

Effectively Overcoming RF PCB Antenna Design Issues

The article shares some practical tips for dealing with issues like noise sensitivity and tight impedance margins when designing antennas for an RF PCB.

Demand is on the rise for RF PCBs due to the current trends of wireless electronics, IoT sensors, smartphones, robotics, and more. An RF PCB operates at a very high frequency of 100 MHz and above.

Designing an RF circuit involves issues like noise sensitivity and tight impedance margins. When you add an RF antenna to the circuit, it brings additional complexities to the board's signal integrity. Choosing an experienced PCB manufacturer is one of the key factors in building high-quality RF PCBs.

An RF antenna transmits and receives electromagnetic radiation in a wireless circuit. The signal range depends on the antenna design, placement, enclosure, and ground plane size. To avoid signal corruption and interference from nearby digital components, a circuit designer must follow the PCB supplier's recommended [RF PCB design](#) guidelines. Correct antenna design and layout strategies can ensure analog signal integrity in mixed-signal PCBs.

RF Antenna: Basic Elements and Types

An effective antenna design—paired with a good PCB layout—can substantially increase the operating range of an RF PCB. An antenna converts the electrical energy fed from a signal generator into electromagnetic radiation. Primarily, an antenna is a conductor that transmits the signal into the free space. The length of the conductor should be a multiple of the signal wavelength to radiate the signal effectively.

The basic elements of an antenna include a conductive radiator that emits the signal, a reference plane to determine the antenna's orientation, an antenna feedline that provides the input signal to the antenna, and an impedance-matching network to ensure signal integrity and maximum power transfer at the set carrier frequency.

Different types of antennas are available:

- A PCB antenna is a trace etched on the circuit board in any specific shape depending on the design requirement

and space constraints. It's a 2D structure in the same PCB plane and can be easily manufactured.

- A wire antenna is a 3D extension over the PCB and protrudes into the space. It provides the best performance and higher RF range due to its external exposure.

- Chip antennas are suitable for compact PCB products, but they're expensive. These antennas are sensitive to RF GND size and may require an extra matching network for antenna tuning.

It's recommended to select the antenna type based on the RF range to be covered, cost allocated, the board size, and required radiation intensity in a particular direction.

Design, Placement, and Layout issues

After choosing the antenna design type, one must understand the effects of placement, enclosure, and ground plane size on the RF antenna performance.

Antennas are sensitive to the PCB RF ground plane size and the plastic casings. The resonant frequency reduces with an increase in the effective capacitance due to a larger RF ground size or a bigger plastic casing.

To achieve better RF performance, the return path should be uniform. If the ground plane is interrupted, the return signal may switch to the smallest path available and affect the impedance matching between the antenna and the radio.

Antenna placement must be aligned with the RF product's final orientation to reach maximum radiation in the chosen direction. The matching network should be carefully designed as multiple parameters can affect the antenna's impedance. While radiating the signal, strong coupling may occur between feeding lines and radiating elements, which can impact the antenna performance.

In a high-power application like space systems, the feed circuit to the antenna should be immune to the corona discharge effect, which can damage the antenna itself. Prior simulation and testing of the antenna with the designed feed circuit is recommended to avoid such issues.

Guidelines for Designing RF PCB Antennas

The guidelines for RF PCB antenna design assist in ensuring there's efficient radiation from the antenna. They also suggest placement methods for maximum isolation between the analog and digital sections as well as offer electromagnetic-compatibility tips in resisting unnecessary signal reception from neighboring devices. Here are some guidelines for designing better RF PCB antennas:

- The antenna should be placed in one corner of the PCB with enough clearance from the remaining part of the circuitry.
- Maintain strict antenna keep-out areas in every layer of the PCB stackup. No components, planes, or traces should be placed in the RF antenna keep-out area.
- The ground pattern varies based on the type of antenna used in the RF PCB design. It's recommended to follow the corresponding antenna manufacturer's reference design to include appropriate ground patterns.
- Antennas should be placed away from any plastic casing on the PCB to avoid variation in its resonant frequency.
- No metallic enclosures are allowed to cover the RF product completely. The antennas are always kept out of such enclosures.
- The antenna should be placed in line with the overall product orientation to facilitate maximum radiation.
- No cable can cross the antenna trace on the PCB positioned on the same side of the antenna.
- Include ground clearance from the antenna with a minimum width as per the manufacturer's reference document.
- Always follow the manufacturer's datasheet while designing the antenna matching network, considering the right antenna trace length, final plastic enclosure placement, and other recommendations.
- Provisions for a matching network are suggested to be included in the RF PCB design using 0E (or zero-ohm) resistors, in case of unknown antenna impedance (due to substrate differences, ground variations, or proximity of plastic casings).
- To provide isolation to the RF-sensitive elements on the board, it's advised to use shielding structures like ground pours, via fences, waveguide routing techniques, etc. Also, the latest EDA tools support simulations to understand the radiation effects on the PCB, which can be used while designing the RF circuits.

Conclusion

To effectively overcome RF PCB antenna design issues, it's necessary to adhere to the above guidelines and work with an experienced RF PCB manufacturer.

Manufacturers can offer layout suggestions like using inner PCB layers as PWR-GND planes to provide shielding effects, packing critical signals to avoid interference from

noise signals, and many others. If the manufacturer has enough experience in RF PCB manufacturing or [PCB prototyping](#), they can come up with great suggestions that might help you.

Recently, the software tools used for PCB designs also help ensure the isolation and signal integrity of the RF design.

Suresh Patel has worked as a sales engineer and in other management roles at [Mer-Mar](#). He brings 25 years of experience in sales of printed circuit boards, technical services for clients, and business management.