Practical Considerations for PCB Impedance Measurements

John Coonrod, Rogers Corporation

Abstract

It is common for PCBs used for high frequency RF or high speed digital applications, to be tested for an impedance value prior to shipping the board. The controlled impedance board is typically specified to a nominal impedance value with a given tolerance. Some time ago a tolerance of ± 10% was considered acceptable however recently and with more demanding applications, the impedance tolerance is often much narrower. Understanding the many aspects of impedance is beneficial to the PCB fabricator and designer, who are often required to make a judgment for product shipment based on the impedance measurement. There are several different types of impedance however characteristic impedance is normally specified for a controlled impedance PCB. This circuit property has many variables and some variables are related to the PCB manufacturing process, some are associated with material properties and some variables are due to measurement techniques. Additionally, some of these variables are more or less dominate for a particular type of circuit design and / or construction.

This paper will give an overview of the basic theory for impedance, with an emphasis on characteristic impedance for circuits of different design types. Microstrip, grounded coplanar waveguide (GCPW) and stripline structures will be discussed with their unique impedance attributes. PCB fabrication related variables, as well as material variables, will be illustrated using modeling software and verified with measured results. The variables associated with impedance measurement are many and details will be given for several related issues. One variable, often not recognized, is masking and that is how the impedance value of a circuit can be altered due to an impedance spike which is located prior to the body of the circuit. Masking can cause inaccuracies for impedance measurements and there are ways to minimize this concern, which will be illustrated. The impact on impedance resolution and accuracy due to rise time will also be demonstrated with measured examples.

Introduction

PCB materials used for RF/microwave or high-speed digital applications often undergo impedance testing prior to being shipped to a customer. PCB materials for RF/microwave applications, for example, are often formed into circuits requiring 50-Ω transmission lines, such as microstrip and stripline, while circuits for CATV applications are at 75 Ω. Testing can determine if the PCB materials are within a certain tolerance of a required impedance value. At one time, for example, ±10% was considered an acceptable impedance tolerance. But as high-frequency and high-speed circuits advance, acceptable tolerances become tighter. PCB impedance measurements are a way to ensure PCB designers and fabricators are working with circuit materials with electrical characteristics similar to those represented in the circuit modeling software being used to design a new circuit.

A PCB with a controlled impedance is a combination of dielectric and conductor materials assembled to achieve a target impedance, such as 50 Ω. A controlled-impedance PCB can be described by a number of difference impedances, although it is probably best known by its characteristic impedance. The characteristic impedance is influenced by many variables, some related to material properties and some to the PCB manufacturing process, and the measured value of the characteristic impedance can even be affected by the way that the impedance measurements are performed and/or type of measurement. The type of high-frequency/high-speed circuit design can also influence how strongly each variable will affect the impedance and how precisely it can be measured.

Different transmission lines are used in RF/microwave PCBs, each with a unique physical format and its own manner of achieving an impedance for a given circuit structure. Modeling software uses the physical formats of RF/microwave transmission lines such as microstrip, stripline, and grounded coplanar waveguide (GCPW) and the dimensions of different circuit structures for each transmission line combined with the known parameters of a PCB material to determine impedance. Of course, the modeling software is only as good as the measurements that support it.

The different transmission lines produce impedances in different ways, as a result of such things as dielectric thickness, conductor thickness, and relative position to the ground plane. The impedance is a result of a number of variables, some material related and some related to the fabrication of the PCB. By reviewing some basic impedance theory as well as PCB-fabrication-related variables and material-based variables that can affect impedance, it may be possible to control the effects of those variables on PCB impedance. Exploring PCB impedance measurements can also provide great insight into the role of the PCB material in achieving a particular circuit's characteristic impedance.

Impedance measurements are subject to many variables which can influence final values. One variable that is often overlooked is masking, which refers to how the impedance value of a circuit can be altered due to an impedance spike occurring prior to the circuit under test. It can result in impedance measurement inaccuracies but, fortunately, there are ways to minimize the effects of masking on test results. Some measurement examples will be used to show the impact of masking on impedance measurement resolution and accuracy as a result of how rise time is affected during a measurement.

Investigating Impedance

Impedance exists in many forms in high-speed/high-frequency circuits, including input impedance, wave impedance, surface impedance, characteristic impedance, and frequency-dependent impedances. Characteristic impedance is typically a reference when building or testing PCBs. Several simple formulas are available for understanding how an impedance results for a particular structure, not necessary for calculating the actual impedance of the circuit or structure. Equation 1 is one of these simple formulas:

$$Z_0 = \sqrt{\frac{L}{C}} \qquad (1)$$

The depiction of a microstrip line in Fig. 1 helps to demonstrate how the characteristic impedance, Z_0, is a function of the circuit impedance (L) and capacitance (C). The schematic circuit diagram shows the additional circuit parameters for resistance, R, and conductance, G, helping to explain how physical features of microstrip are related to its impedance. A microstrip signal conductor is mainly characterized by inductance and resistance while the dielectric substrate, which separates the microstrip circuit's signal and ground planes, is mainly characterized by capacitance and dielectric conductance.

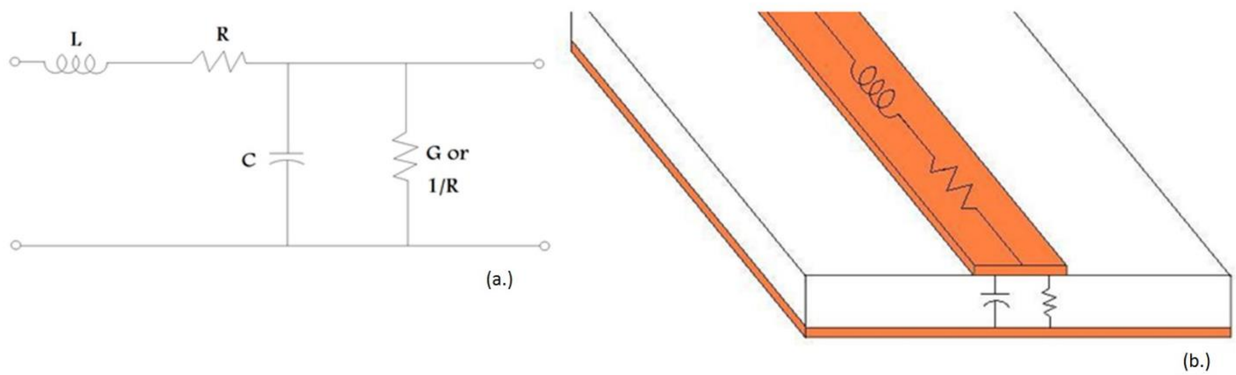


Figure 1. Schematic diagram (a) representing the circuit elements of (b) a microstrip transmission line.

The transmission line's different circuit attributes (RLGC) are frequency dependent and have some associated losses. The conductor losses are related to R in the schematic diagram while the dielectric losses are related to the G component in the RLGC circuit. By showing the microstrip transmission line as a lumped-element representation, using the schematic diagram of Fig. 2 and Eq. 2, the frequency-dependent losses can be included.

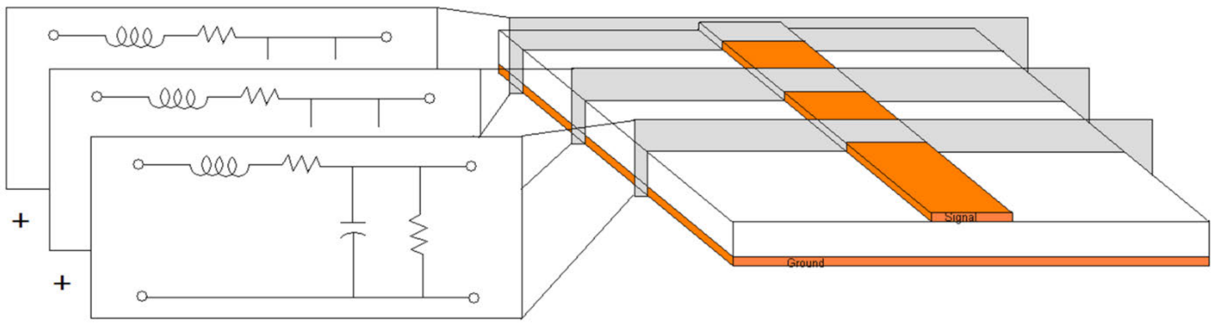


Figure 2. Lumped element representation of the microstrip transmission line circuits.

$$Z_0 = \sqrt{\frac{R + j\omega L}{G + j\omega C}} \tag{2}$$

Equation 1 provides a straightforward way to understand impedance-related relationships when designing or troubleshooting impedance issues. For higher-frequency circuits, 50 Ω is generally considered as a reference characteristic impedance. When a circuit has a higher C value, which will result in a decrease in impedance, it is called a capacitive circuit. When the circuit has an increase in L, which will result in an increase in impedance, it is referred to as an inductive circuit.

Physical differences in a circuit will impact the values of L and C in the circuit (in Eq. 1) and thus the impedance of the circuit. An increase in the conductor width will cause an increase in the parallel-plate capacitance of the circuit and a decrease in the transmission-line impedance. An increase in the distance between the conductive planes (due to a thicker substrate) will cause a decrease in capacitance and an increase in the microstrip transmission-line impedance as shown in Fig. 3.

$$Z_0 = \sqrt{\frac{L}{C}}$$

$$C = \frac{Dk \cdot Area}{thickness}$$

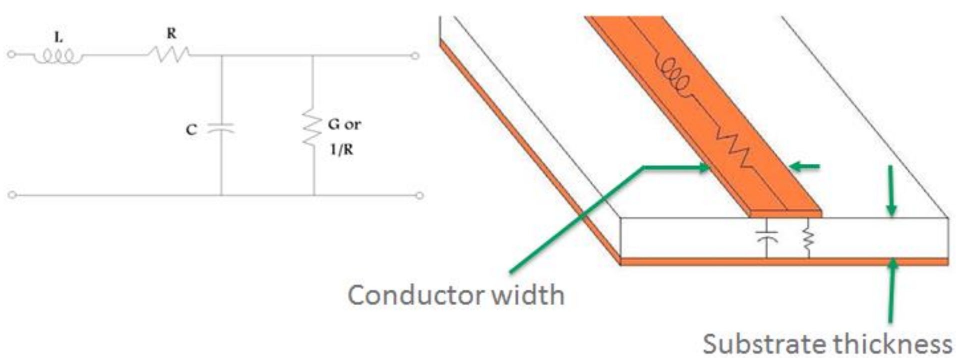


Figure 3. Microstrip transmission line circuit with RLGC notations and reference to alterations of the circuit dimensions which impact impedance.

