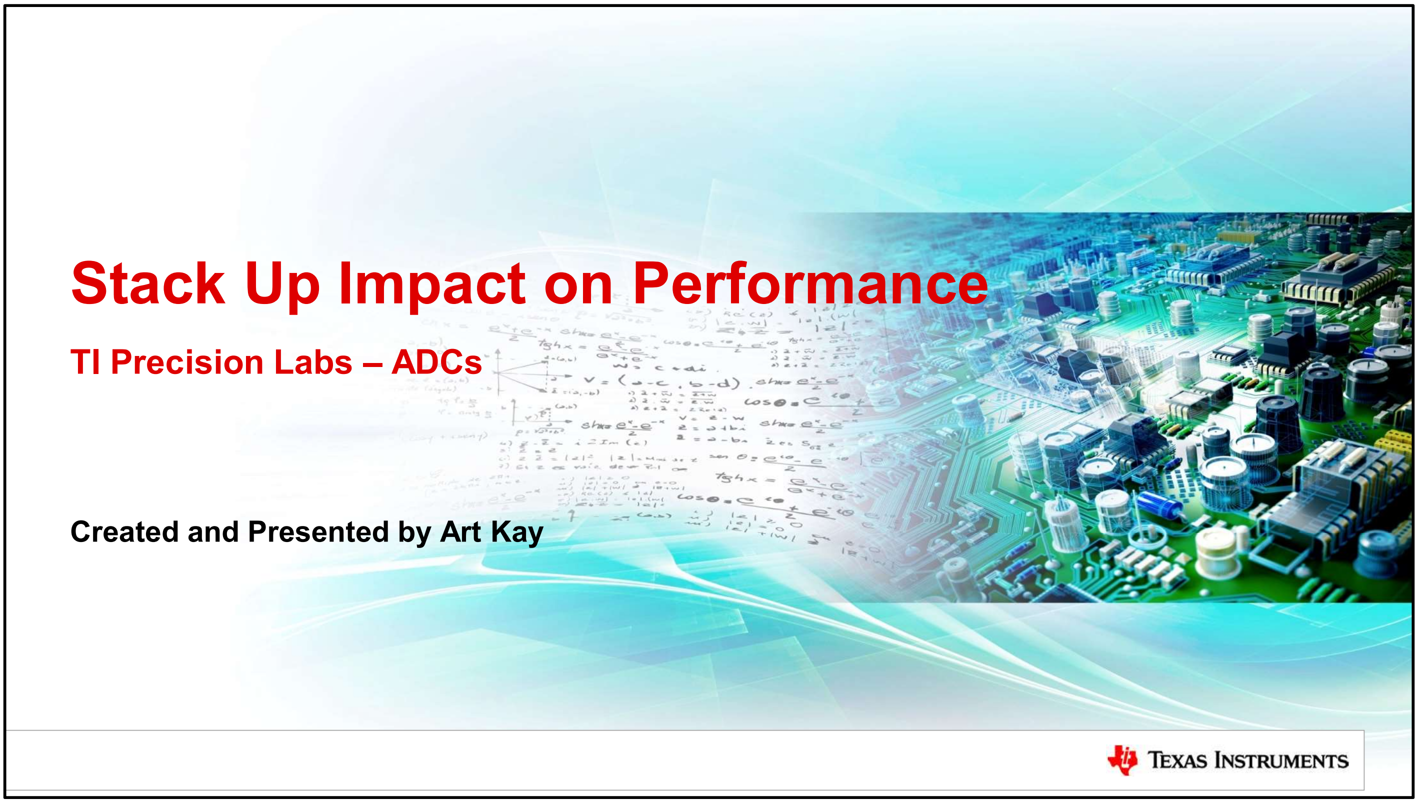
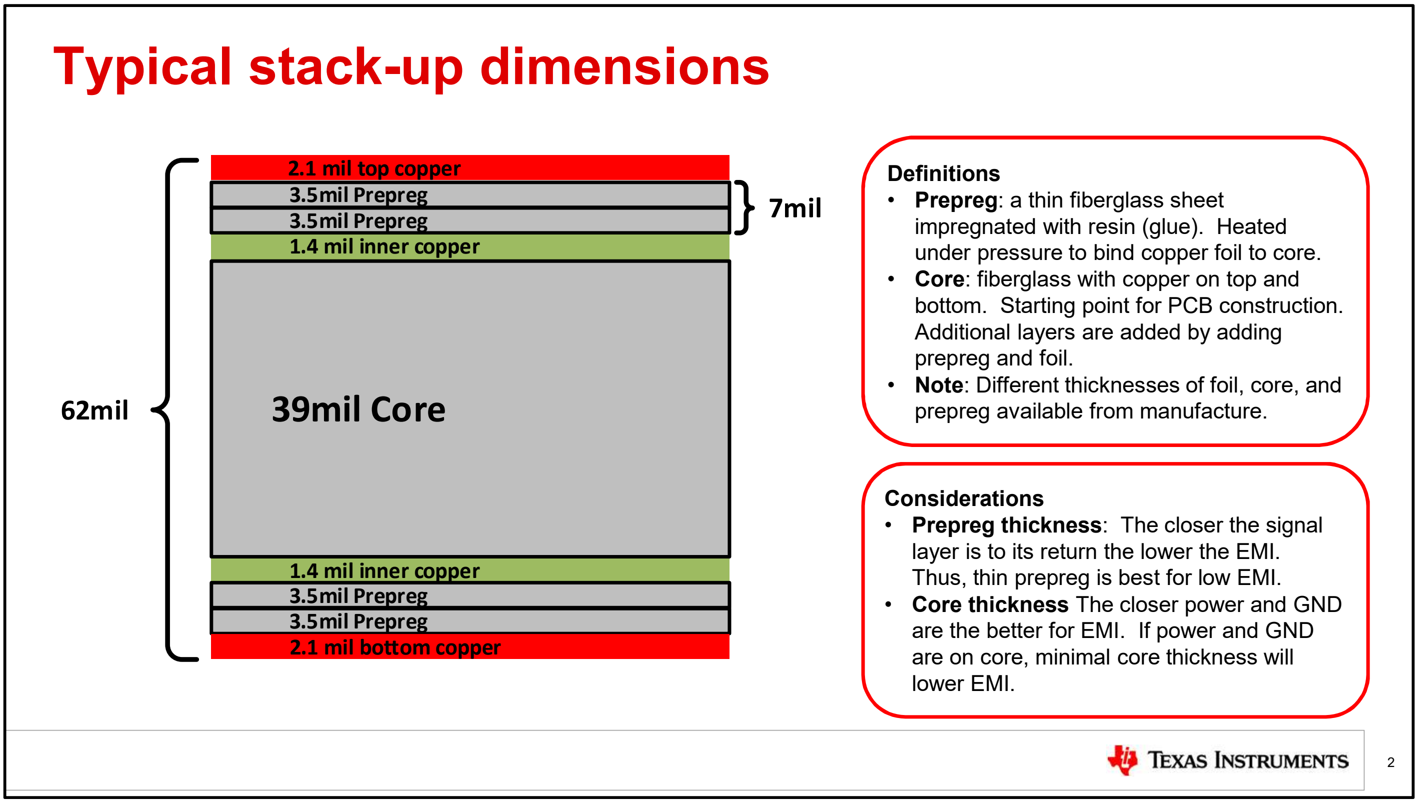
