

Characterization and reduction of line-to-line crosstalk on printed circuit boards

by

Joshua Adam Welch

B.S., Kansas State University, 2016

A THESIS

submitted in partial fulfillment of the requirements for the degree

MASTER OF SCIENCE

Department of Electrical and Computer Engineering
College of Engineering

KANSAS STATE UNIVERSITY
Manhattan, Kansas

2018

Approved by:

Major Professor
Dr. William B. Kuhn

Copyright

© Joshua Welch 2018.

Notice: This manuscript has been authored using funds under the Honeywell Federal Manufacturing & Technologies under Contract No. DE-NA-0002839 with the U.S. Department of Energy. The United States Government retains and the publisher, by accepting the article for publication, acknowledges that the United States Government retains a nonexclusive, paid-up, irrevocable, world-wide license to publish or reproduce the published form of this manuscript, or allow others to do so, for United States Government purposes.

Abstract

An important concern for high speed circuit designs is that of crosstalk and electromagnetic interference. In PCB board-level designs, crosstalk at microwave frequencies may result from imperfections in shielding of PCB interconnects or more generally transmission lines. Several studies have been done to characterize and improve the isolation between PCB transmission lines for both digital and RF circuits. For example, previous studies in the microwave region have examined the effect that line type, line length, and separation have on crosstalk and suggest that without full shielding, the upper limit of isolation is on the order of 60dB for traditional board-level lines [1].

In order to more fully characterize crosstalk and improve isolation above 60 dB, this thesis studies signal-to-ground-plane separation, considers advanced line types, and examines the effect of 3D shielding. Results are presented from 100MHz to 30GHz for the traditional transmission line structures of microstrip, CPW, differential pair and CPW differential pair. This study shows that with a halving of distance between signal and ground planes, isolation between transmission lines can be reduced by as much as 20dB, making this one of the best ways to improve performance. Advanced methods of shielding are then presented. Direct launch stripline and single-sided CPW improve upon existing crosstalk reduction techniques, while split shielding and ablation of dielectric PCB material are also proposed.

The data and additional crosstalk reduction techniques discussed in this thesis serve two purposes. One: with a more complete understanding of the effects that transmission line types and parameters have on crosstalk, engineers can quickly identify potential crosstalk issues and resolve them before manufacturing. Second, this thesis presents the engineer with four new additional techniques that may become available in advanced manufacturing environments. Such

techniques can further reduce crosstalk and may allow for isolation values to approach 100 dB at the PC board level.

Table of Contents

List of Figures	vii
List of Tables	ix
Acknowledgements	x
Dedication	xi
Chapter 1 - Introduction and Background	1
1.1 – Electromagnetic Compatibility, Crosstalk and Modern PCB Boards.....	1
1.2 – Additive Manufacturing and Laser Ablation	2
1.3 – Prior Work	4
1.4 – Thesis Structure	5
Chapter 2 - Signal and Ground Plane Separation Isolation Effects	7
2.1 – Background	7
2.2 – Mathematical Model of Crosstalk VS Signal/Ground Separation.....	8
2.3 – Experimental Setup.....	12
2.4 – Experimental Results	14
2.4a – Microstrip Experimental Results.....	15
2.4b – CPW Experimental Results.....	15
2.4c – Differential Pair Experimental Results	16
2.4d – CPW Differential Pair Experimental Results.....	17
2.5 – Comparison of Different Transmission Lines and Conclusion	18
Chapter 3 - Direct Launch Stripline and Signal Isolation.....	21
3.1 – Introduction to Stripline.....	21
3.2 – Stripline VS Direct Launch Stripline.....	22
3.3 – Experimental Setup.....	24
3.4 – Experimental Results	26
3.4a – Traditional Launch Stripline with Unshielded Transitions	27
3.4b – Direct Launch Stripline Results	28
3.5 – Comparison of Stripline and Direct Launch Stripline	31
3.6 – Conclusion	31
Chapter 4 - Single Sided Coplanar Waveguide (SS-CPW)	33

4.1 – Background	33
4.2 – General Design of SS-CPW	34
4.3 – Experimental Setup	36
4.4 – Experimental Results	37
4.4a – Results with Signal to Ground Plane Separation of 6.7mils	37
4.4b - Results with Signal to Ground Plane Separation of 53.7mils.....	39
4.5 – Example Use Case for SS-CPW	41
4.6 – Conclusion	42
Chapter 5 - Ablation of Dielectric PCB Layers	44
5.1 – Introduction to Laser Ablated Removal of Top Dielectric	44
5.2 – Mathematical Theory of Dielectric Removal for Capacitive Coupling.....	45
5.3 – Experimental Setup.....	46
5.4 – Experimental Results	47
5.4 – Conclusion	55
Chapter 6 - Split Shielding VS Traditional Shielding	56
6.1 – Introduction.....	56
6.2 – Simulation Results	58
6.3 – Experimental Setup.....	62
6.4 – Experimental Results	63
6.5 – Shielding Aperture and Isolation	65
6.6 – Conclusion	66
Chapter 7 - Conclusions and Future Directions	67
7.1 – Conclusions.....	67
7.2 – Future Work.....	67
References.....	69

List of Figures

Figure 1: 3d printed RF shields from the University of Arkansas.....	3
Figure 2: Top dielectric of PCB removed courtesy of the University of Nebraska.....	3
Figure 3: Example of crosstalk test structure.....	4
Figure 4: Lumped component model of crosstalk	8
Figure 5: Magnetic fields from aggressor line interfering with victim line.....	9
Figure 6: Electric fields between aggressor and victim line when H is large.....	10
Figure 7: Signal to ground plane test board.....	12
Figure 8: Landed GSG probe on CPW transmission line.....	13
Figure 9: Example measurement setup.....	14
Figure 10: FEXT of microstrip to microstrip at varying ground distances.....	15
Figure 11: FEXT of CPW to microstrip at varying ground distances	16
Figure 12: FEXT of differential pair to microstrip at varying ground distances	17
Figure 13: CPW differential pair to microstrip at varying ground distances.....	18
Figure 14: Comparison of FEXT for transmission lines with ground 6.7mils below.....	19
Figure 15: Comparison of FEXT for transmission lines with ground 60.4mils below	19
Figure 16: General design of walled stripline transmission line.....	21
Figure 17: Direct launch stripline with connectors sunken into board	23
Figure 18: Direct launch stripline with connectors and shielding	24
Figure 19: Traditional stripline test board	25
Figure 20: Direct launch stripline test board prior to adding top shielding.....	25
Figure 21: Closeup of traditional vs direct launch.....	26
Figure 22: Coupling of traditionally launched stripline configurations to microstrip	27
Figure 23: Coupling of traditionally launched stripline to stripline configurations	28
Figure 24: FEXT stripline to stripline no via fencing.....	29
Figure 25: FEXT for direct launch stripline to stripline with via fencing	30
Figure 26: FEXT stripline to stripline with via fence and flanks	31
Figure 27: Traditional design of a CPW transmission line.....	33
Figure 28: General design of SS-CPW transmission line (from [17]).....	35
Figure 29: SS-CPW FR4 test board.....	36

Figure 30: Comparison of FEXT at 6.7mils (from [17])	37
Figure 31: Comparison of reflection at 6.7mils (from [17]).....	38
Figure 32: Comparison of insertion loss at 6.7mils (from [17]).....	39
Figure 33: Comparison of FEXT at 53.7mils (from [17])	40
Figure 34: Comparison of reflection at 53.7mils (from [17]).....	40
Figure 35: Example use case of SS-CPW (from [17]).....	41
Figure 36: Test Board with Half Trench Ablation (Ablation courtesy of UNL)	46
Figure 37: Measurements being made with absorber material in place.....	48
Figure 38: FEXT of microstrip at 6.7mils	48
Figure 39: Reflection of microstrip at 6.7mils.....	49
Figure 40: FEXT of microstrip at 53.7mils	50
Figure 41: Reflection of microstrip at 53.7mils.....	51
Figure 42: FEXT of CPW with signal to ground plane separation of 6.7mils.....	52
Figure 43: Reflection of CPW at 6.7mils.....	53
Figure 44: FEXT of CPW with signal to ground plane separation of 53.7mils.....	53
Figure 45: Reflection of CPW at 53.7mils.....	54
Figure 46: Example of shielding with separate cavities	56
Figure 47: 3d printed RF shields (Courtesy of UAR).....	57
Figure 48: Simulation setup for shielding test with split shield vs single shield.....	59
Figure 49: Simulated FEXT with separation of 30mils	60
Figure 50: Simulated FEXT with separation of 60mils	60
Figure 51: Simulation of FEXT with separation of 120mils	61
Figure 52: Split shield test board	62
Figure 53: Test board with clamped on shielding.....	63
Figure 54: Measurement of FEXT with separation of 40mils	64
Figure 55: Measurement of FEXT with separation of 120mils	64
Figure 56: Possible aperture locations on test board	66

List of Tables

Table 1: Signal trace width in mils	14
Table 2: Width comparison of SS-CPW (from [17]).....	42

Acknowledgements

The author would like to give a special thanks to Dr. William Kuhn for the opportunity to join this research project. It has truly been a wonderful experience. The time and advice you have given me over the course of my graduate career is invaluable and will not be forgotten.

The author would also like to thank Dr. Yongfeng Lu and Timothy Carlson from the University of Nebraska for the laser ablation of the printed circuit boards found in chapters 3 and 5. The author would also like to thank Dr. Alan Mantooth and Zeke Zumbro from the University of Arkansas for the 3d printed RF shields found in chapter 6. Without the work of both of these groups, research into advanced PCB fabrication techniques would not have been possible.

Finally, the author would like thank the committee members Dr. Gruenbacher and Dr. Prakash for the feedback concerning my research. It was very much appreciated.

Dedication

I dedicate this thesis to my wife Rachel for all her love and support throughout my academic journey. I also would like to dedicate this thesis to my parents, Mark and Jacqueline Welch for their love and patience with me. Thank you for instilling in me the value of education.

Chapter 1 - Introduction and Background

1.1 – Electromagnetic Compatibility, Crosstalk and Modern PCB Boards

A paramount concern for engineers of high speed circuits is signal integrity and electromagnetic interference (EMI). Electromagnetic interference is defined as interference from outside radio-frequency waves on a circuit. Radio waves not only come from the surrounding environment but also from nearby circuits on the same printed circuit board (PCB). If not properly accounted for, EMI can prevent the correct operation of a circuit. A specific case of electromagnetic interference is crosstalk. Crosstalk is when an active line's signal couples to a victim line. While this thesis will focus primarily on crosstalk, results can be extended to other forms of EMI.

A traditional way of dealing with electromagnetic interference is to enclose the PCB in a RF shield structure. In general, this structure is machined out of a single piece of metal. To reduce crosstalk from components on the same board, separate chambers are often milled into the single shield. While traditional RF shielding does a good job of protecting a circuit from ambient interference, it does not prevent crosstalk between transmission lines within the same cavity. Since traditional RF shielding is milled from a single piece of metal, it can also add cost, weight, and size to the final product.

In addition to shielding, RF engineers can utilize more complex transmission line structures to reduce crosstalk. Examples of commonly used structures include co-planar waveguide, stripline and differential pair. When compared to microstrip, each of these structures reduces crosstalk and increases the overall isolation of the circuit. However, due to varying degrees of complexity, these structures are not always practical to implement. Even when using structures that increase isolation, RF engineers must have a firm understanding of how spacing

between lines, shielding and apertures, and other adjacent components and circuits affect crosstalk between transmission lines. Hence, these issues form the focus of this thesis.

This research first seeks to characterize the effect that different ground plane distances have on traditionally used transmission line structures. This research, coupled with previous work will give designers a more complete idea of how design and layout decisions affect isolation. In addition, four new methods are presented to further reduce crosstalk and allow for more design flexibility. Direct launch stripline and single-sided coplanar waveguide methods seek to extend existing transmission line methods by increasing both ease of use as well as isolation. Methods three and four seek to utilize cutting edge post production techniques including dielectric material ablation and addition of microwave absorbing materials to increase isolation.

1.2 – Additive Manufacturing and Laser Ablation

In this thesis, two advanced production techniques enabled by 3d fabrication methods are used to increase isolation on PC boards. Additive manufacturing, specifically in the form of 3d printing was used to make RF shield structures. Additive manufacturing allows for more intricate shield structures which can more easily conform to circuit board structures when compared to traditionally manufactured shields. In addition, additive manufacturing allows for rapid prototyping and turn around [2]. An example of 3d printed RF shielding is shown in Figure 1.



Figure 1: 3d printed RF shields from the University of Arkansas

Unlike 3d printing, laser ablation is a subtractive process. Laser ablation can be used to selectively remove solid materials from a surface. By controlling the power of the laser, the depth of material removed can also be controlled. In this research, laser ablation was used to remove the dielectric material surrounding traces on a PCB as shown in Figure 2. By removing the dielectric material, capacitive coupling can be reduced. In addition, RF absorbing material can be put in its place to further reducing coupling. Preliminary experiments with these methods are also presented later in this thesis.

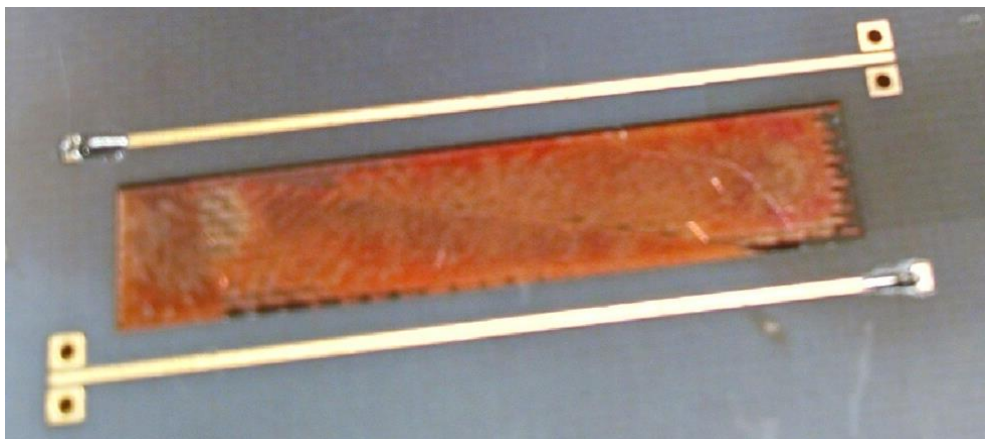


Figure 2: Top dielectric of PCB removed courtesy of the University of Nebraska

1.3 – Prior Work

Even though the mechanisms of crosstalk are well understood, crosstalk continues to be an important area of research. Crosstalk is defined as the amount of signal from an aggressor line that is received by a victim line. In general, crosstalk is divided into near-end crosstalk (NEXT) and far-end crosstalk (FEXT).

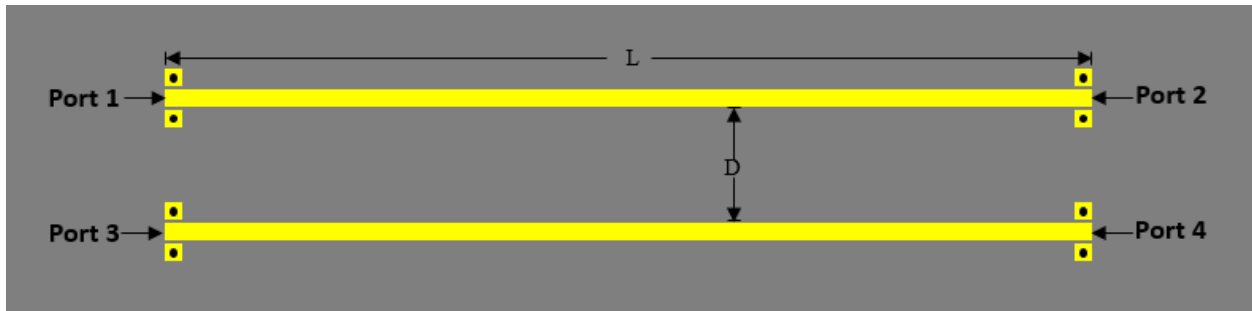


Figure 3: Example of crosstalk test structure

In Figure 3, NEXT can be define as the signal from Port 1 coupling over to Port 3 and FEXT can be define as signal from port 1 coupling over to port 4. During FEXT measurements, port 2 and port 3 were terminated in 50Ω . Since NEXT can only be reduced by increasing the distance between the two lines (D), it is not often studied [3]. Unlike NEXT, several factors can influence FEXT and therefore it is the primary focus of this research.

Further work has been done to characterize the isolation effectiveness of commonly used transmission lines on typical low cost FR4 boards. Specifically, microstrip, co-planar waveguide (CPW), and stripline transmission lines have been tested in [1]. It was found that for every doubling of distance between transmission lines isolation generally increased by 10 to 12dB. When comparing the traditional types of transmission lines it was found that CPW provides 10dB of improvement in insulation over microstrip and stripline provides 20dB of improvement over CPW [1,4].

Due to the widespread popularity of co-planar waveguide, extensive research has been done on optimal design practices. When compared to traditional microstrip, CPW is much more effective at reducing crosstalk [5,6]. Previous research has found that co-planar waveguide is most effective when the signal to ground flank distance is kept to a minimum and via spacing is less than a quarter wavelength at the highest frequency [7]. A variant of CPW known as asymmetric CPW has also been modeled. Asymmetric CPW is a CPW structure in which ground flanks are not spaced equally from the center signal trace [8]. While studied, no practical applications for asymmetric CPW have been found. However, the study of these CPW and its variations led to the creation of a new type of transmission line structure known as single-sided CPW that is studied in chapter 4 of this thesis.

Extensive study has also been done on RF shielding. While it may be intuitive to place an RF shield around a circuit to reduce interference and crosstalk, methods to characterize the RF shielding effectiveness have been studied [9]. In connection to RF shielding, work has also been done to both characterize the effect of and provide guidance on shielding apertures [10]. If incorrectly designed, an aperture can allow signal to escape the shielded structure and interfere with other circuits. It has been shown that a single aperture can reduce shielding effectiveness to 0dB.

1.4 – Thesis Structure

This thesis is structured into seven chapters. Chapter one has provided a brief introduction into electromagnetic interference and crosstalk in regards to PCB design and a general review of prior work done on crosstalk interference. Additionally, a brief overview of additive manufacturing and laser ablation as it relates to PCB board isolation has been included. Chapter two discusses the separation between signal and ground plane as it relates to isolation

between transmission line structures. Specifically, microstrip, coplanar-wave guide, differential pair, and coplanar-waveguide differential pair are studied. Chapter three explores the idea of direct launch stripline as well as the isolation effectiveness in comparison to more widely used microstrip line structures. Chapter four presents a variant of traditional coplanar waveguide called single-sided coplanar waveguide (SS-CPW). Unlike traditional coplanar waveguide which has ground flanking both sides of the signal trace, SS-CPW has one ground flank. In chapter five, the concept of ablating a dielectric layer on a PCB board to increase signal isolation is explored in some detail. Both the theory behind it and experimental results are provided. Chapter six describes the idea of utilizing a split shield over a traditional shield to increase isolation. A discussion on shielding aperture is also contained in chapter five. Finally, chapter seven summarizes the results found in the previous chapters as well as highlights important results and findings. Thoughts on future direction and possible areas of additional study conclude chapter seven.

Chapter 2 - Signal and Ground Plane Separation Isolation Effects

2.1 – Background

In general, when designing a PCB it is better to make the ground plane as close to the signal plane as possible. When close together, the signal current and the ground image current are close enough together to cancel the far magnetic field which in turn can increase isolation. However, engineers do not always have control over the distance between the signal and ground planes. For low cost, quick turn PCB, manufacturers determine the board stackup and engineers must be able to adapt and design appropriately. While all structures can be affected by the distance between signal and ground layers, transmission line structures are particularly sensitive to changes in signal and ground plane separation.

When placing a transmission line, engineers must design the line to have a certain characteristic impedance in order to reduce reflections and maintain signal integrity. For many RF designs this characteristic impedance is 50Ω. The characteristic impedance of a lossless transmission line is determined by equation (2.1).

$$Z_0 = \sqrt{\frac{L}{C}} \quad (2.1)$$

In order to arrive at the correct values of L and C, the dimensions of the transmission line must be scaled appropriately for the given PCB board material properties. For a set characteristic impedance, as the distance between the signal and ground planes increase, the width of the signal trace must also increase. All of these factors will also influence crosstalk.

2.2 – Mathematical Model of Crosstalk VS Signal/Ground Separation

For closely spaced lines, crosstalk is caused by inductive and capacitive coupling from an aggressor line to a victim line. Figure 4 depicts the typical lumped component model for an aggressor line coupling to a victim line [3].

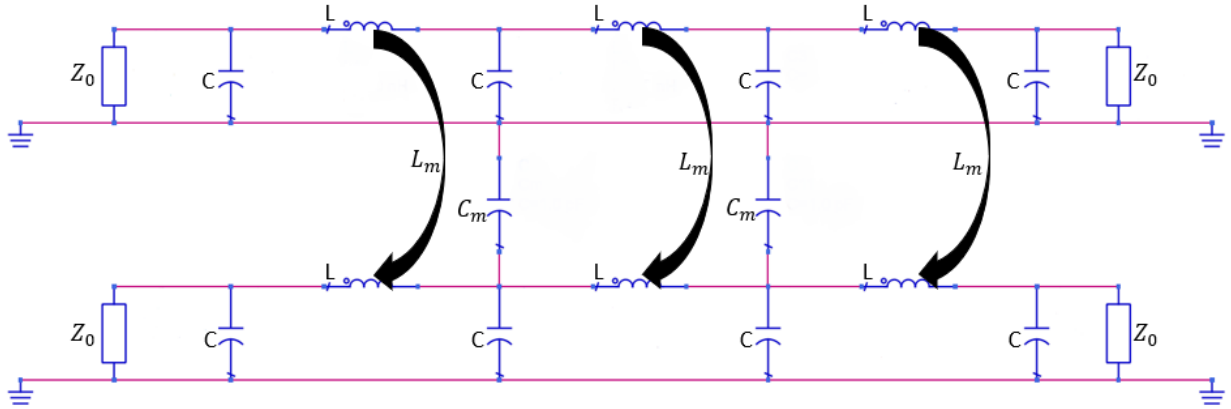


Figure 4: Lumped component model of crosstalk

For the case of inductive coupling, as the distance between signal and ground layers increases coupling becomes worse. Inductance from an aggressor line to a victim line can be defined by the equation for mutual inductance shown in equation (2.2) [11].

$$L_m = \frac{N\Psi}{I_{Aggressor}} \quad (2.2)$$

In equation (2.2) $I_{Aggressor}$ is defined as current through the aggressor line, N as the number of coil turns and Ψ is defined as flux. Flux is defined as the amount of magnetic field that passes through a surface. A formal definition of flux is given by equation (2.3) where B is the magnetic field [11].

$$\Psi = \oint \mathbf{B} \cdot d\mathbf{S} \quad (2.3)$$

Combining equations (2.2) and (2.3) leads to equation (2.4).

$$L_m = \frac{\oint \mathbf{B} \cdot d\mathbf{S}}{I_{Agressor}} \quad (2.4)$$

Figure 5 shows an example of what the magnetic field of a microstrip line might look like. The surface that the flux passes through is defined by the length and signal to ground plane separation of the victim transmission line.