Impedance testing on a PCB is typically performed with the aid of a time-domain reflectometer (TDR) or by using the TDR function of a vector network analyzer (VNA). A 50-Ω cable can be used as a reference for TDR measurements. The impedance difference between the sample circuit and the 50-Ω cable determines the impedance of the circuit under test. Figure 4 shows examples of TDR measurement traces for open and short circuits and how the 50-Ω cable serves as the reference part of each measurement.

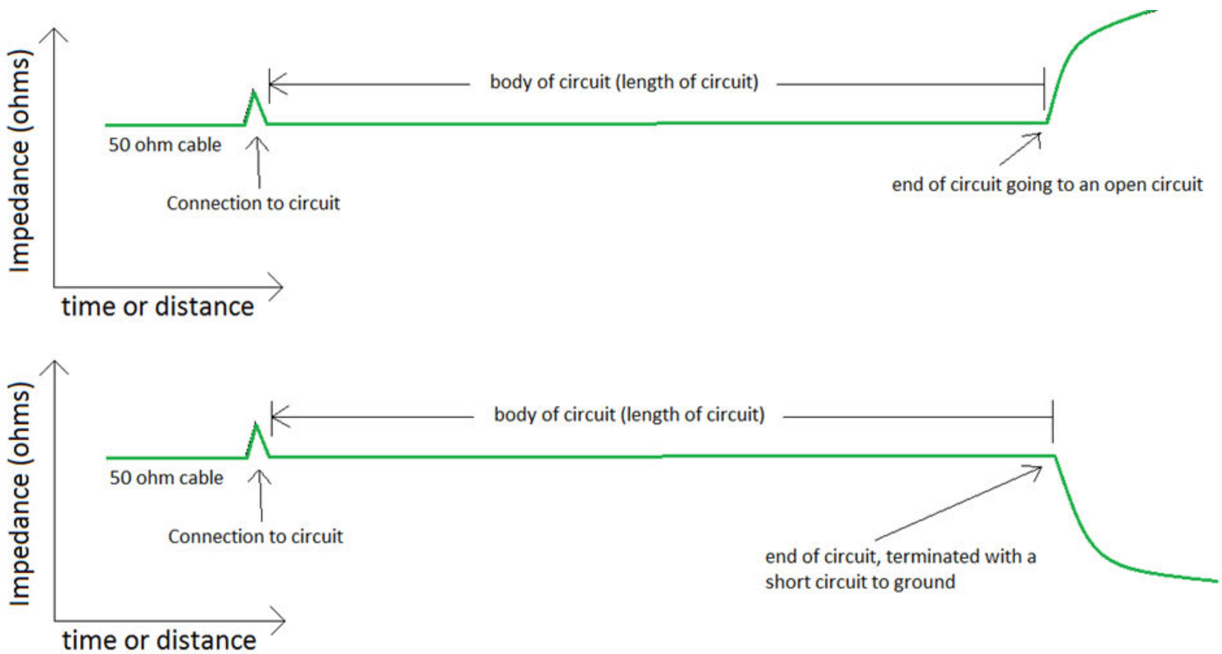
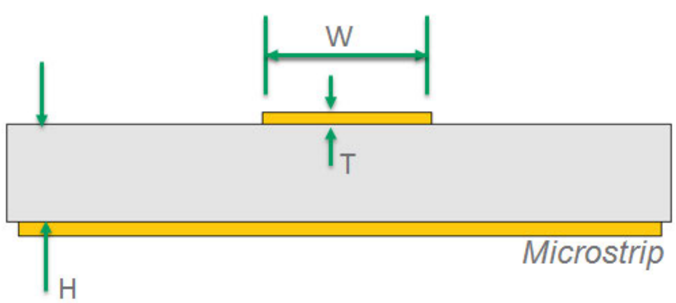


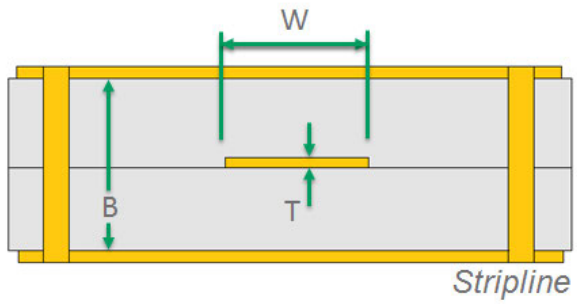
Figure 4. Example of TDR trace for 50-Ω circuits terminated with an open (top trace) and a short (bottom trace).

The inductive spike at the connection between the 50-Ω cable and the circuit under test is an assumption only and sometimes this junction results in a capacitive dip. Ideally, there would be no impedance mismatch at the connection between the cable and the circuit under test, although some impedance mismatch is not unusual at this interface. The impedance mismatch can result in signal reflections at the junction which in turn results in degraded return-loss performance. The return-loss degradation will typically limit the bandwidth of the circuit.

For simple impedance calculations using closed-form equations, several references are available for some common transmission-line structures. These closed-form equations can be used to create a simple program for impedance calculations or used in a spreadsheet to perform the calculations. Figure 5 offers two references for closed-form impedance equations which can be used for microstrip and stripline transmission-line circuitry.



$$Z_0 = \frac{87}{\sqrt{Dk + 1.41}} \ln\left[\frac{5.98H}{(0.8W + T)}\right]$$



$$Z_0 = \frac{60}{\sqrt{Dk}} \ln\left[\frac{1.9B}{(0.8W + T)}\right]$$

Figure 5. Closed-form equations which can be used for simple impedance calculations for microstrip and stripline transmission-line circuits.

The closed-form equation used for the microstrip structure at the top of Fig. 5 is from a well-known reference [1] on microstrip modeling. The stripline equation is from a different reference, [2]although it has been found to be relatively accurate.

Several variables impact the impedance of a high-frequency/high-speed circuit, with the most influential being the width of the signal conductor, the dielectric constant (Dk) of the circuit substrate material, the thickness of the substrate, and the thickness of the copper conductor. Understanding how these different variables can impact PCB impedance is essential to designing or troubleshooting a high-speed/high-frequency PCB, and Table 1 provides details from an impedance-modeling software program on how the different variables can affect the impedance of a circuit.

Table 1. Results from impedance-modeling software showing the hierarchy of the impedance variables.

The most significant variables for impedance differences are:

1. Substrate thickness
2. Conductor width
3. Copper thickness
4. Dk

Microstrip transmission line circuit using 20mil thick high frequency laminate

Dk	Substrate Thickness (mils)	Copper Thickness (mils)	Conductor width (mils)	Characteristic Impedance (ohms)	Difference of impedance (ohms)	Comment
3.50	20	2	43	50.07		Baseline for comparisons
3.50	18	2	43	46.86	3.21	Substrate is 10% thinner than baseline
3.45	20	2	43	50.39	0.32	Dk lower by 1.4% from baseline
3.50	20	1	43	50.70	0.63	Copper thickness reduced by 1mil from baseline
3.50	20	2	42	50.78	0.71	Conductor width reduced by 1mil from baseline

The first row of information for all of the variables in Table 1 is baseline or reference data for the model of a microstrip transmission-line circuit, to show how changes to the variables change the performance of the circuit. The starting point, listed across Row 1, is a reference circuit based on PCB substrate with Dk of 3.5, substrate thickness of 20 mils, copper thickness of 2 mils, and conductor width of 43 mils. This results in a characteristic impedance of 50.07 Ω. In Row 2, the only thing that has been changed from the reference model is a 10% change in substrate thickness, to 18 mils. This drops the characteristic impedance by 3.21 Ω, to 46.86 Ω.

In Row 3, all is the same as for the reference model in Row 1, except that the Dk of the circuit substrate has been reduced from 3.50 to 3.45. This results in a change of 0.32 Ω in the characteristic impedance of the microstrip transmission line, to 50.39 Ω. The change in Dk value was chosen to represent the tolerance of ±0.05 that is common for high-volume, high frequency PCB materials with Dk of 3.5.

Row 4 of Table 1 shows the effects on impedance when changing the overall copper thickness by a realistic value of 1 mil. The overall copper thickness is a combination of the base copper thickness of the laminate combined with the additional copper plating which the PCB fabricator will apply during the manufacturing process. Row 5 shows the effects on PCB impedance when the conductor width is changed by 1 mil, which is also a realistic variation commonly found in PCB fabrication processes.

The differences in how these variables impact PCB impedance show that there is a hierarchy to these variables, with the most influential variable for impedance being substrate thickness. Next in line for PCB impedance is conductor width, followed by copper thickness and then variations in Dk. This rating of the variables assumes a test circuit with 20-mil-thick microstrip transmission line. If the same models were used with a 10-mil-thick microstrip transmission line, the same trends would be found for the variables that impact impedance. For the 10-mil-thick circuit, the amplitudes of the differences of impedance are higher for features related to the conductor. A thinner circuit is more impacted by conductor effects such as copper thickness or conductor width than a thicker circuit.

