

RF Design Considerations

By: Andrew Chen, VP of Technology

With Contributions from Sean Querry, Electrical Engineer Intern

December 16, 2015

Version 1.0

RF hardware design is complex and oftentimes a challenge for embedded designers. For instance, parasitic effects that are not typically a concern in low-frequency applications can have a major impact on radio frequency performance as you move into higher frequency ranges. Different wireless hardware designs also require special certification requirements and designing and testing wireless hardware takes special skills and equipment. These are only a few factors that must be considered.

REVISION HISTORY

Version	Date	Notes	Approver
1.0	16 Dec 2015	Initial Release	Andrew Chen

CONTENTS

Introduction	4
What is RF?.....	4
Model for a Resistor	5
Different Package	5
Model for an Inductor	6
Model for a Capacitor.....	6
PCB Design Considerations.....	6
Characteristic Impedance Matching.....	6
Transmission Line	7
Antenna Placement	7
Types of Antennas	7
Parasitic Effects during Design	9
DFX Considerations	11
Design for Passing Certification (FCC/CE/IC)	11
Design for Manufacturing and Test	11
Conclusion.....	11

INTRODUCTION

RF hardware design is complex and oftentimes a challenge for embedded designers. For instance, parasitic effects that are not typically a concern in low frequency applications can have a major impact on radio frequency performance as you move into higher frequency ranges. Different wireless hardware designs also require special certification requirements and designing and testing wireless hardware takes a special set of skills and equipment. These are only a few factors that must be considered. This white paper explores some of the key components of RF design and their implications on system performance. Topics include a review of RF and the characteristics engineers should be aware of, basic components and potential parasitic effects, PCB design characteristics including impedance matching, transmission line, and antenna selection and placement, and DFX considerations.

WHAT IS RF?

RF, short for radio frequency, commonly refers to wireless communications occupying the radio frequency portion of the electromagnetic spectrum. RF has many applications as you can see in [Figure 1](#), but of what characteristics must users be aware? For wireless communications, frequencies can range from several hertz (Hz) to over 300 gigahertz (GHz). Devices such as AM/FM radios, remote controls, TV, cellphones, GPS, and some medical devices operate in relatively low frequency ranges, typically less than one GHz. Compared to frequencies in the five GHz or ten GHz range, these low frequencies offer a wider coverage area for a given transmit power; they also have the advantage of being able to propagate through dense objects, not requiring line of sight. For example, cellphones are designed to operate within the 900 megahertz (MHz) to 1.8 GHz range because they need to penetrate walls and other dense objects for indoor coverage. At 2.4 GHz, signals have difficulty going through dense objects like concrete and metal, which attenuate the signal. These harsh environments cause difficulty for 2.4 GHz signals, therefore roaming and antenna diversity are necessary to ensure devices stay connected. Again, a drawback of higher frequencies is that signals require line of sight for operation. However, higher frequencies offer the advantage of higher available channel bandwidth which translates to higher data throughput.

For example, Wi-Fi designs using the 802.11 a/b/g/n standards have a minimum 20 MHz channel bandwidth in the 2.4/5 GHz range. With 802.11ac, the channel bandwidth ranges from 80 MHz to 160 MHz in the 5 GHz range only. This allows for an increased data rate but if the frequency increases, the fractional bandwidth decreases making system design more difficult. Also, higher frequency signals attenuate much more quickly in air. In short, high frequencies cannot drive a very high output power. To compensate for this however, you can easily design a higher gain antenna with little size penalty. Thus, with high frequency designs, you cannot deliver a high output power, but a higher gain antenna can be used to compensate for the weaker signal.