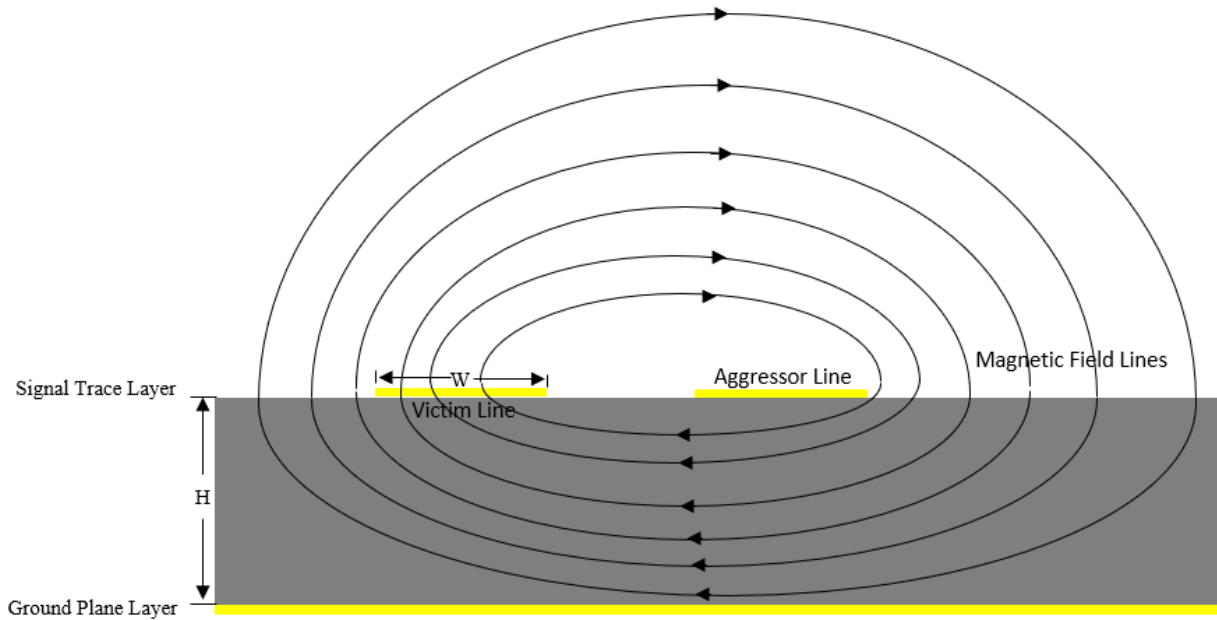


Figure 5: Magnetic fields from aggressor line interfering with victim line

For a victim line of length l and signal to ground plane separation H , equation (2.4) can be approximated by equation (2.5) to the extent the field is uniform. Here, K is a constant that depends on the distance between the lines, and to a lesser degree the height H .

$$L_m = \frac{lH}{I_{Agressor}} K \quad (2.5)$$

Since from equation (2.5) mutual inductance is determined by the length and signal to ground plane separation, mutual inductance will increase as well.

Much like inductive coupling, capacitive coupling will also increase as the distance between the signal and ground layers increases. Fundamentally, capacitance is the result of an electric field existing between two conductors. Shown in Figure 6 is an example of the electric field between aggressor and victim line.

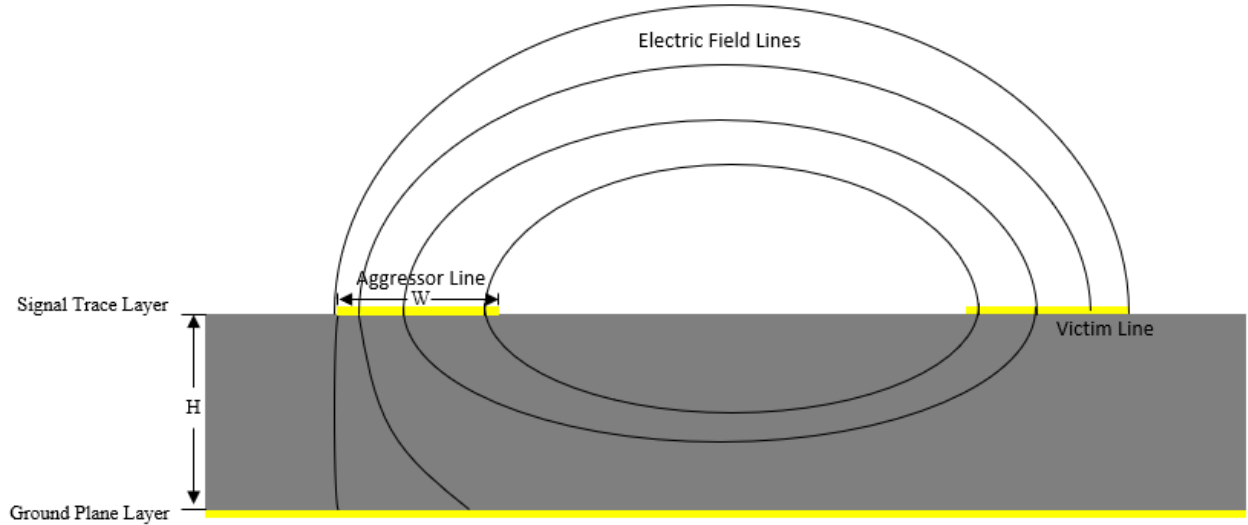


Figure 6: Electric fields between aggressor and victim line when H is large

To a first order, capacitive coupling can be modeled using the equation for a parallel plate capacitor. The capacitance of a parallel plate capacitor in free space is given by equation (2.6) where S is the surface area of the plates and d is the distance between them [11].

$$C = \frac{\epsilon_0 S}{d} \quad (2.6)$$

Depending on the distance between ground and signal plane (H) two capacitive coupling cases develop. When H is small, most of the electric field on the dielectric side will terminate on the ground plane. Given that the surface area is that of the transmission line and that the electric field still forms above the line in free space, the following equation is approximately valid:

$$C \approx \frac{\epsilon_0 l W}{d} \quad (2.7)$$

However, when H is sufficiently large, the electric field in the dielectric will also tend to couple over to the victim line as shown in Figure 6. Considering that half of the electric fields are in the dielectric and the surface area is that of the transmission line, equation (2.6) is modified into the following equation:

$$C \approx \frac{(1+\epsilon_R)\epsilon_0 LW}{d} \quad (2.8)$$

Last, since the distance between the parallel plates is not constant, it will be approximated by taking the arc length from the midpoint of each transmission line. The arc length between two transmission lines with separation S and width W can be approximated by:

$$d \approx \frac{1}{2}\pi(S + W) \quad (2.9)$$

Combining equations (2.7) and (2.9) describes the capacitance between two transmission lines when H is small to a first order. This is shown in the following equation:

$$C_m \approx \frac{2\epsilon_0 LW}{\pi(S+W)} \quad (2.10)$$

Likewise, combining equations (2.8) and (2.9) gives equation (2.11) which describes the capacitance between two transmission lines to a first order when H is large.

$$C_m \approx \frac{2(1+\epsilon_R)\epsilon_0 LW}{\pi(S+W)} \quad (2.11)$$

Since the width of line increases as the distance between signal and ground layers increases and since it sets the surface area of the parallel plates, capacitance will increase as the distance between the signal and ground plane increases. Furthermore, when H is sufficiently large, capacitance will further increase due to the electric field in the dielectric (for small S).

A general equation for far-end crosstalk (FEXT) can be found in [3] where Len is the distance between the two lines, RT is the rise time of the signal, v is the speed of the signal, C_L is the capacitance per length and L_L is the inductance per length of the signal trace.

$$FEXT_{dB} = 20 \log \left(\frac{Len}{RT} * \frac{1}{2v} * \left(\frac{Cm}{C_L} - \frac{Lm}{L_L} \right) \right) \quad (2.12)$$

As seen from equation (2.12), FEXT occurs when there is a dominant amount of either mutual capacitance or mutual inductance. When capacitive coupling dominates, FEXT should theoretically increase by a maximum of between 6-19.5dB for every doubling of distance between signal and ground plane given a dielectric constant of 3.667. Equation (2.12) also suggests that when the mutual capacitive and inductive values achieve the same value, FEXT can be eliminated.

2.3 – Experimental Setup

To validate the theory above and assess more complex geometry, transmission lines of varying type and varying signal to ground distances were fabricated and measured for crosstalk performance. Testing was performed on a single low cost FR4 PCB board shown in Figure 7.

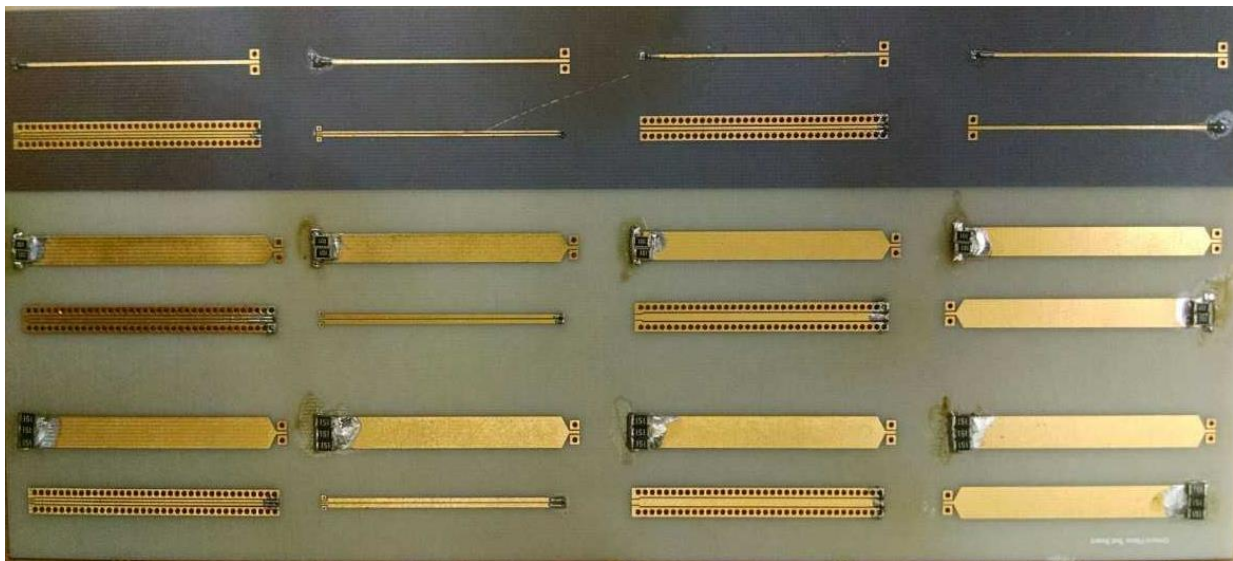


Figure 7: Signal to ground plane test board

Measurements were made using a Keysight N5245A vector network analyzer. To work correctly with the network analyzer, all lines were designed to have a characteristic impedance of 50Ω . Four different transmission line structures tested were: microstrip, CPW, differential pair, and CPW differential pair. Coupling from each transmission line under test and a microstrip line was measured. For transmission line structures without via flanks, vias were added to one end of the transmission line to properly land the $500\mu\text{m}$ ground signal ground (GSG) probes. An example of GSG probes landed on a CPW transmission line is shown in Figure 8. The opposite ends of the transmission lines were terminated in 50Ω to eliminate any reflections on the line.

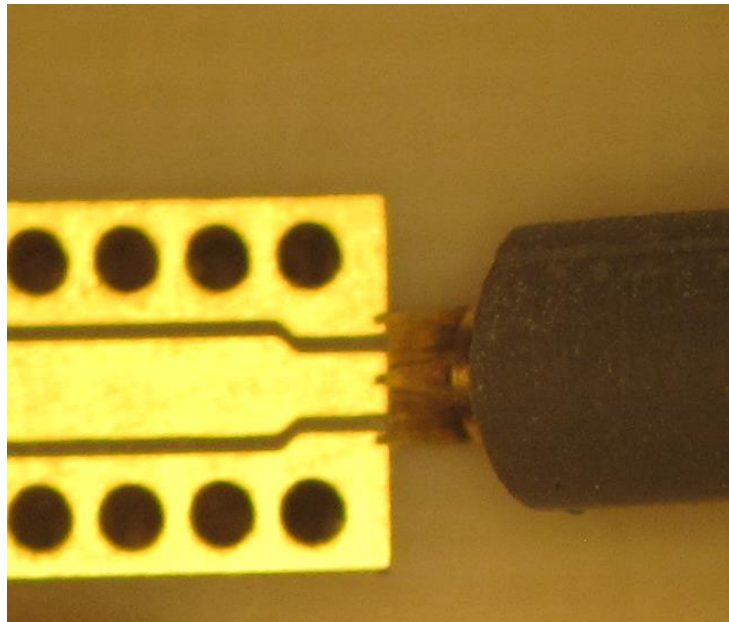


Figure 8: Landed GSG probe on CPW transmission line

An example measurement setup with the Keysight N5245A vector network analyzer and the GSG probes is shown in Figure 9.

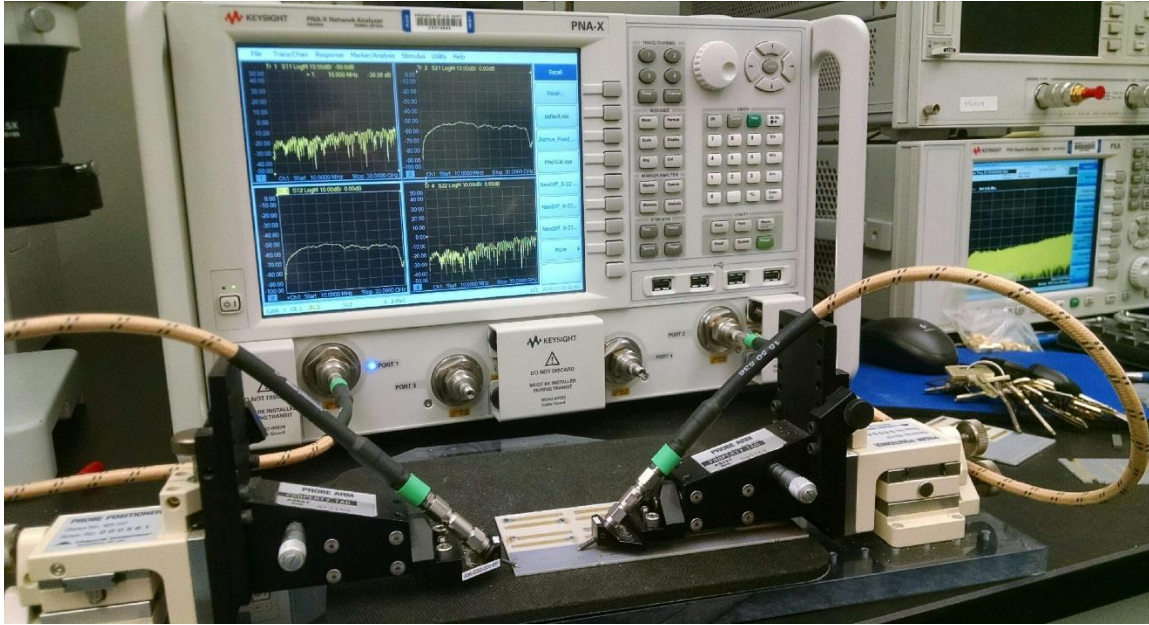


Figure 9: Example measurement setup

For each transmission line tested, measurements were made for ground plane distances of 6.7mils, 53.7mils and 60.4mils from the signal trace. Line length for all cases was 1000mils (2.54cm). Spacing for signal trace to ground flanks for co-planar lines was fixed at 4mils by the FR4 fabrication rules. Table 1 shows the width of each transmission line structure at the 3 different ground distances.

Table 1: Signal trace width in mils

	Microstrip	CPW	Differential Pair	CPW Differential Pair
6.7mil Ground Distance	13.5	12	16	16
53.7mil Ground Distance	120	33	35	20
60.4mil Ground Distance	136	35	36	20

2.4 – Experimental Results

In this section results for each transmission line structure are recorded and analyzed separately. For a comparison between different transmission line structures please see section 2.5.

2.4a – Microstrip Experimental Results

Figure 10 depicts the measured far end crosstalk (FEXT) from microstrip to microstrip at three separate ground distances.

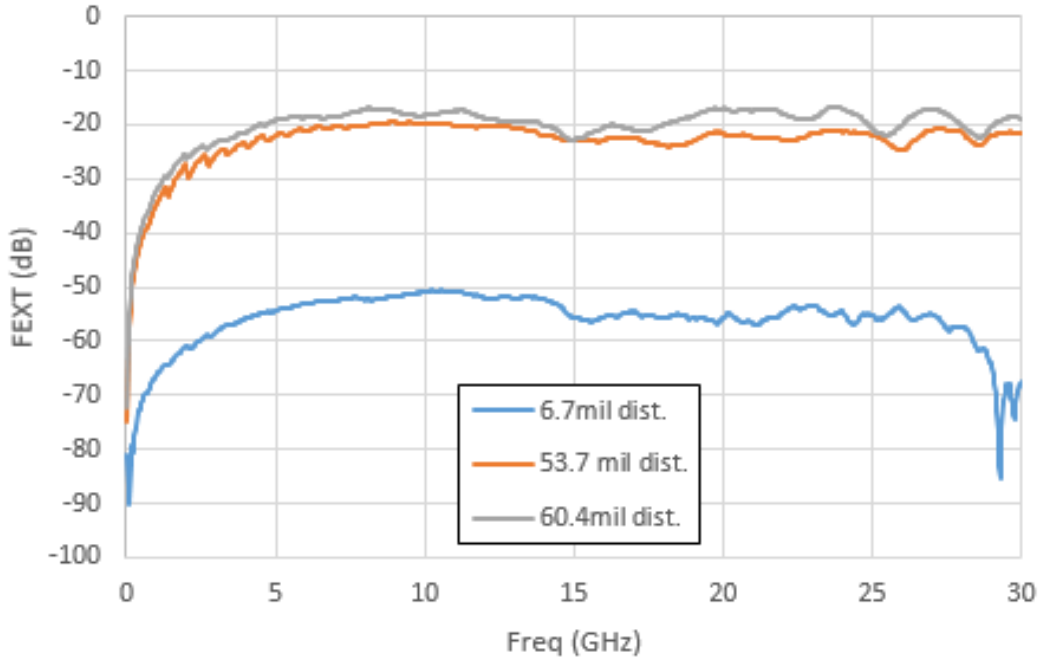


Figure 10: FEXT of microstrip to microstrip at varying ground distances

When the distance between the signal and ground plane is increased from 6.7mils to 60.4mils isolation decreases by approximately 40dB. This translate to a decrease in isolation of 8dB for every doubling of distance between signal and ground plane. It appears from these results that capacitive coupling dominates.

2.4b – CPW Experimental Results

Figure 11 depicts FEXT from CPW to microstrip at three separate ground distances. Due to the ground flanks, CPW should eliminate most of the capacitive coupling since the electric field of the aggressor line will terminate on the ground flanks instead of propagating to the victim line.

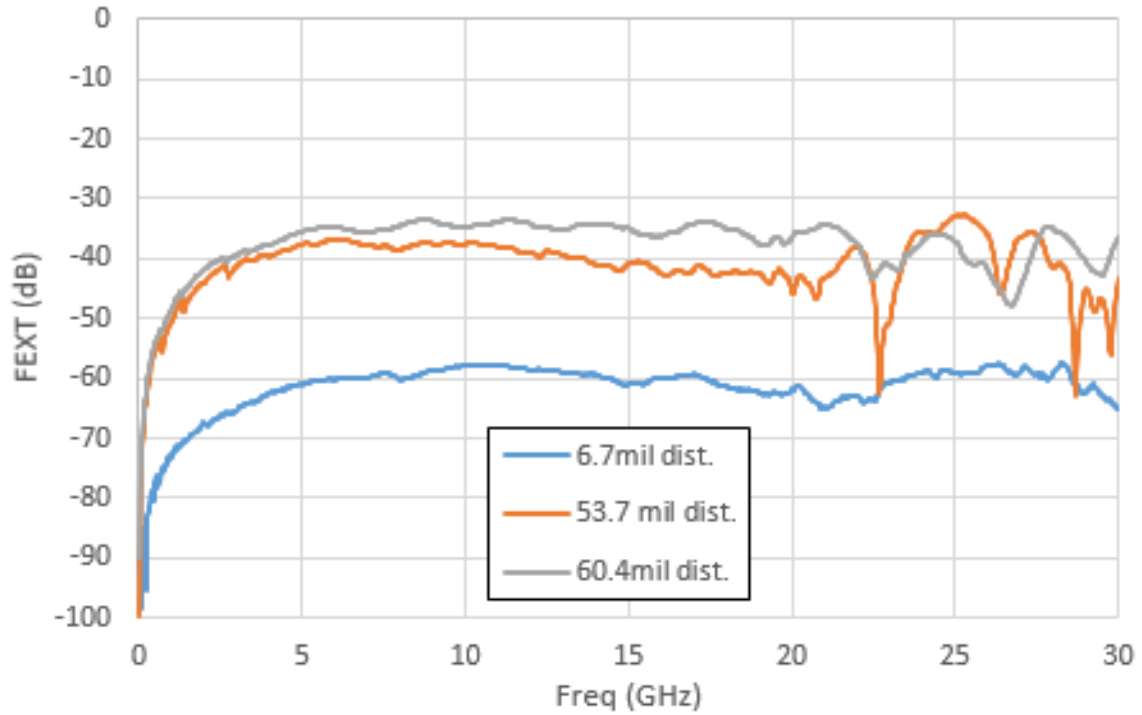


Figure 11: FEXT of CPW to microstrip at varying ground distances

As the distance between the signal and ground plane is increased from 6.7mils to 60.4mils isolation decreases by approximately 25dB.

2.4c – Differential Pair Experimental Results

Figure 12 depicts FEXT from differential pair to microstrip at three separate ground distances.

