Electroplating of a Printed Circuit Board

Introduction

The printed circuit board (PCB) is the heart of almost any electronic product, carrying the components and copper wires supporting its functionality. This tutorial demonstrates how to simulate copper deposition on a PCB.

This model is also embedded in the *Printed Circuit Board Electroplating Designer* app, which adds a tailor-made user interface. In the app, functionality from the Optimization Module is also used to optimize various process parameters in order to, for instance, maximize the deposition rate for a given current distribution uniformity target. In the documentation for the app you can also read more about the PCB fabrication process in general.

The PCB pattern in the example is defined by imported ECAD files. The example requires an ECAD Module license.

Model Definition

The model uses the **Secondary Current Distribution** interface to simulate the current distribution in the deposition cell. Butler-Volmer kinetics is used on both electrode surfaces.

A Total Current boundary condition is used at the anode whereas the cathode is grounded (set to zero electronic potential).

The model geometry is shown in Figure 1. The anodes are a set of stretched blocks at the top of the geometry. The cathode is the PCB pattern located at the center bottom of the cell. An isolating screen with an aperture is placed between the anodes and the cathode to control the current distribution on the PCB. The deposition pattern also contains certain "dummy" parts (current thieves), not used in the final PCB product, that are used in order to make the deposition rate on the PCB more uniform.

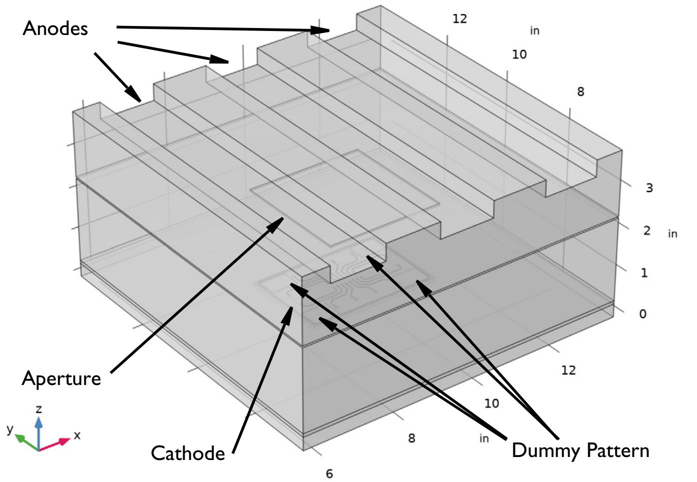

Figure 1: The model geometry.

The conductivity of the metal of the anodes and cathode is very high compared to that of the electrolyte and it is assumed that the electric potential in the metal is constant. The variations in the activation overpotential are therefore caused by the potential in the electrolyte at the surface of the electrodes. Under these assumptions, the electrodes are treated as boundaries in the simulations.

The Secondary Current Distribution interface solves for the electrolyte potential, ϕ_l (V), according to:

$$\mathbf{i}_l = -\sigma_l \nabla \phi_l$$
$$\nabla \cdot \mathbf{i}_l = 0$$

where $\mathbf{i}_l$ (A/m^2) is the electrolyte current density vector and σ_l (S/m) is the electrolyte conductivity, which is assumed to be a constant.

The default Insulation condition is used for all boundaries except the anode and cathode surfaces:

$$\mathbf{n} \cdot \mathbf{i}_l = 0$$

where $\mathbf{n}$ is the normal vector, pointing out of the domain.

The main electrode reaction on both the anode and the cathode surfaces is the copper deposition/dissolution reaction,

$$\mathrm{Cu}^{2+} + 2e^{-} \Leftrightarrow \mathrm{Cu(s)}.$$

A Butler-Volmer Expression is used to model this reaction; this sets the local current density to

$$i_{\mathrm{loc}} = i_0\left(\exp\left(\frac{\alpha_{\mathrm{a}} F\eta}{RT}\right) - \exp\left(-\frac{\alpha_{\mathrm{c}} F\eta}{RT}\right)\right)$$

Note that the local current density is positive at the anode surface and negative at the cathode surfaces, depending on the sign of the overpotential, η (V), defined as

$$\eta = \phi_s - \phi_l - E_{\mathrm{eq}} \tag{1}$$

where E_{eq} (V) is the equilibrium potential of the copper dissolution/deposition reaction and ϕ_s (V) is the potential of the electronic phase of the electrode.

On both the anode and the cathode the electrolyte current density is set to the local current density of the copper deposition reaction:

$$\mathbf{n} \cdot \mathbf{i}_l = i_{\mathrm{loc}} \tag{2}$$

The anode is grounded in the model whereas the cathode electric potential is solved for by an additional equation in order to fulfill a total current condition on the boundary according to

$$\int i_{\mathrm{loc}} = I_{\mathrm{total}} = I_{\mathrm{avg}} A_{\mathrm{cathode}} \tag{3}$$

The model is solved in a stationary study.

