PCB Trace Impedance matching

TI Precision Labs – ADCs

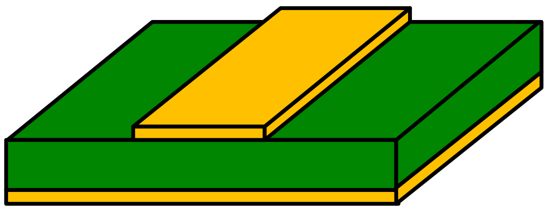
Created and Presented by Art Kay

TEXAS INSTRUMENTS

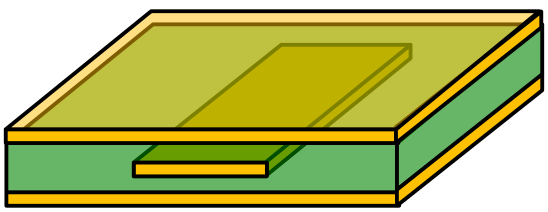
Hello, and welcome to the TI precision labs video on PCB trace impedance matching. This is part of a larger series on PCB layout for good EMC. This series is specifically intended to cover mixed signal designs where the digital signals are less than 100 MHz and clock rise times are greater than 1 ns. In this video we will discuss how the PCB trace characteristic impedance is determined by its geometry. We will see how matching the driver to trace impedance can impact the rise time, overshoot, and ringing on digital communications signals. This is important because the overshoot and ringing can create noise problems that impacts sensitive analog and in some cases can even cause digital communications issues.

Microstrip vs Stripline

Microstrip



Stripline

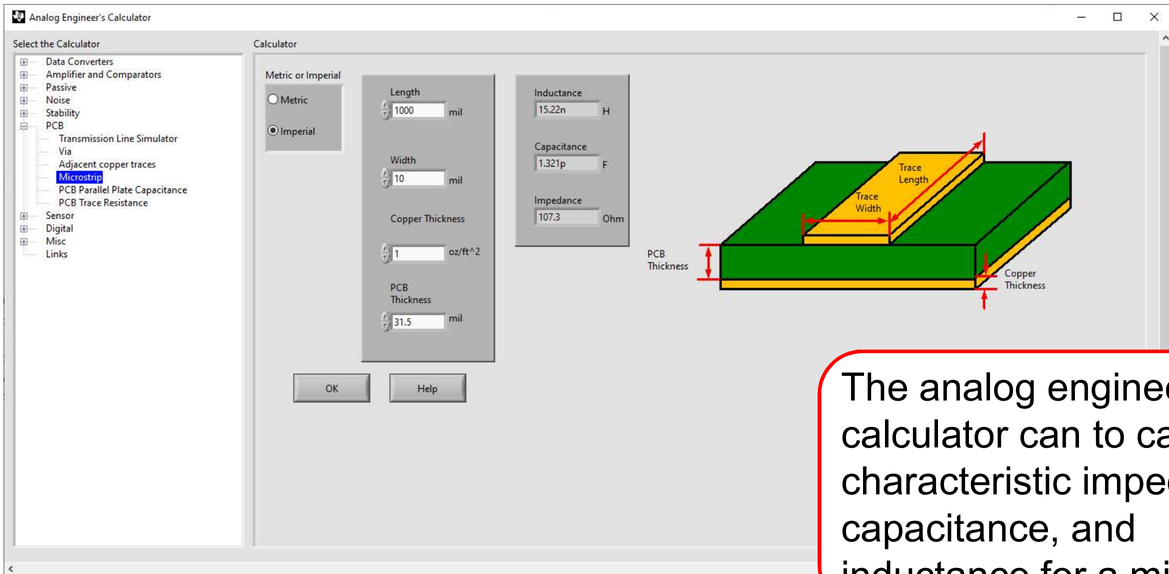


PCB Transmission Line Geometry
Microstrip - a PCB trace above a GND or Power plane. It's width, dielectric thickness, and dielectric type determine the characteristic impedance.

