Introduction
PCB Design for Good EMC

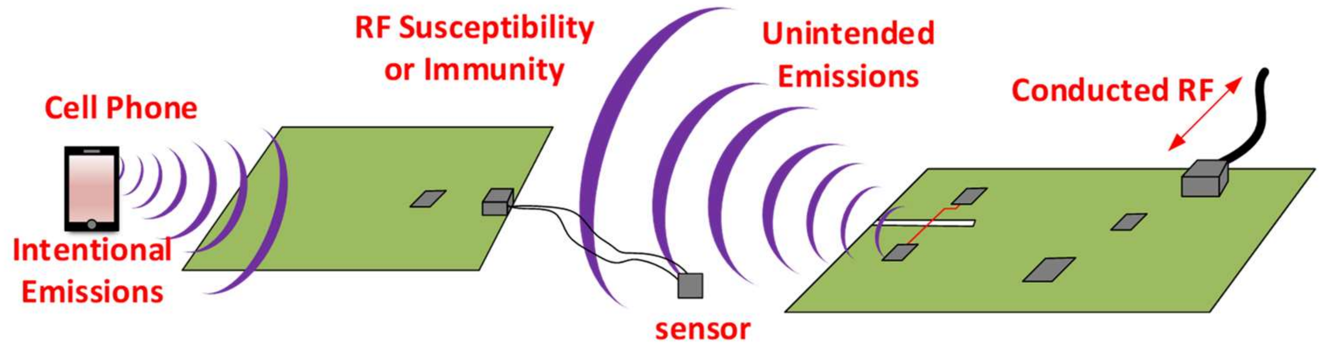
TI Precision Labs – ADCs

Created and Presented by Art Kay

Texas Instruments

Hello, and welcome to the precision labs series on designing Electromagnetic Compatible PC board layouts. Electromagnetic Compatibility, abbreviated EMC, is the ability of your board or system to operate properly in the presence of RF interference as well as assuring that your system does not interfere with other systems. Initially, the series will cover a simple qualitative view of PCB design methods that minimize your design's RF emissions and susceptibility to interference. Later, specific EMC test standards and methods will be discussed. 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. The series will focus on four layer boards although the methods could expanded to other layer counts. This introductory video will cover some of the basic definitions and equations related to EMC design. The video series will also provide measured test results to support the EMC theory. This video will cover the types of experiments, test tools, and test methods used throughout the series. The initial content in this series is fairly basic, but later more advanced concepts will be covered. So, for the more experienced engineers I hope you stay with us for the more advanced content. Without further delay, let's get started!

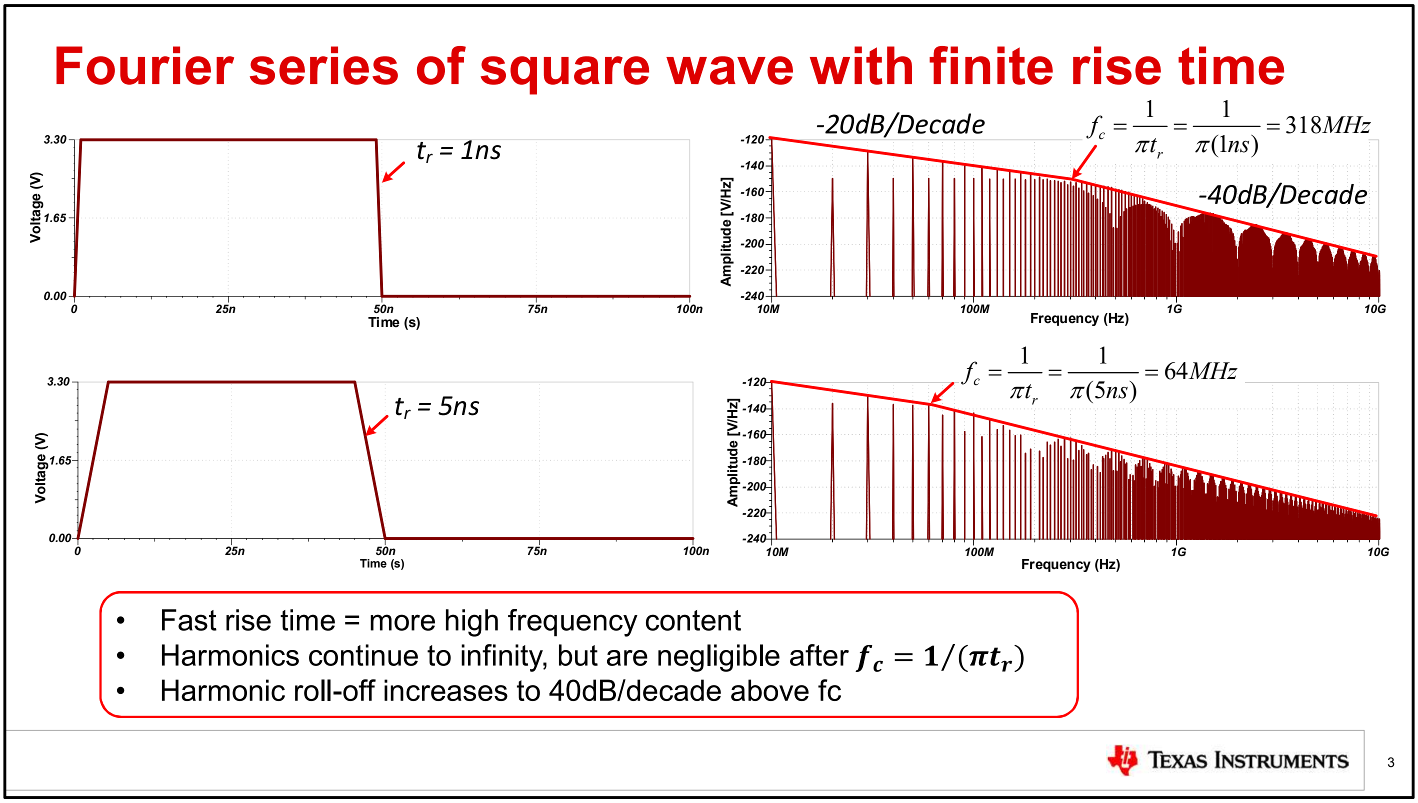
Definitions



- **RF Radiated Emissions** – Electromagnetic wave leaving a device to propagate through the air.
- **Conducted Emissions / Immunity** – RF signals propagate through cables rather than the air.
- **RF Susceptibility** – How sensitive is the device to the RF noise.
- **Intentional Emissions** – Cell Phone, **Unintended** – switching device or PCB layout issue

