APPLICATION NOTE

ANE012 | Coaxial PCB Connector
PCB-Transmission Line Design Guide

Olan Tsai

01. INTRODUCTION

The function of a RF (Radio Frequency) transmission line is a directing RF signal from a source to a load with minimal loss and distortion. Impedance plays an important role in signal transmission. Even if the trace impedance already matches, adding a connector can cause an impedance mismatch and thus signal distortions due to reflections. The purpose of this application note is to outline methods for improving signal transmission when a connector is added on a PCB.

1.1 Impedance

Impedance matching determines the transmission line performance. Characteristic impedance of a standard transmission line is expressed as:

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

According to the formula above, the main factors include: **R** (resistance per meter), **L** (inductance per meter), **C** (capacitance per meter), **G** (conductance per meter), and last but not least the angular frequency **ω**.

1.2 Types of RF Transmission line on PCB

The choice of the appropriate transmission line depends on the application.

Embedded Strip Line

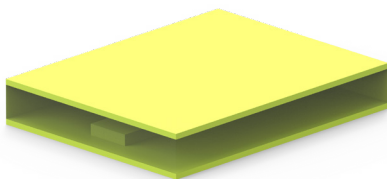There are ground planes above and below, and the dielectric around the trace is homogeneous.

Figure 1 *Embedded Strip Line*

Microstrip Line

The dielectric material above the trace is different from that below the trace. There must be a gound plane below the trace, while a grould plane above the trace may be omitted. Commonly applied on test PCBs
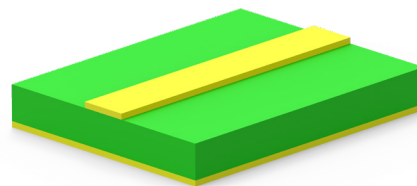

Figure 2 *Microstrip Line*

Coplanar Waveguide with Ground

The complanar wave guide basically consists of the signal trace with the two adjacent GND traces on the left and right. Additionally there might be a ground under the coplanar waveguide to reduce the parasitic discontinuities. This structure is common use in applications that need to connect to a RF connector or an antenna. The ground plane additionally helps to shield the transmission line against parasitic couplings to close metal objects
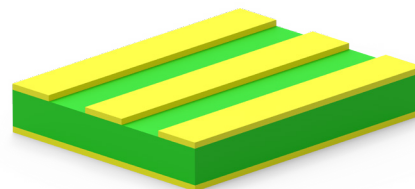

Figure 3 *Coplanar Waveguide*

02. RF TRANSIMISSION LINE DESIGN PARAMETER

The coplanar waveguide with ground is typically found on PCBs with a coaxial connector; therefore, we use this RF transmission line as example.
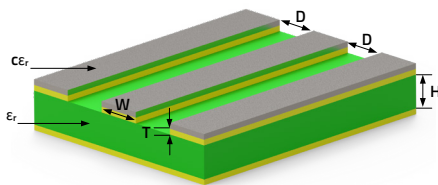

Figure 4 *Coplanar Waveguide*

Factors that determine the PCB transmission-line characteristic impedance are:

ε_r	Dielectric constant of the substrate
W	Conductor width
H	Substrate hight
T	Conductor thickness (copper thickness)
D	Distance of the signal conductor to the coplanar ground strips
$C\varepsilon_r$	Dielectric constant of the coating (solder resist)

Table 1 *Influencing factors on characteristic impedance*

Suggested design flow for a RF transmission line is as follows:

a) Select the PCB substrate:
  - For example : FR1, FR4, CEM1, ... etc.

b) Define the PCBs properties:
  - ε_r: Substrate dielectric constant
  - H: Substrate hight
  - T: Trace thickness
  - $C\varepsilon_r$: Coating dielectric constant

c) Control and adjust the value of W and D to reach the required characteristic impedance.

d) If step (c) cannot be achieved, please return to step (b) to select a different substrate height and trace thickness. Return to (a) when the newly selected properties in step (b) still cannot achieve the desired impedance.

03. RELATIONSHIP FOR EACH PARAMETER WITH CHARACTERISTIC IMPEDANCE

It is important to understand how each factor influences the transmission line.

Each paragraph below indicates how the characteristic impedance will change when only one of these factors is adjusted.

