DIABLO MOUNTAIN RESEARCH, LLC. 741 BUENA VISTA PLACE WALNUT CREEK, Ca 94597 TEL: 925 788-4807

PCB Trace Resistance at DC

All of us that design board level products like to use the PCB (Printed Circuit Board) to implement resistors, inductors, capacitors, transmission lines, filters, couplers, hybrids, equalizers, thermal control … We all recognize the importance of the PCB layout, and the physical characteristics of the PCB material in our design work. This technical note covers the fundamentals of a trace's resistance at DC. Also touched on are caveats, limitations, current handling, and an example of a mitigating technique when a trace's resistance is used for current measurement.

The plating weight of copper on a printed circuit board refers to the number of ounces of copper deposited on a flat surface per square foot. For most applications, weight tends to be a relatively unimportant unit of measurement. The nominal thickness for "standard" weights (prior to any solder plating) is:

Copper weight (oz)	Thickness (inch)	Thickness Tolerance (inch)
½	0.0007	±0.0001
1	0.0014	±0.0002
2	0.0028	±0.0003

The resistance of a trace or plane (in ohms) can then be calculated from:

$$R = \frac{(0.6788 \bullet 10^{-6}) \bullet L}{WT}$$

where

$$W = \textit{width of the trace (in.)}$$
$$L = \textit{length of the trace (in.)}$$
$$T = \textit{copper thickness (in.)}$$

Note: This equation uses the nominal copper foil resistivity of 1.724 microhm-cm = 0.6788 microhom-in. Pure copper resistivity is 1.678 micro ohm-cm.

For **½ oz** copper, the formula can be simplified to:

$$R = \frac{(969.714 \bullet 10^{-6}) \bullet L}{W}$$

(L and W in inches).

For **1 oz** copper, the formula can be simplified to:

$$R = \frac{(484.857 \bullet 10^{-6}) \bullet L}{W}$$

(L and W in inches).

Ohms per Square:

We often simplify the resistance calculation by using the concept of ohms per square. Given a specific conductor thickness, and setting $\frac{L}{W}=1$, (a simple square), we can then generate a simplified, approximate, and easy to remember table:

Copper weight (oz)	Approximate Ohms per Square (mΩ)	*Ohms per Square (mΩ)*
½	1	*0.9697*
1	0.5	*0.4848*
2	0.25	*0.2424*

For example: a 10 mil wide trace, 1 inch long on a ½ ounce copper layer is simply 100 squares, thus its resistance will be approximately 100 milliohms. Recall that copper traces will not have a perfect rectangular cross section; due to etching, their cross section will be trapezoidal. The trapezoidal cross section will result in a moderately higher resistance. Another example: the resistance of a solid 1oz copper plane on a board that is 3"x3", or 24"x24", would be approximately ½ milliohm.

Temperature Coefficient of Resistance

The nominal temperature coefficient of resistance (TCR) of copper is ~0.393%/°C. This is a whopping big TCR, compare this (3930ppm/°C) to the <100ppm/°C TCR for a cheap 0805 5% SMT resistor.

Outside Layers (Top and Bottom Trace Thickness)

Due to plating, the outside layers of a PCB can be additionally thick, another 0.002" thick would not be uncommon. Additionally, there may be additional thin coating (nickel, gold, …).

Temperature Rise of Traces

Fortunately, there has been a recent update in estimating the temperature rise of traces due to current flow and the resultant power dissipation. This update is well past due. For the past 55 years, a set of vintage graphs has been used. These are readily available on the web, and were collected into a MIL spec: MIL-STD-275. (also available on the web) This very old data is also presented in IPC-2221.

The new data is available in IPC-2152, *Standard for Determining Current-Carrying Capacity in Printed Board Design*. For an introduction and overview, see the Jan 2011 Printed Circuit Design and Fab (http://www.pcdandf.com/) article by Michael Jouppi. Mr. Jouppi states that the old graphs result in oversized conductors. The newer data includes our much used multilayer boards, which can have a large effect on the current handling of a trace. Recall that the thermal conductivity of copper is about 1000 times greater than FR4.

