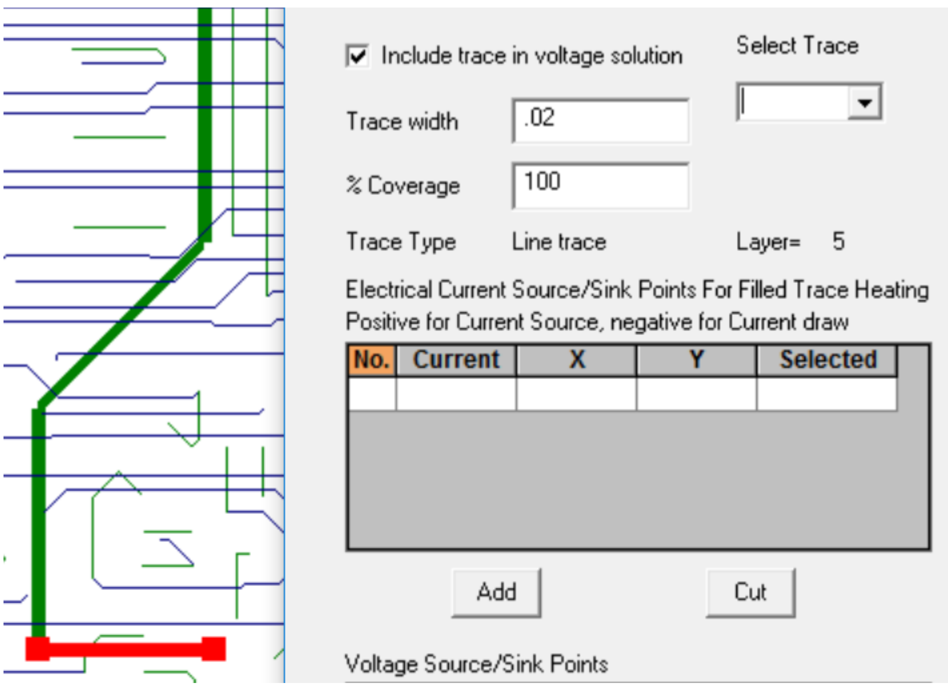
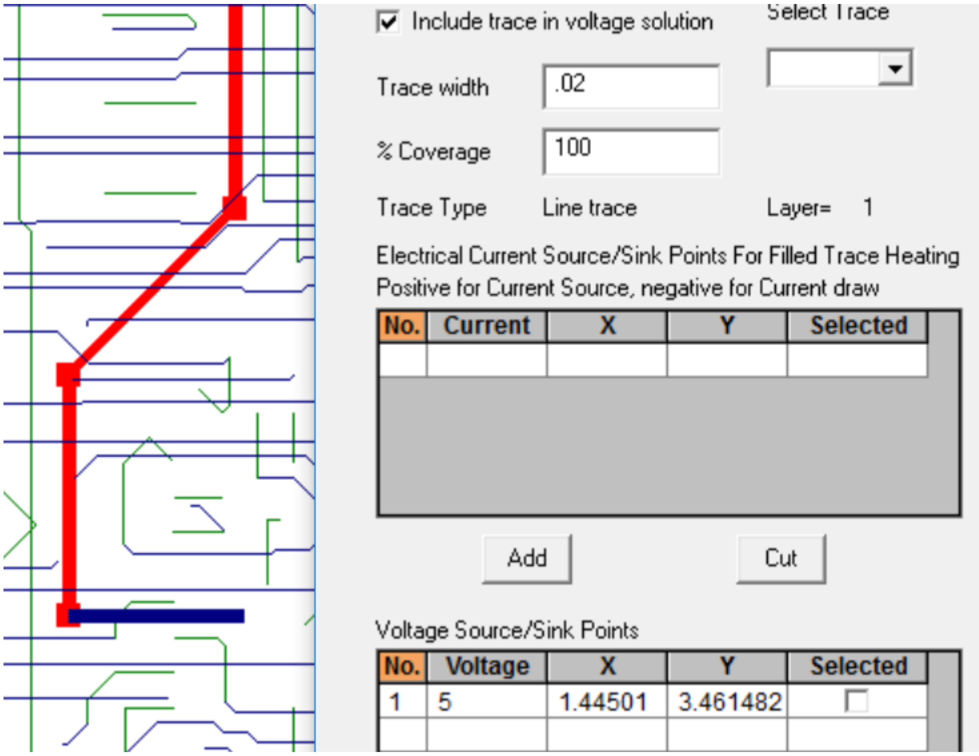
SolariaPCB new Trace Heating Feature

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.

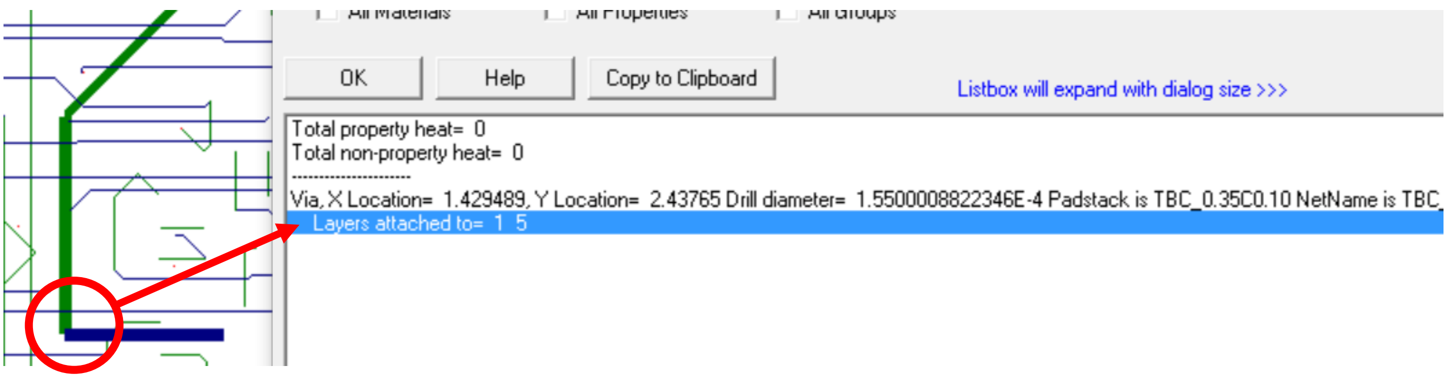
Electrically connecting traces on different layers

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.

