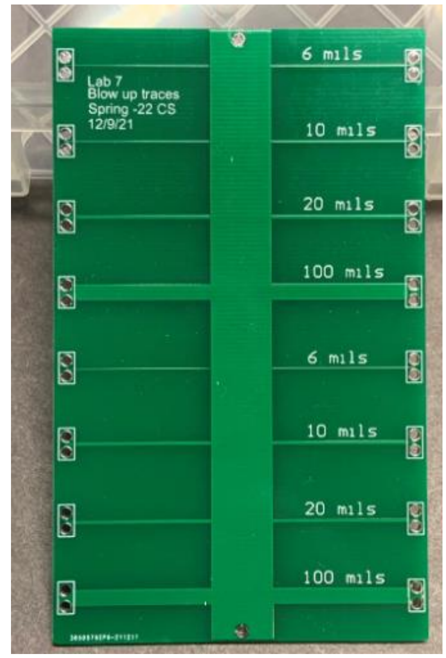
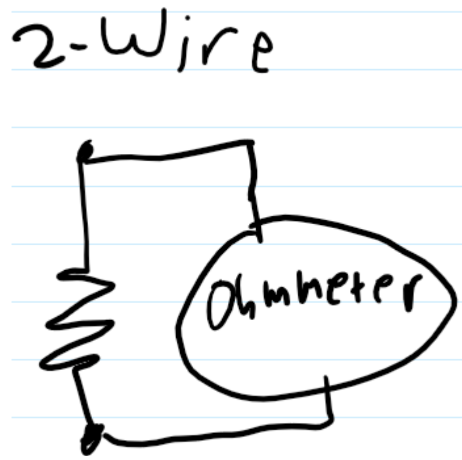
Nathan Herrington
ECEN 3730
Professor Blum
10/4/2023

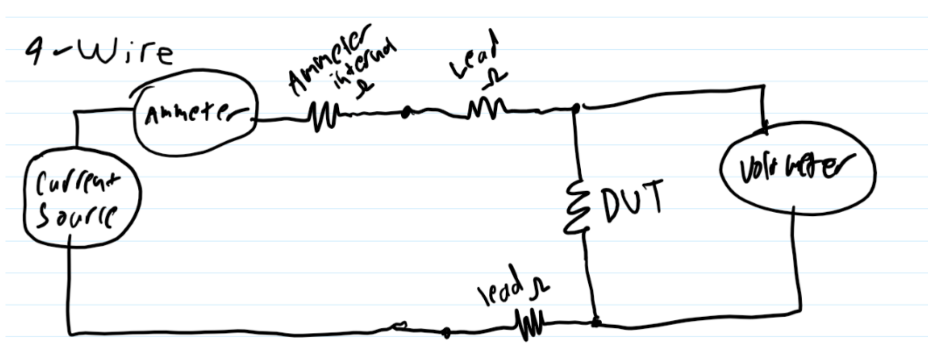
Lab 11 & 12 Report

During these labs, I ran 2 experiments to better understand the properties of trace on a pcb. During the first lab, I studied and measured the resistance of 4 different widths of trace. Traditionally, one would use an ohmmeter to measure these resistances, but, due to the very small scale of the resistance from the trace, this method isn't possible. This 2-wire method isn't applicable in this instance because there is also a resistance in each of the leads and an internal resistance in the ohmmeter. These resistances are very small, but are also either of the same magnitude or larger than the resistances due to the trace, meaning one can't confidently measure the resistance of the trace. A different 4-wire method is used to measure smaller resistances. This method involves supplying current through the DUT, measuring that current with an ammeter and the voltage drop across the DUT. Using these 2 values, one can calcuate the resistance of the DUT by dividing the voltage by the current. A good voltmeter will have a sufficiently high impedance that there will be no significant drop in voltage in the leads. Circuit diagrams of both methods are shown to the right.





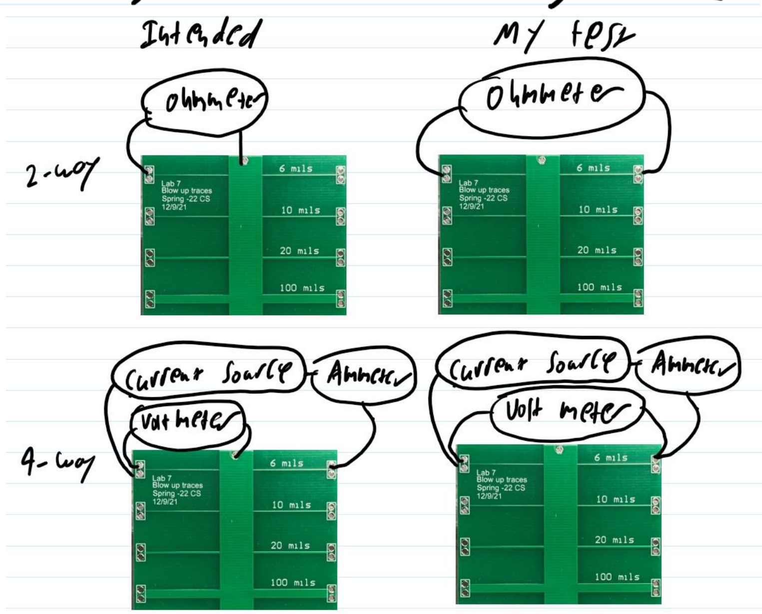
I calculated the expected resistance of the traces of 4 different widths under the assumption that its 1 oz copper. The table at the beginning of the next page shows the estimated resistances, 2-wire measurements, and 4-wire measurements. Unfortunately, there is a large error between my estimated resistance and my



