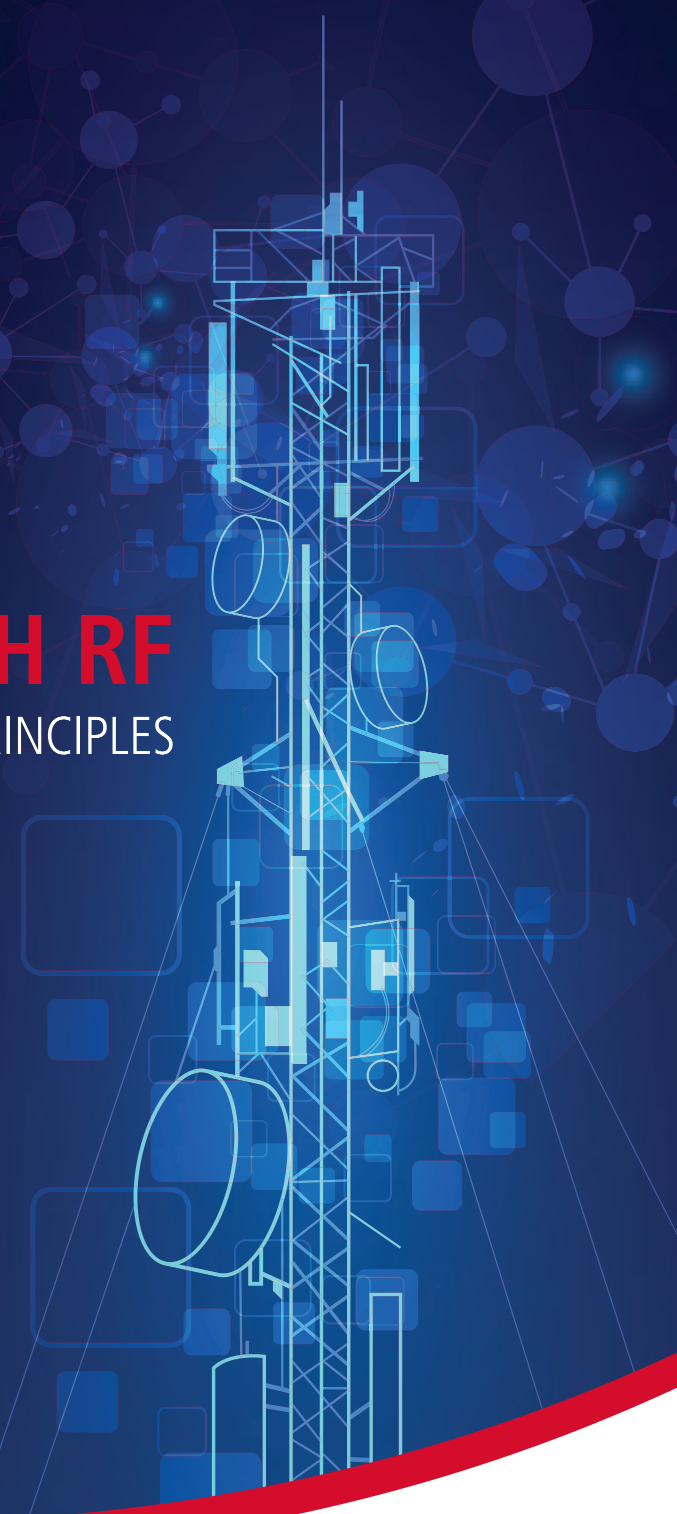
MAKING WAVES WITH RF

A PCB ENGINEER'S GUIDE TO RF DESIGN PRINCIPLES



EMA Design Automation®

THE WAVE OF THE FUTURE

MAKING WAVES WITH THE ELECTROMAGNETIC SPECTRUM

Radio Frequencies (RF) are everywhere. We can remotely start our cars from the comfort of our living room. We can answer phone calls from smart watches. We can video chat with loved ones on opposite ends of the world. We can transmit commands to a robot that's on a different planet altogether. All of these things (and much, much more) are made possible through the manipulation of radio signals.

While RF seems to be "everywhere" (it literally is). It doesn't mean getting it right is easy. This guide will help you get acquainted with RF, what it is, how it works, and what designers need to consider when including RF functionality into their projects.