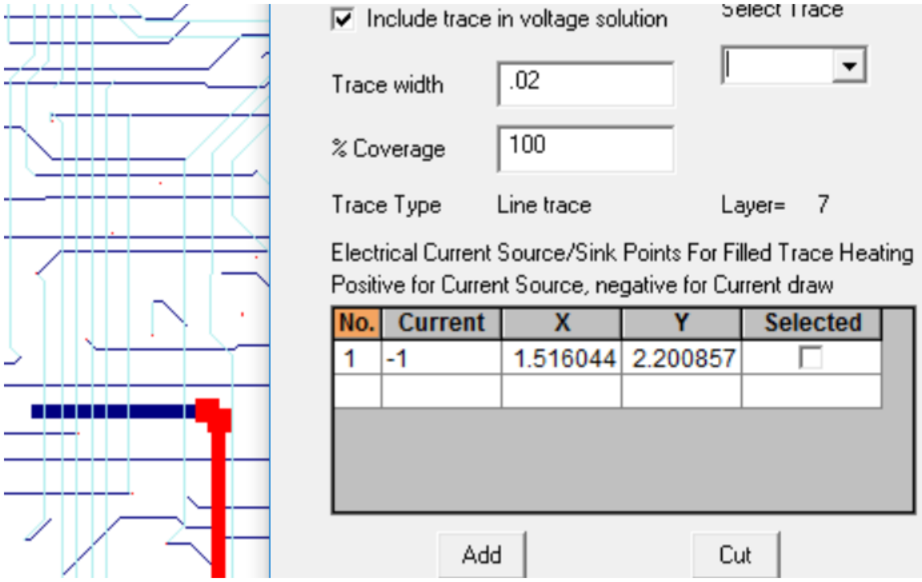




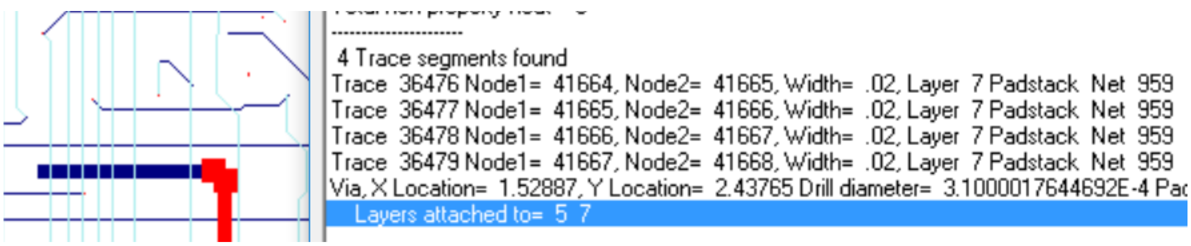
To make sure that the traces on the different layers will be electrically connected, select Information>Element. Make sure Vias are viewed in PCB>View Entities. Hit Select then click on the intersection of the two traces, as shown below. In this case it shows that the via is connecting layers 1 and 5.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



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.



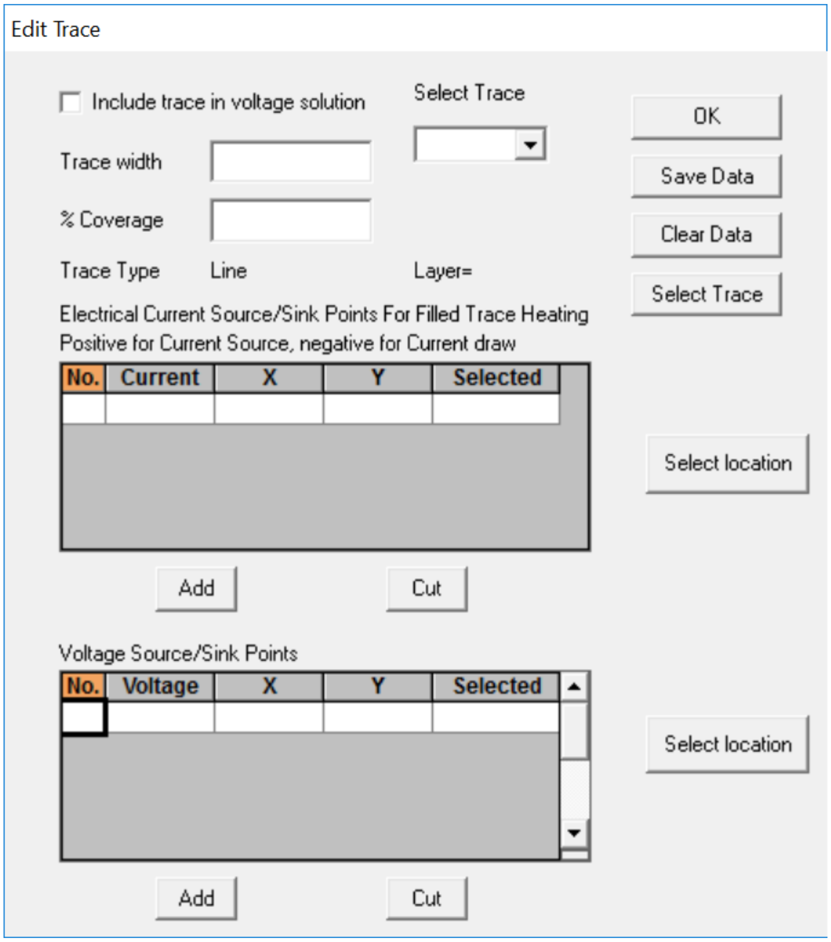
Again, in Information>Element, selecting the via at the intersection of the Layer 5 and layer 7 trace shows that there is a via connecting layers 5 and 7.



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.



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.

