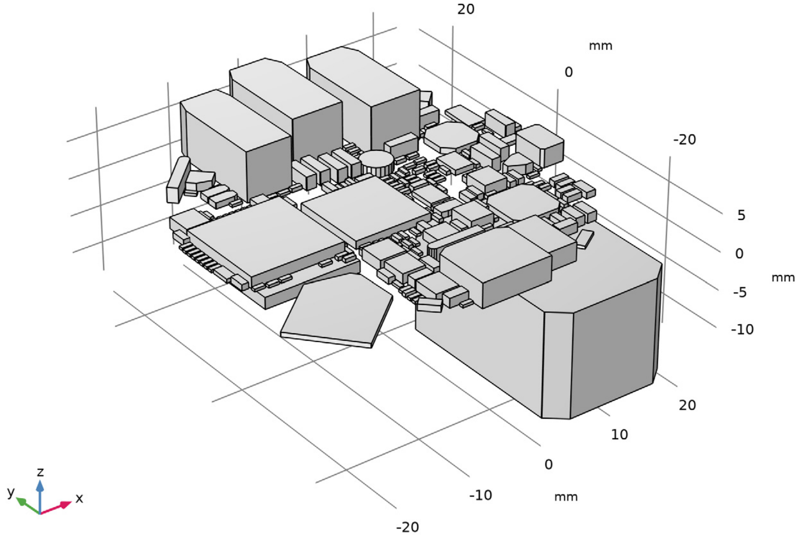
GEOMETRY 1

Import 1 (imp1)

The geometry sequence generated by the add-in includes the operations for creating the simplified representation of the components and the PCB where the board's outline, thickness and layer distribution are preserved.

1 In the **Model Builder** window, expand the **Component 1 (comp1)** > **Geometry 1** node, then click **Import 1 (imp1)**.

2 In the **Settings** window for **Import**, click **Build Selected**.



The first import node is set up to generate the components. The outlines are read from the file and extruded to create the component domains.

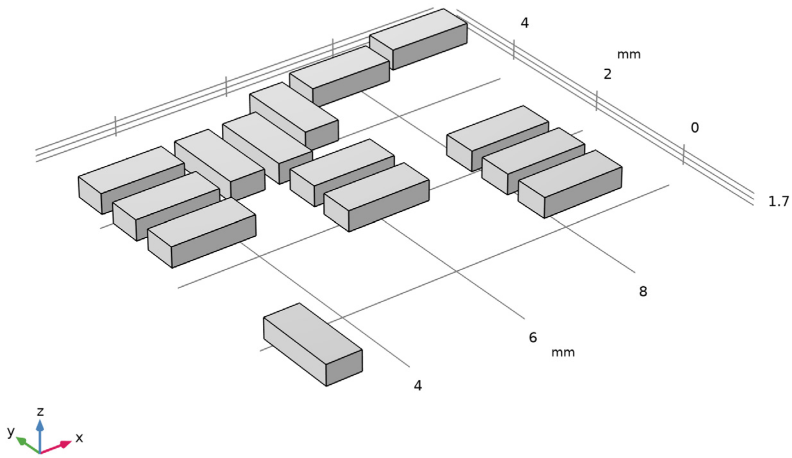
Extract 1 (extract1)

1 In the **Model Builder** window, click **Extract 1 (extract1)**.

2 In the **Settings** window for **Extract**, click **Build Selected**.

Since the import generates domains for all components, the **Extract 1** feature is used to retain only those selected earlier.