Frequency < 100MHz, t_r > 1ns
For this kind of application we don't need controlled dielectric (FR4 type is fine). We also don't need to choose trace thickness for 50Ω impedance. However, it is important to terminate properly.

The characteristic impedance of a line is the impedance seen by a wave front traveling down the line. Impedance mismatches between the driver, transmission line, and load will cause reflections that translate into overshoot and ringing on digital signals. The geometry of a PCB trace relative to its ground return plane Will determine its characteristic impedance. In the slide you can see the micro strip and strip line configurations. The trace width, dielectric thickness, and dielectric constant can be used to calculate the characteristic impedance. Transmission line theory is a very deep subject and we were only brush over it briefly in this section. The references provided at the end of this presentation provide a much more detailed coverage of this subject

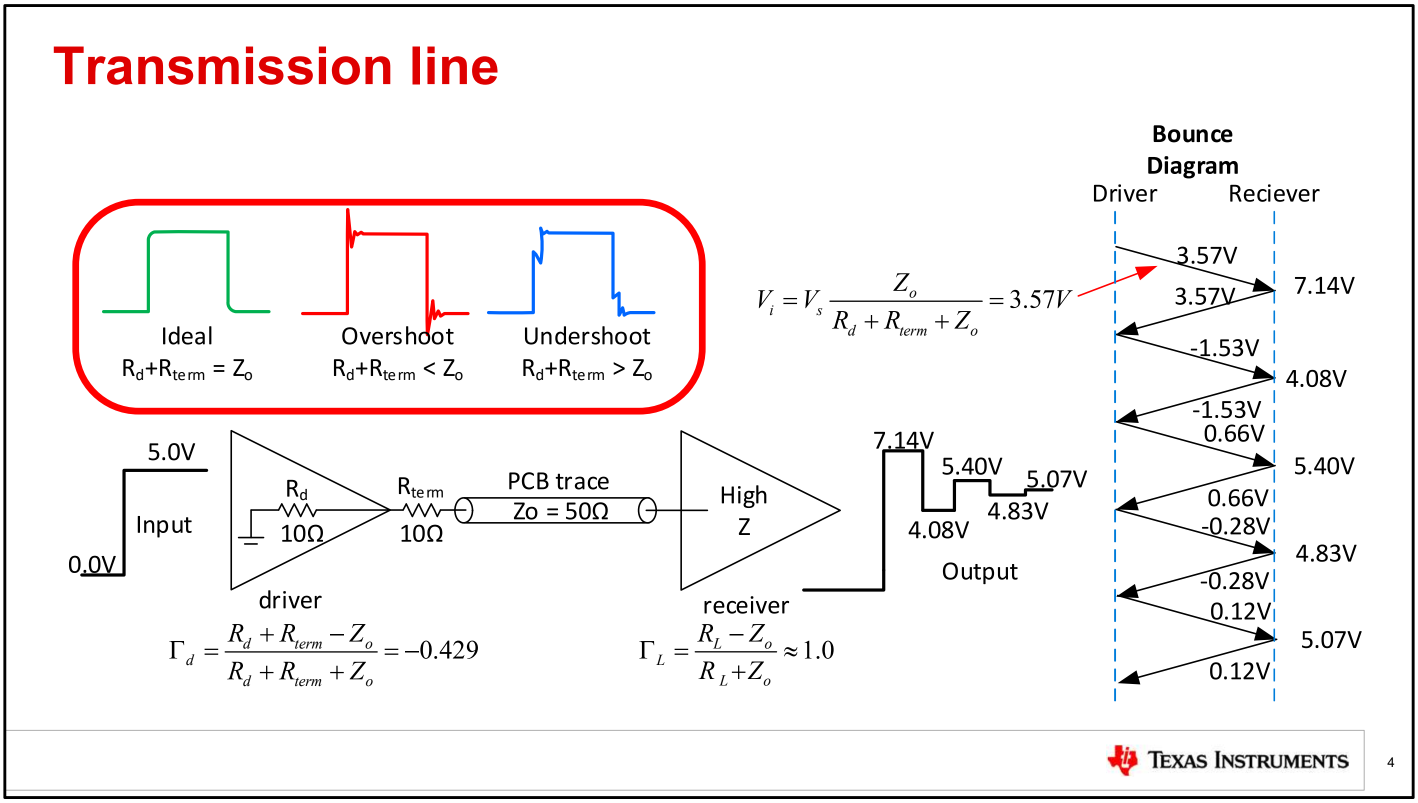
PCB Trace impedance



The analog engineer's calculator can to calculate characteristic impedance, capacitance, and inductance for a microstrip

