Low Noise Printed Circuit Board Design

Matt Affeldt

November 16, 2012

Design Team 6 – ECE480

Keywords:

Low Noise, Impedance, Capacitance, PCB,

Printed Circuit Board, layout, design

Summary:

This application note is intended to be a guide for low noise, high efficiency designs of printed circuit boards (PCB). This includes mainly layout considerations of PCBs to limit EMI between components as well as dealing with some non-board generated noise issues.

Table of Contents

Introduction

In this application note, how to design a printed circuit board (PCB) to be low noise in order to improve the quality of the desired circuit will be described. It is assumed that the reader has some background knowledge of the circuits, but the basics of printed circuit boards will be described. Designing low noise printed circuit boards requires avoiding excess impedance, stray capacitance, and antennas. Any large impedances, stray capacitance or antennas can cause noise, instability or both. Noise is typically a high-frequency signal on top of the desired signal. This noise can often interfere with measurement techniques and can introduce a significant amount of inaccuracy in the results depending on how noisy the circuit is. Instability, on the other hand, is much worse. Instability can cause complete failure of a circuit and potentially damage components in the circuit. Stability concerns are commonly found with operational amplifiers, and are also a concern when using most active components such as microcontrollers and power converters. Poor planning can lead to oscillating signals which can eventually grow, swinging voltage from ground to the power supply voltage and creates large current. This application note will detail how to reduce noise, improve performance and avoid circuit instability.

Printed Circuit Board Background

A printed circuit board is a layer of copper that has been cut or etched in order to create electrical connections between pads. These pads will then be connected to a component in order to realize a specific circuit in a cleaner, more compact form than a bread board (or proto-board). These copper layers can be stacked with an insulating epoxy in between to allow very complex circuits to be realized on a dimensionally small board. Each layer will only make the board slightly thicker but not any wider or longer. Each connection from one pad to another is called a trace. When it is necessary to connect one trace to another trace on a different layer, a via can be used which is a hole that is plated with conductive material. Finally, in multi-layer boards it is common to have 'power planes' such as ground and the power supply voltage to make it easy to power and ground all the components. There is no perfect way to design a printed circuit board. There are some specific do's and don'ts but generally speaking, design choices involve tradeoffs in performance between two or more attributes.

Printed Circuit Board Design

This section will describe how to design PCBs to reduce internal noise generated by the circuit components and physical layout of the board. This includes suggestions to reduce impedance across traces as well as how to keep the paths across the ground plane very low impedance, avoid stray capacitance, avoid unwanted inductance, and avoid creating antennas. This section also discusses some brief stability issues and how to design a PCB to avoid situations that can result in circuit instability.

Reducing Impedance

High impedance is the easiest noise source to mitigate, below are 2 examples of how to reduce impedance-based noise from the circuit.

1. **Avoid long traces.** Long traces are inherently more resistive than short traces. Below in Figure 1 shows a bad example of how to lay out a component. There is a long trace between power and the capacitor (which is polarized). Figure 2 shows that it is possible, by rotating the capacitor 180° and moving it to the right, to make the trace much shorter so as to result in a lower impedance for the trace.

