



**SIGNAL
MICROWAVE**

Transparent Connections for 5G and WiGig Testing

Bill Rosas, CTO

August, 2018

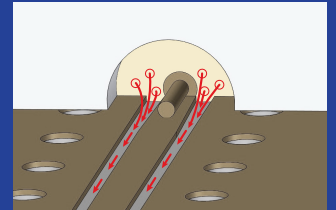
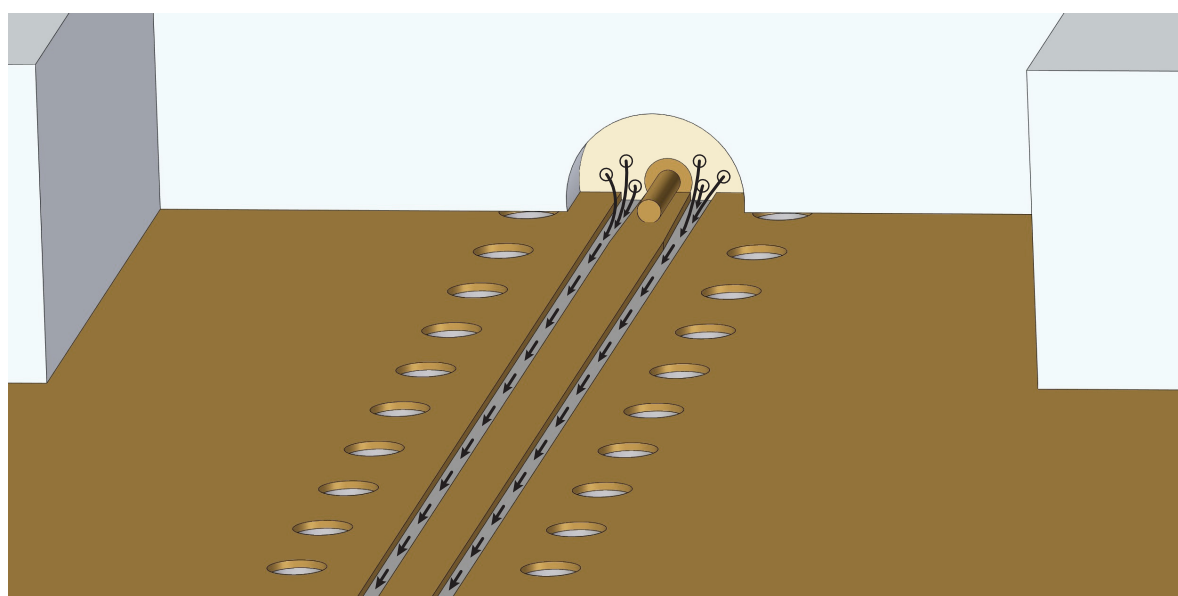




Table of Contents

Introduction	1
3D Tools Available to Designers.....	1
The Need for Using 3D Models	2
Designs Start with the Board Material	2
Transmission Line Designs.....	3
40 GHz and 70 GHz Launch Differences.....	4
Top Ground Launch for Multi-Layer Boards.....	4
Example of a Launch Design	5
How to Set-Up the Simulation.....	6
Comsol Settings for the Simulation	6
Optimizing the Connector to PCB Transition.....	7
Sources of Discontinuity:.....	8
The Pin Sitting on the Signal Line	8
RF Advantages of a Solderless Connection	8
Issues with Soldering	8
The Ground Contact with the PC Board	8
A Need for a Trained Operator of the Software.....	9
Sensitivity Analysis	9
Conclusion.....	9
Appendix: List of Signal Microwave Reference Designs	10
References	10



Representation of energy flow from the connector to the printed circuit board.



Introduction

Critical in many new projects is a transparent interface to the board - one you do not even notice is there. This connector to board interface is referred to as the board “launch”, where the RF energy transitions from the connector to the board. Shown in (Figure 1) is a typical design with 2.92 mm interface 40 GHz connectors on the edge of the board. For designers moving to higher frequencies, into the 20 GHz range and higher, just treating an SMA connector as a lumped 50 ohm element and mechanically joining it to the PCB or using a standard footprint will no longer suffice. For these designs it is now common that 3D electromagnetic software tools, such as supplied by COMSOL®, are used.

