

Insertion Loss Comparisons of Common High Frequency PCB Constructions

John Coonrod, Rogers Corporation

Printed Circuit Boards (PCB's) have been used for many years in low loss, high frequency microwave applications and many of these circuits have become increasingly complex. Often these complex circuits are just combinations of much simpler microwave PCB constructions and understanding the basic structures can be very advantageous for the PCB designer and fabricator. Each of these microwave structures have different loss mechanisms and this paper will focus on the three most common microwave PCB structures and their loss properties. The three structures are: microstrip, coplanar and stripline. Initially a basic overview of relative electromagnetic concepts will be given and following will be comparisons regarding loss performance of the different structures, using electromagnetic modeling software and measured data.

The science of electromagnetics is extremely vast and as related to microwave PCB technology it is generally thought of as dynamic electromagnetics of parallel plate structures. This focused field of electromagnetics is also very deep and diverse and the following discussion will be tailored to the concepts which affect loss for parallel plate technology.

The total loss of a transmission line is called insertion loss and is the summation of several other losses. A simplified view of these losses follows:

$$\alpha_T = \alpha_C + \alpha_D + \alpha_R + \alpha_L \quad (1)$$

The total loss (insertion loss, α_T) is made up of conductor (α_C), dielectric (α_D), radiation (α_R) and leakage losses (α_L). Leakage losses are typically associated with semiconductor grade materials and are normally not an issue for microwave PCB circuits. The materials used in PCB technology generally have very high volume resistance and due to that property, leakage losses are not a concern. There are exceptions, however for the scope of this paper leakage losses will be dismissed.

Radiation loss is how much energy is lost from the circuit to the surrounding environment or how much energy is radiated off of the transmission line. Radiation loss is design dependent and some circuit designs are more or less susceptible to this type of loss. These losses will be worse at impedance transitions as well as signal launch areas. Radiation losses are frequency, dielectric constant (Dk) and thickness dependent. For a given transmission line circuit with radiation loss issues, the loss will be much higher at higher frequencies. The same circuit, when using a thinner substrate, will have less radiation loss. Finally the Dk can alter the radiation loss and a substrate with a higher Dk will develop less radiation loss.

The microstrip transmission line is susceptible to radiation loss and this will be discussed in more detail later; however there is a simple mathematical expression derived^[1] for radiation loss which may assist in understanding the different tradeoffs:

$$\alpha_r = 60 \left(\frac{2\pi h}{\lambda_0} \right)^2 F(eff) \quad (2)$$

$$F(eff) = 1.0 - \frac{\epsilon_{eff} - 1}{2.0 \sqrt{\epsilon_{eff}}} \log \left(\frac{\sqrt{\epsilon_{eff}} + 1.0}{\sqrt{\epsilon_{eff}} - 1.0} \right) \quad (3)$$

The symbols α_r , h , λ_0 and ϵ_{eff} are the radiation loss, height (thickness) of the substrate, wavelength and the effective dielectric constant respectively.

Dielectric losses are generally associated with the dissipation factor (Df) of the substrate used to make the PCB. In some cases the addition of a soldermask can increase dielectric loss as well as any other dielectric additive to a circuit. As a general reference, most high frequency circuit materials are considered low loss and have a Df number less than 0.005.

Conductor loss has many variables and can be difficult to properly consider. Skin depth is a contributor to conductor loss and that is the amount of conductor which is used by the electrical current. At DC, 0 Hz, the electric current will use the entire cross-sectional area of a wire. As the frequency increases, the current will only use the outer skin of the wire and at microwave frequencies the skin depth is often much less than 2 microns (80 micro-inches). As an example, the skin depth in a copper conductor at 1 GHz is 2.08 microns (82 micro-inches) and at 10 GHz is 0.66 microns (26 micro-inches). The mathematical relationship for skin depth is given:

$$\delta = \sqrt{\frac{1}{\pi \cdot f \cdot \mu \cdot \sigma}} \quad (4)$$

The symbols for skin depth (δ) are f , frequency, μ is permeability and σ is conductivity. Permeability is the property of a material to alter magnetic fields and most dielectrics used in the PCB industry have the permeability of free space. There are some metals used as plated finish for circuits which have a much high permeability and that would reduce the skin depth. Additionally it can be seen in equation 4 that the conductivity of the metal can influence the skin depth and if the metal is lower in conductivity (more resistive) then the skin depth will increase.

Another issue related to conductor loss is the surface roughness of the conductor. When a conductor surface is rough this will cause a longer wave propagation path and will create more losses. The losses are related to parasitic inductance due to surface inductance of current following in partial loops^[2] in the metal profile when viewed at high magnification. Essentially when the skin depth is approximately the same dimension as the copper surface roughness profile, the roughness will have a significant impact on the conductor losses.

The three common microwave transmission line circuits to be discussed are shown in figure 1.

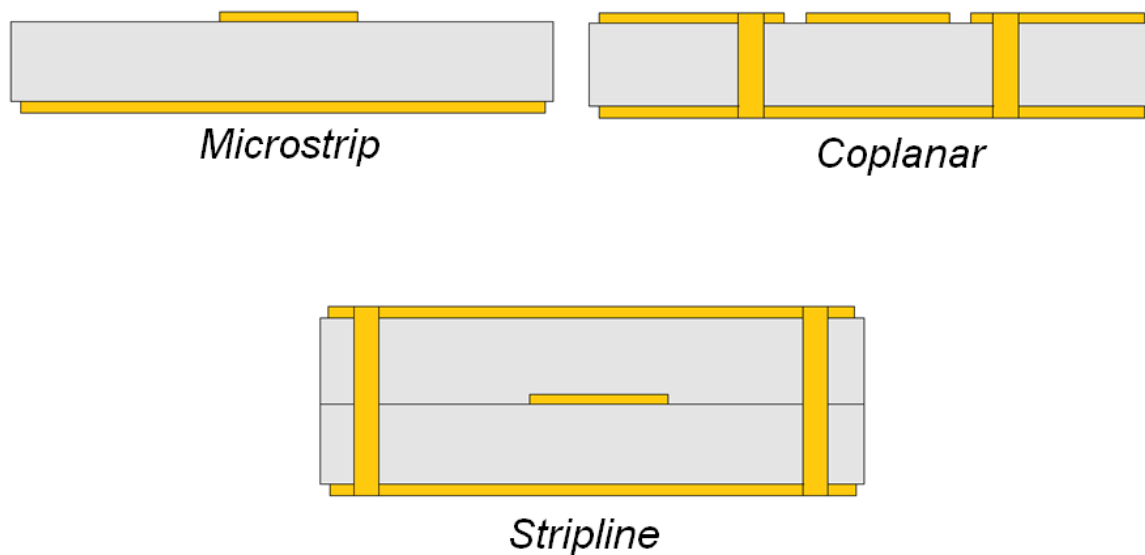


Figure 1. Cross-sectional view of three common transmission line circuits

A microstrip transmission line, shown in the upper left corner of figure 1, is a simple two copper layer circuit with the signal conductor on top and a ground plane on the bottom. This structure is probably the most common microwave PCB structure used, however, it is often the outer layer of a multilayer PCB circuit.

