Application Note

Simple Methods to Reduce EMI from a PCB Trace

Preston Vanderlight

Abstract

As data communication speed increases and higher frequencies are required to transmit information, electromagnetic (EM) emissions start to become a design requirement. This application note provides simple steps to reduce electromagnetic interference (EMI) while minimizing design complexity and cost.

Table of Contents

Trademarks

All trademarks are the property of their respective owners.

1 Introduction

As operating frequencies in modern electronic systems continue to increase, the EMI generated by PCB traces becomes an increasingly critical design consideration. At high frequencies, PCB traces no longer act as simple wires but instead function as transmission lines capable of radiating significant electromagnetic energy. This radiated energy can interfere with nearby circuits, degrade signal quality, and cause compliance issues with electromagnetic compatibility (EMC) standards.

The challenge of managing EMI becomes particularly acute in designs featuring long traces, high clock speeds, or sensitive analog circuits. Traditional design approaches that work well at lower frequencies prove inadequate, as signal reflections and overshoot become pronounced, and high-frequency components of square waves radiate efficiently.

This application note presents practical methods for reducing EMI emissions from PCB traces through two primary techniques: proper signal termination and strategic signal filtering. By implementing series source-side termination to match transmission line impedance and employing RC filtering to attenuate high-frequency components, designers can effectively minimize signal reflections, overshooting, and unwanted radiation. This document provides step-by-step procedures for characterizing source impedance, calculating termination resistance values, and selecting appropriate filter parameters.

2 Detailed Description

2.1 Signal Integrity

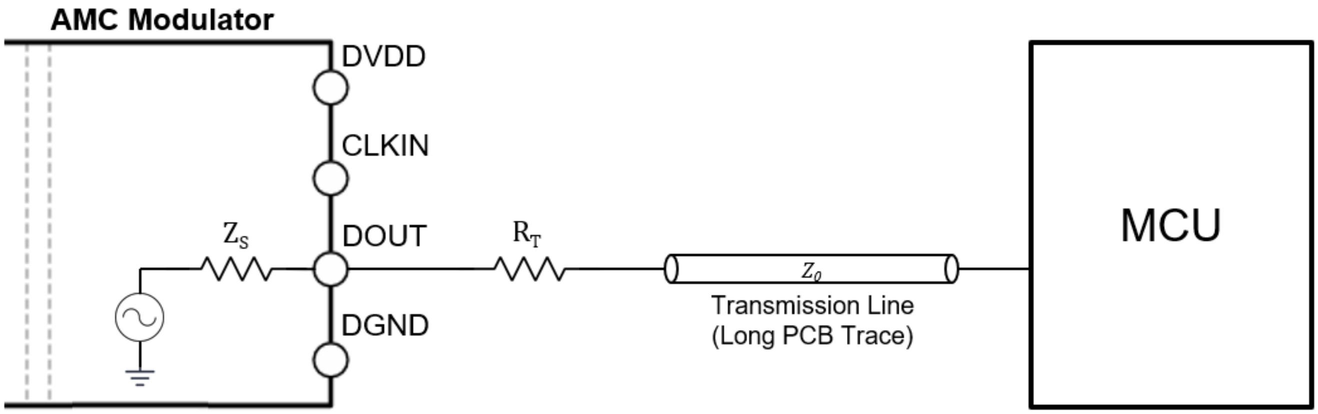
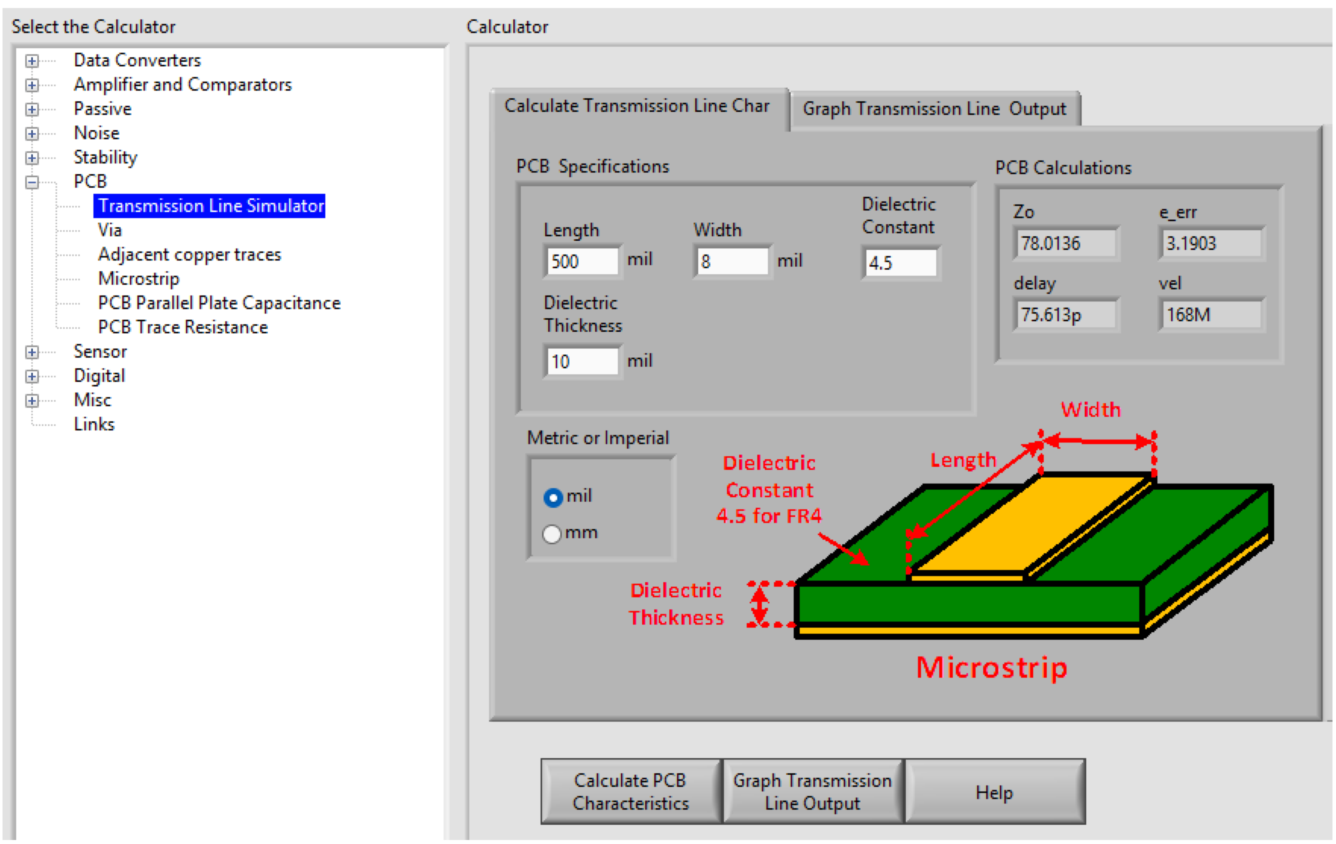