3D Tools Available to Designers

Designing a connector to board launch design is getting easier with 3D electro-magnetic (EM) software tools that have been available for years. Recently increased support from microwave connector suppliers are available as well.

Many microwave connector companies now provide 3D models of their higher frequency connectors that can be imported into 3D EM software tools. Some of these firms provide these models in a format that is only compatible with a specific 3D EM software package, a fee may also be charged to receive the model. Signal Microwave models, on the other hand, are generic and can be used in any of the 3D electromagnetic software packages.

PCB Board material information is also available in many 3D solver packages. The user can then simulate the 3D connector model with the PCB stackup and signal trace. Signal Microwave goes beyond that by supplying reference designs for common high-frequency board materials. These reference designs, which are optimized by Signal Microwave, come in a common CAD format and can be imported into many 3D software tools. Customers are able to start with these reference designs and evaluate them using their own software tools. The design can then be used as is, or the design can be modified to better fit the customer's requirements for final performance.

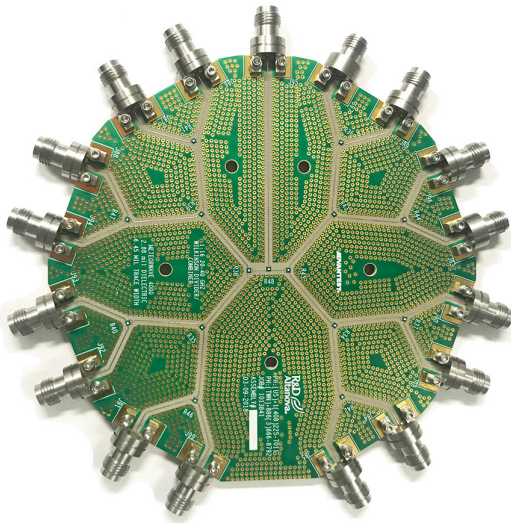


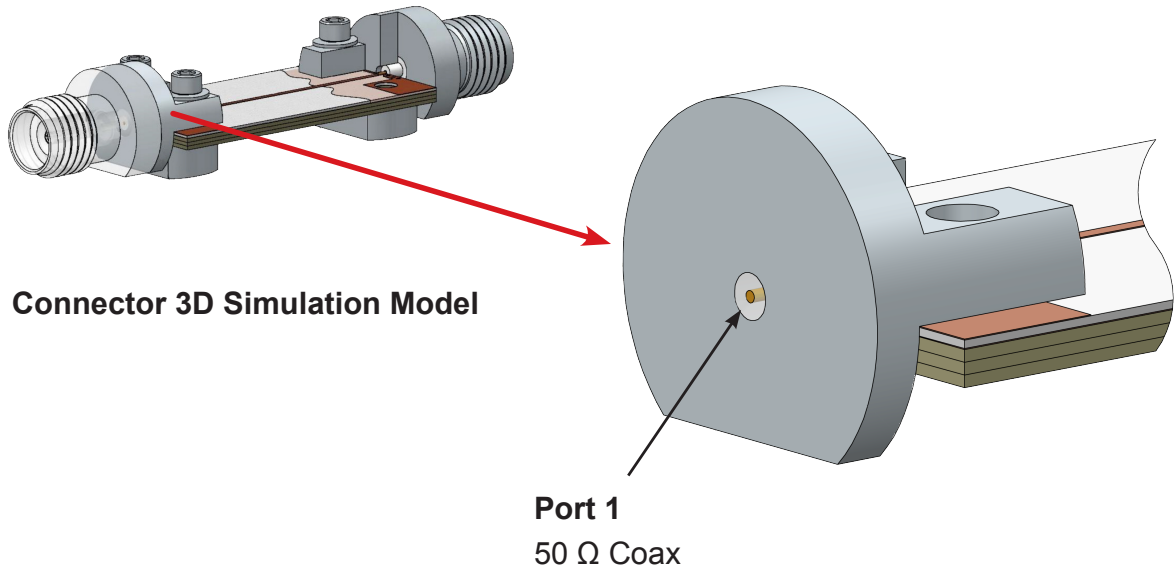
Figure 1

**20–40 GHz 1:16 Wilkinson Divider
(Courtesy of Advantest)**



The Need for Using 3D Models

In 2D circuit simulators, the components on a PCB that operate at high frequencies are modeled as planar circuits with limited support for 3D features like vias. This is not the case with a coaxial connector to PC board transition. The transmission line geometry in a connector is coaxial (circular) and on a board it is planar. To correctly model how these two transmission line types directly interact requires a 3D EM solver to address the complexities of the change in transmission line mode from coaxial to planar.



