PCB DESIGN ERROR CONTRIBUTIONS IN CURRENT SENSE RESISTORS

Purpose:

Having a high reliable Current Sense Resistor (CSR) component in your design is always a priority for a successful output. However, the value of this component will be diminished if you do not have a good optimization of your PCB. The more you minimize your design error contributions, the more superior performance you will achieve. This paper will discuss some of the factors to consider when designing a PCB

PCB Stack-up Layers

One factor to consider in your design is the number of layers on your PCB. The number of layers on your PCB will have an impact on the performance of the component. For example, a single layer PCB (Figure 1) does not provide as much thermal conductivity as a multilayer PCB (Figure 2). Copper dominates PCB thermal conductivity so a multilayer board with solid Cu on all layers will be more conductive. The number of layers is important in your design, however, other factors like the number of vias and the material thickness also play a crucial role on your PCB thermodynamic considerations.

Layer	Material	Thickness
Copper Layer #1	2 oz. Copper	0.068mm
FR4 Layer #1	FR4	0.073mm

Figure 1

Layer	Material	Thickness
Copper Layer #1	1 oz. Copper	0.034mm
FR4 Layer #1	FR4	0.073mm
Copper Layer #2	1.5 oz. Copper	0.051mm
FR4 Layer #2	FR4	0.073mm
Copper Layer #3	2 oz. Copper	0.068mm
FR4 Layer #3	FR4	0.076mm
Copper Layer #4	2 oz. Copper	0.068mm
FR4 Layer #4	FR4	0.073mm
Copper Layer #5	1.5 oz. Copper	0.051mm
FR4 Layer #5	FR4	0.073mm
Copper Layer #6	1 oz. Copper	0.034mm

Figure 2

Optimizing the PCB Design

Current sense resistors are known for their very low ohmic value (a few mΩ or less), with high accuracy. Due to the low resistance value, factors like the resistance of the solder joints, size of the component pads and PCB component density becomes a large portion of the sense element resistance and can contribute to your measurement error and thermal behavior of the PCB.

Understanding the design of your PCB, as well as the density of your component layout will allow you to reduce your thermal impacts. The test below will show the effect of these considerations.

Thermal Test

Purpose:
To determine the heat rise of TFT's MPC0402 Current Sense Resistor based on different PCB configurations and stack-ups.

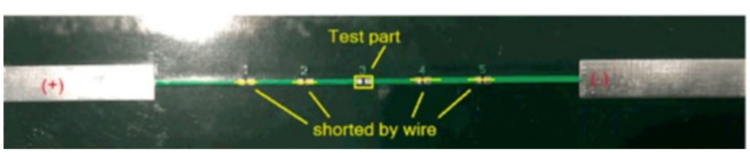
Request:
We have run this test on two different resistor configurations and three different stack-up boards as described below. Both configurations utilize the same PCB layout. The only change on the stack up designs is the copper thickness of the outermost layers, as described in this report.

Board Design:

Configuration #1 (Red Box- five resistors):
The parts are positioned on the board in a single line. There are five resistors in series which are positioned end to end. The spacing between the components was dictated by our customer who helped determine the conditions for this testing based on their real-life design considerations. Please see (picture 1) below that shows this configuration.

Configuration # 1 Five Resistors
Picture 1

Configuration#2 (Yellow Box – Single resistor):
We placed one single part in the middle of the designed test board, and we shorted the remaining resistors electrode pads out by wire.