3.1 ε_r - Substrate Dielectric Constant

Many different types of materials can be used as PCB substrate. Each substrate has its own dielectric constant. The user needs to be aware of "PCB Material Dielectric Constant" given by manufacturers.

On the other hand, material dielectric constant will vary not only between vendors but with the frequency, too.

PCB Type	Dielectric Constant
FR-1	5.5-6
FR-4	4.2-4.7
CEM-1	4.6
CEM-3	5.0

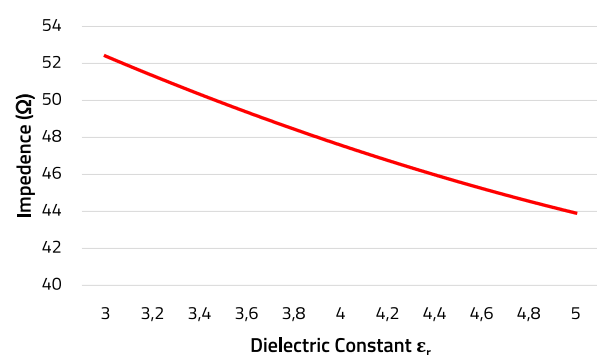Table 2 *Dielectric constants for common PCB materials*

Figure 5 *Influence of the dielectric constant of the substrate on the characteristic impedance*

The impedance (Z_0) is inversely proportional to the dielectric constant (ε_r)

3.2 W - Trace Width

For trace width design, the user needs to consider four things:

1. Narrow traces are more difficult to control
2. Wider traces need more space on the PCB.
3. Actual trace width may not be as designed due to manufacturing tolerance.
4. Because the wave impedance and the height of the dielectric material is fixed, these parameters will decide the trace width.
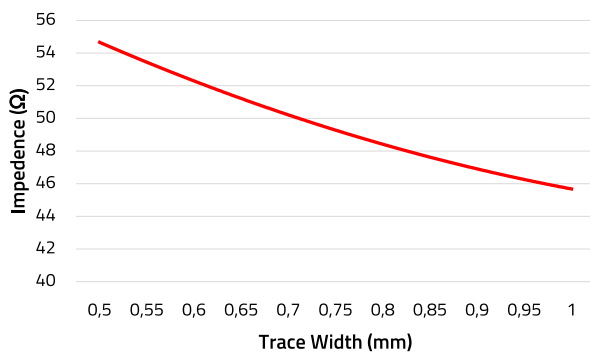

Figure 6 *Influence of the trace width on the characteristic impedance*

The impedance (Z_0) is approximately inversely proportional to the trace width (W).

3.3 H_ Substrate Height

The substrate height is the distance between the ground and signal layer of a PCB. The characteristic impedance reduces if the copper layers are closer together and the other parameters are fixed. To maintain the impedance of the trace, the trace width can be reduced. This in turn saves space on PCB. But please note, if narrower tracks are used, the tolerance of the impedance will be greater due to the undercutting during manufacturing process!
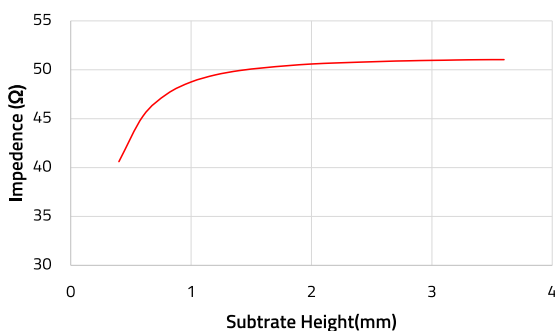
Figure 7 *Influence of the substrate height on the characteristic impedance*

The impedance (Z_0) increases more slowly when a certain substrate height (H) is exceeded.