Figure 2-1. Series Terminated Transmission Line

The goal of series termination is to have the sum of the source impedance (Z_S) and the termination resistance (R_T) equal to the characteristic impedance of your transmission line (Z_O). To determine the impedance of your transmission line, use tools such as TI's Analog Engineer's Calculator or Altium to calculate the trace impedance based on the physical characteristics of the transmission line.



The next step is to characterize the output impedance (Z_{out}) of your driver. Sometimes, the output impedance of the device is not readily available from a datasheet and must be characterized. A simple setup to do this is shown in Figure 2-2.

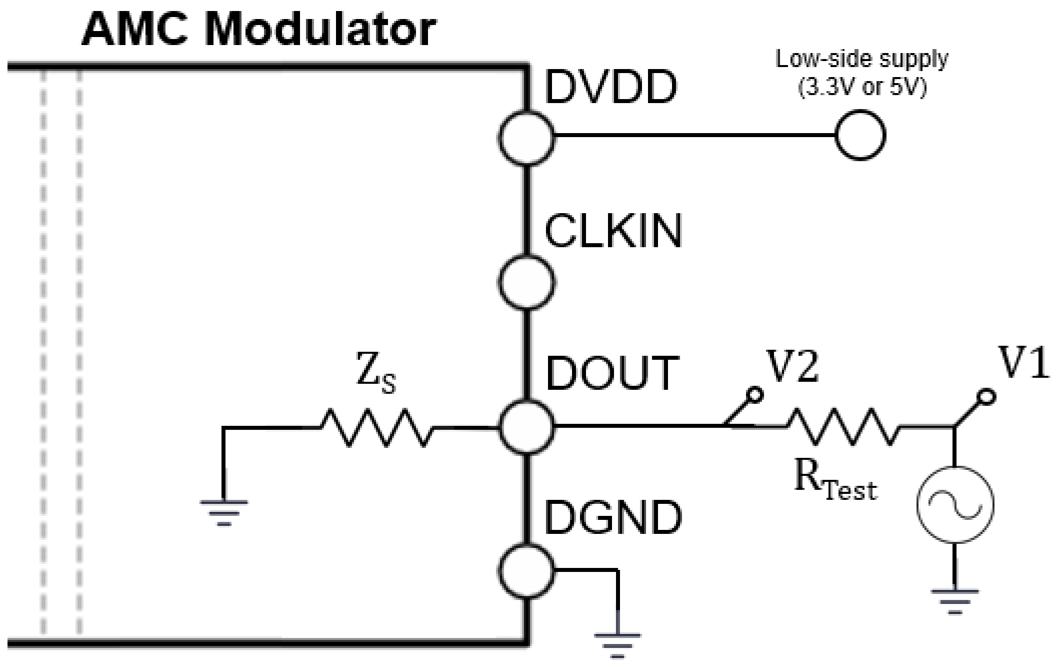


Figure 2-2. Device Impedance Characterization Setup