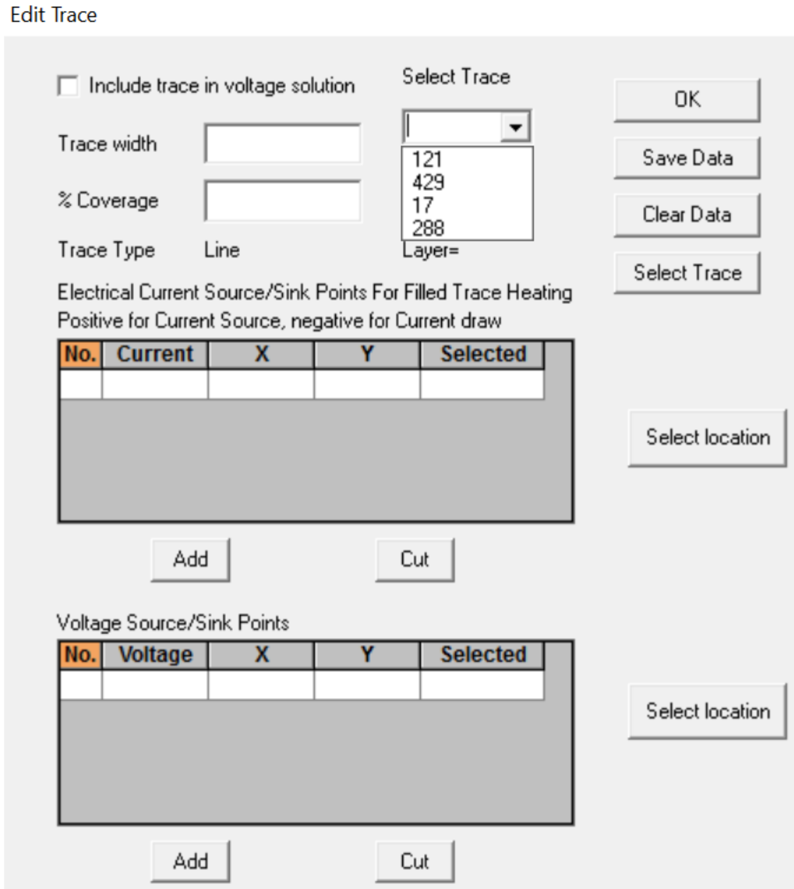
Changing the trace width

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.

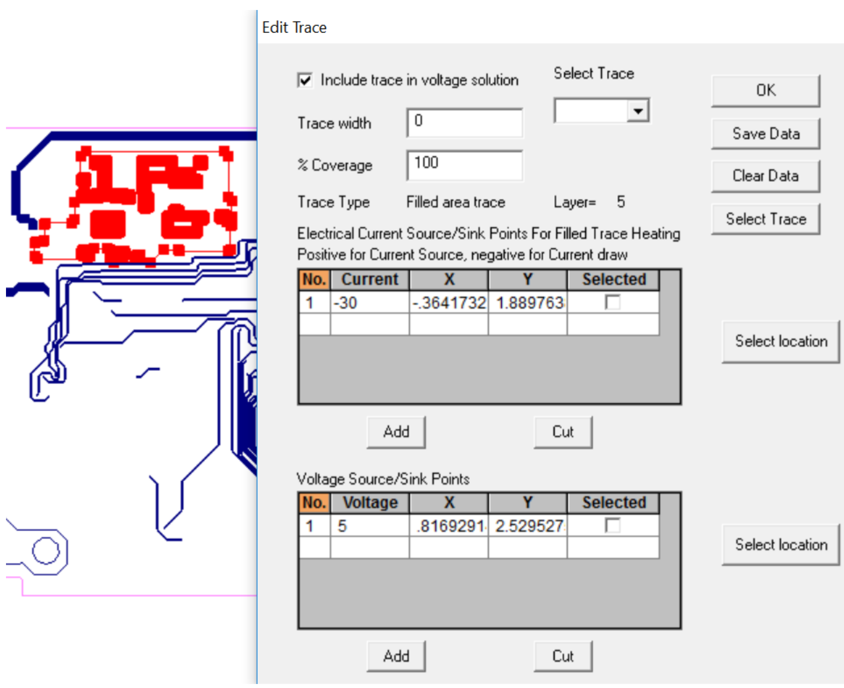
Changing the percent coverage

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.

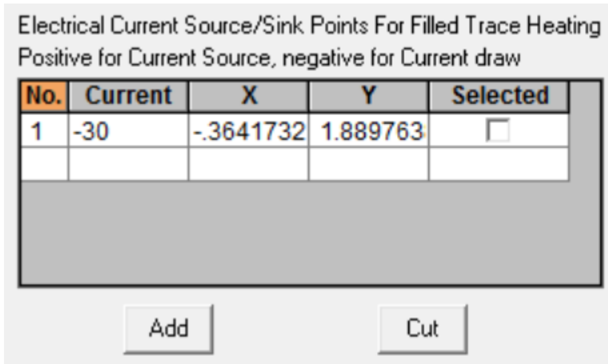


The trace will be highlighted and information on the trace will be populated in the dialog. If the trace wasn't highlighted, then make sure that the layer the trace is on iv visible in PCB>View Settings.



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.



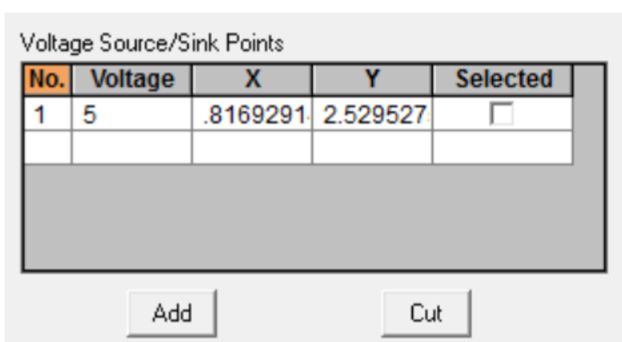
To define a new current source or sink, first hit the Add button. If you had not already checked "Include trace in voltage solution" then you will be asked if you want the trace to be. Next check the Selected checkbox in that row then hit either one of the Select Location buttons to the right. Next,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