When postprocessing the solution the deposition thickness, s (m), at the PCB is calculated according to

$$s = \frac{i_{\mathrm{loc}}}{I_{\mathrm{avg}}} s_{\mathrm{target}} \tag{4}$$

where s_{target} (m) is the target mean deposition thickness for the whole cathode.

The time needed to achieve this thickness, t_{dep}(m), is related to s_{target} according to

$$t_{\mathrm{dep}} = s_{\mathrm{target}}\frac{nF}{I_{\mathrm{avg}}}\frac{\rho}{M} \tag{5}$$

where M is the mean molar mass (63.55 g/mol) and ρ is the density (8960 kg/m^3) of the copper atoms and n (=2) is the number of participating electrons.

Results and Discussion

Figure 2 shows the current density on the cathode, excluding the dummy pattern, for an average current density of 2 A/dm^2, and Figure 3 shows the corresponding deposited thickness for a target deposition thickness of 10 µm.

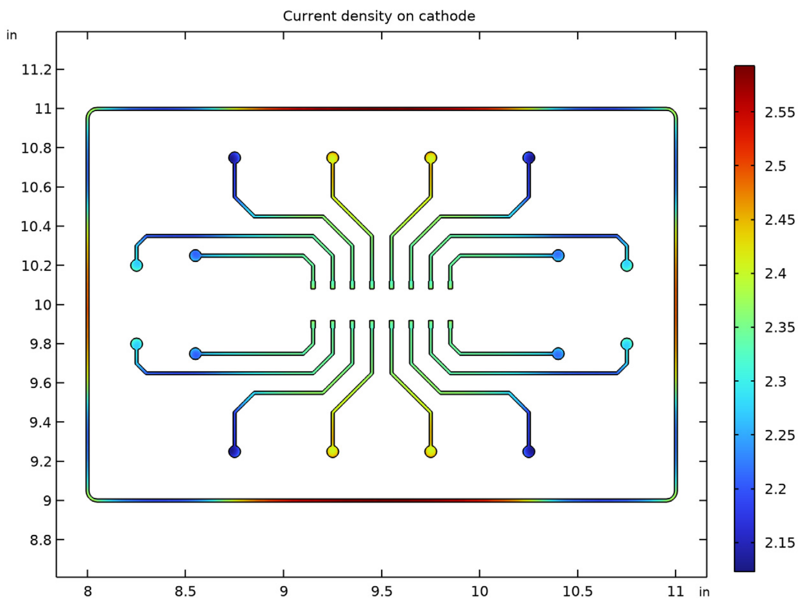

Figure 2: Current density at PCB pattern, excluding the dummy pattern (current thief).

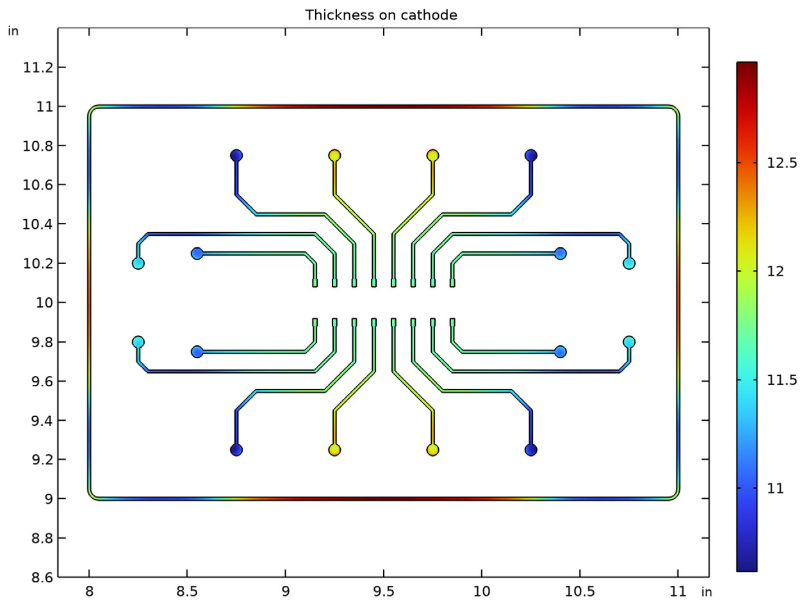

Figure 3: Deposition thickness of the PCB pattern for a target thickness of 10 µm.

Figure 4 shows the effect of the aperture on the field lines, and the thickness for the whole cathode including the dummy pattern. The deposition thickness is lower than 10 µm for the dummy parts of the PCB.