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



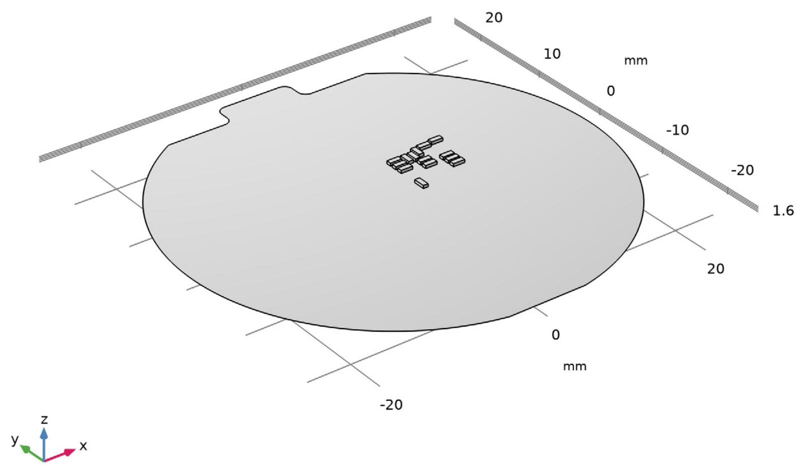
Import 2 (imp2)

1 In the **Model Builder** window, click **Import 2 (imp2)**.

2 In the **Settings** window for **Import**, click  **Build Selected**.

This second import node is configured to import one dielectric layer with the thickness set to zero. The resulting face is used at the next step to extrude the board geometry.

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

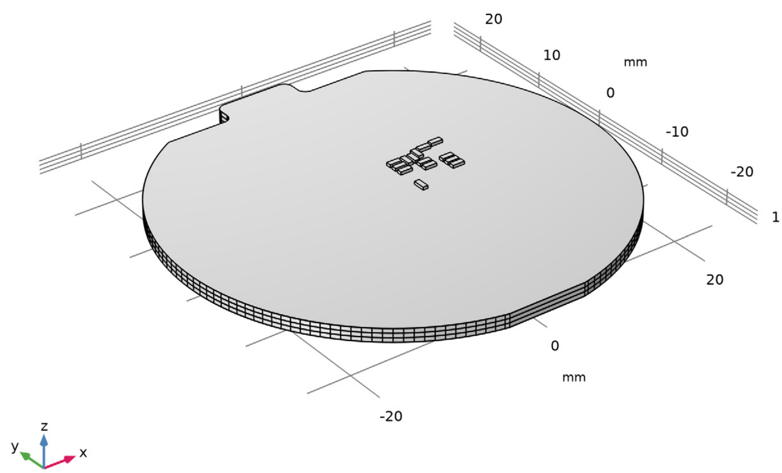


Extrude 1 (ext1)

1 In the **Model Builder** window, click **Extrude 1 (ext1)**.

2 In the **Settings** window for **Extrude**, click **Build Selected**.

The distances specified in the Distances table for **Extrude 1** correspond to the layer elevations in the PCB file. Thus, in the resulting object the separate domains represent the layers from the PCB file.

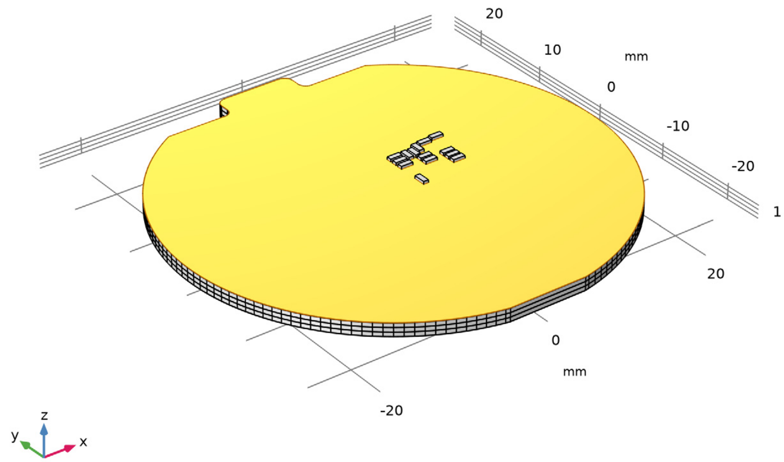


Box Selection 1 (boxsel1)

1 In the **Model Builder** window, click **Box Selection 1 (boxsel1)**.

2 In the **Settings** window for **Box Selection**, click **Build Selected**.

This coordinate-based selection has been added for selecting the topmost boundary of the board which is used in the meshing sequence.



Interpolation 1

The add-in generates a space-dependent interpolation functions that mathematically models the effect of the board's traces and vias allowing for the substitution of detailed geometry with equivalent material property fields.