The fact that variations in substrate material Dk have less impact on PCB impedance than some of the other variables may cause many designers to rethink their PCB impedance troubleshooting routines. Many designers intuitively think that Dk control is the most important step in achieving PCB impedance control, and that is typically not the case. The model in Table 1 does assume the use of a high-frequency circuit laminate with good control of Dk and tolerance of ±0.050. But even if the difference in Dk was doubled in the model, the hierarchy of how the variables impact impedance would remain the same. When troubleshooting PCB impedance issues, unwanted variations in substrate thickness should be one of the first things to consider.

If the same model experiment was performed with stripline transmission-line circuits, a similar trend in the hierarchy of circuit variables affecting PCB impedance would be found, with Dk tolerance being the variable of least significance regarding circuit impedance. In addition, if a GCPW model was evaluated for the influence of the variables on impedance, the trend would be similar. But due to the strong coupling fields in the coplanar layer, copper thickness variations would be much more influential in affecting impedance than with microstrip or stripline circuits.

Circuit material moisture absorption is a characteristic that has often been connected to changes in impedance. Certainly, circuit materials high in moisture absorption have been known to suffer in terms of insertion-loss performance and phase response due to moisture absorption. But moisture absorption may not have as significant an impact on impedance as on these other circuit characteristics. To better understand this, two different types of circuit materials were evaluated under controlled conditions: a ceramic-filled PTFE circuit material with low moisture absorption and a PPE-based circuit material with higher moisture absorption.

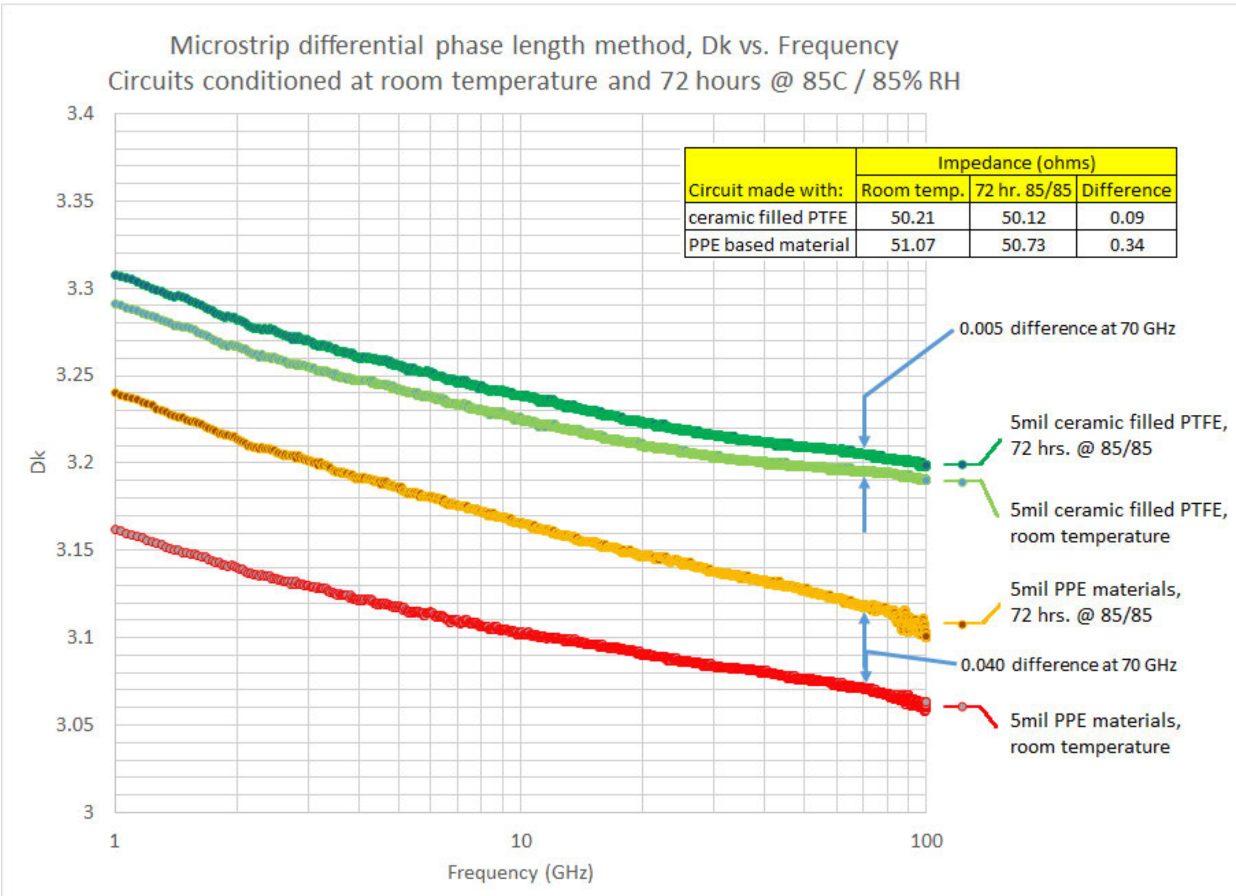


Figure 6. Dk versus Frequency curves from testing two different sets of microstrip transmission line circuits with materials that have different moisture absorption properties.

Testing was performed on the two different circuit materials by means of the differential phase-length method,[3] with results plotted in Fig. 6. The circuits were first tested under room-temperature conditions and then tested again after being conditioned at +85°C and 85% relative humidity (RH) for 72 hours. The ceramic-filled PTFE circuit material exhibits moisture absorption of 0.05% while the PPE-based circuit material has moisture absorption of 0.30%. Even though 0.30% is not considered a high value for moisture absorption, it is high enough to make a substantial difference for the phase response of the circuit.

Figure 6 plots Dk versus frequency, with Dk values back-calculated from the differential phase-length measurements of the circuits. Even though the Dk values shifted significantly for the PPE-based materials due to moisture absorption, the impedance values (shown in the table within Fig. 6) were only slightly affected. This reason goes back to the hierarchy of the impedance variables, where a change in Dk is not a major influence for a change in impedance.

Another impedance-related issue is the rise time of the impedance measurement and the potential impact it has on impedance measurement accuracy. It has been reported that, in some cases, a right angle in the conductor routing has minimal or no impact on the impedance of the circuit. This may be the case if the measurement equipment rise time is too slow to detect the impedance anomaly caused by the right-angle bend. As a general practice for RF applications, the right-angle bend for conductor routing is chamfered to maintain a minimal difference in area between signal and ground planes.

When such conductor chamfering is done correctly, there is no difference in the parallel-plate capacitance in the bend area, which translates to no difference in the transmission-line impedance. However, if measurement rise times are too slow, bends representing impedance anomalies in the conductor routing will not be obvious. When the rise time is faster, impedance anomalies caused by such bends can be detected, as seen in Fig. 7.

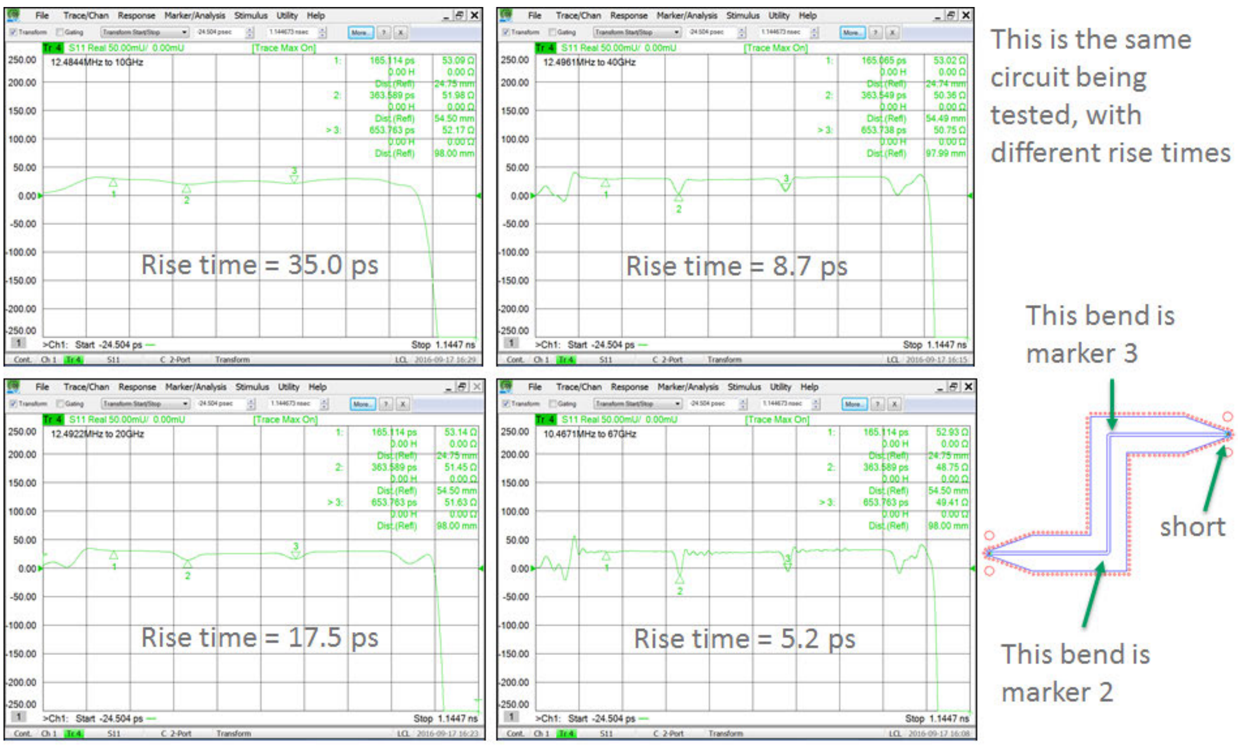


Figure 7. Impedance curves (green curves) for a stripline circuit with two right-angle bends in the conductor routing made using vector analyzers with different rise times.

The screen shots shown in Fig. 7 are in the time domain, with the x-axis representing time or distance. The left side of the screen shot is where the test signal enters the circuit under test from the test connector. Marker 2 indicates where the first right-angle bend occurs and marker 3 shows where the second right-angle bend is located. The circuit is terminated with an electrical short, seen as the abrupt decrease in impedance near the right-hand side of the screen plot.

