SolariaPCB new Trace Heating Feature

The new SolariaPCB trace heating feature will calculate the heat dissipated in traces that are carrying current. When an ODB++ file is imported, every trace in the board is imported. The electrical engineer designing the board may have concerns on particular traces that are carrying a significant amount of current. IPC has guidelines but they do not account for all the local effect of complex boards. With this new feature, you can define Voltage sources and Current sources and sink on individual traces. You then define the rest of the board as you would normally do, defining component data and dissipation, adding thermal contacts so on. When you're done with the board, simply select PCB>Build Board Model as you normally do. You will notice now a few more dialogs will come up, but they are just to give you a status. When done, you will, just as you usually do, have a ready to solve board Solaria model. But now it will include the power dissipated by all the traces you have defined, as well as component and others you have defined.

Electrically connecting traces on different layers

When an ODB++ file is imported, all vias are imported. For each via, the layers it is electrically connected to is imported. When calculating voltages, vias are electrically simulated. In the example below, a trace on layer 5 and another on layer 1 is selected as being included in the voltage solution. You can see that a Voltage of 5 Volts is defined on trace 1.

	🔽 Include trac	e in voltage sol	ution Se	elect Trace	
	Trace width	.02		•	
	% Coverage	100			
	Trace Type	Line trace	La	ayer= 5	
=	Electrical Currer Positive for Curr	nt Source/Sink ent Source, ne	Points For Fi gative for Cu	lled Trace Hea rrent draw	ting
	No. Curren	t X	Y	Selected	
	A	dd	Cu	ıt [
	Voltage Source	/Sink Points			

Include trace	e in voltage so	lution Se	elect I race	
Trace width	.02		•	
% Coverage	100			
Trace Type	Line trace	La	iyer= 1	
Electrical Curren Positive for Curre	t Source/Sink ent Source, ne	Points For Fil gative for Cu	lled Trace Heal rrent draw	ting
No. Current	X	Y	Selected	
A	Id	Cu	ıt	
Ad	ld Sink Points	C.	ıt	
Ad Voltage Source/ No. Voltage	ld Sink Points	C. Y	it Selected	
Voltage Source/ No. Voltage 1 5	ld Sink Points X 1.44501	Cu Y 3.461482	t Selected	

To make sure that the traces on the different layers will be electrically connected, select Information>Element. Make sure Vias are viewed in PCB>View Entities. Hit Select then click on the intersection of the two traces, as shown below. In this case it shows that the via is connecting layers 1 and 5. Therefore these two traces will be electrically connected in the voltage solution.

+	
+	OK Help Copy to Clipboard Listbox will expand with dialog size >>>
\square	Total property heat= 0
\square	I otal non-property heat= U
	Via, X Location= 1.429489, Y Location= 2.43765 Drill diameter= 1.5500008822346E-4 Padstack is TBC_0.35C0.10 NetName is TBC
	► Layers attached to= 1 5

In this example, a trace on layer 7 is also selected. You can see below that a current draw of 1.0 Amp is defined.

✓ Include trace	in voltage sol	ution 56	ect Frace	
Trace width	.02		•	
% Coverage	100			
Trace Type	Line trace	La	iyer= 7	
Electrical Current	Source/Sink	Points For Fil	led Trace Hea	iting
Positive for Curre	nt Source, ne	gative for Cui	rent draw	
No. Current	nt Source, ne X	gative for Cui Y	Selected	
No. Current	nt Source, ne X 1.516044	2.200857	Selected	
No. Current	nt Source, ne X 1.516044	2.200857	Selected	
No. Current	1.516044	2.200857	Selected	
No. Current	1.516044	2.200857	Selected	

Again, in Information>Element, selecting the via at the intersection of the Layer 5 and layer 7 trace shows that there is a via connecting layers 5 and 7.