Designs Start with the Board Material

Connectors are compatible with a wide range of board thicknesses and dielectric constants. The purpose of the board will determine the combination of board material, dielectric constant, and dielectric thickness requirements. The common board materials being used in the industry for high frequency designs range in thicknesses from 4 mils (.004") to 12 mils (.012") thick with relative dielectric constants in the range of 3.0 to 3.5. Based on these parameters, today's high frequency connector's geometry match up well.

Here is a quick check list of required knowledge before starting the actual design:

- Frequency range
- List of components and features on the board
- Determine line widths to match
- Determine loss and power requirements
- Choose a transmission line structure
- Choose a suitable substrate
- Choose a suitable connector

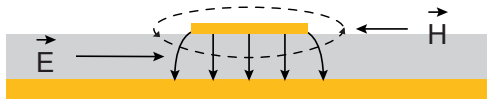


Transmission Line Types

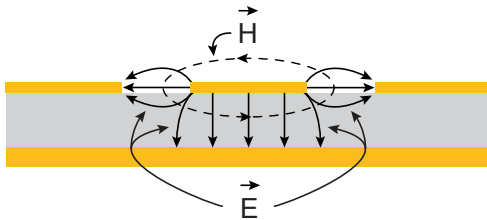
Microstrip Lines:

A microstrip transmission line has its own frequency limitation. The frequency is limited to when the transmission line itself will start to radiate and a drastic increase in loss will occur (references 1, 2). In general for microstrip, the thicker the material the lower the frequency range and the thinner the material the higher the frequency.

To efficiently transfer energy from a connector to a microstrip line on a board, there needs to be electrical connection to the microstrip ground layer. This can be accomplished when the board has one layer of dielectric and the bottom of the board is the ground conductor layer.



Microstrip Transmission Lines



GCPW Transmission Lines

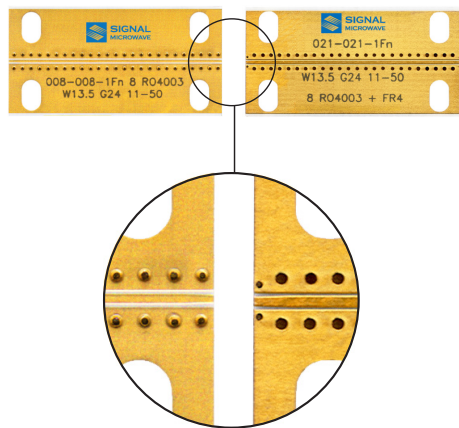
Most boards manufactured today are multi-layer so there is no direct access to the ground plane. Signal Microwave launch designs account for this and transition the ground connection through a short GCPW section. This uses the top ground connection of the connector to get the energy into the board's transmission line, then transfers the ground connection to the internal ground layer.

Grounded Coplanar Waveguide (GCPW) Lines:

A coplanar transmission line is when the ground conductors and the signal line conductors are in the same plane. The energy is between the conductors and traveling mostly in air, with some of the energy traveling through the dielectric directly under the gaps in the line.

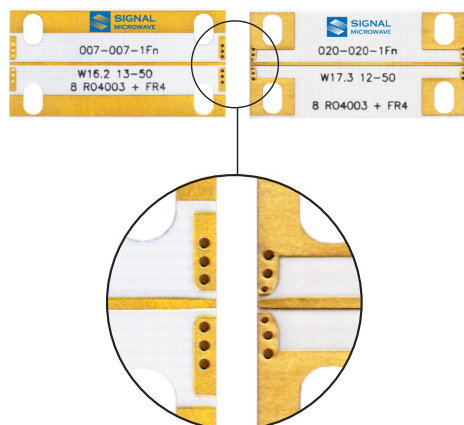
When a bottom ground is added to the structure, a microstrip mode of transmission also occurs where the energy is in the dielectric between the signal line and the ground plane. This mode does not occur at the launch unless there is a direct connection from the connector to the ground plane in the board. This is addressed in Signal Microwave launch designs as will be explained later.