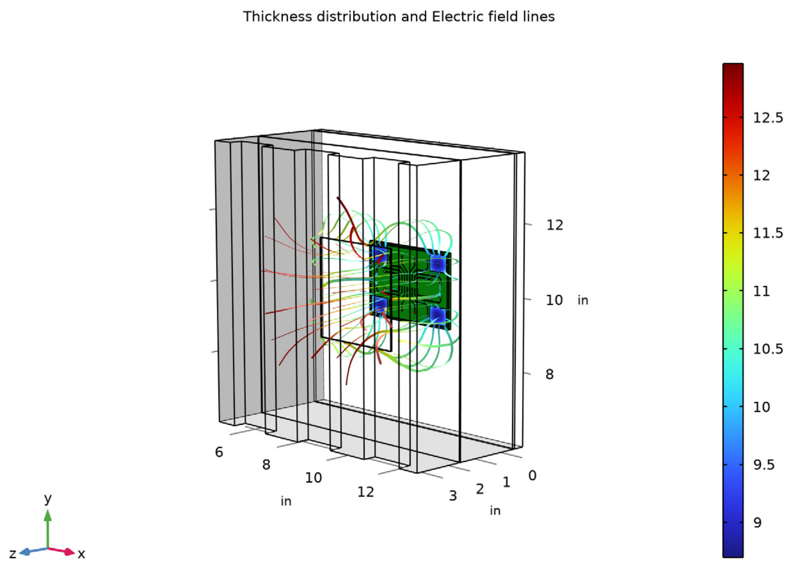

Figure 4: Field lines and thickness on the cathode, including the dummy pattern.

Application Library path: `Electrodeposition_Module/Tutorials/pcb_designer`

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 **Electrochemistry> Primary and Secondary Current Distribution>Secondary Current Distribution (cd)**.
3 Click **Add**.
4 Click **Study**.

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

6 Click **Done**.

GLOBAL DEFINITIONS

Load parameters from a file.

Parameters 1

1 In the **Model Builder** window, under **Global Definitions** click **Parameters 1**.

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

3 Click **Load from File**.

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

GEOMETRY 1

This model utilizes a premade geometry file containing a PCB pattern imported from an ECAD file. The model geometry is available as a parameterized geometry sequence in a separate MPH-file. If you want to build it from scratch, follow the instructions in the section Appendix — Geometry Modeling Instructions. Otherwise load it from file with the following steps.

1 In the **Geometry** toolbar, click **Insert Sequence** and choose **Insert Sequence**.

2 Browse to the model's Application Libraries folder and double-click the file `pcb_designer_geom_sequence.mph`.

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

Use the transparency button to see the entire geometry clearly.

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

Create some selections that will be used during model setup.

Electrolyte swept mesh region 2

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

2 In the **Settings** window for **Box Selection**, type `Electrolyte swept mesh region 2` in the **Label** text field.

3 Locate the **Box Limits** section. In the **z minimum** text field, type `0`.

4 In the **z maximum** text field, type `0`.

Electrolyte swept mesh regions

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

2 In the **Settings** window for **Union Selection**, type `Electrolyte swept mesh regions` in the **Label** text field.

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

4 In the **Add** dialog box, in the **Selections to add** list, choose **Electrolyte swept mesh region 1** and **Electrolyte swept mesh region 2**.

5 Click **OK**.

Electrolyte swept mesh regions (unisel2)

1 In the **Model Builder** window, click **Electrolyte swept mesh regions**.

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

PCB top dielectric

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

2 In the **Settings** window for **Adjacent Selection**, type `PCB top dielectric` in the **Label** text field.

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

4 In the **Add** dialog box, select **Cathode** in the **Input selections** list.

5 Click **OK**.

6 In the **Settings** window for **Adjacent Selection**, locate the **Input Entities** section.

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

8 Click **Add**.

9 In the **Add** dialog box, select **Cathode** in the **Input selections** list.

10 Click **OK**.

PCB top dielectric (adjsel1)

1 In the **Model Builder** window, click **PCB top dielectric**.

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

PCB top

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

2 In the **Settings** window for **Union Selection**, type `PCB top` 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 **Cathode** and **PCB top dielectric**.

6 Click **OK**.

PCB top (unisel3)

1 In the **Model Builder** window, click **PCB top**.

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

PCB without cathode

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

2 In the **Settings** window for **Difference Selection**, type `PCB without cathode` in the **Label** text field.

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

4 In the **Add** dialog box, select **PCB** in the **Selections to add** list.

5 Click **OK**.

6 In the **Settings** window for **Difference Selection**, locate the **Input Entities** section.

7 Click **Add**.

8 In the **Add** dialog box, select **Cathode** in the **Selections to subtract** list.

9 Click **OK**.

10 In the **Settings** window for **Difference Selection**, locate the **Geometric Entity Level** section.

11 From the **Level** list, choose **Boundary**.

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

13 In the **Add** dialog box, select **PCB** in the **Selections to add** list.

14 Click **OK**.

15 In the **Settings** window for **Difference Selection**, locate the **Input Entities** section.

16 Click **Add**.

17 In the **Add** dialog box, select **Cathode** in the **Selections to subtract** list.

18 Click **OK**.

DEFINITIONS

Add an integration coupling variable and load variables from a text file.