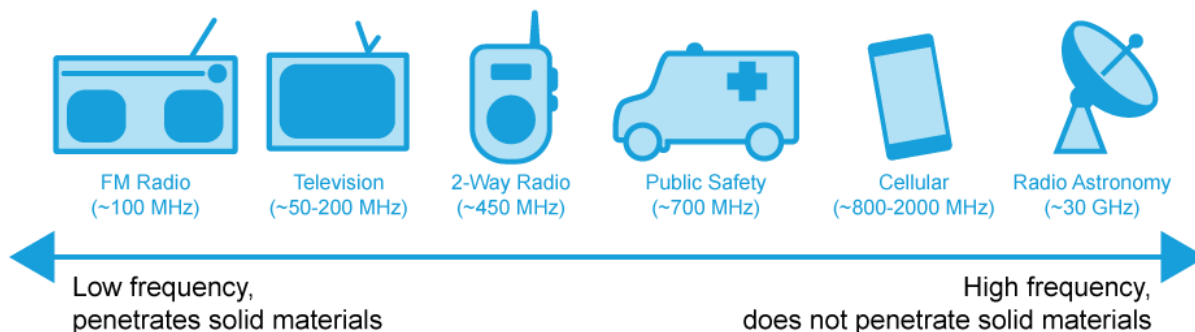


Figure 1: Uses of the RF spectrum

BASIC COMPONENTS

In any electronics hardware design there are three basic passive components: resistor, inductor, and capacitor. In digital designs these components can generally be assumed as ideal and parasitic effects can be neglected. This is not the case with RF. In RF designs, frequency-dependent behavior must be taken into account, therefore the component models become more complex. How do we manage this?

Model for a Resistor

The model for a resistor is complicated as shown in Figure 2. Its behavior is frequency-dependent; at DC or low frequency, the inductive and capacitive effects can be ignored. However that is not the case for higher frequencies. The higher the frequency, the greater the likelihood of parasitic effects impacting the signal. When designing circuits for the 2.4 GHz and 5 GHz frequency bands used for 802.11 Wi-Fi, these effects must be considered.

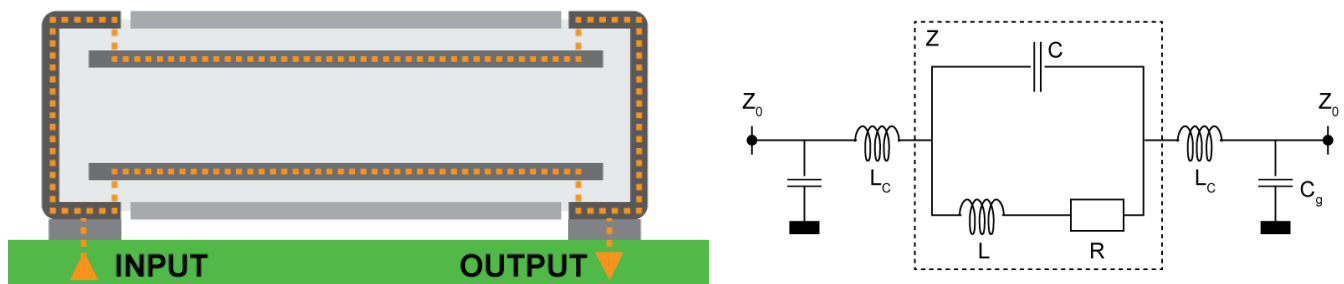
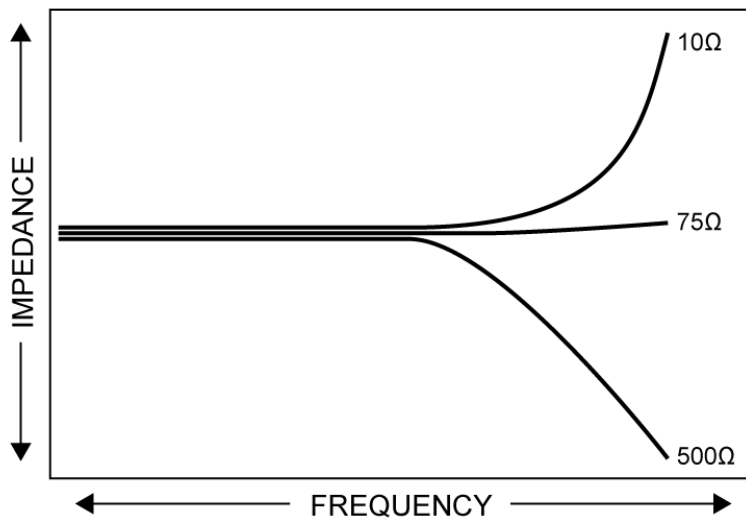


Figure 2: Typical high-frequency performance electrical model for resistor

Different Package

When designing RF circuits, we must consider that a resistor's high frequency parasitic effects can be dependent on package type and size. For example, larger packages such as the 0805 are not suitable when used at higher frequencies due to these effects being particularly pronounced (Figure 3).