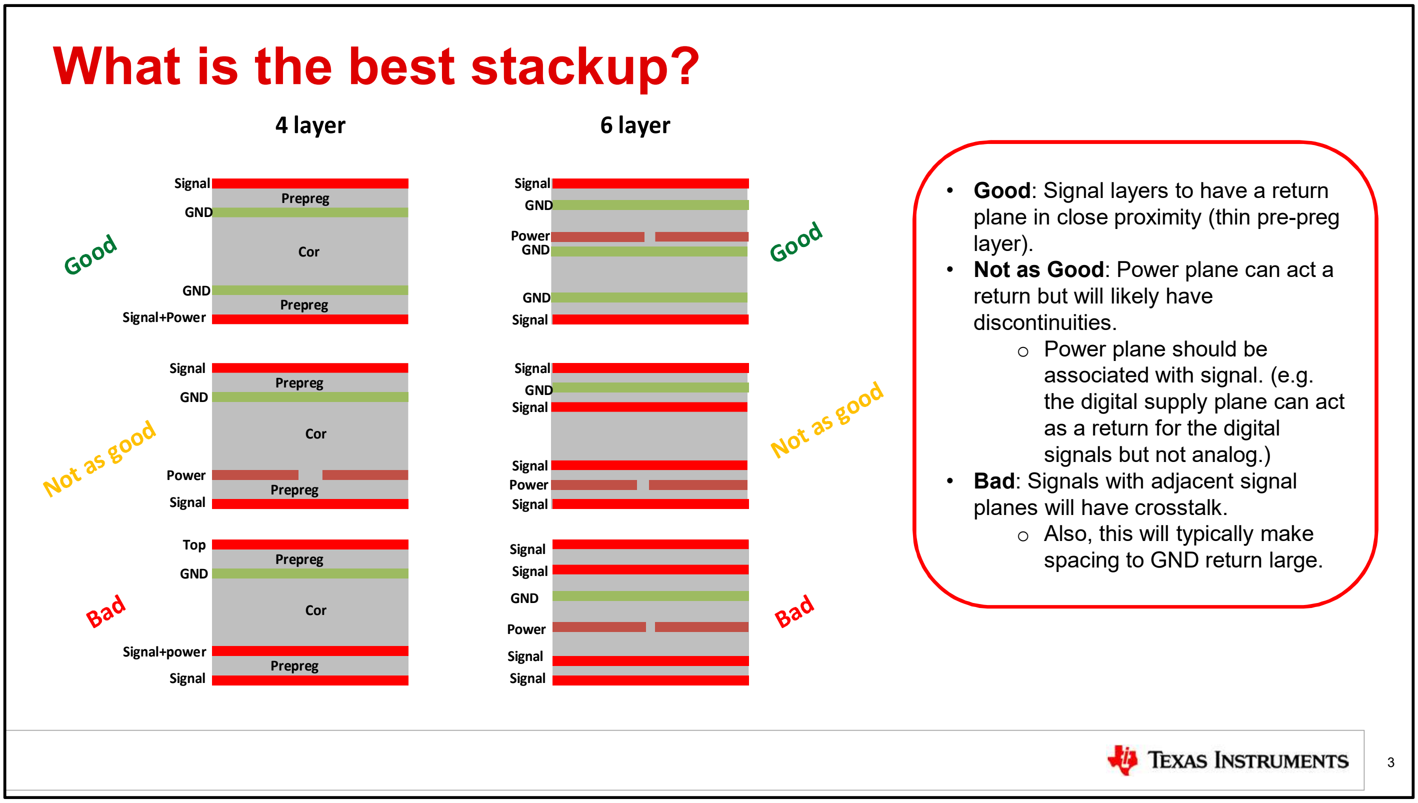