Integration 1 (intop1)

1 In the **Definitions** toolbar, click **Nonlocal Couplings** and choose **Integration**.

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

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

4 From the **Selection** list, choose **Cathode**.

Variables 1

1 In the **Model Builder** window, right-click **Definitions** and choose **Variables**.

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

3 Click **Load from File**.

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

MATERIALS

Add a material to specify the electrolyte conductivity.

Electrolyte

1 In the **Model Builder** window, under **Component 1 (comp1)** right-click **Materials** and choose **Blank Material**.

2 In the **Settings** window for **Material**, type `Electrolyte` in the **Label** text field.

3 Locate the **Material Contents** section. In the table, enter the following settings:

Property	Variable	Value	Unit	Property group
Electrolyte conductivity	sigmal_iso ; sigmalii = sigmal_iso, sigmalij = 0	50	S/m	Electrolyte conductivity

SECONDARY CURRENT DISTRIBUTION (CD)

Define the physics settings in the Secondary Current Distribution interface.

Electrode Surface 1

1 In the **Model Builder** window, under **Component 1 (comp1)** right-click **Secondary Current Distribution (cd)** and choose **Electrode Surface**.

2 In the **Settings** window for **Electrode Surface**, locate the **Boundary Selection** section.

3 From the **Selection** list, choose **Cathode**.

4 Locate the **Electrode Phase Potential Condition** section. From the **Electrode phase potential condition** list, choose **Total current**.

5 In the $I_{1,\mathrm{total}}$ text field, type `-ItotCathode`.

Electrode Reaction 1

1 In the **Model Builder** window, click **Electrode Reaction 1**.

2 In the **Settings** window for **Electrode Reaction**, locate the **Electrode Kinetics** section.

3 From the **Kinetics expression type** list, choose **Butler-Volmer**.

4 In the i_0 text field, type `i0`.

5 In the α_a text field, type `alphaa`.

Electrode Surface 2

1 In the **Physics** toolbar, click **Boundaries** and choose **Electrode Surface**.

2 In the **Settings** window for **Electrode Surface**, locate the **Boundary Selection** section.

3 From the **Selection** list, choose **Anode**.

Electrode Reaction 1

1 In the **Model Builder** window, click **Electrode Reaction 1**.

2 In the **Settings** window for **Electrode Reaction**, locate the **Electrode Kinetics** section.

3 From the **Kinetics expression type** list, choose **Butler-Volmer**.

4 In the i_0 text field, type `i0`.

5 In the α_a text field, type `alphaa`.

Initial Values 1

1 In the **Model Builder** window, under **Component 1 (comp1)> Secondary Current Distribution (cd)** click **Initial Values 1**.

2 In the **Settings** window for **Initial Values**, locate the **Initial Values** section.

3 In the *phil* text field, type `phil_initial`.

MESH 1

Generate the mesh as follows.

Free Triangular 1

In the **Mesh** toolbar, click **Boundary** and choose **Free Triangular**.

Size

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

2 In the **Settings** window for **Size**, locate the **Element Size** section.

3 From the **Predefined** list, choose **Fine**.

Free Triangular 1

1 In the **Model Builder** window, click **Free Triangular 1**.

2 In the **Settings** window for **Free Triangular**, locate the **Boundary Selection** section.

3 From the **Selection** list, choose **PCB top**.

Size 1

1 Right-click **Free Triangular 1** and choose **Size**.

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

3 From the **Selection** list, choose **Cathode**.

4 Locate the **Element Size** section. From the **Predefined** list, choose **Extra fine**.

Swept 1

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

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

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

4 From the **Selection** list, choose **Electrolyte swept mesh regions**.

5 Click to expand the **Source Faces** section. Click **Paste Selection**.

6 In the **Paste Selection** dialog box, type `9` in the **Selection** text field.

7 Click **OK**.

8 In the **Settings** window for **Swept**, click to expand the **Sweep Method** section.

9 From the **Face meshing method** list, choose **Triangular (generate prisms)**.

Size 1

1 Right-click **Swept 1** and choose **Size**.

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

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

4 Click **Paste Selection**.

5 In the **Paste Selection** dialog box, type `9` in the **Selection** text field.

6 Click **OK**.

7 In the **Settings** window for **Size**, locate the **Element Size** section.

8 From the **Predefined** list, choose **Finer**.

Distribution 1

1 In the **Model Builder** window, right-click **Swept 1** and choose **Distribution**.

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

3 In the **Number of elements** text field, type `round((PCBThickness/1.5[mm]>=1)*PCBThickness/1.5[mm]+(PCBThickness/1.5[mm]<1),0)`.