40 GHz Test Board 70 GHz Test Board

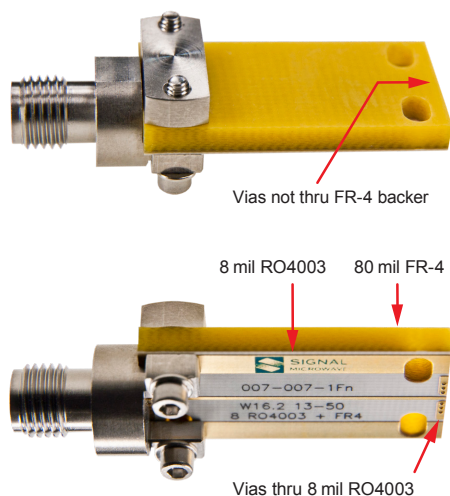


Grounded Coplanar Waveguide Test Board

40 GHz Test Board 70 GHz Test Board



Microstrip with Top Ground Launch



Board Construction Details

40 GHz and 70 GHz Launch Differences

The launch design will have its own frequency range limitation. The sizing and spacing launch geometry is proportional to the wavelength of the energy that is traveling through the transition. Having the smaller geometry is a challenge to the limitations of board fabrication processes and leads more inconsistency in the performance of the high frequency boards after fabrication. The lower frequency boards are not as sensitive to the variations inherent in board processing and perform more consistently between individual boards after fabrication.

On the left are 40 GHz and 70 GHz bandwidth grounded coplanar waveguide (GCPW) and microstrip test boards. These are standard Signal Microwave test boards made using 8 mil thick Rogers RO4003 material with an FR-4 stiffener.

Shown in the close-ups are comparisons of the 40 GHz launch designs and 70 GHz launch designs. The 70 GHz launch has different size vias, notably the first vias on either side of the trace, and they are spaced closer to the trace and the edge of the board. The 40 GHz launch designs have larger vias and are not spaced as tight.

Top Ground Launch for Multi-Layer Boards

Top Ground Launch

For both microstrip and GCPW transmission lines, when implemented in a multi-layer board configuration, the ground layer for the microwave transmission line is buried. The connector will not have access to it no matter how the connector is configured.

The launch designs developed by Signal Microwave incorporate a short section to transition the ground connection from the top of the board to the buried ground layer.

The small set of vias at the edge of the microstrip board is a GCPW section used to transfer the energy from the connector to the board using only the top ground connection of the connector.

Signal Microwave Board Construction

The 8 mil RO4003 top layer is processed separately including vias and plating. Then the RO4003 layer is laminated to the FR-4 layer which acts as a stiffener. The FR-4 layer does not have through vias that can cause resonances and no back drilling is required.



Example of a Launch Design

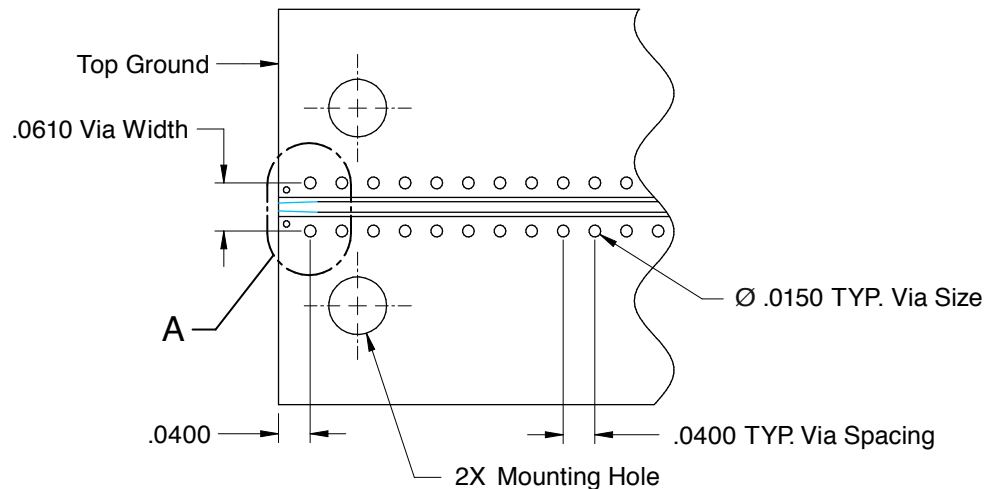
All of these designs are for multi-layer boards where there is not direct access to the microwave transmission line ground plane. This ground plane is buried in the board directly under the transmission line details on the top of the board.