To measure the device impedance, Z_S, make sure the device is powered on and apply a test voltage to V1. Now measure the voltage drop across R_{Test} and calculate the current and voltage supplied to the device. Note that the device's impedance increases at higher frequencies. To simplify the calculation, we can assume the impedance as just a resistance for most applications.

After you have calculated your source and PCB impedance, use this formula to calculate the impedance of your termination resistor: $R_T = Z_0 – Z_S$. Figure 2-3 through Figure 2-5 show examples of an under, over, and perfectly damped system. An underdamped system has overshoot and ripples, the overdamped system has a longer settling time, and the perfectly damped system is the ideal.

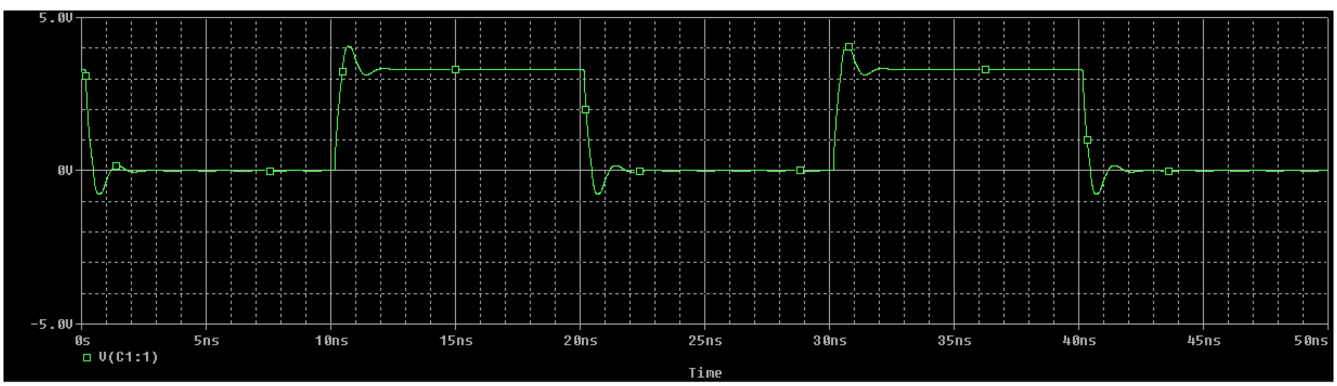


Figure 2-3. Underdamped System (Z_S + R_T < Z_0) Resulting in Overshooting

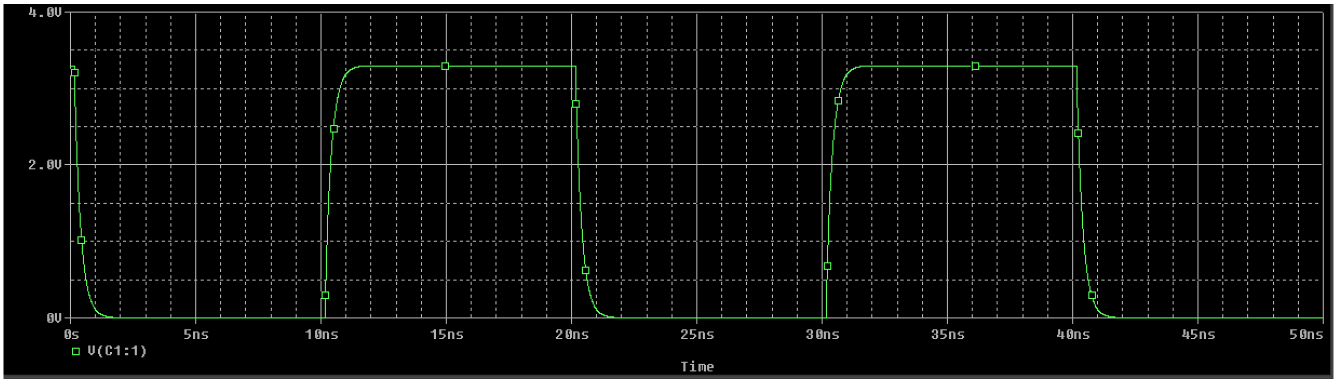


Figure 2-4. Perfectly Damped System ($Z_S + R_T = Z_0$)

