# Component thermal modeling in SolariaPCB

Contents

There are many ways to represent a component in SolariaPCB. It all depends on how accurate you want to represent it.

1. How to define component thermal representation

The thermal representation of a component is defined in the Component Thermal Properties dialog. Thermal contact between the component case and the board surface can be calculated by the software if lead/pad/ball geometry and material is defined in the Component Geometry dialog. These dialogs can be accessed several ways.

a. PCB>Define Active Components

For a specific component, if you click on the component Thermal or Geometry cell a button will appear. If you click on the button the Thermal or Geometry dialog for that component will come up. To define the lead, pad or ball material, select the ^^ cell for that component. When enough information has been defined to thermally characterize that component, the cell will turn green.
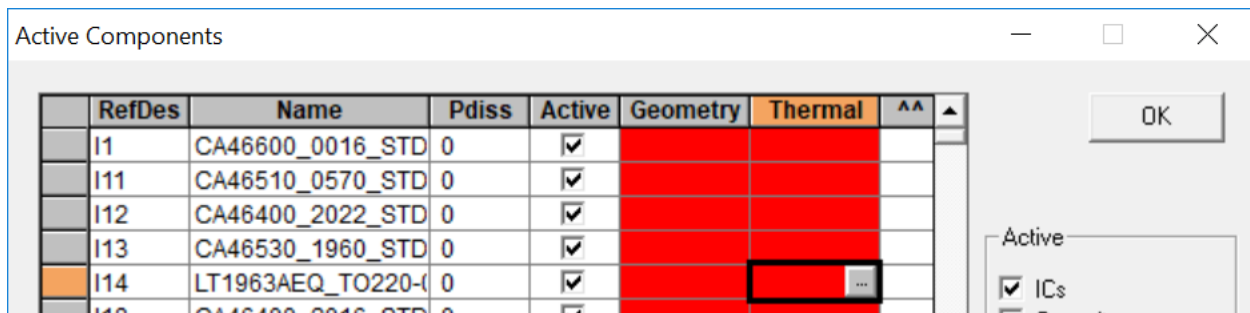
b. Edit a specific component

Select PCB>Edit>Edit component on PCB. Select the component then hit the Working Set button.

c. Add Component To Lib dialog

The PCB>Add Components To Lib dialog, in addition to adding Master Library components to the board library and adding board library components to the Master Library, it can also be used to access the Thermal and Geometry dialogs for a specific component. Select the component on the right side then hit the Edit button.

2. Defining Active components

Components that are thermally important to the simulation should be defined as active. Important components are those where you spend the time to define a thermal resistance and power dissipation. The shape of all Active components are meshed into the PCB. This is needed to accurately simulate them. Therefore, non-critical components, typically like resistors, capacitors, test points and connectors should NOT be defined as active. If they are

then the mesh of the PCB will be unnecessarily large. Components are defined as Active in the PCB>Define Active Components dialog.

3. Defining a component thermal representation

a. The simplest representation

At a minimum, the local board surface temperature will be given for every component on the board. This is independent is it is defined as Active or has a power dissipation defined. If you do define a power dissipation for a component that is not defined as Active, the power dissipation will be added to the board surface at the component location. The shape of non-Active components is not meshed into the board.

b. Defining a Case Node

In the Component Thermal Properties dialog, you can add a single Node and define it as CASE. You should define the component lead/pad/ball material and geometry. The thermal resistance from the Case to the board surface will be calculated using this information and the Case node will be thermally connected to the PCB surface. In this case, the component should be defined as Active. The component power dissipation will be applied to the Case node.

c. Defining Case and Board Nodes

If you know the Case to Board thermal resistance, the create CASE and BOARD nodes. Then create a CASE to BOARD resistance. In this case, the component should be defined as Active. The component power dissipation will be applied to the Case node. No calculations will be done in this scenario.

d. Defining Junction and Case Nodes

If you have a Junction to Case thermal resistance for a component, then create a JUNCTION and a CASE node. Then create a JUNCTION to CASE resistance. In this case, the component should be defined as Active. The component power dissipation will be applied to the Junction node. You should define the component lead/pad/ball material and geometry. The thermal resistance from the Case to the board surface will be calculated using this information and the Case node will be thermally connected to the PCB surface.

e. Defining other thermal representations

If you have a different thermal representation, then you can just create the Nodes needed and the Resistances between the nodes and SolariaPCB will create them in the Solaria model.

f. DELPHI component thermal model

Five nodes and eight resistors component model

	Junction	Bo	Bi	To	Ti
Junction		6.27	0.7	0.75	0.001
Bo			15.57	8.33	
Bi				4.58	2.74

In the Component Thermal Properties dialog, five Nodes are generated, and eight Resistors are generated as shown below.

