Design and Assembly of Edge Launch mm Wave Connectors

INTRODUCTION

Successfully launching an RF connector to a PCB at millimeter wave frequencies requires a carefully engineered plan for design, simulation, and installation. The key factors for low relfection performance at high frequency include:

1. RF connector and simulation
2. PCB landing pad design and simulation
3. Soldering process and fixtures

If any of thes factors are neglected, the electrical performance of the launch may not be optimized. In this document, we will analyze these parameters to see how they affect RF performance.

RF CONNECTOR DESIGN AND SIMULATION

The goal of any RF connector design is to eliminate or reduce reflections, maintain a constant impedance and provide a robust mechanical connection. A standard 2.92 mm connector uses a customized insulator designed to maximize signal propagation and mechanical support (Figure 1 below).

While this design is electrically efficient, it enables the possibility of introducing FOD into the air line during soldering (Figure 2 below), which causes unwanted reflections.

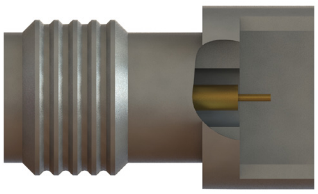

Figure 1: 1521-60051

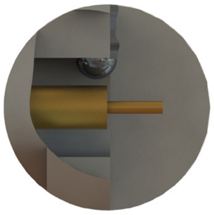

Figure 2: Solder in Air-Line

To combat this, Figure 3 (below) shows an alternate design that utilizes an insulator at the exit of the connector. The insulator provides a barrier against solder and other types of FOD. With a simple re-design of the connector body and contact, the electrical effects of the insulator can be mitigated.

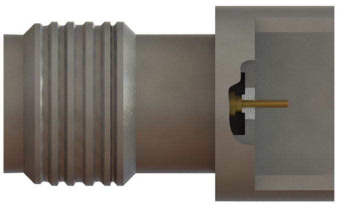

Figure 3: 1521-60067

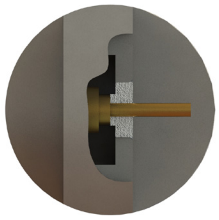

Figure 4: Cross-Section Showing Insulator

The exit diameter should closely match the transmission line width of the PCB making a smooth transition from the connector to PCB.

Phone: (561) 840-1800

Email: sales@svmicro.com

PCB LANDING PAD DESIGN AND SIMULATION

PCB footprints should be designed to match an RF connector to a specific PCB stackup. The characteristics of a PCB that need to be considered for proper matching in an edge launch connector for CPW launch are:

1. Dielectric constant and thickness of signal and ground layer (S1 and S2)
2. Trace width (W)
3. Gap width (G)
4. Ground layer thickness and material (T1, T2, T3)
5. Additional plating and masking

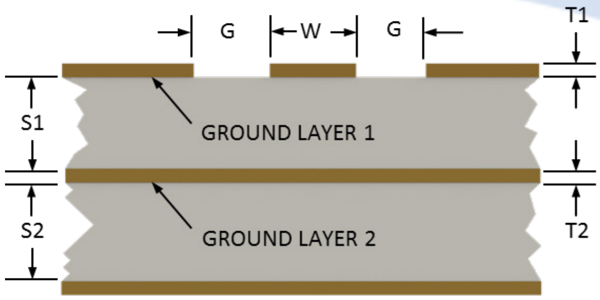

Figure 5

Trace and gap width and substrate thickness control the impedance of the transmission line. There are many calculators available to properly determine these dimensions, but attention should be paid to the correlation between theoretical performance of a simulation and measured performance of a finished product.

The difference between an accurately matched connector and footprint and one that has not been optimized is pronounced, especially at high frequencies. Simple modification to the landing pattern to maintain 50 ohms impedance throughout the pin to transmission line can be modeled and analyzed in electromagnetic simulation software.

The below images show an example of what a matched transmission line might look like compared to a mis-matched transmission line. Our electrical engineers provide footprint design optimization service at no charge.