3.4 T - Trace Thickness (height of the copper layer)

3.5 Trace Thickness (height of the copper layer)

Copper is commonly used as the PCB metal layer in layer heights of 0.5 oz/ft^2 (0.017mm), 1 oz/ft^2 (0.035mm) or 2 oz/ft^2 (0.070mm). The characteristic impedance reduces, if the Cu copper thickness (height of the copper layer) is increased.
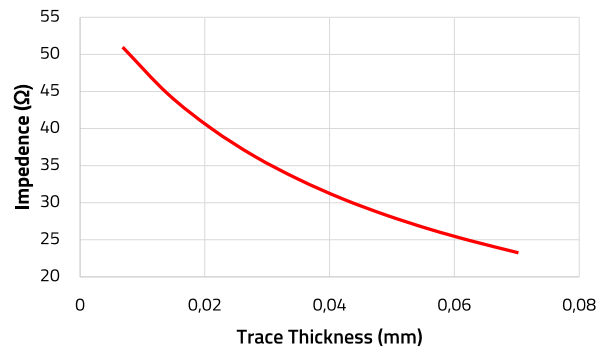
Figure 8 *Influence of the conductor thickness (layer thickness of the copper) on the characteristic impedance*

The impedance (Z_0) is approximately inversely proportional to the trace thickness (layer thickness of the copper T).

3.6 D - Distance between Trace and Coplanar Ground

This factor is very sensitive. It can influence the impedance by 1~2 Ω with only 0.01 mm variation, hence the PCB manufacturing precision needs to be checked when doing this adjustment.
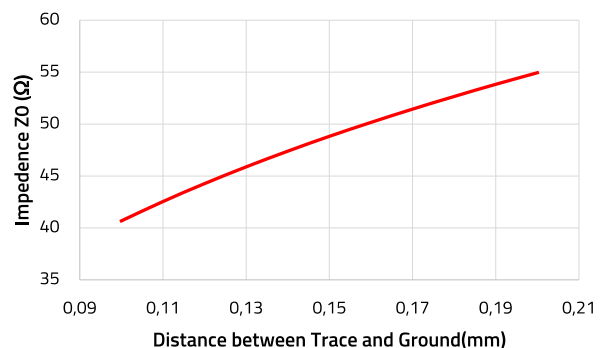

Figure 9 *Influence of the distance from the signal conductor to the coplanar ground strip on the characteristic impedance*

The impedance (Z_0) is roughly proportional to the distance between the signal conductor and the coplanar ground strips (D).

3.7 $C_{\varepsilon r}$ - The coating material's permittivity also has an influence on characteristic impedance. It should be considered early in the design stage in order to avoid deviation from calculated values.
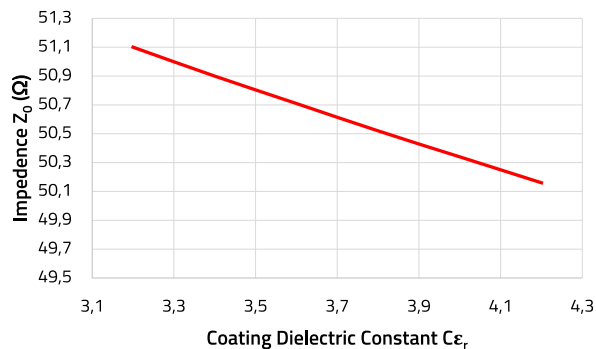

Figure 10 *Influence of the dielectric constant of the coating on the characteristic impedance*

The impedance (Z_0) is approximately inversely proportional to the dielectric constant of the coating ($C\varepsilon r$).

04. CONNECTOR PIN CONTACT TO PCB

When the connector center pin is connected to the PCB trace, it adds a certain height on top of the PCB trace due to its defined size. This height creates a discontinuity and should be fixed by adjusting the trace width on PCB.

4.1 End Launch Round Post Pin
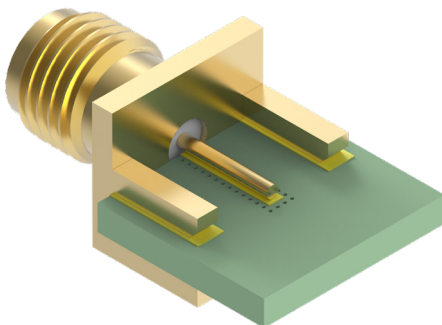
Figure 11 *Round post connectors have a larger center pin which creates significant discontinuity after soldering*

To match with the "microstrip line", the trace width needs to be changed to fit with the round post pin.
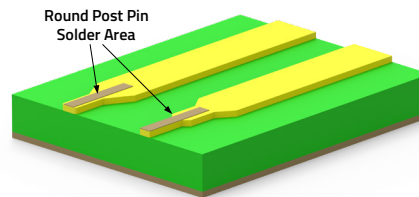
Figure 12 *Exemplary optimization of the trace geometry for improved matching*

If the impedance value is too low from the ideal matching impedance, try to use the shape shown on the left side of figure 12. The trace width near the connector center pin must be reduced. Making the trace width narrower at the end will make a better match.