	i anal hair property hoar a
	4 Trace segments found
	Trace 36476 Node1= 41664, Node2= 41665, Width= .02, Layer 7 Padstack Net 959
<u> </u>	Trace 36477 Node1= 41665, Node2= 41666, Width= .02, Layer 7 Padstack Net 959
<u> </u>	Trace 36478 Node1= 41666, Node2= 41667, Width= .02, Layer 7 Padstack Net 959
	Trace 36479 Node1= 41667, Node2= 41668, Width= .02, Layer 7 Padstack Net 959
	Via, X Location= 1.52887, Y Location= 2.43765 Drill diameter= 3.1000017644692E-4 Pac
	Layers attached to= 5-7

So in this case, the current applied to the trace on layer 7 will travel through the via to the trace on layer 5. It will travel down the trace on layer 5 and through the via connecting the trace on layer 1 to the voltage at the other end of the trace on layer 1, completing the circuit.

Defining a trace to be included in the voltage solution

Select PCB>Edit Trace.

Edit Trace					
Include trac	e in voltage sol	ution S	elect Trace		OK
Trace width			•		Save Data
% Coverage					Clear Data
Trace Type Electrical Currer Positive for Curr	Line ht Source/Sink ent Source, nei	La Points For Fi gative for Cu	ayer= illed Trace Heal irrent draw	ting	Select Trace
No. Curren	t X	Y	Selected		Select location
Voltage Source	dd /Sink Points > X	Y	Selected	▲ 	Select location
A	dd	Ci	ut	-	

See below for adding a current source or sink or a Voltage source. Remember to hit the Save Data button after making any changes to the dialog.

Changing the trace width

After selecting a trace, simply change the trace width displayed in the dialog. The width of an area or filled trace cannot be changed. If the line trace width is increased to a value larger than the smallest trace segment length, then the automesher will have problems meshing it correctly.

Changing the percent coverage

There may be times where an area or filled trace isn't 100 percent copper. It may be an EMI shield created by a field of lines. If it is about 50% copper, then the % Coverage should be changed to 50. When the model is built, the thickness of the Plate elements representing the trace will be derated by this value. So if the layer trace thickness is defined as 0.0014 inch, then the thickness in the Property will be 0.0007 inches.

If traces have already been defined, select the one you want to view or edit from the Select Trace drop down list.

Edit Trace					
🔲 Include trace ir	n voltage solu	ition Se	elect Trace		ок
Trace width			21	9	ave Data
% Coverage		1	23 7 88	0	lear Data
Trace Type L Electrical Current S	.ine Source/Sink F	La Points For Fil	yer= led Trace Heating	S	elect Trace
Positive for Curren	t Source, neg	ative for Cur	rent draw		
No. Current	x	Y	Selected	!	Select location
Add		Cu	t		
Voltage Source/Si	ink Points			_	
No. Voltage	X	Y	Selected		Select location
Add		Cu	t		

The trace will be highlighted and information on the trace will be populated in the dialog. If the trace wasn't highlighted, then make sure that the layer the trace is on iv visible in PCB>View Settings.

Edit Trace	
✓ Include trace in voltage solution Select Trace	ОК
Trace width 0	Save Data
% Coverage 100	Clear Data
Trace Type Filled area trace Layer= 5 Electrical Current Source/Sink Points For Filled Trace Heating Positive for Current Source, negative for Current draw	Select Trace
No. Current X Y Selected 1 -30 3641732 1.889763 □	Select location
Add	
Voltage Source/Sink Points No. Voltage X Y Selected 1 5 .8169291 2.529527	Select location
Add	

Defining a Current source or sink

The figure below shows a 30 Amp current sink located at the X and Y location on the board as shown. A current source should have a positive value. This is one that is supplying a fixed amount of current to the board. A current sink should have a negative value. This is one the draws a fixed amount of current. This would be like the pin of a component.