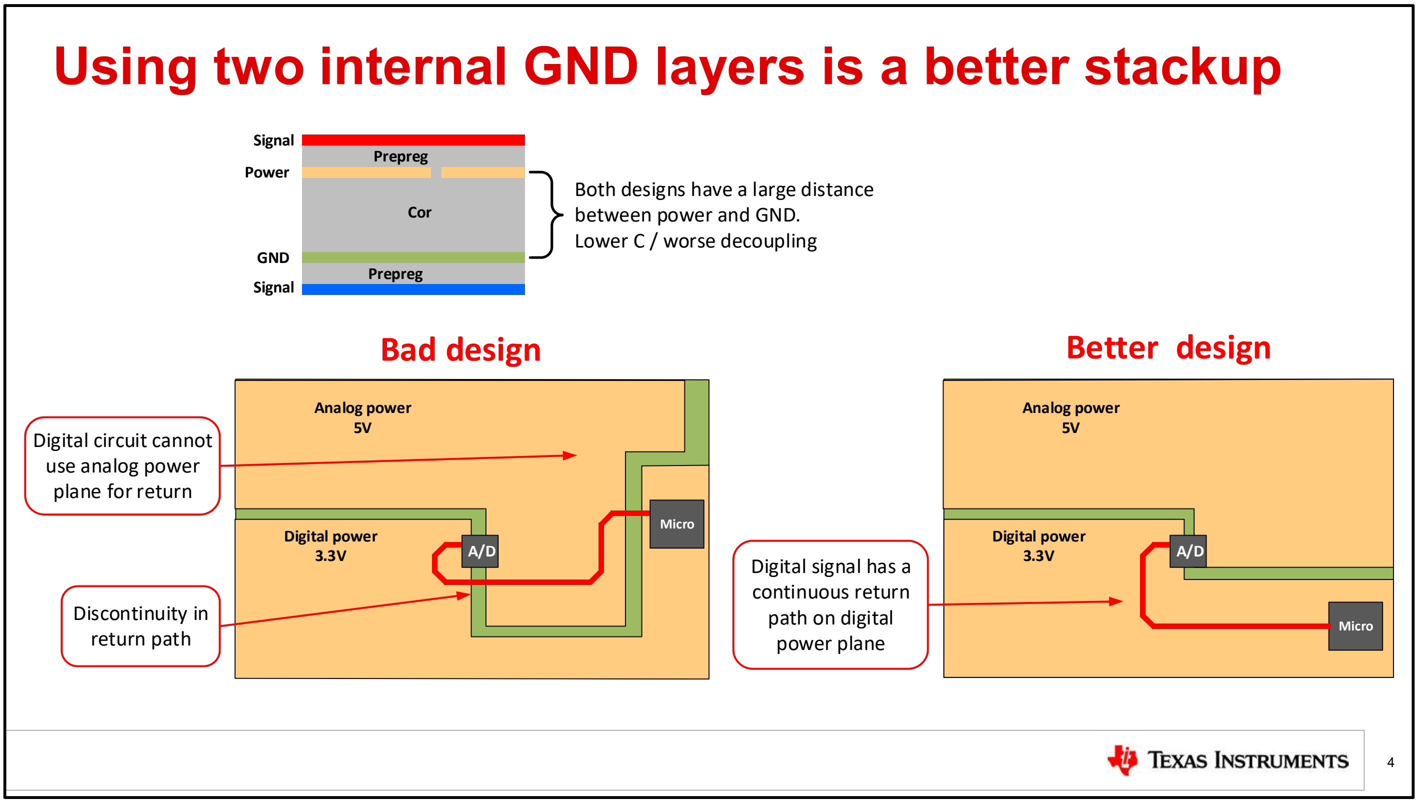
Hello, and welcome to the TI Precision Lab video covering how a PCB layer stackup impacts the system EMC performance. This is part of a larger series on PCB layout for good EMC. This series is specifically intended to cover mixed signal designs where the digital signals are less than 100 MHz and clock rise times are greater than 1 ns. This video looks at how changing the order and function of layers can have significant impacts on performance. Lets start by considering reviewing some PCB terminology.