To launch the energy to the board, the coplanar mode without the bottom ground is used first. Once the energy is on the board, the vias guide the energy down to the dielectric underneath for the portion of the energy that will be in the microstrip mode.

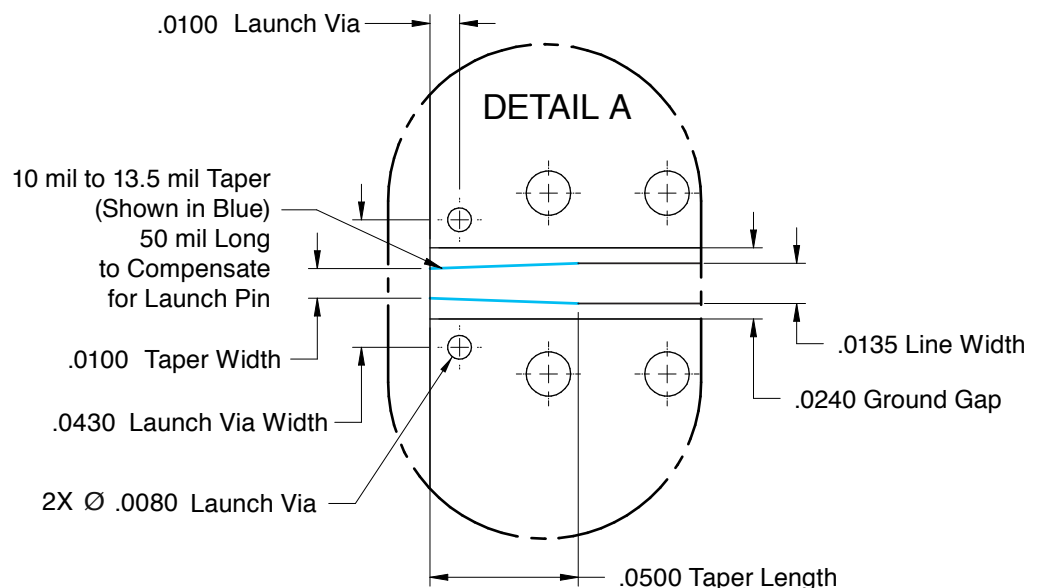
Key elements of the design include:

- Via size, placement, and spacing
- Circuit ground spacing to match the connector
- Signal line width to create a 50 ohm transmission line
- Compensation of the signal line (taper) to account for the connector pin