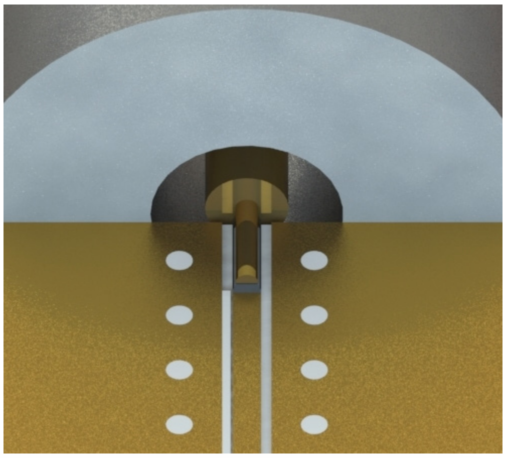

Figure 6: Mismatched Connector/Trace

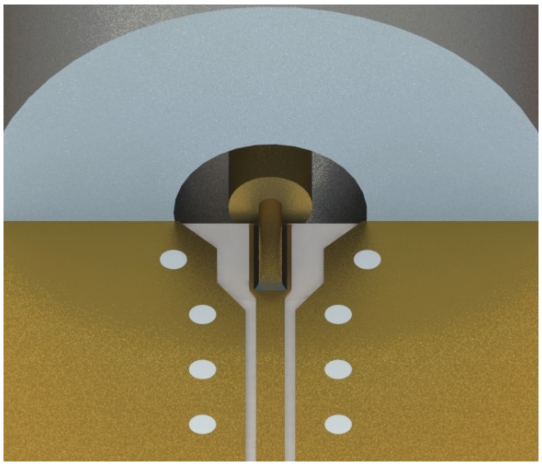

Figure 7: Matched Connector Trace

SOLDERING PROCESS AND FIXTURES

One of the most difficult parameters to control during hand soldering is localized solder temperature. Excessive heat can deform the insulator creating electrical and mechanical problems. Too little heat can create a brittle 'cold solder joint' when all the metal surfaces are not brought above the melting temperature of the solder.

Figure 8

To draw heat away from the solder joints and hold the center pin in place while soldering, we recommend using mechanical solder buttons. Standard solder buttons are available from SV Microwave.

The ideal connector to PCB joint contains no air gap between the edge of the PCB and the connector. Edge plated PCBs or PCBs with castellated vias can help with this, but this is not always an option. There are techniques that can be used to mitigate the potential for a gap on a standard (non-edge plated) application. In the below simulation study, you can see that even a .001" gap can cause electrical issues.