The screen shot in the upper left, using a slower rise time, does not show the impact of the right-angle bend on the impedance. When the rise time is doubled or made faster in the bottom left screen shot, slight impedance anomalies appear at markers 2 and 3, where the right-angle bends are located. When the rise time speeds up further, in the upper-right screen shot, the impedance anomalies for the right-angle bends are more apparent. Finally, the bottom-right screen shot was made with the fastest rise time, and it shows how the right-angle bends in the conductor routing cause a capacitive dip as indicated at markers 2 and 3.

Measurements in Fig. 7 were all on the same circuit, with the only differences in the test set being in impedance measurement rise time. The bottom-right screen shot shows a difference in the depths of the capacitive dips caused by the right-angle circuit bends at markers 2 and 3. If the circuit is disconnected, flipped around, reconnected, and tested again, the impedance curve will appear the same, with a larger capacitive dip at marker 2 than at marker 3. Both right-angle bends in the conductor routing are exactly the same. So, in theory, they should have the same depth for impedance; however, they do not. The right-angle bend located closest to the connector will have the larger capacitive dip, with the following bend showing a smaller capacitive dip, and this is due to masking. Basically, masking is a phenomenon where a significant impedance anomaly will lessen an impedance spike which follows. Also, if the launch point from the cable to the circuit has a large impedance anomaly, it can corrupt the impedance curve which follows.

A good example of impedance masking is from a study [4] on transitioning a signal conductor from one copper layer to another within a stripline circuit. The study featured a dual stripline circuit with three viaholes in the body of the circuit and connectors on both ends. Different variables were studied to learn how to minimize the impedance anomaly for the signal transition; however, impedance masking was apparent for several issues. One obvious impedance masking event was observed when the same circuit was tested with the analyzer's gating function turned on or off. The gating function allows the removal of an impedance anomaly by gating-out a portion of the impedance curve. Figure 8 shows two impedance curves for the same circuit, one measured with gating on and one with gating off.

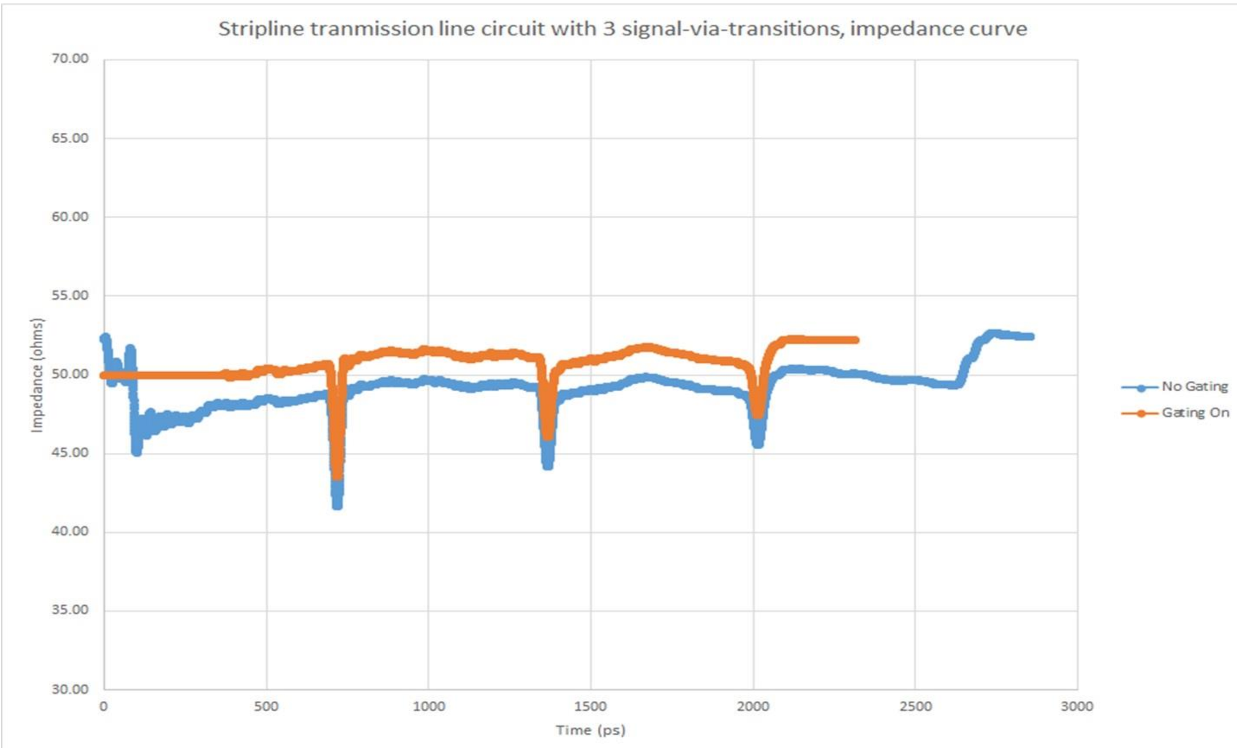


Figure 8. Two impedance curves for the same stripline circuit showing the effects of using an analyzer's gating function to remove impedance masking.

The blue curve in Fig. 8 is the dual stripline circuit with the connector signal launch at the far left and far right of the impedance trace. The blue curve shows the impedance curve as it was measured initially. The orange curve plots the impedance for the same circuit, except that gating was applied in the signal launch area of the connector-circuit interface. Gating removed the strong capacitive dip at the signal launch area. The overall impedance curve increased when the capacitive dip was removed. The capacitive dip in the signal launch area caused masking on the rest of the blue impedance curve, resulting in a lower-than-actual impedance reading.

Conclusions/Summary

PCB impedance is affected by many variables; understanding these variables can help when developing a new circuit design or when troubleshooting an existing PCB. The variables affect impedance in a hierarchical manner, some with greater impact than others, and knowing the magnitude that each variable can have on PCB impedance can assist in both circuit design and troubleshooting later on. When testing circuits for impedance, knowledge of the masking effect can be vital for ensuring that the measured impedance values meet the accuracy and resolution needed to assist both PCB design and troubleshooting efforts.

References

[1] Hammerstad and O. Jensen, "Accurate Models for Microstrip Computer-Aided Design," 1980 IEEE MTT-S International Symposium digest, May 1980, Washington, DC IEEE catalog #80CH1545-3MTT, pp 407-409.

[2] S.B. Cohn, “Characteristic Impedance of the Shielded-Strip Transmission Line,” IRE Transactions on Microwave Theory & Techniques(MTT), July 1954, pp 52 – 57.

[3] John Coonrod, “Understanding the Variables of Dielectric Constant for PCB Materials Used at Microwave Frequencies,” European Microwave Week 2011, October 2011.

[4] John Coonrod, “High Frequency RF Electrical Performance of Plated Through Holes Vias,” IPC APEX 2016, March 2016.

Practical Considerations for Impedance Measurements of PCBs

John Coonrod

Rogers Corporation

Advanced Connectivity Solutions

Agenda

- Basic impedance definition and concepts
- PCB structures, impedance attributes
  - *Microstrip, Grounded Coplanar Waveguide (GCPW) and stripline*
- Practical variables which impact impedance of PCB’s
- Rise time; impedance resolution and accuracy
- Masking; impedance accuracy