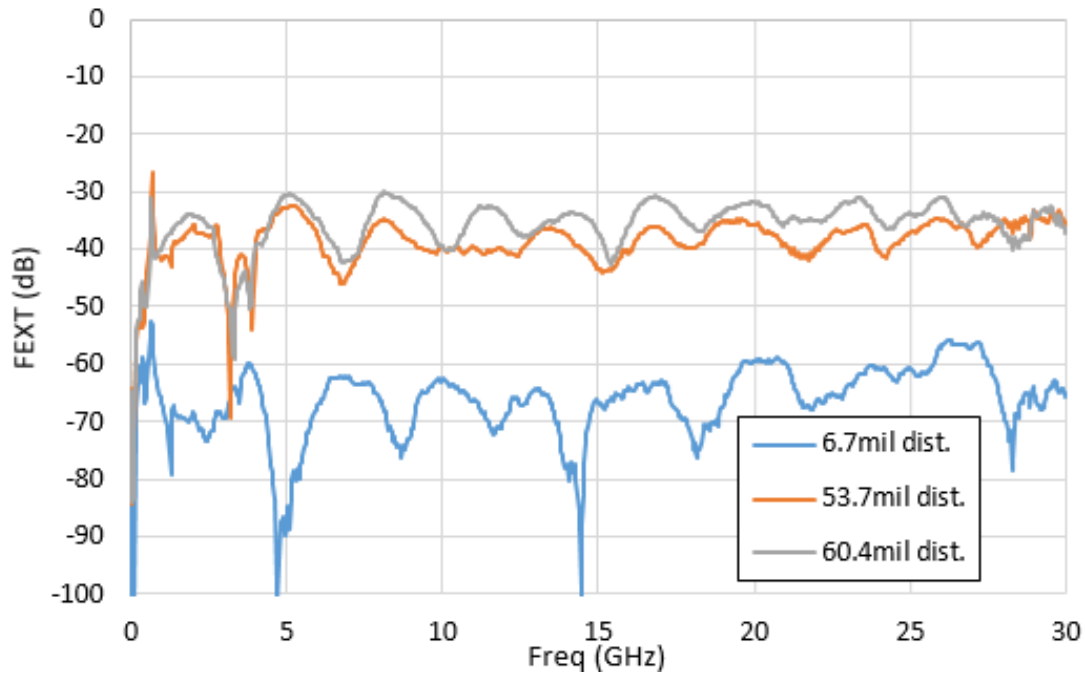


Figure 12: FEXT of differential pair to microstrip at varying ground distances

As the distance between the signal and ground plane is increased from 6.7mils to 60.4mils isolation decreases by approximately 20-25dB.

2.4d – CPW Differential Pair Experimental Results

Figure 13 depicts FEXT from CPW differential pair to microstrip at three separate ground distances. With the inclusion of the ground flanks, the electric field that is not canceled as a result of the differential pair will terminate on the ground flanks.

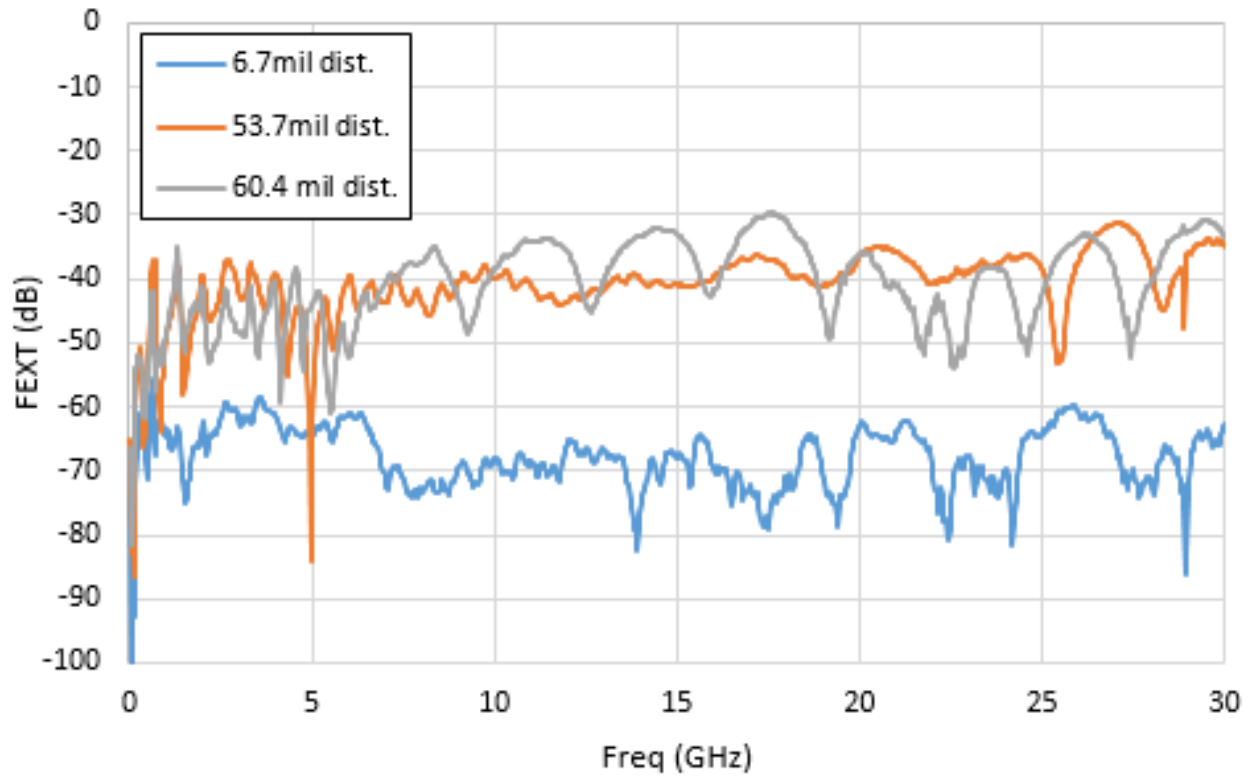


Figure 13: CPW differential pair to microstrip at varying ground distances

As the distance between the signal and ground plane is increased from 6.7mils to 60.4mils isolation decreases by approximately 15-25dB. When compared to traditional differential pair, the ground flanks if the CPW differential pair provided a minimum increase in isolation.

2.5 – Comparison of Different Transmission Lines and Conclusion

Figures 14 and 15 depict a comparison of the different transmission line structures with a ground to signal plane separation of 6.7 and 60.4mils.

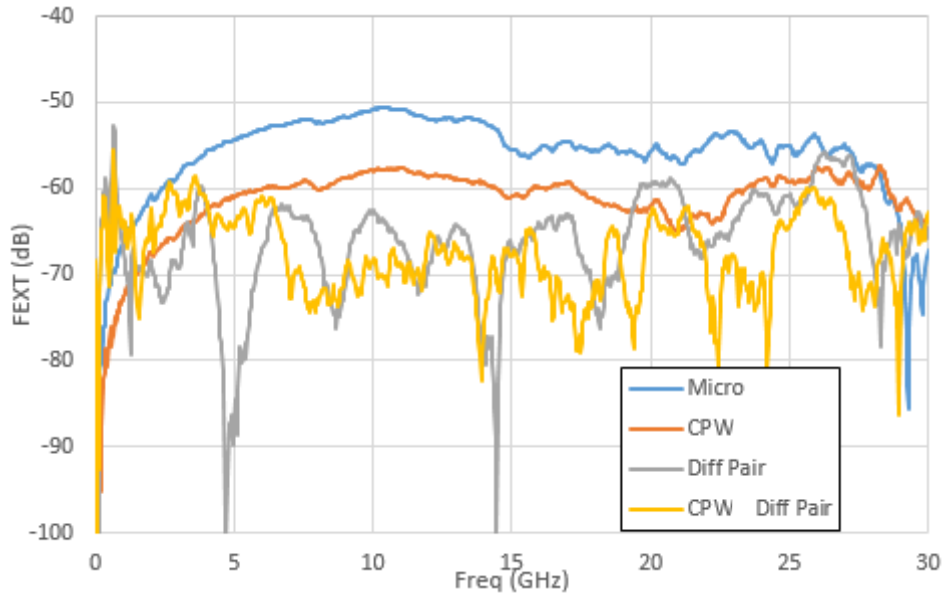


Figure 14: Comparison of FEXT for transmission lines with ground 6.7mils below

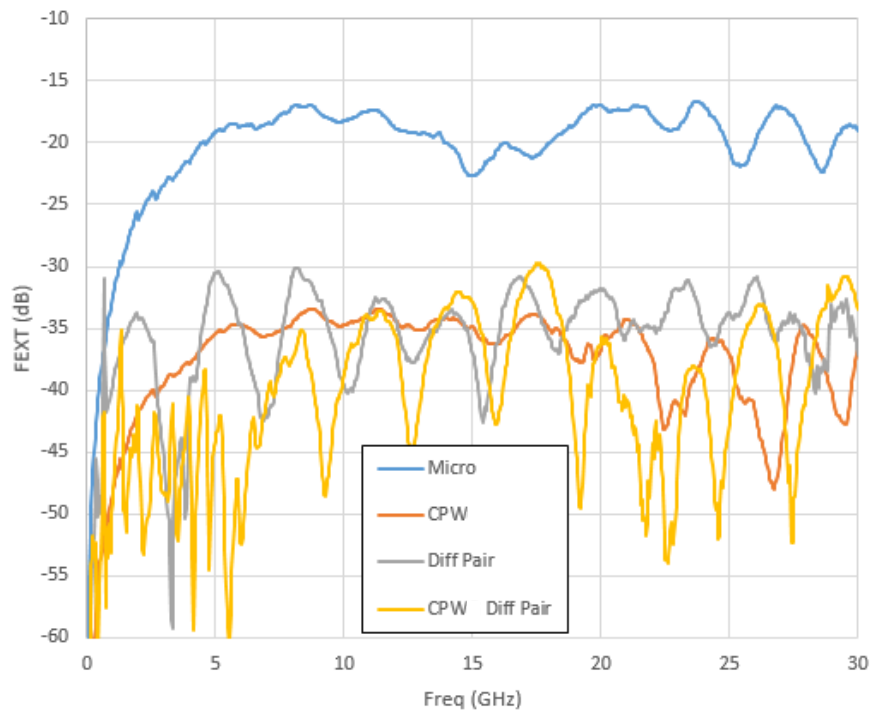


Figure 15: Comparison of FEXT for transmission lines with ground 60.4mils below

When the distance between the signal and ground plane is kept to a minimum, CPW differential and differential pair provide superior isolation. As the distance between signal and ground plane

increases CPW provides relatively the same amount of isolation as CPW differential pair and differential pair. For all cases microstrip provides the least amount of isolation.

When Figures 14 and 15 are compared another trend begins to emerge. Transmission line structures that minimize the width provide greater isolation. With microstrip, the signal trace width increases from 13.5 to 136mils and the isolation decreases by approximately 40dB. Due to the ground flanks, CPW reduces the increase of the width of the signal trace to only 35mils. Due to the reduced width, isolation only decreases by 25dB.

As expected, the closer together the ground and signal planes are placed the higher isolation becomes. This holds true regardless of the transmission line structure. However, when a large distance between the signal and ground plane cannot be avoided, transmission line structures that minimize the increase in signal trace width provide better isolation. Finally, adding ground flanks to a traditional differential pair provides only a minimum increase in isolation.

Chapter 3 - Direct Launch Stripline and Signal Isolation

3.1 – Introduction to Stripline

For circuit designs with higher isolation requirements than traditional CPW or differential pair transmission lines can provide, designs may incorporate stripline transmission line structures. Unlike microstrip, CPW, and differential pair which have the signal trace on a top or bottom board layer, stripline buries the signal trace in between two ground planes. In general, stripline may refer to several types of transmission line structures with a signal trace suspended between two ground planes. General stripline structures have found uses in both digital and RF circuit designs.

While the two ground planes do not necessarily need to be stitched together with vias for a transmission line to be considered stripline, via walls are recommended in order to increase isolation. In previous studies via stitched stripline has also been referred to as walled stripline, rectangular stripline and rectax [12,13]. These studies have also demonstrated to some extent the isolation benefits of using walled stripline. Figure 16 provides both a top down and cross-sectional view of a general walled stripline structure with via walls.

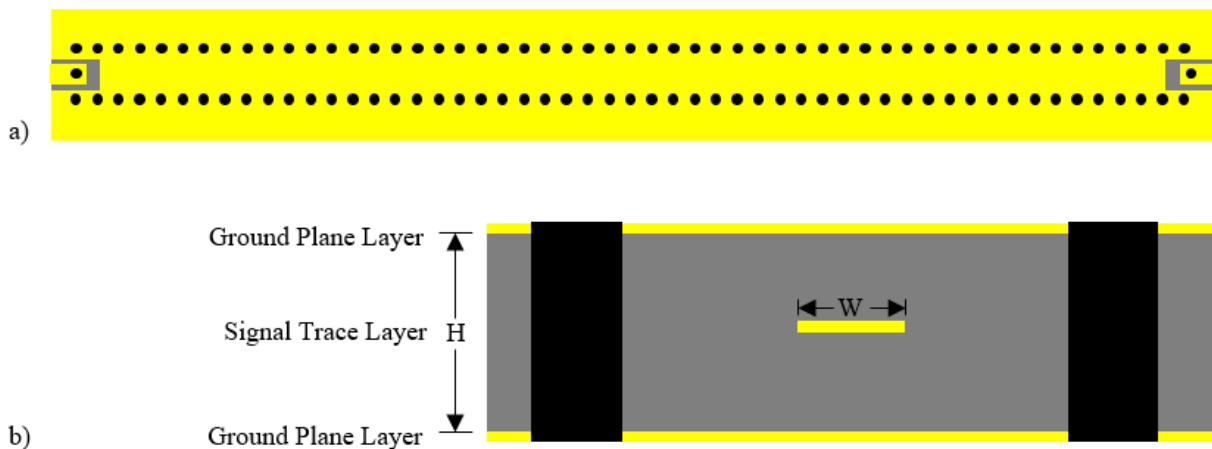


Figure 16: General design of walled stripline transmission line

In general, the characteristic impedance of stripline is determined to a first order by the line width and distance away from the two ground planes that the signal trace is. To increase isolation, via walls may be added as shown in Figure 16. To prevent a resonance from occurring when utilizing via walls, the vias should be spaced less than a $\frac{1}{4}$ wavelength of the highest frequency. Traditionally, to connect components mounted on the topside or backside of a PCB to the stripline structure, a via transition must be used to allow access from the internal signal trace to either the front or back surface layers.

It has been shown in previous work that stripline can increase isolation by as much as 20dB over traditional CPW [4]. While stripline provides superior isolation, it can introduce several issues as well. As alluded to earlier, one prevalent issue with stripline has to do with the via transition. At the via transition, the signal can radiate and isolation can be lost. In addition to the isolation issue, the transition from top layer to the internal signal layer can result in a substantial impedance discontinuity. This impedance discontinuity can result in the signal being reflected back to the source at certain frequencies. To correct these issues, direct launch stripline was studied in this work.

3.2 – Stripline VS Direct Launch Stripline

As mentioned previously, the via transition from the top board layer to the inner signal layer of stripline can cause signal dropout due to the impedance discontinuity and radiation losses. Previous studies have shown that in order to prevent strong reflections, the via stub and transition geometry should be kept to less than $\frac{1}{20}$ of the wavelength of the highest frequency. To avoid a complete dropout, the via stub must be kept to less than $\frac{1}{4}$ of the wavelength of the highest frequency [1]. A significant amount of research has gone into exploring ways improve the via transition from microstrip and CPW to stripline [14, 15]. However, none of these

techniques have been entirely successful. Here we have explored removing the via transition all together.

Direct launch stripline explores the technique of completely removing the via transition and sinking components onto the same layer as the signal trace. Once sunken into the board, the exposed layers can be shielded to further increase isolation. An example of this is shown in Figure 17. In order to expose the signal layer, laser ablation was done on the board by Dr. Yongfeng Lu and Timothy Carlson of UNL.

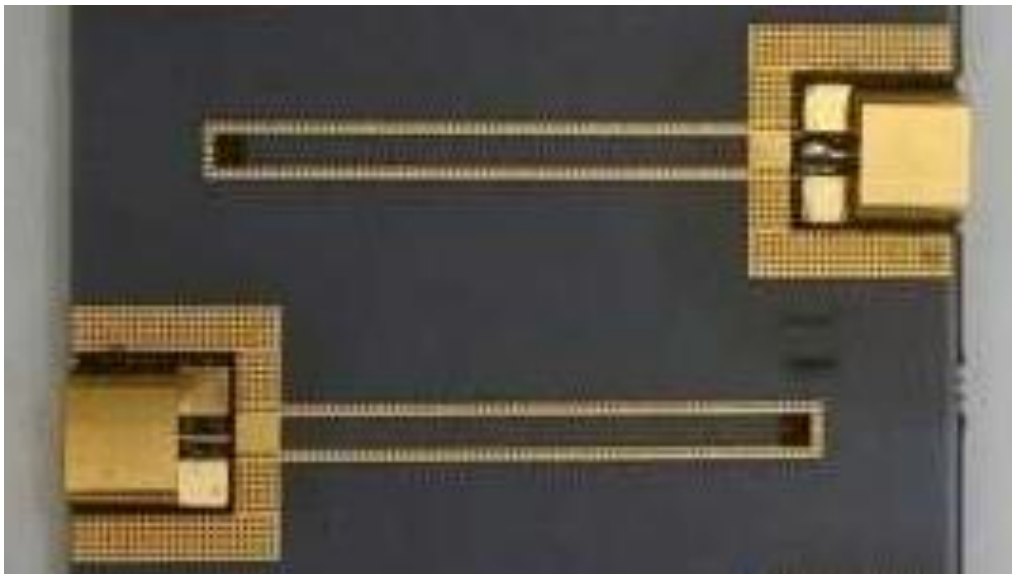


Figure 17: Direct launch stripline with connectors sunken into board

While for research purposes, a laser was used to remove the top dielectric and conductor layers to allow for direct launch into the stripline's signal trace, in the future technologies and the ability to 3D print circuit boards may be incorporated. Once components have been soldered into the board, the top layer can be shielded with 3d printed or applied covers to increase isolation. An example of this shielding is shown in Figure 18.

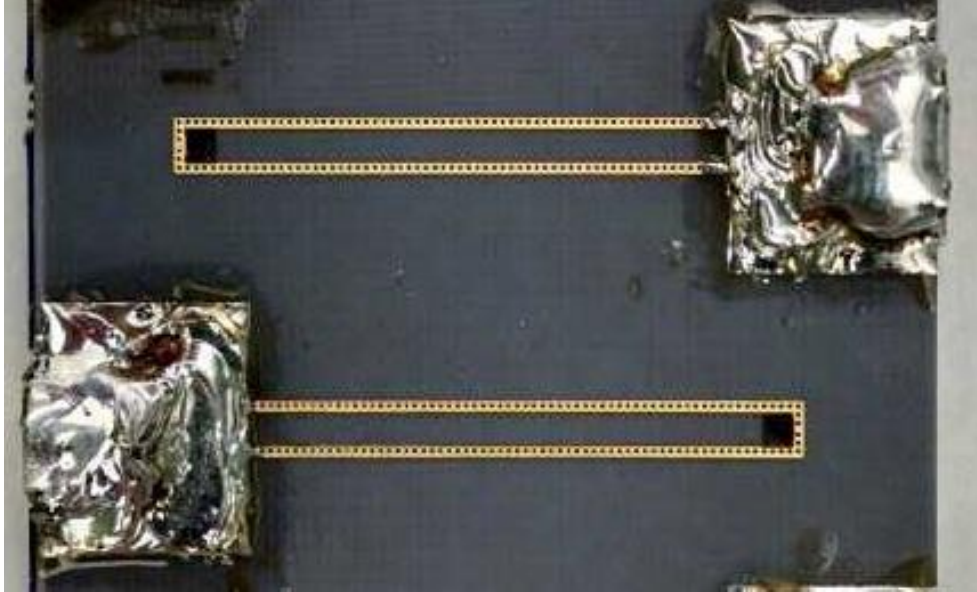


Figure 18: Direct launch stripline with connectors and shielding

In Figure 18, once the connectors were soldered to the board, a shield was constructed around them to fully enclose them and prevent the signal from radiating. Without this full-enclosure shielding, significant signal leakage resulted thru the aperture [16].

3.3 – Experimental Setup

In order to test both direct launch and traditionally launched stripline, two test boards were constructed. For consistency, both boards were designed on low cost FR4 with the same stack up and from the same manufacturer. Figure 19 depicts the test board used to test traditionally launched stripline.

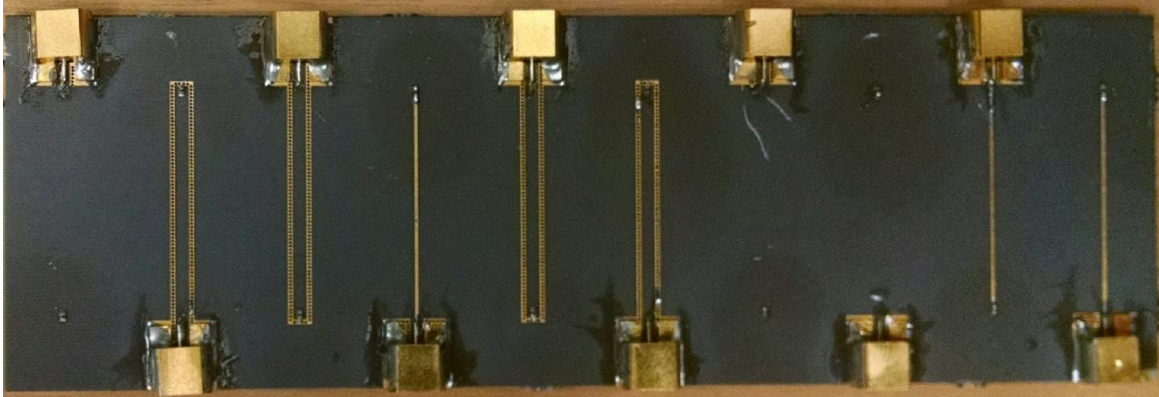


Figure 19: Traditional stripline test board

Figure 20 depicts the test board used to test direct launch stripline. In order to sink the components into the board, a laser was used to ablate the top layers. Once the components were soldered into place, a shield was placed over the connectors to eliminate apertures that degrade shielding effectiveness.

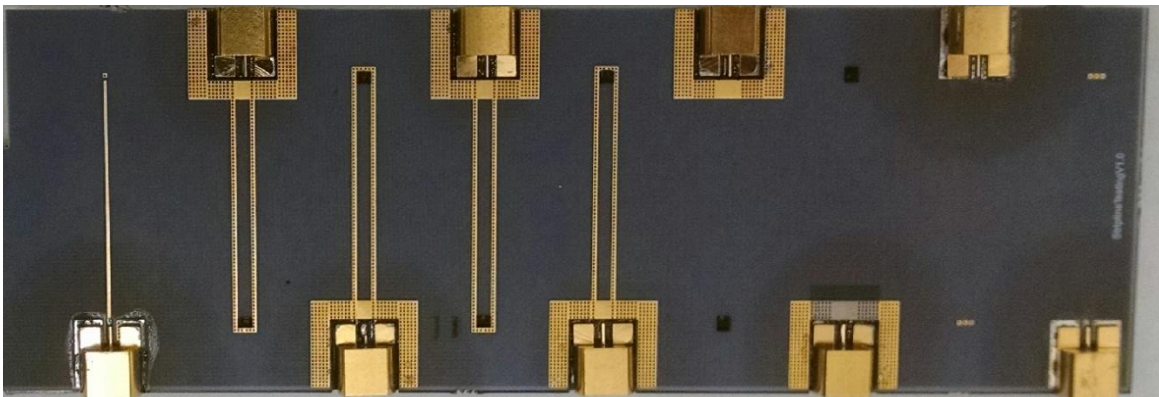


Figure 20: Direct launch stripline test board prior to adding top shielding

Figure 21 depicts the difference between the traditional launch of stripline used on the test board depicted in Figure 19 and the direct launch of stripline used on the test board depicted in Figure 20. Traditional stripline launch requires that the connector be mounted on the top layer and a via used to access the signal trace on the internal layer to the center signal pin of the connector. Direct launch allows for the connector to be placed on the same layer as the signal

trace and no via is required to connect the internal signal layer to the connector's signal pin. Furthermore, a via wall is enclosed the connector so that the component can be completely shielded.