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

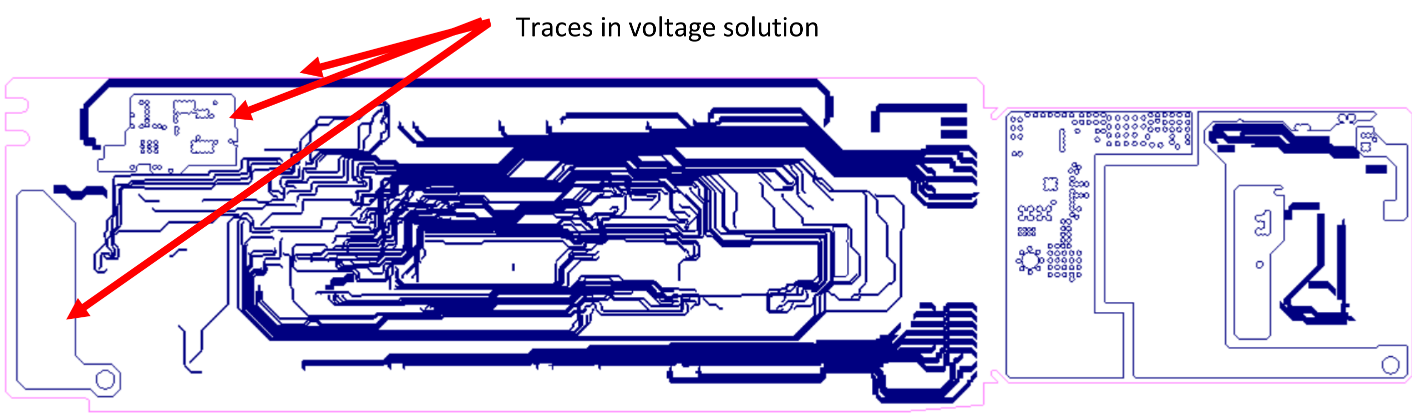
Select anyplace in the row of the Voltage source you want to remove then hit the Cut button.

How to remove a trace from the voltage solution

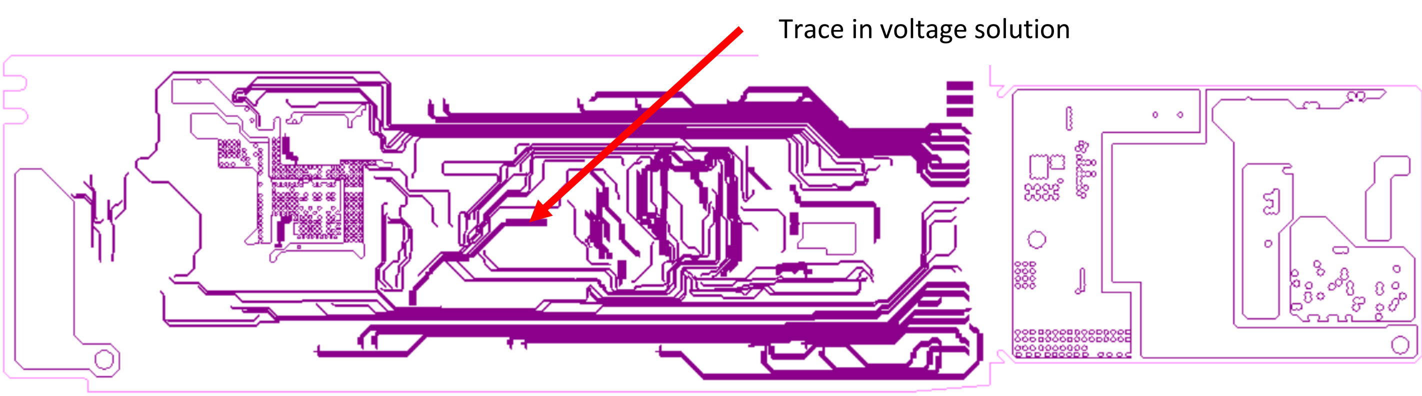
In the Edit Trace dialog, select the trace you want to disable. Uncheck “Include trace in voltage solution”. Remember next to hit the Save Data button.

How traces are represented in the Solaria model

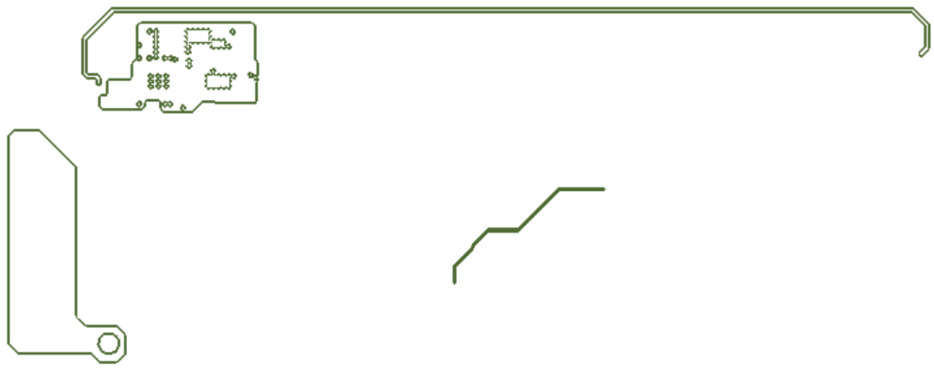
Shown is layer 5 of the PCB.



Shown below is layer 11 of the PCB.

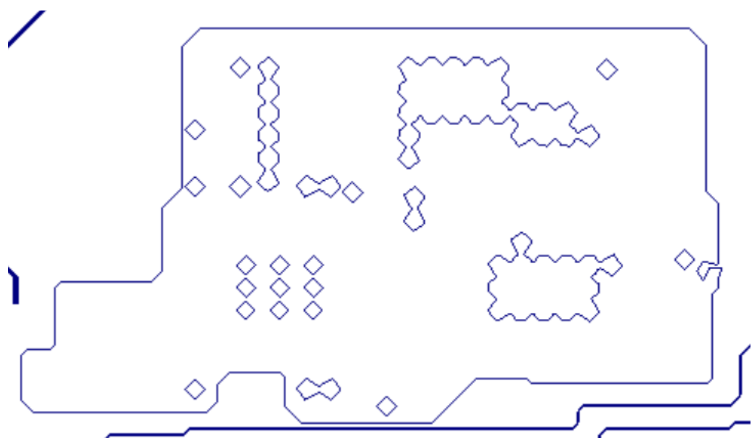


The four traces shown above will be explicitly meshed in the board. Below are shown Solaria Line elements that were generated from the traces selected.

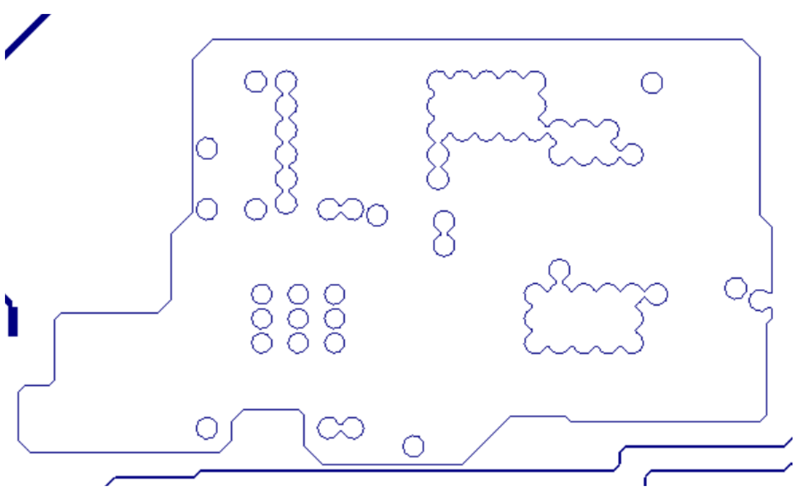


Filled or area trace meshing

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.