Distribution 2

1 Right-click **Swept 1** and choose **Distribution**.

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

3 From the **Selection** list, choose **Electrolyte swept mesh region 2**.

4 Locate the **Distribution** section. In the **Number of elements** text field, type `((PCBOffset-PCBThickness)/2[mm]>=1)*(PCBOffset-PCBThickness)/2[mm]+((PCBOffset-PCBThickness)/2[mm]<1)`.

Swept 2

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

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

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

4 From the **Selection** list, choose **Aperture**.

5 Locate the **Source Faces** section. From the **Selection** list, choose **Aperture source**.

6 Locate the **Sweep Method** section. From the **Face meshing method** list, choose **Triangular (generate prisms)**.

Size 1

1 Right-click **Swept 2** and choose **Size**.

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 **Aperture source**.

5 Locate the **Element Size** section. From the **Predefined** list, choose **Finer**.

Distribution 1

1 In the **Model Builder** window, right-click **Swept 2** and choose **Distribution**.

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

3 In the **Number of elements** text field, type `(ApertureThickness/1.5[mm]>=1)*ApertureThickness/1.5[mm]+(ApertureThickness/1.5[mm]<1)`.

Free Tetrahedral 1

1 In the **Mesh** toolbar, click **Free Tetrahedral**.

2 In the **Model Builder** window, right-click **Mesh 1** and choose **Build All**.

STUDY 1

Finally, compute the results.

Solution 1 (sol1)

1 In the **Study** toolbar, click **Show Default Solver**.

2 In the **Model Builder** window, expand the **Solution 1 (sol1)** node, then click **Stationary Solver 1**.

3 In the **Settings** window for **Stationary Solver**, locate the **General** section.

4 In the **Relative tolerance** text field, type `1e-6`.

5 In the **Model Builder** window, click **Study 1**.

6 In the **Settings** window for **Study**, locate the **Study Settings** section.

7 Clear the **Generate default plots** check box.

8 In the **Study** toolbar, click **=** **Compute**.

RESULTS

Create some datasets that will be used during postprocessing.

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

Cathode

1 In the **Model Builder** window, expand the **Results>Datasets** node.

2 Right-click **Results>Datasets** and choose **Surface**.

3 In the **Settings** window for **Surface**, type `Cathode` in the **Label** text field.

4 Locate the **Parameterization** section. From the **x- and y-axes** list, choose **Expression**.

5 Locate the **Selection** section. From the **Selection** list, choose **Cathode**.

Cathode copper layout

1 In the **Results** toolbar, click **More Datasets** and choose **Surface**.

2 In the **Settings** window for **Surface**, type `Cathode copper layout` in the **Label** text field.

3 Locate the **Parameterization** section. From the **x- and y-axes** list, choose **Expression**.

4 Locate the **Selection** section. From the **Selection** list, choose **PCB copper layout**.

PCB without cathode

1 In the **Results** toolbar, click **More Datasets** and choose **Surface**.

2 In the **Settings** window for **Surface**, type `PCB without cathode` in the **Label** text field.

3 Locate the **Selection** section. From the **Selection** list, choose **PCB without cathode**.

Walls

1 In the **Results** toolbar, click **More Datasets** and choose **Surface**.

2 In the **Settings** window for **Surface**, type `Walls` in the **Label** text field.

3 Locate the **Selection** section. Click **Paste Selection**.

4 In the **Paste Selection** dialog box, type `1-5, 7-8, 11-12` in the **Selection** text field.

5 Click **OK**.

Thickness on cathode

First, plot the thickness on the cathode copper layout.

1 In the **Results** toolbar, click **2D Plot Group**.
2 In the **Settings** window for **2D Plot Group**, type `Thickness on cathode` in the **Label** text field.
3 Locate the **Data** section. From the **Dataset** list, choose **Cathode copper layout**.

Surface 1

1 Right-click **Thickness on cathode** and choose **Surface**.
2 In the **Settings** window for **Surface**, click **Replace Expression** in the upper-right corner of the **Expression** section. From the menu, choose **Component 1 (comp1)>Definitions>Variables>thickness_cathode - Thickness on cathode - m**.
3 Locate the **Expression** section. From the **Unit** list, choose **µm**.

Thickness on cathode

1 In the **Model Builder** window, click **Thickness on cathode**.
2 In the **Settings** window for **2D Plot Group**, click to expand the **Title** section.
3 From the **Title type** list, choose **Label**.