Texas Instruments 2

Here are some basic definitions for electromagnetic compliance. In some cases, a circuit that is designed to operate at a low frequency can omit high frequency RF signals. Frequently, this can happen because of poor printed circuit board design techniques. For example driving a 50 MHz digital communication signal across a gap in the ground plane can cause RF emissions. If these emissions leave the printed circuit board and travel through the air they are considered to be radiated emissions. On the other hand, if the emissions travel through a cable, they would consider be considered conducted emissions. RF emissions can be picked up by other circuits and impact the circuit performance. A circuit subjected to the RF emissions may be sensitive or insensitive to this issue. For example, a long PC board trace or cable can act as a good antenna and will be especially susceptible to our a pick up. The degree to which a circuit is sensitive to RF pick up is called RF susceptibility or immunity of the circuit. Finally it should be noted that some circuits will intentionally radiate, whereas some radiation is unintended. For example, a cell phone is designed to transmit RF signals, so it is an intentional radiator. On the other hand, the digital communication we mentioned previously was not intended to radiate RF noise. One of the main goals of this content is to provide methods to minimize the unintended emissions, and improve the immunity of circuits to RF noise.



This slide shows how a 10MHz digital square wave can have frequency content to gigahertz. Technically, the waveform we are considering here is a trapezoid wave as the waveform has finite rise and fall times. The Fourier theorem states that any repetitive signal can be represented as an infinite series of sinusoidal waves called a Fourier series. The Fourier series for a trapezoidal waveform is shown at the right. The frequency of the square wave is 10MHz. This means that the Fourier series will have a fundamental component at 10MHz then will have frequency content to theoretically infinite frequency. Fortunately, the harmonic content decreases rapidly so that from a practical perspective the harmonics are small at some high frequency. Analyzing the Fourier series shows that the roll-off of the harmonics is 20dB/decade for frequency just above the fundamental, then increase to 40dB/decade at higher frequencies. The point where the attenuation changes from 20 to 40db/decade is considered to be the RF emissions bandwidth. After the bandwidth limit the harmonics are considered to be insignificant from an emissions perspective. The bandwidth is calculated by taking one over pi times the rise time. Thus the rise time of the square wave is the key factor that determines how far in frequency the harmonics extend. Both square waves shown here have the same fundamental frequency but have different rise times. The first example has a fast 1ns rise time and the bandwidth limit is 318MHz. The second has a slower rise time of 5ns and its bandwidth limit is 64MHz. Later we will see measured results that show RF emissions for different rise times.

Wavelength and velocity calculations

$$\nu = \frac{c}{\sqrt{\varepsilon_r}} = \frac{300 \cdot 10^6 \ \frac{m}{s}}{\sqrt{4.5}} = \left(1.414 \cdot 10^8\right) \frac{m}{s}$$

$$\lambda = \frac{\nu}{f} = \frac{\left(1.414 \cdot 10^8\right) \frac{m}{s}}{10\ GHz} = 14\ mm \qquad (551\ mil)$$