As frequency increases, impedance is introduced to varying degrees, based on the resistance of the resistor.

Figure 3: High frequency electrical characteristic

Model for an Inductor

Like resistors, the behavior of inductors is frequency dependent. Again, at DC or low frequencies, parasitic effects are minimal but at high frequencies, they must be considered. As shown in Figure 4, parasitic effects include series resistance across the inductor, series capacitance between the turns of the inductor coil, and even parallel capacitance with the adjacent circuit. Even the inductance value itself can change rapidly with frequency. Thus, high frequency effects on inductor performance must be considered when designing RF circuits.

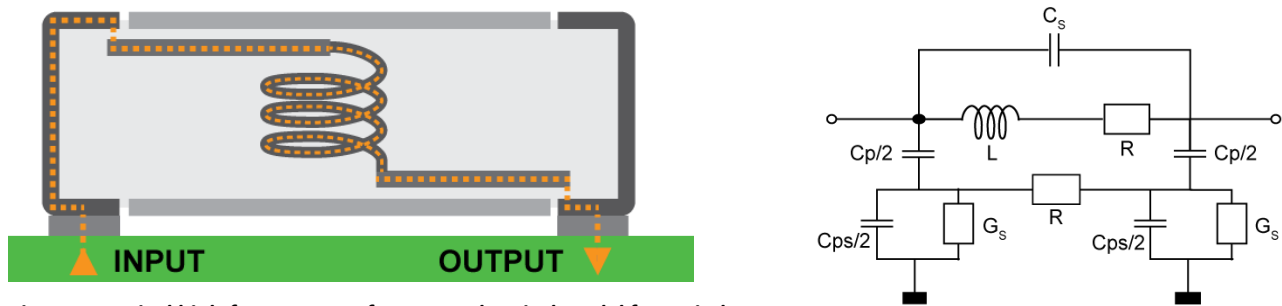


Figure 4: Typical high-frequency performance electrical model for an inductor

Model for a Capacitor

As with the resistor and inductor, the high-frequency model for a capacitor is more complex than a simple capacitance and the performance is dependent on frequency. Inductive effects must be considered as well as series and parallel resistances. Understanding this behavior is critical when designing a high-frequency circuit with bypass capacitors or DC blocking capacitors in an RF path. For RF grounding, designers must consider what kind of RF signal will be used in the design.

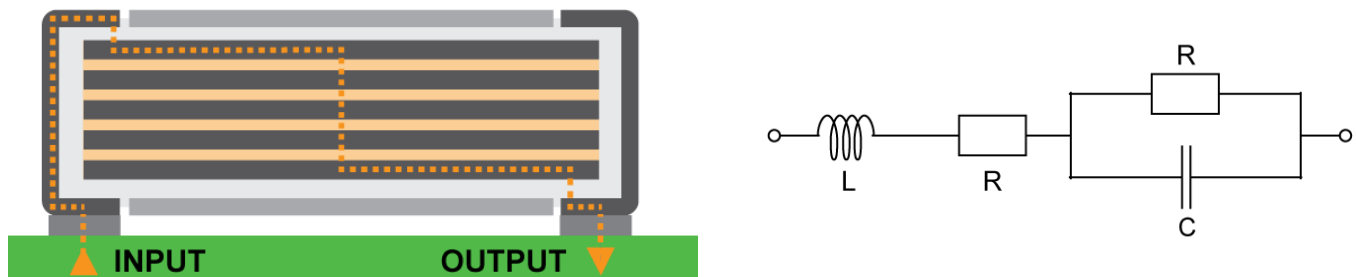


Figure 5: Typical high-frequency performance electrical model for a capacitor

In short, designing standard digital circuits tends to be simpler as parasitic effects of passive components can be neglected. In RF designs however, designers must consider the frequencies of the signals that will be used in their circuits and the effects those signals will have on their components.

PCB DESIGN CONSIDERATIONS

Characteristic Impedance Matching

Characteristic impedance matching is critical when designing a PCB layout. As shown in Figure 6, when a wave encounters an interface between two materials of different densities, some of the wave is transmitted through the interface and some travels back in the form of a reflection. Material density in this example is analogous to

characteristic impedance. For an RF circuit, an impedance mismatch results in reflections and thus a loss in power delivered to/from the antenna. This loss in power translates directly to a reduced EIRP and sensitivity. In high power devices, reflected signals can sometimes cause oscillation or more importantly, can damage your device.