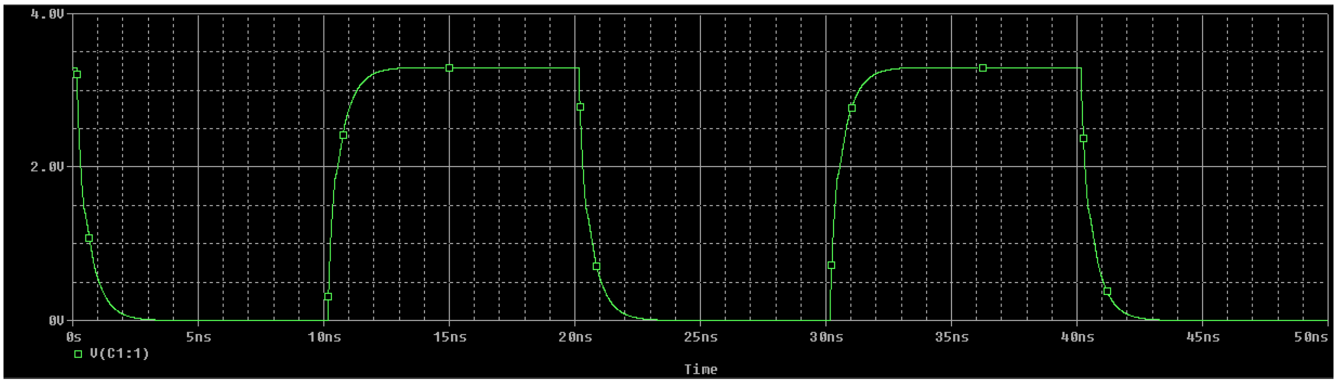


Figure 2-5. Overdamped System ($Z_S + R_T > Z_0$) Resulting in Undershooting

2.2 Signal Filtering

After properly terminating your PCB trace, EMI can still be an issue. By altering the shape of the signal waveform, you can attenuate high frequency components of your signal by rounding the square wave. This can be achieved with an RC filter at the source-side of the PCB trace.

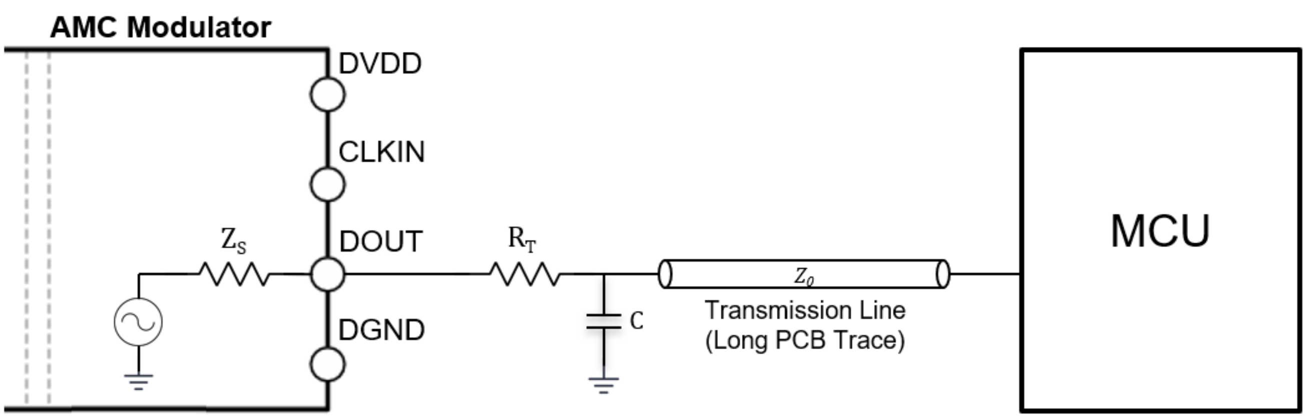


Figure 2-6. RC Filtering Scheme

First, determine what your output signal frequency is. The capacitor value is calculated based on the RC cutoff frequency:

$$f = \frac{1}{2\pi R_T C} \tag{1}$$

This frequency represents the -3db cutoff point of the signal. A standard cutoff frequency to start with is twice your square wave frequency. If larger high-frequency attenuation is required, you can use a lower cutoff frequency. Figure 2-7 through Figure 2-9 show the effects of lower RC cutoff frequencies on a square wave.