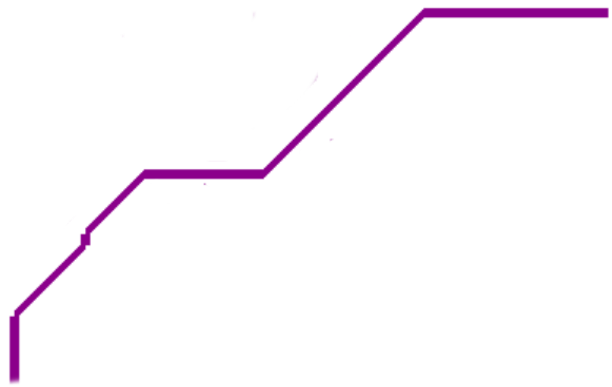
In the figure below a value of 8 was used.



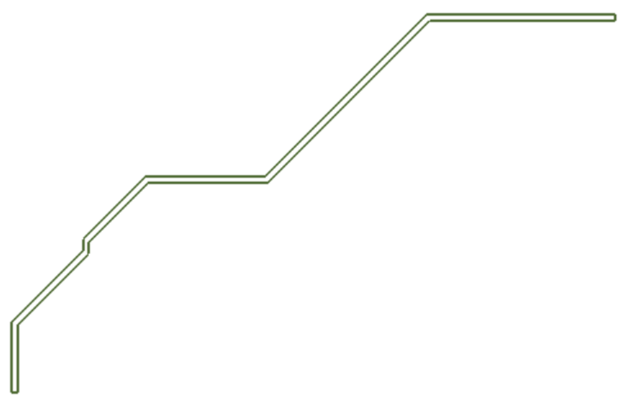
Using a value of 8 will result in a much finer mesh with very little different in voltages calculated as well as temperatures. Having a finer mesh will result in a longer Solaria model build time. But it is your choice.

Line trace meshing

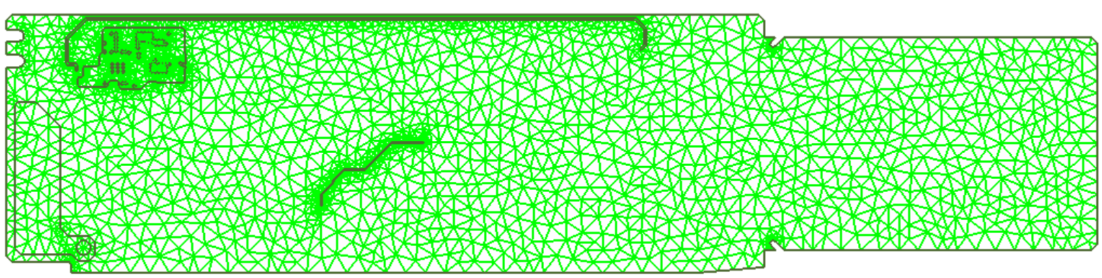
Solaria Line elements are generated along the exact perimeter of the line. The figure below shows the line imported from the ODB++ file. It is simply a polyline connecting points with a defined width.



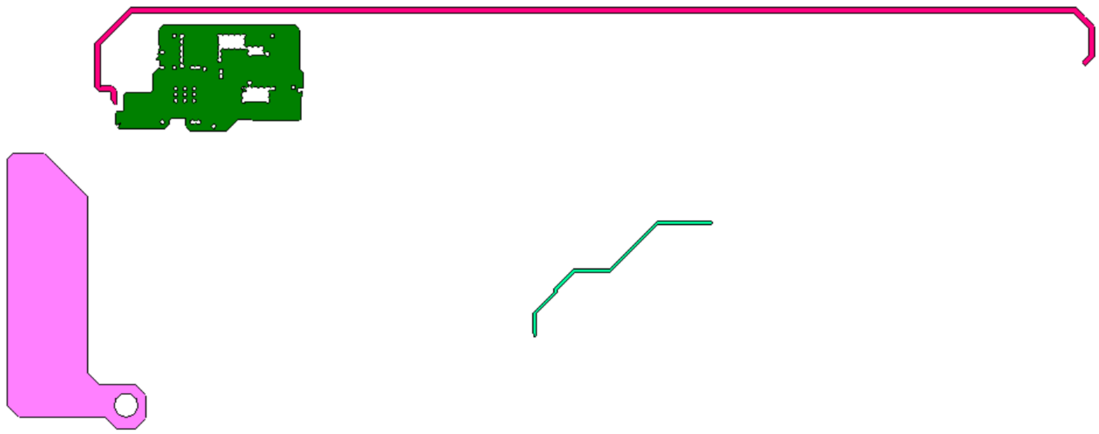
The figure below shows the Solaria Line elements along the perimeter of the line trace shown above.



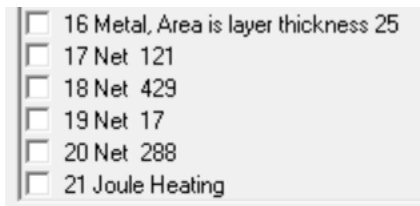
The figure below shows the board mesh with only the board outline and the four traces included.