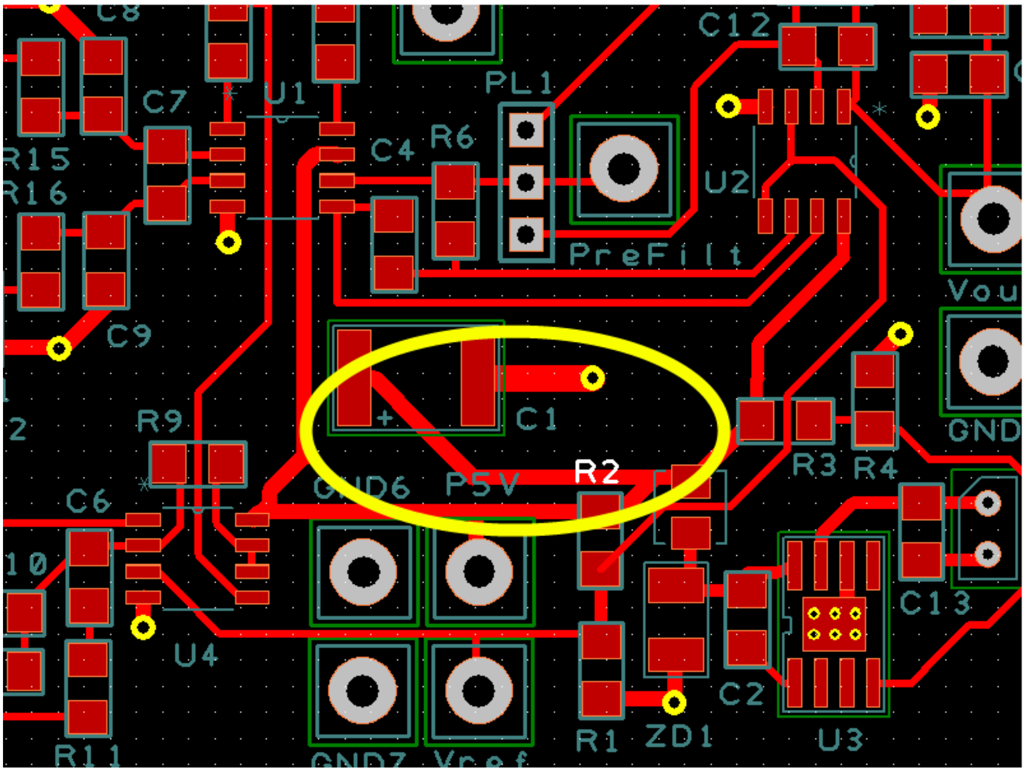


Figure 1. Long trace for connecting capacitor to power

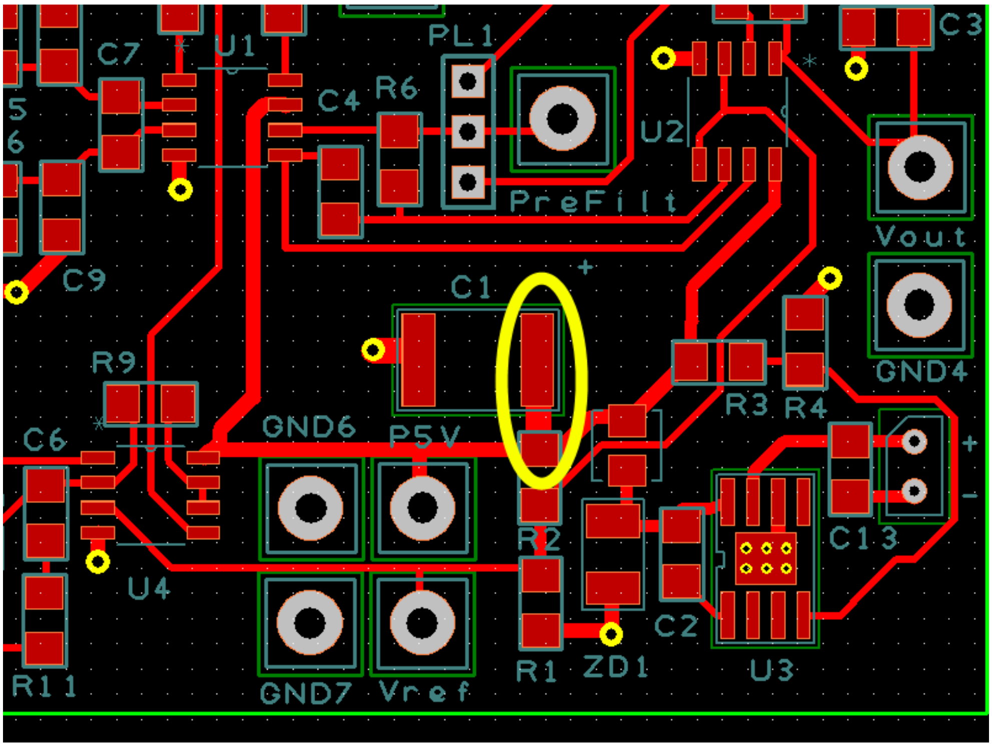


Figure 2. Shorter trace for capacitor to power

2. **Avoid cutting the ground plane.** The following figure illustrates a top layer and ground plane. As shown by the letter C, the ground plane has a large slot in it. This creates a large impedance for any voltage difference between the top and bottom of the board. By cutting the holes for the component, more like letter D, the area between the pins allows current to flow. This significantly reduces the impedance of the ground plane. In addition, traces in the ground plane should be avoided whenever possible. When traces in the ground plane are a necessity, avoid making the traces as shown in A. This again cuts the ground plane into left and right sections with a large impedance between the two halves. The letter B shows a much better path, where it goes around the outside of the ground plane. Although this does increase the impedance of trace B, it is better than increasing the impedance of the entire ground plane.