Configuration # 2 Single Resistor
Picture 2

Board Copper Thickness Stack-up

Copper Thickness - 0.5 oz

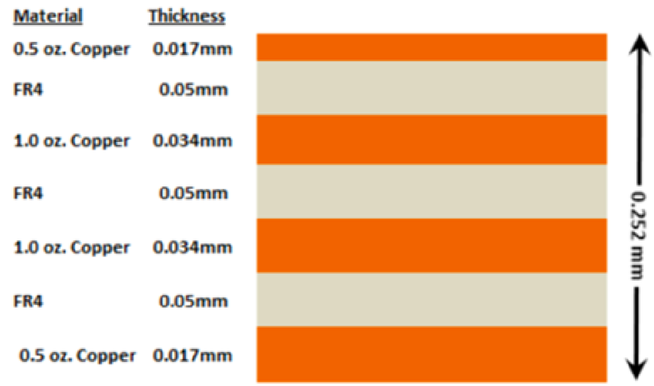


Figure 3

Copper Thickness - 1.0 oz

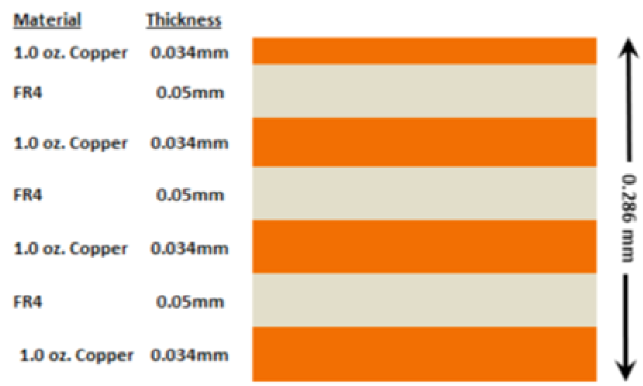


Figure 4

Copper Thickness - 1.5 oz

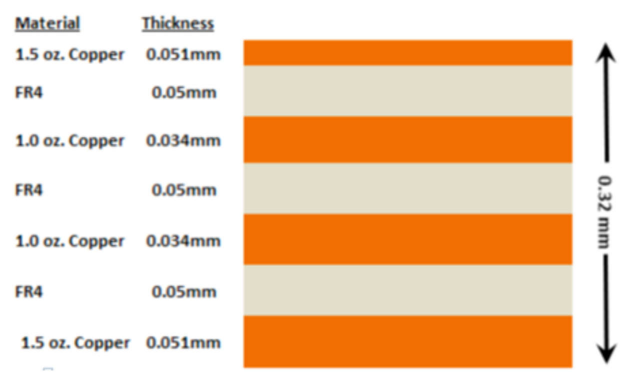


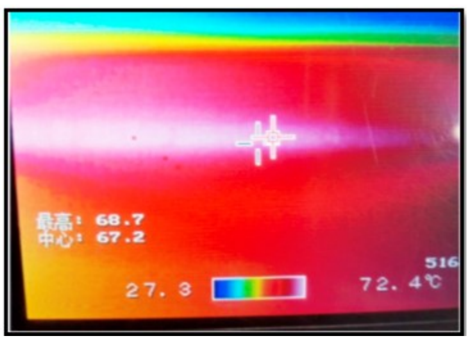
Figure 5

Testing Procedure:

1. We tested the parts using two different resistor configurations and three different stack -up boards. We mounted 5 resistors on to TFT's 0.5oz thickness test board using configuration #1(five resistors). We applied 6 amps across the parts, and we let the parts stabilize for 5 min. Using the thermal camera, we captured the maximum temperature of the board, and based on that we calculated the max temperature rise of the corresponding configuration.
2. Using a new test board, we mounted one resistor on to TFT's 0.5oz thickness PCB using configuration # 2 (single resistor). We applied 6 amps across the part, and we let the part stabilize for 5 min. Using the thermal camera, we captured the maximum temperature of the board, and based on that we calculated the max temperature rise of the corresponding configuration.
3. We then repeated all the steps above using TFT's 1.0 oz thickness and 1.5 oz thickness test boards.