Microstrip circuits have the three loss mechanisms, previously discussed, which make up the total insertion loss. Each of these losses will be more or less significant depending on the thickness of the circuit. For a thin circuit, the conductor losses will dominate and if it is a rough copper the conductor losses will be very significant.

If the microstrip circuit uses a thicker substrate, the conductor losses will be less impactful. An example of how these different microstrip losses sum to the composite insertion loss, when using the same material and at different thicknesses, is shown in figure 2.

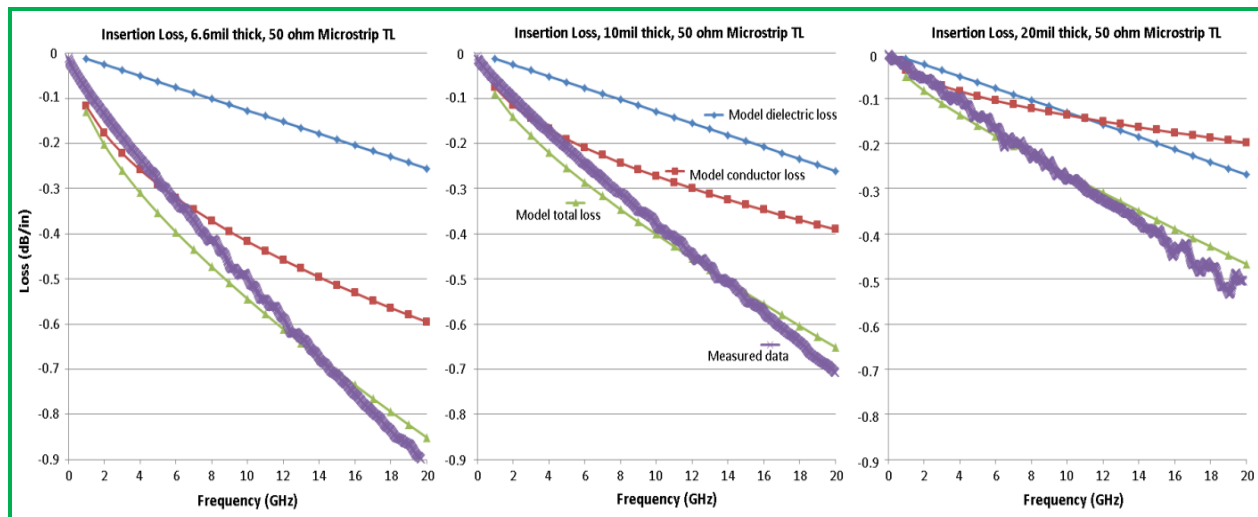


Figure 2. 50 ohm microstrip transmission line insertion loss , showing the different components of the insertion loss, when using the same material at different thicknesses.

The legend for all three curves is shown on the middle curve and that is a transmission line built on a 10mil thick RO4350B™ substrate. The same substrate was used for each of these curves and the model used in the curve generations is a free software from Rogers Corporation's Technology Support Hub website called MWI-2010. The curve on the far right is using the thickest substrate (20mil) and has the lowest insertion loss. The 20mil thick circuit also shows the dielectric loss and conductor loss to be very similar. In contrast the curve on the far left is the thinnest circuit, has the highest insertion loss and the conductor loss dominates. It can also be seen across all three curves that the dielectric loss does not change much with thickness. This is indicative of a stable low loss, high frequency circuit material.

In figure 2 the radiation losses were ignored however to demonstrate the effects of these losses regarding thickness and frequency dependency, figure 3 has a modified model which includes radiation loss.

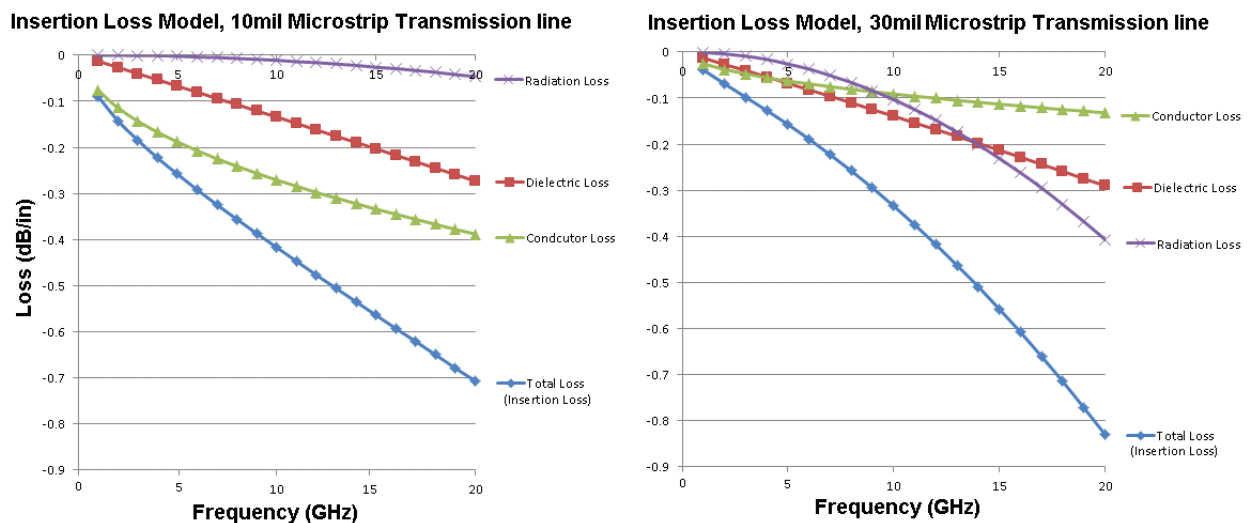


Figure 3. Radiation losses included for the 10mil and 30mil thick microstrip transmission line circuit.

It can be seen on the left chart that the 10mil thick circuit model of total insertion loss compares very accurately to figure 2 of the same circuit, with the measured data performance of an actual circuit. Figure 3 shows very clearly that radiation losses

are minimal with the thinner circuit at 20 GHz, however, the curve of the thicker circuit shows radiation losses to be significant at the same frequency.

There are several different types of coplanar circuit constructions, although when used at microwave frequencies, the most commonly used circuit is the conductor backed coplanar waveguide (CBCPW). These circuits are also called Grounded Coplanar Waveguide (GCPW) and offer improved performance over microstrip at higher frequencies. The losses of a CBCPW are usually worse at lower frequencies than a microstrip, however when designed correctly, the coplanar circuit will have nearly no radiation losses at high frequencies.

Coplanar circuits are similar to microstrip circuits in construction, where they are both two copper layer circuits. The CBCPW circuit shown in the upper right of figure 1 has plated through hole via's to electrically connect the top ground planes to the bottom ground plane. The pitch along the length of the conductor run of these grounding via's is extremely important. If the via's are not close enough together there will be an increase in parasitic parallel plate inductance and that will raise the impedance as well as give an erratic insertion loss curve. An excellent paper^[3] which addresses the effects of grounding via's in CBCPW suggest a pitch distance of $1/8$ wavelength of the highest operating frequency.

