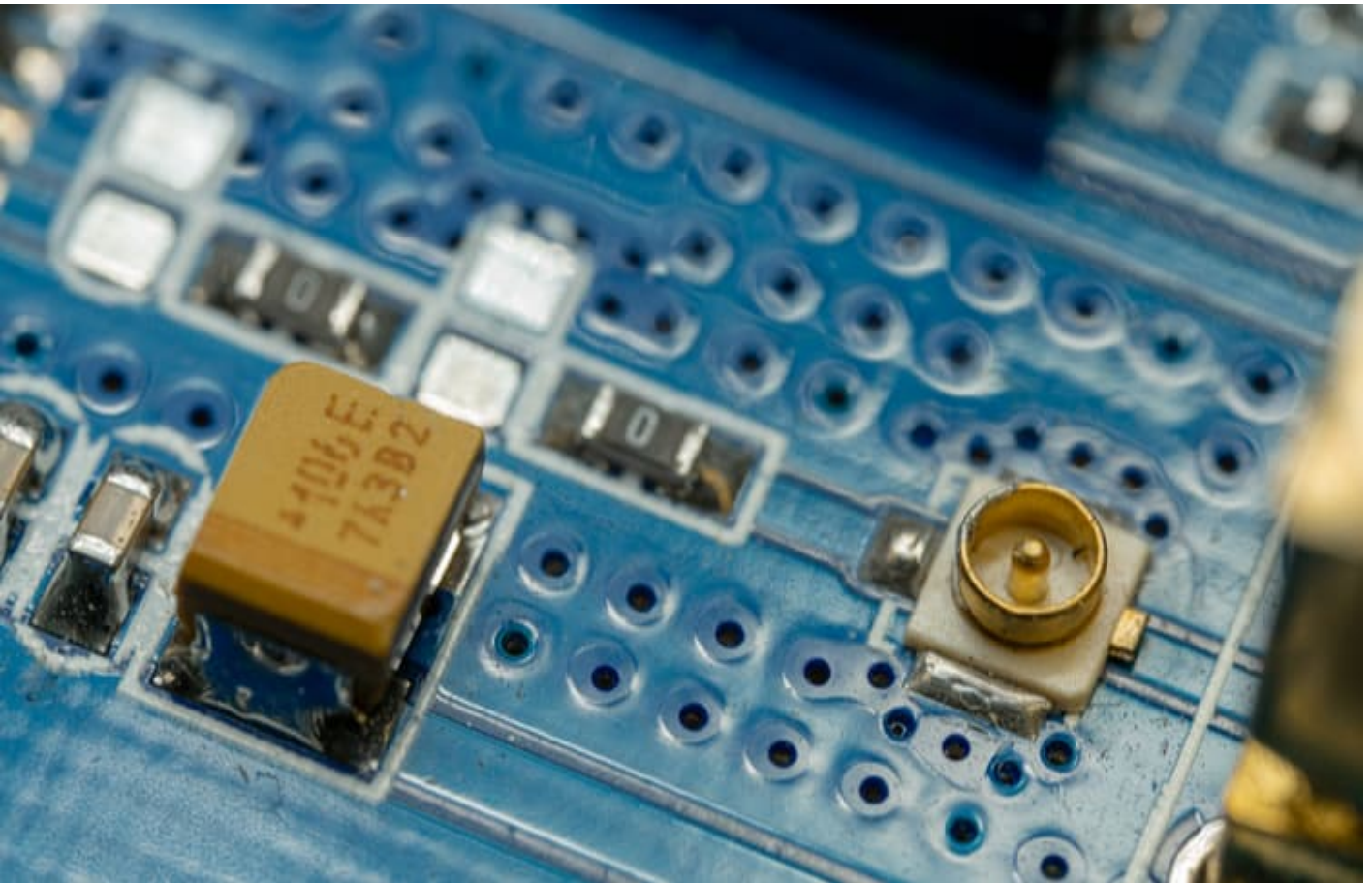


RF Antenna Design and Layout Tips for Your PCB

Cadence System Analysis

Key Takeaways

- RF antennas come in many form factors, ranging from flat chip antennas integrated into ICs to copper antennas printed directly on a PCB.
- Creating a layout with one or more antennas requires ensuring isolation between different circuit blocks in your PCB.
- When you need to design an RF antenna, you should use CAD tools that help you design isolation structures, transition structures, and even printed antennas for your PCB.



This SMA connector makes a coaxial connection with an RF antenna.