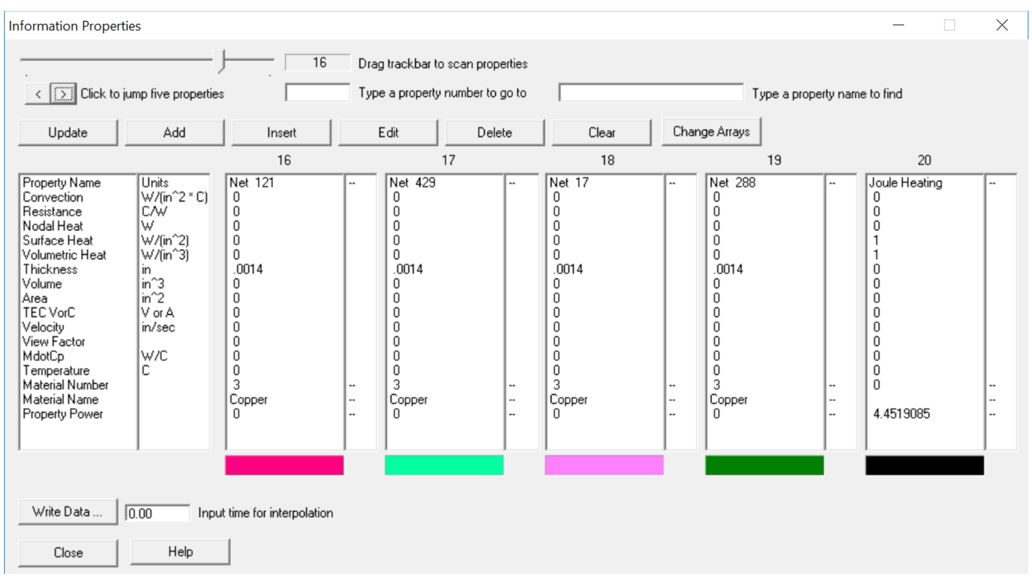
When the board is meshed, each trace references its own Property. The elements generated are on the same layer in the model the actual trace is on.



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.



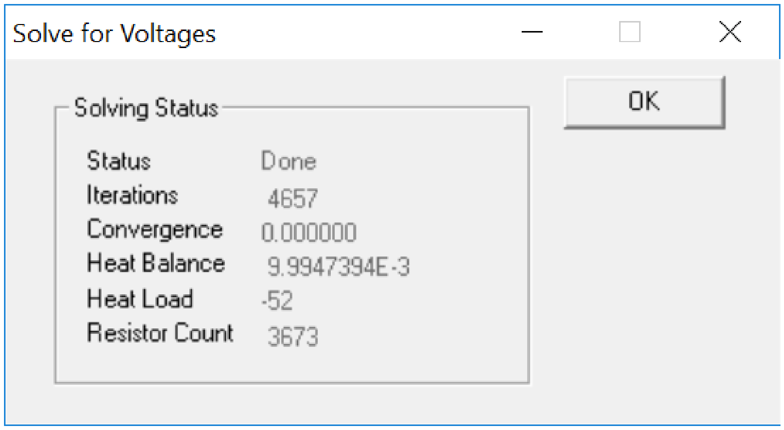
All surface heat loads generated reference the Joule Heating Property. If you select Information>Property you can see the total heat generated by the traces. In this case, 4.45 Watts.



		16	17	18	19	20
Property Name	Units	Net 121	Net 429	Net 17	Net 288	Joule Heating
Convection	W/(in^2 * C)	0	0	0	0	0
Resistance	C/W	0	0	0	0	0
Nodal Heat	W	0	0	0	0	0
Surface Heat	W/(in^2)	0	0	0	0	1
Volumetric Heat	W/(in^3)	0	0	0	0	1
Thickness	in	.0014	.0014	.0014	.0014	0
Volume	in^3	0	0	0	0	0
Area	in^2	0	0	0	0	0
TEC VorC	V or A	0	0	0	0	0
Velocity	in/sec	0	0	0	0	0
View Factor		0	0	0	0	0
MdotCp	W/C	0	0	0	0	0
Temperature	C	0	0	0	0	0
Material Number		3	3	3	3	0
Material Name		Copper	Copper	Copper	Copper	
Property Power		0	0	0	0	4.4519085

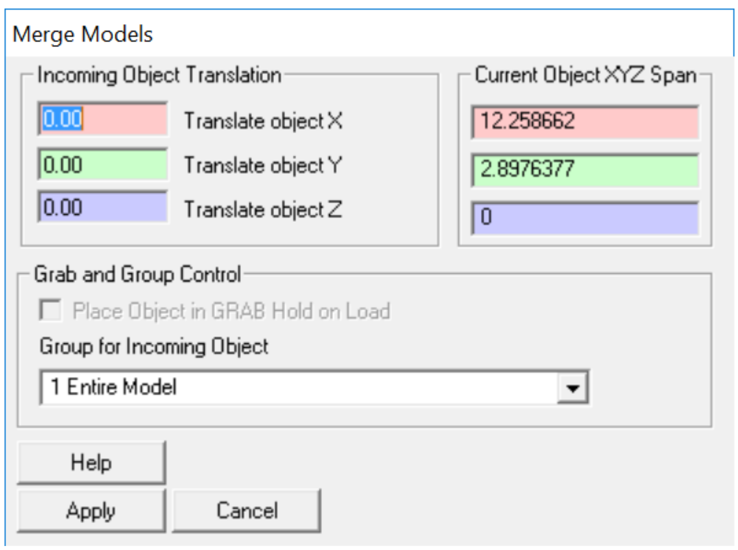
Solving for voltages

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.

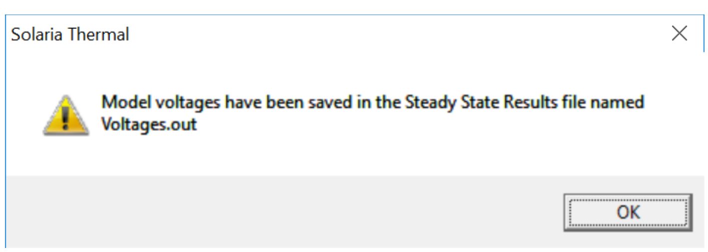


After solving for voltages, the Boundary Nodes and the Nodal Heat Loads are deleted from the model. All trace Properties are automatically changed to reference the material defined for the metal layers.

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.



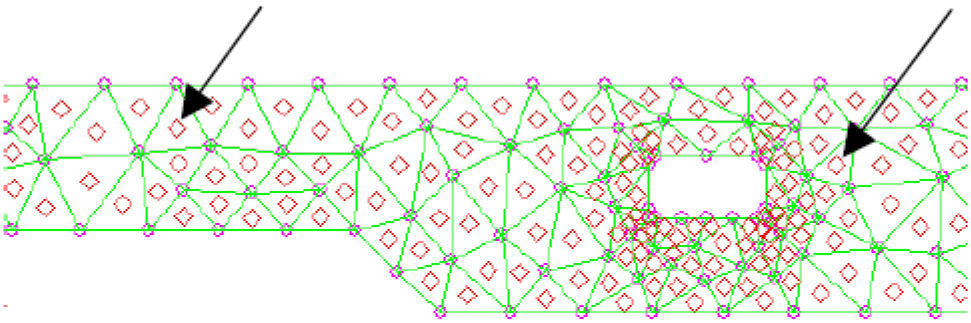
Voltages are saved to a file so they can be imported later.