$$\frac{\lambda}{10} = \frac{14\ mm}{10} = 1.5\ mm \qquad (55.1\ mil)$$

Convert frequency to wavelength for fr4

C – speed of light in vacuum 300*10^6 m/s

ν – velocity in medium

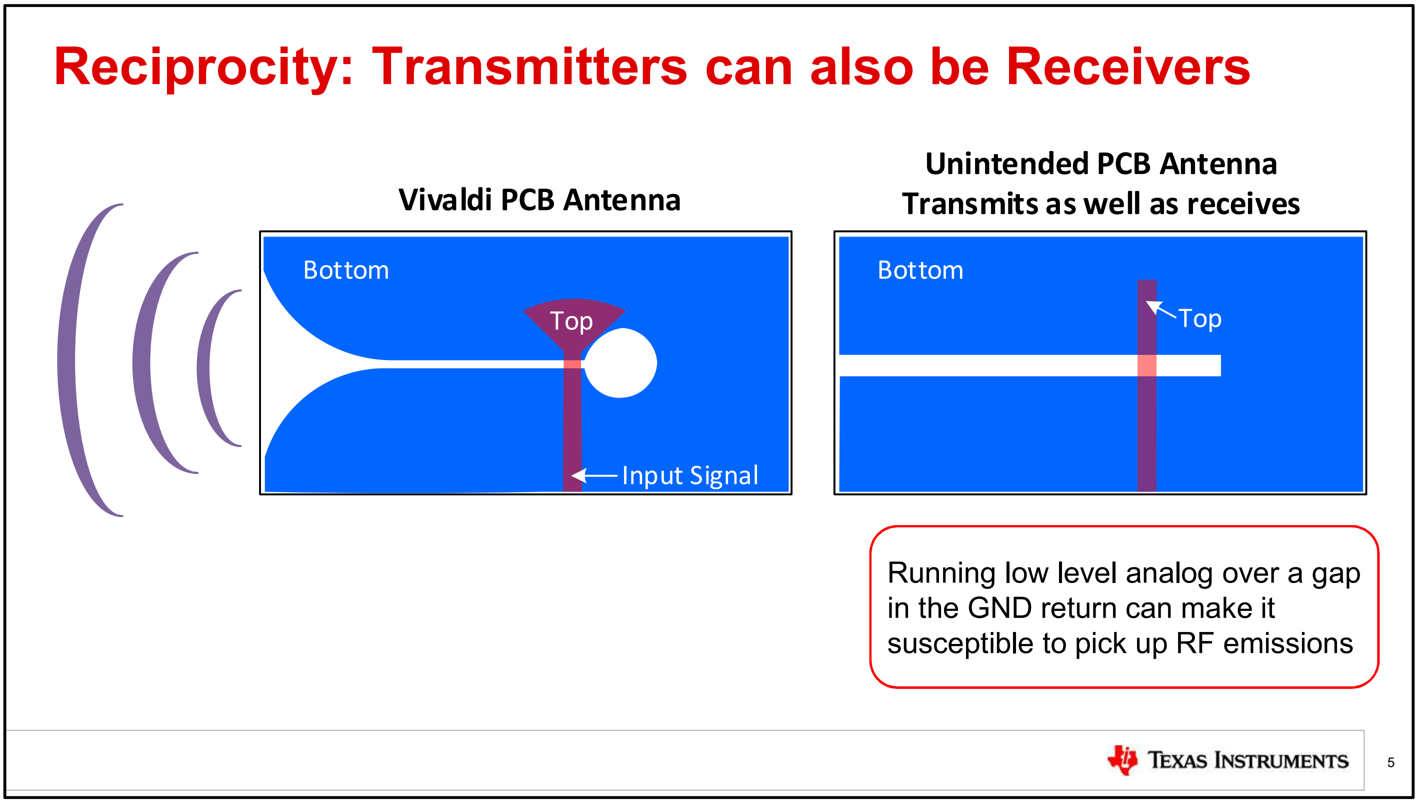
ε_r – relative permeability 4.5 for FR4

λ – wavelength in meters

$\frac{\lambda}{10}$ is short from an RF perspective

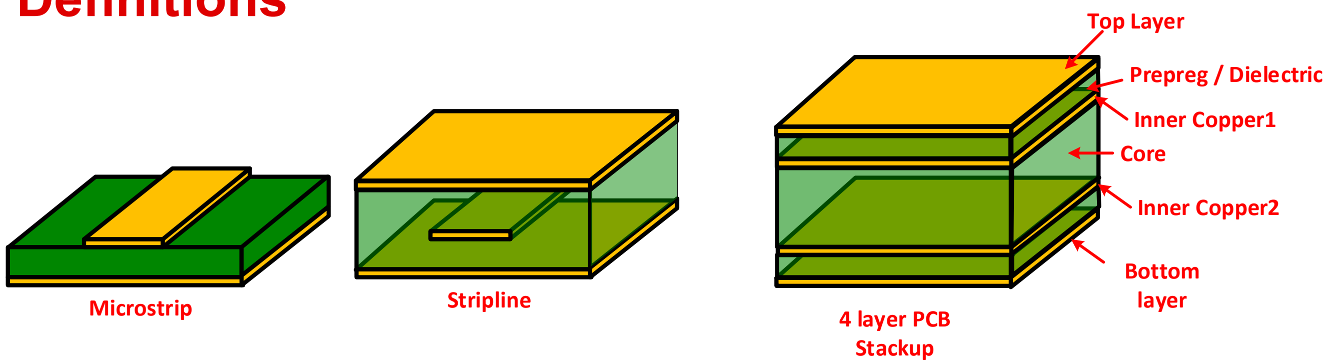
$\frac{\lambda}{4}$ multiples makes a good antenna

Here we present equations for velocity and wave length for electromagnetic waves for printed circuit boards. The velocity is determined by dividing the speed of light by the square root of the relative permeability of the PC board dielectric. FR4 is the most common type of dielectric, and it has a relative permittivity of approximately 4.5. This number is not well controlled and can vary significantly. Higher frequency boards often use better controlled dielectric materials such as Johnson. In any case applying the equation using FR4 dielectric, the velocity is 1.414×10^8 m/s. The wavelength of the electromagnetic signal is determined by dividing the velocity by the applied frequency. So for a 10 GHz signal, the wave length is 14 mm. The wave length is important for PC board design as it is an indication of how good a trace or cable acts as an antenna. Traces that are multiples of a quarter wave length make good antennas. Traces that are shorter than a 10th of a wave length are considered short from an RF perspective. We will refer back to these equations throughout this presentation.



One of the key concerns is that we do not want to create unintended emissions with our PCB design. Unintended emissions can happen when long PCB traces act like antenna. Also, it is important to have a continuous ground plane beneath your PCB trace. If there is a slot in the ground plane as shown on the right, this can cause emissions. We will discuss this in detail later, but for now I want you to know that any structure that works well as an antenna for emissions will also work well as a receiver. The principle that antenna will work equally well as a transmitter or receiver is called reciprocity. Keep in mind that when we cover PCB design problems that cause emissions, the same design may also create noise immunity issues. In the example above, driving a digital signal across a split in the ground plane will cause emissions. Placing a low level analog signal above a split isn't likely to cause emissions issues, but through reciprocity it will act as a good receiver and will pick up RF noise. Finally, if you wanted to create an intentional transmitter such as a Bluetooth device, one way to create an antenna is to drive a signal across a split in the GND plane. The picture on the left is a Vivaldi PCB antenna where a signal is driven on the top layer across a split in the GND plane. Note how similar this structure is to the unintended antenna. We will return to the example of driving signals across gaps in the GND return path as it is one of the most common EMC design errors.