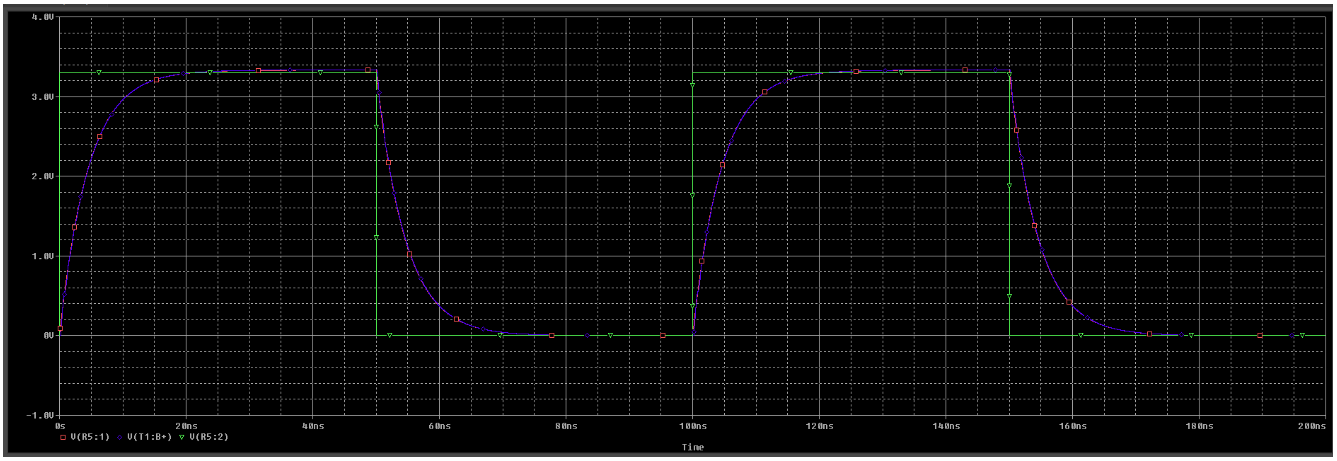


Figure 2-7. Cutoff Frequency = 2 × Square Wave Frequency

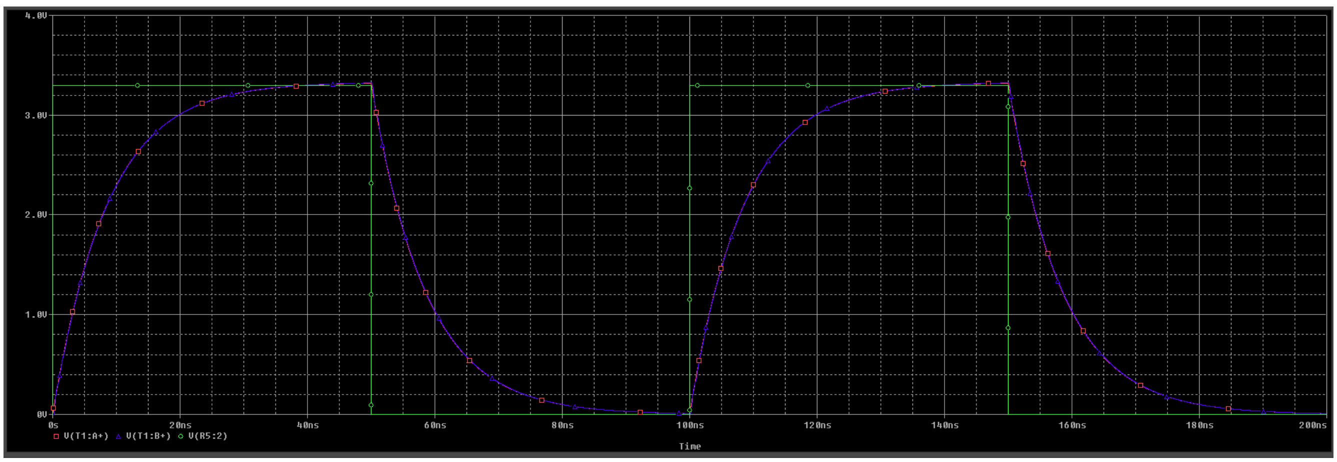


Figure 2-8. Cutoff Frequency = Square Wave Frequency

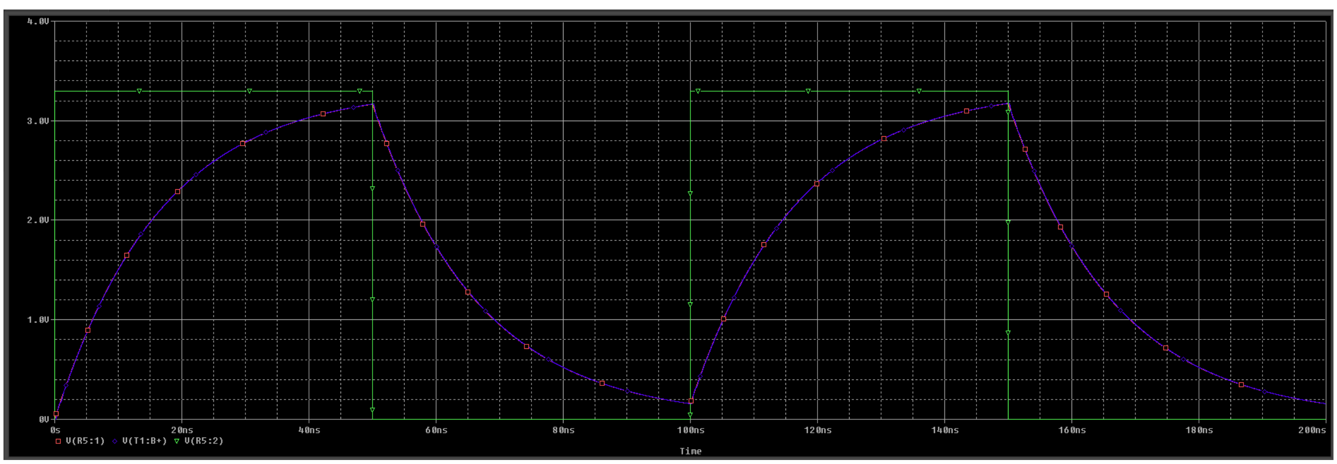


Figure 2-9. Cutoff Frequency = ½ Square Wave Frequency

2.3 Shielding

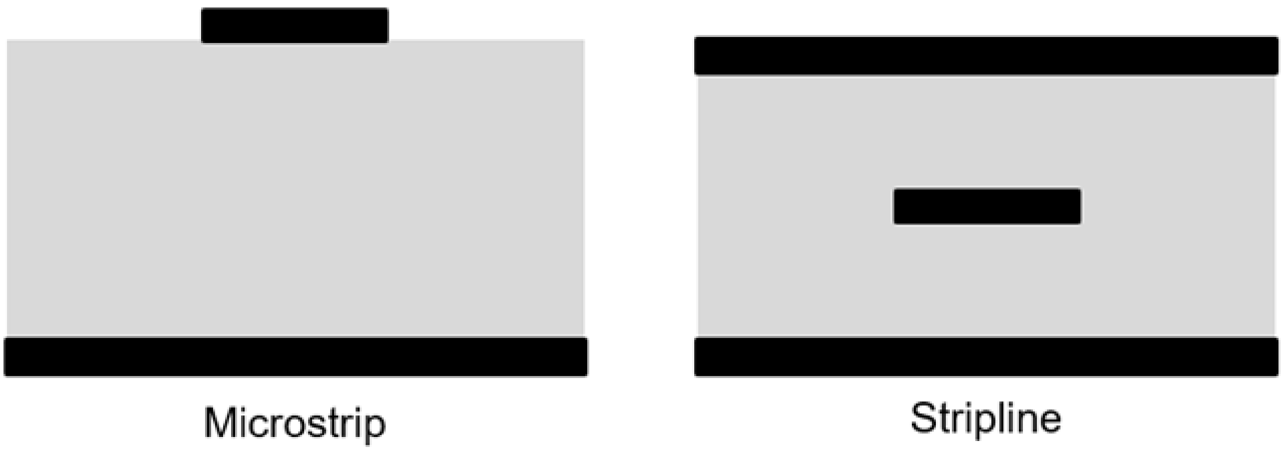


Figure 2-10. Microstrip vs Stripline Cross Section

Another way to reduce EMI is shielding. By inclosing noisy signal traces between copper planes emissions can be reduced. It is best if the copper planes above and below the trace are ground planes, but one can also be a power plane if necessary to satisfy design constraints. If additional EMI reduction on a stripline is desired, the methods in this document can be used. The impedance calculation for a stripline trace is more involved but can be done with many PCB design programs.