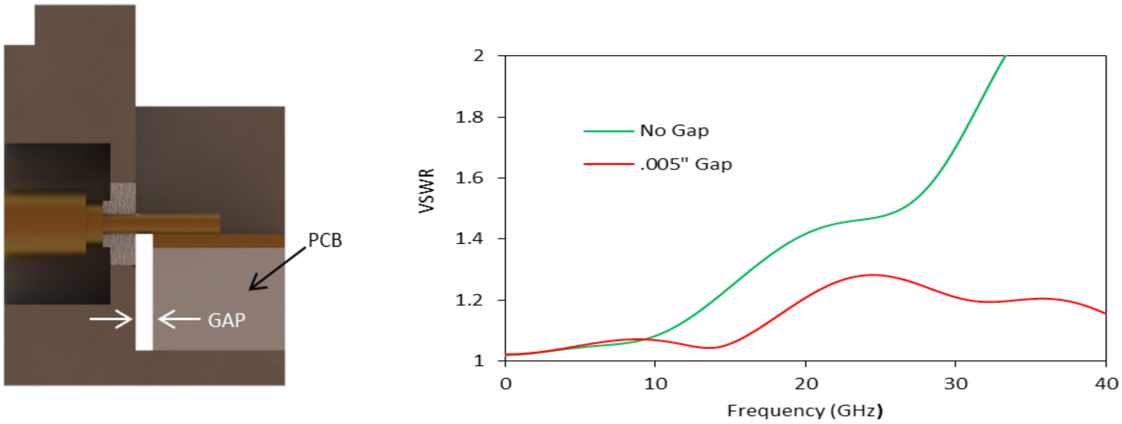

Figure 9: Electrical Simulation of Gap

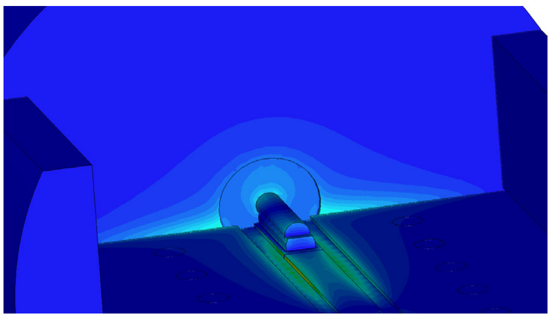

Figure 10: E-Field Distribution with Gap

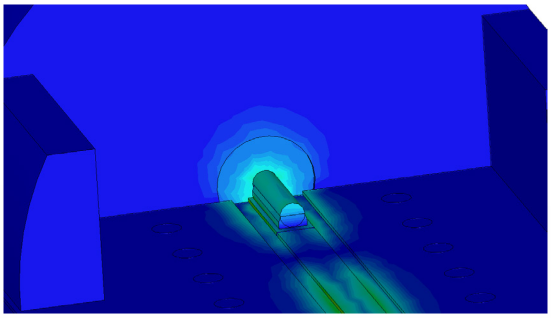

Figure 11: E-Field Distribution without Gap

In order to reduce the potential for a gap, the edge of the PCB should be as uniform as possible. Post processing (grinding down) any stubs or non-uniformities can help with this. When hand soldering, the use of a soldering fixture is recommended to ensure the connector remains ideally positioned throughout the soldering process. SV Microwave can custom design and build soldering fixtures for your specific application.

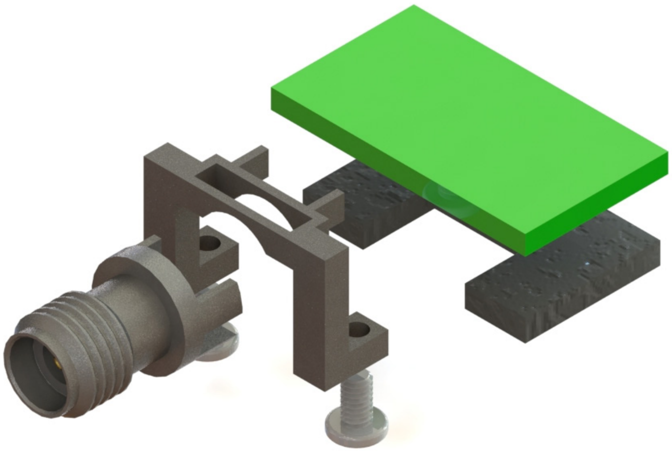

Figure 12

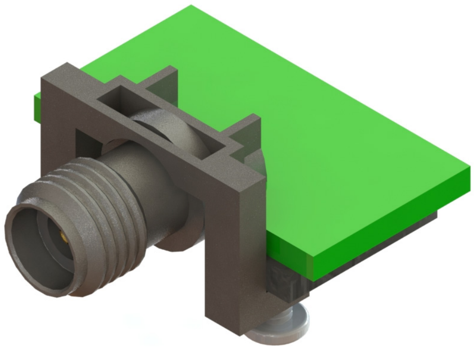

Figure 13

Ideally the soldering fixture clamps the connector in place in the Z-axis and holds it flush the edge of the PCB in the X and Y axis. Often custom fixtures are required as no two connector/fixture combinations are the same.

It is important to ensure that the connector body is making electrical contact at all points. This can be visually verified by confirming that solder fillets exist in all joints between the landing pad and the connector body. Improper grounding can cause insertion loss ‘suckout’ as shown in the plot below.

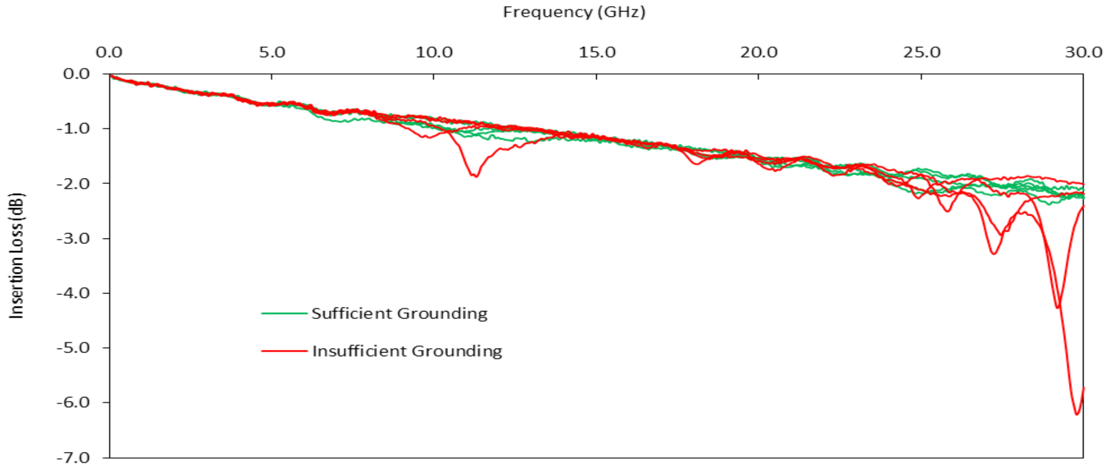

Figure 14