

Op Amp PCB Layout – Mixed Signals, Grounding & Bypass Capacitors

TI Precision Labs – Op Amps

Prepared and presented by Daniel Miller



1

Hello and welcome to the TI precision labs series on PCB layout techniques for op amps. In this lecture, we'll discuss some common questions and best practices regarding mixed signals, grounding, and bypass capacitors for op amp layouts.

What we will cover

1. Separate analog and digital signals
2. Use proper grounding
 1. Ground planes
 2. Ground vias
3. Place bypass capacitors near supply pins
 1. Noise suppression
 2. Transient currents

2

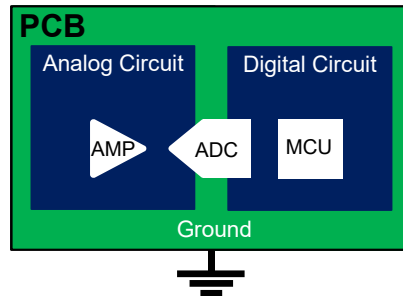


In the first video of this series, we took a look at parasitic impedances of op amp traces including their causes, effects, and solutions. We will now consider some of the broader concerns with op amp PCB layouts.

We'll start by reviewing the importance of separating analog and digital signals to avoid unwanted coupling into amplifier circuits. From there we'll cover some tips on proper grounding. Grounding in and of itself is a vast topic with quite a bit of depth. However, we will focus on some of the most important grounding practices here. Finally, we'll consider the use of bypass capacitors on the supply pins, a technique which is well known but not always well understood.

Separate analog and digital signals

1. High speed digital signals can couple into analog traces
2. Separate analog and digital traces
3. Separate analog and digital current return paths
4. Partition board into analog and digital sections with common ground



3

 TEXAS INSTRUMENTS

In the previous video, we covered the vulnerabilities of parallel traces to unwanted signal coupling. For example, two signal traces in parallel can form the two metallic plates of a capacitor and allow for capacitive coupling. Similarly, two signal traces in parallel can experience inductive coupling by Faraday's law.

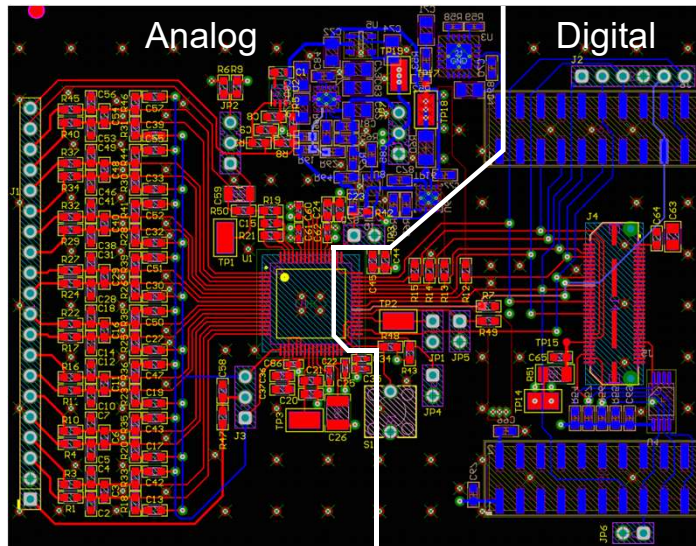
These possibilities are even more of a concern when digital and analog signals are placed near each other. The fast and sharp changes of digital voltage levels along with their high speed current pulses can create problems for more sensitive analog circuits. This is true not only for analog signals, but also for power traces and planes.

Consequently, it is important to keep the signal traces, power traces, and current return paths of analog and digital circuits separate whenever possible. Keeping the traces separate can be straightforward. But ensuring the current return paths do not cross can be more challenging. For this reason, the easiest solution is typically to partition the board into analog and digital sections. This simple strategy ensures the analog circuitry is kept apart from the digital circuitry.

It is important to note that this does not mean two independent ground planes. While the return current paths for analog and digital signals should be separated, make sure that the analog and digital grounds are still tied together with a solid

ground plane. There should be only one ground node. We will cover grounding in more detail shortly.