Definitions



- **Prepreg / Dielectric** – The insulative fiberglass material used between the core and top or bottom foil (or between multiple cores). FR4 is a common dielectric
- **Core** – rigid base material laminated with copper on one or two sides
- **Stackup** – the order and number of layers in the PCB. (e.g. 4 layer stackup: Top = signal, Inner1 = GND, Inner 2 = Power, and Bottom = signal2).
- **Microstrip** – a signal trace above a ground or power plane
- **Stripline** – a signal trace sandwiched between two planes

Here we introduce some basic PCB layout definitions. The figure on the left is a four layer PCB with a top and bottom layer as well as two internal layers. Generally the internal layers are reserved for ground and power planes and the outer layers are used for signal traces. The functional order and number of layers on a PCB is called a stackup. So, for example a four layer board may have a stackup of: top signal, inner1 ground, inner2 power, and bottom signal. The layers in a PCB are separated by a fiber glass insulative material called dielectric. The most common dielectric material use is FR4. Higher frequency designs may require more expensive dielectrics such as Johnson, but we will only cover FR4. A PCB core is a rigid dielectric material with two copper layers on it. A four layer board is built by adding a thin sheets of dielectric material called prepreg on the top an bottom of the board followed by a top and bottom layer of copper. An important point here is that the core thickness is generally large compared to the prepreg thickness. Thus, the top and bottom layer are very close to the inner layers. This is important from an EMC perspective.

Beyond understanding basic PCB construction, it is important to know a few common PCB geometries. The microstrip is a single PCB trace above a copper plane, and the stripline is a PCB trace placed between two return planes. The length and width of the trace as well as the dielectric thickness will impact various PCB characteristics such as impedance. We will refer back to the microstrip and stripline configuration through this material.

Mixed signal examples

Rapid transients generate the most noise and RF emissions

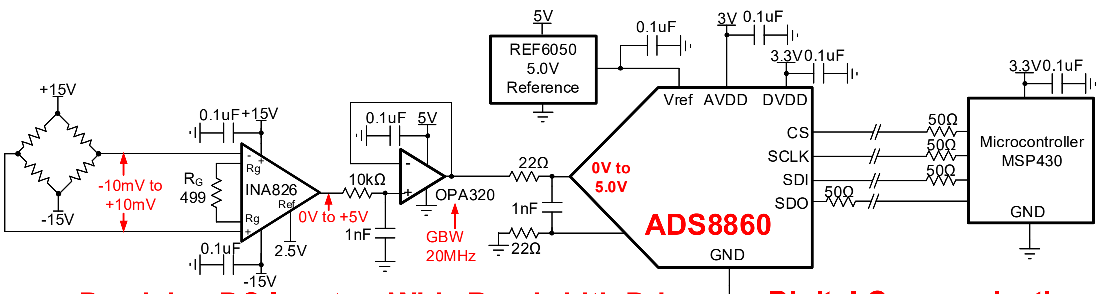
- Digital and switched transients
- ADC input and Reference input have switching transients

For low noise consider:

- Continuous GND return for fast signals
- Decoupling for high speed devices
- Minimize overshoot and reflections on digital signals
- Digital overshoot can cause data integrity issues as well as noise

Low level analog is most sensitive

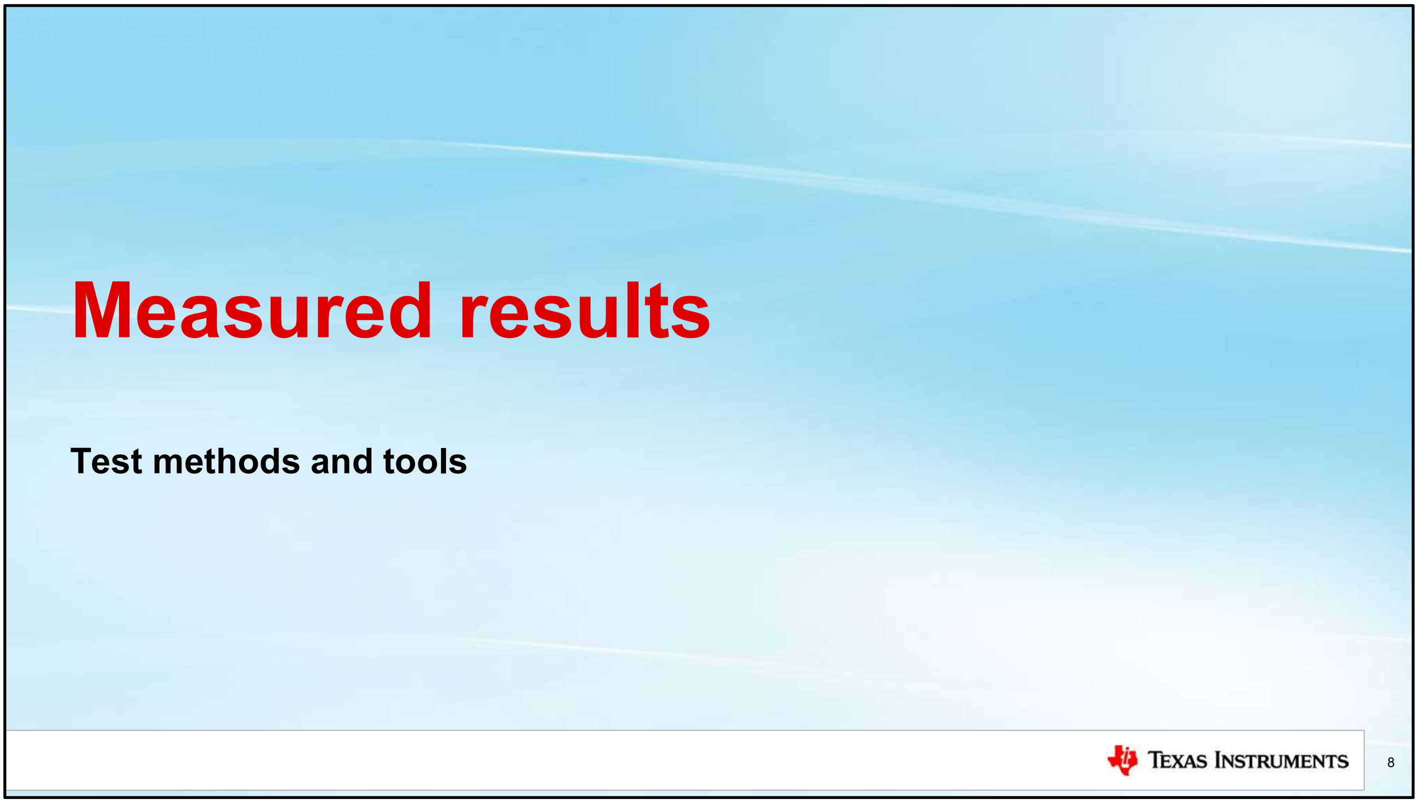
- Make traces short as possible (long traces act as antenna)
- Keep traces as far away from transients as possible
- Keep differential lines symmetrical (CMRR)
- EMIRR and filtering