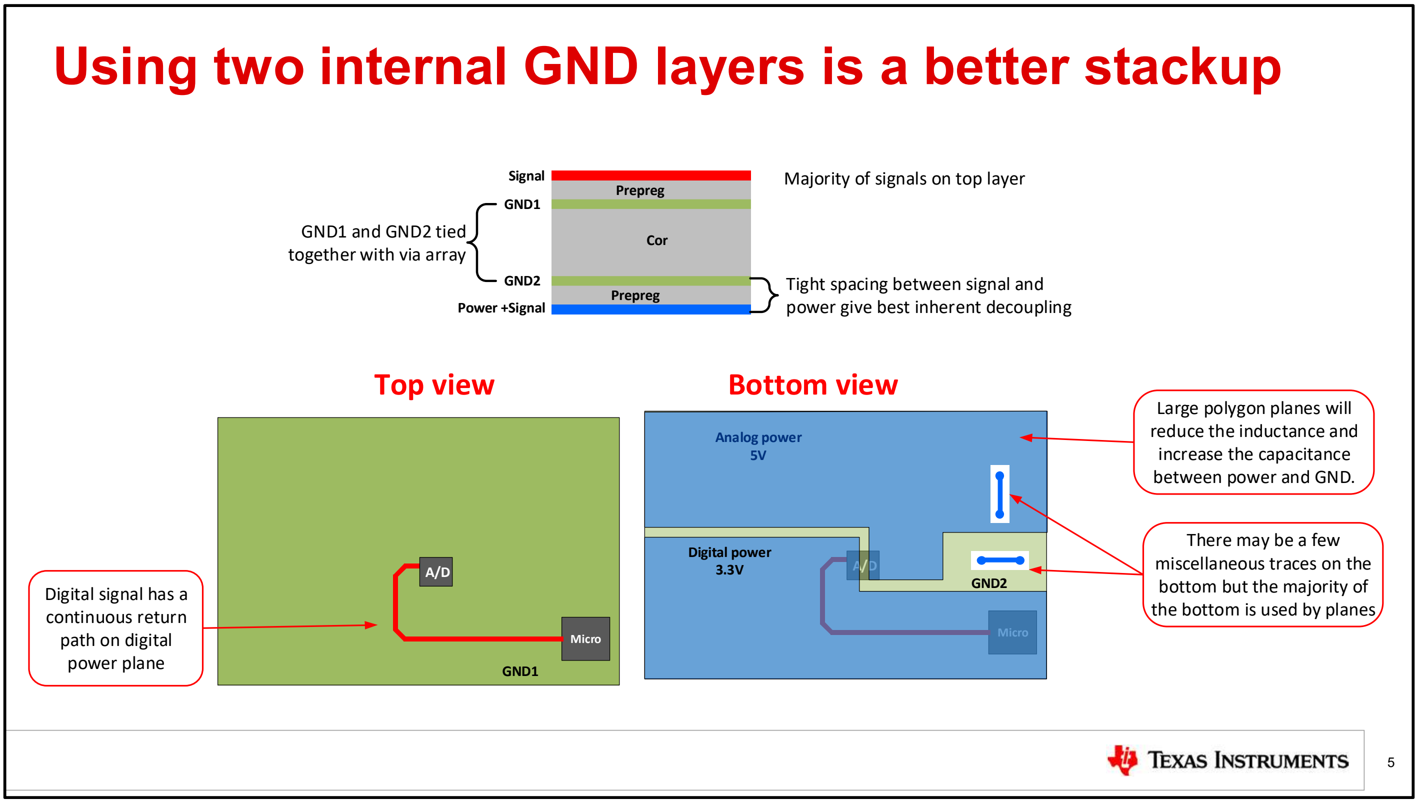
This slide provides a reminder of some key definitions for PCB construction. This example covers a four layer board but could be applied to higher layer counts. The core of a PCB is a rigid fiberglass panel with copper foil on the top and bottom. The core is basically a two layer board. To expand to higher layer counts a thin sheet of fiberglass with glue, called prepreg, is used to attach a top and bottom foil. This example PCB is shows very common thicknesses. For example, a 62mil board is considered to be a standard board thickness, and 7 mil prepreg is very common. Normally the core is fairly thick and the prepreg is thin. Thus, the top layer is very close to the top of the core, and the bottom layer is close to the bottom of the core. For good EMI performance the signal layers should be as close as possible to their associated ground return layer. Using the top is a signal layer, and the adjacent inner layer is use as the ground return is a good choice as the prepreg is generally thin. For good power supply decoupling, it is helpful to keep the power and ground plane close together as well. If the core is used for the power plane and ground plane, it can be difficult to keep the spacing tight as the core is usually a thicker material.