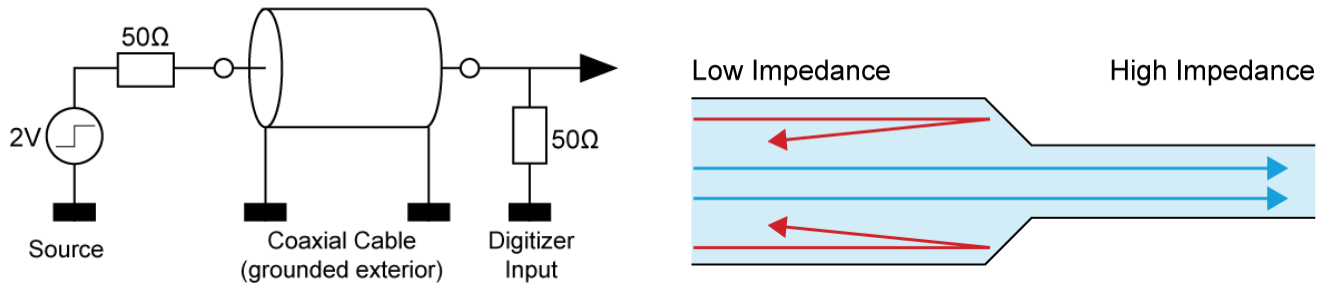


Figure 6: Impedance for a transmission line

PCB layout is the key to RF performance. When designing a PCB layout, designers must consider trace impedance, PCB material, and stack-up thickness. For example, consider a design that has a 50 ohm RF trace, a 100 ohm differential pair, and a 90 ohm USB impedance. You must trade off your impedance width and gap with your board stack because impedance is not only defined by the trace width but also by the substrate thickness, or the thickness of your RF trace related to your reference layer. The trace width and the board stack thickness define your impedance trace.

Transmission Line

Another factor to consider is what type of transmission line you will use in your design. For instance, several types of transmission lines that can be used include:

- Microstrip Line
- Coplanar Waveguide (CPW)
- Coplanar Waveguide with Ground (CPWG)
- Strip Line

Certain transmission lines are better choices for Design for Manufacturing (DFM). The transmission line type can also be chosen to reduce the junction effect, which is caused by abrupt changes in RF trace width. For example, if you run an impedance simulation on your board stack for a microstrip, you'll likely see a very narrow microstrip trace width. In this case you'll need to change to another transmission line type such as CPW or CPWG. CPW doesn't rely on a second ground layer, instead it uses a ground plane on the same layer as the signal trace. With the microstrip line, a second copper ground layer is embedded into the PCB. So trade-offs can be made with different transmission lines and material thicknesses that allow you to attain an adequate RF 50 ohm trace, reducing junction effects with components installed on that trace.

Antenna Placement

Types of Antennas

Antennas play an important role on RF system performance. First you'll want to consider antenna type. What type of antenna is suitable for your design? Antenna types include:

- Monopole Antenna

- Inverted-F
- Dipole Antenna/Sleeve Dipole

Sensitivity to board size is an important factor when choosing an antenna. For example, a monopole antenna is an antenna working with an ideal, relatively large ground which is considered part of the antenna. This makes the monopole antenna sensitive to ground size and shape, meaning that the ground can greatly affect the antenna's performance.

Inverted-F antennas, also called PIFA antennas, have an arrangement that makes them less sensitive to ground.

Lastly there are dipole or sleeve antennas. These antennas have a positive current on one side and a negative current on the other, thus establishing their own ground reference. Of the three antenna types listed here, the dipole is least sensitive to ground.

All antennas require some amount of space for placement. When deciding on antenna placement, the surrounding materials must be considered, particularly conductive materials, as they affect the performance of the antenna.