- There are several different categories for impedance

    - *Wave impedance*

    - *Input impedance*

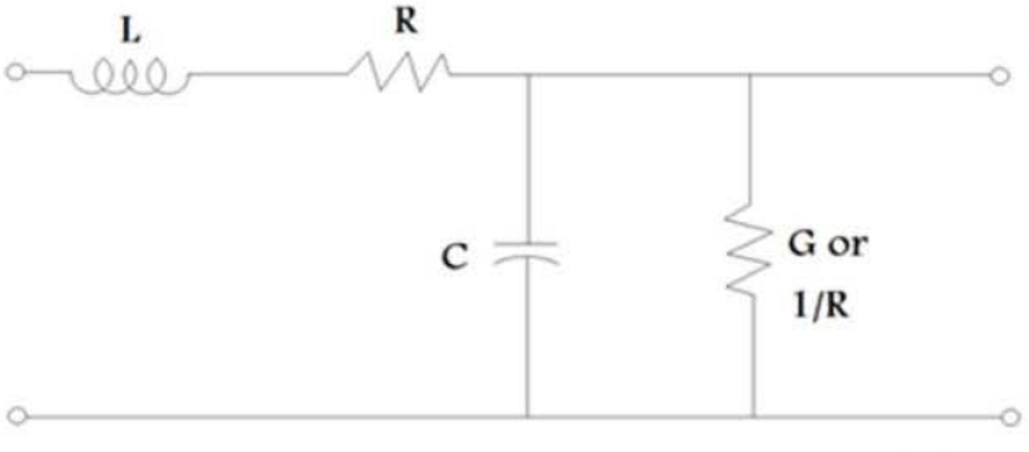
    - *Characteristic impedance*

    - *Surface impedance*

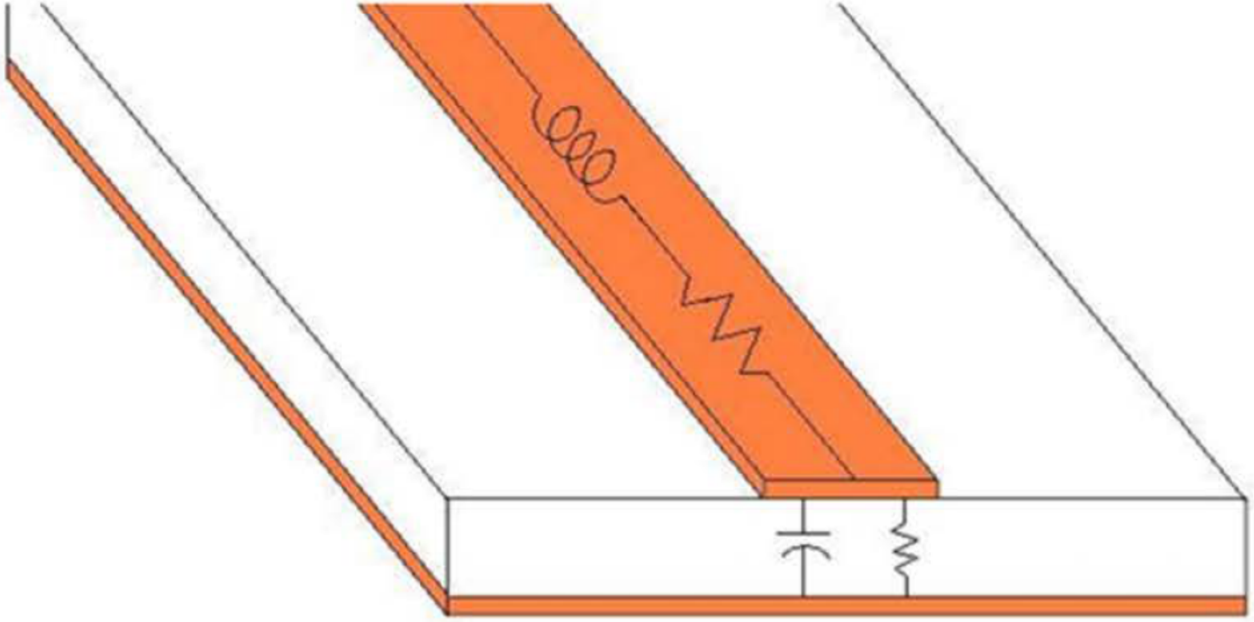
    - *Frequency dependent impedance*

- The impedance most commonly specified for a “controlled impedance PCB” is characteristic impedance

Basic impedance definition and concepts



(a.)



(b.)

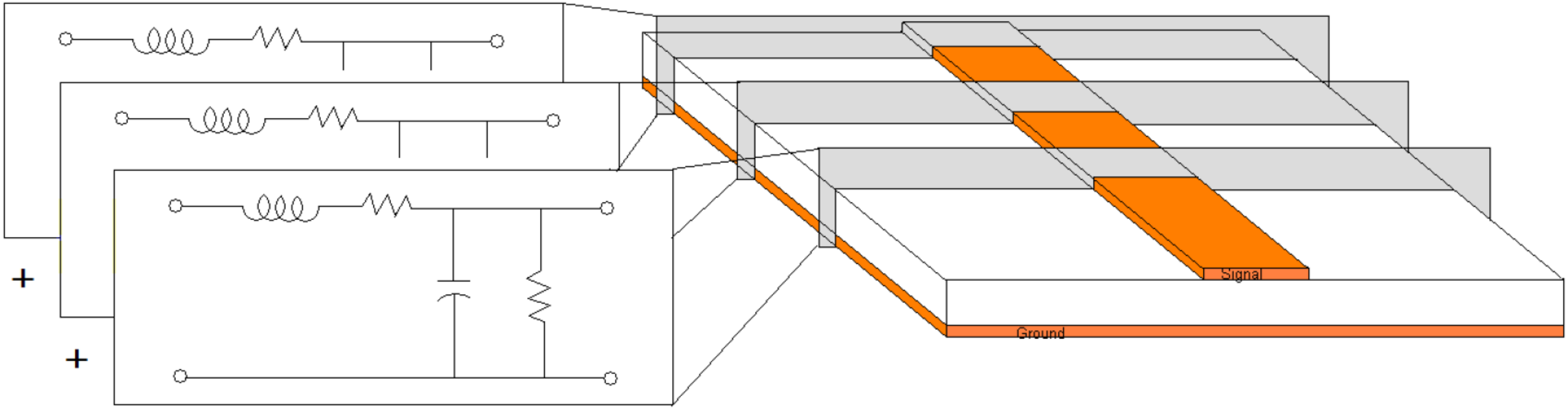
$$Z_0 = \sqrt{\frac{L}{C}}$$

Basic impedance definition and concepts

- A better definition for characteristic impedance is accounting for the lumped element aspects of a circuit where differential-circuits are added up
- This relationship is frequency dependent and does include conductor loss and dielectric loss

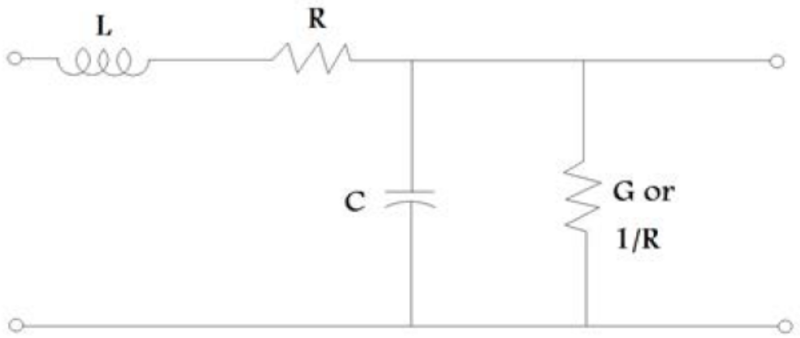
$$Z_0 = \sqrt{\frac{R + j\omega L}{G + j\omega C}}$$

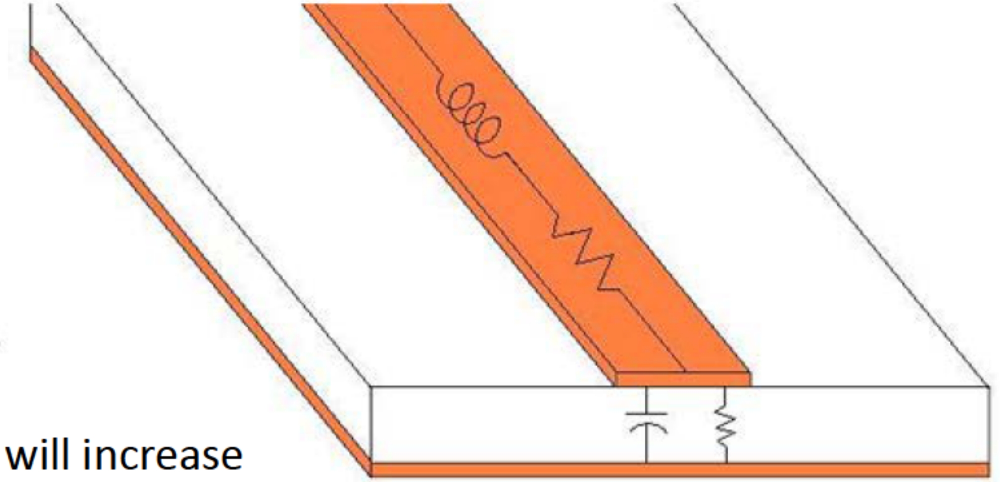
R is related to conductor loss, G related to dielectric loss and ω is frequency (ω = 2πf)



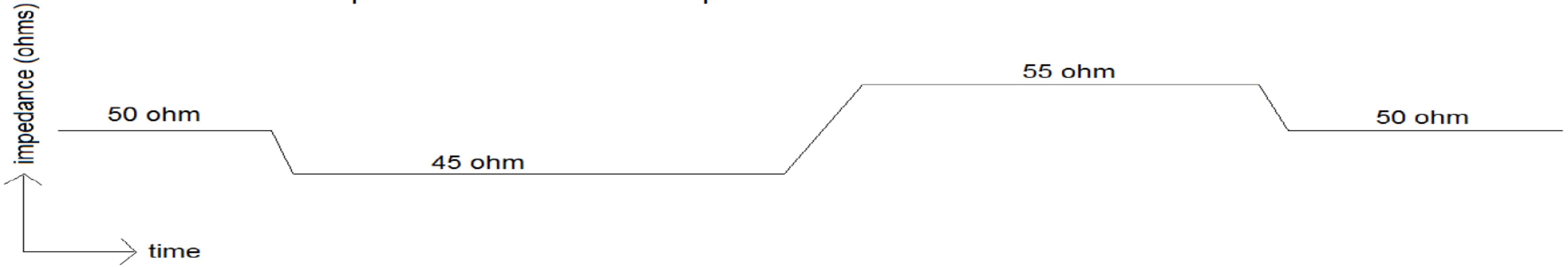
- The following formula can be used for practical relationships of characteristic impedance:

$$Z_0 = \sqrt{\frac{L}{C}}$$



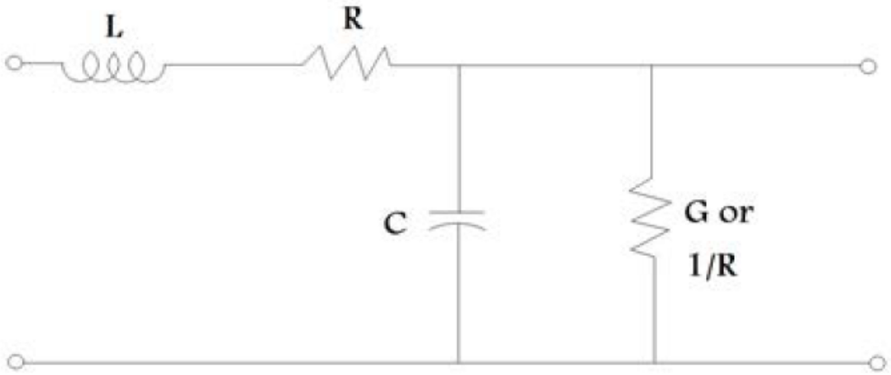


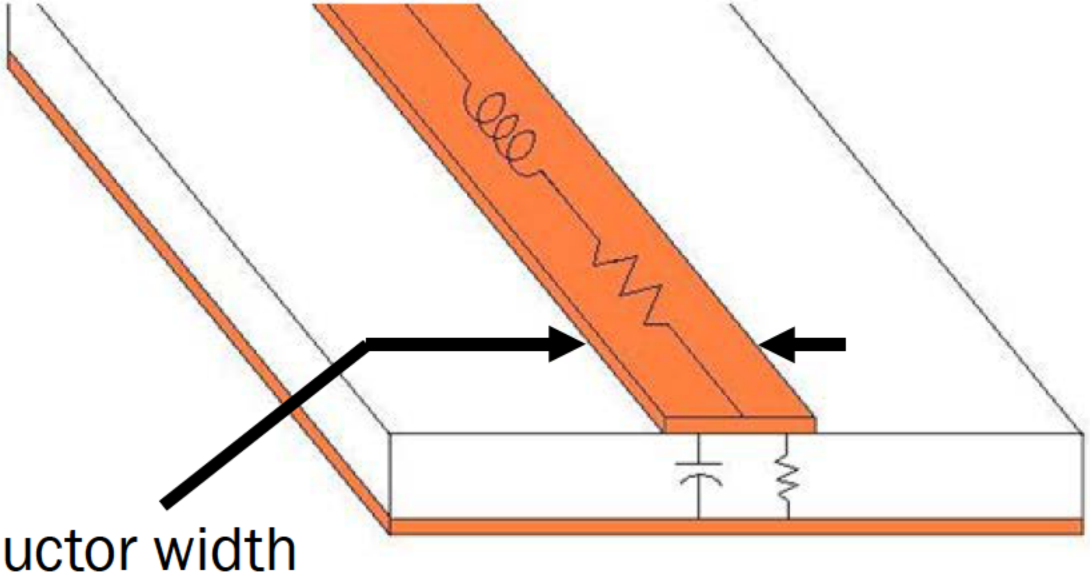
  - With an increased inductance, the impedance will increase
  - With an increased capacitance, the impedance will decrease
  - Impedance curve example shown below assumes a 50 ohm system, the 45 ohm portion could be due to an increase in capacitance and the 55 ohm portion could be due to an increase in inductance



- The following formula can be used for practical relationships of characteristic impedance:

$$Z_0 = \sqrt{\frac{L}{C}}$$



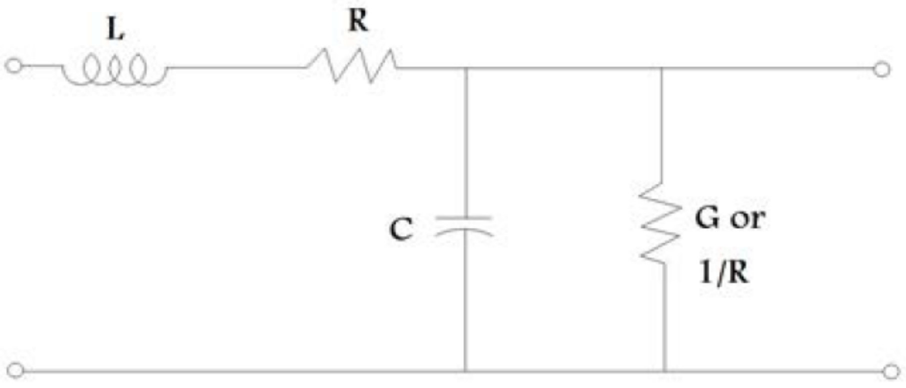


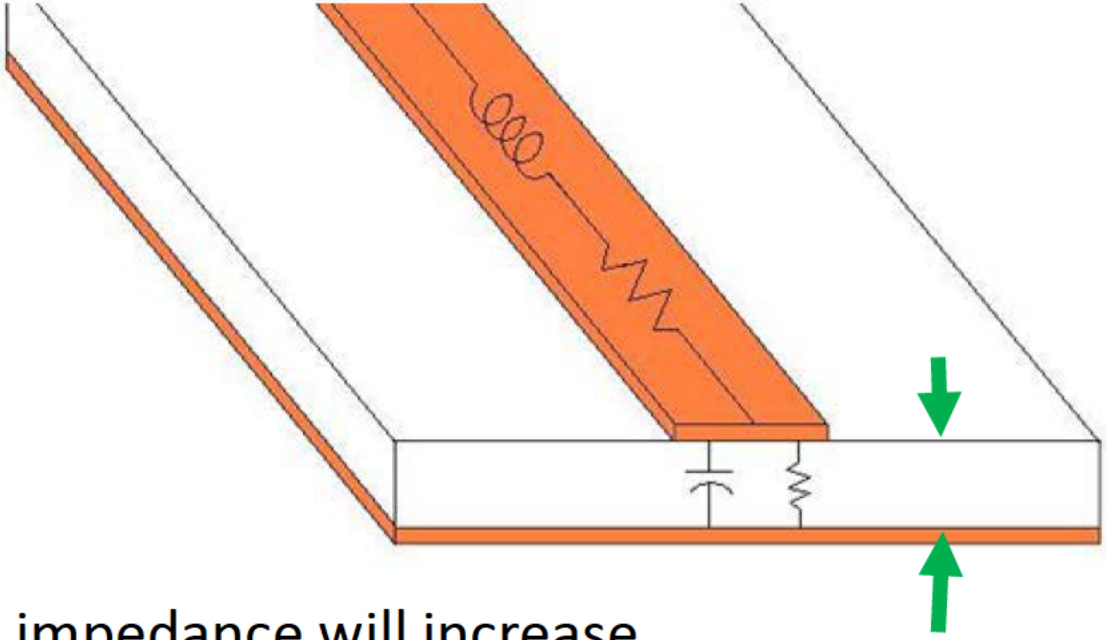
- Narrower conductor, increases inductance and the impedance will increase
- Wider conductor, increases capacitance and the impedance will decrease

- The following formula can be used for practical relationships of characteristic impedance:

$$Z_0 = \sqrt{\frac{L}{C}}$$

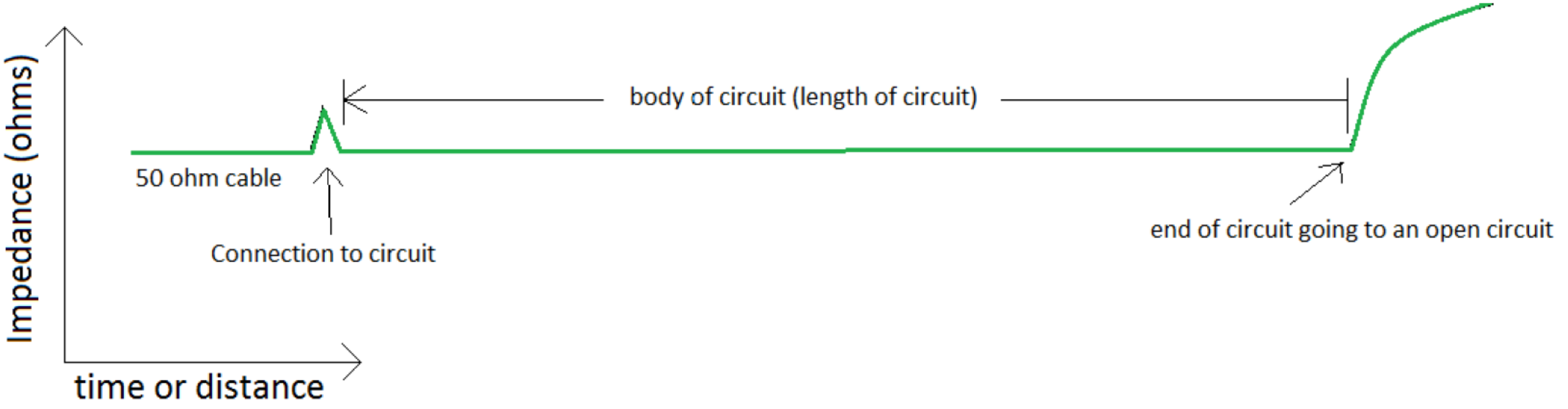
$$C = \frac{Dk \cdot Area}{thickness}$$

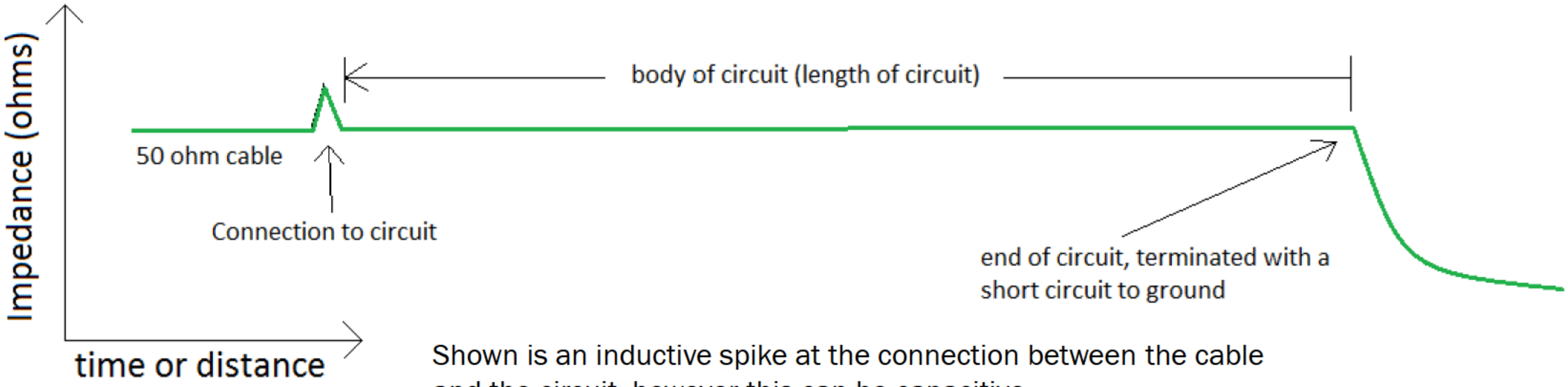




- Thicker substrate, capacitance decreases and the impedance will increase
- Thinner substrate, capacitance increases and the impedance will decrease
- A wider conductor can also be thought of as an increased area between the signal plane and the ground plane. The increased area will increase capacitance