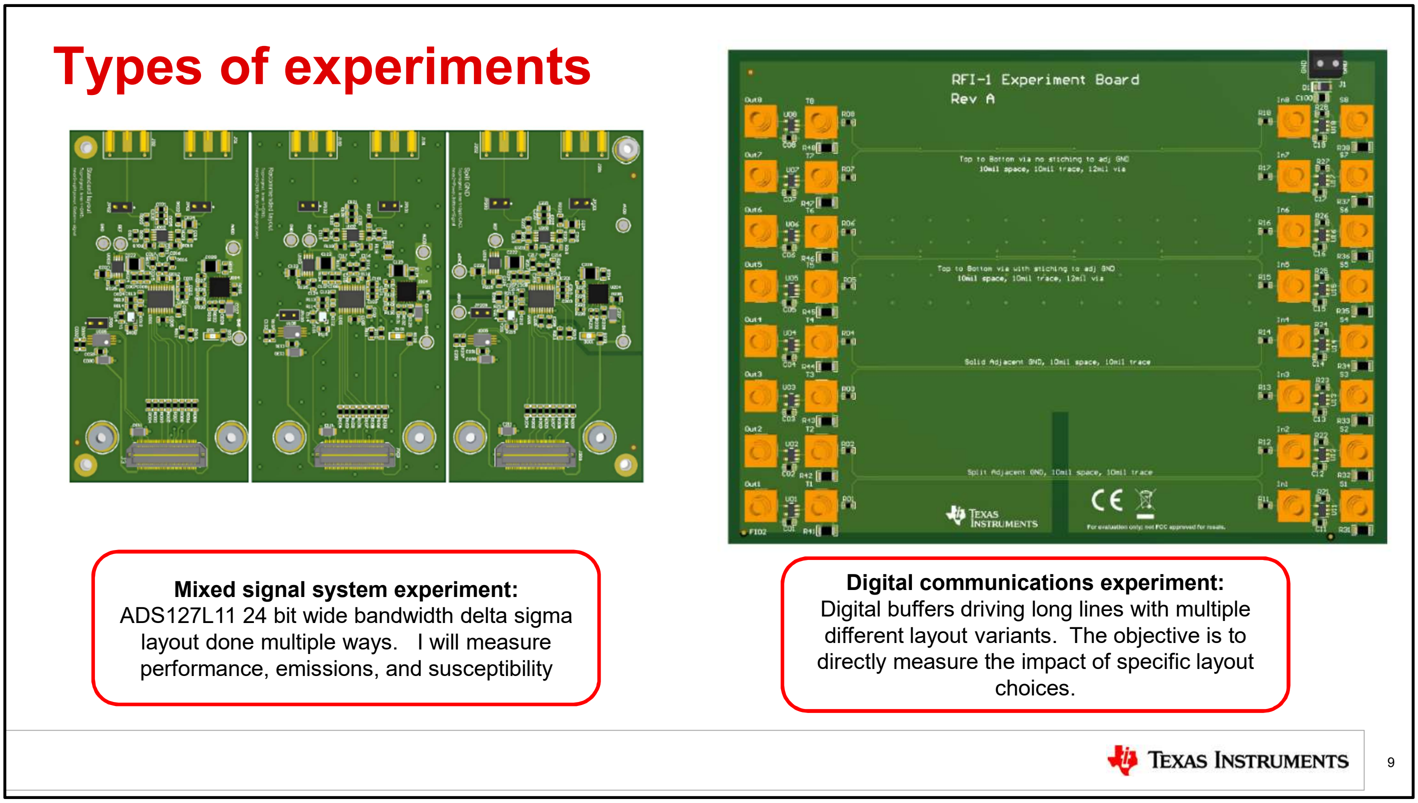
Precision DC Input **Wide Bandwidth Driver** **Digital Communications**