https://www.ti.com/tool/ANALOG-ENGINEER-CALC

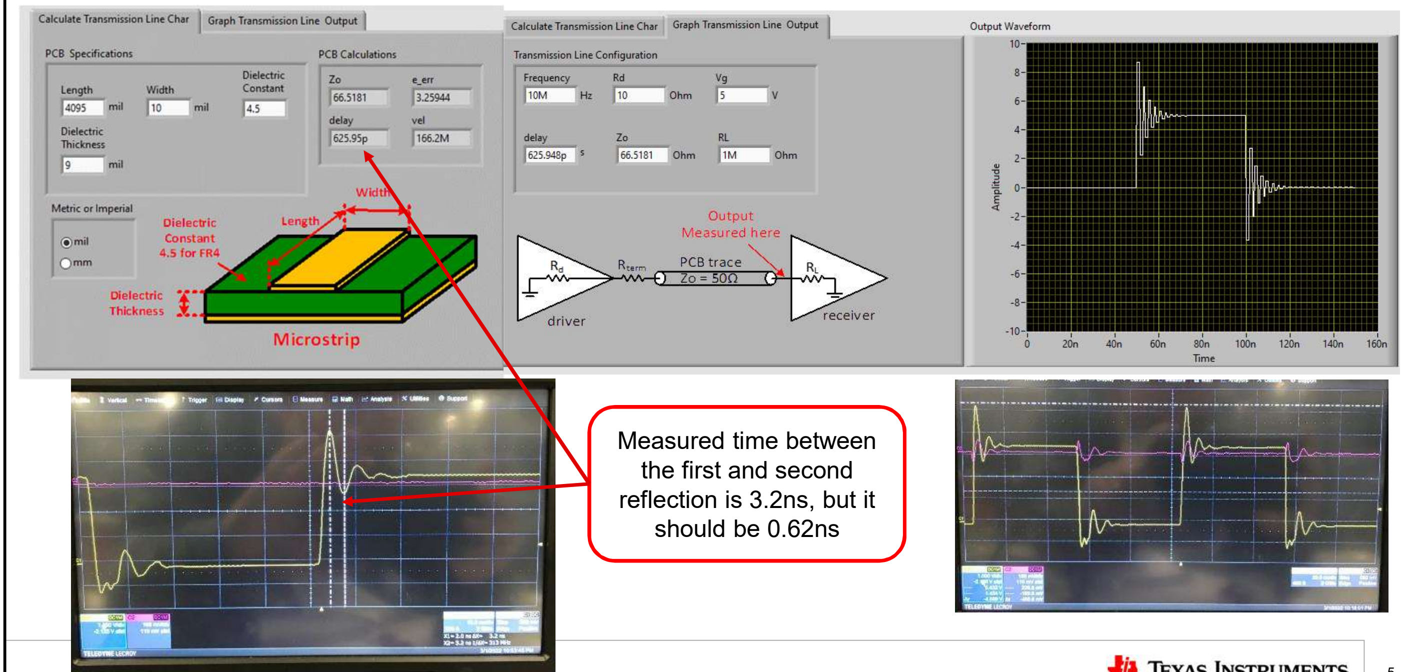
The analog engineers calculator provides a simple way to find the characteristic impedance for a micro strip configuration. Simply enter the width, length, dielectric thickness, and dielectric coefficient to find the characteristic impedance. The most common characteristic impedance is 50 homes. For most PC board traces the characteristic impedance will range from 20 ohms to 200 ohms. For this example, the characteristic impedance is 107 ohms. The link at the bottom of the page can be used download the analog engineer's calculator.