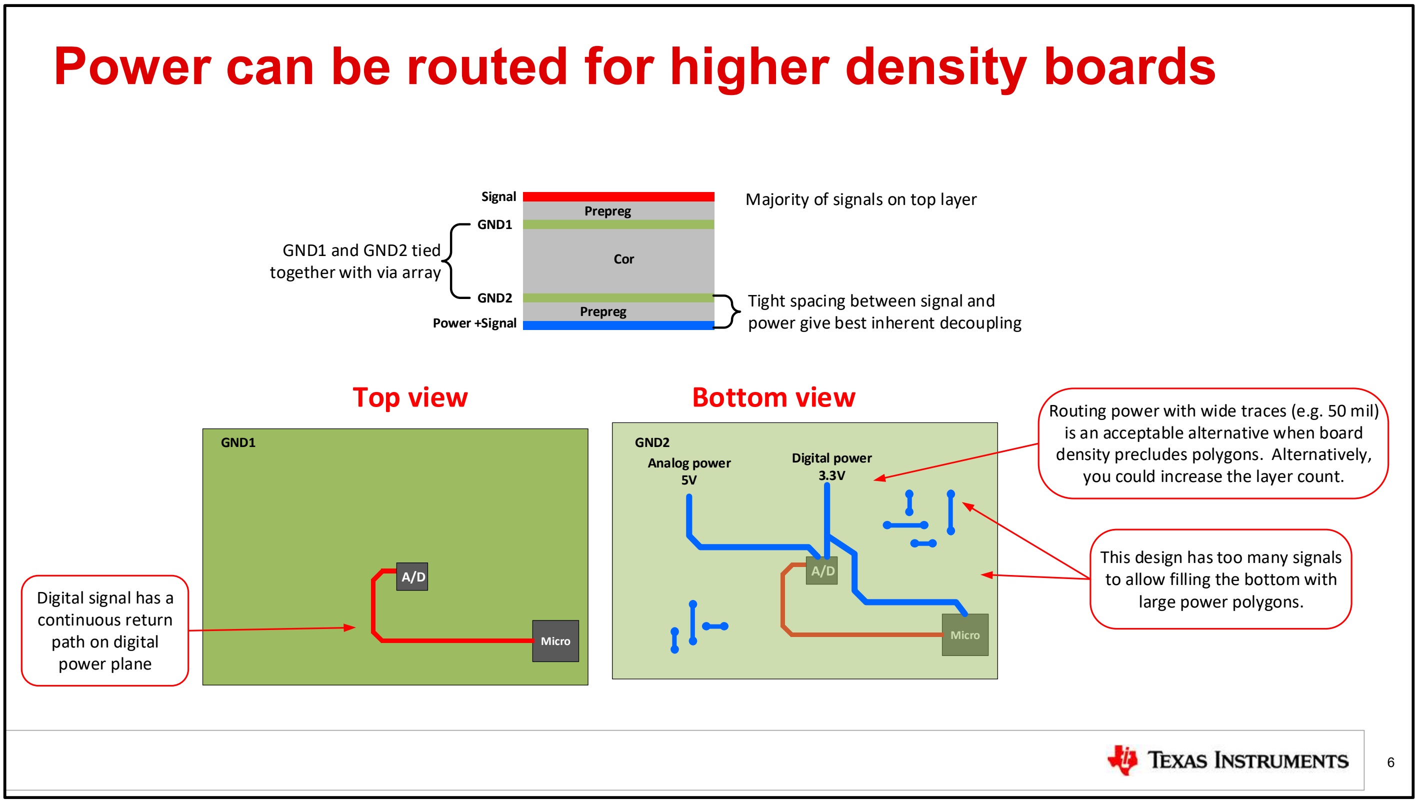
Here we compare different stackups to see which is best. The worst stackups will use two adjacent layers with signals on each layer. In this case the two signals can interfere with each other and the ground reference is typical far from the signals making the return currents spread out. A common approach used on four layer boards is to use power and ground as internal layers, and have the signals on the outer layers. The power rail at the bottom will act as the return path for signals on the bottom. One possible issue with this approach is that the power plane may be split into multiple power rails. In this case, there will be discontinuities or slots in the return path and stitching capacitors will be required to bridge the gap. Also when transitioning from the top layer to the bottom layer the return path will need to transition from ground to power, and this can also cause RF emissions. The best approach for EMI on a four layer board is to use two internal ground planes. In this case, signals would be assigned to the top layer, and power and signals would be on the bottom layer. This method has the advantage of keeping the power rails very close to the ground return path, which will improve decoupling. The method also eliminates the need for stitching capacitors as the ground return is solid where as the power plane in the previous example was split to accommodate multiple power rails. Also, when signals on the top an bottom transition layers the two ground layers are connected together with stitching via so emissions are minimal. The six layer and higher layer count printed circuit boards follow the same principles. In short, you never want signal layers without adjacent ground return paths. Also, keeping power close to a ground layer will improve decoupling.