Overview of the Complete Board Layout



Critical Details for Electrical Performance





How to Set-Up the Simulation

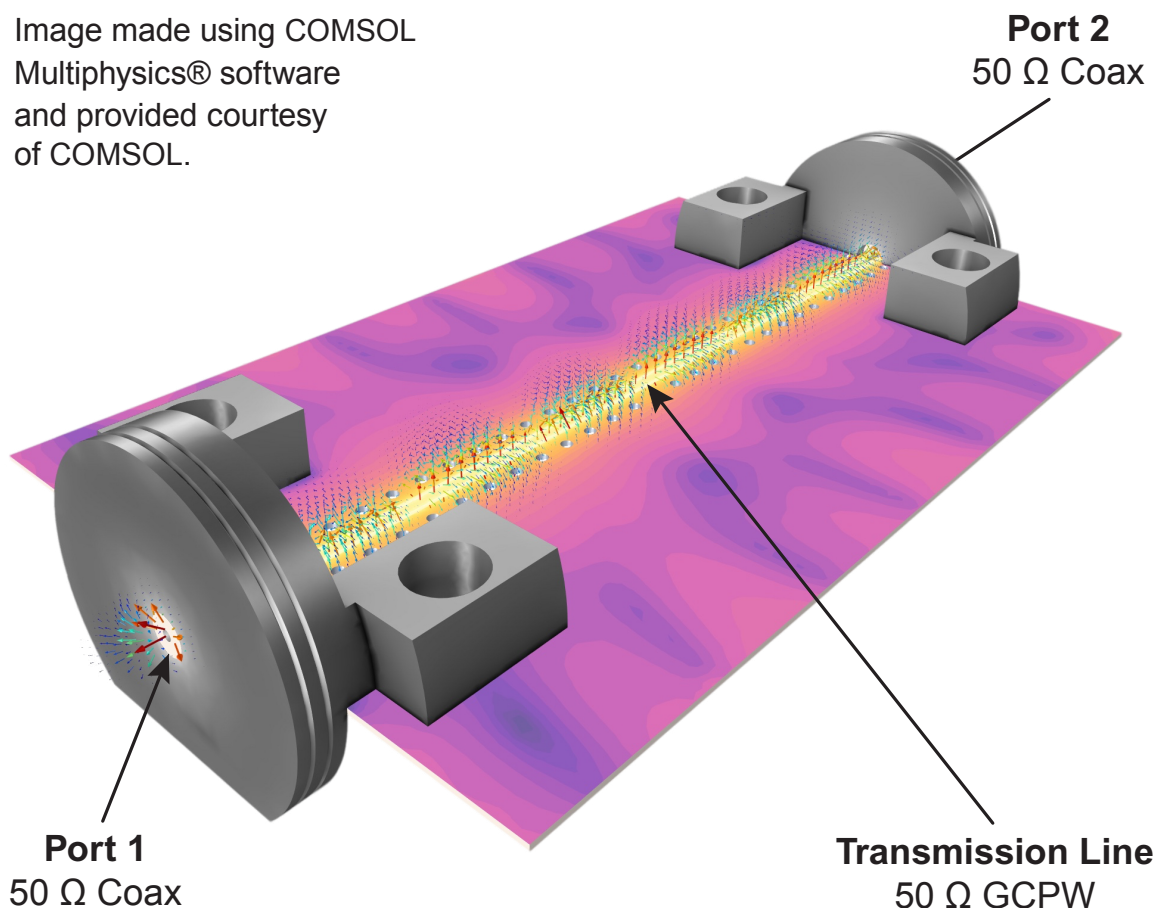
Figure below shows a short transmission line structure with 3D models of the connector on either side. Signal Microwave supplies 3D models of the launch portion of their board mount connectors. COMSOL Multiphysics® software is a 3D modeling package with has many of these models in their library along with common board materials. Using such a structure it is easy to set up a 2-port simulation model which can be used to optimize board designs.

Creating a through line with connectors on either end and creating the ports at the end of each connector simulation model will create a simulation of a structure that can be measured. The most efficient way to optimize the transition is by defining the ports as close as possible on either side of the section to be optimized. This would require creating a port in the transmission line of the board.

Comsol Settings for the Simulation

The domain between the inner and outer conductor of the edge launch connector is filled with dielectric. That dielectric is Neoflon and has a Dk of 2.5. All metal parts including the edge launch connectors, GCPW, metalized vias, and ground plane are modeled as Perfect Electric Conductor (PEC) for low frequency prototypes.

Image made using COMSOL Multiphysics® software and provided courtesy of COMSOL.