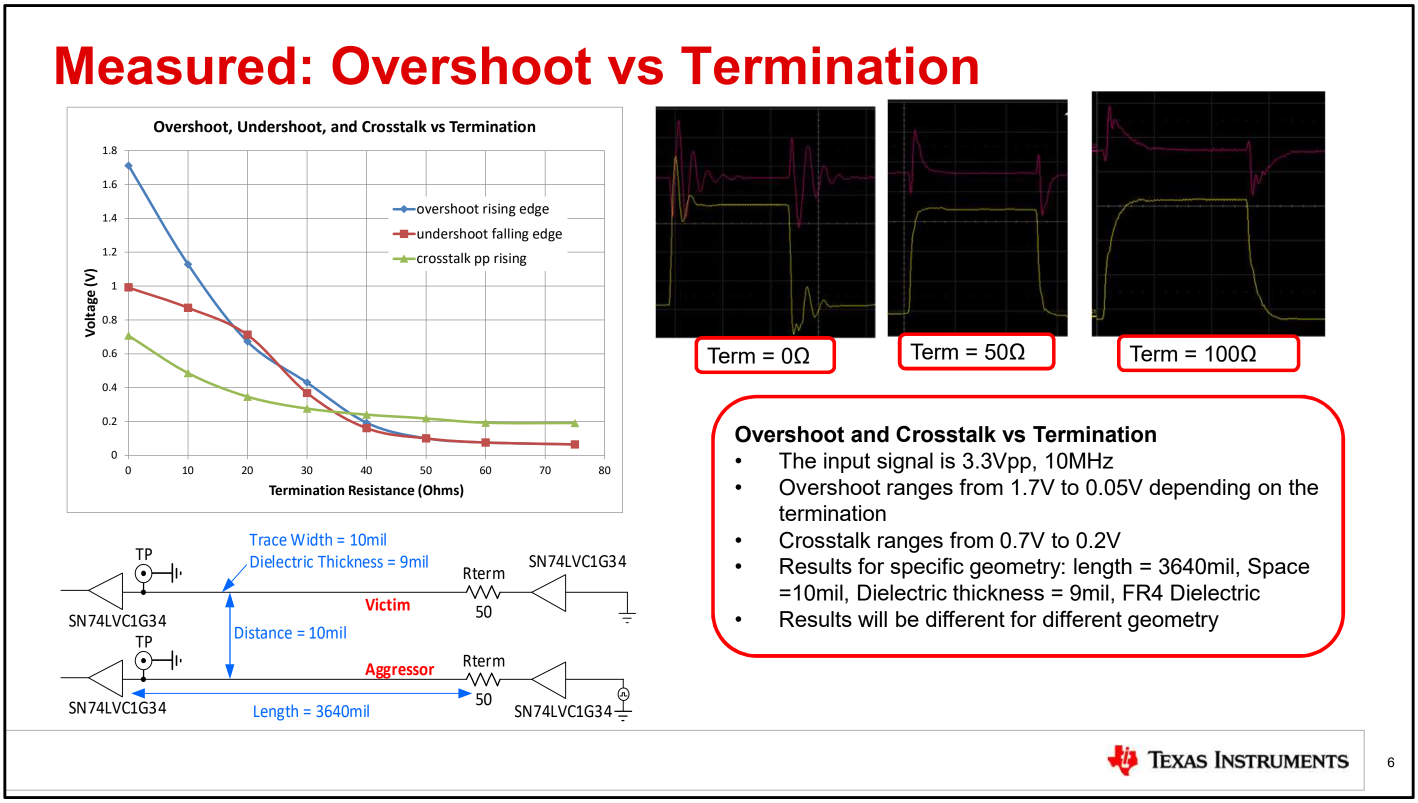
This slide shows some very basic theory for transmitting a digital signal across a PCB transmission line. Transmission lines reflections will happen whenever there is an impedance mismatch in the signal path. To avoid reflections the driver impedance must match the PC trace impedance and the load Impedance. Generally the load is the input of a gate which is effectively an infinite impedance, so this will always have a full reflection. The driver will have an internal impedance Rd and an external termination resistance Rterm. The external termination resistance is used to match the PCB trace impedance. If Rd + Rterm matches the PCB trace impedance, the signal delivered to the load will not have overshoot or undershoot. If Rd + Rterm is less than the trace impedance there will be overshoot, and it is greater than the trace impedance they will be undershoot.