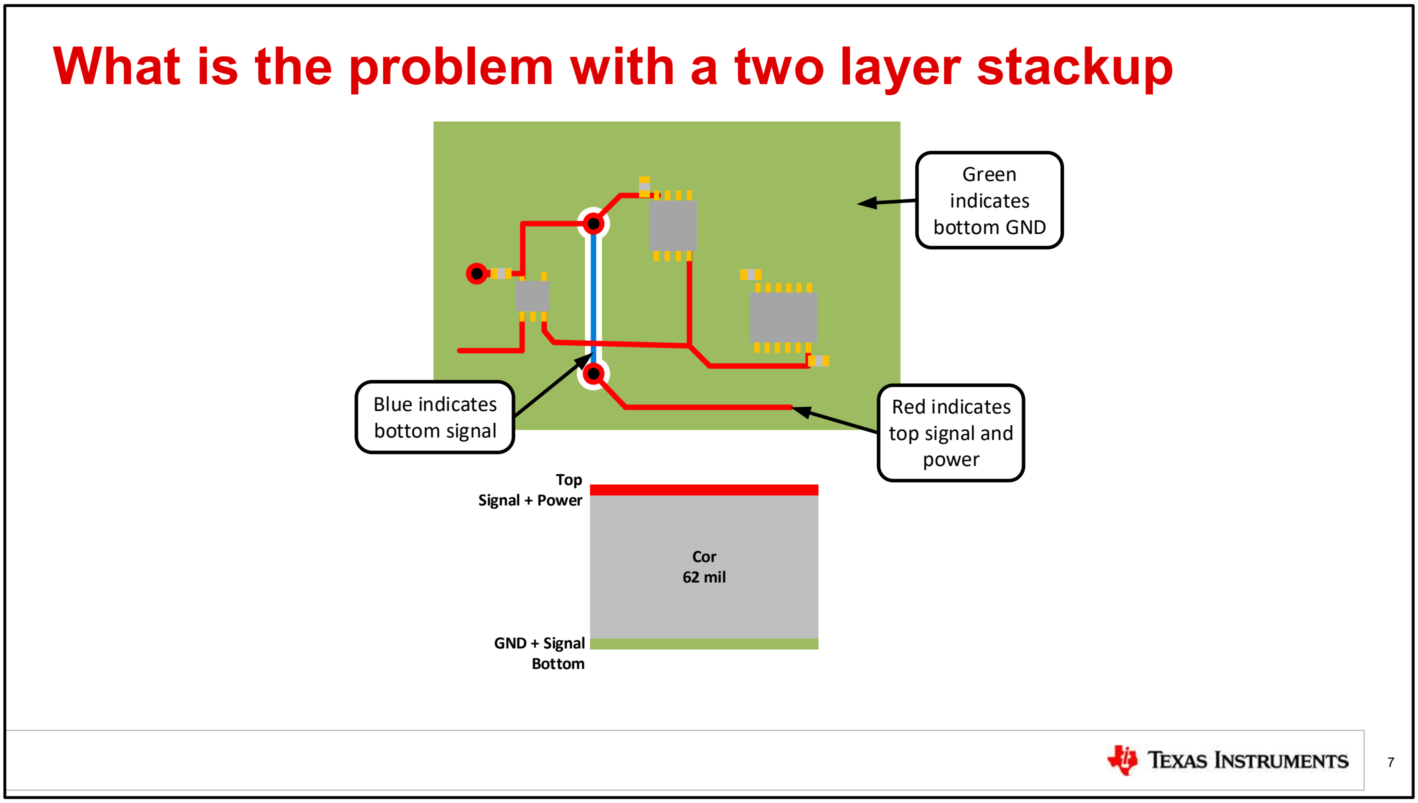
This example shows how using a power layer adjacent to a signal layer can cause problems. This method can work but you need to be careful to avoid routing signals above splits in the power plane. In this example power is split into a 3.3V digital plane and a 5V analog plane. The design on the left can create RF emissions because the digitals signal is routed over splits in the return path. On the right the layout was adjusted to keep a continuous digital power plane beneath the digital I/O line. For this design the return current will flow through the digital 3.3V power plane, not the ground plane.