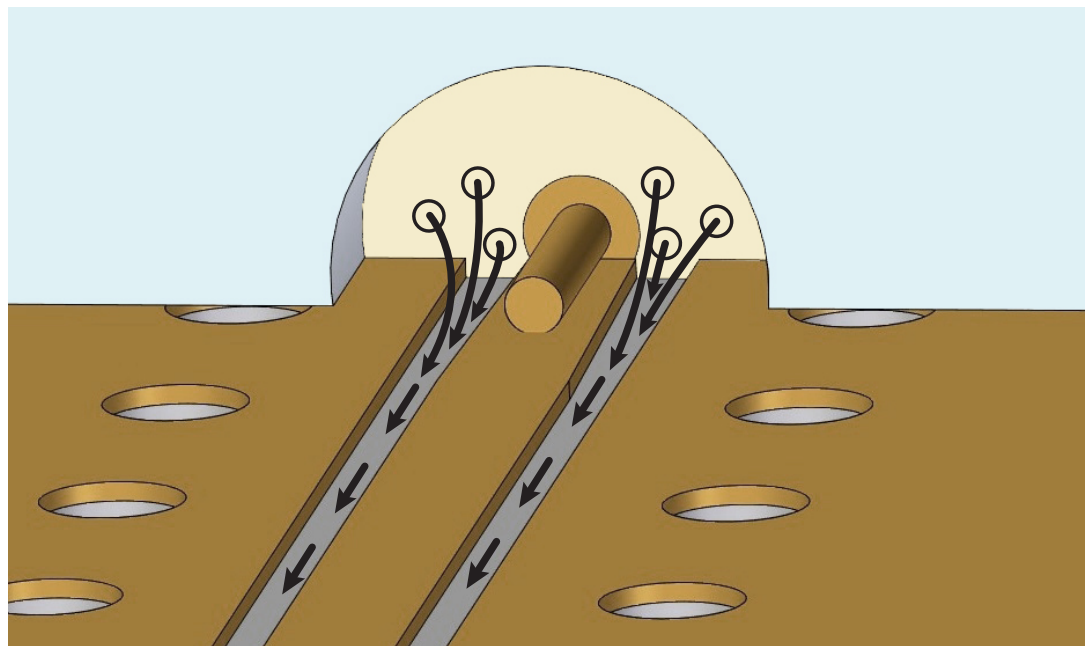


The lossless PEC condition is replaced by Transition Boundary Condition and Impedance Boundary Condition to include loss factors and surface roughness at higher frequencies. The entire modeling domain is defined by Scattering Boundary Condition that represent an open space. The physics-controlled mesh is chosen to get all domains meshed with a tetrahedral mesh with a maximum element size of five elements per wavelength so that the wave is well-resolved. The maximum mesh size is automatically scaled by the material properties in the dielectric substrate. The parts in the edge launch connectors are meshed manually with a finer resolution to provide good resolution of the curved surfaces.

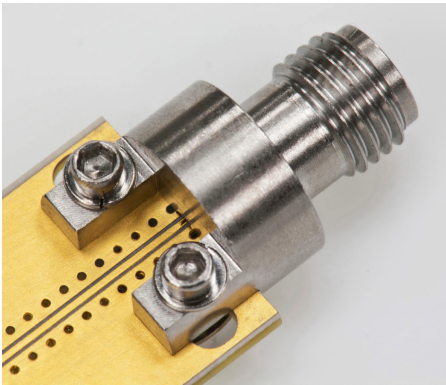
Optimizing the Connector to PCB Transition

Below is a representation of the energy flow from the coax to the PCB. This is a simplified representation showing only the energy in the air portion of the transmission line. A significant amount of the energy is in the substrate and transitions from the dielectric of the connector to the substrate between the top and bottom conductors.

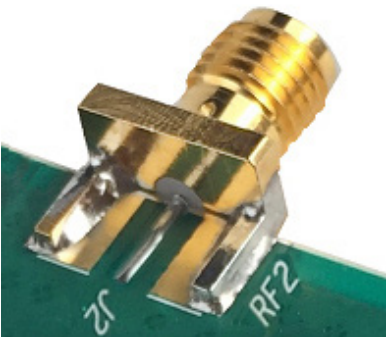
To fully capture the complexity of this transition at higher frequencies, a 3D simulation tool is required. Once the model is created and the ports are set to measure the energy flow (S-parameters) an optimization of the launch design can be done.



Representation of energy flow from the connector to the printed circuit board.



Signal Microwave Solderless Connector



Typical Edge Mount Soldered Connector

Sources of Discontinuity:

The Pin Sitting on the Signal Line

The pin of the connector adds capacitance to the section of the board where it is electrically connected. This increases if the pin is soldered so soldering is not recommended. The signal line of the PC board where the connector makes contact needs to be adjusted to reduce the effect of this extra capacitance.

RF Advantages of a Solderless Connection

Soldering is the traditional way of creating the electrical connection between the connector and the PC board. When modeling the transition a predictable geometry is required for the model to be valid, especially at a point as critical as where the pin makes contact with the board.

Issues with Soldering

Any soldered added will change the electrical characteristics at high frequencies by adding capacitance to the transmission line. This can be accounted for in the model but soldering itself is inconsistent compared to a solderless contact so will lead to more deviation in performance.

The Ground Contact with the PC Board

For a GCPW the top ground spacing can be matched to the ground contact points of the connector. For a microstrip line there is no top ground so there is not much to optimize to improve performance. If a short GCPW section is added to edge of a microstrip line, this gives a structure that can be optimized to the connector.