Separate analog and digital signals – Example 1



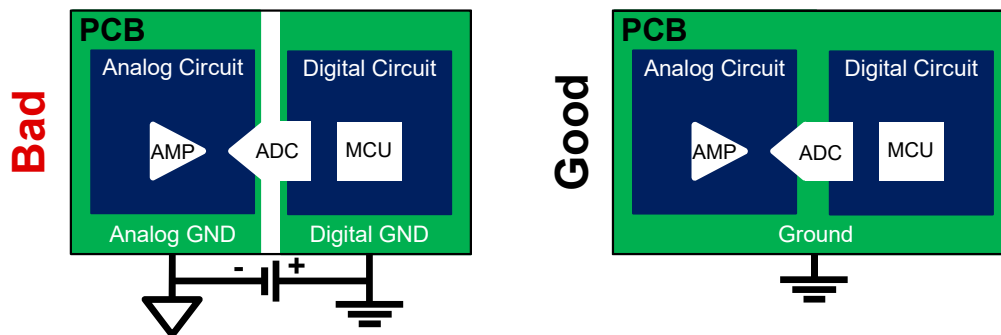
4

 TEXAS INSTRUMENTS

Here's an example of a board under development. In the center of the board is an analog to digital converter, or "ADC". The ADC has analog signals feeding in from the left and digital signals going out from the right. Top layer signals are in red and bottom layer signals are in blue. Notice that all the analog signals with the corresponding components and traces are kept on the left side of the board and all the digital signals are kept on the right. This is a good way to reduce unwanted signal coupling.

Use proper grounding

1. At least one solid ground plane
2. Avoid splitting ground planes, partition instead
3. Analog and digital share ground, but circuits are still separate
4. Place vias to ground near ground pins



5

 TEXAS INSTRUMENTS

When separating analog and digital sections of the board, the natural follow-up question is “What should be done about the ground node?” Grounding is a complicated topic with a lot of good practices to follow and poor grounding can lead to all sorts of performance issues that can be hard to diagnose. So it’s a subject that should not be ignored.

Fortunately, there are a few, straightforward techniques that can be used to prevent many problems. For one, it’s a good idea to have at least one solid ground plane rather than a series of ground traces. This allows current return paths to find more convenient routes with less parasitic impedance and reduces the risk of current loop issues. On a similar note, it is important to keep the ground plane as unbroken as possible to maximize this advantage.

As mentioned before, the analog and digital sections of the board should be partitioned so that the return current paths on the ground plane do not cross each other causing unwanted noise coupling from the digital circuitry into the analog circuitry. At the same time, having the same ground plane for the analog and digital circuits helps to ensure a universal ground voltage for the system. So keep the ground plane connected, but don’t intermingle the analog and digital circuits.

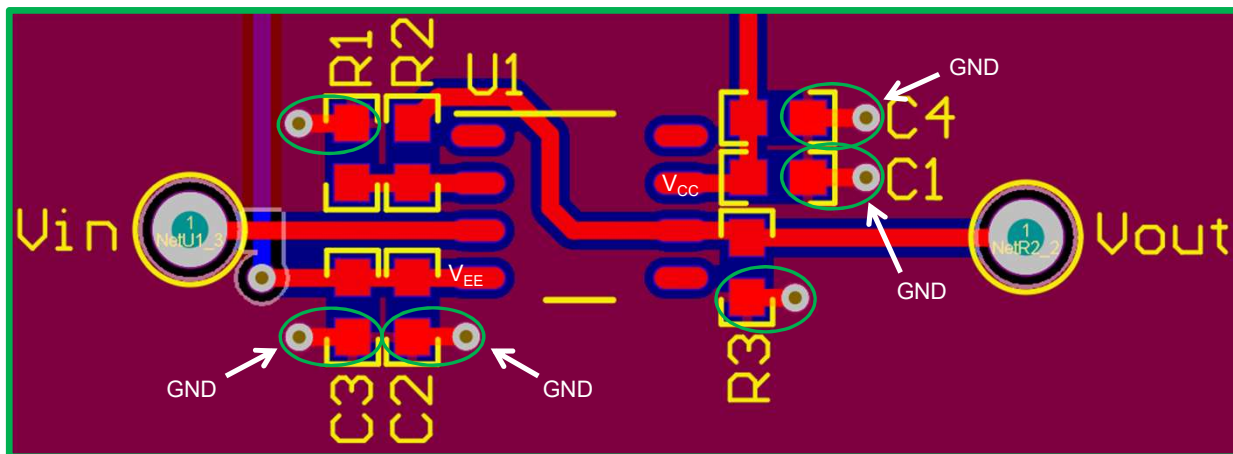
In the example on this slide, you can see an issue caused by poor grounding. On

the left side, the board is designed to keep the analog and digital circuits separate. This is good and helps to avoid crossing of analog and digital traces, supplies, and current return paths. However, the board has two completely split ground planes that are not connected. Because of this splitting of the ground nodes, current that is attempting to return to one side of the board from the other will have to find an external return path. For example, the current return path of the ADC may have to take a long route through an external supply. This results in a lot of parasitic impedance, which can ruin signal integrity and also create an unpredictable voltage difference between the ground nodes.