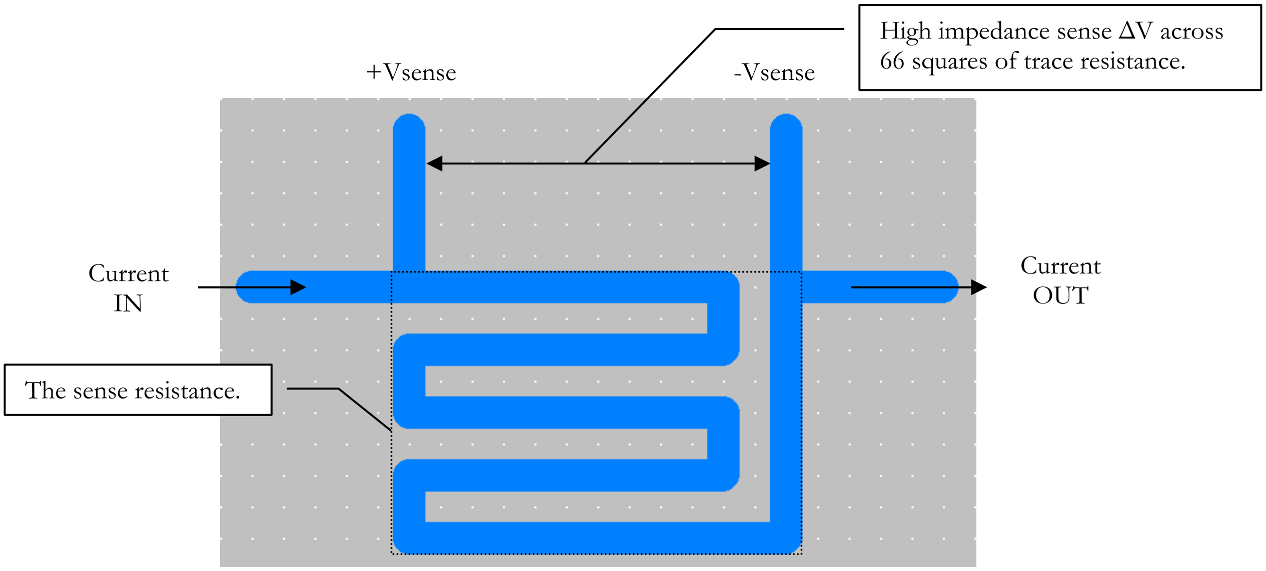
Ignoring a trace's power dissipation can result in problems. We have all seen examples of PCB traces that have experienced over-currents. The traces have become hot enough to either fuse, or, more impressively, to have delaminated from the board. Obviously, a situation we want to avoid.

PCB Trace Loss at High Frequency:

Recall that we can pretty much neglect the DC resistance loss of a trace at high frequency. High frequency losses are dominated by skin effect and dielectric losses.

Kelvin Sense

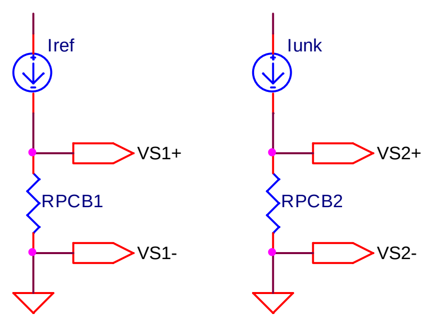
A common use of a trace's resistance is to sense moderately large current flow. When this is done, it is often important to use a Kelvin Sense technique, as illustrated in the simple example below:



Compensation & Tolerance Mitigation Techniques

When using a trace's resistance, it is important to take into account its large TCR, as well as processing variance. *Also, remember that the outside trace's thickness can be significantly different than what the standard copper weight would imply.* For example, to mitigate the TCR variation, one could use an appropriately positioned temperature sensor to monitor the trace's temperature, and then compensate.

There are numerous techniques for dealing with circuit elements that have inconvenient tolerances and temperature coefficients. The most commonly used technique is to measure the unknown and ratio compare to a known reference. As an example, let's assume we want to use a PCB trace to measure a circuit current. We could implement this by using two trace resistances, one for the unknown, and one for the reference current as illustrated below:



By inspection, we see that:

$$\frac{V_{S2}}{V_{S1}} = \frac{I_{UNK} \bullet R_{PCB2}}{I_{REF} \bullet R_{PCB1}} \quad \text{and therefore:} \quad I_{UNK} = I_{REF} \bullet \frac{R_{PCB1}}{R_{PCB2}} \bullet \frac{V_{S2}}{V_{S1}}$$

The PCB traces should be co-located on the same layer, thermally identical, and sized as needed so that the processing physical tolerances will not be an issue. V_{S1} and V_{S2} can then be processed with either analog or digital techniques.