For a coplanar waveguide, the impedance value can be increased by increasing the distance between the trace and its coplanar ground (GND). It is also possible to reach the ideal impedance by reducing the width of signal trace.
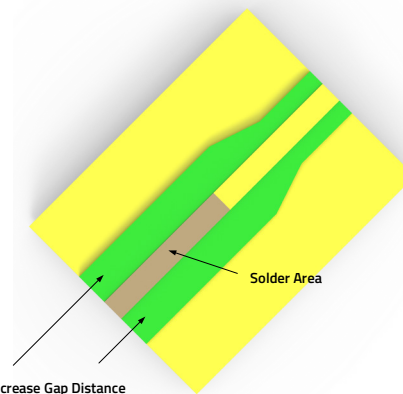
Figure 13 *Example of increase the gap distance to improve impedance matching*

4.2 End Launch Flat Tab Pin

This connector usually has a smaller center pin and has less influence on impedance discontinuity after soldering. When the flat pin makes the impedance lower, it can be raised by reducing the width of trace or increasing the gap distance between trace and its coplanar ground (GND).
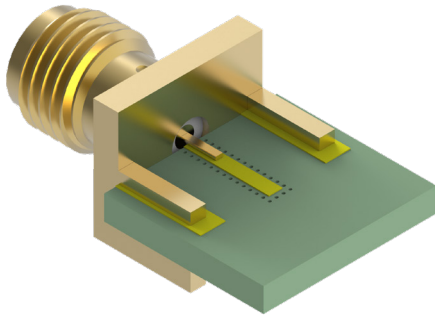
Figure 14 *End Launch Flat Tab Pin*
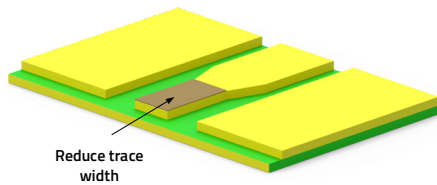

Reduce trace width

Figure 15 : *Example of reduced trace width to improve impedance matching*

4.3 THT Pin & SMT Pin

THT and SMT connectors are more difficult to reach the ideal match. This is often caused by the gap after the connector is soldered on to PCB. The gap will make the impedance value higher (Figure 16).
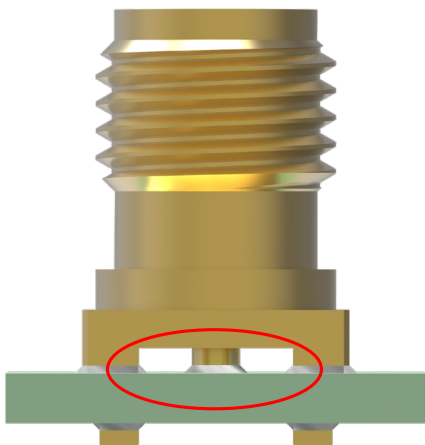

Figure 16 *Gap between PCB and connector*

Furthermore, THT and SMT connectors are mostly oriented perpendicular to the PCB surface. This leads to a lower impedance due to capacitive couplings. With THT signal pins, the extended pins cause a lower impedance; here, capacitive couplings are also the cause.

Mismatching of the impedance occurs in general and so also with THT and SMT connectors already at a very short distance. This makes it difficult to compensate for this mismatch by correcting the traces. Due to this challenge, THT

and SMT connectors are normally not used for applications in very high frequency ranges (typ. > 5 GHz).