TEXAS INSTRUMENTS 7

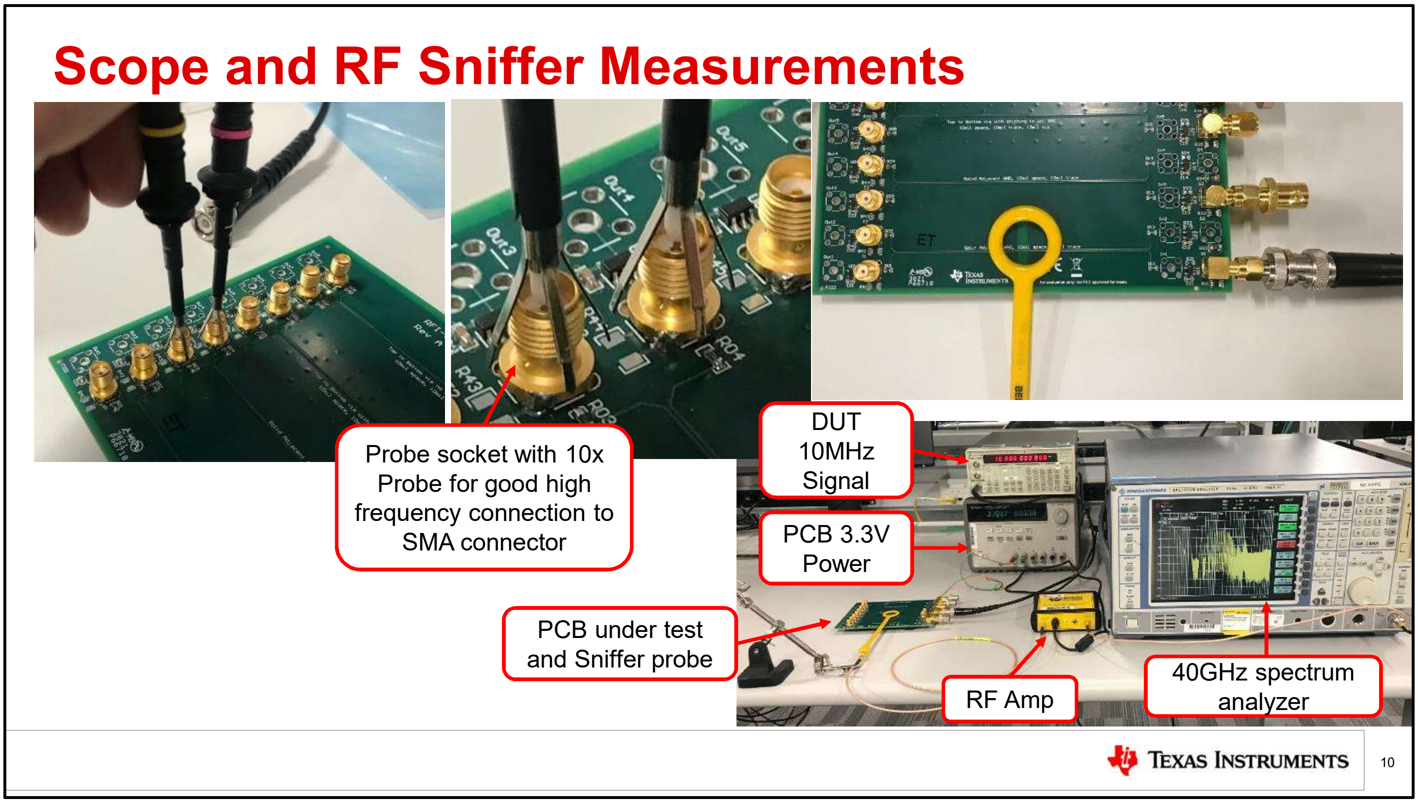
This slide shows a typical mixed signal system. A Low level analog signal from a bridge sensor is applied to an instrumentation amplifier which is followed by a high speed op amp. The op amp drives an analog to digital converter which is connected to a precision voltage reference and micro controller. In this system look for rapid transients as the most likely noise sources for RF emissions. The digital I/O signals, the reference input, and analog to digital converter input all have switching transients associated with them. There are many techniques that can be used to minimize noise. For example, taking care to make sure that the ground return path is continuous is very important. Proper layout and selection of decoupling capacitors is also useful for minimizing noise. From a digital perspective minimizing overshoot and reflections reduces noise and cross talk. The sensitive low-level analog may not be a source of noise, but it can be easily corrupted by noise causing large measurement errors. Minimizing trace length for low level signals and keeping them away from transients can help prevent noise coupling. Using differential lines and using proper filtering can also reduce the impact of noise on analog devices. Throughout today’s presentation all these concepts will be discussed in detail with emphasis on proper PC board layout.