The coupling between the signal conductor and the adjacent ground planes can have an impact on conductor loss as it is related to copper roughness. As previously discussed, a conductor with a high profile will cause more conductor losses for a microstrip and because of that some material suppliers have offered the same high frequency laminate with smoother copper. The smooth copper will reduce the insertion loss on microstrip and CBCPW, however the benefits will be less realized on the CBCPW. The amount of benefit from the smooth copper will vary for the CBCPW for two reasons. One is the fact that when the coupling is tight, or a narrow space between the ground-signal-ground on the top layer, there will be more electric fields in the air between the coupled features and less at the base of the conductors where the roughness has an effect on loss. The second item is the normal trapezoidal effects of the conductor from the circuit pattern etching process. When the trapezoidal effects are significant and the ground-signal-ground conductor walls are slanted away from each other, the current density and the electric fields move down near the base of the conductor and then the roughness will have more impact on losses.

A study was done comparing microstrip and CBCPW transmission lines with different copper types, which had significantly different surface roughness. It was found that the microstrip showed more benefit than the CBCPW when comparing a smooth copper to a rough copper surface for insertion loss. The results are summarized in figure 4.

Insertion Loss Comparisons between microstrip and CBCPW with different copper surface roughness All circuits are 50 ohm transmission lines

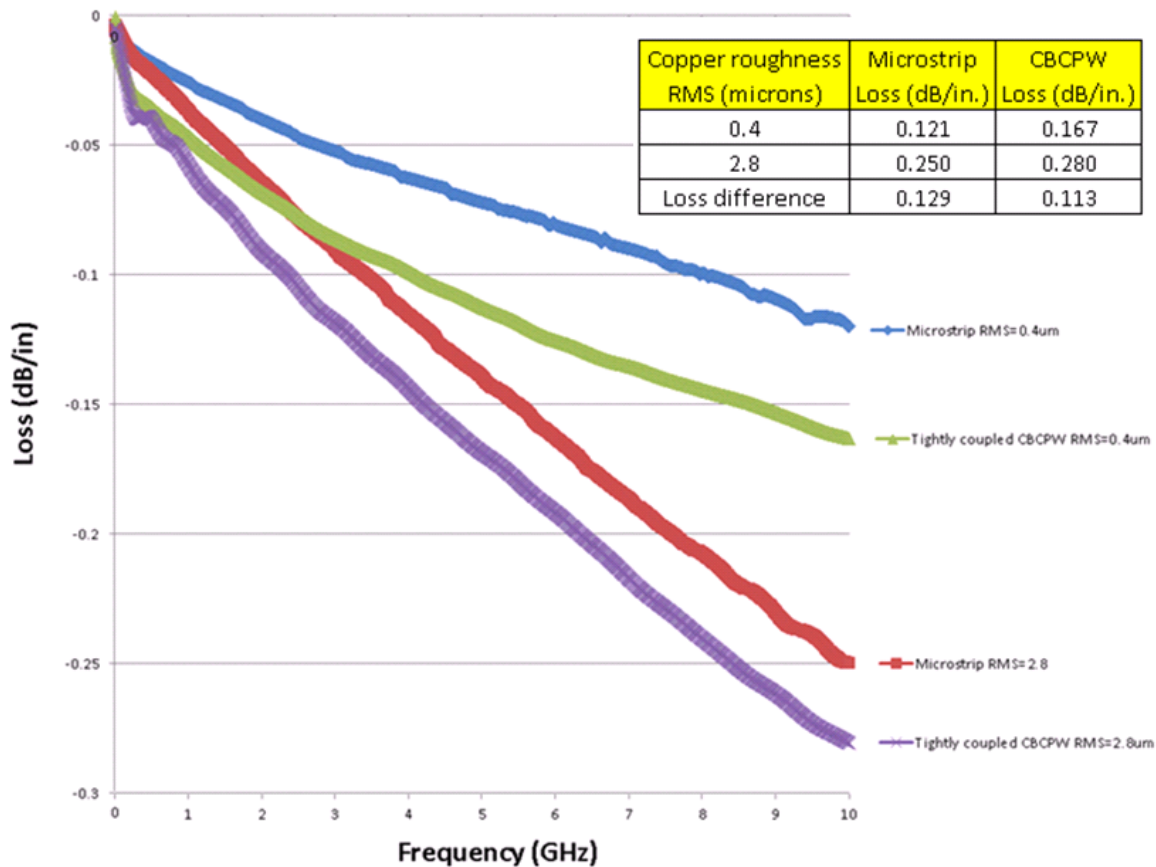


Figure 4. Comparison of smooth (RMS=0.4μm) vs rough (RMS=2.8μm) copper surface using microstrip and CBCPW transmission line circuits.

The insertion loss difference for the microstrip circuits shown in figure 4 have more improvement due to the smooth copper, as compared to the CBCPW circuits. The coplanar circuits had very little trapezoidal effects which means the difference between the smooth and rough copper were less significant than the microstrip and if the trapezoidal effects were more prevalent then the loss difference would have been more obvious between the smooth and rough copper.

Coplanar circuits have many benefits, however the trapezoidal effects can confuse the conductor loss issue associated with copper surface roughness. More information on this topic can be found from a technical article^[4] where this study was conducted in more detail.

Stripline circuits are used extensively in the microwave PCB applications. These are three copper layer circuits as shown in the bottom drawing of figure 1. The signal layer is buried with dielectric material on top and bottom of it. The outer two ground layers are electrically connected by the use of plated through hole via's and there is the same concern for grounding via pitch as stated for the CBCPW circuits.

Stripline circuits have more loss than microstrip and typically more than CBCPW. The reason stripline is used is there are no radiation losses and there is perfect isolation of the signal conductor from other electrical influences, when designed correctly. Stripline circuits are capable of very high frequency applications, however the conductor loss can be difficult to understand regarding copper surface roughness.

In the case of stripline, there are four copper-substrate interfaces and depending on the fabrication method, these interfaces will typically have very different surface roughness. If the stripline circuit is made from a laminate for the bottom two

conductive layers and prepreg-copper-foil for the top layer, the copper surface roughness of the different layers will be different. The bottom laminate will have the same copper surface roughness at the bottom copper-substrate interface and the next interface up from there. However that same copper layer on the top side will have a much smoother surface. The last copper-substrate interface for the top copper layer is dependent on what type of copper was used to bond the copper foil with the prepreg.

Both microstrip and CBCPW circuits have some benefit over stripline from using air, because air has the lowest possible loss. In the case of stripline, the dielectric surrounds the signal conductor and that material can have an impact on the benefits of smooth copper when comparing copper of different surface roughnesses. Using materials with a higher Df, the dielectric losses will be a higher percentage of the overall insertion loss, and any improvement to the conductor loss due to copper surface roughness is less significant. Figure 5 shows several models of stripline circuits using low-loss and mid-loss materials combined with rough and smooth copper surfaces. The low-loss materials have a Df of 0.0037 and the mid-loss material has a Df of 0.0090.