Area under the die must be defined. If not, defaults to 0.5 of the package area.


4

Below shows the Solaria thermal model that is generated from the above definition.
Convection elements are generated to represent the thermal contact between the component
inside and outside areas to the PCB surface.

Convection to outer area

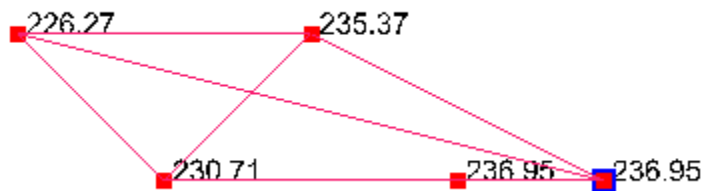BOTTOM_OUTER

BOTTOM_INNER

Convection to inner area

Z
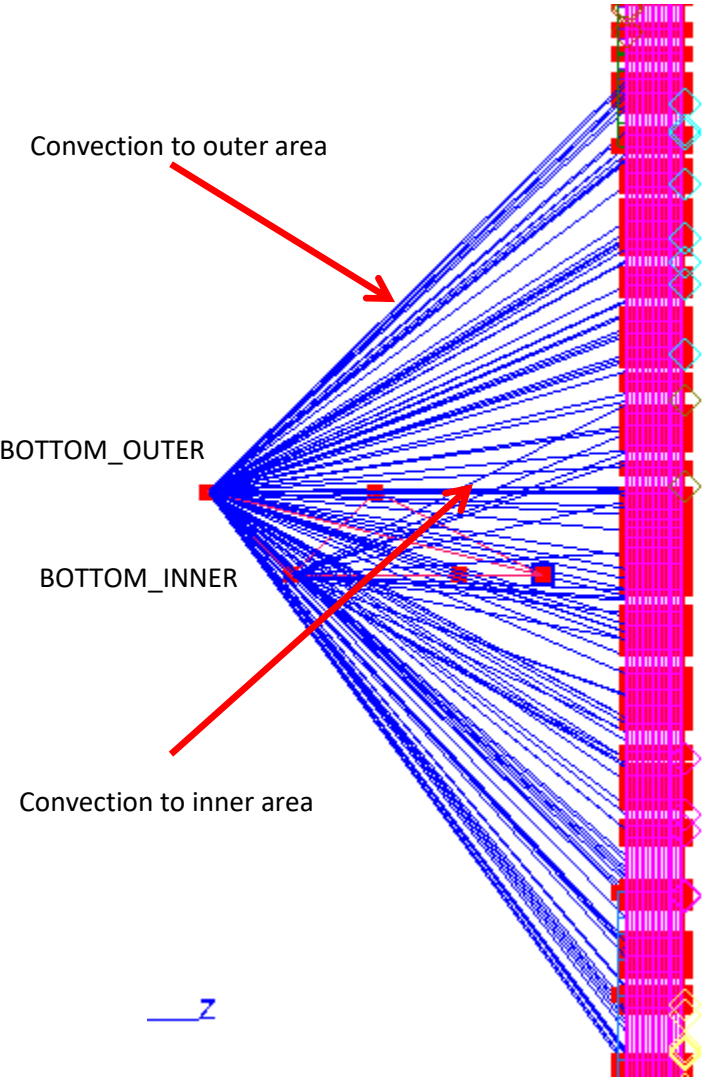
The component outline as well as the inner area of the component are meshed into the PCB to ensure accurate contact areas are used.
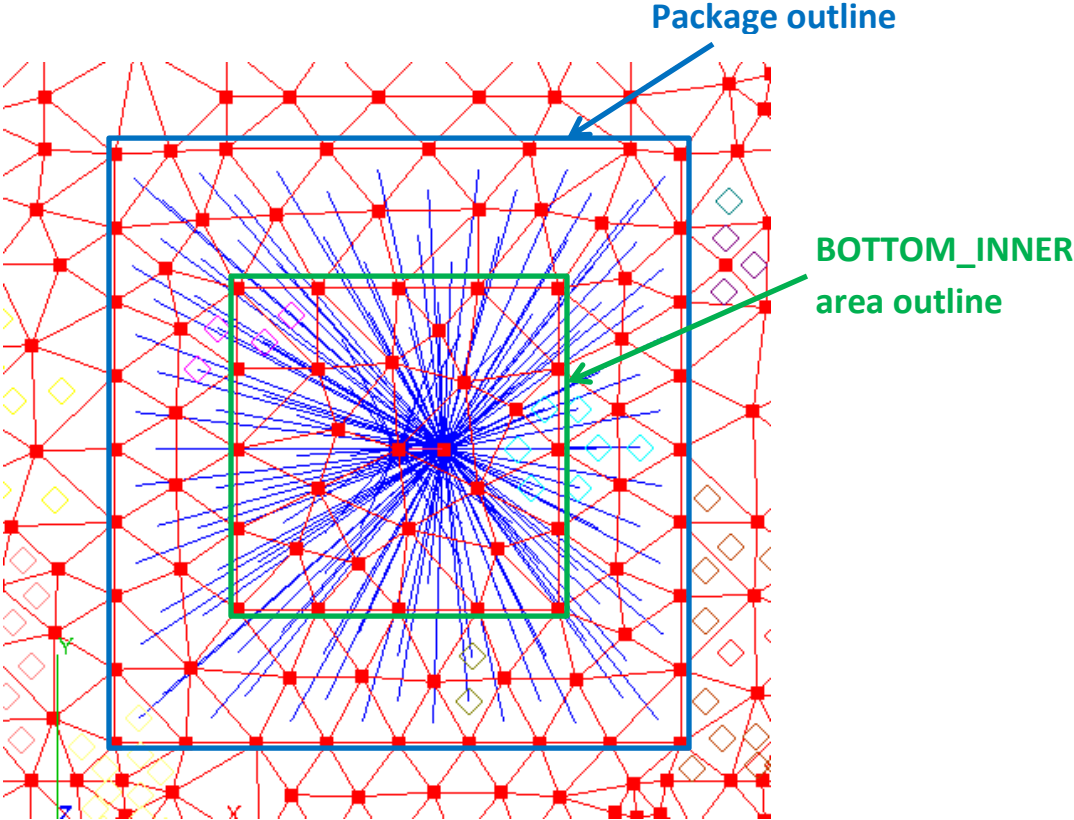
Below shows the Nodes and Resistors that are generated.
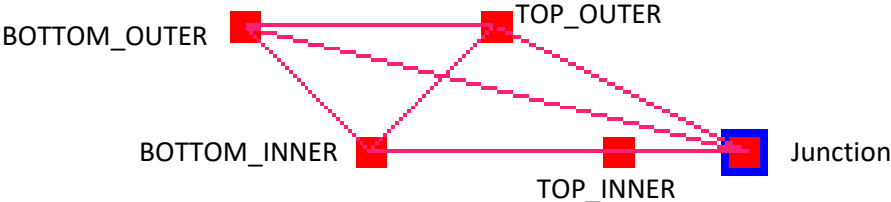
Below are node temperatures displayed in Solaria.