This slide shows how two internal ground planes can be used in a four layer board. In this case most of the signals will be routed on the top layer and the power planes are on the bottom layer. Some signal traces can be routed on the bottom but the majority of the bottom layer is dedicated to power. This stackup has the advantage of keeping the power plane close to its ground return which will improve decoupling as the AC impedance between power and ground is low. Also, signals transitioning from top to bottom layers will not require stitching capacitors.

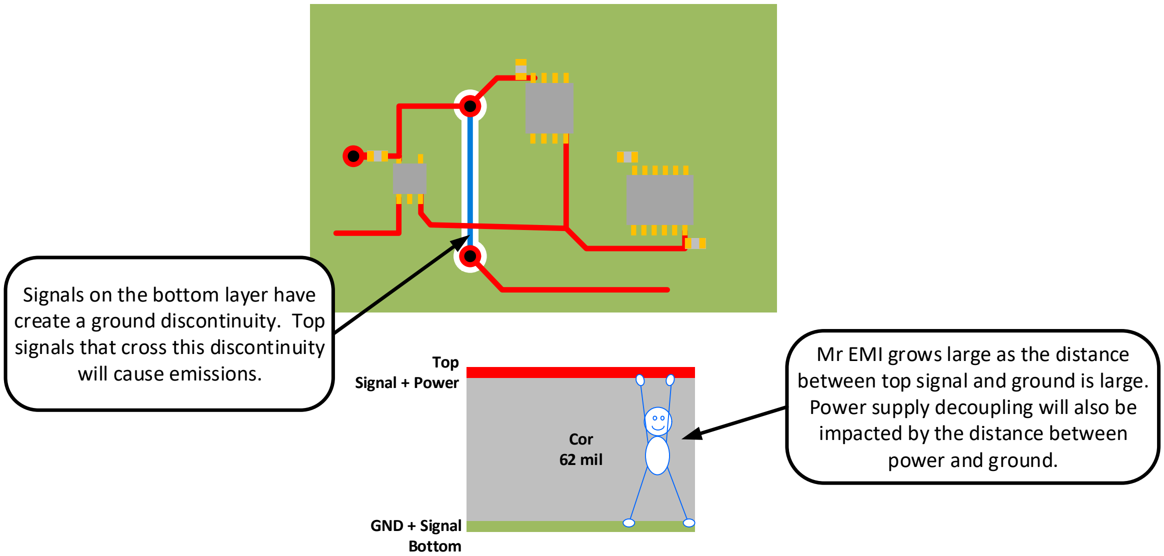


The example from the last slide can be modified so that power is routed as thick traces rather than planes. If the transient currents on the power rails is not significant routing power is a reasonable approach. The advantage of this approach over the power planes is that more space is available for signal routing on the bottom layer.