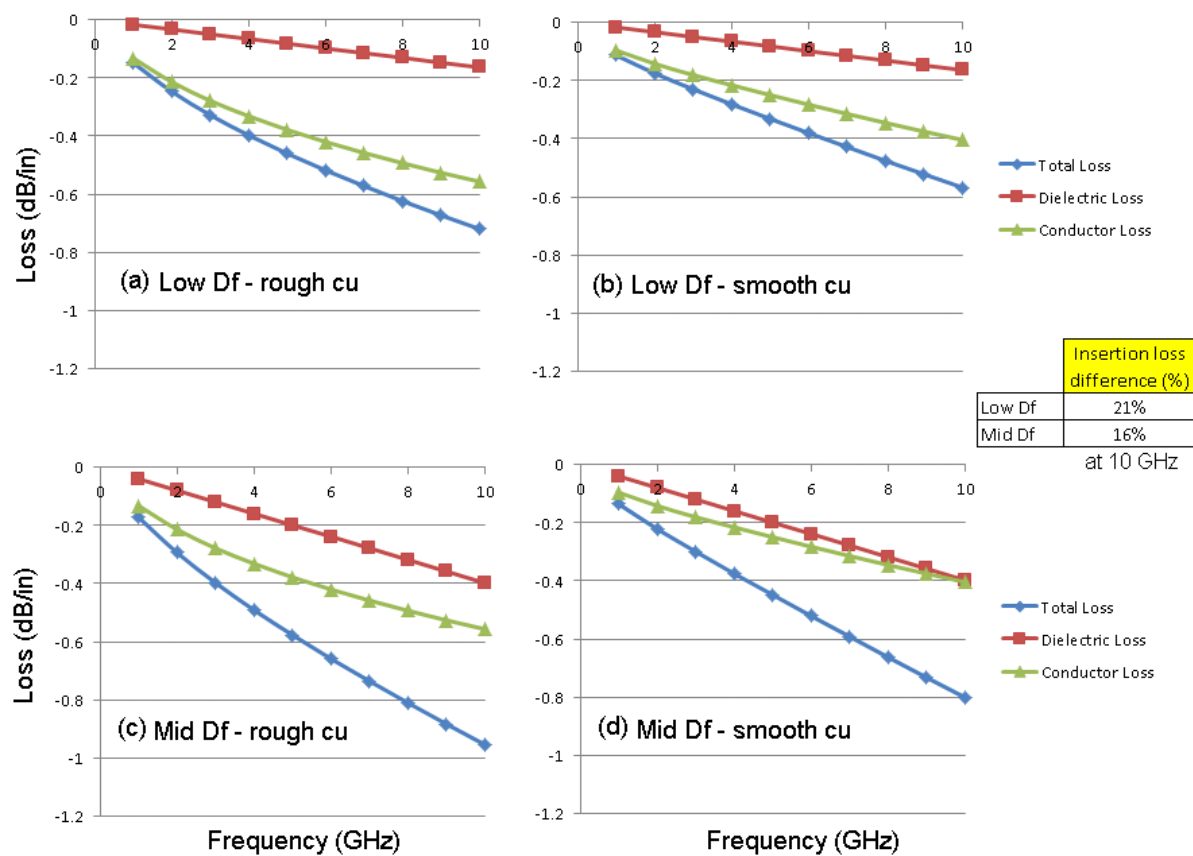


Figure 5. Stripline model comparisons of insertion loss differences when using smooth and rough copper and materials that are low-loss Df and mid-loss Df.

In figure 5 it can be seen that the low-loss material shows a greater improvement in insertion loss due to the same change from copper being rough to smooth. The percentage difference in insertion loss is shown only to demonstrate the magnitude of the difference in loss, however percentage it not a valid comparison for insertion loss differences and is best done in reference to dB or power level differences.

The three most common microwave PCB structures have been shown to have insertion loss which is made up of different components. Understanding the different components of insertion loss for each of these transmission line circuits enables the designers to make better choices about loss budgets. The properties of high frequency circuit materials are a key consideration and it is highly recommended for the designer to consult their material supplier for new microwave applications.

References

- [1] B. C. Wadell, "Transmission Line Design Handbook," ARTECH HOUSE, INC., p. 99, 1991
- [2] A.F. Horn, J.W. Reynolds, J.C. Rautio, "Conductor Profile Effects on the Propagation Constant of Microstrip Transmission Lines", *IEEE Transactions on Microwave Theory and Techniques*, 2010
- [3] B. Rosas, "Optimizing Test Boards for 50 GHz End Launch Connectors: Grounded Coplanar Launches and Through Lines on 30-mil Rogers RO4350B with Comparison to Microstrip," Southwest Microwave, Inc., Tempe, AZ, 2007, www.southwestmicrowave.com.
- [4] J. Coonrod, B. Rautio, "Comparing Microstrip and CPW Performance", *Microwave Journal*, July 2012.

RO4350B is a licensed trademark of Rogers Corporation

Insertion Loss Comparisons of Common High Frequency PCB Constructions

*John Coonrod
Rogers Corporation
Advanced Circuit Materials Division*

Insertion Loss Comparisons of Common High Frequency PCB Constructions

Topics

- Basic Electromagnetic Concepts regarding PCB Losses
- Loss overview of the most common Transmission Line Circuits
 - Microstrip
 - Coplanar
 - Stripline

Insertion Loss Comparisons of Common High Frequency PCB Constructions

Basic Electromagnetic Concepts regarding PCB Losses

$$Loss_{Total} = Loss_{Conductor} + Loss_{Dielectric} + Loss_{Radiation} + Loss_{Leakage}$$

- Total loss of a transmission line is Insertion Loss
- Insertion loss is made up of different types of losses
- Due to high volume resistivity most PCB materials do not have issues with leakage loss
- In general:
 - Radiation loss is design specific
 - Dielectric and conductor loss are material specific
 - There are fabrication issues which can alter these losses