A “bounce diagram” is a method used to calculate the transmission line reflections so that we can predict the waveform delivered to the load. Let’s do a bounce diagram example for system with overshoot. For this example a 5V step is applied to the transmission line, and Rd + Rterm is 20 ohms. The reflection coefficient, gamma, is calculated at the driver side as -0.429 and the receiver side as 1.0. The receiver will generally have the reflection coefficient of 1.0 as it is effectively an open circuit and will completely reflect the waveform. When the 5V step is first applied a voltage divider forms between the driver impedance and the trace impedance so that the signal on the PCB trace is 3.75V. Based on the trace length, it will take some time for the signal to travel to the receiver. When the signal reaches the receiver there is a 100% reflection and the initial signal seen at the receiver is 7.14V. The reflected signal travels back to the driver where is reflected by -0.429 because of the impedance mismatch. This reflection travels back to the receiver and causes the signal to drop to 4.08V. This process continues and you see the output waveform shown. If you did the same calculation with a matched impedance you would not see any overshoot. If you used a very larger termination resistance you would see undershoot. Now let’s take a look at a simple calculator that can automatically do this calculation.

Reflection calculations

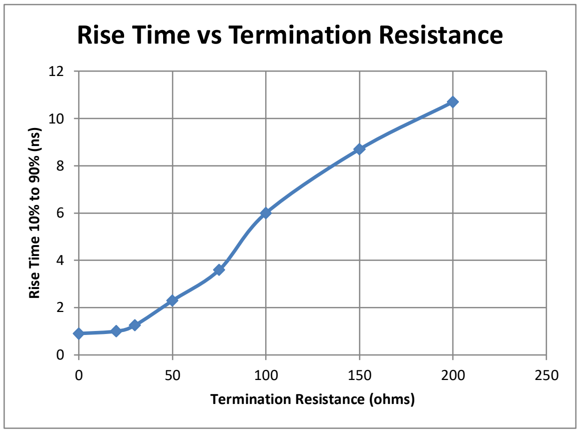


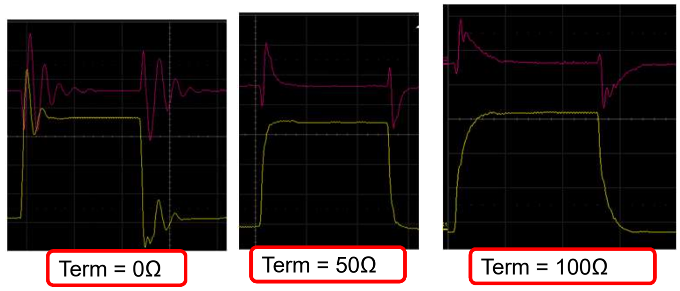
This side shows how the analog engineers calculator can be used to predict reflections. In this example, the length, dielectric constant, and dielectric thick thickness are entered. The signal frequency, amplitude, and termination impedance are also provided used to predict the reflections. Adjusting the calculator parameters is a quick and easy way to get an intuitive feel for how transmission lines work. In this example, we use the calculator with the real world PC board specifications for a measured experiment. You can see that the measured results look very similar to the predictive results but do not match perfectly. In the next few slides will look at a summary of multiple experiments related to adjusting the termination impedance.