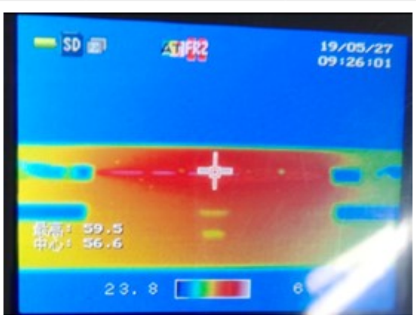
Testing Results 0.5 oz:

Configuration # 1 (Five Resistors)		
Room Temp	Max Temp Board	Calculated Temp Rise
26 °C	68.7 °C	42.7 °C



Picture 3

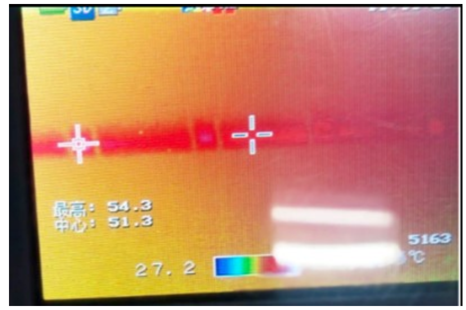
Configuration #2 (Single Piece)		
Room Temp	Max Temp Board	Calculated Temp Rise
26 °C	59.5°C	33.5 °C



Picture 4

Testing Results 1.0 oz:

Configuration # 1 (Five Resistors)		
Room Temp	Max Temp Board	Calculated Temp Rise
26 °C	54.3°C	28.3 °C



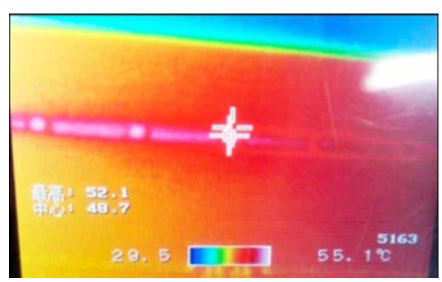
Picture 5

Configuration #2 (Single Piece)		
Room Temp	Max Temp Board	Calculated Temp Rise
26 °C	43.4°C	17.4 °C

Picture 6

Testing Results 1.5 oz:

Configuration # 1 (Five Resistors)		
Room Temp	Max Temp Board	Calculated Temp Rise
26 °C	52.1°C	26.1 °C



Picture 7

Configuration #2 (Single Piece)		
Room Temp	Max Temp Board	Calculated Temp Rise
26 °C	39.6°C	13.6 °C

Picture 8

Conclusion:

1. After testing TFT's MPC0402 parts on three different stack-up boards, as expected, the test board with 1.5OZ Cu top and bottom layers had the best heat dissipation performance.
2. You can use this DEO as a guideline to show the concept of PCB design and its thermodynamic factors. However, every design is unique and will have different impacts depending on your board design and component layout. The proximity of components to one another will change the rate of heat rise within the resistor so please consider this when designing your board.

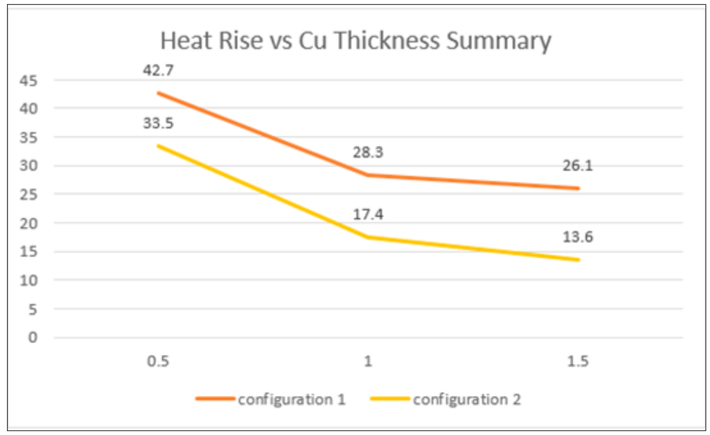


Figure 6

Visit our virtual booth on the electronica website or learn more about our products and offerings at thin-film.com.