Insertion Loss Comparisons of Common High Frequency PCB Constructions

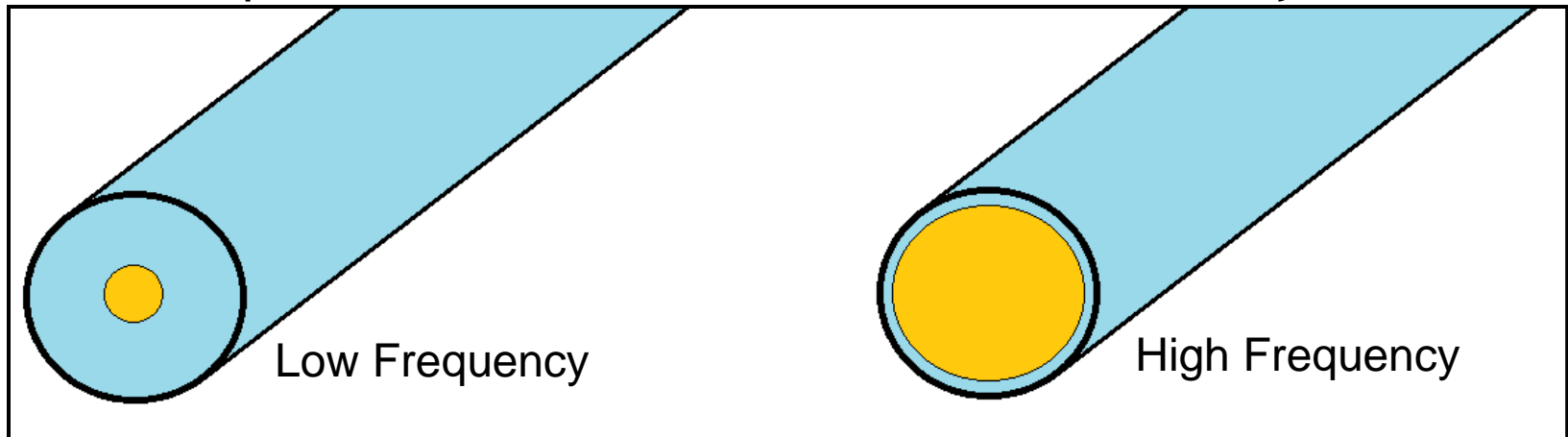
Basic Electromagnetic Concepts regarding PCB Losses

At DC or 0 Hz frequency the entire conductor is used by the electric current

For AC the outer skin of the conductor is used by the current

At higher frequencies less of the conductor is used by the current

Wire example, where blue is the amount of wire used by the current



Insertion Loss Comparisons of Common High Frequency PCB Constructions

Basic Electromagnetic Concepts regarding PCB Losses

A skin depth (δ) for a microwave circuit is typically very thin

example 1: 1 GHz has a skin depth in copper of 0.082mils (82 micro-inches)

example 2: 10 GHz has a skin depth of 0.026 mils (26 micro-inches)

$$\delta = \sqrt{\frac{2}{\omega \mu \sigma}} \quad \text{or} \quad \delta = \sqrt{\frac{1}{\pi f_{\text{freq}} \mu \sigma}} \quad \begin{array}{l} \sigma \text{ is conductivity} \\ \mu \text{ is permeability (magnetic property)} \end{array}$$

With lower conductivity (more resistive), there will be an increase in skin depth

Materials with higher permeability will decrease the skin depth

Insertion Loss Comparisons of Common High Frequency PCB Constructions

Basic Electromagnetic Concepts regarding PCB Losses

Conductor loss:

Skin depth is a component of conductor loss

Due to the effects of skin depth on conductor loss, this loss is frequency dependent

When the skin depth is near the same dimension as the copper surface roughness, the roughness will increase conductor loss

Dielectric loss:

The dissipation factor (Df) of the substrate is the major contributor for dielectric loss

Soldermask and other additive materials can add to dielectric loss

Radiation loss:

Radiation loss is the energy dissipated from the circuit

Some circuit designs are more prone to radiation loss

There is more radiation loss at higher frequencies, thicker substrates and lower Dk

Insertion Loss Comparisons of Common High Frequency PCB Constructions

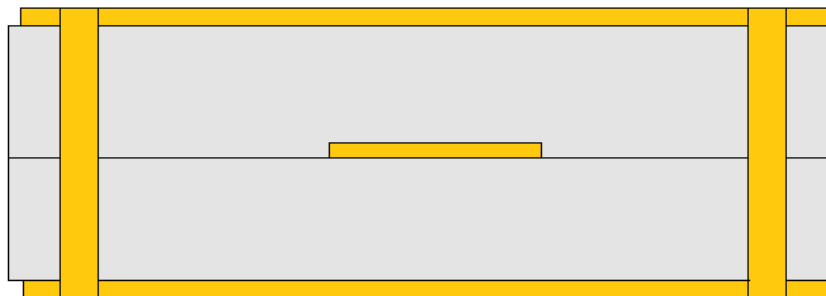
Loss overview of the Most Common Transmission Line Circuits



Microstrip



Coplanar



Stripline

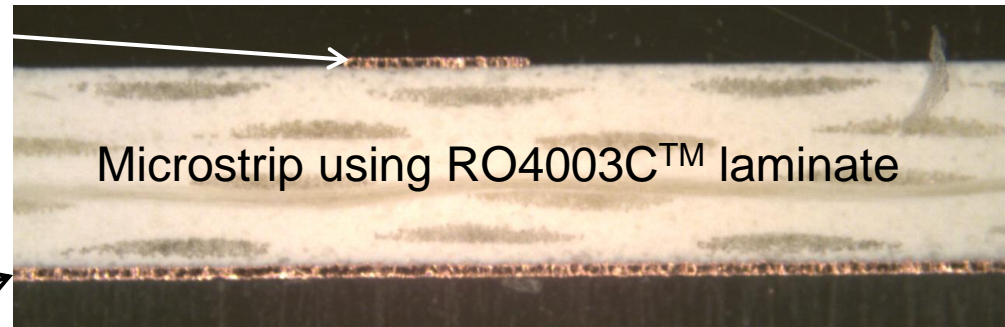
Insertion Loss Comparisons of Common High Frequency PCB Constructions

Loss overview of the Most Common Transmission Line Circuits

Microstrip Losses

Signal conductor

Ground plane



Microstrip using RO4003C™ laminate

Microstrip has dielectric loss, conductor loss and radiation loss possible

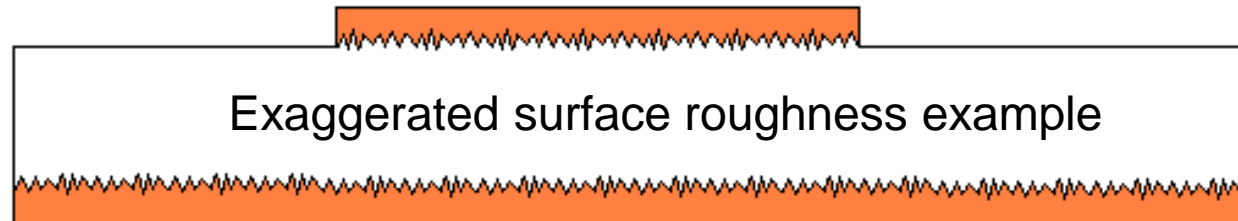
Dielectric loss is mostly due to dissipation factor (Df)

If soldermask is added to the signal layer, the Df associated with the soldermask will increase the dielectric constant

Insertion Loss Comparisons of Common High Frequency PCB Constructions

Loss overview of the Most Common Transmission Line Circuits

Microstrip Losses



Conductor losses are dependent on:

- frequency
- thickness of substrate
- copper surface roughness

Conductor losses will increase when a finish is applied to the copper

- Enig (electroless nickel / immersion gold), nickel is less conductive than copper
- Tin, solder are less conductive than copper