How the signal is routed from a **THT** connector changes the impedance:

Extended center contact
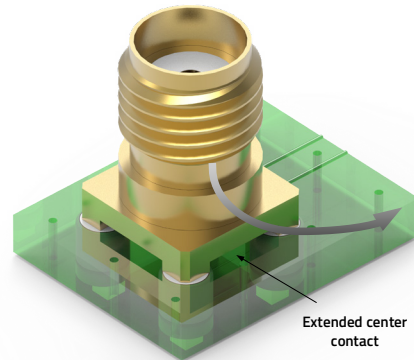
Figure 17 *Signal route: Connector interface PCB trace*
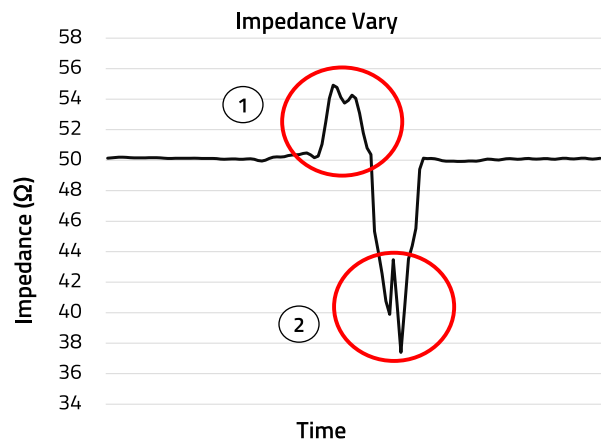
Figure 18 *Impedance in time domain*

1. Gaps make the impedance higher

2. Vertical area and extended center contact make the impedance lower

How the signal is routed from a **SMT** connector changes the impedance:
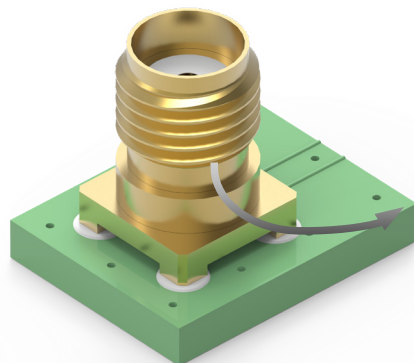
Figure 19 *Signal route: Connector interface PCB trace*
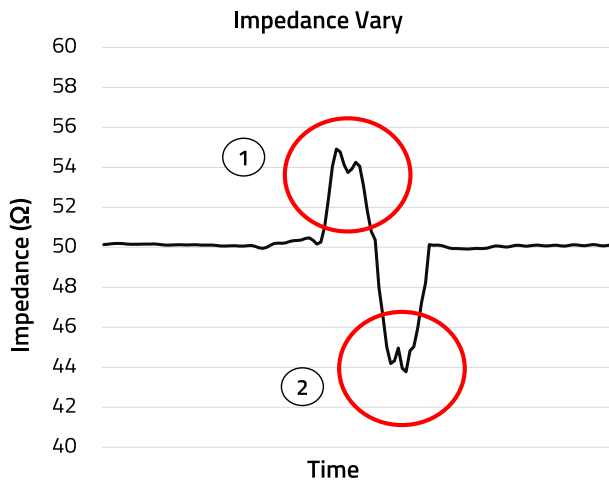
Figure 20 *Impedance in time domain*

1. Gaps make impedance higher

2. Vertical area makes impedance lower

Figures 17 to 20 show the impact of the connector construction and its connections to the PCB in the time domain. It was measured with a time-domain reflectometer (TDR) which determines the impedance of the measured construction by observing the reflected waveform. The change of the impedance over time can be recalculated via ε_r to the point of reflection.

How to Improve:

1. Gaps make the impedance higher: Use solder paste to fill up gaps to improve the impedance
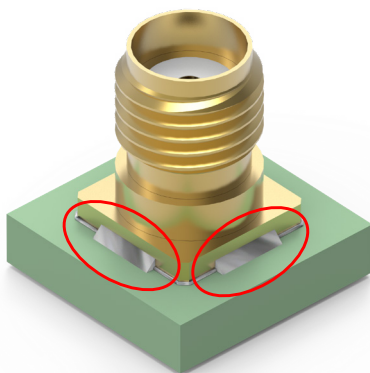
Figure 21 *Use solder paste to fill up gaps*