On the right, the PCB design demonstrates proper board partitioning. The analog and digital sections are separated, but there is a common ground connection. This is not the only way to set a common ground. But, it is a simple and effective method. One technique that can be used to ensure this result is to initially route the board with two separate ground planes: one for digital signals and one for analog signals. Then once the routing is completed, the ground planes can be connected. In this manner the analog and digital sections are kept partitioned while having a common ground.

Finally, make sure to place vias to connect ground planes with other ground pours and traces. If different points on the ground node become separated by a long path or are altogether disconnected, they can develop voltage differences. In other words, not every point labeled "GND" will actually be at 0V, just as when the ground planes are split. Ground vias help to tie the ground node together and keep it at a consistent voltage on a multi-layer board. For this reason, it is especially important to place a ground via close to every component pin or terminal that is connected to GND.

Use proper grounding - Example 2



6

 TEXAS INSTRUMENTS

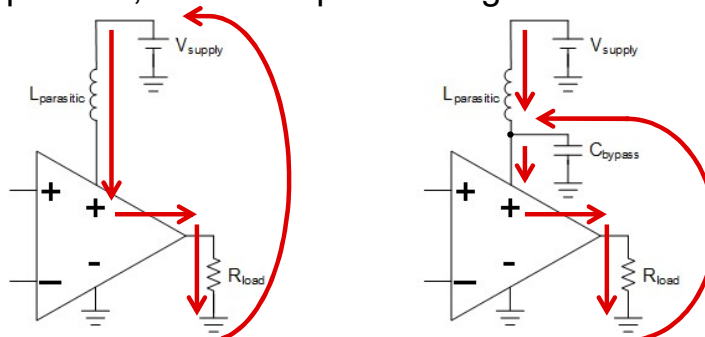
Let's take a look at some of the grounding techniques in practice. Here we have an op amp layout in development. The op amp pins, denoted in red, are on the top layer and are surrounded by a solid ground plane. On the bottom layer in blue is another solid ground plane. The signals shown here are all analog and no digital communication is present.

Notice how the bottom ground plane is kept as solid and unbroken as possible. With the exception of a single trace, all other traces and components are limited to the top layer. This makes it easier for return current paths to traverse the bottom layer without having to take unnecessarily long paths that might be prone to noise coupling or parasitic impedance. Also notice that there are several components with vias to GND. These vias connect the ground pin of the components to the surrounding ground plane on the top layer and the ground plane on the bottom layer. This helps to ensure all portions of the ground node are actually at the same voltage.

Once the layout is approaching completion, the designer may choose to also add stitching vias. Stitching vias are vias placed at regular intervals throughout the board that help to tie the ground node together.

Place bypass capacitors near supply pins

1. Suppress supply noise
2. Provide transient current
3. Place capacitors as close to power pins as possible
4. If multiple capacitors, smaller capacitance goes closer



7

 TEXAS INSTRUMENTS

The last topic to be covered in this video is that of bypass capacitors. Bypass capacitors are placed near op amp supply pins and are connected from the supply to GND. Many engineers know that bypass capacitors are important, but not why they are important.

Amplifiers are designed with clean, low-impedance power supplies in mind. In practice though, the voltage supplies are not always as clean or as low-impedance as desired. At low frequencies, amplifiers have good power supply rejection ratios, often abbreviated in data sheets as “PSRR.” In other words, the output of the amplifier is not heavily affected by a changing supply, assuming the output remains in its linear operating region between the supply voltages. However, the natural PSRR of an amplifier drops with frequency. Thus, higher frequency noise on the supply more easily affects the op amp’s output. To combat this undesired outcome, bypass capacitors can be placed between the supply pins and ground. This allows the noise arriving to the amplifier to find a low impedance path to ground at high frequency.