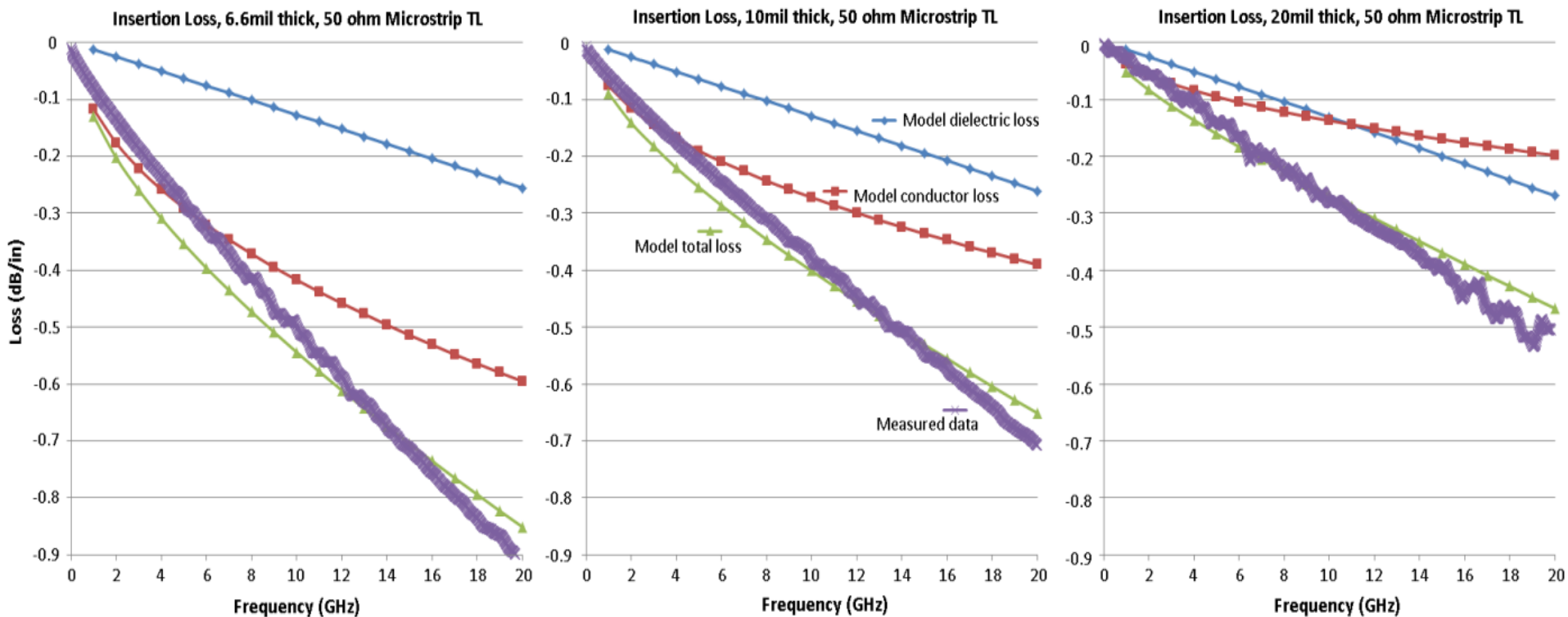
Silver is more conductive than copper, however silver used in PCB industry is an alloy and not pure silver, extremely thin and the skin depth is more shallow than copper

Insertion Loss Comparisons of Common High Frequency PCB Constructions

Loss overview of the Most Common Transmission Line Circuits

Microstrip Losses

Example of three microstrip transmission lines, exactly the same, except using different substrate thickness



Insertion Loss Comparisons of Common High Frequency PCB Constructions

Loss overview of the Most Common Transmission Line Circuits

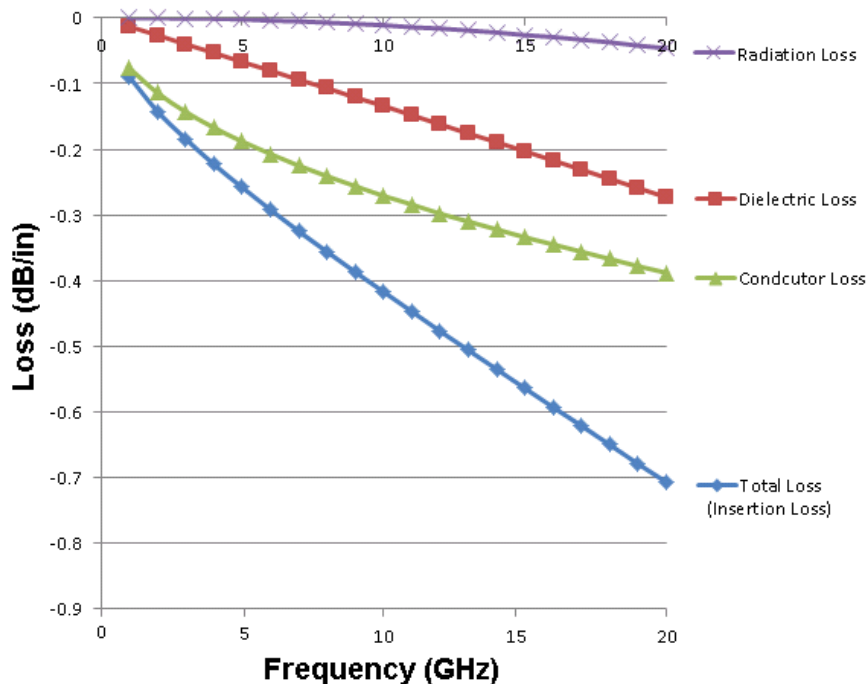
Microstrip Losses

Microstrip can have more or less radiation loss due to the circuit design

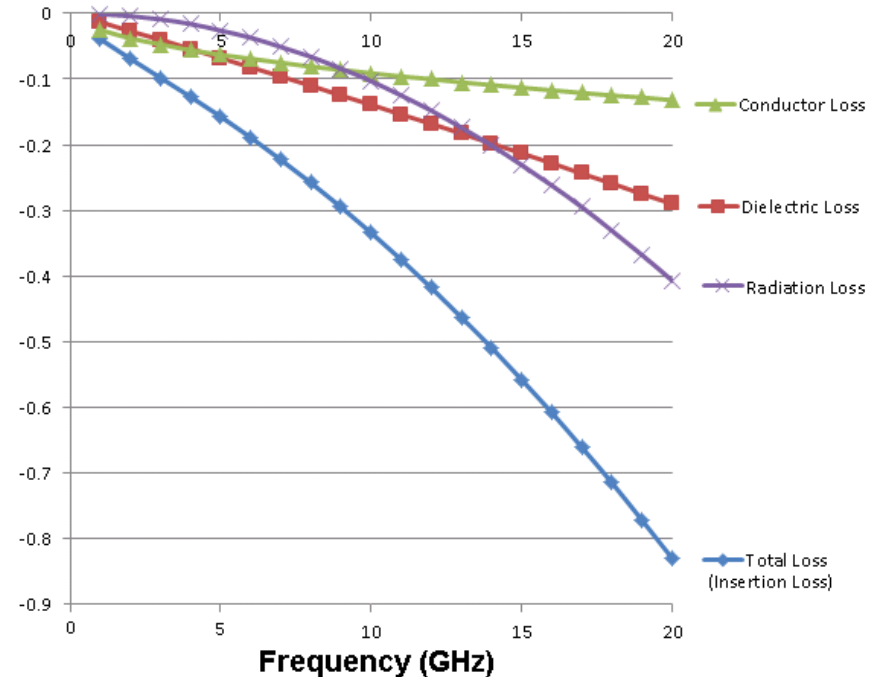
An impedance transition will generate some radiation loss