2. Vertical area makes the impedance lower: Increase the gap between the contact pin and the GND to reduce the impedance
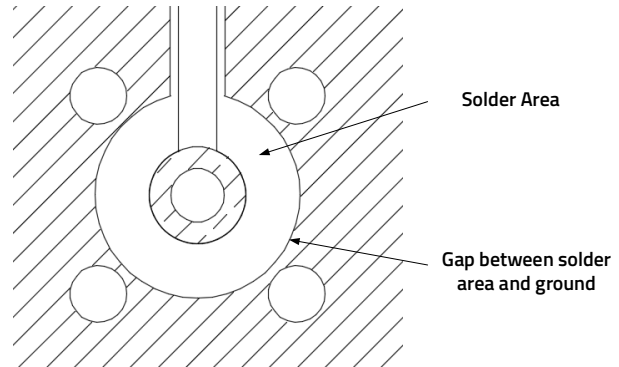
Figure 22 *Layout modification to improve the matching*

3. Extended connector legs make the impedance lower. Extended connector legs not only make the impedance value lower, they might also cause an antenna effect. Methods to solve this are:
   - Select the connector pin length to better match PCB thickness
   - Cut long legs for a significant improvement

05. OTHER FACTORS AFFECTING THE IMPEDANCE

Following a few points that will influence the transmission line impedance when making a PCB.

Following a few points that will influence the transmission line impedance when making a PCB:

5.1 PCB cut Line (End Launch Coaxial Connector Issue)

Fertigungstoleranzen bei der Nutzentrennung machen sich beim Fertigungsprozess der Leiterplatte bemerkbar. Wenn die Masselage nicht bis ganz an den Rand der Leiterplatte reicht, kann die Abweichung dazu führen, dass ein Spalt zwischen Massefläche und Steckverbinder entsteht und eine Fehlanpassung verursacht.
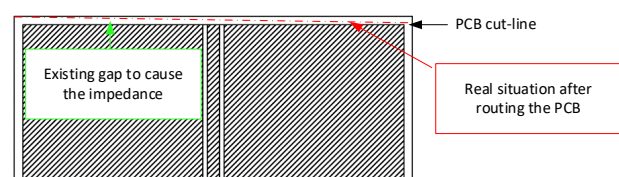
Figure 23 *Ground not extended to end of the PCB*
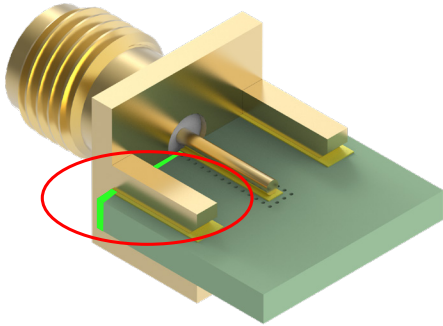
Figure 24 *After soldering the connector, a gap exist without ground*

We would suggest extending the ground layer on the connector side to the edge. If PCB routing tolerances issues occur, there will be no gap after the connector is soldered to the PCB.
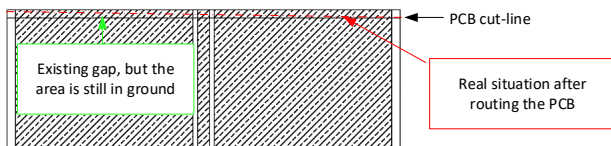
Figure 25 *Extended ground to the end of the PCB*

5.2 PCB Trim Line (End Launch Edge Card Coaxial Connector Issue)

The PCB routing cannot cut a sharp right angle and the round corners might cause a gap between the connector and the PCB. Because they could not meet closely this causes an impedance discontinuity (figure 26). To fix this, we suggest using Ø0.6-2 mm relief holes in the corners in order to remove the corner radius, as displayed in figure 27
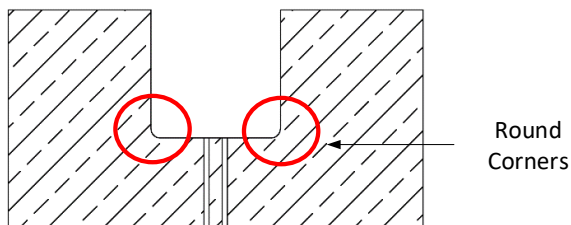
Figure 26 *Round corners cause poor mechanical matching of the connector*
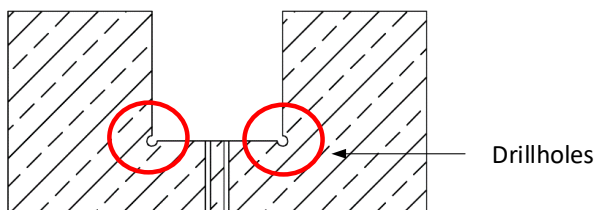
Figure 27 *Additional holes improve the fit of the connector.*