Current density on cathode

Next, plot the current density on the cathode copper layout.

1 Right-click **Thickness on cathode** and choose **Duplicate**.
2 In the **Settings** window for **2D Plot Group**, type `Current density on cathode` in the **Label** text field.

Surface 1

1 In the **Model Builder** window, expand the **Thickness on cathode 1** node, then click **Results>Current density on cathode>Surface 1**.
2 In the **Settings** window for **Surface**, click **Replace Expression** in the upper-right corner of the **Expression** section. From the menu, choose **Component 1 (comp1)>Secondary Current Distribution>Electrode kinetics>cd.iloc_er1 - Local current density - A/m²**.
3 Locate the **Expression** section. In the **Expression** text field, type `-cd.iloc_er1`.
4 In the **Unit** field, type `A/dm^2`.
5 Select the **Description** check box.
6 In the associated text field, type `Current Density on Cathode`.

Current density on cathode

1 In the **Model Builder** window, click **Current density on cathode**.

2 In the **Current density on cathode** toolbar, click **Plot**.

Thickness distribution and Electric field lines

Next, plot the thickness distribution and the electric field lines.

1 In the **Home** toolbar, click **Add Plot Group** and choose **3D Plot Group**.

2 In the **Settings** window for **3D Plot Group**, type `Thickness distribution and Electric field lines` in the **Label** text field.

Surface 1

1 Right-click **Thickness distribution and Electric field lines** and choose **Surface**.

2 In the **Settings** window for **Surface**, locate the **Data** section.

3 From the **Dataset** list, choose **Cathode**.

4 Locate the **Expression** section. In the **Expression** text field, type `thickness_cathode`.

5 From the **Unit** list, choose **µm**.

Surface 2

1 In the **Model Builder** window, right-click **Thickness distribution and Electric field lines** and choose **Surface**.

2 In the **Settings** window for **Surface**, locate the **Data** section.

3 From the **Dataset** list, choose **PCB without cathode**.

4 Locate the **Coloring and Style** section. From the **Coloring** list, choose **Uniform**.

5 From the **Color** list, choose **Custom**.

6 On Windows, click the colored bar underneath, or — if you are running the cross-platform desktop — the **Color** button.

7 Click **Define custom colors**.

8 Set the RGB values to 9, 118, and 9, respectively.

9 Click **Add to custom colors**.

10 Click **Show color palette only** or **OK** on the cross-platform desktop.

Surface 3

1 Right-click **Thickness distribution and Electric field lines** and choose **Surface**.

2 In the **Settings** window for **Surface**, locate the **Data** section.

3 From the **Dataset** list, choose **Walls**.

4 Locate the **Expression** section. In the **Expression** text field, type `1`.

5 Locate the **Coloring and Style** section. From the **Coloring** list, choose **Uniform**.

6 From the **Color** list, choose **White**.

Streamline 1

1 Right-click **Thickness distribution and Electric field lines** and choose **Streamline**.

2 In the **Settings** window for **Streamline**, locate the **Streamline Positioning** section.

3 In the **Number** text field, type 50.

4 Locate the **Selection** section. From the **Selection** list, choose **Cathode**.

5 Locate the **Coloring and Style** section. Find the **Line style** subsection. From the **Type** list, choose **Ribbon**.

Color Expression 1

1 Right-click **Streamline 1** and choose **Color Expression**.

2 In the **Settings** window for **Color Expression**, locate the **Coloring and Style** section.

3 Clear the **Color legend** check box.

Thickness distribution and Electric field lines

1 In the **Model Builder** window, under **Results** click **Thickness distribution and Electric field lines**.

2 In the **Settings** window for **3D Plot Group**, click to expand the **Title** section.

3 From the **Title type** list, choose **Label**.

4 In the **Thickness distribution and Electric field lines** toolbar, click **Plot**.

Appendix — Geometry 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**.

GLOBAL DEFINITIONS

Parameters 1

1 In the **Model Builder** window, under **Global Definitions** click **Parameters 1**.

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

3 Click **Load from File**.

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

GEOMETRY 1

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

2 In the **Settings** window for **Geometry**, locate the **Units** section.

3 From the **Length unit** list, choose **in**.

PCB

1 In the **Geometry** toolbar, click **Block**.

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

3 Locate the **Size and Shape** section. In the **Width** text field, type `PCBWidth+2*PCBMargin`.

4 In the **Depth** text field, type `PCBHeight+2*PCBMargin`.

5 In the **Height** text field, type `PCBThickness`.

6 Locate the **Position** section. In the **x** text field, type `PCBxMin-PCBMargin`.

7 In the **y** text field, type `PCByMin-PCBMargin`.

8 In the **z** text field, type `PCBOffset-PCBThickness`.

9 Locate the **Selections of Resulting Entities** section. Select the **Resulting objects selection** check box.

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

Work Plane 1 (wp1)