4. A component sitting on a Stiffener

Often a component that dissipated a lot of heat is placed on a metallic rail to help heat transfer to a side wall. A Stiffener in SolariaPCB is used to represent the metallic rail. Below shows a component, blue, sitting on a Stiffener, gray.
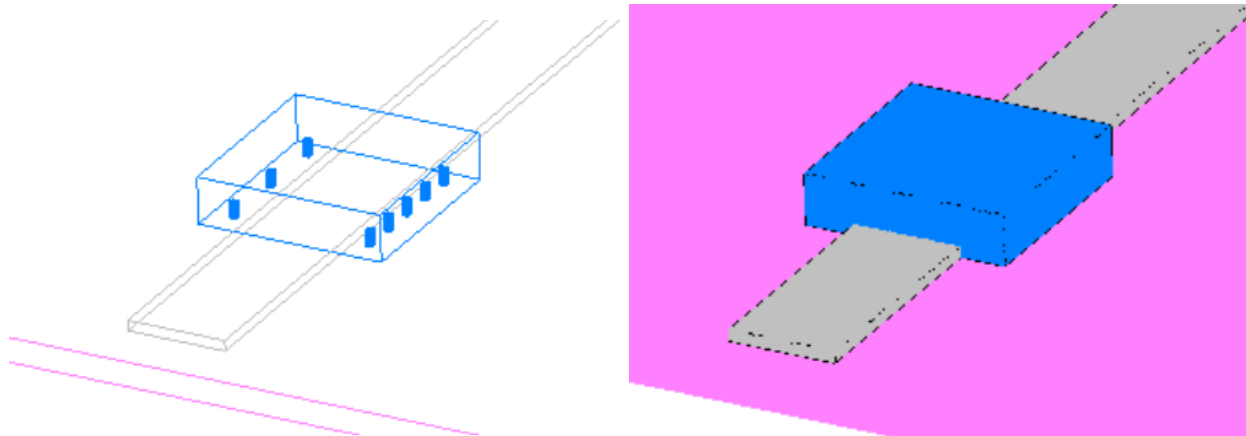

Shown in the dialog below, the component is bonded to the PCB. The thickness of the bond is the Clearance value in the component Geometry. But since the Stiffener is under the Component, when creating the Solaria model, SolariaPCB recognizes this and attaches the component to the top of the Stiffener.
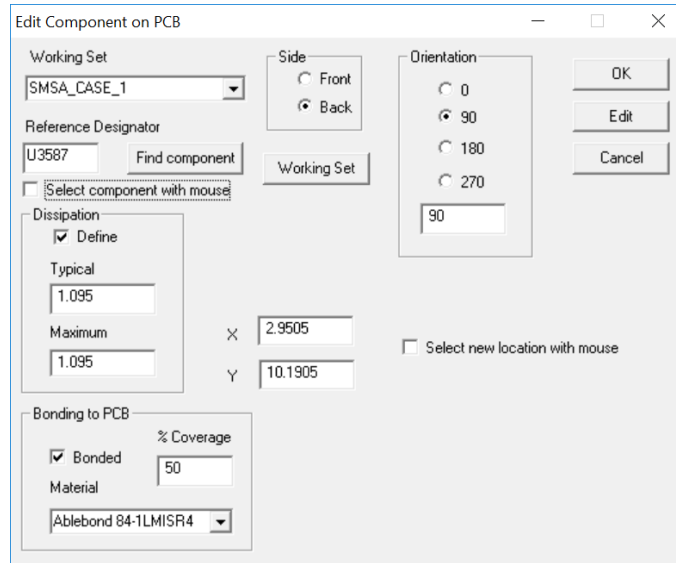
The figure below shows a side view of the component in Solaria. The purple is the Stiffener. The horizontal pink lines are the board layers. The lines from the Case node to the Stiffener and the PCB surface are Convection elements representing thermal contact due to the bond to the Stiffener and the leads to the PCB.
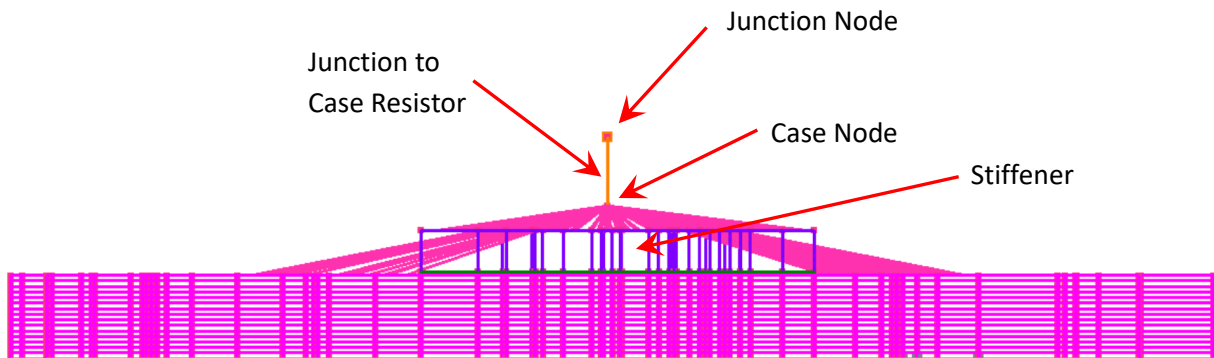