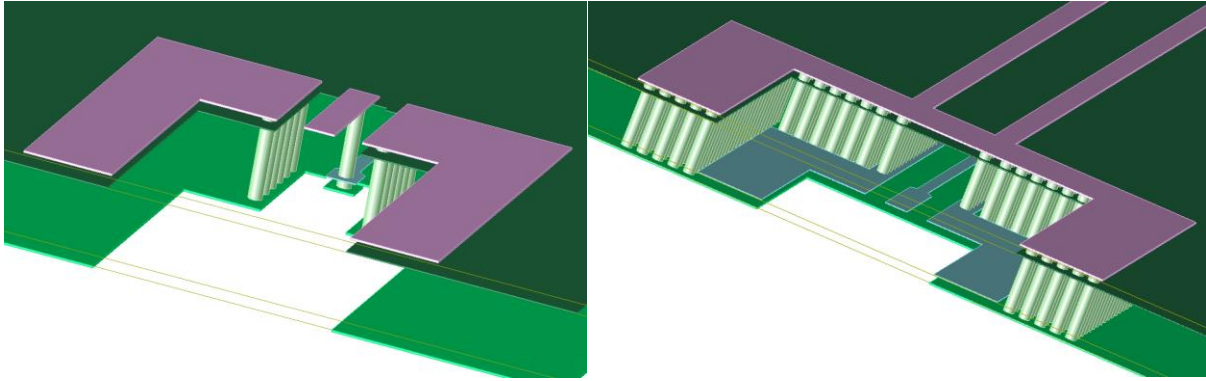


Figure 21: Closeup of traditional vs direct launch

3.4 – Experimental Results

All measurements were made using a Keysight N5245A vector network analyzer. Instead of GSG probes, connectors were used to probe all structures. All transmission line structures were designed to be 1000mils long as measured from the end of the connector.

Experimental results are divided into two main sections. In section 3.4a, results are presented for the traditional stripline launch case. Section 3.4a also explores isolation when a mixture of stripline and other transmission line structures such as microstrip are used. In section 3.4b, results are presented for direct launch stripline. Results for partial shielding, and complete shielding of direct launch stripline are also presented in this section. Due to the elimination of the via transition, it is expected that the direct launch stripline should substantially improve isolation when compared to traditional stripline.

3.4a – Traditional Launch Stripline with Unshielded Transitions

Figure 22 depicts coupling from different stripline structures to microstrip. Three different types of stripline were tested: stripline without via walls, stripline with via walls, and stripline with via walls and flanks. A flank refers to the rectangular conductor surrounding the ground stitching vias. This flank is similar in design to the flank structures of CPW but it is internal to the board. Overall, stripline without via walls performed the worst while stripline with via fence and flank on the signal layer performed the best. Without any via fencing, stripline performed slightly worse than even traditional microstrip. However, when via walls were utilized, stripline outperformed microstrip on average by 10dB.

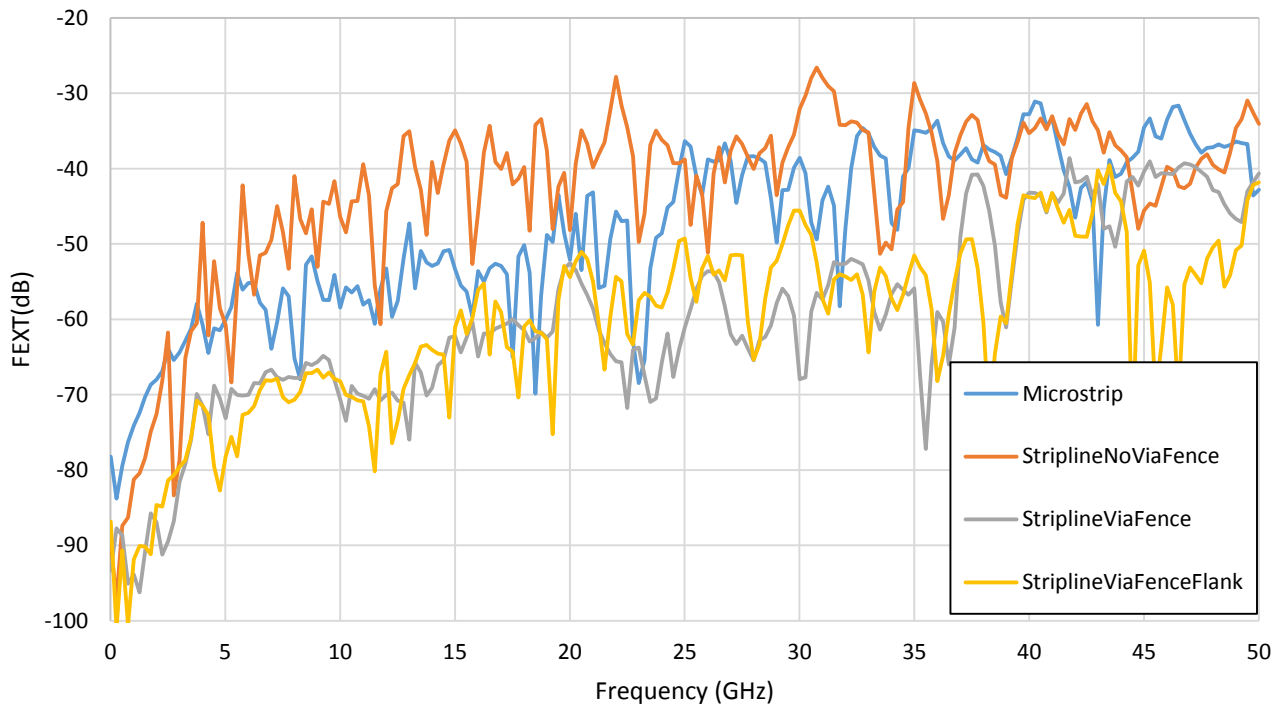


Figure 22: Coupling of traditionally launched stripline configurations to microstrip

Unlike the previous figure, Figure 23 depicts FEXT between different pairs of stripline structures to stripline structures. Once again, stripline without any via fencing performed the

worst, averaging only about 35dB of isolation above 10GHz. The poor performance in isolation can be attributed to the increase in capacitive coupling. The addition of a flank on the signal layer had almost no effect on isolation as stripline with via walls and stripline with via walls and flank on the signal layer performed identically. When via walls were utilized, isolation improved by approximately 30dB over non via walled stripline.

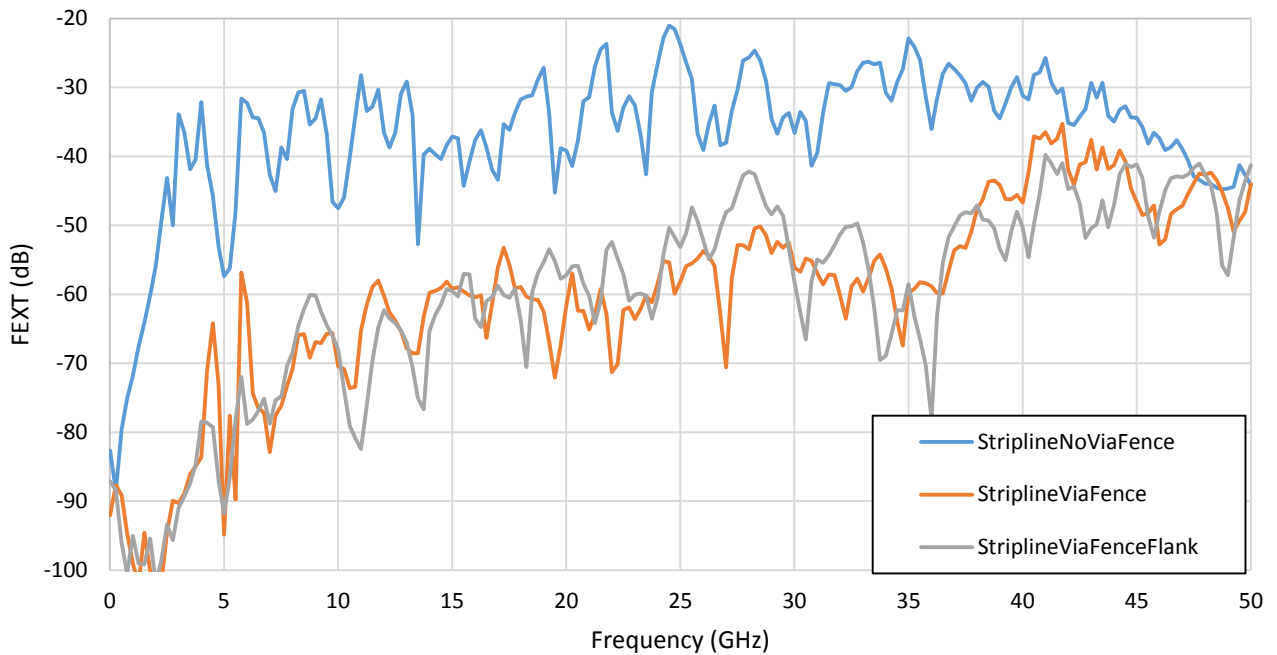


Figure 23: Coupling of traditionally launched stripline to stripline configurations

3.4b – Direct Launch Stripline Results

In section 3.4b, results are presented for direct direct launch stripline. In addition, results are presented for various forms of shielded direct launch stripline. Shown in Figure 24 are the FEXT measurements for stripline to stripline when a via fence was not used on either of the stripline transmission lines. Baseline refers to direct launch stripline without any shielding or absorber material added. Without the via fence, the signal from the aggressor was still allowed to propagate through the dielectric material to the victim line. This was made evident by the fact

that when the connector launch point was completely shielded, the isolation did not increase. The isolation achieved without fully enclosed stripline is no better than CPW.

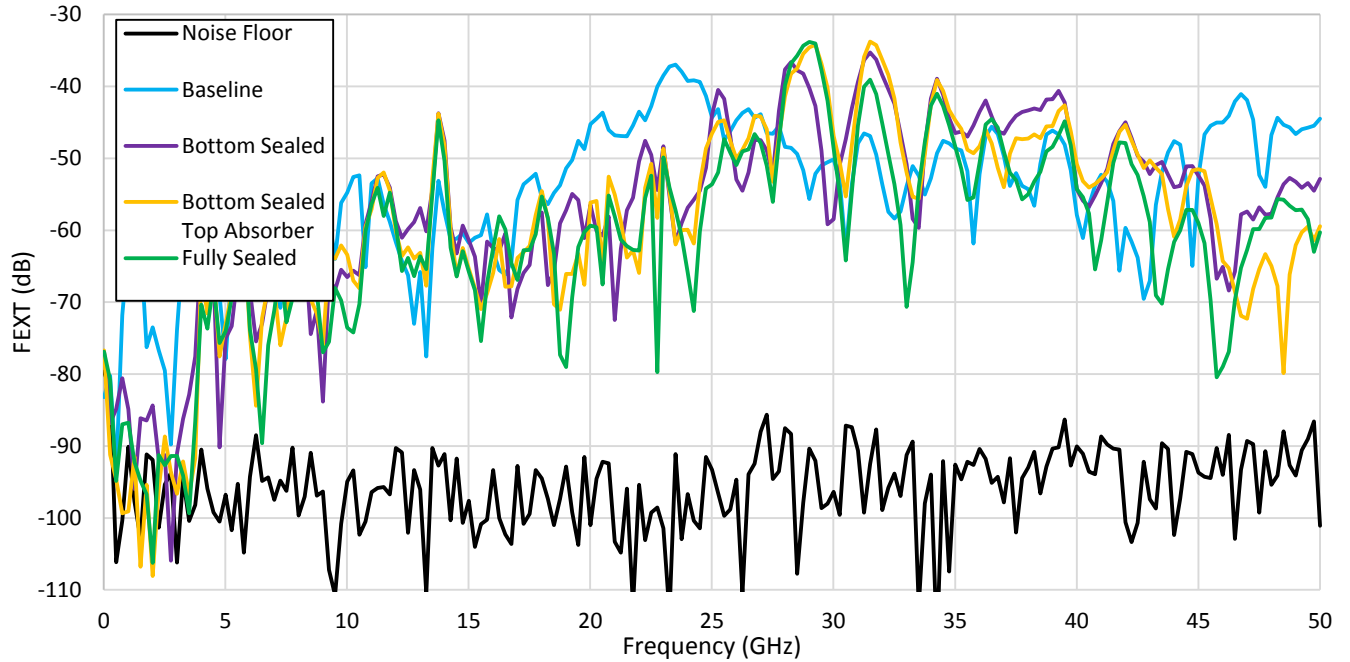


Figure 24: FEXT stripline to stripline no via fencing

Figure 25 presents results for FEXT between two stripline transmission lines with via walls surrounding both. In contrast to Figure 24, when fully shielded, stripline greatly increases isolation over all top layer transmission lines. When fully shielded, direct launch stripline can achieve isolation approaching that of the noise floor (>90dB).

When full shielding cannot be achieved, the addition of RF absorber material can result in isolation on average greater than 80dB. However, when care is not taken to partially or fully shield stripline, isolation on average of only 65dB can be achieved. Overall, when compared to stripline without via walls, the average isolation is increased by up to 30 or 40dB when via walls are added.

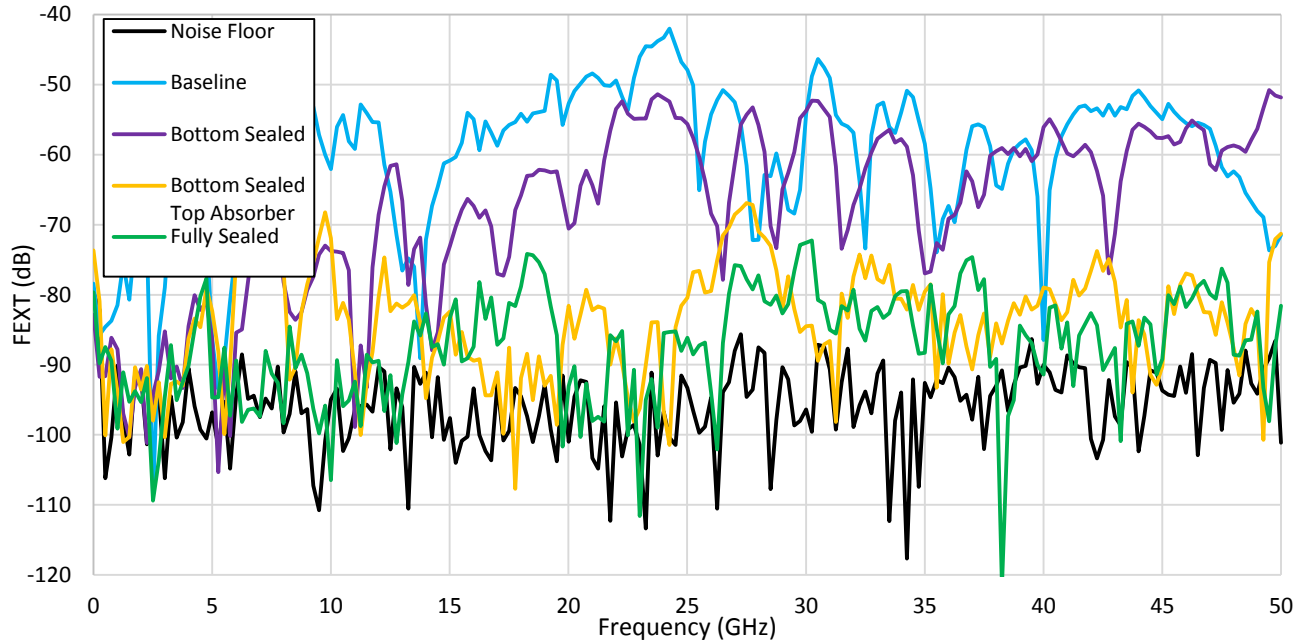


Figure 25: FEXT for direct launch stripline to stripline with via fencing

Figure 26 depicts the FEXT of stripline to stripline when both via fences and flanks on the signal layer are utilized. Overall, when compared to the previous results the addition of a flank on the signal layer did not appreciably improve isolation. At most, the addition of the flank increased isolation of stripline by 5dB. Once again, when fully sealed isolation approached that of the noise floor.

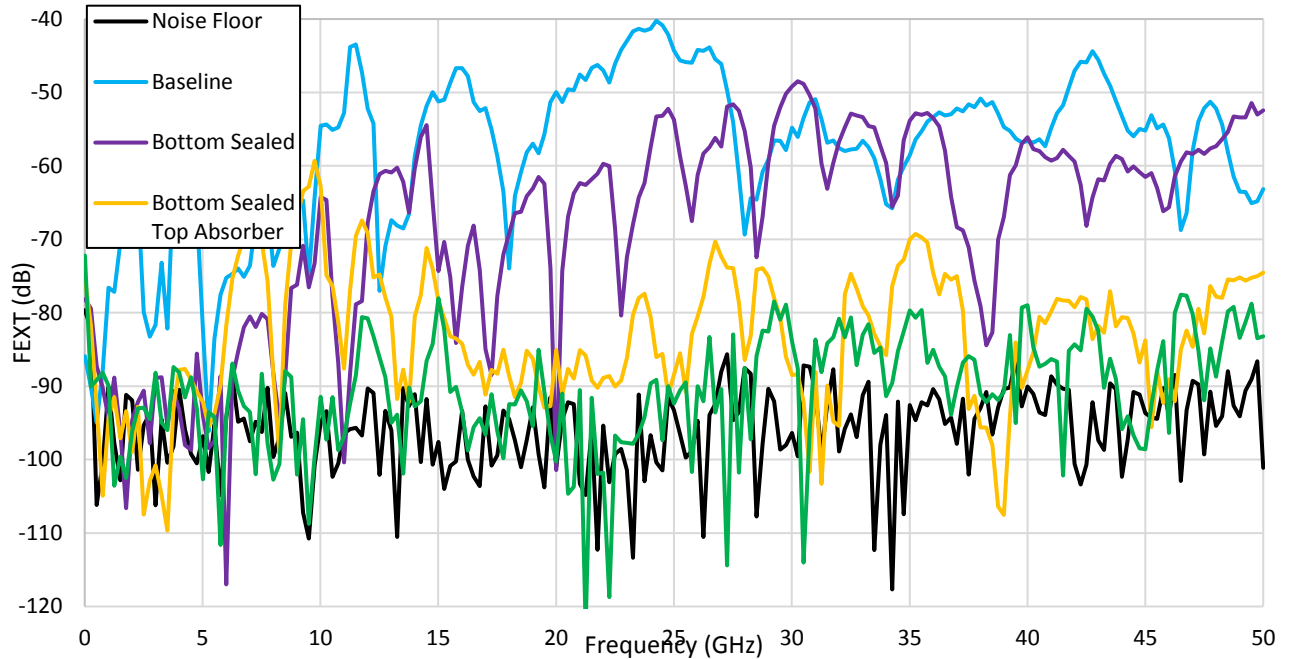


Figure 26: FEXT stripline to stripline with via fence and flanks

3.5 – Comparison of Stripline and Direct Launch Stripline

A comparison of direct launch stripline and traditional stripline illustrated the issue that via transitions cause in terms of isolation. For cases when via walls are used, elimination of via transitions can improve isolation by as much as 30dB. Additionally, the use of direct launch stripline allows for the ability to completely shield structures. When structures are completely shielded, isolation can approach that of noise floor.

3.6 – Conclusion

For stripline to effectively increase isolation, via transitions must be dealt with appropriately. When the transitions are eliminated, isolation can improve by as much as 30dB. Furthermore, direct launch stripline allows for complete shielding of components for even greater isolation. When direct launch stripline is fully shielded, isolation approaching that of the noise floor can be achieved. Finally, it has been shown that via wall are extremely important for

stripline to be effective. Adding via walls can improve isolation by as much as 40dB when compared to stripline without via walls.

Chapter 4 - Single Sided Coplanar Waveguide (SS-CPW)

4.1 – Background

A widely known and used technique to reduce crosstalk in a circuit is to replace microstrip lines with ground backed coplanar waveguide lines (CPW or GB-CPW). An example of a CPW transmission line is shown in Figure 27. A CPW line consists of a signal trace flanked on both sides by grounds with a ground plane underneath. In typical board designs, the top and bottom surface grounds are “stitched” together with a series of vias. CPW reduces crosstalk significantly by terminating the electric fields from the aggressor line on the ground flanks instead of on the signal trace of the victim line.

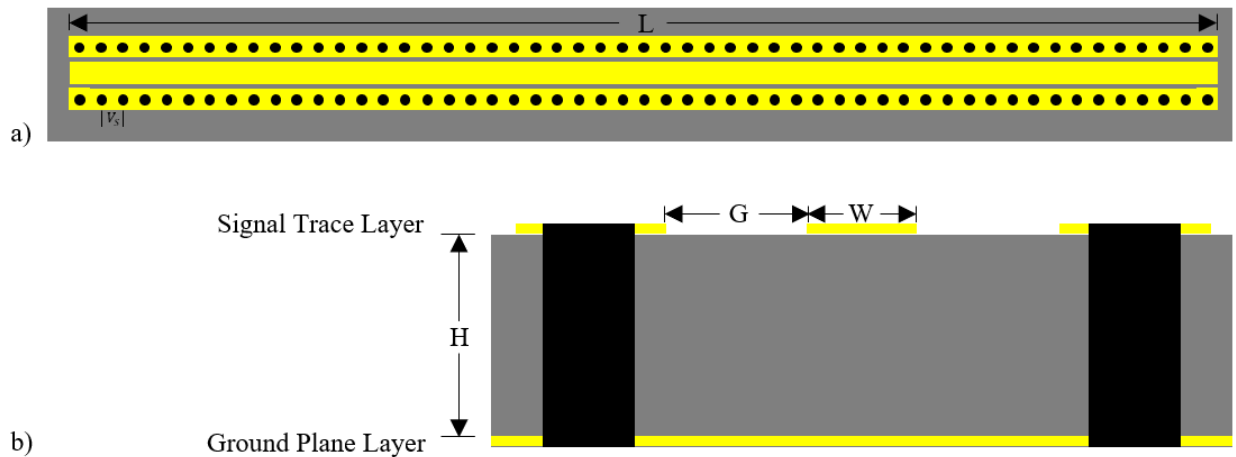


Figure 27: Traditional design of a CPW transmission line

With the addition of ground flanks, it has been shown that CPW can increase isolation by as much as 10dB when compared to microstrip [4]. While CPW does increase isolation, certain considerations must be taken when using this typical line. Due to requiring two ground flanks, CPW can increase the overall size of the design as well as complicate the addition of filters, stubs and other complex designs. Furthermore, unlike microstrip, CPW can prevent the easy

modification of a circuit once it has been manufactured. For these reasons, research continues to be done on the improvement of CPW [7].