1 In the **Geometry** toolbar, click **Work Plane**.

2 In the **Settings** window for **Work Plane**, locate the **Plane Definition** section.

3 In the **z-coordinate** text field, type `PCBOffset`.

4 Locate the **Selections of Resulting Entities** section. Find the **Cumulative selection** subsection. Click **New**.

5 In the **New Cumulative Selection** dialog box, type `PCB copper layout` in the **Name** text field.

6 Click **OK**.

7 In the **Settings** window for **Work Plane**, click **Show Work Plane**.

Work Plane 1 (wp1)>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 `example_pcb.tgz`.

5 Click **Import**.

6 Find the **Layers to import** subsection. In the table, clear the **Import** check box for **Dielectric**.

GEOMETRY 1

Work Plane 1 (wp1)

In the **Model Builder** window, collapse the **Component 1 (comp1)>Geometry 1>Work Plane 1 (wp1)** node.

If 1 (if1)

1 In the **Model Builder** window, right-click **Geometry 1** and choose **Programming>If + End If**.

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

3 In the **Condition** text field, type `UseDummy`.

Work Plane 2 (wp2)

1 In the **Geometry** toolbar, click **Work Plane**.

2 In the **Settings** window for **Work Plane**, locate the **Plane Definition** section.

3 In the **z-coordinate** text field, type `PCBOffset`.

4 Locate the **Selections of Resulting Entities** section. Find the **Cumulative selection** subsection. Click **New**.

5 In the **New Cumulative Selection** dialog box, type `PCB dummy layout` in the **Name** text field.

6 Click **OK**.

7 In the **Settings** window for **Work Plane**, click **Show Work Plane**.

Work Plane 2 (wp2)>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 `example_pcb_dummy_pattern.tgz`.

5 Click **Import**.

6 Find the **Layers to import** subsection. In the table, clear the **Import** check box for **Dielectric**.

End If 1 (endif1)

1 In the **Model Builder** window, under **Component 1 (comp1)>Geometry 1** click **End If 1 (endif1)**.

2 In the **Settings** window for **End If**, click **Build All Objects**.

Cathode

1 In the **Model Builder** window, right-click **Geometry 1** and choose **Selections>Union Selection**.

2 In the **Settings** window for **Union Selection**, type `Cathode` 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 **PCB copper layout** and **PCB dummy layout**.

6 Click **OK**.

Bath

1 In the **Geometry** toolbar, click **Block**.

2 In the **Settings** window for **Block**, type `Bath` in the **Label** text field.

3 Locate the **Size and Shape** section. In the **Width** text field, type `BathWidth`.

4 In the **Depth** text field, type `BathHeight`.

5 In the **Height** text field, type `BathDepth`.

6 Locate the **Position** section. In the **x** text field, type `PCBxMin-(BathWidth-PCBWidth)/2`.

7 In the **y** text field, type `PCByMin-(BathHeight-PCBHeight)/2`.

8 Locate the **Selections of Resulting Entities** section. Select the **Resulting objects selection** check box.

Electrolyte swept mesh region 1

1 In the **Geometry** toolbar, click **Block**.

2 In the **Settings** window for **Block**, type `Electrolyte swept mesh region 1` in the **Label** text field.

3 Locate the **Size and Shape** section. In the **Width** text field, type `BathWidth`.

4 In the **Depth** text field, type `BathHeight`.

5 In the **Height** text field, type `PCBThickness`.

6 Locate the **Position** section. In the **x** text field, type `PCBxMin-(BathWidth-PCBWidth)/2`.

7 In the **y** text field, type `PCByMin-(BathHeight-PCBHeight)/2`.

8 In the **z** text field, type `PCBOffset-PCBThickness`.

9 Locate the **Selections of Resulting Entities** section. Select the **Resulting objects selection** check box.

Work Plane 3 (wp3)

1 In the **Geometry** toolbar, click **Work Plane**.

2 In the **Settings** window for **Work Plane**, locate the **Plane Definition** section.

3 From the **Plane type** list, choose **Face parallel**.

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

5 On the object **blk2**, select Boundary 4 only.

6 Click **Show Work Plane**.

Work Plane 3 (wp3)>Rectangle 1 (r1)

1 In the **Work Plane** toolbar, click **Rectangle**.

2 In the **Settings** window for **Rectangle**, locate the **Size and Shape** section.

3 In the **Width** text field, type `BathWidth/6`.

4 In the **Height** text field, type `BathHeight`.

5 Locate the **Position** section. In the **xw** text field, type `-BathWidth/2+BathWidth/6/2`.

6 In the **yw** text field, type `-BathHeight/2`.

Work Plane 3 (wp3)>Array 1 (arr1)

1 In the **Work Plane** toolbar, click **Transforms** and choose **Array**.

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

3 Select the object **r1** only.

4 In the **Settings** window for **Array**, locate the **Size** section.

5 In the **xw size** text field, type 3.

6 Locate the **Displacement** section. In the **xw** text field, type BathWidth/3.

GEOMETRY 1

Work Plane 3 (wp3)

In the **Model Builder** window, collapse the **Component 1 (comp1)>Geometry 1>Work Plane 3 (wp3)** node.