Below shows a 3D view of the component.
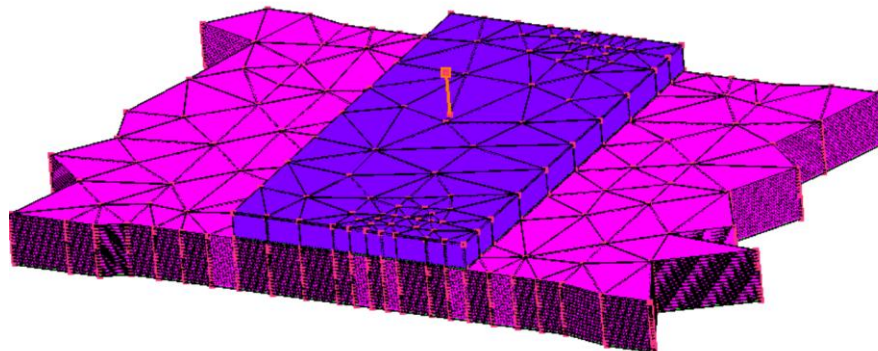
The figure below shows only Convection elements with the setting in Settings>General for Surface Couplings checked. Instead of drawing lines between the Node and surfaces, convection surfaces are drawn. You can see Convection elements connect the component case to the top of the Stiffener as well as the surface of the PCB.
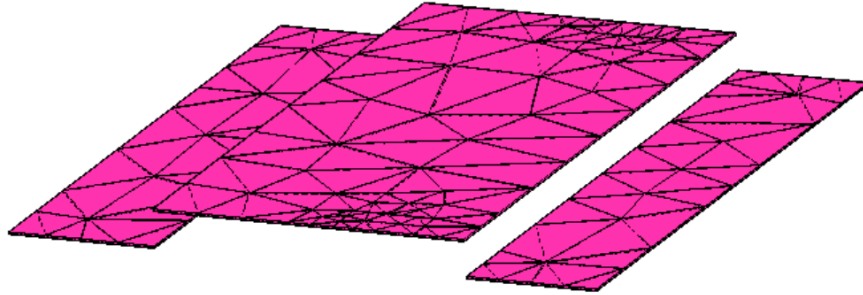
5. A component sitting in a Cutout

There are times when part of the PCB is cut away to directly attach a component to a layer inside the PCB. This could be a power or ground layer or it could be a metal plate bonded between two boards.