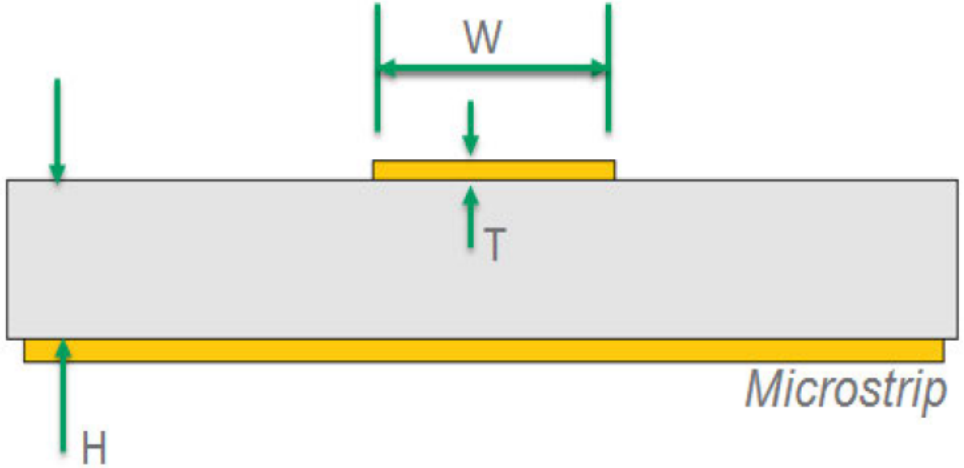
- Typical impedance curves



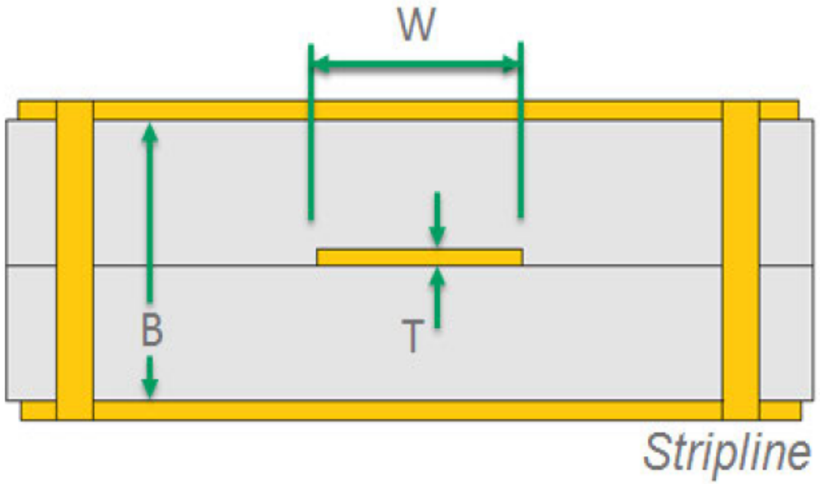


Shown is an inductive spike at the connection between the cable and the circuit, however this can be capacitive

- The following formulas are common closed form equations which can be used to write a simple program or used in a spreadsheet program, to get the approximate impedance



$$Z_0 = \frac{87}{\sqrt{Dk+1.41}} \ln\left[\frac{5.98H}{(0.8W+T)}\right]$$



$$Z_0 = \frac{60}{\sqrt{Dk}} \ln\left[\frac{1.9B}{(0.8W+T)}\right]$$

- This gives the general hierarchical influence of variables which impact impedance values

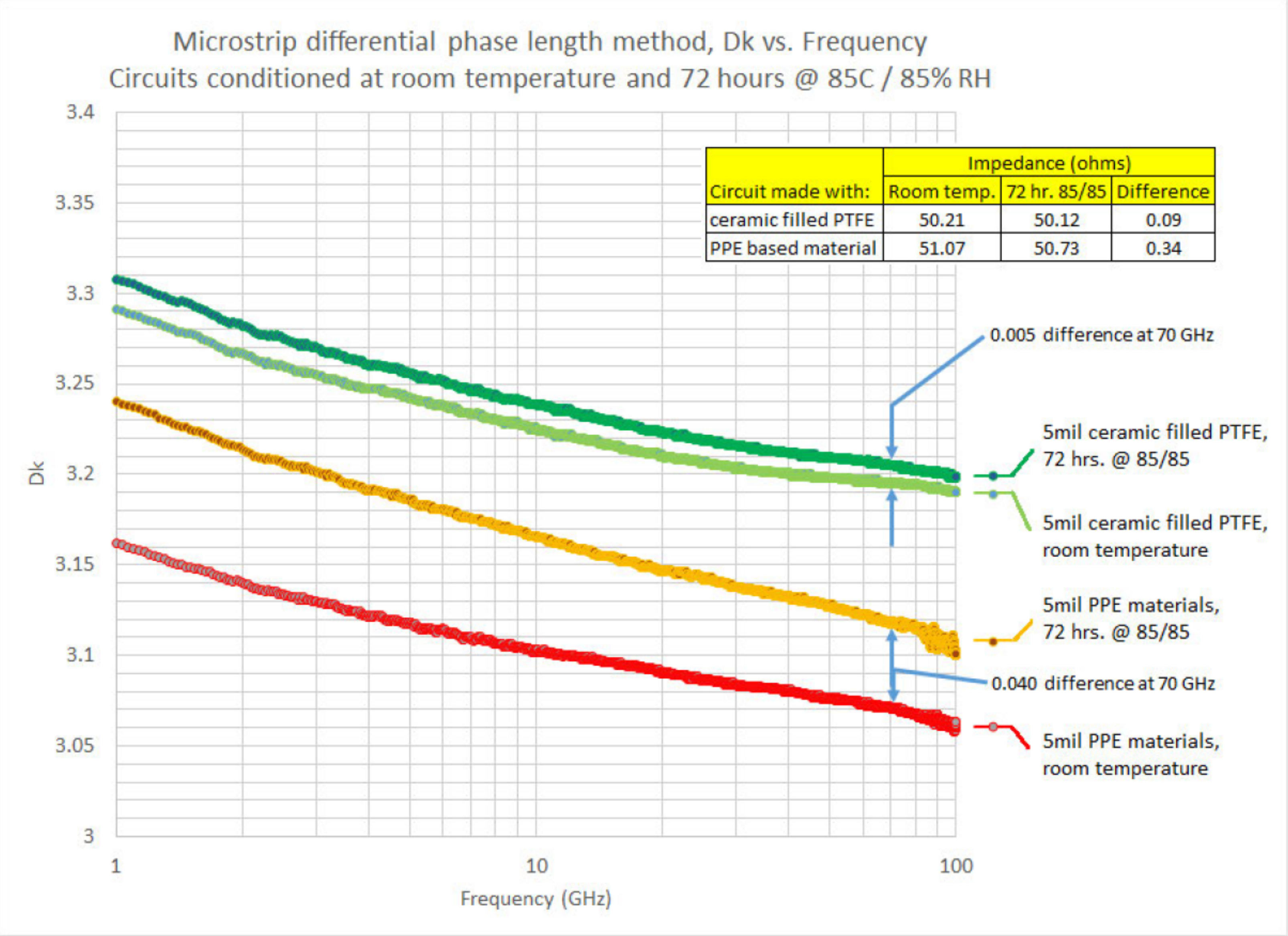
The most significant variables for impedance differences are:

1. Substrate thickness
2. Conductor width
3. Copper thickness
4. Dk

Microstrip transmission line circuit using 20mil thick high frequency laminate

Dk	Substrate Thickness (mils)	Copper Thickness (mils)	Conductor width (mils)	Characteristic Impedance (ohms)	Difference of impedance (ohms)	Comment
3.50	20	2	43	50.07		Baseline for comparisons
3.50	18	2	43	46.86	3.21	Substrate is 10% thinner than baseline
3.45	20	2	43	50.39	0.32	Dk lower by 1.4% from baseline
3.50	20	1	43	50.70	0.63	Copper thickness reduced by 1mil from baseline
3.50	20	2	42	50.78	0.71	Conductor width reduced by 1mil from baseline

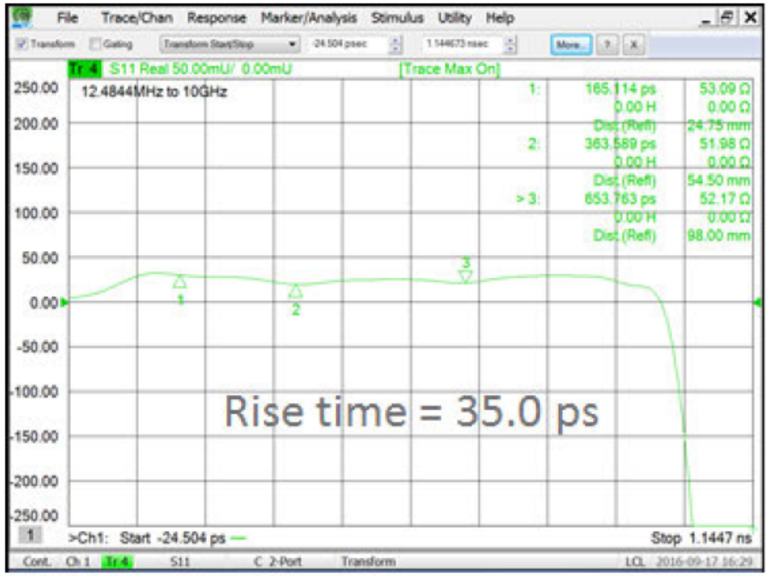
- Environmental conditioning and effects on circuits using materials with different moisture absorption properties