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

The functions that define the material properties are listed in the Functions table.

MATERIALS

Material 3 (mat3)

1 In the **Model Builder** window, expand **Component 1 (comp1)>Materials** node, then click **Material 3 (mat3)**.

2 In the **Settings** window for Material, locate the **Material Contents** section.

For each material property defined in the table, the Value column contains the corresponding interpolation function.

ADD MATERIAL

While the add-in has configured the material properties of the board, it remains to choose an appropriate material for the imported resistors.

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** > **Alumina**.

4 Click the **Add to Component** button in the window toolbar.

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

MATERIALS

Alumina (mat4)

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

2 From the **Selection** list, choose **Extract 1**.

HEAT TRANSFER IN SOLIDS (HT)

Heat Source 1

1 In the **Physics** toolbar, click **Domains** and choose **Heat Source**.

2 In the **Settings** window for **Heat Source**, locate the **Domain Selection** section.

3 From the **Selection** list, choose **Extract 1**.

4 Locate the **Heat Source** section. From the **Heat source** list, choose **Heat rate**.

5 In the P_0 text field, type `14*0.0625[W]`.

Heat Flux 1

1 In the **Physics** toolbar, click **Boundaries** and choose **Heat Flux**.

To more easily select the boundaries subjected to natural convection, create a selection in the geometry that encompasses all exterior boundaries of the board and components.

GEOMETRY 1

Form Union (fin)

1 In the **Model Builder** window, under **Component 1 (comp1)** > **Geometry 1** click **Form Union (fin)**.

2 In the **Settings** window for **Form Union/Assembly**, click **Build Selected**.

By first building the Form union node, you ensure that the Adjacent Selection feature added in the next step is inserted after Form Union. This is important to avoid selecting boundaries that are between the components and the board.

Exterior boundaries

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

2 In the **Settings** window for **Adjacent Selection**, type `Exterior boundaries` in the **Label** text field.

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

4 In the **Add** dialog, in the **Input selections** list, choose **Extract 1** and **Extrude 1**.

5 Click **OK**.

HEAT TRANSFER IN SOLIDS (HT)

Heat Flux 1

1 In the **Model Builder** window, under **Component 1 (comp1)** > **Heat Transfer in Solids (ht)** click **Heat Flux 1**.

2 In the **Settings** window for **Heat Flux**, locate the **Boundary Selection** section.

3 From the **Selection** list, choose **Exterior boundaries**.

4 Locate the **Heat Flux** section. From the **Flux type** list, choose **Convective heat flux**.

5 In the h text field, type `5[W/(m^2*K)]`.

6 In the T_{ext} text field, type `20[degC]`.

SOLID MECHANICS (SOLID)

In the **Model Builder** window, under **Component 1 (comp1)** click **Solid Mechanics (solid)**.

Rigid Motion Suppression 1

1 In the **Physics** toolbar, click **Domains** and choose **Rigid Motion Suppression**.

2 In the **Settings** window for **Rigid Motion Suppression**, locate the **Contributing Points** section.

3 From the list, choose **From selection input**.

4 Locate the **Point Selection** section. From the **Selection** list, choose **Extrude 1**.

MULTIPHYSICS

Thermal Expansion 1 (te1)