5.3 End Launch Connector and the Backside of a PCB

When an end launch connector is placed on a PCB, if the copper layer is not extended to the PCB edge, there would be a small gap in between. Use of solder paste to fill up the gap can be helpful to impedance control (figure 28).
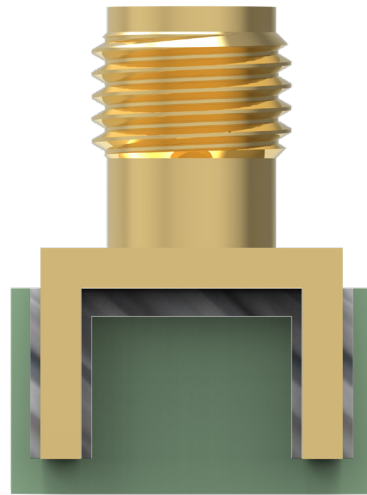
Figure 28 *Use solder paste to fill up the gap*

5.4 Durchkontaktierung

When using a coplanar waveguide design, it is suggested to add some via holes to the coplanar GND surrounding the trace, as shown in figure 29. The via holes connect the upper traces and the bottom layer which is the ground layer. Depending on the working frequency, the vias distance should designed to make sure that the coplanar GND-traces keep their GND potential.
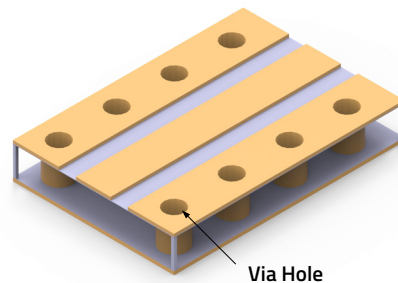
Figure 29 *Via implementation from the coplanar ground traces to the ground (GND) layer*

06. SIMULATION EXAMPLES

60312202114509

PCB End Launch Jack Round Post for 1.6mm PCB
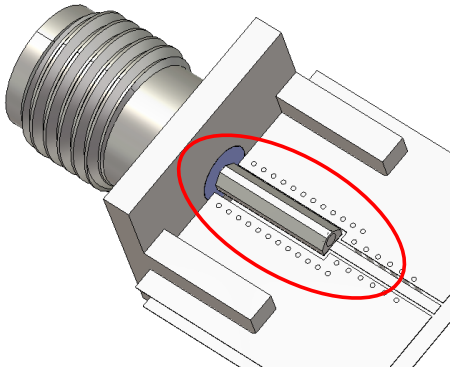
Figure 30 *Original Design*
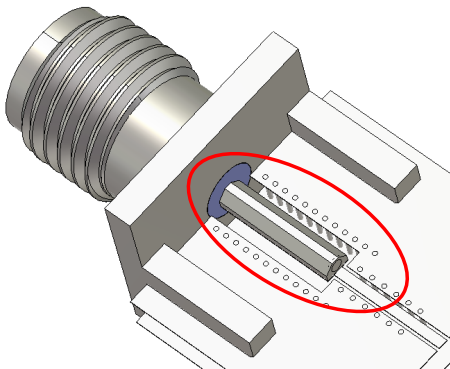
Figure 31 *Matching Design*

Increase the gap between trace and ground to increase the impedance and reach impedance matching.

Result of TDR measurement
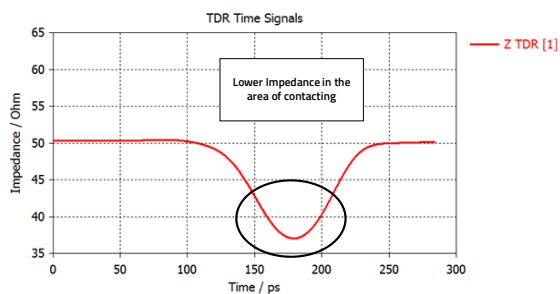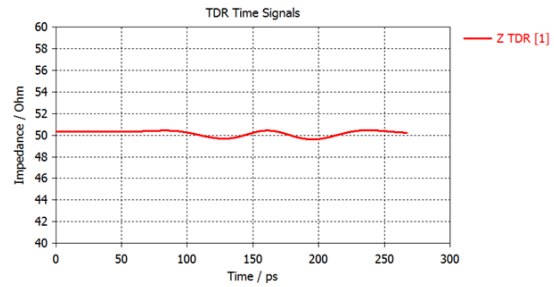