Antenna selection also depends on the system in which the antenna will be used. Here is a list of systems and their ideal antenna numbers and placement:

- **Single-input, single-output (SISO)** - This system uses only one antenna. SISO systems are usually quite sensitive to location. Performance is easily affected by the multipath effect. In a SISO system, some locations generate what is called a constructive effect and other locations generate a destructive effect. For example, a car's FM radio is usually a single antenna system. As the car moves along the road, you may receive a clear signal one location and static noise in another. SISO systems with a single antenna are the easiest to design and are inexpensive.
- **SISO with antenna diversity** – In this configuration, the system has two antennas. A SISO system with a single antenna can only receive a signal at one point in space with no redundancy. However, a SISO system with antenna diversity support has two antennas, either one of which can be used at any point in time. This allows the system to switch antennas if the performance of one antenna is lacking. The system always switches to the best antenna to overcome the multipath problem. If your system supports antenna diversity, it is better to use two antennas. The rule of thumb is to place the antennas at least a quarter of a wavelength apart. As a rough estimate, a quarter of a wavelength is three centimeters in the 2.4 GHz band and 1.5 centimeters in the 5 GHz band.
- **Multiple-input, multiple-output (MIMO)** – These systems use multiple antennas to receive and transmit concurrently. For example, if you are using a 2X2 MIMO system, you need two antennas; this configuration is called a two data stream system. MIMO systems must have adequate isolation between each antenna. Typically, approximately 25 dBm isolation gives you better signal quality and thus better throughput.

How can you achieve higher isolation? The first and easiest method is to increase the distance between the antennas. Move the antennas as far away from each other as possible. Longer antenna distance provides better antenna isolation. The second method is to adjust the antenna polarization. For example, if you have two dipole antennas, you can adjust them so that they form a 90-degree angle, one in the horizontal polarization and the other in the vertical polarization. This way, even at very short ranges, you can still achieve 25 dBm isolation.

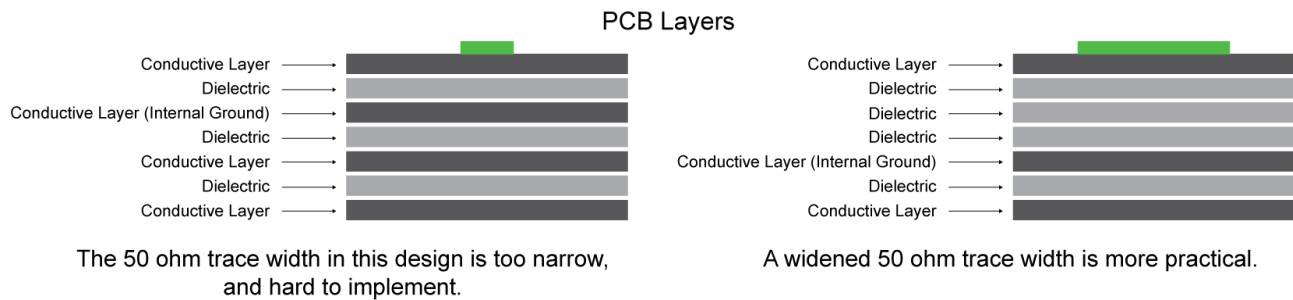
- **Multi-com with multiple antennas**- Multi-com systems use two different standards, such as Wi-Fi and Bluetooth, in one product. Multi-com systems require multiple antennas. Wi-Fi and Bluetooth operate at the same frequencies so adequate isolation between the antennas helps avoid interference and makes for better multi-com coexistence. For example, Wi-Fi products can handle a maximum of -20 dBm input signal

and a maximum transmit power (TX) of approximately +20 dBm. Normally, Bluetooth can receive a maximum input signal of -10 dBm and TX power is typically limited to approximately 4 dBm. Therefore, you need approximately 25~30 dBm isolation between Wi-Fi antennas and Bluetooth antennas. This provides increased performance when Wi-Fi and Bluetooth operate concurrently.

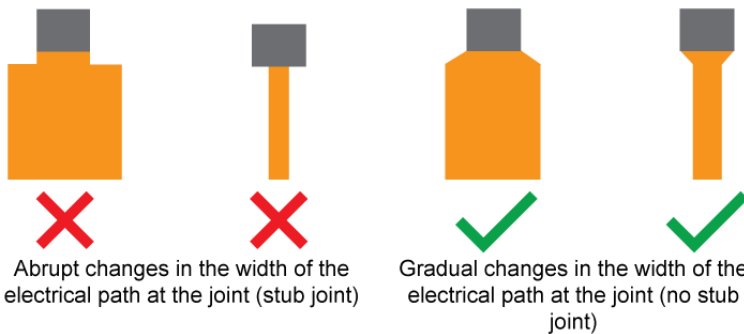
Parasitic Effects during Design

When designing an RF system that operates at a high frequency such as the 5 GHz band, it is necessary to take parasitic effects into consideration. It is important to understand from where the parasitic effects are coming. When designing the PCB layout, you must consider landing pad size in order to reduce junction effects, as well as parasitic capacitances which could exist at your operating frequency.