The largest discontinuities occur at the section between the launch pin of the connector and the edge of the board. To simulate this the TEM mode field is excited at the coaxial type lumped port, port 1. The field then propagates through the port 1 edge launch connector and excites a planar mode on the GCPW. The symmetric electric field in the coplanar waveguide is confined from the center conductor outwards to the conductors on each side and underneath. The listener port, port 2, is also terminated by the same type of coaxial lumped port.



A Need for a Trained Operator of the Software

The person running the simulation will need to know RF transmission line basics. With this knowledge they can properly oversee the implementation and then results of the simulation. The software can quickly and easily explore a range of design features as defined by the user and provide data for each specific combination of features. It cannot, for example, tell the user that a feature not in the design needs to be added, or a feature in the design needs to be removed. These decisions need to be made by the user and the design modified for further analysis through simulation. In this way the software tool can be used to identify on an optimized design and explore sensitivities in the design.

Sensitivity Analysis

The user will need to have an understanding of board processing techniques and variances inherent in the board fabrication process. Understanding the weaknesses and strengths in holding tolerances in the fabrication process can guide the user in the sensitivity analysis. Try to make the most difficult board processing features the least sensitive in the design.

Conclusion

Projects will benefit from a well-designed board launch. Without the efficient and well behaved transition of energy into and out of a board, the determination of the performance of a board, or components on a board, cannot be properly determined. In test, yields can increase when components are not failed due to the degradation of test results due to the connector to board transition performance. In design verification the test results accuracy will be improved as well. High performance board launches and transmission lines are achievable in projects where their performance is critical. While not “transparent”, it is close enough that the connector transition does not interfere with board measurements.



Appendix:

List of Signal Microwave Reference Designs

2.88 mil	MetWave4000 Microstrip 70 GHz	Layout 065
4 mil	RO3850 Microstrip 70 GHz	Layout 050
4 mil	RO3850 GCPW 70 GHz	Layout 050
4.2 mil	Megtron 6 Microstrip 50 GHz	Layout 047
4.9 mil	AstraMT-77 Microstrip 110 GHz	Layout 044
4.9 mil	AstraMT-77 GCPW 110 GHz	Layout 045
5 mil	RO3003 Microstrip 70 GHz	Layout 041
5 mil	RO5880 Microstrip 70 GHz	Layout 032
5 mil	RO5880 GCPW 70 GHz	Layout 031
5 mil	RO5880 GCPW 70 GHz	Layout 066
6.46 mil	Megtron 6 70 GHz	Layout 070
7.3 mil	RO4350 LoPro Microstrip	Layout 039
8 mil	RO4003 Microstrip 40 GHz	Layout 007
8 mil	RO4003 GCPW 40 GHz	Layout 008
8 mil	RO4003 Microstrip 70 GHz	Layout 020
8 mil	RO4003 GCPW 70 GHz	Layout 021
5 mil	RO5880 Microstrip 70 GHz	Layout 031
10 mil	RO4835 Microstrip 70 GHz	Layout 032
10 mil	RO4350 Microstrip 70 GHz	Layout 033
8 mil	RO4360G2 Microstrip 40 GHz	Layout 073
8 mil	Megtron 6 Microstrip 70 GHz	Layout 034
8.7 mil	RO4003C LoPro 40 GHz	Layout 063
10 mil	RO4350 GCPW 70 GHz	Layout 035
10 mil	RO3003 Microstrip 70 GHz	Layout 040
10 mil	isolaAstra Microstrip 70 GHz	Layout 036
10 mil	Isola AstraMT GCPW 70 GHz	Layout 038
10 mil	isola Tachyon Microstrip 70 GHz	Layout 037
10.32 mil	Megtron 6 Microstrip 70 GHz	Layout 071
12 mil	RO4360G2 GCPW 40 GHz	Layout 048
12 mil	RO4360G2 GCPW 40 GHz	Layout 052

References

1. B. Rosas "Optimizing Test Boards for 50 GHz End Launch Connectors: Grounded Coplanar Launches and Through Lines on 30 mil Rogers 4350 with Comparison to Microstrip," Southwest Microwave Inc., Tempe, Arizona, 2007.
2. B. Rosas "The Design and Test of Broadband Launches up to 50 GHz on Thin and Thick_Substrates," Southwest Microwave Inc., Tempe, Arizona, 2007.