

Precision Analog Applications Seminar

PCB Layout Tips for High Resolution

Section 9

Layout of printed circuit boards (PCBs) could be an all-day discussion. Rather than give you a set of rules to follow, which almost always have exceptions, we'll look here at some general principles and then review a number of actual board layouts that illustrate common mistakes.

Precision Analog Applications Seminar

PCB Layout Tips

Outline

- ◆ **LSB size**
 - Impact on layout decisions
- ◆ **Basic PCB Design Principles**
 - For High Resolution
- ◆ **PCB Layout Reviews**
 - Learn from the misfortune of others!

Precision Analog Applications Seminar

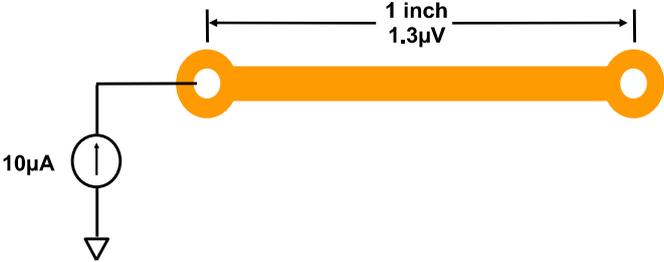
PCB Layout Tips LSB Size and Its Impact

- ◆ **Signal Range Is Critical**
 - **±10V is a 20V Range**
 - 16 Bits: $20V/65,536 = 305\mu V$ Per LSB
 - 24 Bits: $20V/16,777,216 = 1,192nV$ Per LSB
 - **±2.5V is a 5V Range**
 - 16 Bits: $5V/65,536 = 76.3\mu V$ Per LSB
 - 24 Bits: $5V/16,777,216 = 298nV$ Per LSB
 - **±0.020V is a 0.040V Range**
 - 16 Bits: $0.040V/65,536 = 0.610\mu V$ Per LSB
 - 24 Bits: $0.040V/16,777,216 = 2nV$ Per LSB

Where ±10V gives you an LSB of 305μV, a 5V signal only gives you 76.2μV for a 16-bit LSB. When gain is added into the system, the resolution can sink into the noise.

Precision Analog Applications Seminar

PCB Layout Tips
High Resolution Measurements



The diagram illustrates a PCB layout tip for high-resolution measurements. It shows a 10µA current source connected to a 1-inch trace. The voltage drop across the trace is 1.3µV. The trace is highlighted in orange, and the current source is represented by a circle with a vertical line and a downward arrow.

- ◆ 1 inch (7 mil) trace of 1/2 oz copper with 10µA of current => voltage drop of 1.3µV
- ◆ **4 LSBs at 24 bits!**

You must be careful with the PCB layout for a 12-bit design. For a 16-bit design you have to do everything correctly to achieve that performance; it should be obvious that a 24-bit design will be affected by any errors in the layout.

Precision Analog Applications Seminar

PCB Layout Tips Basic PCB Design Principles

- ◆ Separate Analog and Digital Signals
- ◆ AGND and DGND connected at ADC/DAC
- ◆ Provide Good Ground Return Paths
- ◆ High Frequency Bypassing
- ◆ Minimize Inductance
- ◆ Control Thermocouples
- ◆ Differential Signal Measurements

The separation of the Analog and Digital section of the PCB keeps noisy digital from the low level analog. However, they must come together at the ADC or DAC. It is important that the Analog Ground and the Digital Ground be connected together at the ADC or DAC. This allows a quick return for the ground currents as the analog and digital portions of the device communicate.

Inductance is determined by the size of the loop of current. Providing a path for return currents next to the signal trace will reduce the inductance. That is the advantage of a solid ground plane. If a reasonably solid plane can't be achieved with two layers, then more layers should be used.

Precision Analog Applications Seminar

PCB Layout Tips PCB Layout Process

- ◆ Start with a Split Ground Plane
- ◆ Digital Signals over the Digital Plane
- ◆ Analog Signals over the Analog Plane
- ◆ Then as a Final Step – Remove the Split
- ◆ The Signal Return Currents will flow only Next to the Signal Trace. (A larger path will have increased inductance)

Since AGND and DGND are connected at the ADC/DAC and also at the power supply, there is a ground loop created. By having a solid ground plane this can be eliminated. The analog signals will still just flow in the analog portions of the ground plane and the digital signals will only flow in the digital portion of the ground plane. The reason why they will not travel throughout the board, is because a larger path has a larger inductance. The signals will travel through the path of lowest impedance.

Precision Analog Applications Seminar

PCB Layout Tips

Offset Voltages

- ◆ A temperature differential with any two metals will create a thermocouple
- ◆ This includes PCB feedthroughs
 - A different number of feedthroughs for both sides of a differential signal will create an offset that varies with temperature.

Your board design can create additional offset voltages because of the traces and feedthroughs. To minimize offset it is good to route the signal traces differentially next to each other so that they will see the same thermal gradients and the same number of feedthroughs.