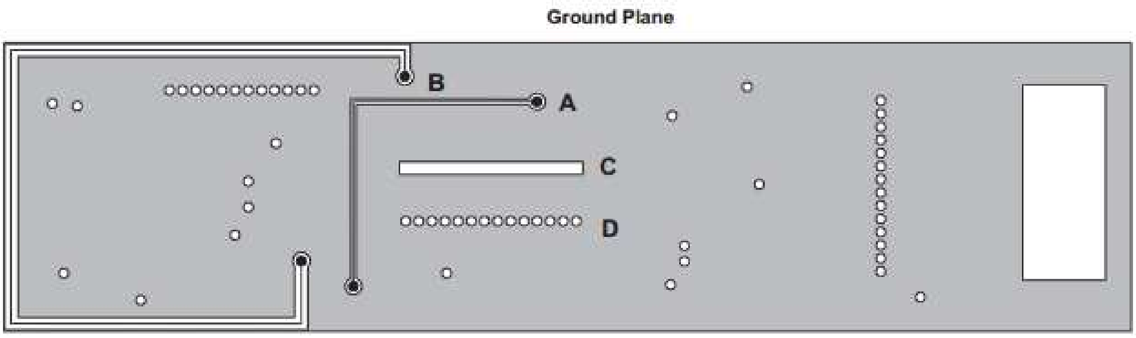


Figure 2. Ground Plane Trace Styles

Avoiding Stray Capacitance

This section will describe two techniques to help reduce the stray capacitance. It is important to be mindful of stray capacitance, which is always present, due to stability and noise issues. Operational amplifiers have stability issues when the input node is capacitive, and will cause oscillation on the output of the amplifier.

1. **Keep power lines away from sensitive nodes:** Power traces carry a high voltage and the amount of charge on a capacitor is equal to the voltage difference multiplied by capacitance. Although the capacitance will be small due to the physical layout, the potentially large voltage difference between a power trace and signal trace will cause charge to build up. When the signal voltage changes the charge buildup can be injected into the circuit causing noise in the circuit. As shown in Figure 3, the positive and negative power traces go around the circuit to avoid the most critical node, the input node, to the op-amp (circled in yellow).

2. **Cut the ground plane:** *NOTE: This suggestion comes at a price and should only be used when necessary. Since cutting the ground plane increases the impedance of the ground plane, only do it for nodes that are very sensitive to capacitance such as input nodes to an operational amplifier.* In Figure 3, the input node to the operational amplifier is circled. As already discussed, inputs to operational amplifiers are particularly sensitive to capacitance which can cause circuit instability. Multi-layer PCB are essentially capacitors, conductive plates separated by an insulator, and therefore every node above the ground plane will have some capacitance. To mitigate this capacitance, the ground plane (and all other planes below or above the node) can be cut, removing the conductive plate that the node would couple with.