There are a few things you can do with a PCB design to work around parasitic capacitances. First you can remove the inner ground referenced copper layer. Removing this copper increases the distance to the reference ground which reduces the capacitance value.



The second option is to reduce stub junctions as much as possible. Stubs can cause problems with signal integrity when operating signal frequencies rise above approximately 100 MHz.



Third, reduce any abrupt changes in your trace width. A 90 degree bend has an adverse effect on an RF signal. Smooth transitions and curves are best. Think of the trace as a water pipe. If the pipe bends at a 90 degree angle, then the water does not flow smoothly.

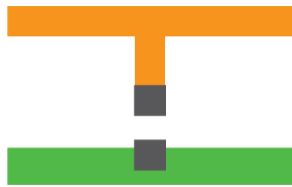


Abrupt changes in trace width or trace angle, not optimal

More gradual change is better, but not an ideal design

Most gradual change is the best design

Fourth, reduce the junction on the matching circuit.

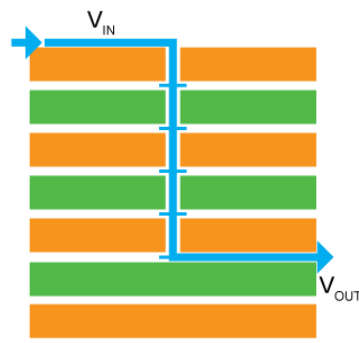
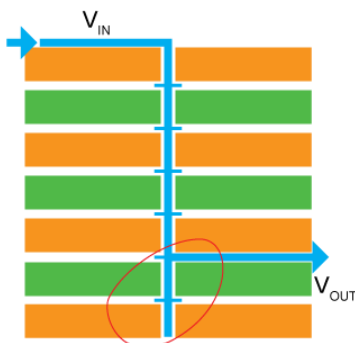


X Extended junction introduces possibility of parasitic effects

✓ Reduced junction is a better design

Lastly, make sure the return ground path is clean. For RF signals, the ground return path is under/follows the RF signal trace. Sometimes when designing layouts, engineers do not consider signal path and forget that the signal returns through the ground. Ground continuity and ensuring the ground path is clean is critical for the design.

The ground plane must not be broken or interrupted during transmission line routing. Ground vias are necessary to ensure that the RF trace has proper ground reference. Ground vias prevent accrual of parasitic ground inductance caused by ground-current return paths. They also help to prevent cross-coupling between RF and other signal lines across the PCB. It is also important to consider bias and ground layers. The layers assigned to the system bias (DC supply) and ground must be considered in terms of the return current for the components. The general rule of thumb is to not have signal routed on layers between the bias layer and the ground layer. Ground is the key to signal integrity.



X Additional cross-layer via causes signal disturbance

✓ Properly terminated cross-layer via reduces parasitic effects

There are three S's that are key to PCB designs with regard reducing parasitic effects: short, straight, and smooth as possible.

DFX CONSIDERATIONS

Design for Passing Certification (FCC/CE/IC)

When designing, you must think about what certifications the product must pass. It is important to always consider impedance matching and how to reduce the unwanted signal; if it is not managed properly, you will likely fail certification tests for FCC, CE, IC, etc. Keep the three S's in mind. Keep the design trace short, straight, and smooth.

Design for Manufacturing and Test

The product must be tested during manufacturing. High frequency pogo pins are quite expensive and it is difficult to maintain impedance matching across a wide frequency range. Use probe connectors as much as possible to get a stable test result. To measure 2.4 GHz signals, a 5 mm length pogo pin using a G-S-G probe structure may still work. For 5 GHz signals, a test connector is required for improved impedance matching.

CONCLUSION

In RF designs, there are many factors that must be taken into consideration in order for the design to be successful. While it can be challenging, following best practices and understanding key components and their implications on system performance can help you create a successful design. Always consider impedance matching. Dielectric materials and board thickness can be used to optimize the RF trace width, reducing junction effects while maintaining the characteristic impedance requirement. Strong impedance matching prevents reflections that cause issues with emissions and test accuracy.