Precision Analog Applications Seminar

PCB Layout

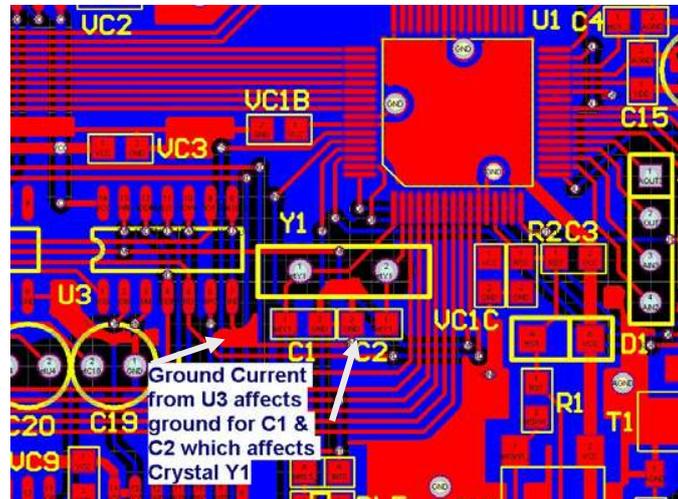
A Rogue's Gallery

We'll now look at a few examples of circuit layouts that make some common mistakes.

Precision Analog Applications Seminar

PCB Layout Tips

Layout Review #1

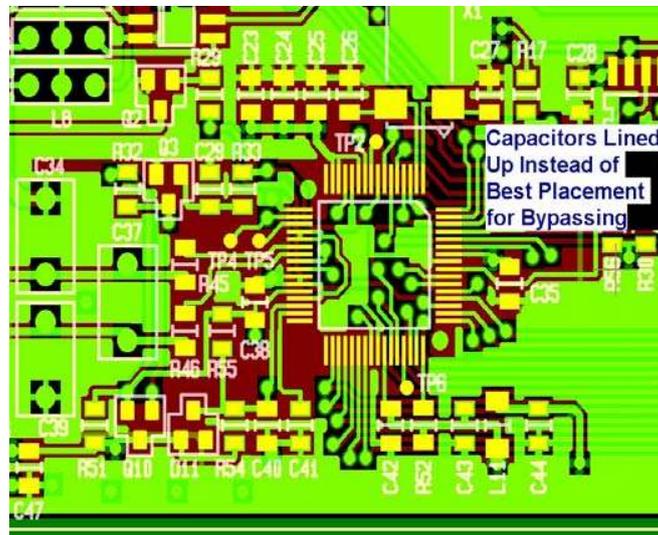


The ground current from U3 was routed to the ground point for the crystal (Y1) capacitors and then to the rest of the ground in the system. This caused a significant effect on the oscillator and made the clock very unstable.

Precision Analog Applications Seminar

PCB Layout Tips

Layout Review #2

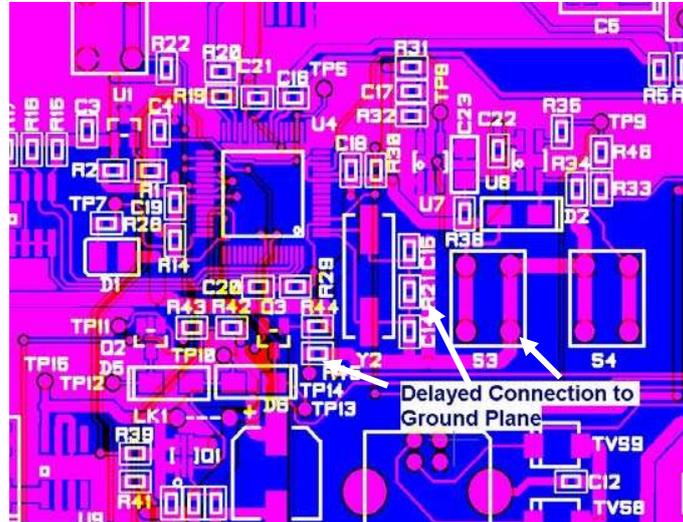


If we position capacitors and components with an order and goal of symmetry, the results could be unnecessary inductance and resistance that are introduced which will reduce the performance of the system.

Precision Analog Applications Seminar

PCB Layout Tips

Layout Review #3

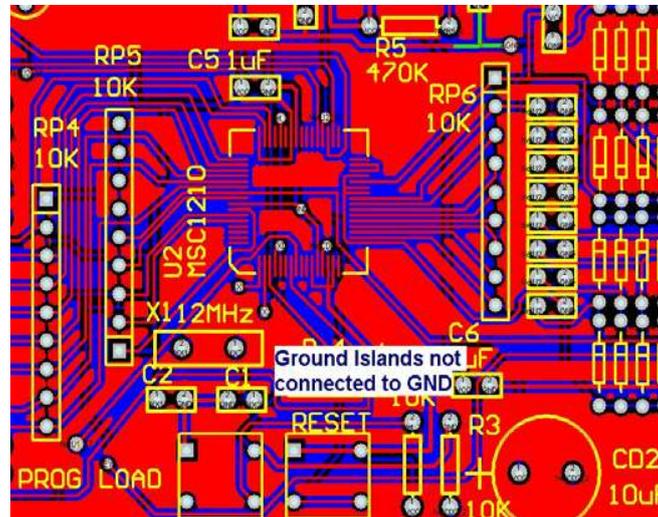


Here this board is designed with a reasonably solid ground plane and rather than connecting directly with feedthroughs to ground plane, there are large traces to a centralized ground connection point. The extra inductance and resistance, even with wide traces, will reduce the performance of the board compared to feedthroughs (or multiple feedthroughs) placed close to the component's ground connection.

Precision Analog Applications Seminar

PCB Layout Tips

Layout Review #4



These top planes seem to be just an area fill for the gaps in the traces. When the plane areas on the top layer are not connected to the ground plane on the bottom, that means that the ground plane on the bottom doesn't have a continuous path available for current. If these top ground planes were connected to the bottom ground plane, the ground return currents would have a lower inductance path.