Here is an example of a two layer board. For this type of board the best stackup is to put power and signal on the top layer and use the bottom layer as a solid ground plane. However, it can often be difficult to route the board with all the signals and power on the top, so frequently some traces will be routed on the bottom. What are some common issues associated with this kind of stackup?

What is the problem with a two layer stackup



Texas Instruments 8

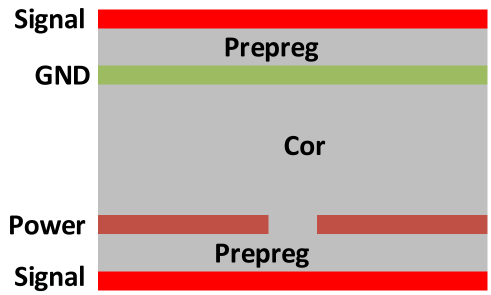
One significant issue with two layer boards occurs because some traces need to be routed on the bottom layer. The trace on the bottom causes a discontinuity in the ground path. When signals on the top layer pass over this discontinuity, the ground return currents area forced to rout around the split and RF emissions happen. Another issue occurs because the top and bottom layer are physically separated by a significant distance. A typical thickness for a two layer boards is 62 mil. This thickness is substantially greater than the distance between the top layer and core on a four layer board. Increasing the separation between layers will cause the fields beneath the trace to spread out. Remember Mr. EMI. He will conform to the top and bottom of the wave guide. Spreading out of the fields will increase crosstalk between digital and sensitive analog traces. Of course, if the two layer board is made to be thin this problem is minimized, but this may not be practical from a mechanical strength perspective.

Thanks for your time! Please try the quiz.

That concludes this video – thank you for watching! Please try the quiz to check your understanding of this video's content.

Quiz: Introduction PCB Design for Good EMC

1. What are some potential problems with the stackup below?
   a) For proper shielding the ground should always be on the bottom layer.
   b) The power layer should never have a split.
   c) The power and ground plane are separated by a wide distance. This degrades the high speed decoupling.
   d) The power plane is split, so signals on the bottom layer that cross the split can have RF emissions issues.
   e) Answers a, and b are problems.
   f) Answers c, and d are problems.



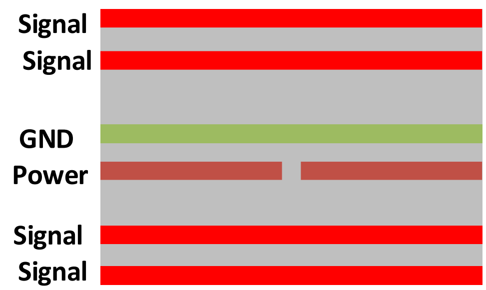
Question 1, What are some potential problems with the stackup below?

The correct answer is “f”, Answers c, and d are problems. Keeping the ground and power close together reduces the high frequency impedance between the planes. This low impedance will short out power supply noise. In this case the power and ground are separated by a large distance, so the high speed decoupling will be degraded.

Also, the power plane is split. This split can cause issues if signals are driven across the split.

Quiz: Introduction PCB Design for Good EMC

2. What are some potential problems with the stackup below?
   a) The two adjacent signals do not have an adjacent ground return, so they will potentially interfere with each other.
   b) Ground and power are too close, they should be separated further to prevent power supply noise from coupling into ground.
   c) None of the above, this is a good stackup.



Question 1, What are some potential problems with the stackup below?

The correct answer is “a”, The two adjacent signals do not have an adjacent ground return, so they will potentially interfere with each other. In this case the signals on the top two layers will reference the ground plane and all the return currents will mix causing crosstalk. On the bottom two layers the signals are referenced to the power plane, and again, the return currents will mix. Also, the bottom layer has a split and this could also cause issues if signals cross the split.

Thanks for your time!

TEXAS INSTRUMENTS 12

That's all for todays video. Thanks for watching.

© Copyright 2022 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