This slide shows measured results for a transmission line where the termination resistance is adjusted across a wide range. The schematic of the circuit tested is in the lower left of the slide. A 74LVC1G34 gate is used as the transmitter and receiver. One line, called the aggressor, is driven with a 3.3V 10MHz square wave. The other line, called the victim, is driven by a logic low. The aggressor will cause crosstalk on the victim. For the experiment we will look at overshoot, and crosstalk between the two traces. Theoretically, the overshoot should be zero when the termination resistance plus driver impedance is equal to the trace impedance. The calculated impedance for this trace is approximately 66ohms. Assuming the drive impedance is 10 ohms the termination resistance should be 56ohms. Looking at the oscilloscope waveforms you can see that for zero ohms of termination resistance there is a great deal of overshoot and ringing. The 50 ohm termination has a very fast rise time but no significant overshoot. The 100ohm termination has a slow rise time. The graph summarizes the experimental results over a wide range of termination resistance. The experimental results show that the calculated trace impedance is approximately correct as a 50 ohm termination produces optimal results.

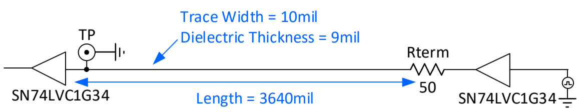
Measured: Rise time vs termination





Rise Time vs Termination

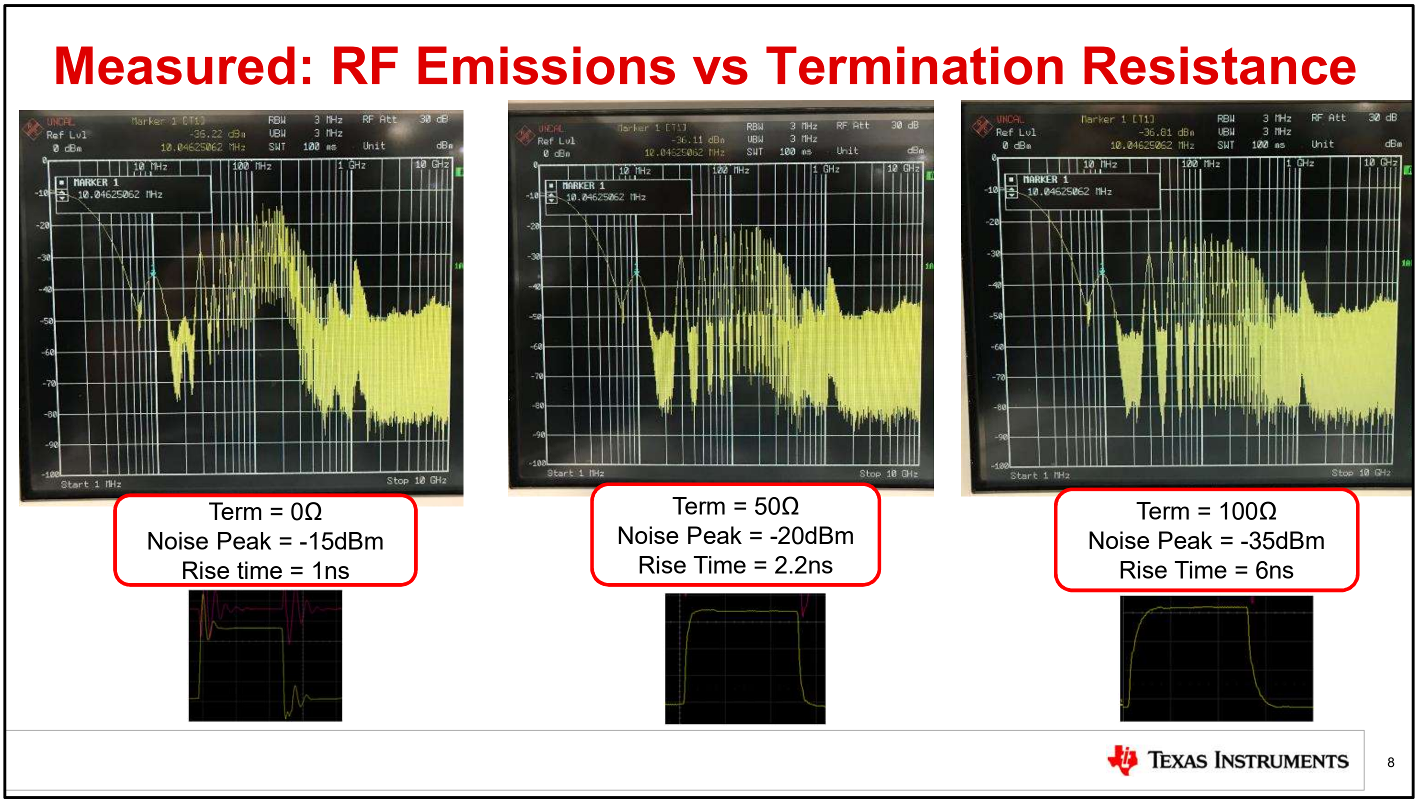
- The input signal is 3.3Vpp, 10MHz
- Rise time ranges from 1ns to 11ns depending on the termination
- Results for specific geometry: length = 3640mil, Space =10mil, Dielectric thickness = 9mil, FR4 Dielectric
- Results will be different for different geometry