Electrical Current Source/Sink Points For Filled Trace Heating Positive for Current Source, negative for Current draw									
No.	Current	Х	Y	Selected					
1	-30	3641732	1.889763						
Add Cut									

To define a new current source or sink, first hit the Add button. If you had not already checked "Include trace in voltage solution" then you will be asked if you want the trace to be. Next check the Selected checkbox in that row then hit either one of the Select Location buttons to the right. Next, using the mouse, select one of the highlighted locations on the trace. Next enter the value of the current source or sink under the Current column.

How to remove a current source or sink

Select anyplace in the row of the current source or sink you want to remove then hit the Cut button.

Defining a Voltage source

The figure below shows a 5 Volt source at the X and Y location on the board as shown.

Voltage Source/Sink Points								
No.	Voltage	Voltage X		Selected				
1	5	.8169291	2.529527					
		1	_	1				
	Add		Cu	ıt				

To define a new Voltage source, first hit the Add button. If you had not already checked "Include trace in voltage solution" then you will be asked if you want the trace to be. Next check the Selected checkbox in that row then hit either one of the Select Location buttons to the right. Next, using the mouse, select one of the highlighted locations on the trace. Next enter the value of the Voltage source under the Voltage column.

How to remove a current source or sink

Select anyplace in the row of the Voltage source you want to remove then hit the Cut button.

How to remove a trace from the voltage solution

In the Edit Trace dialog, select the trace you want to disable. Uncheck "Include trace in voltage solution". Remember next to hit the Save Data button.

How traces are represented in the Solaria model

Shown is layer 5 of the PCB.



Shown below is layer 11 of the PCB.



The four traces shown above will be explicitly meshed in the board. Below are shown Solaria Line elements that were generated from the traces selected.



Filled or area trace meshing

When importing an ODB++ file, one option is to define the "Number of segments in a circle". In the figure below, the default value of 4 was used.



In the figure below a value of 8 was used.



Using a value of 8 will result in a much finer mesh with very little different in voltages calculated as well as temperatures. Having a finer mesh will result in a longer Solaria model build time. But it is your choice.

Line trace meshing

Solaria Line elements are generated along the exact perimeter of the line. The figure below shows the line imported from the ODB++ file. It is simply a polyline connecting points with a defined width.



The figure below shows the Solaria Line elements along the perimeter of the line trace shown above.



The figure below shows the board mesh with only the board outline and the four traces included.



When the board is meshed, each trace references its own Property. The elements generated are on the same layer in the model the actual trace is on.



This makes it easy to put the traces into Groups. When putting elements into a Group by Property, you can check each trace individually or check Joule Heating to select them all.

16 Metal, Area is layer thickness 25
17 Net 121
18 Net 429
19 Net 17
20 Net 288
21 Joule Heating

All surface heat loads generated reference the Joule Heating Property. If you select Information>Property you can see the total heat generated by the traces. In this case, 4.45 Watts.

Information Proper	ties											-		×
< [] Click to	jump five propertie:	- 16 s	Drag Type	g trackbar ti e a property	o scan prope number to g	erties jo to	[ту	vpe a prope	rty nam	e to find		
Update	Add	Insert	E	Edit	Dele	e	Clear	Chan	nge Arrays					
		16			17		18			19		2	0	
Property Name Convection Resistance Nodal Heat Surface Heat Volumetric Heat Thickness Volume Area TEC VorC Velocity View Factor MdotCp Temperature Material Name Property Power	Units W/(in^2 * C) C/W W/(in^2) W/(in^3) in in^2 V or A in/sec W/C C	Net 121 0 0 0 0 0 0 0 0 0 0 0 0 0 0 0 0 0 0		Net 429 0 0 0 0 0 0 0 0 0 0 0 0 0 0 0 0 0 0 0			Net 17 0 0 0 0 0 0 0 0 0 0 0 0 0 0 0 0 0 0 0		Net 288 0 0 0 0 0 0 0 0 0 0 0 0 0 0 0 0 0 0			Joule Heat 0 0 1 1 0 0 0 0 0 0 0 0 0 0 0 0 0	ing	
Write Data	0.00 Inpu	t time for interpolation												•
Close	Help													