Below shows the PCB>Add Cutout dialog where a Cutout is created using the Diagonal Points option. Since the component is on the backside of the PCB, the Cutout starts at the Back side of the PCB and extends to Node 12 in the PCB.
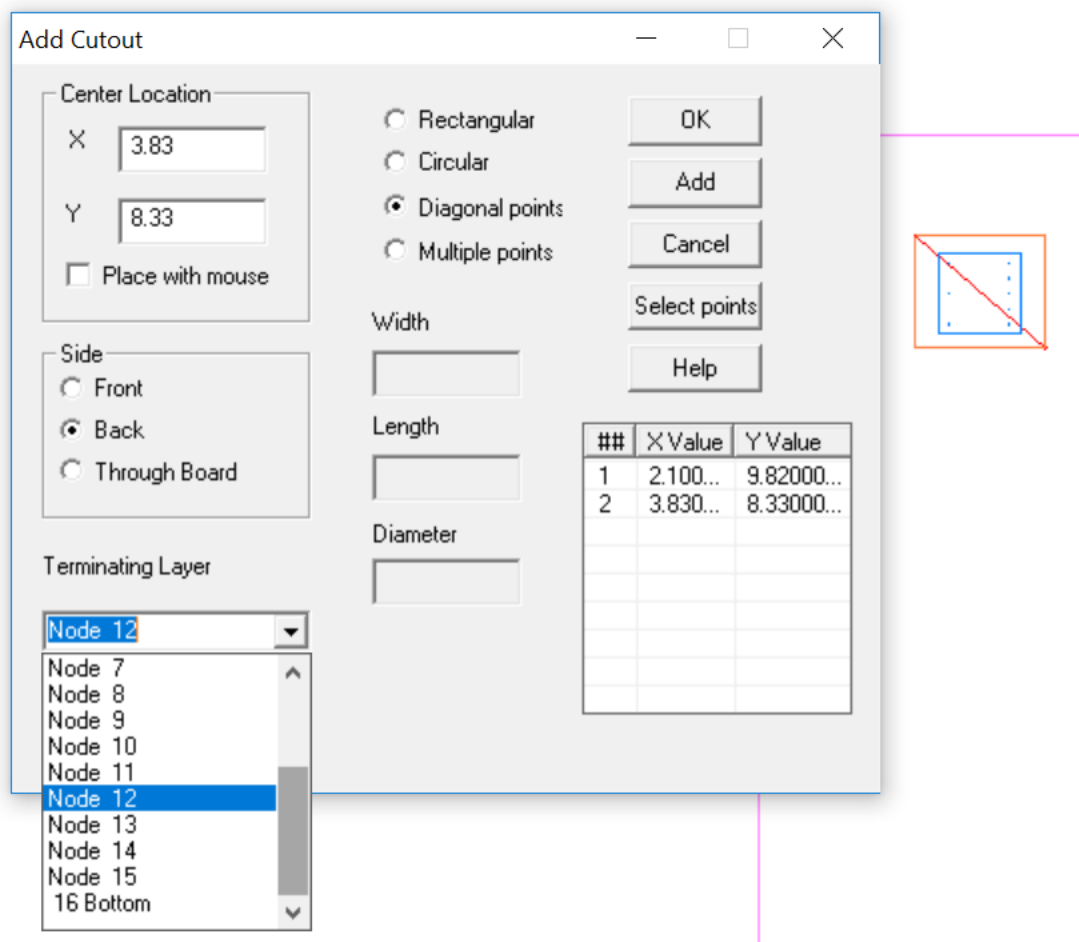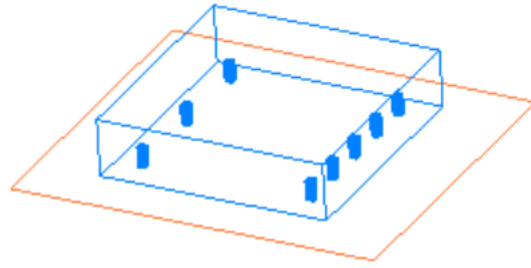
The figure below shows the Cutout on the back side of the board in the Solaria model.
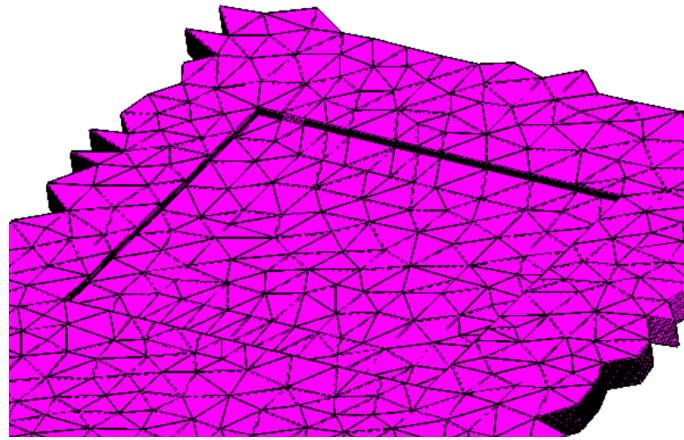
Below shows that the thermal contact from the leads and the bond go to the topmost exposed surface of the back side of the board.