2.4 Results

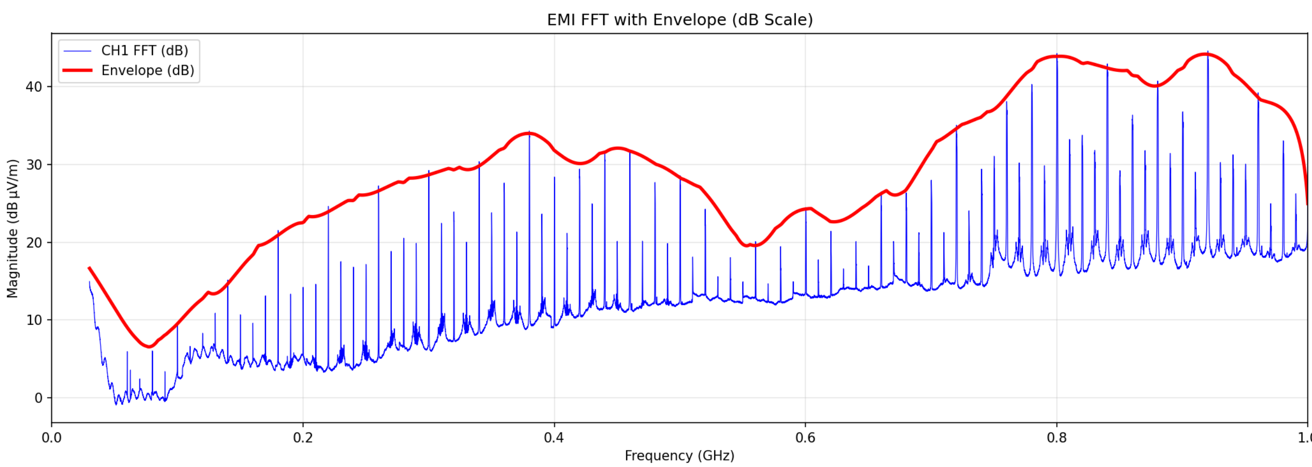


Figure 2-11. Example of Enveloping a CISPR Plot

Figure 2-11 shows a plot using enveloping to trace the top of the CISPR emissions plot. This is a clever method to simplify CISPR plots and allows them to be overlaid for easy comparison.