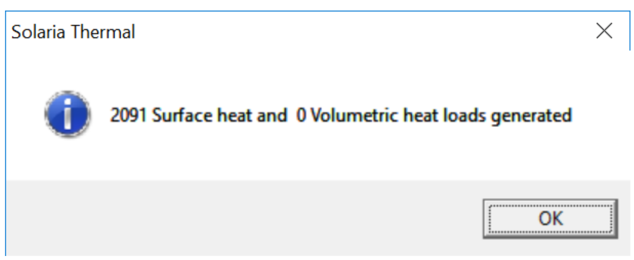
Solving for heat load based on the voltage drop

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.



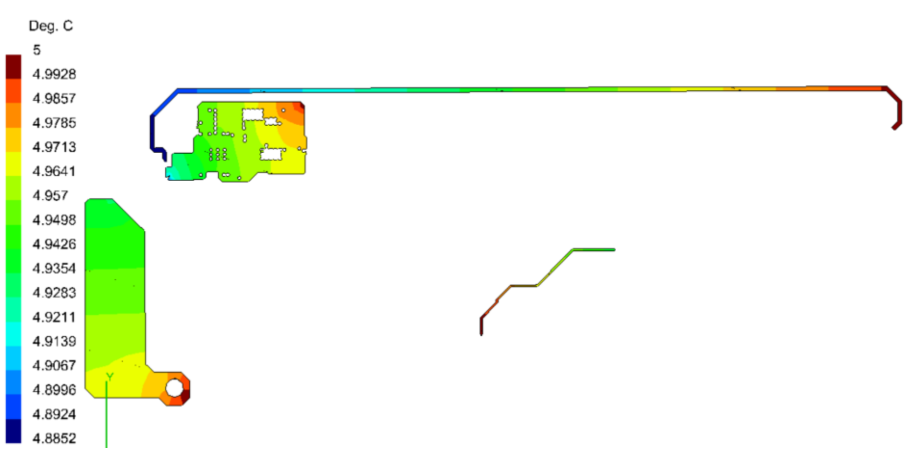
The following dialog will come up.