Nathan Herrington
ECEN 3730
Professor Blum
10/4/2023

measured resistance. This is likely a result of the fact that I calculated the resistance of a 2.5 inch long trace instead of the 1 inch long trace that was intended for the experiment. I then measured using the full width of the board to measure the resistance instead of measuring from the test point at the top to the test point at one of the sides. These both are different from how this board was intended to be tested. The 4-way measured resistance is a more accurate resistance for the length of trace that I intended to test. A drawing of the difference between the intended usage and my usage is shown to the right.

	estimate	Bench DMM 2-wire	4-wire
6 mil	208 1/3 mΩ	250 mΩ	190 mΩ
10 mil	125 mΩ	210 mΩ	75 mΩ
20 mil	62.5 mΩ	190 mΩ	36 mΩ
100 mil	12.5	170 mΩ	7.5 mΩ



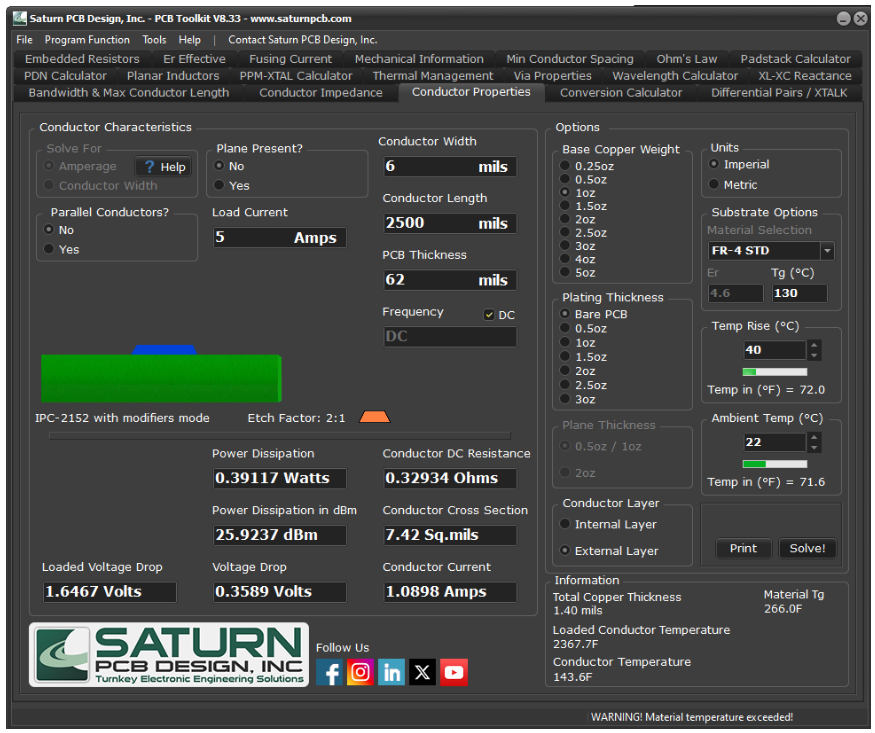
In the next experiment, I tested the limits of what currents I could safely push through a trace. I tested 2 traces with widths of 6mil and 100mil. The table below shows the currents I found each trace could handle before getting warm, hot, and smoking.

	warm	hot	smoke
6m	1.6 A	2.4	4.6
100m	5.6A	8.6	couldn't

Based on these results, I would feel safe putting 2A through a 6mil trace because that leaves room for more than 2A of error in my calculations while designing a circuit

Nathan Herrington
ECEN 3730
Professor Blum
10/4/2023

before the trace would start to get hot enough to begin damaging the circuit. I put this safe limit above my measured warm amperage because I counted the trace as warm the moment I felt the trace get warmer than the surrounding PCB, meaning there is still a current buffer before the trace gets hot. For the 100mil trace, I would feel comfortable pushing 6A and I couldn't get the trace to start smoking because I reached 10A on my power supply, and the power supply couldn't push more than that. I chose 6A as a safe current because, again, I measured the warm current very early. This means I could underestimate my current load by about 3A before the trace gets hot, and even more before the trace begins to smoke. The difference between the current ratings is the reason why power traces on a PCB should be wider than signal traces. The amperage for the entire circuit may pass through a power trace, meaning it needs to have a much higher rating that the signal traces that it supplies. 100mil is overkill for the majority of circuits, so 20mil trace would be a more reasonable width for a power line to increase trace density.



Unfortunately, when I checked my findings against Saturn PCB Toolkit, I found a very large disparity between what amperage I found to be safe and what amperage is safe. I should have found that the trace was very hot at 1A, with the trace being 40 degrees C higher than the ambient temperature, but instead I found that it was starting to get warm at 1.6A. This is because the Saturn PCB Toolkit is calculating a straight, 2.5in trace while my board had a large section of copper down the middle that connected all of the traces. This meant my test was able to distribute significantly more heat than the simulated trace due to the larger surface area.