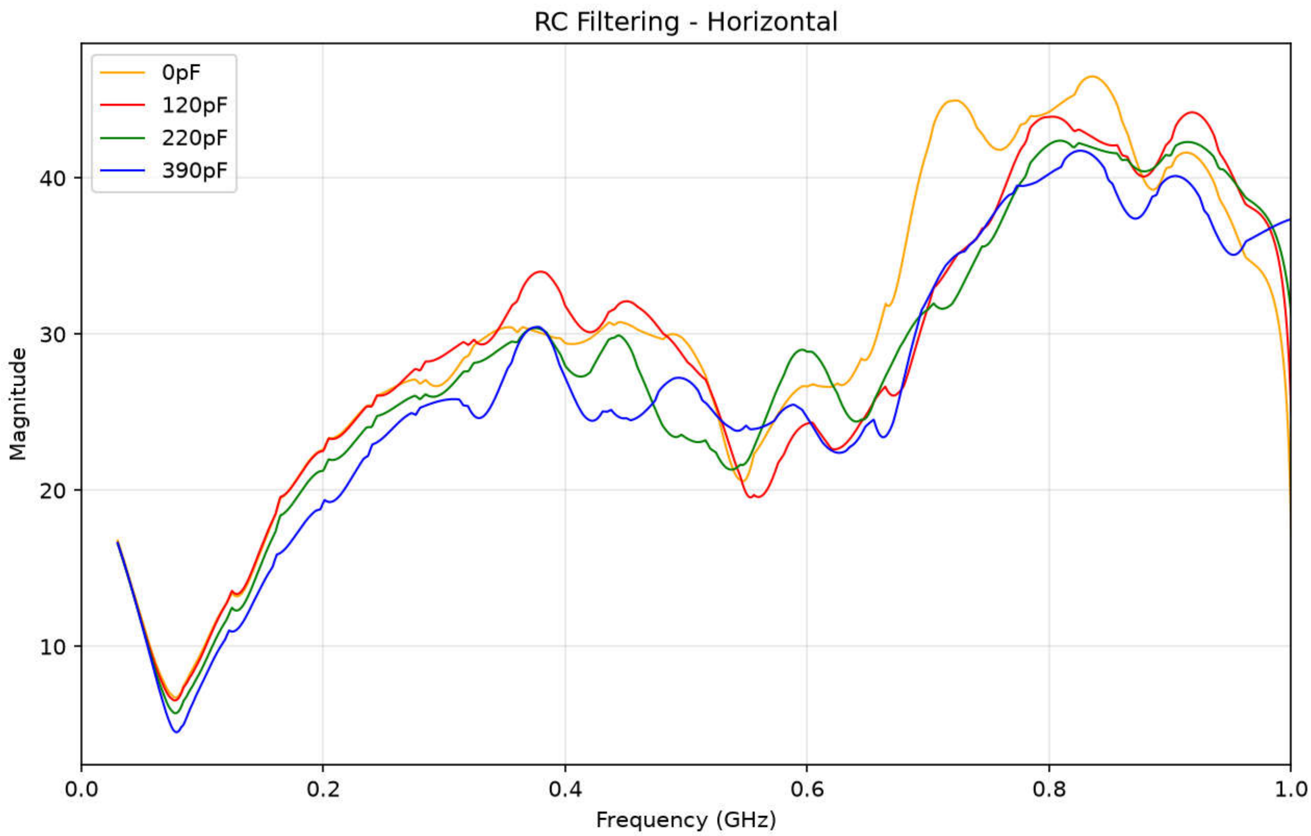


Figure 2-12. RC Filtering Results with Different Sized Capacitors

Figure 2-12 shows the results of the RC filtering in CISPR emissions. The graph overlays the envelopes of the different sized capacitors in RC filters. As seen in the plot, using a larger capacitor lowers the RC filters cutoff frequency, which brings down the EMI. This effect is more pronounced at high frequencies. Note the 0pF line is just series terminated with no capacitor.

Figure 2-13. CISPR Test Setup

Figure 2-13 shows the test setup for the CISPR32 standard, The CISPR11 standard was used over the course of creating this application note. The only major differences are the lack of anechoic tiles between the device under test and the receiver and the frequency measurement range. The device was rotated in increments of 90° for this test and the highest magnitude for each frequency was recorded.

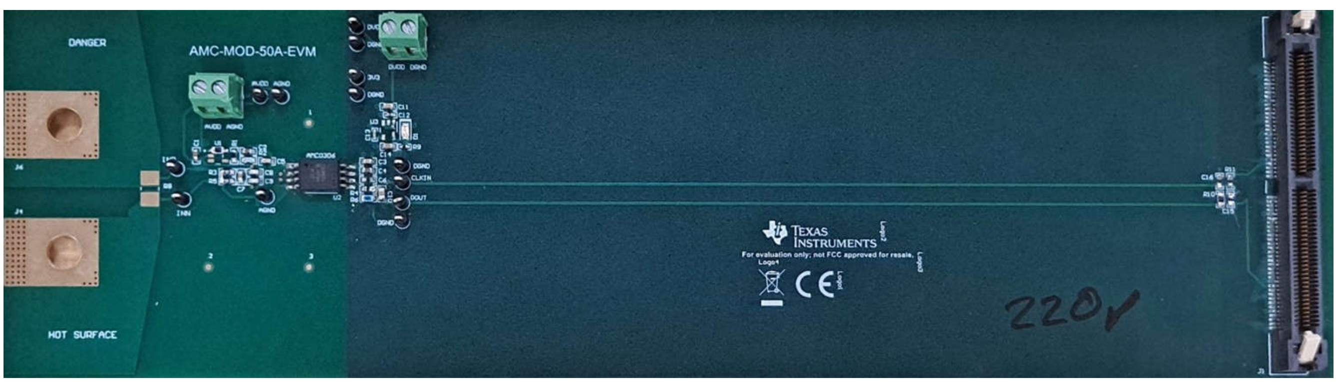


Figure 2-14. Device Under Test

Figure 2-14 shows the device used to test EMI in the chamber. One of long traces was connected to the digital output of the AMC0306M25 and the other was left unconnected. Batteries were used to power the device during CISPR testing to reduce noise from the power source.

3 Summary

By applying series source-side termination and RC filtering to PCB traces, designers can achieve measurable reductions in EMI emissions. Testing in a CISPR chamber demonstrated that RC filtering is an effective tool to reduce EMI emissions from a long PCB trace. By reducing the cutoff frequency of the filter and sacrificing some signal integrity, even greater reduction of EMI can be achieved.

4 References

1. Texas Instruments, *PCB Design Guidelines For Reduced EMI*, application note.
2. Texas Instruments, *AN-903 A Comparison of Differential Termination Techniques (Rev. B)*, application note.