These days, it's hard to think of a consumer product that doesn't contain an antenna. Even my garage door opener can connect to my phone via Bluetooth or WiFi. Each time a new RF antenna gets added to a PCB layout, it can create a new headache for RF designers, especially as analog design skills start to become critical again. With so many RF capabilities being added to new PCBs, how can designers ensure the signals in their system are not corrupted and signal integrity is preserved?

Thankfully, there are some simple design choices you can make to help ensure your RF signals are not degraded by nearby digital components. These same design choices will help prevent multiple analog signals from interfering with each other. While there are plenty of topics in RF design to consider when designing mixed signal or all-RF systems, antenna design and layout are probably two of the most important. Here's what you need to know about RF antenna design in your [PCB layout](#) and how to ensure analog signal integrity.

RF Antenna Design Basics

There are a few basic points to follow when designing a custom antenna or choosing a COTS antenna for use in your RF PCB. All RF antennas share a

few particular characteristics that should be considered during the design phase. Every antenna needs the following elements:

- **Floating conductive radiator:** This is the antenna element from which radiation will be emitted.
- **Reference:** The reference plane or element for an antenna helps determine the structure's directionality in each antenna mode.
- **Feedline:** The feedline routes the input signal from an RF component into the radiating antenna element.
- **Impedance matching network:** An antenna normally has ~10 Ohm impedance, so it needs to be matched to the feedline impedance to prevent reflection and ensure [maximum power transfer](#) at the desired carrier frequency and bandwidth.

There are many standard antenna designs that have been thoroughly studied. You can find many reference designs online, which can then be copied into your PCB layout. You can also find many design formulas for standard antenna structures in microwave engineering textbooks. Finally, if you want to use a COTS RF antenna, there are many inexpensive designs you can find on the market for low cost. No matter which RF antenna you choose to use, you'll need to carefully place it in your layout to prevent interference between board sections.

RF Antenna Layout Tips

Once you've designed your antenna, it's time to figure out where it should be placed on the PCB. RF designers should take some tips from mixed-signal designers (most RF boards are really mixed-signal boards) in order to prevent interference between multiple sections in the RF front end, back end, and digital sections.

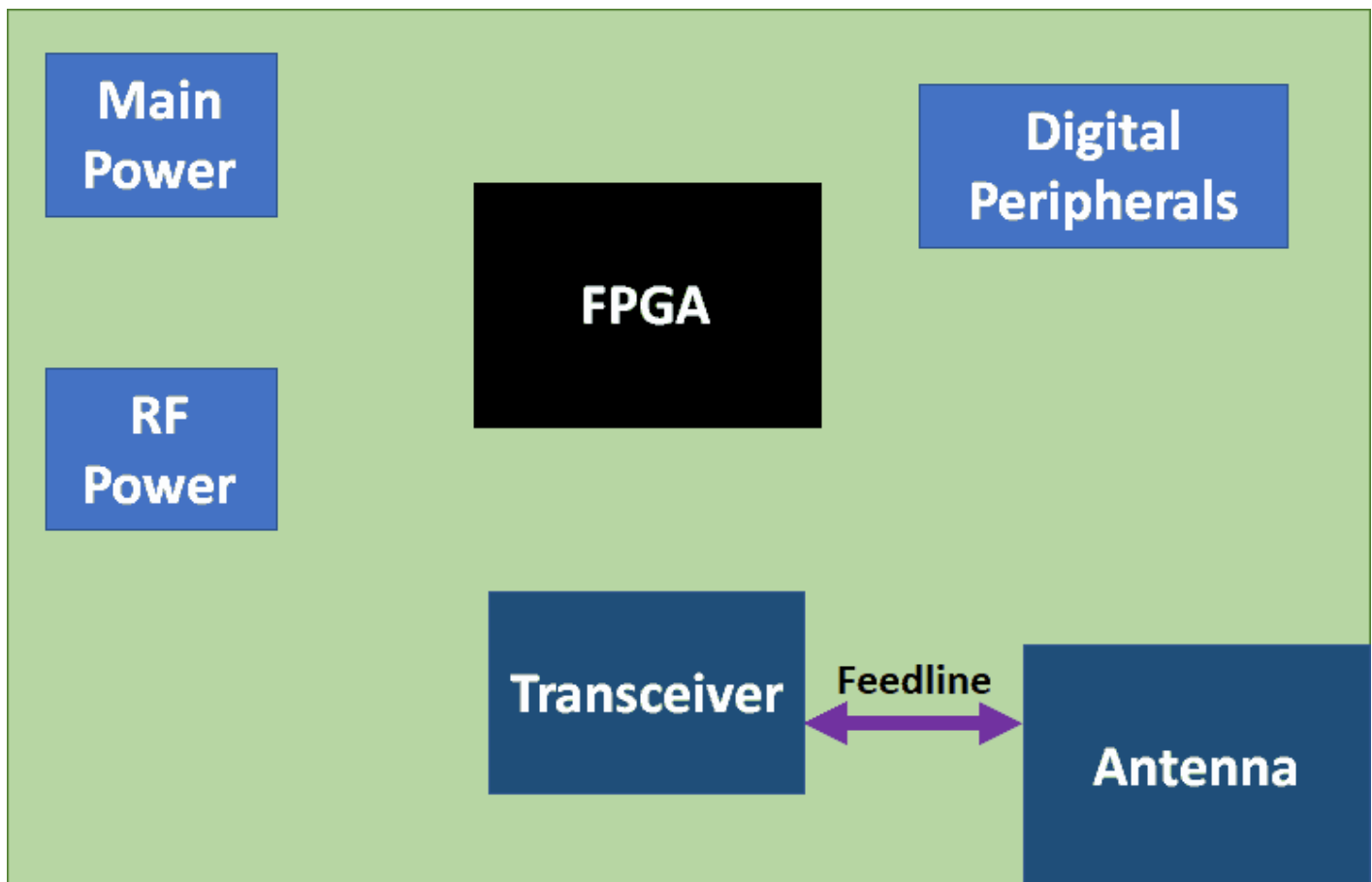
- **Efficient radiation:** The goal here is ensure radiation from antenna elements travels away from the board without being picked up by other structures in the PCB layout.