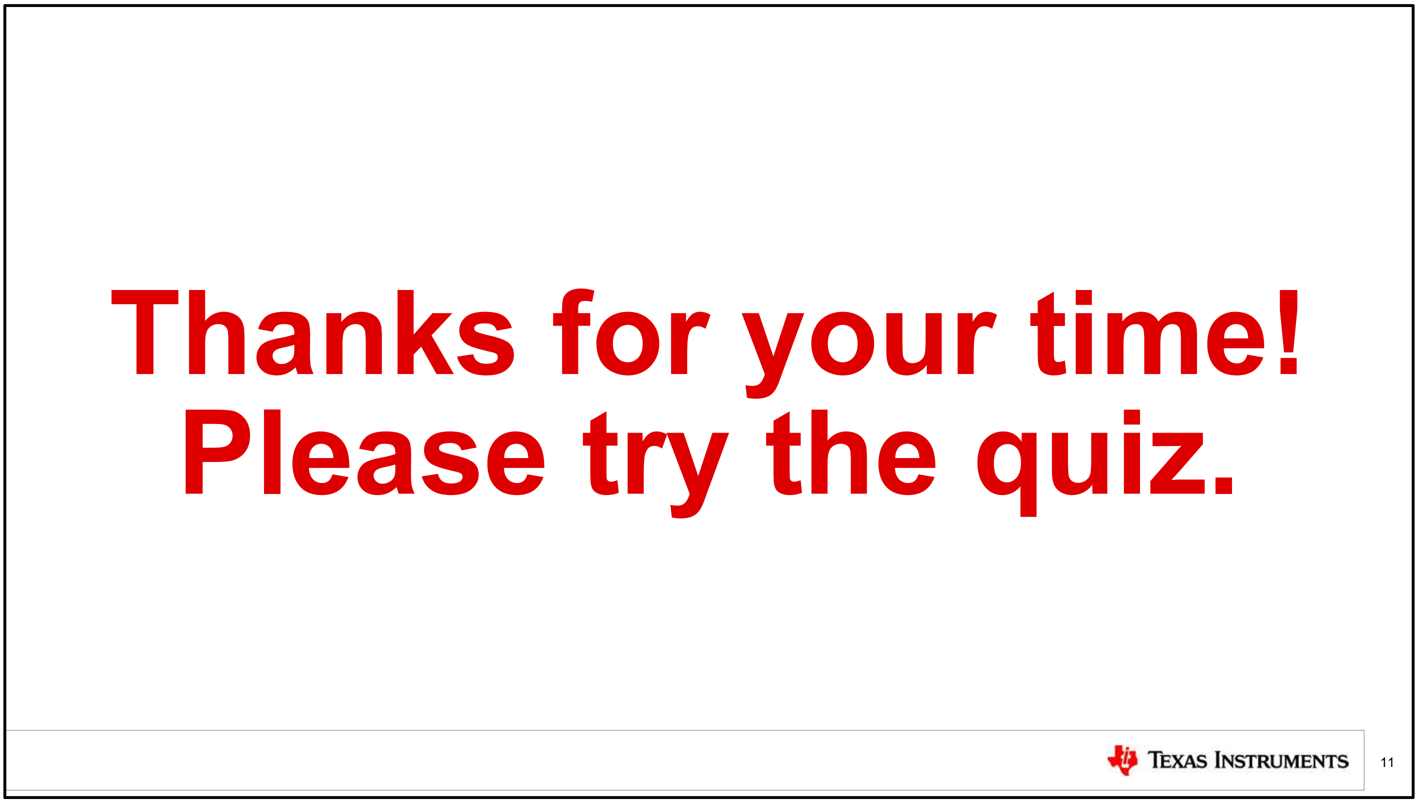
I built and measured some PCB experiments to support the theory in todays presentation. The experimental results will be discussed throughout the presentation. Here I will do a brief introduction to show the hardware and discuss the measurement methods.



Two different categories of experiments were developed for this content. One type of experiment uses a full analog signal chain similar to what was previously introduced. This circuit is duplicated multiple times with the same schematic, but different layouts. The idea here is to compare good versus bad layout methods to show how the analog performance, RF emissions, and RF subsebtebility are impacted. The second category of experiments are looking at digital communications. Typically a long trace is driven by digital signals and the performance impact of trace spacing, grounding, and via placement on crosstalk and emissions is considered.



This slide shows the real world test equipment used for the experimental results in this presentation. To provide a good high frequency oscilloscope connection a probe socket configuration is use to connect to an SMA connector. To measure RF emissions EMI sniffer probes are connected to an RF amplifier and a wide bandwidth spectrum analyzer. The sniffer probes measure the near field RF emissions on the PCB. Sniffer probes are good for localizing a source of RF emissions on a PCB. For product EMI compliance testing far field RF emissions will be measured using an antenna in an anechoic chamber. We will not cover this today, but the reference section provides a link to an excellent application note covering many of the different compliance tests.



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. Which would have higher frequency RF emissions: a 10MHz square wave with a 1ns rise time, or a 50MHz square wave with a 5ns rise time.
   a) 10MHz square wave with a 1ns rise time
   b) 50MHz square wave with a 5ns rise time

2. What kind of EMC problem would you expect with a low level DC analog signal.
   a) It may cause an unintended transmitter
   b) It may act as an unintended receiver

3. (T/F) Reciprocity states that a structure that works well as an antenna for transmitting will work equally well as a receiving antenna.
   a) True
   b) False

Texas Instruments 12

Question 1, Which would have higher frequency RF emissions: a 10MHz square wave with a 1ns rise time, or a 50MHz square wave with a 5ns rise time.

The correct answer is “a”, 10MHz square wave with a 1ns rise time. Remember that the square wave will have frequency content out to high frequency. The rise time of the square wave will determine how wide in frequency this signal will spread out. The equation 1/(pi*tr) can be used to estimate the upper frequency limit of the square wave.

Question 2, What kind of EMC problem would you expect with a low level DC analog signal.

The correct answer is “b”, It may act as an unintended receiver

Question 3, true or false, Reciprocity states that a structure that works well as an antenna for transmitting will work equally well as a receiving antenna.

The correct answer is “a”, true. For this reason low level signals are subjectable to RF pickup and higher level AC signals may act as unintended transmitters on PCB structures that act as unintended antenna.

Quiz: Introduction PCB Design for Good EMC

4. What is the wavelength for a 100MHz frequency on an FR4 PCB?
   a) 12mm
   b) 251mm
   c) 1414mm
   d) 2522mm

5. What is the minimum PCB trace length that would make a good antenna for a 100MHz frequency on FR4 PCB.
   a) 61mm
   b) 354mm
   c) 692mm
   d) 821mm

Texas Instruments 13

Question 4, What is the wavelength for a 100MHz frequency on an FR4 PCB?

The correct answer is "c", 1414mm. To make this calculation first calculate the velocity by dividing 300×10^6 m/s by the square root of the dielectric coefficient of the PCB. In this case the PCB is FR4 and it has a dielectric coefficient of 4.5. The velocity in this case is 1.414×10^8 m/s. The velocity is then divided by the frequency of the waveform to find the wavelength which is 1414mm in this case.

Question 5, What is the minimum PCB trace length that would make a good antenna for a 100MHz frequency on FR4 PCB.

The correct answer is "b", 354mm. This question is related to the previous question. In the previous question we determined that the wavelength for a 100MHz signal is 1414mm. A good antenna is multiples of a quarter wavelength. Dividing 1414mm by 4 gives 354mm.

Thanks for your time!

See references in video PDF

TEXAS INSTRUMENTS 14

That’s all for todays video. References and acknowledgments for this video series are provided in the PDF of this video. Thanks for watching.