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

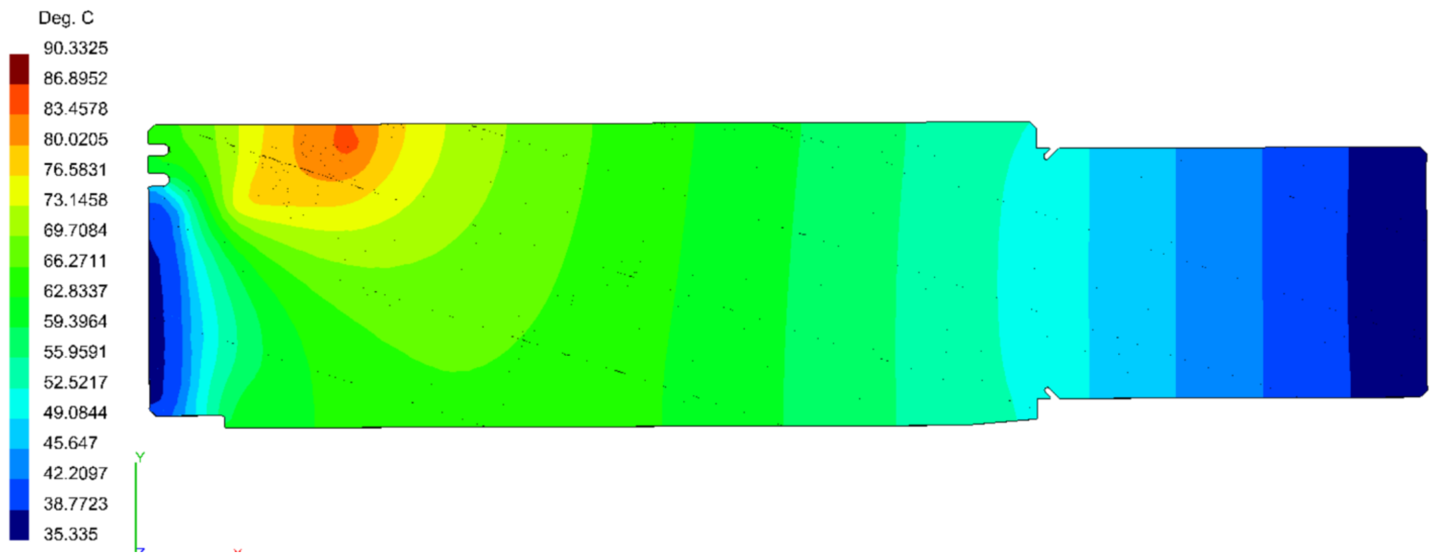
Viewing voltage drop in the traces

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

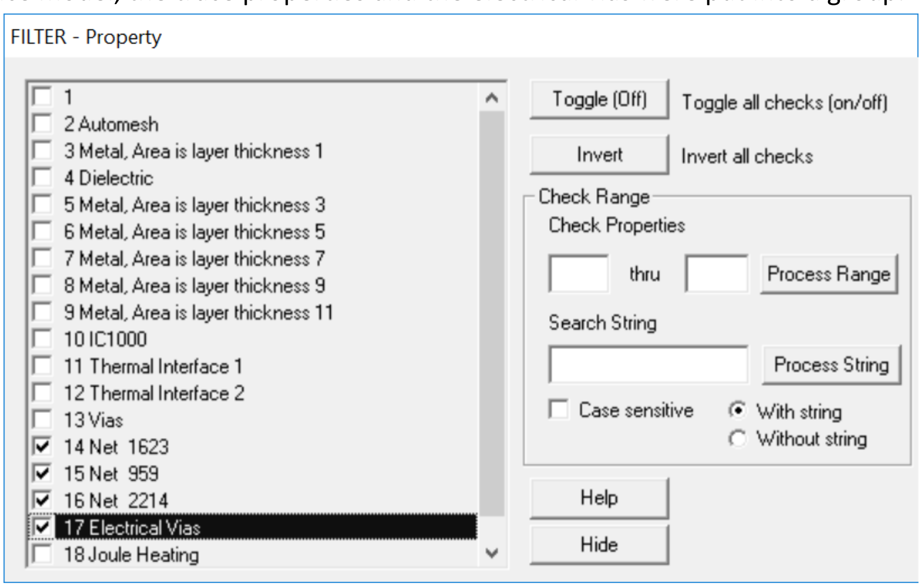


Solving for temperatures

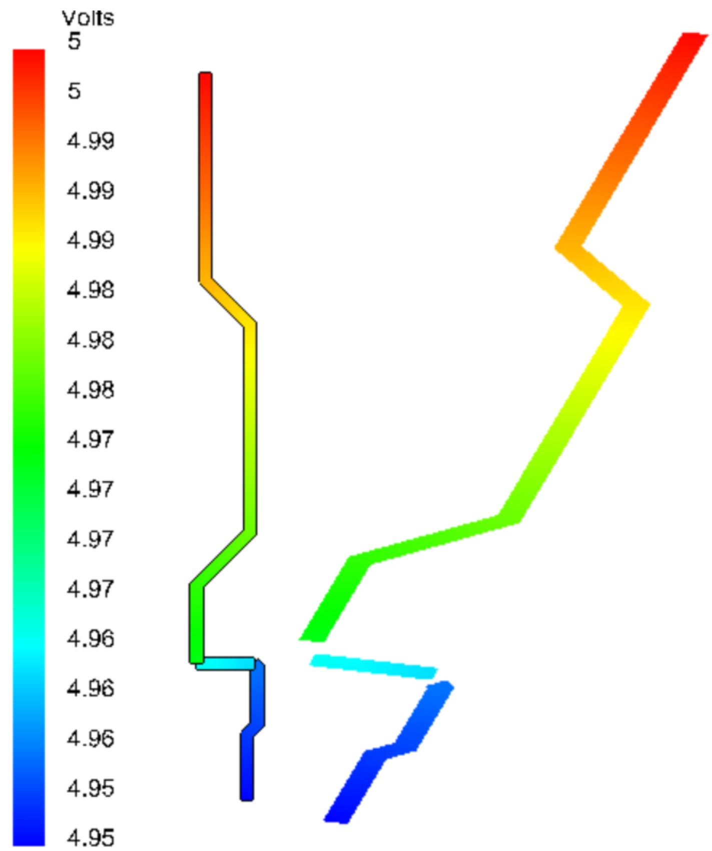
The Solaria model created is ready to be solved for temperatures, assuming that you have added boundary conditions like a Thermal Contact.



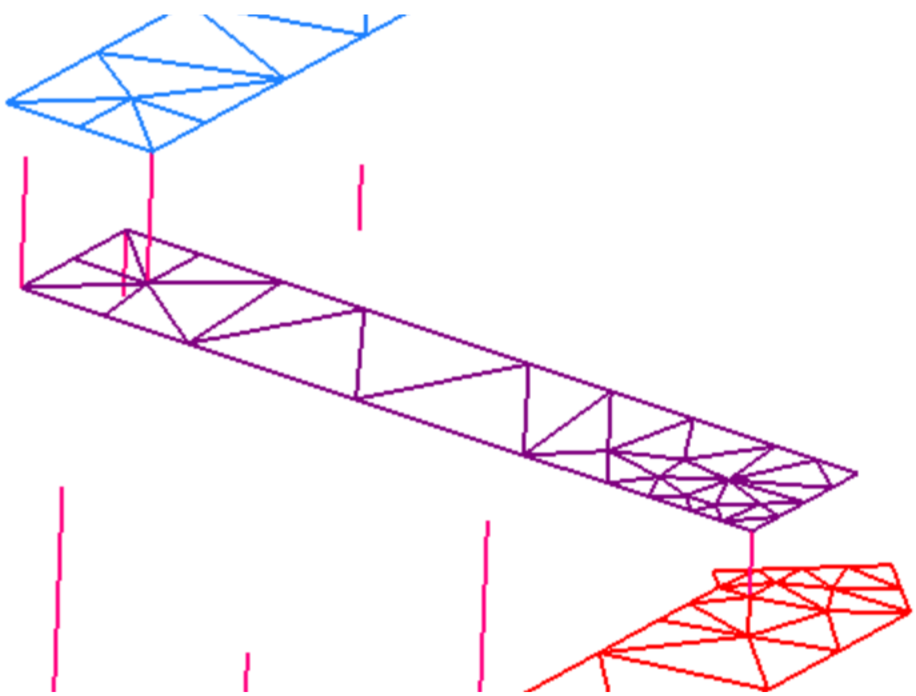
For the multi-layer trace model, the trace properties and the electrical vias were put into a group.



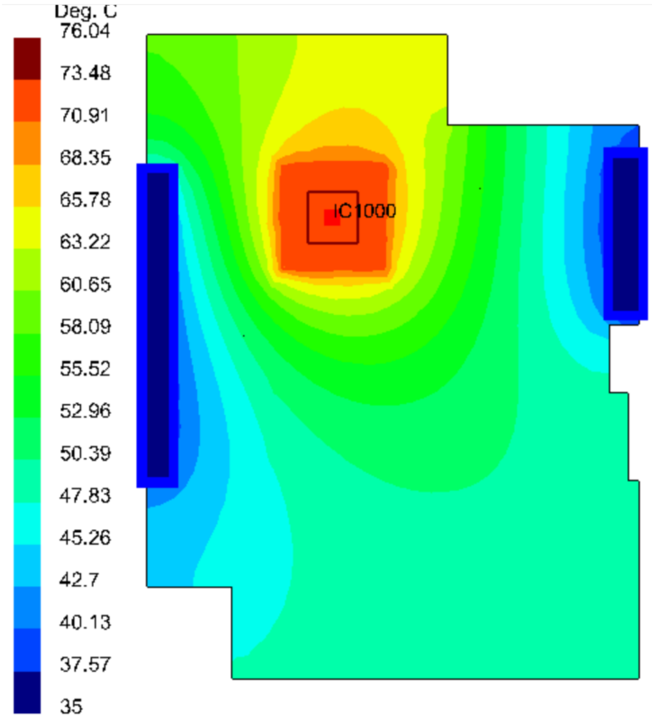
Displaying temperatures shows the Voltages on the traces.



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.