Figure 32 *TDR measurement (Original Design)*

Figure 33 *TDR measurement (Matching-Design)*
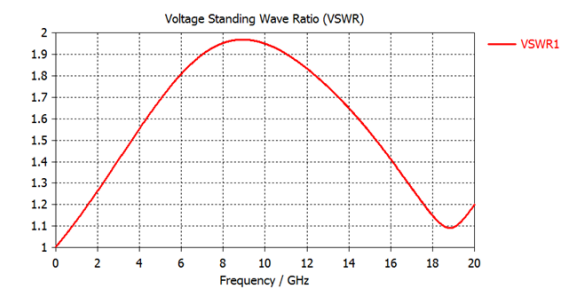
Result of S-Parameter measurement
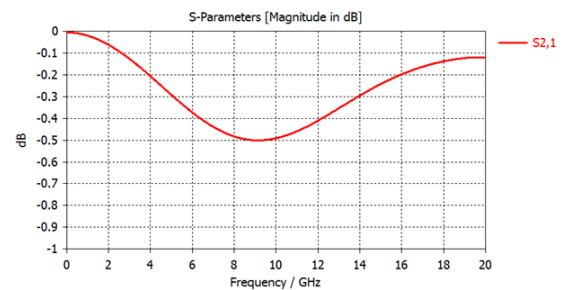

Figure 34 *VSWR measurement (Original Design)*

Figure 35 *Return Loss measurement (Original Design)*
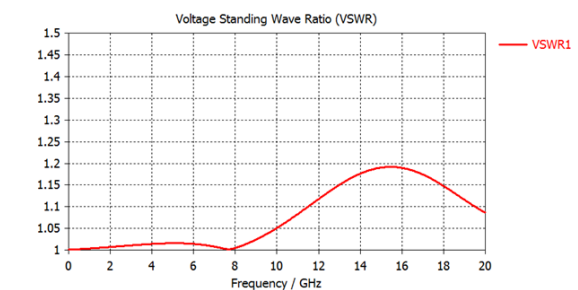
Figure 36 *VSWR measurement (Matching Design)*
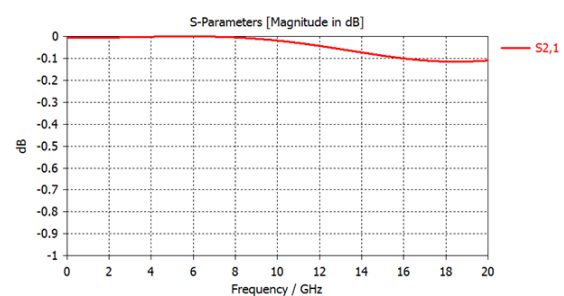
Figure 37 *Return Loss measurement (Matching Design)*

07. CONCLUSION

Simulation software is very useful when designing a transmission line, Software like AppCAD, for example, can simulate transmission lines with different factors. Using the result from AppCAD, software like HFSS can provide full-wave simulations, including both connector and PCB. Overall, using simulation software can save a lot of time.

Knowing the preciseness of PCB production will also help the user design a transmission line because most transmission lines are very narrow. If the PCB has wide tolerances from production, the user's test results will have huge difference compared to software simulations. Sometimes, roughness of copper layer is a factor that affects impedance of transmission line.

The quality of a transmission line is one part of transmission quality in a product. Users now should understand how to enhance transmission line quality after reading through these suggestions.

IMPORTANT NOTICE

USEFUL LINKS

Application Notes
www.we-online.com/appnotes

REDEXPERT Design Plattform
www.we-online.com/redexpert

Toolbox
www.we-online.com/toolbox

Product Catalog
www.we-online.com/products

CONTACT INFORMATION

appnotes@we-online.com
Tel. +49 7942 945 – 0

Würth Elektronik eiSos GmbH & Co. KG
Max-Eyth-Str. 1 · 74638 Waldenburg
Germany
www.we-online.com