1 In the **Model Builder** window, under **Component 1 (comp1)** > **Multiphysics** click **Thermal Expansion 1 (te1)**.

2 In the **Settings** window for **Thermal Expansion**, locate the **Model Input** section.

3 From the T_{ref} list, choose **User defined**. In the associated text field, type `20[degC]`.

MESH 1

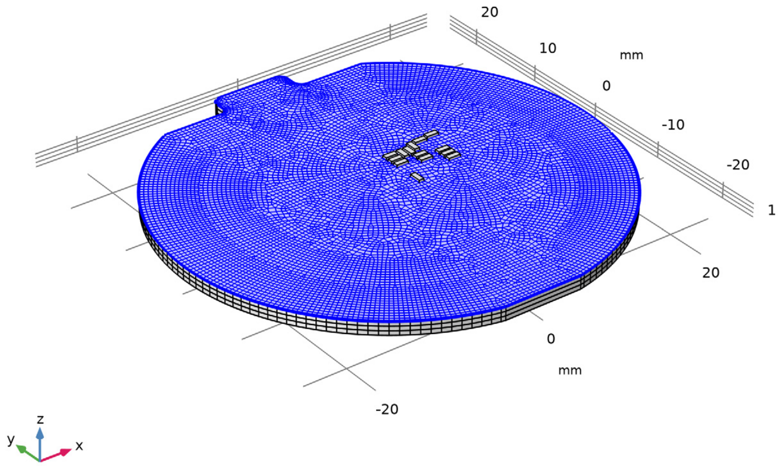
Free Quad 1

In the **Model Builder** window, expand the **Mesh 1** node.

Size 1

1 In the **Model Builder** window, expand the **Free Quad 1** node.

2 Right-click **Size 1** and choose **Build Selected**.

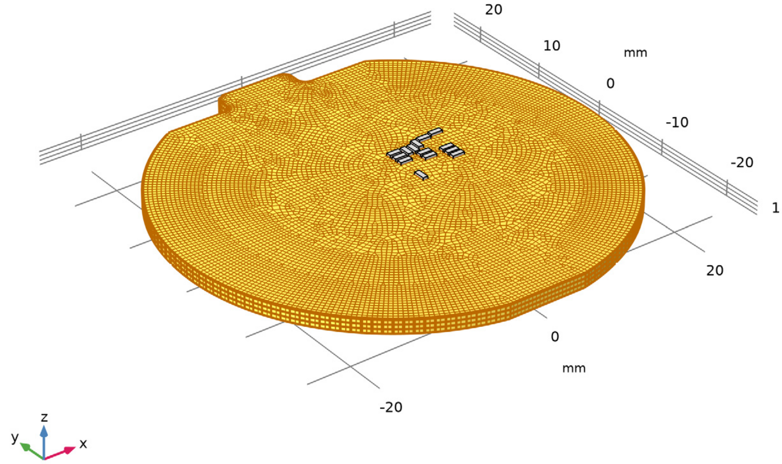


The add-in created a mesh that takes into account the 3D grid used for generating the interpolation data. Specifically, the maximum element size is configured to be five times the interpolation grid size, and the minimum element size is set to equal the grid size. You can adjust these parameters to perform a mesh sensitivity study and determine an optimal resolution for your analysis. However, it is important to note that meshing with element size finer than the underlying interpolation grid will not yield further gains in solution accuracy.

Swept 1

1 In the **Model Builder** window, expand the **Component 1 (comp1)** > **Mesh 1** > **Swept 1** node.

2 Right-click **Swept 1** and choose **Build Selected**.



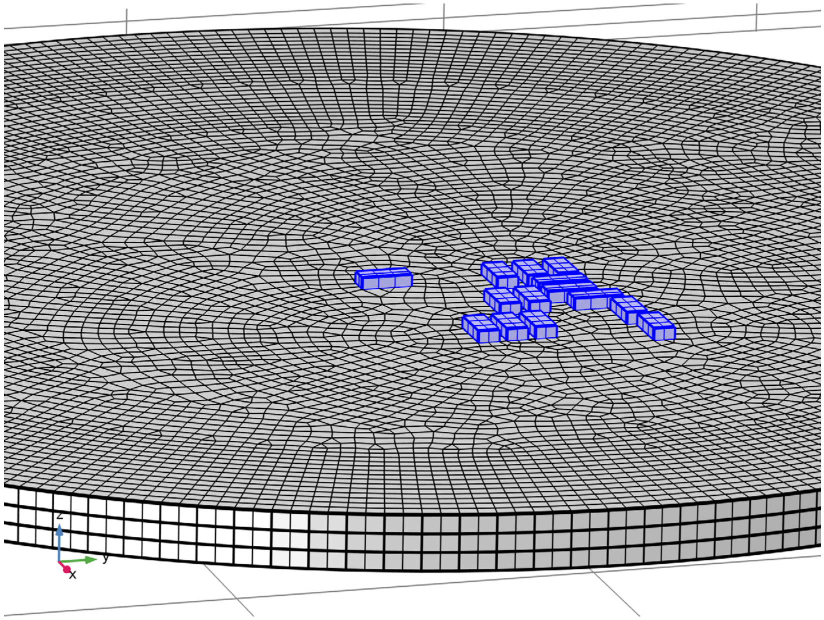
Notice that the **Swept 1** operation applied one element layer per domain. The number of elements in the sweep direction can be changed in **the Distribution 1** attribute to **Swept 1**.