Radiation losses are frequency, Dk and thickness dependent

Insertion Loss Model, 10mil Microstrip Transmission line



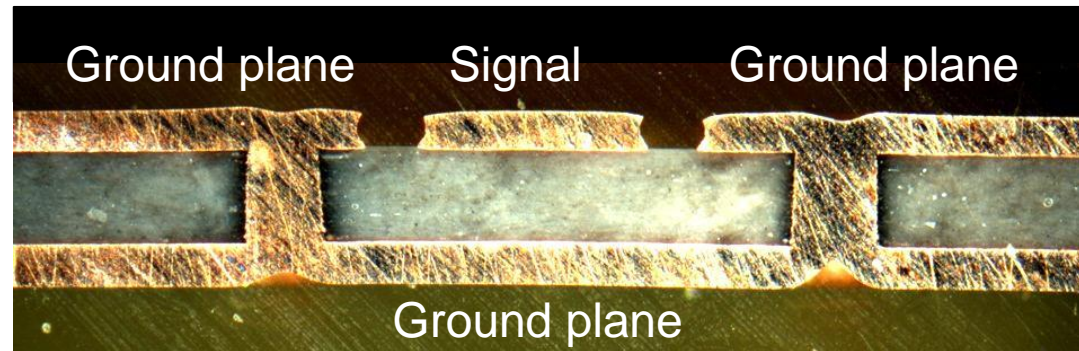
Insertion Loss Model, 30mil Microstrip Transmission line



Insertion Loss Comparisons of Common High Frequency PCB Constructions

Loss overview of the Most Common Transmission Line Circuits

Coplanar Losses



There are many different types of coplanar constructions

Conductor backed coplanar wave guide (CBCPW) or grounded coplanar waveguide (GCPW), are most often used at microwave frequencies for PCB technology

CBCPW:

- have more conductor losses than microstrip

- have much less radiation losses than microstrip

- grounding via pitch is critical for loss considerations

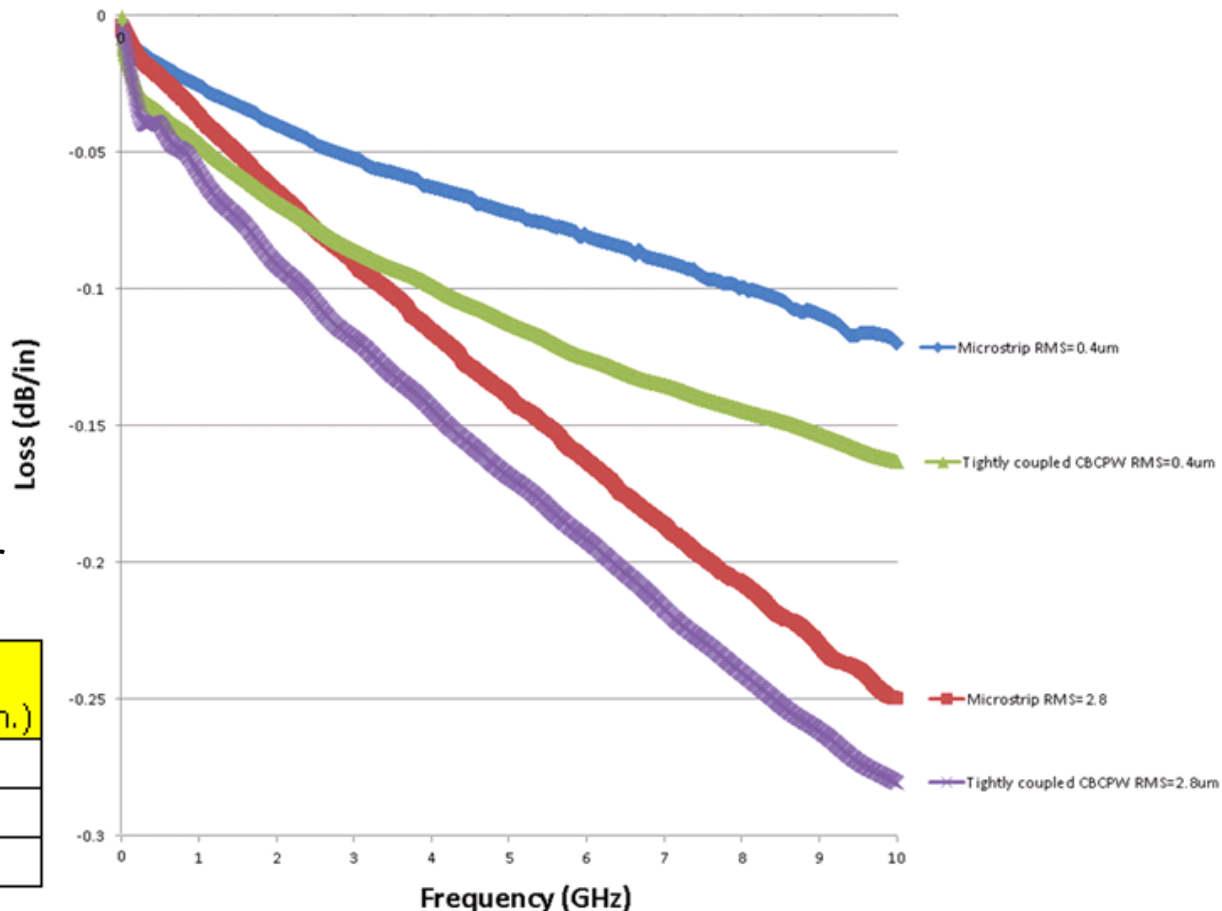
- losses are much more sensitive to conductor trapezoidal effects

- coupling (ground-signal-ground spacing) varies loss effect due to roughness

Insertion Loss Comparisons of Common High Frequency PCB Constructions

Loss overview of the Most Common Transmission Line Circuits

Insertion Loss Comparisons between microstrip and CBCPW
with different copper surface roughness
All circuits are 50 ohm transmission lines