In addition to CPW, a variation of CPW known as asymmetric CPW has also been the subject of study [8]. Much like CPW, asymmetric CPW consists of a signal trace and two ground flanks with a ground plane underneath. Unlike CPW in which the ground flanks are placed equal distance away from the signal trace, asymmetric CPW places the signal trace closer to one of the ground flanks. Benefits from this configuration have not been found in the literature.

Unlike asymmetric CPW, SS-CPW (developed in the course of this research [11]) is a variation of CPW that seeks to provide more design flexibility while still maintaining the ability to provide superior isolation benefits when compared to traditional microstrip. Unlike CPW, SS-CPW utilizes only one ground flank to increase isolation. The inclusion of the single ground flank allows the electric field to terminate on the ground flank and reduce the crosstalk between transmission line structures by reducing capacitive coupling. By using only one ground flank however, board size is kept to a minimum and more complicated structures such as stubs and vias can be added to the transmission line. A general design guide for SS-CPW can be found in the following section.

4.2 – General Design of SS-CPW

Figure 28 shows both a top down and cross-sectional view of a single-sided CPW transmission line. As the name implies, a SS-CPW transmission line consists of a signal trace and ground flank with a ground plane underneath. Again, much like CPW, the ground flank is connected to the underlying ground plane through the use of vias.

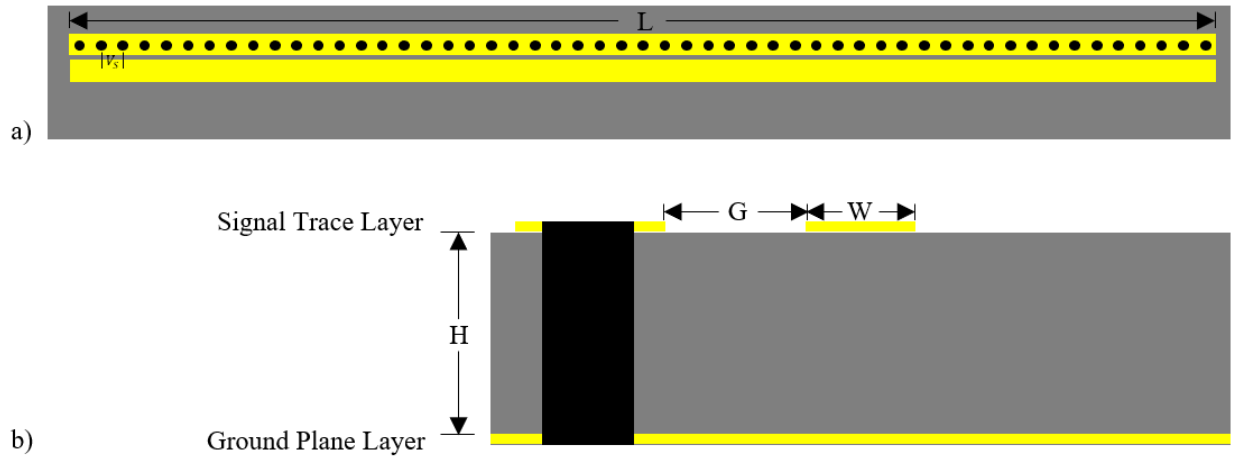


Figure 28: General design of SS-CPW transmission line (from [17])

Much like with traditional CPW, to prevent resonances from occurring in the circuit, the spacing between vias (V_S) should be kept to less than $\frac{1}{4}$ of the wavelength of the highest frequency [7]. In order to maximize the isolation effect of the ground flank, the distance between the signal trace and ground flank (G) should be kept to a minimum. Additionally, it is important to place the ground flank of SS-CPW on the side of the transmission line facing the victim to benefit from the increase in isolation.

While no concise formula currently exists to determine the line width (W) needed for a given characteristic impedance, a traditional CPW transmission can be used as a guide to estimate the correct dimensions needed. For a traditional CPW line, the characteristic impedance is largely determined by the separation of the center signal trace and the two ground flanks. When designing a SS-CPW line with the same separation between signal trace and ground flank G , the width of the signal trace should be scaled 10-20% larger than that of a CPW line to account for only one ground flank.

4.3 – Experimental Setup

To validate the effectiveness of SS-CPW, FEXT measurements were made from four test cases and the results compared. In general, the effectiveness of a SS-CPW transmission line was compared to that of microstrip and CPW. Each test case consisted of a FEXT measurement from the line under test to that of a microstrip line. In addition to FEXT, the reflection and insertion loss of each transmission line structure type was recorded. Lines had a fixed length of 1000mils and the end of each line not being probed was terminated in a 50 Ω load. Test structures were designed for signal to ground separations of both 6.7mil and 53.7mils and all structures were designed to have a characteristic impedance of 50 Ω . Testing was performed on a single low cost FR4 board using the Keysight N5245A vector network analyzer. Figure 29 depicts the test board used.

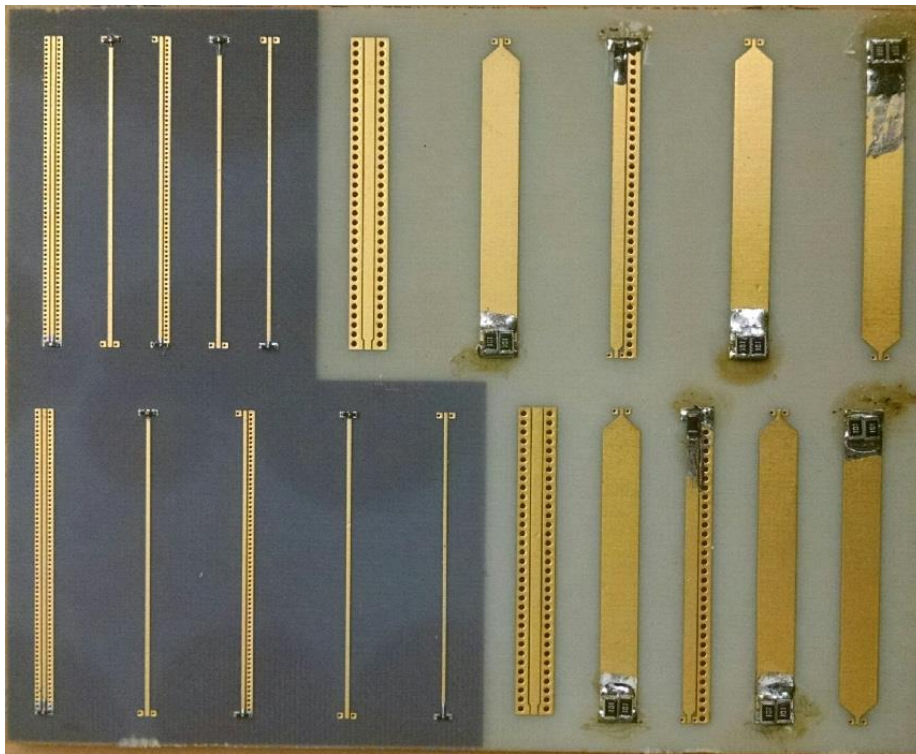


Figure 29: SS-CPW FR4 test board

4.4 – Experimental Results

In this section, experimental results comparing SS-CPW to the traditional transmission line structures of microstrip and CPW are recorded and analyzed. This section contains results for two different signal to ground plane separations of 6.7 and 53.7mils. While the general effect that signal to ground plane separation has on isolation was explored in Chapter 2, it is covered again in this section in the context of its effect on SS-CPW.

4.4a – Results with Signal to Ground Plane Separation of 6.7mils

Figure 30 depicts the measured crosstalk for each transmission line type under test to a SS-CPW line. As expected, microstrip provided the least amount of isolation while CPW provided approximately 10dB more isolation than microstrip. When the single ground flank of the SS-CPW was placed on the side closest to the microstrip victim line, SS-CWP improved isolation by approximately 7dB when compared to microstrip. However, when the ground flank was placed on the opposite side, SS-CPW only provided a nominal increase in isolation of 3dB over microstrip.

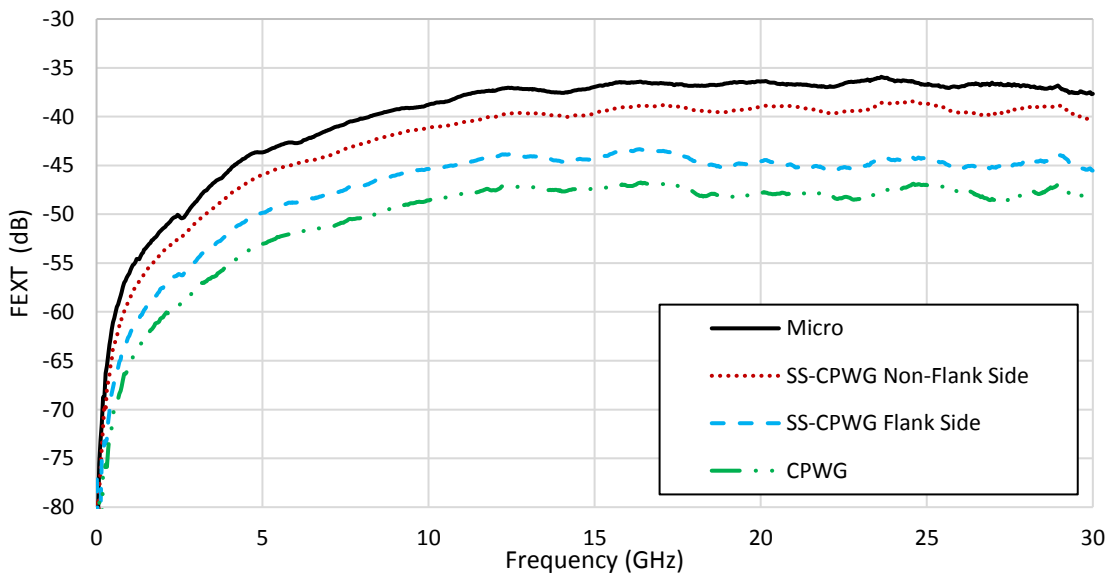


Figure 30: Comparison of FEXT at 6.7mils (from [17])

Figure 31 provides a comparison of the measured reflection of each transmission line structure. For a signal to ground plane separation distance of 6.7mils all three test structures performed relatively the same. A low reflection number suggests that the characteristic impedance of the transmission lines was close to the target impedance. In this case, the characteristic impedance of all three test structures was close to the desired characteristic impedance of 50Ω .

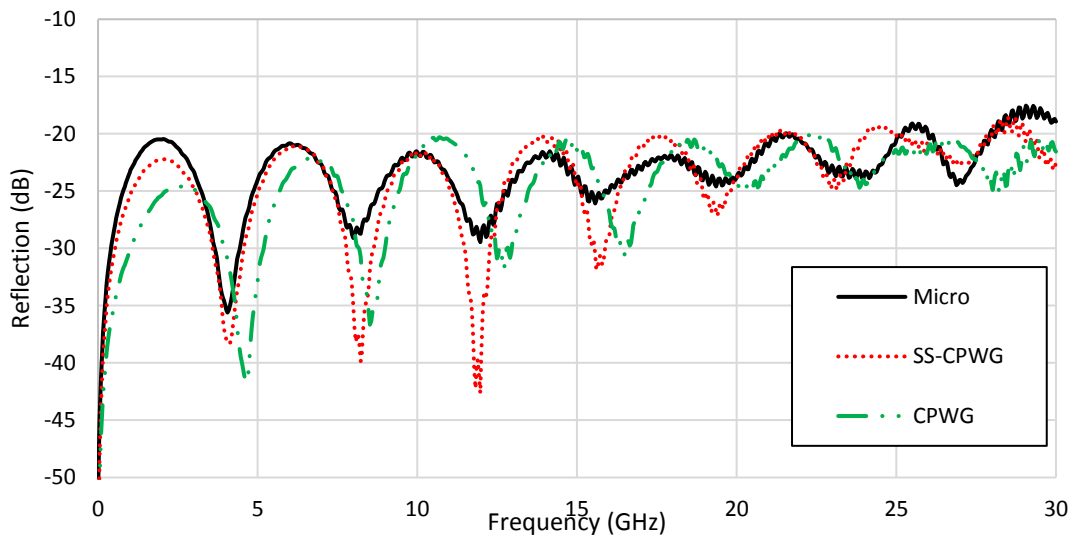


Figure 31: Comparison of reflection at 6.7mils (from [17])

Figure 32 is a comparison of the insertion loss of the different transmission line structures. Insertion loss is a measure of signal power loss as the signal propagates down the transmission line. SS-CPW outperforms traditional CPW for the entire frequency range by as much as 0.7dB. In addition, for frequencies less than 15GHz, SS-CPW outperforms microstrip by a slight margin.

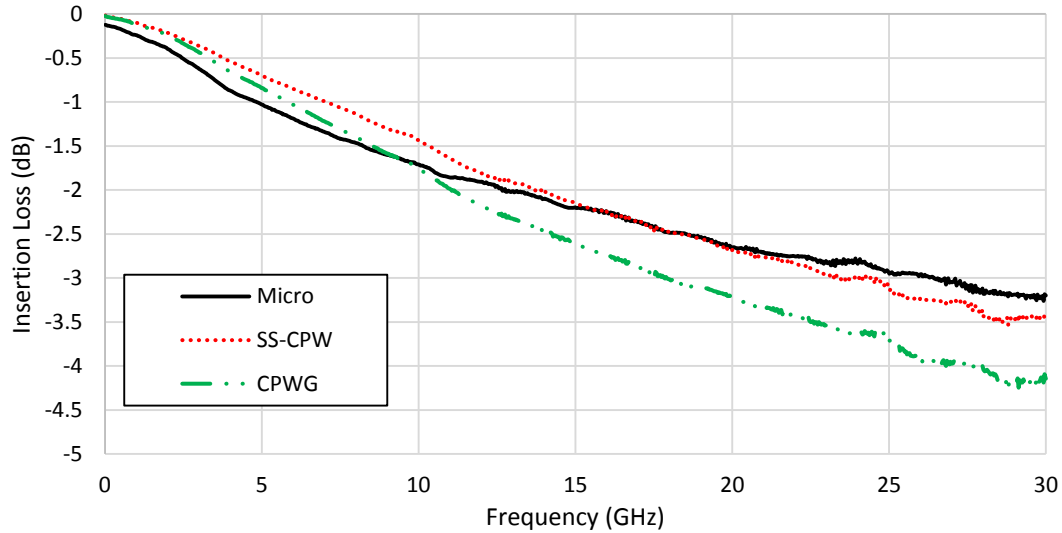


Figure 32: Comparison of insertion loss at 6.7mils (from [17])

4.4b - Results with Signal to Ground Plane Separation of 53.7mils

In general, as the distance between signal and ground plane increases, isolation between aggressor and victim line should decrease. This is once again the case for SS-CPW. Figure 33 depicts the FEXT of transmission line structures with a signal to ground plane separation of 53.7mils. Both CPW and SS-CPW increase isolation by as much as 12dB when compared to microstrip. Once again, the importance of the ground flank placement is evident. If the ground flank is placed on the opposite side of the victim line, SS-CPW provides the isolation as microstrip. However, when the ground flank is placed on the side closest to the victim line, SS-CPW provides isolation benefits comparable to that of CPW.

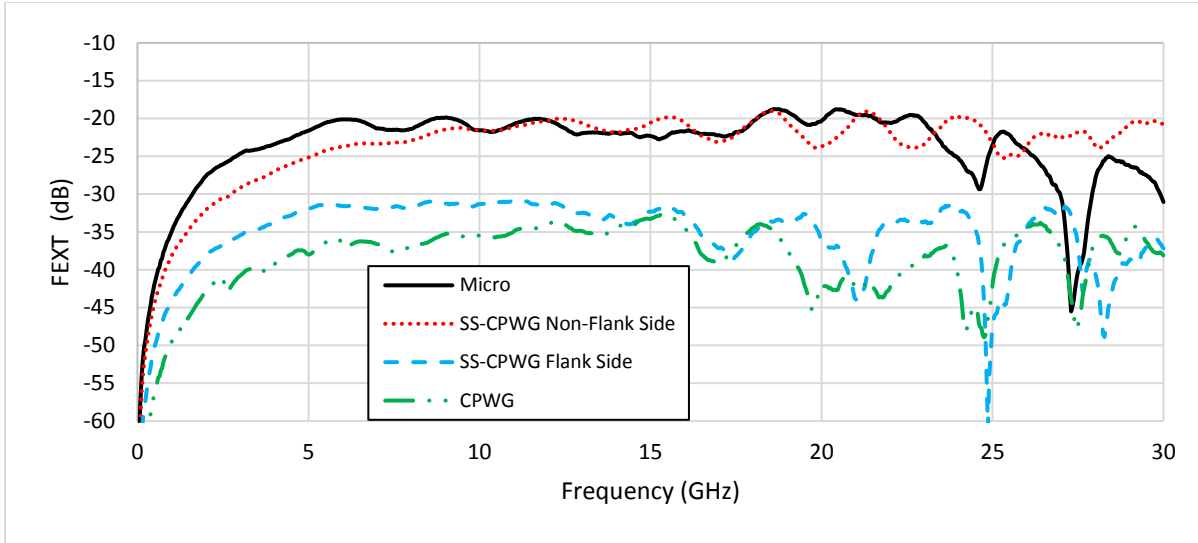


Figure 33: Comparison of FEXT at 53.7mils (from [17])

Figure 34 depicts the reflection measurement for each transmission line structure at 53.7mils. A potential reason for the decreased reflection performance of SS-CPW, is that the characteristic impedance of the SS-CPW did not closely match the target impedance of 50Ω . With more optimal dimensions, the reflection measurement of SS-CPW may further be improved.

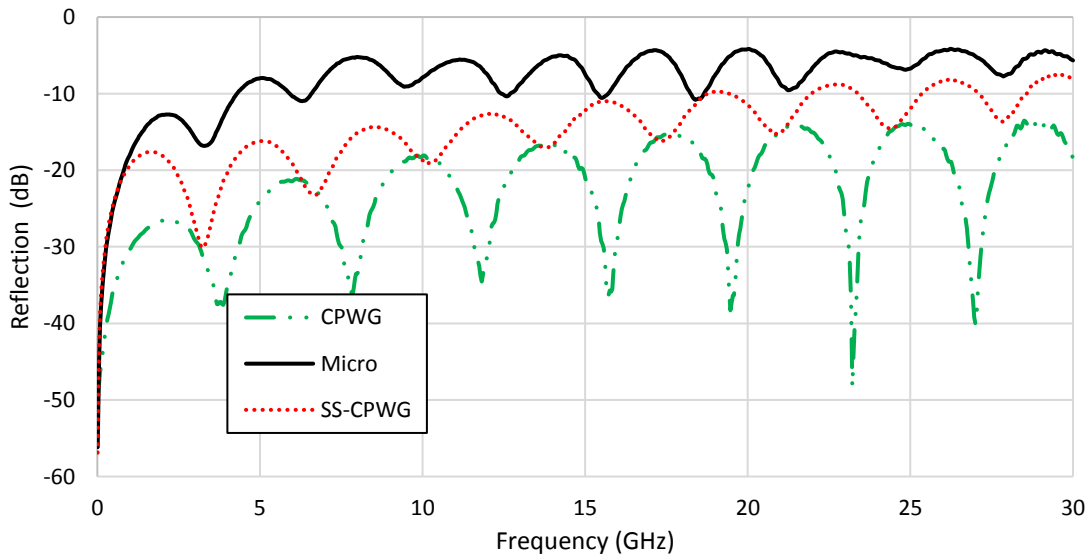


Figure 34: Comparison of reflection at 53.7mils (from [17])

4.5 – Example Use Case for SS-CPW

As mentioned in the introduction, one of the driving forces for the development of SS-CPW was design flexibility. Due to utilizing one ground flank, SS-CPW allows for the addition of stubs and filters to the circuit. In addition, after board manufacturing, designers are able to manipulate the circuit better than if traditional CPW was utilized. An example of this is shown in Figure 35.

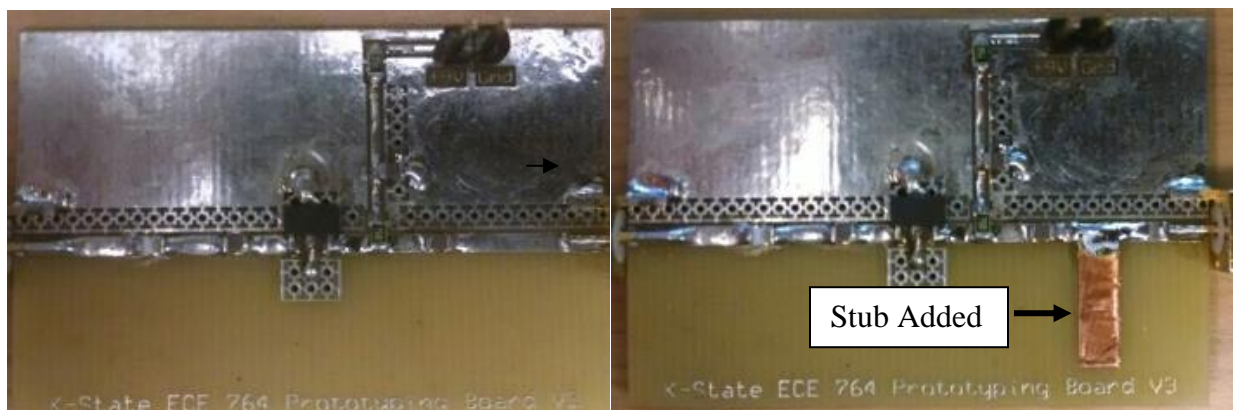


Figure 35: Example use case of SS-CPW (from [17])

Figure 35 depicts a simple amplifier circuit utilizing SS-CPW. After manufacturing and testing it was determined that the output of the amplifier needed to be filtered. Due to the utilization of SS-CPW, a stub was easily added to filter the output.

Another important benefit of SS-CPW was that it consumes less board space when compared to CPW and, under certain conditions microstrip. In order to further illustrate this fact, Table 2 gives the total width in mils of each transmission line test structure for both signal to ground plane separations.

Table 2: Width comparison of SS-CPW (from [17])

	Microstrip	CPW	SS-CPW
6.7mils	13.5	61	37
50.4mils	120	123	86

From Table 2 it is clear that SS-CPW will always consume less board space when compared to CPW due to utilizing only one ground flank when the same ground flank dimensions are used. Further, the total width of SS-CPW is more resistant to changes in distance between signal and ground planes when compared to microstrip. When the separation was increased from 6.7mils to 53.7mils, SS-CPW only increased in width by 32.4% while microstrip increased in width by 690%. Regardless of signal and ground separation, as long as the width of the ground flank is kept constant, SS-CPW will consume less board space when compared to CPW due to only utilizing one ground flank.