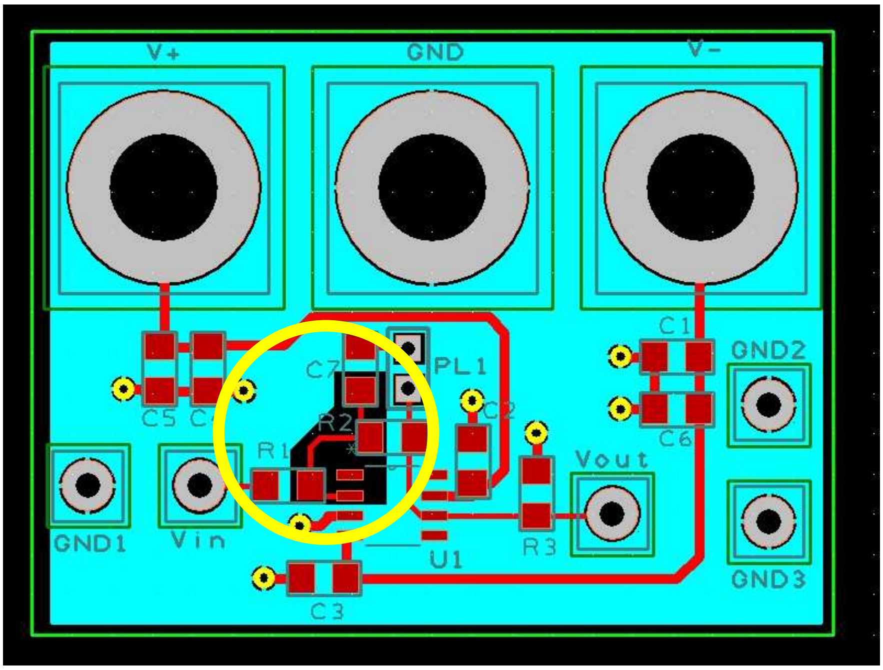


Figure 3. Sample board design

Avoiding Antennas

It is important to note that according to Texas Instruments, an effective antenna trace is at least 1/4 λ for a given frequency. It is given that C = λƒ where C is the speed of light, λ is the wavelength and ƒ is the frequency. This means that as frequency decreases, wavelength increase and therefore the effective size of an antenna increase. For a frequency of 100MHz, a relatively large frequency for circuits on a PCB, the effective wavelength is 75cm. It is unlikely that any component on a printed circuit board is this large and generally the entire board will be less than 75cm. This means that components on the board, and typically the traces as well, will not make for a good antenna however connecting cables are often much longer than 75cm. Any cables connected to a printed circuit board can pick up noise and then transfer that noise to the board. Figure 4 shows two different layouts of connectors on a PCB. In Figure 4, the connectors are on opposite sides of the board. This means that if each cable transfers a different amount of noise to the board, there will be a voltage induced across the entire board which will result in a noisy current traveling through all of the components. A much better layout is shown in part b of Figure 4, where the connectors are right next to each other. Any voltage that is induced across the board will be immediately transferred from one cable to the other and will significantly reduce the amount of noisy current that is traveling through the board.

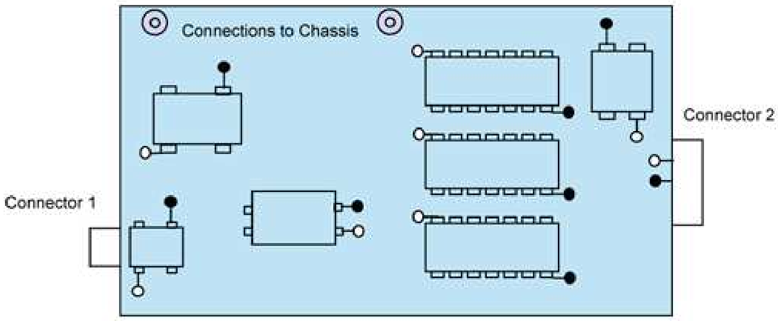
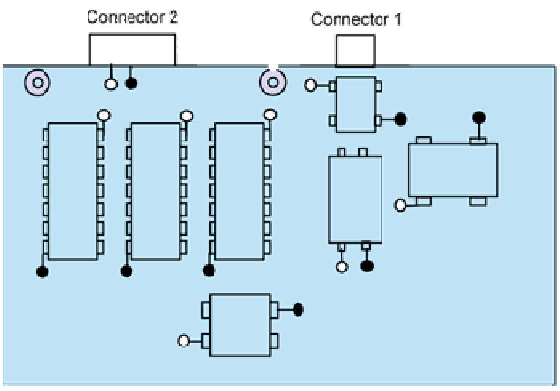


Figure 4. a) Connectors on opposite sides of the board (above) b) Shows connectors next to each other on the board (right)



Conclusions and Recommendations

As stated before, it is important to note that these are all suggestions on how to improve performance of a board. Typically there is a trade of for each suggestion so every scenario should be reviewed individually. Reducing noise in one part of the circuit may cost higher impedance elsewhere. Also, sometimes it is not possible to follow some of these guidelines due to design constraints such as size or the complexity of the circuit. Always be mindful of any specifics of the circuit that is being implemented on a PCB and physical constraints such as connectors that have to plug into another board. For active components, the manufacturer will often have a datasheet including some information on where parts should be placed. For example, Texas Instruments points out that for their amplifiers a decoupling capacitor should be placed as close to the power pin of the amplifier as possible. This suggestion from the manufacture would trump moving the capacitor further away to allow a different trace to be shorter.

References

Texas Instruments Guidelines specifically for EMI:

http://www.ti.com/lit/an/szza009/szza009.pdf

Printed Circuit Board Issues – Chapter 12 of *Basic Linear Design* by Hank Zumbahlen

http://www.analog.com/library/analogdialogue/archives/43-09/EDch%2012%20pc%20issues.pdf

Texas Instruments Presentation on Analog Designs of PCB

http://www.x2y.com/filters/TechDay09kr_hpa_Track2_1_Precision_Analog_Designs_Demand_GoodPCB Layouts%20_JohnWu.pdf