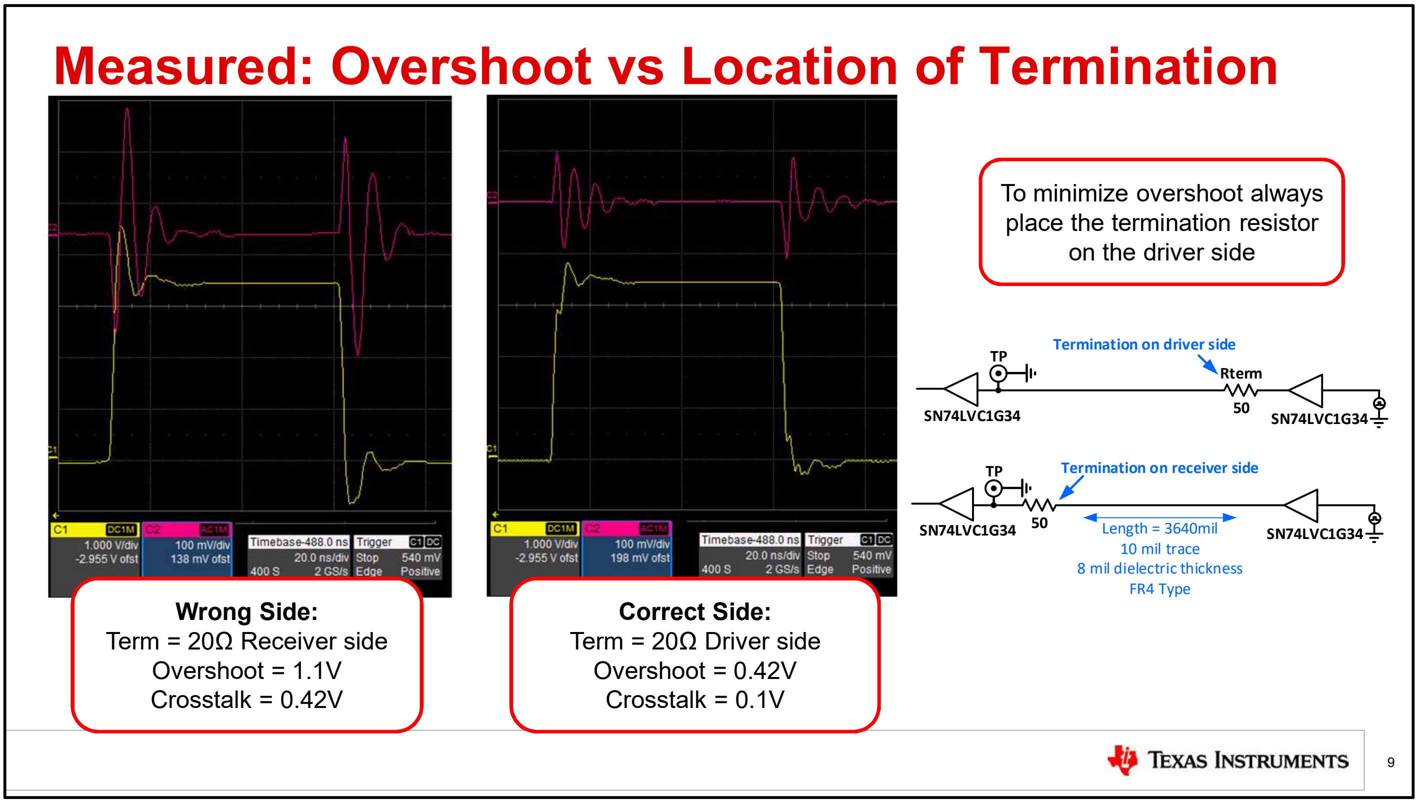
TEXAS INSTRUMENTS 7

Earlier we mentioned that rise time is important in RF systems because it determines how much high frequency noise the square wave generates. The termination resistance will directly impact the signal rise time. For small termination resistances, the signal will rise rapidly and there will be overshoot and ringing. For this circuit, the zero ohm impedance has a rise time of about 1ns. When the termination resistance matches the characteristic impedance well, the rise time will also be fast, but there will be no overshoot and ringing. For this circuit, the 50 ohm impedance has a rise time of about 2ns. For very large impedances the undershoot decreases the rise time. In this case a 100 ohm impedance has a 6ns rise time.

In the next slide we can see how the rise time relates to RF emissions.



Here are the radiated emissions measured with RF sniffer probes. The emissions for the circuit from the last page were measured with 0ohm, 50ohm, and 100ohm termination. Increasing the termination resistance will increases the rise time which decreases the RF emissions. The rise time is 1ns, 2.2ns, and 6ns for 0ohm, 50ohm, and 100ohm respectively. For a 1ns rise time the noise peak is -15dBm. For a 2.2ns rise time the noise decreases to -20dBm, and for a 6ns rise time the noise decreases further to -35dBm. Thus, the overall noise decreased by 20dB or a factor of 10 simply by changing the termination.



One last thing to consider regarding termination impedances. The termination impedance should be near the driver for the best impact. The goal is to match the impedance discontinuity at the output of the gate not at the end of the transmission line. Remember the receiver is a high impedance, so placing a termination in series with an open circuit isn't helpful. In this experiment we compare overshoot and ringing when the termination is placed on the correct and wrong side of the line. The measured results show that placing the termination on the wrong side of the line more than doubles the overshoot. So, please be careful to place the termination near the driver.