4.6 – Conclusion

SS-CPW is a useful transmission line structure for designers wanting to balance isolation requirements and design layout flexibility. It has been experimentally confirmed that when compared to microstrip, SS-CPW can increase isolation by as much as 7dB for sufficiently small signal to ground plane distances. As the distance between the signal and ground plane increases, the isolation benefit that SS-CPW provides can approach that of CPW. It has also been shown that for SS-CPW ground flank placement is extremely important. In order provide the maximum amount of isolation, the single ground flank of SS-CPW must be placed on the side closest to the victim line. It can be concluded that for designs with strict isolation requirements, CPW still provides slightly better isolation when compared to SS-CPW. However, for circuit designs that

emphasize design flexibility, inclusion of stubs and filters, minimal size and still require a high degree of isolation, SS-CPW is a viable option and should be considered in place of traditional CPW.

Chapter 5 - Ablation of Dielectric PCB Layers

5.1 – Introduction to Laser Ablated Removal of Top Dielectric

In addition to exploring a new transmission line structure, post processing techniques were also investigated for their potential ability to reduce crosstalk. A post processing technique refers to any process that occurs after the PCB board has been manufactured. Post processing techniques not only allow for experimentation with cheaper FR4 PCB but also allows for quicker resolution of crosstalk problems in PCB prototype. Instead of having to remanufacture a PCB board, post processing techniques allow for the ability to increase isolation on already existing boards. One post processing technique investigated was selective surface dielectric removal through laser ablation.

Recall that in general crosstalk is composed of both capacitive and inductive coupling. To reduce crosstalk either the inductive or the capacitive coupling must be reduced. Through laser ablation, dielectric material can be removed from a PCB board. By selectively removing dielectric material from the surface of the PCB board between traces, capacitive coupling can be reduced without affecting the characteristic impedance of the transmission line. For cases where the distance between the signal and ground plane is small, the reduction in capacitive coupling is minimal as most electric field lines terminate on the ground plane. However, as the distance between signal and ground layers increases, more capacitive coupling occurs and can be reduced by ablation of the dielectric constant.

In addition to reducing capacitive coupling, removal of dielectric material until the ground plane is exposed can reduce crosstalk resulting from surface wave propagation [18]. As a signal propagates down a transmission line, it forms an EM field pattern, more commonly known as a mode. At low frequencies, dominant modes remain bound to the signal conductor. However,

at higher frequencies the signal will leak out in the form of a surface wave on the ground plane. Not only will this surface wave reduce the power transmitted in the transmission line, it will also cause crosstalk [18]. By exposing the ground plane, EM absorber material can be placed between lines to dampen EM surface waves and in turn further reduce crosstalk.

5.2 – Mathematical Theory of Dielectric Removal for Capacitive Coupling

Recall from chapter 2.2 that capacitive coupling is a significant contributor to crosstalk. The capacitive coupling between two transmission lines can be broken down into two separate cases. When the distance between the signal plane and ground plane is kept small, most of the electric field from the aggressor line will terminate on the ground plane. In this case, removal of the top dielectric will have minimal affect on the capacitive coupling.

When the distance between signal and ground plane is sufficiently large and lines are closely spaced, the electric field in the dielectric material will couple from the aggressor line to the victim line thru the dielectric with permittivity $\epsilon_R \epsilon_0$. As a result, this leads to a capacitance:

$$C_m \approx \frac{2(1+\epsilon_R)\epsilon_0 LW}{\pi(S+W)} \quad (2.11)$$

For a board material such as FR4 with a dielectric constant of 3.7, this can increase the capacitive coupling by as much as a factor of 3 relative to free space. However, when the dielectric material between the two transmission lines is ablated away, the above equation becomes:

$$C_m \approx \frac{4\epsilon_0 LW}{\pi(S+W)} \quad (5.1)$$

When compared with a traditional FR4 board, a selectively ablated FR4 board can in theory reduce capacitive coupling by approximately 40%.

5.3 – Experimental Setup

Testing was performed on a single low cost FR4 PCB and measurements were made using a Keysight N5245A vector network analyzer. Lines were terminated in 50Ω and for transmission line structures without via flanks, ground vias were added to one end of the transmission line to properly land the $500\mu\text{m}$ ground signal ground (GSG) probes. Measurements were made before the board was ablated, and after half trench ablation. Half trench ablation refers to only removing the dielectric material between the two transmission lines as seen in Figure 36. The selective surface dielectric remove was done by laser ablation at UNL by Dr. Yongfeng Lu and Timothy Carlson.

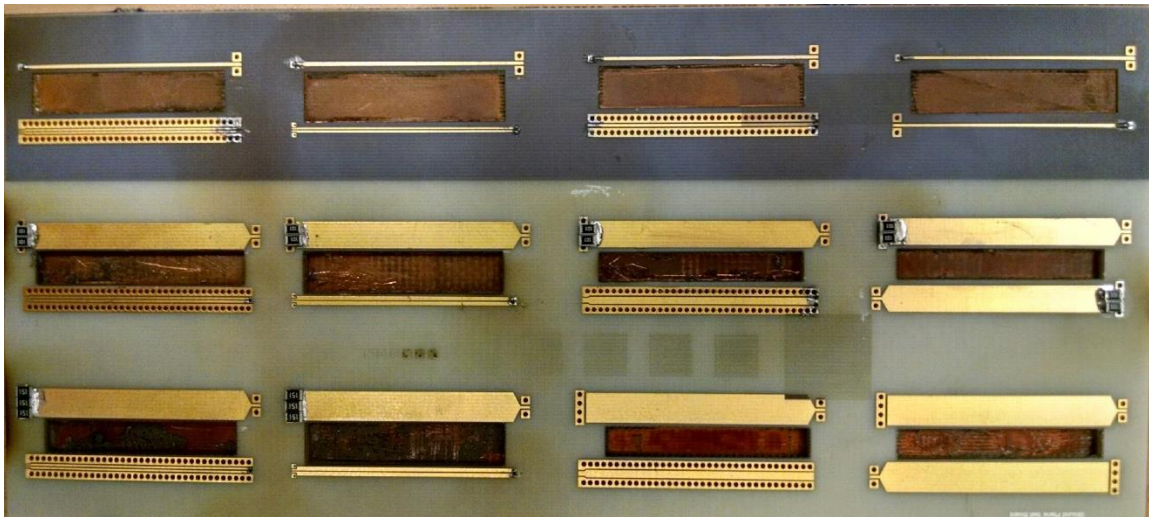


Figure 36: Test Board with Half Trench Ablation (Ablation courtesy of UNL)

The two traditional transmission line structures tested were microstrip and CPW. Coupling between each transmission line under test and a microstrip line was measured. Crosstalk was measured for structures with a signal to ground plane distance of both 6.7 and 53.4mils. Additionally, in order to show that the impedance of the line was not significantly altered from 50Ω , the reflection measurements are also presented.

5.4 – Experimental Results

In the following section results for each transmission line structure are recorded and analyzed separately. Crosstalk and reflection results from 100MHz-30GHz are presented. Reflection results are also presented to provide insight into the effect selective dielectric removal has on the characteristic impedance of transmission lines. The closer to the correct impedance (in this case 50Ω) a transmission is, the less power is reflected back into the network analyzer. If laser ablation significantly changed the characteristic impedance of a transmission line, the reflected power would either increase or decrease significantly.

In addition to measuring the effect of simply removing the dielectric, two thicknesses of commercially available absorber material were tested. Both absorber materials were manufactured by Mast Technologies with the thicker absorber (Part number: MR31-0007-20) material measuring 0.080 inches thick and the thinner absorber material (Part number: MR31-0001-20) measuring 0.010 inches thick. The absorber material was placed directly on the exposed copper ground plane. According to Mast Technologies specifications, the thicker absorber material should dampen lower frequency waves and the thinner absorber material should perform better at the higher frequency. An example of measurements being made with the absorber material is shown in Figure 37.

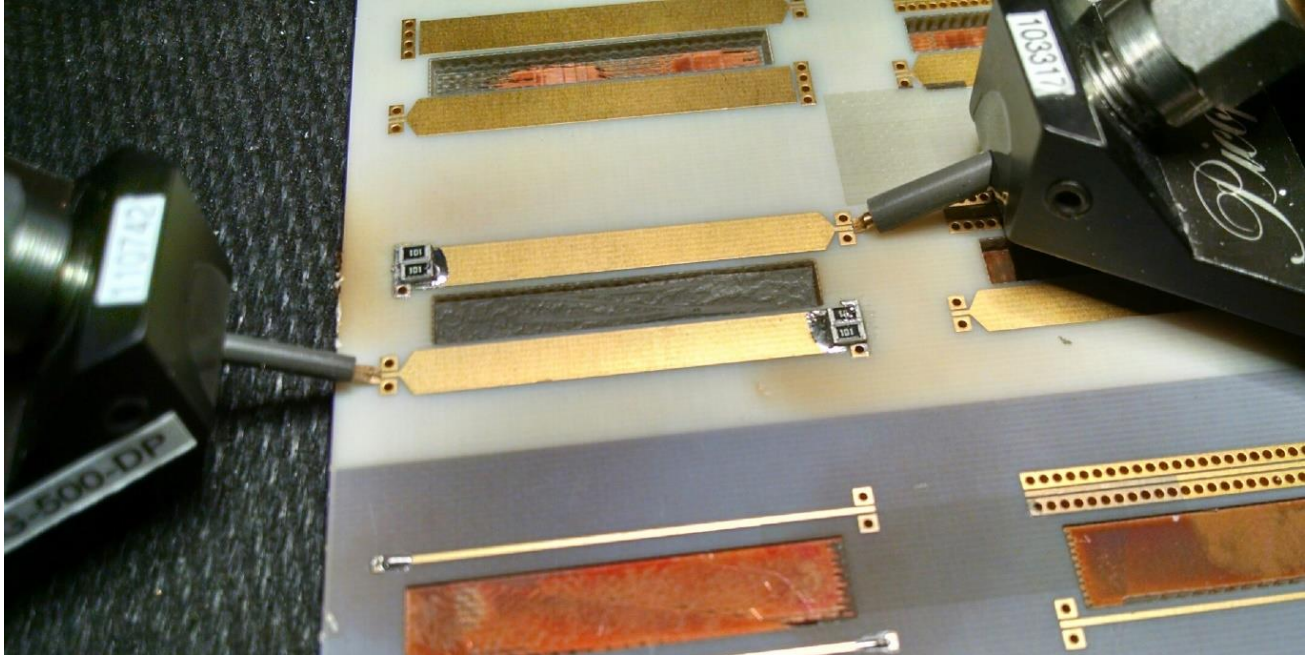


Figure 37: Measurements being made with absorber material in place

Figure 38 depicts far end crosstalk (FEXT) from microstrip to microstrip with a signal to ground layer distance of 6.7mils.

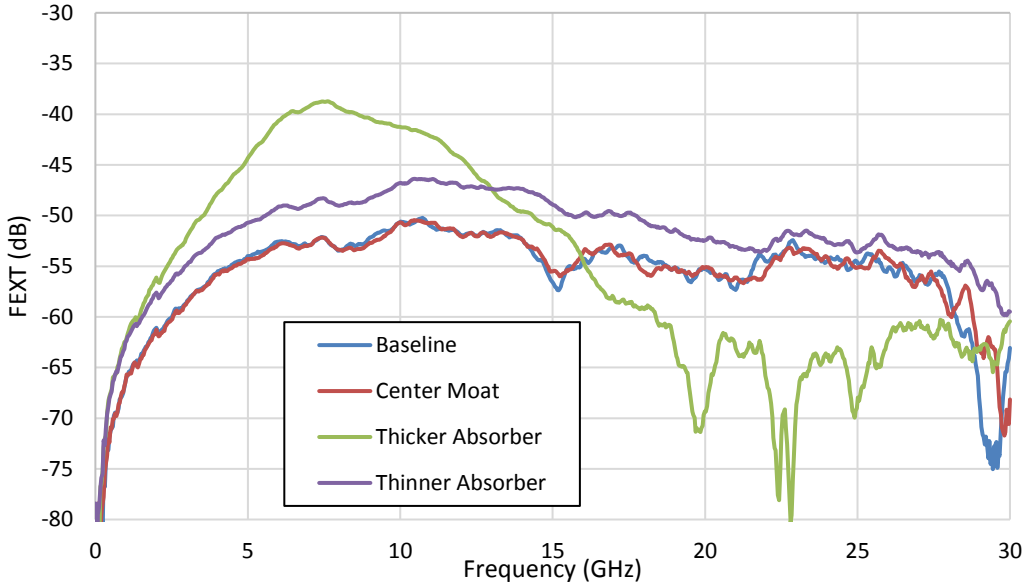


Figure 38: FEXT of microstrip at 6.7mils

It is interesting to note from Figure 38 that at frequencies less than 16GHz, the thicker absorber increases the FEXT. However, at frequencies greater than 16GHz, the thicker absorber material can reduce crosstalk by as much as 9dB. Additionally, it appears that simply ablating the top dielectric material does not seem to have an effect on the FEXT when the signal to ground plane are 6.7mils apart. This fits with the theory that significant capacitive coupling only occurs in the dielectric material when the distance between signal and ground planes is sufficiently large. The ablated board with the thinner absorber performed poorly at all frequencies. In most cases, the thinner absorber increased FEXT when compared to the baseline. Since, surface waves only occur at sufficiently high frequencies, it comes as no surprise that the thinner absorber did not perform well.

Figure 39 depicts the reflection for a microstrip transmission line structures with signal to ground plane separation of 6.7mils. As expected, the reflection did not change when the dielectric material was removed, or the absorber materials were added.

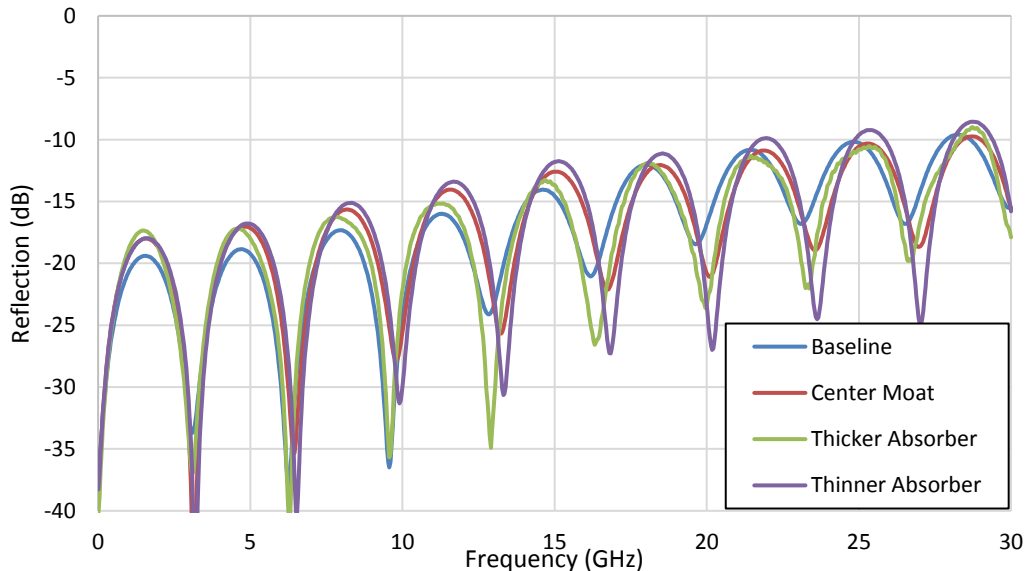


Figure 39: Reflection of microstrip at 6.7mils

Figure 40 depicts the FEXT between two microstrip lines when the signal to ground plane distances is increased to 53.7mils.

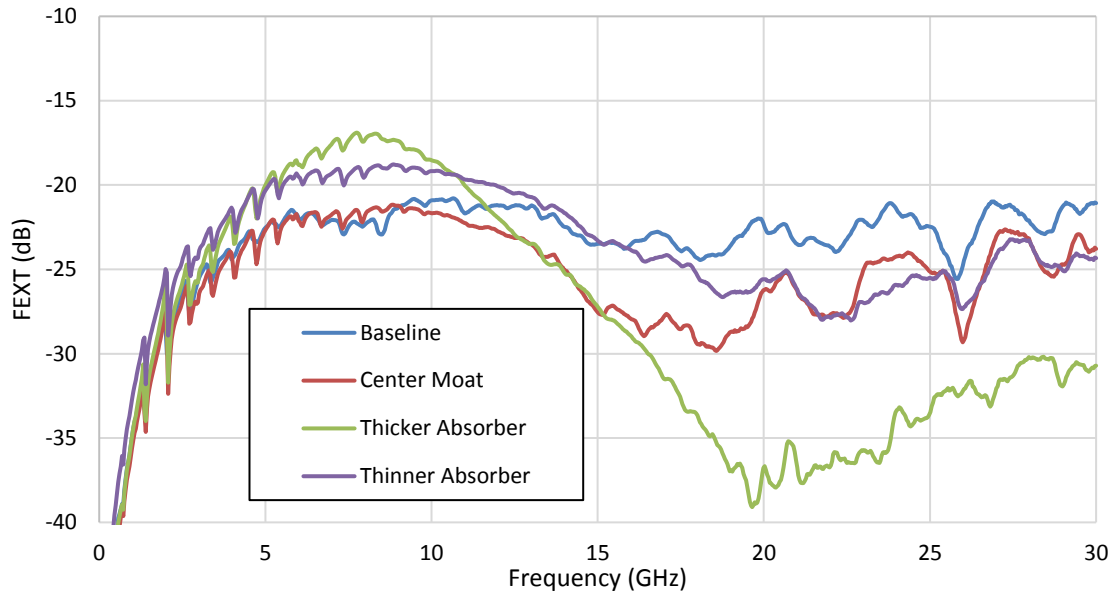


Figure 40: FEXT of microstrip at 53.7mils

As the distance between the signal and ground trace is increased to 53.7mils, ablating the center dielectric material appears to increase isolation by approximately 3-4dB for frequencies greater than 14GHz. Once again this is consistent with theory that significant capacitive coupling in the dielectric material occurs only when the distance between signal and ground plane is large.

For both signal to ground plane separations, the thinner absorber did not seem to improve isolation. In the case of the 6.7mil separation, the ablated substrate with the thinner absorber performed worse even when compared to the baseline board. When the thicker absorber material was used however, isolation increased by 12-14dB at 20GHz when compared to the baseline. Overall, the thicker absorber material performed better than baseline for frequencies greater than 15GHz and worse than baseline at low frequency.

Figure 41 depicts the reflection for a microstrip transmission line structures with signal to ground plane separation of 6.7mils. Once again, the reflection did not change when the dielectric material was removed, or the absorber materials were added.

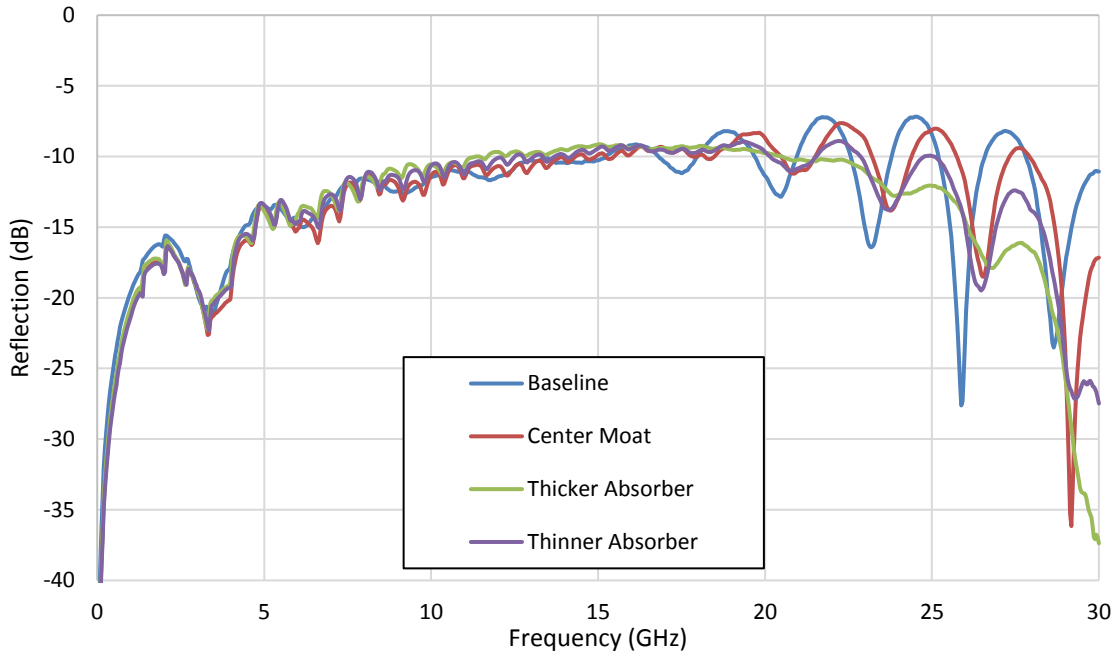


Figure 41: Reflection of microstrip at 53.7mils

Figure 42 depicts the FEXT between a CPW and a microstrip line when the signal to ground plane distances is 6.7mils.

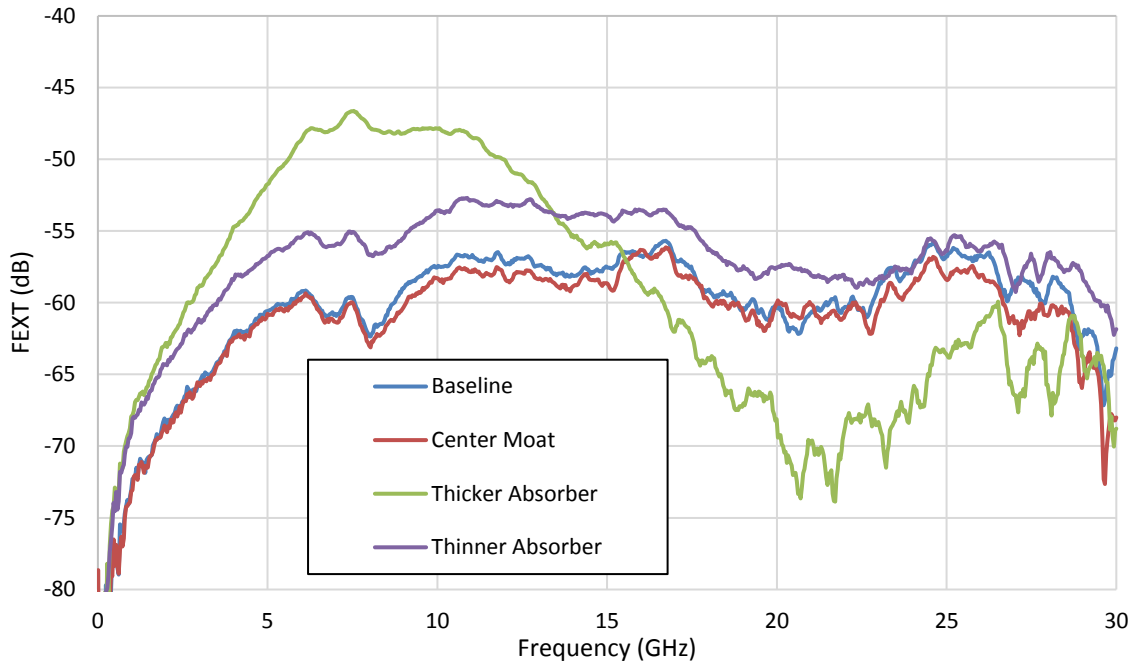


Figure 42: FEXT of CPW with signal to ground plane separation of 6.7mils

As expected from previous results for ground to signal separation of 6.7mils, simply ablating the dielectric substrate had little effect on FEXT. Overall, isolation between aggressor and victim line increased but this was due in large part to the CPW transmission line. However, when the ablated trench was filled with the thicker absorber, FEXT decreased overall for frequencies greater than 16GHz with a maximum decrease in FEXT of 9dB. Much like the microstrip results, the thinner absorber increased crosstalk slightly regardless of the frequency.