- **Isolation:** Similarly, we don't want multiple sections in the [PCB layout](#) interfering with each other.
- **Electromagnetic compatibility (EMC):** Finally, we need to ensure that the layout is resistant to reception of signals from other devices that may emit over a broad range of frequencies.

In a real PCB, most design goals are in competition, but there are two important points to follow that will help you balance these design goals.

Separate Circuit Blocks in Your PCB Layout

This is a fundamental mixed-signal PCB design topic, and it applies just as much to RF antenna layouts. You'll need to place the antenna section in a location on the board that is separate from other circuit blocks. Generally, it is best to place the antenna section near the edge of the board and away from other analog components. This confines strong emission to one location on the board and ensures interference between board sections is minimal.



Gridded system layout on a PCB

The challenge in gridding is to ensure your return paths in different sections do not interfere with each other, as this leads to noise coupling and crosstalk. Field solvers integrated into advanced PCB design tools can help you spot [deviations from return paths](#) as you create your layout. For high frequency designs, use a continuous ground plane structure to ensure consistent return paths.

Isolate Antenna Sections

Modern cell phones and cellular equipment have become the gold standard for RF isolation techniques through the use of creative isolation structures. Very simply, isolation involves placing some shielding around an RF-sensitive element in the board to block propagation of waves between an emitter and receiver. Here are some options you can use in your RF antenna section to isolate components, feedlines, and the antenna from each other or external noise sources.

Isolation structure	Advantages	Disadvantages
Shielding can	High isolation value as long as gaps in the structure are small.	Can be bulky components, or they need to be custom-built.
Via fences	Similar effect as ground pour, but with less board space.	Lower isolation, narrowband only, low cutoff frequency.
Ground pour	For RF antenna feedlines, creates a coplanar waveguide with high isolation.	Takes up board space, not ideal in smaller boards with dense component arrangement.
Waveguide routing	Very high isolation, mode selection can be performed to enable bandwidth-specific routing.	Takes up board space, only appropriate for most critical lines.
Bandgap structures	Can be engineered to provide moderate to high isolation for particular bandwidths.	Ideal for high frequencies, which will take up less board space.

Isolation structures are generally placed between RF elements to block noise coupling and power exchange between them. Determining which isolation structure you should use to ensure RF antenna signal integrity is a complex design problem that has been thoroughly researched. If you’re not an expert at elliptical integrals, you’ll need to rely on an [electromagnetic \(EM\) field solver](#) to determine how these structures affect feedline/RF antenna impedance, as well as the level of isolation these structures provide.

If you have access to an EM field solver, you can use near-field and far-field simulations to identify areas where strong radiation occurs in your PCB layout. Once you identify these areas and which frequencies are being emitted, it’s easier to see which type of isolation strategy you should use. It’s best to work

in the frequency domain directly (FDFD method) rather than using Fourier transforms to convert from FDTD results.

While RF antenna design and layout requires careful attention to detail, this extra caution pays off, as you ensure isolation and signal integrity for your RF design.

If you'd like to keep up-to-date with our [System Analysis content](#), [sign-up for our newsletter](#) curating resources on current trends and innovations. If you're looking to learn more about how Cadence has the solution for you, [talk to us and our team of experts](#).