Next, generate a mesh for the resistor domains that are not meshed by the add-in.

Swept 2

1 In the **Mesh** toolbar, click **Swept**.

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



STUDY 1

Step 1: Stationary

In the **Home** toolbar, click **Compute**.

RESULT TEMPLATES

To visualize the temperature distribution inside the PCB, create a multislice plot.

1 In the **Results** toolbar, click **Result Templates** to open the **Result Templates** window.

2 Go to the **Result Templates** window.

3 In the tree, select **Study 1/Solution 1 (sol1)** > **Solid Mechanics** > **Displacement (solid)** and **Study 1/Solution 1 (sol1)** > **Heat Transfer in Solids** > **Temperature, Multislice (ht)**.

4 Click the **Add Result Template** button in the window toolbar.

5 In the **Results** toolbar, click **Result Templates** to close the **Result Templates** window.

RESULTS

Multislice 1

1 In the **Model Builder** window, expand the **Temperature, Multislice (ht)** node, then click **Multislice 1**.

2 In the **Settings** window for **Multislice**, locate the **Expression** section.

3 From the **Unit** list, choose **°C**.

4 Locate the **Multiplane Data** section. Find the **X-planes** subsection. From the **Entry method** list, choose **Coordinates**.

5 In the **Coordinates** text field, type `6.35[mm]`.

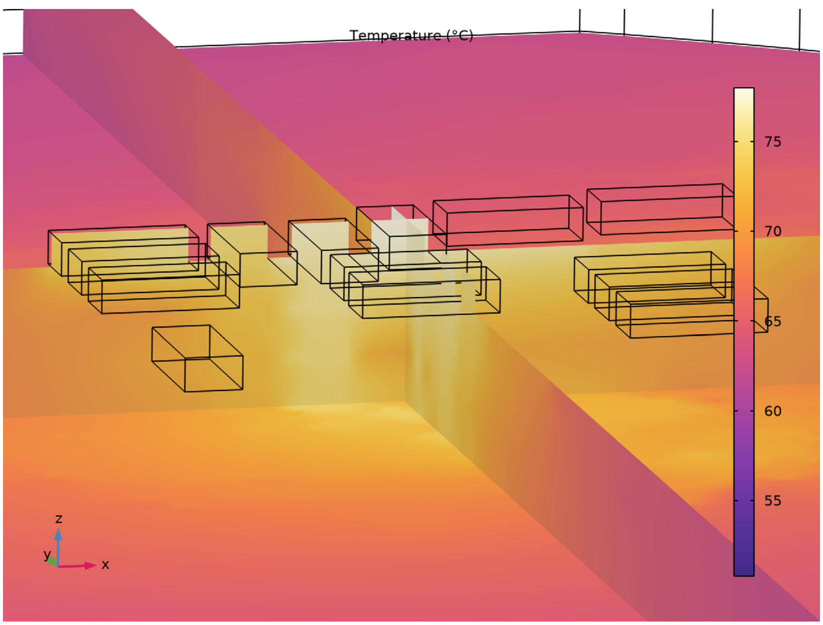
6 Find the **Y-planes** subsection. From the **Entry method** list, choose **Coordinates**.