Figure 43 depicts the reflection for a CPW transmission line structures with signal to ground plane separation of 6.7mils. As shown in the graph, no significant change in reflection was observed.

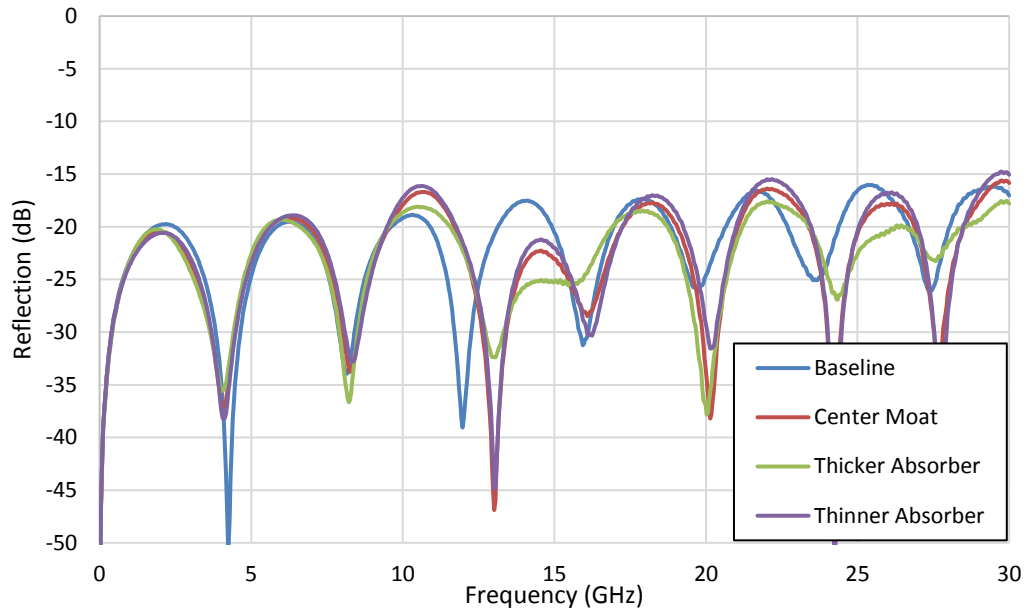


Figure 43: Reflection of CPW at 6.7mils

Figure 44 shows the FEXT between a CPW and a microstrip line when the signal to ground plane distances is increased to 53.7mils.

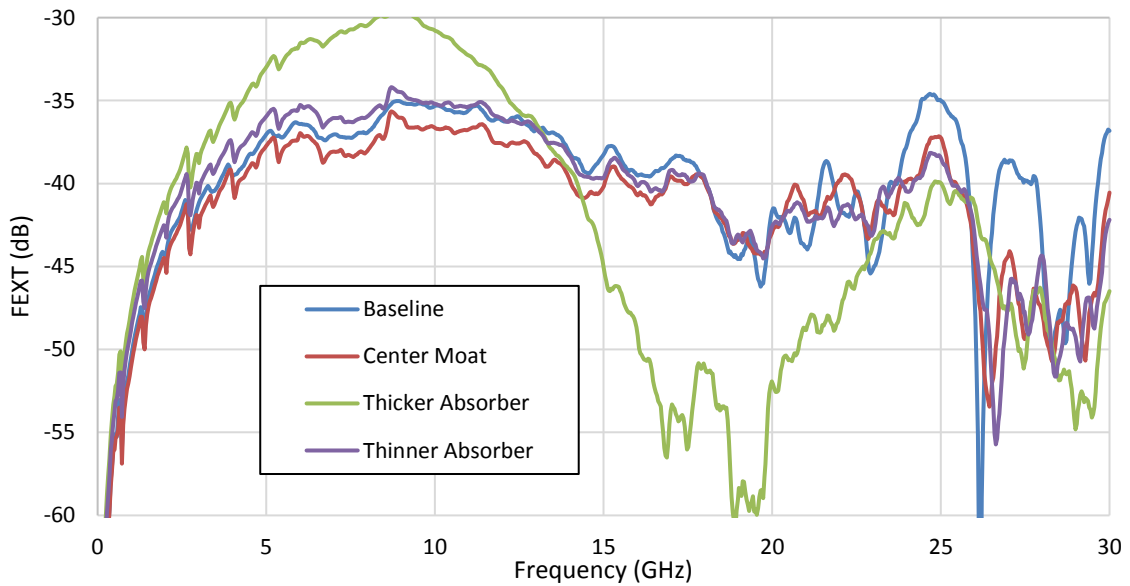


Figure 44: FEXT of CPW with signal to ground plane separation of 53.7mils

As the distance separation was increased to 53.7mils, the ablated board with the thicker absorber outperforms the baseline from 14-25GHz. Results from 25-30GHz were inconclusive. For CPW with signal to ground plane separation of 53.7mils, the ablated board with thicker absorber can increase isolation by a maximum of 14dB. Unlike the microstrip to microstrip transmission lines with signal to ground plane separation of 53.7mils, simply ablating the dielectric material between a CPW and a microstrip line had no effect on FEXT. This was largely due to the CPW ground flanks preventing capacitive coupling from occurring in the dielectric board substrate. The ablated board with thinner RF absorber material had no effect on isolation.

Last, Figure 45 depicts the reflection for a CPW transmission line structure with signal to ground plane separation of 53.7mils. As shown in the graph, no significant change in reflection was observed.

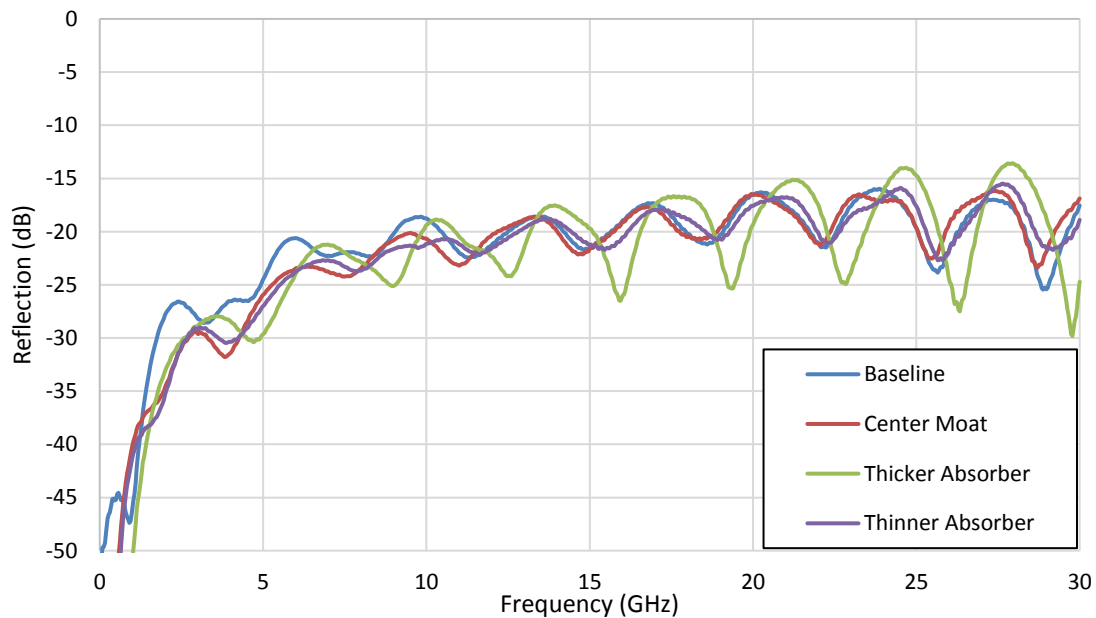


Figure 45: Reflection of CPW at 53.7mils

5.4 – Conclusion

Isolation results have been presented for selective removal of dielectric material. With the addition of thick RF absorber material, both microstrip and CPW transmission lines benefited at higher frequencies. Isolation improved by as much as 14dB for frequencies greater than 14GHz. when the thick RF absorber material was added. However, results also showed that when too thin of an RF absorber was used, isolation decreased. Additionally, it was shown through reflection measurements that the selective removal of dielectric material did not significantly alter the characteristic impedance of the transmission lines. While results showed promise further research needs to be done in order to fully unlock the potential of selective ablation of dielectric material and addition of absorbers to combat surface wave coupling.

Chapter 6 - Split Shielding VS Traditional Shielding

6.1 – Introduction

Recall in chapter 5 that as a signal propagates down a transmission line, surface waves may form at sufficiently high frequencies. These surface waves may in turn cause crosstalk through either inductive or capacitive coupling to the victim line. While as shown in chapter 3 in general the best way to prevent any form of crosstalk would be to completely enclose a victim line in shielding, this is not always feasible. For example, components need to accept input or output on several different lines and with current shielding practices, it may not be possible to individually completely shield each input and output transmission line.

In general, RF shielding practices today consist of shielding a PCB with a single piece of aluminum or some other lightweight metal. In order to increase the isolation between separate circuits on the board, separate cavities may be utilized. These shields are either soldered onto the board or compressed on the board through the use of screws or clips. An example of a traditional shielding structure with cavities is shown in Figure 46.

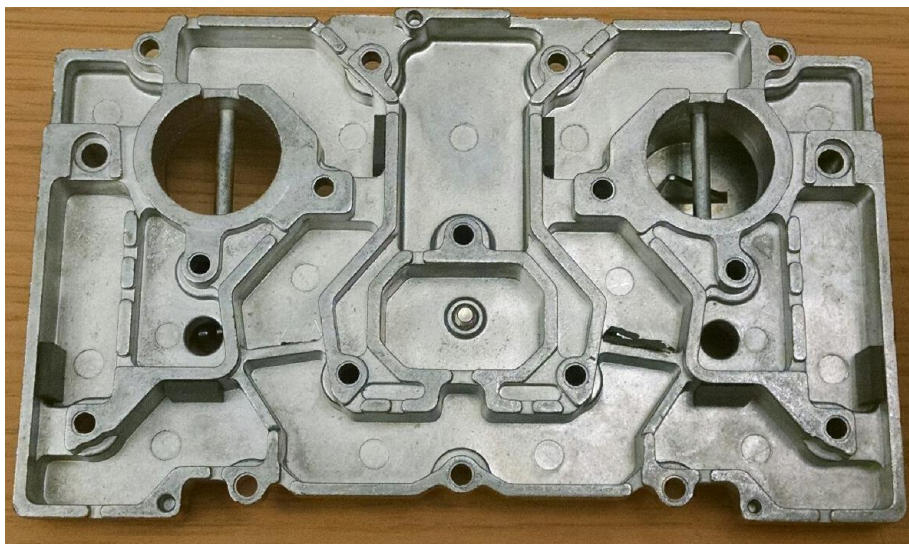


Figure 46: Example of shielding with separate cavities

While separate cavities may reduce the crosstalk between components and transmission lines in separate cavities, it does nothing to prevent crosstalk from occurring within the same cavity or from separate input and output lines of the same surface mount component. If not separately shielded, surface waves can travel along the single shield or cavity wall from the aggressor line to the victim line.

Until now, manufacturing separate shield structures for separate input and outputs of a single component or a single cavity would be cost prohibitive. Furthermore, current milling techniques may not allow for the intricacy necessary to mill a shield structure that precise. However, with the advent of 3d printers, the tools necessary to construct such shields are more readily available and more cost effective. An example of 3d printed shields is shown in Figure 47. All 3d printed shields were designed and manufactured by Dr. Alan Mantooth and Zeke Zumbro at UAR.

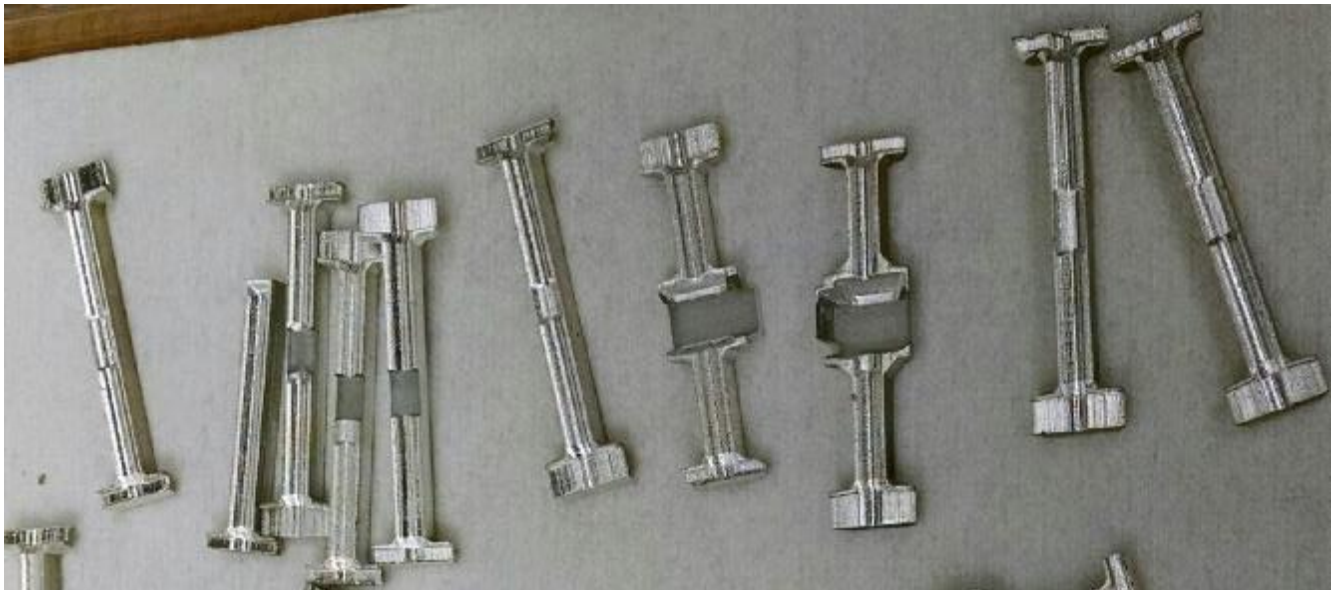


Figure 47: 3d printed RF shields (Courtesy of UAR)

The shields in Figure 47 were printed using a Formlabs SLA printer. Once printed, metal was sputtered onto them to make them electrically conductive. The shields could then be attached to the board using either low temperature solder or compression.

6.2 – Simulation Results

To investigate the role that RF shielding has on the crosstalk between the input and output of a surface mount component, several simulations were performed. Simulations were performed on traces without shielding (baseline), with a single shield over both lines and with separate shields for each line. Three sets of simulations were performed with spacing between the aggressor and victim lines of 30, 60 and 120mils. Separate distances were simulated to mimic different surface component dimensions. Shown in Figure 48 is a picture of the typical test structure setup showing port locations. FEXT is when the signal from port 1 couples to port 4 or when signal from port 3 couples to port 2. Simulations were run from 100MHz to 30GHz and FEXT results were recorded.

The simulator used was based on the finite element method (FEM) and is included with the Advanced Design System software. The simulation was run from 100MHz to 30GHz. To ensure accuracy, the delta error was set to 0.01 and the stop criterion required that at least 2 consecutive passes of the simulation be below the delta error. All other settings were left at default.

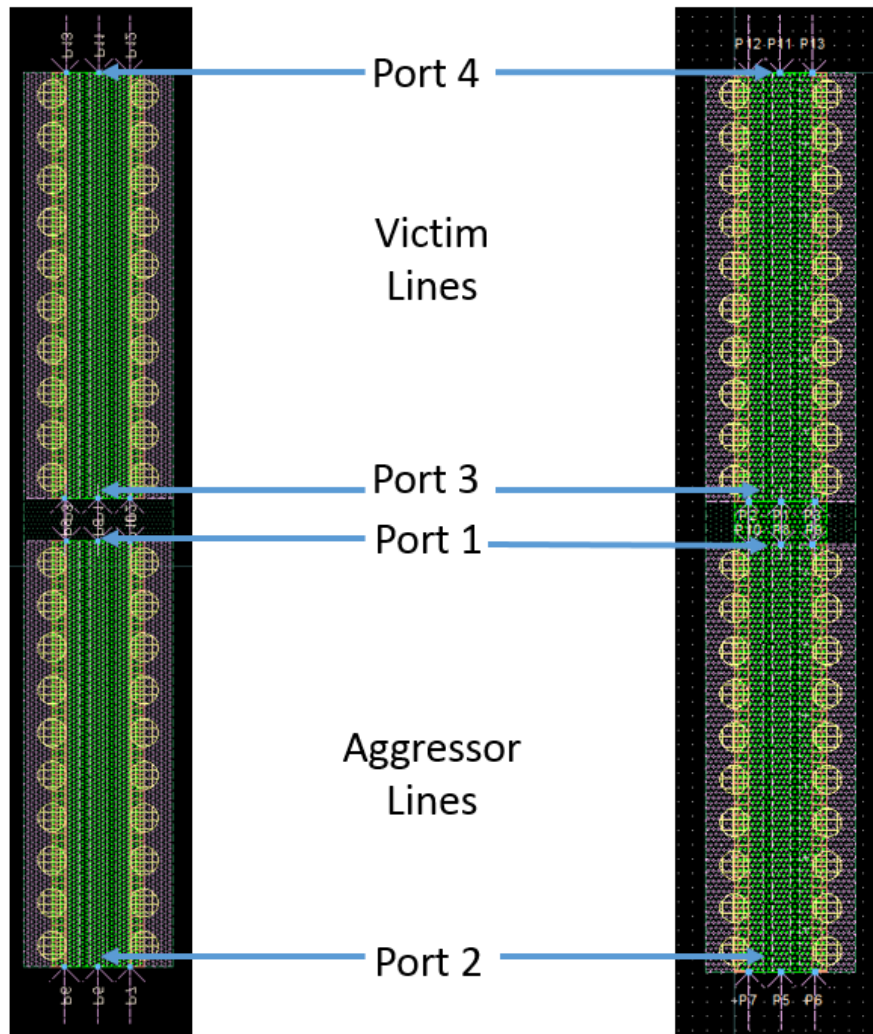


Figure 48: Simulation setup for shielding test with split shield vs single shield

Figure 49 depicts FEXT when there was a separation of 30mils between the aggressor and victim line. As theorized, the single shield did not appear to have any effect on isolation over baseline. When a split shield was used however, an increase in isolation of approximately 20dB was observed.

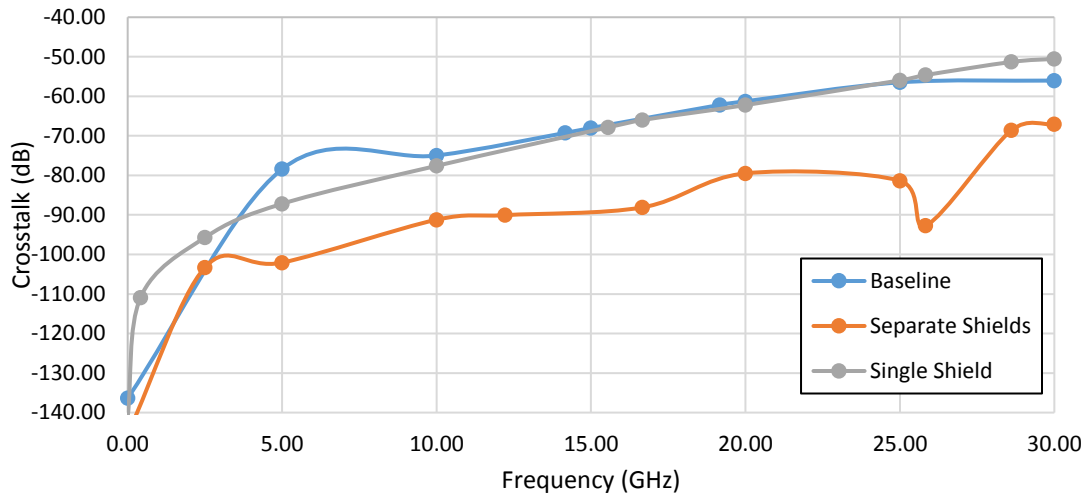


Figure 49: Simulated FEXT with separation of 30mils

Figure 50 depicts the FEXT between aggressor and victim line when the separation was increased to 60mils. When the separation was increased, the improvement that the split shield provided over the single shield was still impressive. The simulation once again showed an increase in isolation of approximately 20dB.

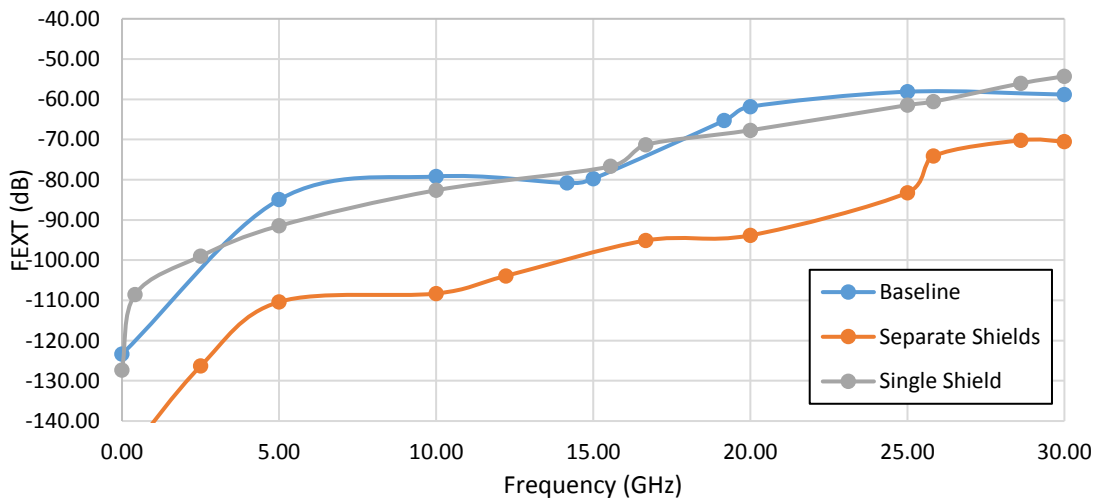


Figure 50: Simulated FEXT with separation of 60mils

Last, Figure 51 depicts the FEXT between the aggressor and victim line when the separation was increased to 120mils. Once again, the increased distance between the aggressor

and victim line did not seem to either increase or diminish the effect that the split shield had on isolation. Once again, isolation increased by 20dB when split shield was utilized when compared to a single shield and baseline.