Thanks for your time!
Please try the quiz.

Texas Instruments 10

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

Quiz: Impedance Matching

1. Which is ***not*** a factor in determining PCB trace impedance?
   a) Trace Length.
   b) Trace width.
   c) Dielectric thickness.
   d) Dielectric material

2. What problem will occur if a termination resistance is too small?
   a) The logic circuit will be susceptible to damage
   b) The signal will have large overshoot and ringing.
   c) The signal will have large undershoot.

TEXAS INSTRUMENTS 11

Question 1, Which is not a factor in determining PCB trace impedance?

The correct answer is “a”, trace length. The trace width, dielectric thickness, and dielectric material are the only factors that set the impedance.

Question 2, What problem will occur if a termination resistance is too small?

The correct answer is “b”, The signal will have large overshoot and ringing.

Quiz: Impedance Matching

3. What are some issues that you would expect from mismatched impedances?
   a) Overshoot and ringing
   b) Excessive crosstalk
   c) Excessive RF emissions.
   d) Data communications errors.
   e) All of the above
   f) None of the above.

4. (T/F) The termination resistor can be placed anywhere on the line for impedance matching.
   a) True
   b) False

Question 3, What are some issues that you would expect from mismatched impedances?

The correct answer is “e” all of the above. Impedance mismatching can cause overshoot, crosstalk, RF emissions, and data communications errors.

Question 4, true or false The termination resistor can be placed anywhere on the line for impedance matching.

The correct answer is “b” false. The termination resistor needs to be placed near the driving gate.

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