*Coplanar
Losses*

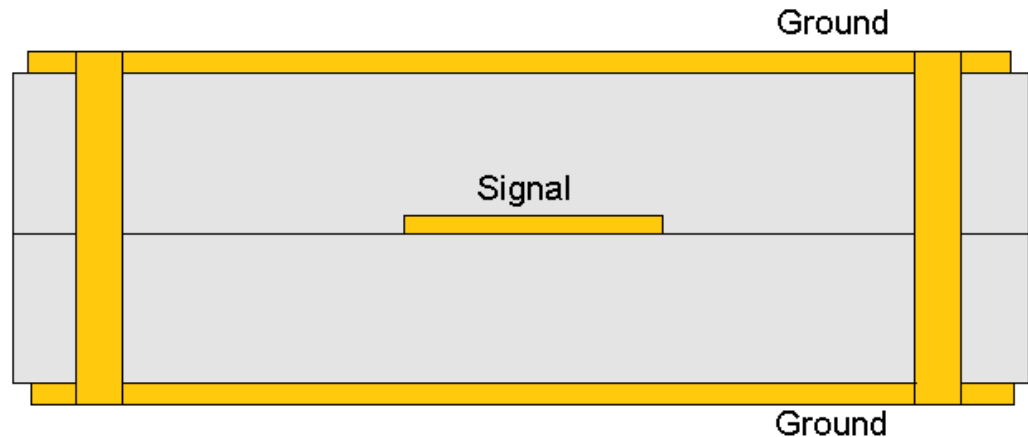
Differences in loss due to copper surface roughness @ 10GHz

Copper roughness RMS (microns)	Microstrip Loss (dB/in.)	CBCPW Loss (dB/in.)
0.4	0.121	0.167
2.8	0.250	0.280
Loss difference	0.129	0.113

Insertion Loss Comparisons of Common High Frequency PCB Constructions

Loss overview of the Most Common Transmission Line Circuits

Stripline Losses



Stripline losses are dielectric losses and conductor losses

If designed correctly, there is no radiation losses with stripline

The pitch distance for the grounding via's is critical for good loss performance

Conductor losses due to copper surface roughness is difficult to understand

- There are 4 copper-substrate interfaces
- Typically these interfaces will have different copper roughness

Dielectric losses are complicated when using different materials or with unbalanced stripline

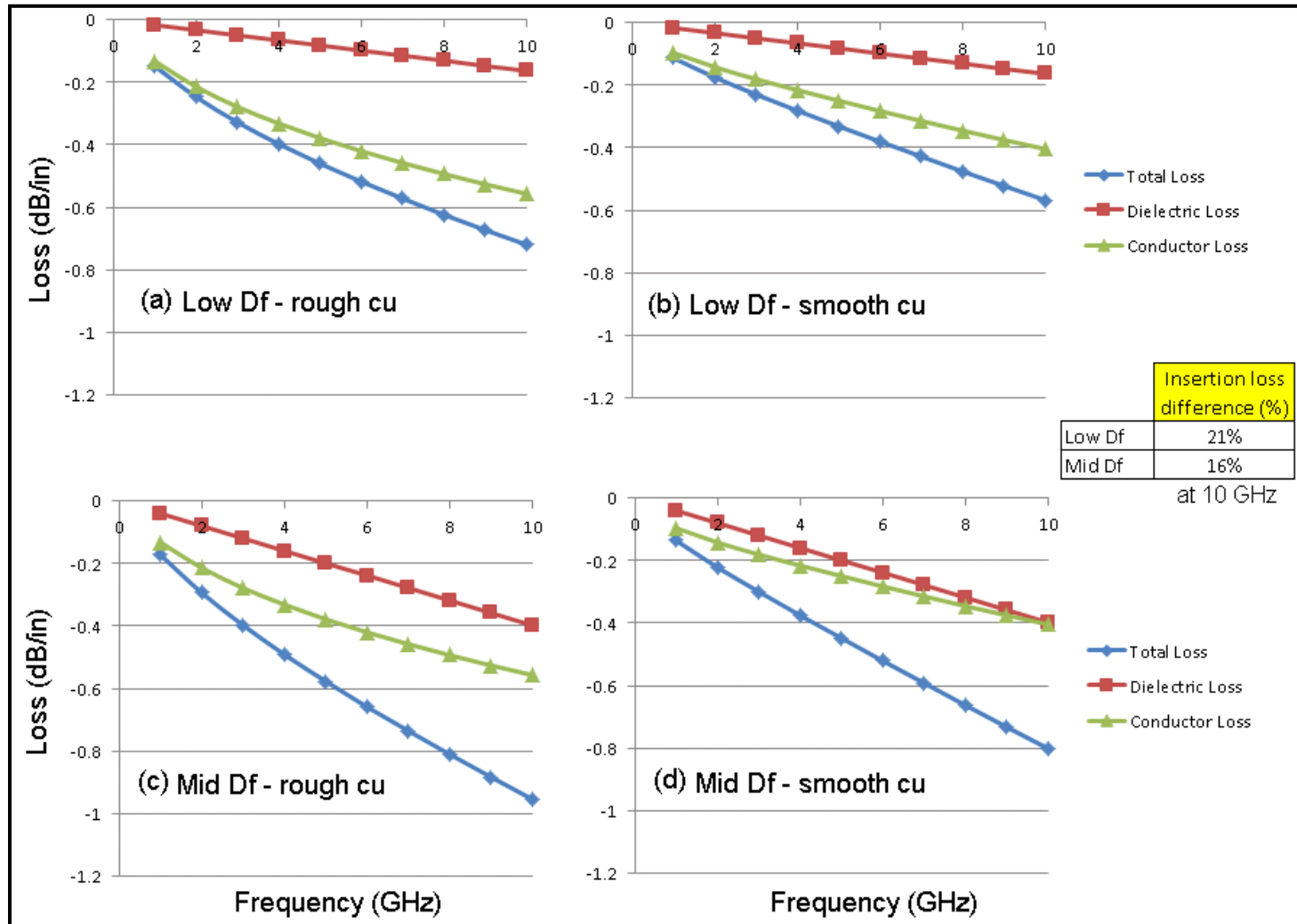
Insertion Loss Comparisons of Common High Frequency PCB Constructions

Loss overview of the Most Common Transmission Line Circuits

Stripline Losses

The loss difference between smooth and rough copper is less significant on stripline as compared to microstrip

Df has a large effect on benefits of smooth copper



Insertion Loss Comparisons of Common High Frequency PCB Constructions

Thank You

References

- [1] B. C. Wadell, "Transmission Line Design Handbook," ARTECH HOUSE, INC., p. 99, 1991
- [2] A.F. Horn, J.W. Reynolds, J.C. Rautio, "Conductor Profile Effects on the Propagation Constant of Microstrip Transmission Lines", *IEEE Transactions on Microwave Theory and Techniques*, 2010
- [3] B. Rosas, "Optimizing Test Boards for 50 GHz End Launch Connectors: Grounded Coplanar Launches and Through Lines on 30-mil Rogers RO4350B with Comparison to Microstrip," Southwest Microwave, Inc., Tempe, AZ, 2007, www.southwestmicrowave.com.

RO4350B and RO4003Care licensed trademarks of Rogers Corporation.

The information in this paper is intended to assist you in working with Rogers' High-Frequency Materials. It is not intended to and does not create any warranties, express or implied, including any warranty of merchantability or fitness for a particular purpose or that any results show in this paper will be achieved by a user for a particular purpose. The user is responsible for determining the suitability of Rogers' High Frequency Materials for each application.