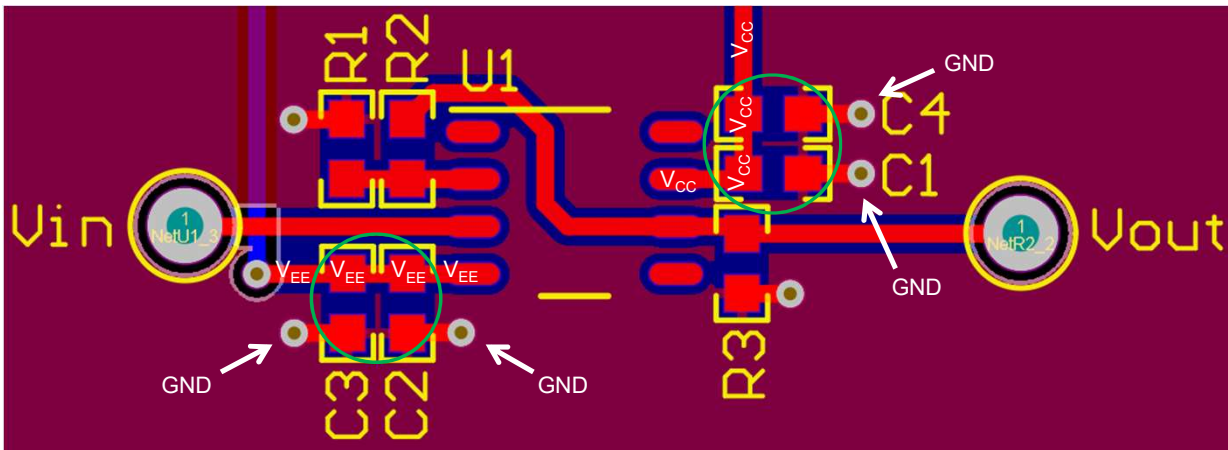
The bypass capacitor also helps the amplifier deal with current transients. During operation, the amplifier may need to provide a sudden change in current to the load. In the absence of bypass capacitors, this surge in current will have to come from one of the power supplies. The current will arrive from the voltage supply, pass

through the op amp and the load, and then return to the supply. If the supply is far away from the amplifier, which may be the case in more complex layouts, then the path of the supply current to the amplifier can have an inductive effect as shown in the circuit on the left. Changes in supply current can lead to changes in the supply voltage seen by the amplifier. This can have bad consequences for the output signal.

Placing a bypass capacitor helps to resolve this issue by providing a short return path for the output current as shown in the circuit on the right. In cases where a transient current is needed by the amplifier, the bypass capacitor can provide this temporary current and a quick return path from ground to the amplifier's supply pin. Thus, the inductive supply path is avoided and the amplifier's supply and output can remain steadier. Remember, that to get this effect you should place your bypass capacitor as close to the amplifier's supply pin as possible. If the bypass capacitor is placed far away from the op amp's supply pin, then it will have a long trace with parasitic inductance to the op amp, thus defeating the purpose of having the bypass capacitor. You should also ensure that the supply trace runs through the capacitor before reaching the amplifier. Again, if the bypass capacitor is not placed between the supply and the amplifier, then the purpose of the capacitor is defeated.

Sometimes, two bypass capacitors of different capacitances are used. The smaller capacitor can act to provide rapid transient currents, while the larger capacitor will filter out a larger band of noise. The use of two bypass capacitors also reduces the equivalent parasitic resistance of the real world capacitors. If multiple capacitors are needed, place the capacitor with the smallest capacitance closest to the pin of the device.

Place bypass capacitors near supply pins - Example 2 continued



8

 TEXAS INSTRUMENTS

In the previous example, we took a look at a simple op amp layout and its grounding setup. The same example layout also shows how to place bypass capacitors.

Take a look at the V_{CC} and V_{EE} pins of the op amp. Notice that they have decoupling capacitors present and that these capacitors are placed as closely to the supply pins as possible. Again, whenever two capacitors are placed on the same supply pin, the smaller capacitor should be closer to the pin than the larger capacitor. For example, common bypass capacitor values include $1\mu\text{F}$ and 100nF with 100nF often being the minimum decoupling capacitance recommended in the data sheet. If both $1\mu\text{F}$ and 100nF capacitors are put near the same supply pin, then the 100nF capacitor should be placed closer.

It is also important to note that the trace from the supply source runs through the capacitors before passing into the supply pins of the amplifier itself. This is important for ensuring the bypass capacitors function as intended.

Finally, remember that supply pins connected to ground, such as the V_{-} pin in a single-supply op amp configuration, do not need bypass capacitors.

Summary

1. Separate analog and digital signals
2. Use proper grounding
 1. Ground planes
 2. Ground vias
3. Place bypass capacitors near supply pins
 1. Noise suppression
 2. Transient currents

9



In summary, strong op amp layouts involve more than simply routing some traces properly. In this presentation, we covered the importance of separating analog and digital signals, using proper grounding, and placing bypass capacitors.

While not an exhaustive list of op amp layout techniques, the list of topics covered by this series of presentations should provide you with a solid foundation and confidence for your op amp PCB layout designs.

Thanks for your time!

10



That's all for now! Thanks for watching.



© Copyright 2018 Texas Instruments Incorporated. All rights reserved.

This material is provided strictly “as-is,” for informational purposes only, and without any warranty.
Use of this material is subject to TI’s **Terms of Use**, viewable at TI.com