7 In the **Coordinates** text field, type `4.4[mm]`.

8 Find the **Z-planes** subsection. From the **Entry method** list, choose **Coordinates**.

9 In the **Coordinates** text field, type `-0.0175[mm]`.

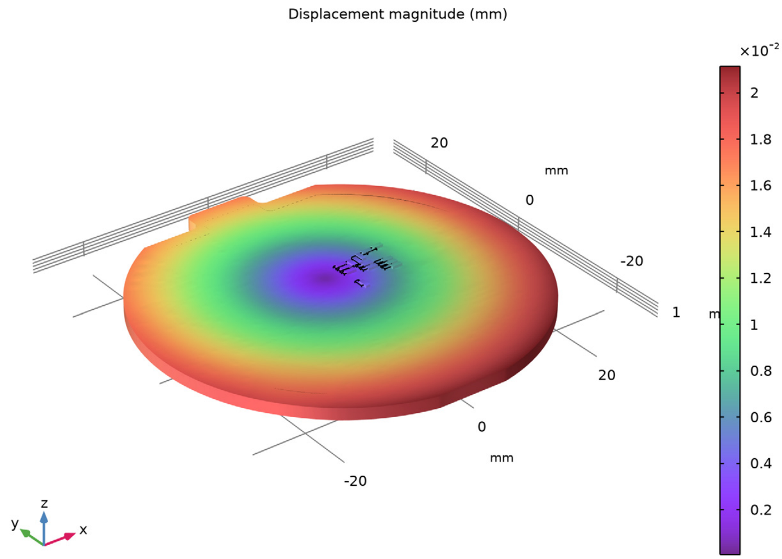
10 In the **Temperature, Multislice (ht)** toolbar, click **Plot**.



Displacement (solid)

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

2 In the **Model Builder** window, under **Results** click **Displacement (solid)**.



Material Properties Plot

Finally, create a slice plot at an elevation that corresponds to one of the interior metal layers to visualize the spatial distribution of a material property obtained by evaluating the interpolation function.

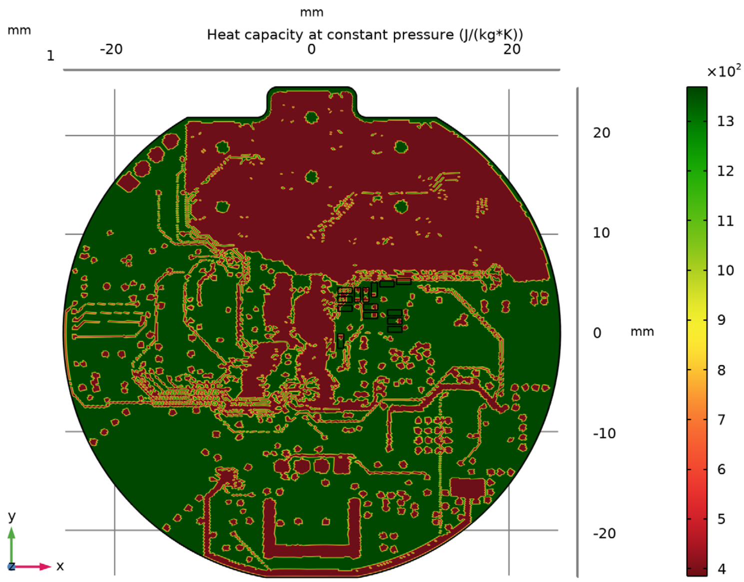
1. In the **Results** toolbar, click **3D Plot Group**.
2. In the **Settings** window for **3D Plot Group**, type `Material Properties Plot` in the **Label** text field.

Slice 1

1. Right-click **Material Properties Plot** and choose **Slice**.
2. In the **Settings** window for **Slice**, locate the **Expression** section.
3. In the **Expression** text field, type `ht.Cp`.
4. Locate the **Plane Data** section. From the **Plane** list, choose **XY-planes**.
5. From the **Entry method** list, choose **Coordinates**.
6. In the **Z-coordinates** text field, type `1[mm]`.
7. Locate the **Coloring and Style** section. From the **Color table** list, choose **Traffic**.
8. From the **Color table transformation** list, choose **Reverse**.

9 In the **Material Properties Plot** toolbar, click **Plot**.

10 Click the **Go to XY View** button in the **Graphics** toolbar.



To compare the displayed plot to the detailed geometry of the traces at the corresponding elevation, see Figure 1.