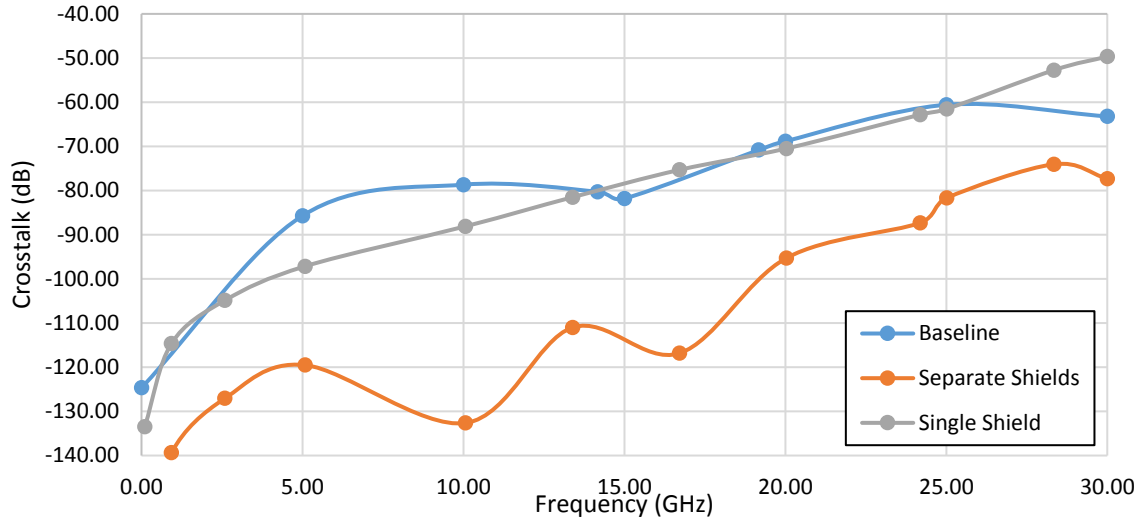


Figure 51: Simulation of FEXT with separation of 120mils

Given that simulation results suggested that utilizing a split shield provided greater isolation, an experiment was performed to confirm the results seen in the simulations.

6.3 – Experimental Setup

To confirm the validity of the simulation results, a test board was constructed. The constructed test board is shown in Figure 52 and was made out of low cost FR4 material.

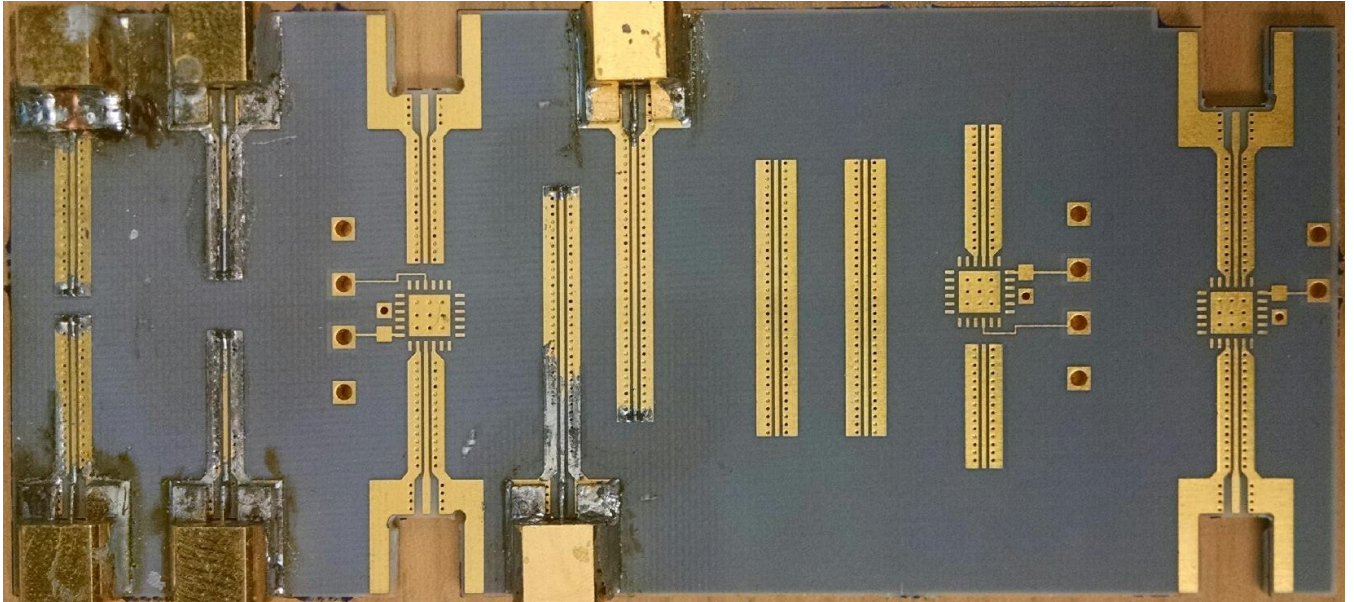


Figure 52: Split shield test board

Measurements were made using the using a Keysight N5245A vector network analyzer and connectors were used instead of the GSG probes. In order to emulate the input and output of a surface mount component, the ends of the transmission lines under test were terminated in 50Ω . FEXT measurements were made without shields (baseline), with a signal traditional shield structure and with a split shield structure. Clamps were used to affix the shielding to the ground flanks of the CPW transmission lines. An example of this is shown in Figure 53.

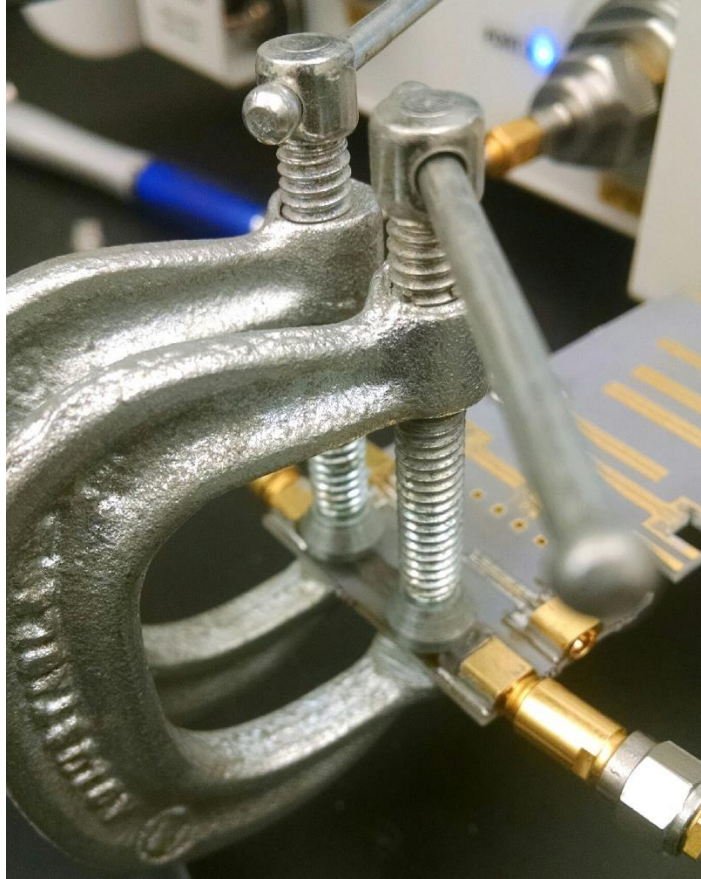


Figure 53: Test board with clamped on shielding

6.4 – Experimental Results

Figure 54 depicts FEXT measurements when the ends of the two transmission lines were spaced 40mils apart. It is clear that the results do not match that of the simulation. Below 15GHz, it appeared that the single shield outperformed both the split shielding and baseline. Above 15GHz, it appeared that shielding had no effect on FEXT.

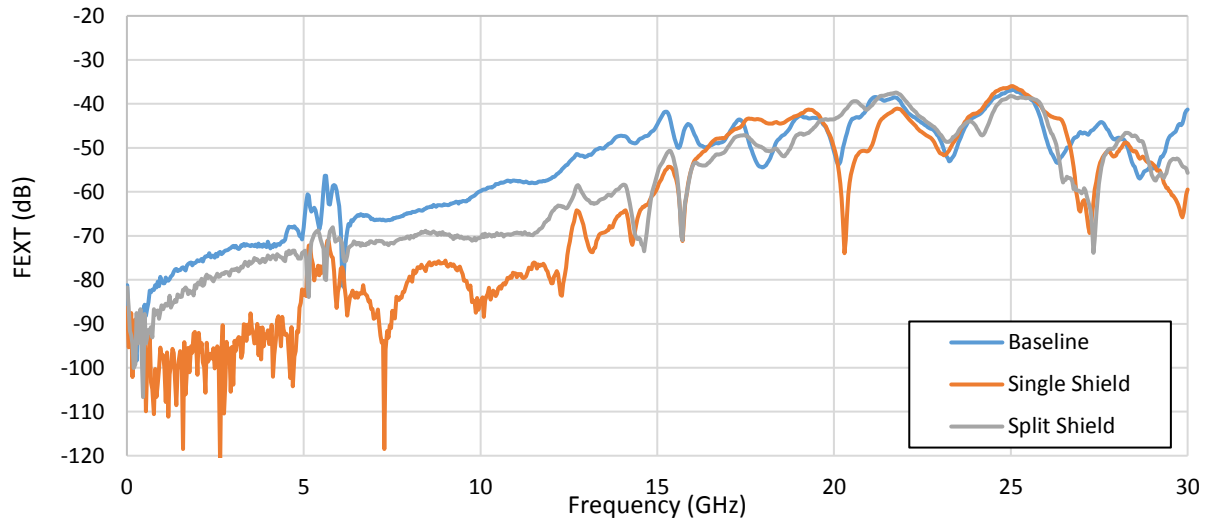


Figure 54: Measurement of FEXT with separation of 40mils

Figure 55 depicts the measured FEXT when the gap between the two transmission lines was increased to 120mils. Once again, the split shields did not appear to have any appreciable benefit over either the single shield or baseline. Furthermore, it appeared that the single shield was no longer more effective than baseline at frequencies less than 15GHz.

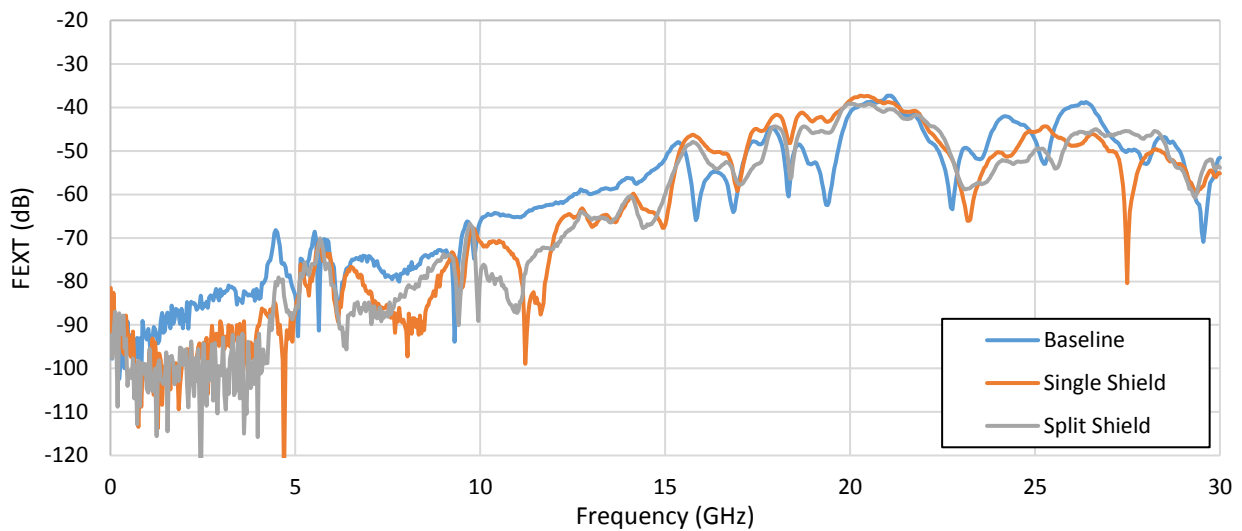


Figure 55: Measurement of FEXT with separation of 120mils

In both cases, the split shielding failed to improve isolation by the 20dB the simulation suggested it would. Even worse, the split shield provided no appreciable increase in isolation when compared to baseline. One possible reason for the discrepancy between the simulation results and the experimental results was the effect shielding aperture had on isolation.

6.5 – Shielding Aperture and Isolation

Shielding aperture can refer to any opening in the shield. To some extent all shielding must have an aperture to allow input and output signals to reach the circuit. When care is not taken to correctly design these openings, a shielding aperture can completely negate any increase in isolation that shielding might provide. A general equation for shielding effectiveness when there are slots equal to or less than $\frac{1}{2}$ wavelength is given by the following equation where λ is the wavelength and l is the length of the slot of the aperture [9]:

$$S = 20 \log \left(\frac{\lambda}{2l} \right) \quad (6.1)$$

From the above equation it can be shown that if the aperture slot length is a $\frac{1}{2}$ wavelength of the frequency, shielding effectiveness is reduced to 0dB. Multiple apertures can further decrease the effectiveness in shielding.

Keeping this in mind, both the split shields and single shield were examined for significant apertures. Figure 56 depicts a side view of the board and shielding where some apertures are present. It appears from the figure that a problem point for the 3d printed shields occurred by the connectors. These problem points are highlighted by the white rectangles around them. In these rectangles, multiple apertures of varying lengths were discovered. These apertures could result in a complete loss of shielding effectiveness for both single and split shields. In the future, in order to more accurately test shielding, unnecessary apertures will need to be completely sealed.

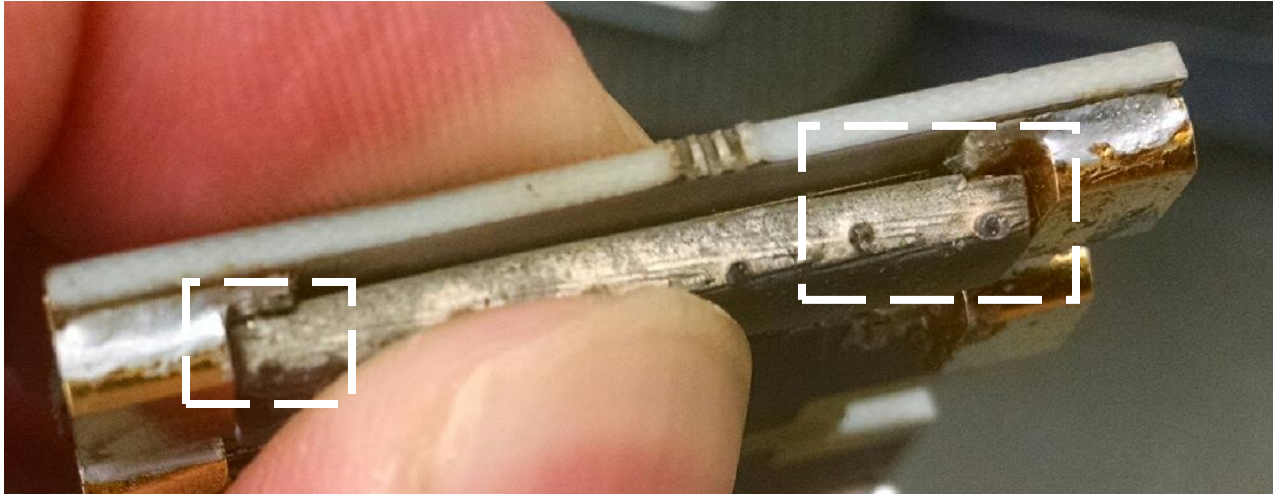


Figure 56: Possible aperture locations on test board

6.6 – Conclusion

While simulation results suggested that separately shielding input and output lines on surface components could increase isolation by approximately 20dB, experimental results do not agree. However, it was discovered that the 3d printed shields for the test board had multiple apertures rendering the shielding ineffective. In order to effectively test the possible isolation benefits of split shielding, apertures must be either completely eliminated or significantly minimized. In addition, finding a better way to affix the 3d printed shields to a PCB board must be developed in order for 3d printed shields to work.

Chapter 7 - Conclusions and Future Directions

7.1 – Conclusions

During this research several important guidelines regarding crosstalk and isolation were discovered. For every doubling of distance between signal and ground plane, isolation will decrease between 6-19dB depending on whether inductive or capacitive coupling dominates. In order for stripline to be effective, special care must be given when designing via transitions as the via transition can reduce the isolation of stripline to that of microstrip. For isolation requirements >60dB, it is recommended that direct launch stripline be utilized and any openings be completely shielded.

Several promising new isolation techniques were also explored in addition to direct launch stripline. For designs seeking to balance isolation requirements with those of board consumption and design flexibility, the newly researched transmission line structure of SS-CPW may be of use. SS-CPW provides 7dB of improvement when compared to microstrip. Due to utilizing a single ground flank, board consumption is reduced relative to traditional CPW and more complex structures such as stubs can easily be added.

Finally, dielectric ablation of the PCB surface coupled with RF absorber material also increased isolation. It was discovered that when a thick absorber material was used, crosstalk could be reduced by as much as 9dB for frequencies greater than 16GHz. It was also shown that the removal of dielectric material around the transmission lines did not have a significant effect on the characteristic impedance.

7.2 – Future Work

Possible future work could include exploring the effect that ground flank width of CPW has on isolation. Currently, general practice is to ground flood the top layer of PCB when using

CPW. However, use of thinner ground flanks for CPW did not seem to have a negative effect on isolation. With the knowledge that at high frequencies surface waves form, separate thinner ground flanks may in fact increase isolation.

For single-sided CPW to be utilized more readily, more work needs to be done to characterize it. Specifically, design rules and equations need to be developed to allow for the design of specific characteristic impedances without simulations. Finally, the effect SS-CPW has on more complex filter structures needs to be explored.

Dielectric ablation presents several possible areas of future research. One area is to measure the effect that a full trench would have on isolation. A possible second area of research would be to explore the effect that dielectric ablation and selective placement of absorber material would have on the directionality and crosstalk of patch antenna arrays.

Finally, more research should be done on the utilization of RF split shields. Current experimental results do not confirm the isolation benefit suggested by simulation. However, after examination of the 3d printed shields, the shields appeared to suffer from aperture issues that may have overpowered any benefits split shielding might have provided. In conjunction, further research should also be conducted on the design and construction of 3d printed shielding for RF applications in general.

References

- [1] Rider, T. (2017). *Crosstalk and EMI on microwave circuit boards*. Master of Science. Kansas State University.
- [2] J. E. Zumbro, H. A. Mantooth, W. B. Kuhn, J. Welch, T. Rider and J. A. Wolf, "Board-level rapid-prototyping shielding for radio frequency transmission lines," *2018 IEEE Radar Conference (RadarConf18)*, Oklahoma City, OK, 2018, pp. 1528-1532.
- [3] E. Bogatin, *Signal and power integrity, simplified*. Boston: Prentice Hall, 2010.
- [4] T. Rider, W. B. Kuhn and A. Wolf, "Crosstalk and EMI in mixed-signal / microwave multi-layer pc boards," *2017 IEEE International Symposium on Electromagnetic Compatibility & Signal/Power Integrity (EMCSI)*, Washington, DC, 2017, pp. 26-31.
- [5] J. Coonrod, "Comparing Microstrip and Grounded Coplanar Waveguide," *Microwave Journal*, 16-Nov-2017.
- [6] J. Coonrod, B. Rautio, "Comparing microstrip and CPW performance", *Microwave Journal*., vol. 55, no. 7, pp. 74-86, Jul. 2012.
- [7] A. Sain and K. L. Melde, "Impact of Ground via Placement in Grounded Coplanar Waveguide Interconnects," in *IEEE Transactions on Components, Packaging and Manufacturing Technology*, vol. 6, no. 1, pp. 136-144, Jan. 2016.
- [8] F. D. Paolo, "Asymmetric Coplanar Strip Analysis Including Conductor Backing," *Microwave Journal*, 17-Jan-2012
- [9] H. W. Ott, *Noise Reduction Techniques in Electronic Systems*. New York: John Wiley & Sons, 1988.
- [10] F. A. Po'ad, M. Z. M. Jenu, C. Christopoulos and D. W. P. Thomas, "Estimation of Electric and Magnetic Shielding Effectiveness of a Metallic Enclosure with Apertures," *2006 International RF and Microwave Conference*, Putra Jaya, 2006, pp. 291-295.
- [11] M. Sadiku, *Elements of Electromagnetics*, New York: Oxford University Press, 2015.
- [12] J. R. Reid, E. D. Marsh and R. T. Webster, "Micromachined rectangular-coaxial transmission lines," in *IEEE Transactions on Microwave Theory and Techniques*, vol. 54, no. 8, pp. 3433-3442, Aug. 2006.
- [13] C. M. Weil and L. Gruner, "High-Order Mode Cutoff In Rectangular Striplines (Short Papers)," in *IEEE Transactions on Microwave Theory and Techniques*, vol. 32, no. 6, pp. 638-641, Jun. 1984.

- [14] M. Leib, M. Mirbach and W. Menzel, "An ultra-wideband vertical transition from microstrip to stripline in PCB technology," *2010 IEEE International Conference on Ultra-Wideband*, Nanjing, 2010, pp. 1-4.
- [15] Shuo Lei, Y. X. Guo and L. C. Ong, "CPW to stripline transitions in LTCC for millimeter-wave applications," *2005 Asia-Pacific Microwave Conference Proceedings*, Suzhou, 2005, pp. 3 pp.-.
- [16] W Kuhn, G Peterson and J Welch, "Broadband Antenna Probe for Microwave EMC Measurements," *27th Conference on Electrical Performance of Electrical Packaging and Systems*, 2018, pp. 199-201
- [17] J Welch and W Kuhn, "Single Sided Coplanar Waveguide," *27th Conference on Electrical Performance of Electrical Packaging and Systems*, 2018, pp. 131-133
- [18] A. A. Oliner, "Leaky waves: basic properties and applications," *Proceedings of 1997 Asia-Pacific Microwave Conference*, Hong Kong, 1997, pp. 397-400 vol.1.