Acknowledgments

Thanks to many TI engineers advice and technical support:

Bob Benjamin, Keith Nicholas, Dale Li, Collin Wells, Thomas Kuehl, and many others

Thanks to the EMC experts for their great literature:

Henry Ott, Eric Bogatin, Rick Hartley, Robert Feranec

Here I'm acknowledging the many people who helped review and develop this material. Also thanks to the Great EMC experts for their excellent videos articles and books.

References: Books

Ott, Henry W. (2009) *Electromagnetic Compatibility Engineering*, John Whiley and Sons

Bogatin, Eric (2010) *Signal and Power Integrity*, Pearson Education, Inc.

Montrose, Mark I. (2000) *Printed Circuit Board Design Techniques for EMC Compliance*, John Whiley and Sons

References: TI-App Notes

Best Practices for Board Layout of Motor Drivers

Excellent reference on good PCB techniques

https://www.ti.com/lit/an/slva959b/slva959b.pdf

IEC 61000-4-x and CISPR 11 Testing With ADS8686S for Industrial Application

Excellent reference for EMI Compliance Testing for Precision ADC Systems

https://www.ti.com/lit/an/sbaa548/sbaa548.pdf

References: Videos

You must Unlearn what You have Learned, Dr Eric Bogatin

Myth of 3 decoupling capacitors, and is GND pour a good idea?

https://www.youtube.com/watch?v=y4REmZlE7Jg

How to Achieve Proper Grounding - Rick Hartley – Altium Live 2019

PCB Stackup impact on performance, return current flow, split vs solid GND, via impedance and transitioning layers

https://www.youtube.com/watch?v=ySuUZEjARPY

Watch How a PCB Layout Change Makes Big Difference, Dr Eric Bogatin & Robert Feranec

Impact of different return paths on crosstalk

https://www.youtube.com/watch?v=Fj9M2CK2cX0

References: Videos

Understanding Crosstalk in PCB Layout, Eric Bogatin & Robert Feranec

Trace impedance: reflections, matching, and crosstalk

https://www.youtube.com/watch?v=5EeQPxRdurk&list=PLXvLToQzgzdeG8r9IEuAq6ft6r4f_1Ub_&index=28

Ground in PCB Layout - Separate or Not Separate? - Rick Hartley & Robert Feranec

Impact of split plane on EMI, GND loop for multiple PCB in system

https://www.youtube.com/watch?v=vALt6Sd9vlY&list=PLXvLToQzgzdeG8r9IEuAq6ft6r4f_1Ub_&index=6

Every PCB Designer Needs To Know About PCB Track Impedance, Dr Eric Bogatin & Robert Feranec

Reflections vs trace geometry, demo of impedance mismatches, simulation of trace impedance and reflections

https://www.youtube.com/watch?v=Fj9M2CK2cX0

References: Videos

3 Simple Tips To Improve Signals on Your PCB, Robert Feranec

Dielectric thickness, balancing the trace branches, adding termination resistor

https://www.youtube.com/watch?v=CDJn-35W8sg

9 Simple Tricks to Improve EMC / EMI on Your Boards - Min Zhang & Robert Feranec

EMC testing: low cost in lab setups, use of ferrites, cable length, capacitor placement

https://www.youtube.com/watch?v=Lf51sx6sC0I&list=PLXvLToQzgzdeG8r9IEuAq6ft6r4f_1Ub_&index=5

What is The Best VIA Placement for Decoupling Capacitors?, Robert Feranec

Simulation used to check via placement on capacitor and power supply connection, top or bottom connection, many capacitors in parallel vs few capacitors, trace length connecting capacitor, polygon connections

https://www.youtube.com/watch?v=Fj9M2CK2cX0

References: Videos

Breaking Bad Habits in PCB Design, Eric Bogatin Altium Live

Trace thickness, return path placement, reduce congestion, proper usage of via, decoupling capacitor placement. Proper scope probe usage for high frequency signals, decoupling resonance

https://www.youtube.com/watch?v=CDJn-35W8sg

Basics of Near Field RF Probes | *E-Field & H-Field* | *How-to use* - w2aew Channel

How to use "sniffer" probes. How near field relates to far field. E-field vs H-field.

https://www.youtube.com/watch?v=ctynv2klT6Q

Keys to Control Noise, Interference and EMI in PC Boards, Rick Hartley – Altium Live 2019

Frequency vs EMI, transmission lines, types of EMI, power distribution, stackup, I/O design for EMI

https://www.youtube.com/watch?v=ZYUYOXmo9UU

References: Videos

PCB Layout & Decoupling - Explained why it's so complicated (Part 1), Robert Feranec

Resonance in decoupling network, ripple in supply from digital switching

https://www.youtube.com/watch?v=5Ca0Eah7eKI

Part 2: PCB Layout & Decoupling - Understanding Impedance, Robert Feranec

How to use “sniffer” probes. How near field relates to far field. E-field vs H-field.

https://youtu.be/Tt8X6_maj6c

Part 3: PCB Layout & Decoupling - Measuring and Fixing , Robert Feranec

Frequency vs EMI, transmission lines, types of EMI, power distribution, stackup, I/O design for EMI

https://www.youtube.com/watch?v=ZYUYOXmo9UU

References

Advanced Circuits, typical PCB dimensions

Prepreg-thickness-chart

General pcb-design-specifications

Example 62 mil pcb-stack-ups

Logic guide: covers the differences between logic families.

http://www.ti.com/lit/pdf/sdyu001

Implications of Slow or Floating CMOS Inputs

http://www.ti.com/lit/pdf/SCBA004

Capacitor Characteristics (ESR, ESL, for MLCC capacitor)

https://ds.murata.co.jp/simsurfing/mlcc.html?lcid=en-us