Anode

1 In the **Geometry** toolbar, click **Extrude**.

2 In the **Settings** window for **Extrude**, type Anode in the **Label** text field.

3 Locate the **Distances** section. In the table, enter the following settings:

Distances (in)
AnodeThickness

4 Select the **Reverse direction** check box.

5 Locate the **Selections of Resulting Entities** section. Select the **Resulting objects selection** check box.

6 From the **Show in physics** list, choose **Boundary selection**.

Difference 1 (dif1)

1 Right-click **Geometry 1** and choose **Booleans and Partitions>Difference**.

2 Select the objects **blk2** and **blk3** only.

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

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

5 From the **Objects to subtract** list, choose **PCB**.

6 Select the objects **blk1** and **ext1** only.

If 2 (if2)

1 In the **Geometry** toolbar, click **Programming** and choose **If + End If**.

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

3 In the **Condition** text field, type UseAperture.

Work Plane 4 (wp4)

1 In the **Geometry** toolbar, click **Work Plane**.

2 In the **Settings** window for **Work Plane**, locate the **Plane Definition** section.

3 In the **z-coordinate** text field, type `ApertureOffset+PCBOffset`.

4 Click **Show Work Plane**.

Work Plane 4 (wp4)>Cross Section 1 (cro1)

1 In the **Work Plane** toolbar, click **Cross Section**.

2 In the **Settings** window for **Cross Section**, locate the **Cross Section** section.

3 From the **Intersect** list, choose **Selected objects**.

4 Find the **Objects to intersect** subsection. Click to select the **Activate Selection** toggle button.

5 From the **Objects to intersect** list, choose **Bath**.

GEOMETRY 1

Work Plane 4 (wp4)

1 In the **Model Builder** window, collapse the **Component 1 (comp1)>Geometry 1>Work Plane 4 (wp4)** node.

2 In the **Model Builder** window, click **Work Plane 4 (wp4)**.

3 In the **Settings** window for **Work Plane**, click **Build Selected**.

Extrude 2 (ext2)

1 In the **Model Builder** window, right-click **Geometry 1** and choose **Extrude**.

2 In the **Settings** window for **Extrude**, locate the **Distances** section.

3 In the table, enter the following settings:

Distances (in)
`ApertureThickness`

4 Click **Build Selected**.

Difference 2 (dif2)

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

2 Select the object **dif1** only.

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

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

5 Select the object **ext2** only.

6 Click **Build Selected**.

Aperture source

1 In the **Geometry** toolbar, click **Work Plane**.

2 In the **Settings** window for **Work Plane**, type `Aperture source` in the **Label** text field.

3 Locate the **Plane Definition** section. In the **z-coordinate** text field, type `ApertureOffset+PCBOffset`.

4 Locate the **Selections of Resulting Entities** section. Select the **Resulting objects selection** check box.

5 Click **Show Work Plane**.

Aperture source (wp5)>Rectangle 1 (r1)

1 In the **Work Plane** toolbar, click **Rectangle**.

2 In the **Settings** window for **Rectangle**, locate the **Size and Shape** section.

3 In the **Width** text field, type `ApertureWidth`.

4 In the **Height** text field, type `ApertureHeight`.

5 Locate the **Position** section. In the **xw** text field, type `PCBxMin-(BathWidth-PCBWidth)/2+(BathWidth-ApertureWidth)/2`.

6 In the **yw** text field, type `PCByMin-(BathHeight-PCBHeight)/2+(BathHeight-ApertureHeight)/2`.

Aperture source (wp5)

1 In the **Model Builder** window, collapse the **Component 1 (comp1)>Geometry 1>Aperture source (wp5)** node.

2 In the **Model Builder** window, click **Aperture source (wp5)**.

3 In the **Settings** window for **Work Plane**, click **Build Selected**.

Aperture

1 In the **Geometry** toolbar, click **Extrude**.

2 In the **Settings** window for **Extrude**, type `Aperture` in the **Label** text field.

3 Locate the **Distances** section. In the table, enter the following settings:

Distances (in)
`ApertureThickness`

4 Locate the **Selections of Resulting Entities** section. Select the **Resulting objects selection** check box.

Form Union (fin)

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