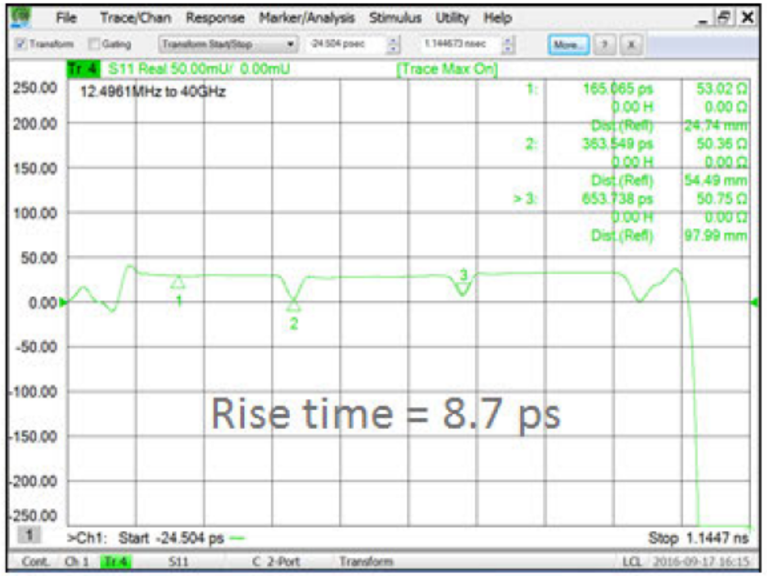


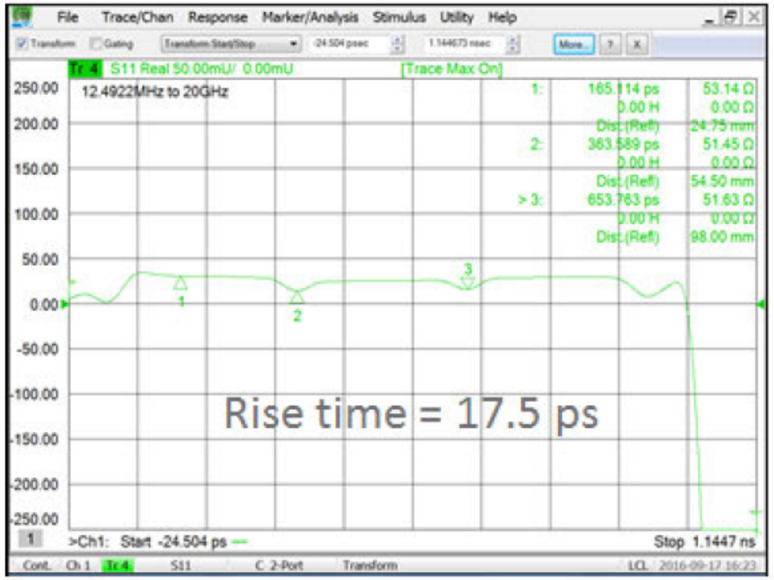
	Impedance (ohms)		
Circuit made with:	Room temp.	72 hr. 85/85	Difference
ceramic filled PTFE	50.21	50.12	0.09
PPE based material	51.07	50.73	0.34

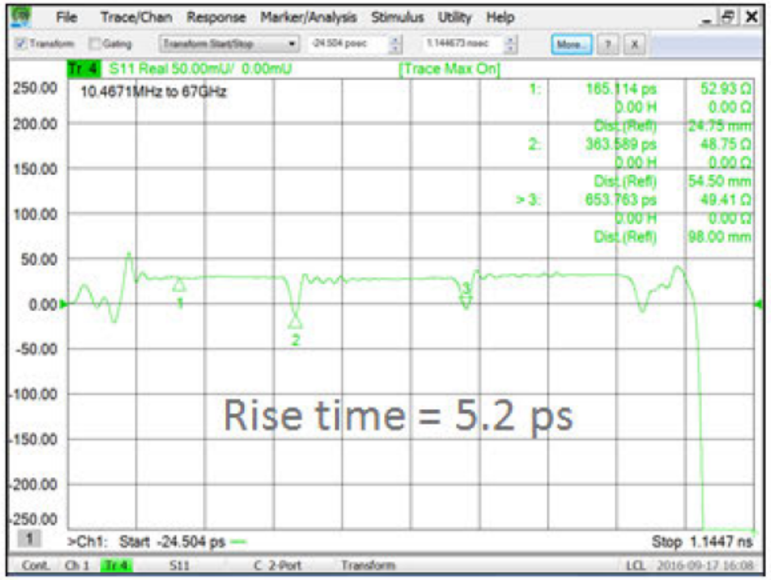
- Impedance measurements of the same circuit, but using different rise times

The right angle bend in the conductor routing causes an increase in capacitance in the isolated area of the bend



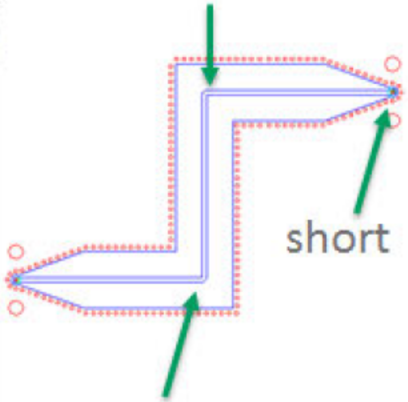






This is the same circuit being tested, with different rise times

This bend is marker 3



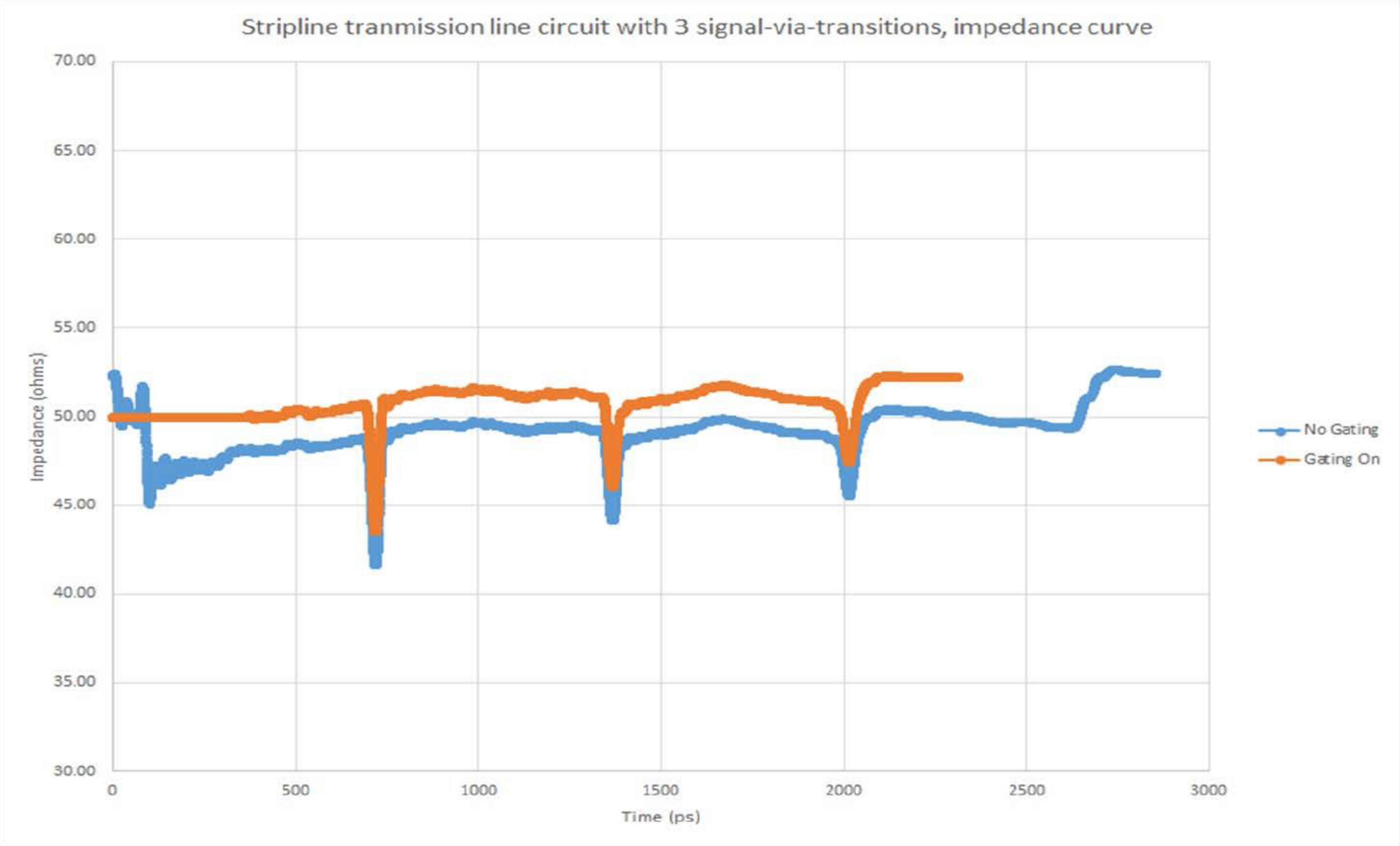
This bend is marker 2

- Impedance curves of the same circuit being tested, however with Gating On the impedance masking effects are minimized

The impedance curve starting on the left side has a strong capacitive dip due to the signal launch (connector-circuit transition)

When Gating is On, the capacitive dip is eliminated and the entire impedance curve shifts up impedance

Masking is where a large impedance anomaly will impact the impedance value of the circuit beyond the impedance anomaly



- Conclusions

  - Understanding the many impedance variables is advantageous for circuit design and troubleshooting

  - There is a hierarchy of material and PCB influences on characteristic impedance
    1. Substrate thickness
    2. Conductor width
    3. Copper thickness
    4. Dk

  - Rise time of the impedance test is critical to resolve accurate impedance values

  - Masking is when a large impedance difference, will cause the following impedance measurements to be less accurate

Practical Considerations for Impedance Measurements of PCBs

Thank You!