Solving for voltages

After the entire Solaria model is built of the board, Boundary Nodes and Nodal Heat Loads are generated on the traces to represent the Current sources and sinks and the Voltage sources. A Voltage source is simply represented by a Boundary node at the defined Voltage value at the node at the defined location. A current source or sink is simply represented with a Nodal Heat Load of the current value defined at the node at the defined location. All traces, filled and line, are represented by 2D triangular Plate elements. Each trace references a different Property where the

thickness is defined. All Properties reference the same Material that has the electrical conductivity of copper at 35C. Only this part of the total Solaria model is sent to the Solaria Finite Difference solve to solve for voltages. The "Heat Load" shown in the dialog below is actually the total current in the model.

Solve for Voltages		—		\times
Solving Status			OK	
Status Iterations Convergence Heat Balance Heat Load Resistor Count	Done 4657 0.000000 9.9947394E-3 -52 3673			

After solving for voltages, the Boundary Nodes and the Nodal Heat Loads are deleted from the model. All trace Properties are automatically changed to reference the material defined for the metal layers.

Next, as with any other SolariaPCB Solaria model build, the following dialog will come up. This dialog allows you to offset the new board model. Hit Apply after changing any values.

Merge Models				
Incoming Object Translation		Current Object XYZ Span		
0.00	Translate object X	12.258662		
0.00	Translate object Y	2.8976377		
0.00	Translate object Z	0		
Grab and Group Control Place Object in GRAB Hold on Load Group for Incoming Object				
1 Entire Mod	el	•		
Help				
Apply	Cancel			

Voltages are saved to a file so they can be imported later.



Solving for heat load based on the voltage drop

The 2D Plate elements are then sent to the algorithm that performs a dV^2/R to generate Surface heat loads. This calculation is performed for each triangular element based on the dV^2/R for each of the three legs of the triangle. R is the electrical resistance of the triangle. The three values are summed up and divided by the triangle area creating a surface heat load.

Where R is the electrical resistance of the meshed element and dV is the local voltage drop.



The following dialog will come up.



Just hit OK then Cancel in the Solaria Model Generation Progress Bar dialog. Switch back to Solaria Mode by hitting the

toolbar button.

Viewing voltage drop in the traces

In Solaria, put the traces into a Group by selecting their Properties. Now you can view Voltages just as you would view temperatures.



Solving for temperatures

The Solaria model created is ready to be solved for temperatures, assuming that you have added boundary conditions like a Thermal Contact.



For the multi-layer trace model, the trace properties and the electrical vias were put into a group.

FILTER - Property	
□ 1 □ 2 Automesh □ 3 Metal, Area is layer thickness 1 □ 4 Dielectric □ 5 Metal, Area is layer thickness 3 □ 6 Metal, Area is layer thickness 5 □ 7 Metal, Area is layer thickness 7 □ 8 Metal, Area is layer thickness 7 □ 9 Metal, Area is layer thickness 11 □ 10 IC1000 □ 11 Thermal Interface 1 □ 12 Thermal Interface 2 □ 13 Vias ✓ 14 Net 1623	 Toggle (Off) Toggle all checks (on/off) Invert Invert Invert all checks Check Range Check Properties thru Process Range Search String Case sensitive With string Without string
 ✓ 15 Net 959 ✓ 16 Net 2214 	Help
✓ 17 Electrical Vias ☐ 18 Joule Heating	→ Hide

Displaying temperatures shows the Voltages on the traces.



The separation in the traces shown above-right, is where the electrical vias. Below shows the elements. The vertical lines are the Resistor elements representing the vias.



Here are the board temperature with power applied to one component in addition to the trace.



To see the effect of only the Joule heating in the traces, in the property for the component, the power was changed to zero and the model was re-solved.