Table of Contents

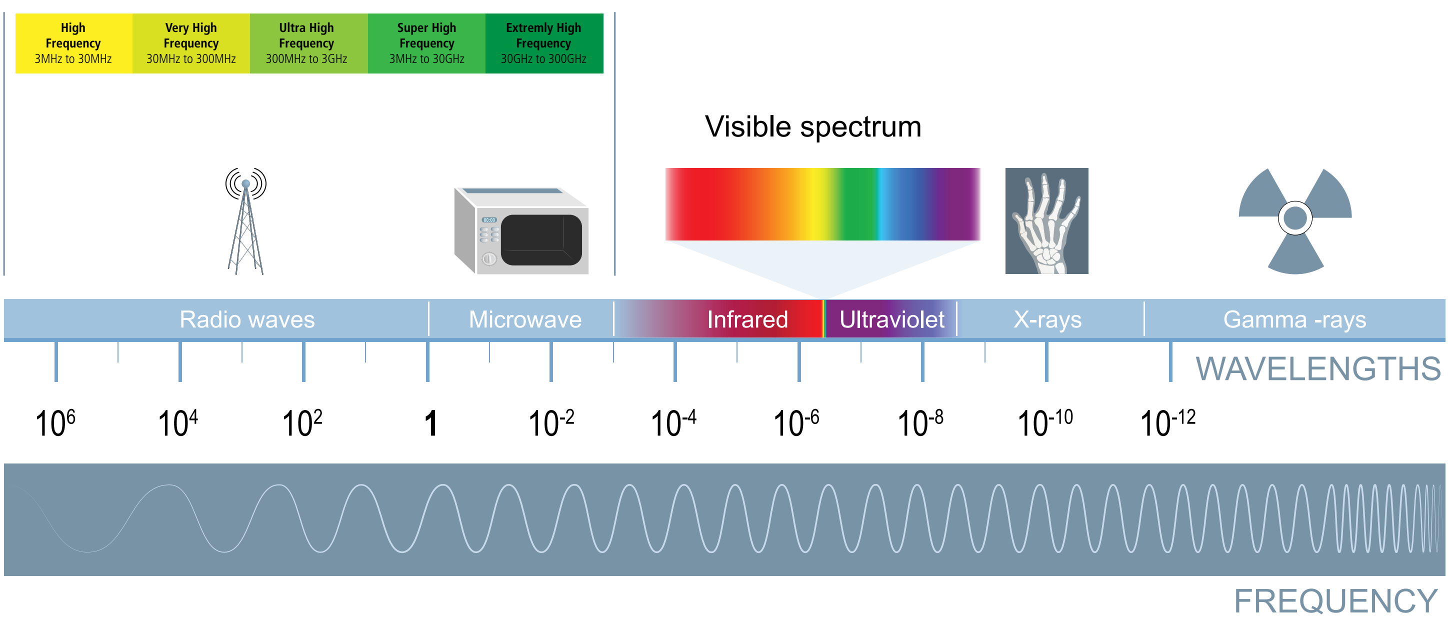
THE "WHAT" AND "WHERE" OF RF

BREAKING DOWN RADIO FREQUENCY

So, what exactly is a radio frequency? The answer to this question can get quite complex, but in basic terms it's electromagnetic radiation with frequencies between 3,000 Hz (3 KHz) up to 300,000,000,000 Hz (300 GHz). The "radio" part simply implies these frequencies are used for communication.

For those of you playing the Wavelength Game™ at home, that's ~100,000 meters (3 KHz) to ~1 millimeter (300 GHz), so roughly 2.5 times around the Earth's equator down to the width of a dime.

If we consider the entire Electromagnetic Spectrum consisting of seven classes, the aforementioned range of frequencies fall within just two of those classes: *Radio Waves and Microwaves*. These two classes comprise the lowest energy per photon (E) values and is where "microwave" derives its name.



The microwave bands are Ultra High Frequency (UHF), Super High Frequency (SHF), and Extremely High Frequency (EHF), which have E values from 1.24 µeV up to 1.24 meV, respectively.

THE “WHAT” AND “WHERE” OF RF

BREAKING DOWN RADIO FREQUENCY

It’s pretty easy to get into the weeds with RF, so let’s instead look at where RF technology exists today.

Walkie-Talkies

Those toy walkie-talkies you got for your kids likely operate between 462 and 467 MHz.

Wifi

Your WiFi operates at 2.4 GHz and/or 5 GHz.

Key fob

In the US, clicking the buttons on your car’s key fob transmits a signal at 315 MHz.

Cellphone

The phone in your pocket you use to look up cat videos communicates wirelessly with giant towers, anywhere from 700 MHz up to 2500 MHz.

Wireless Pay

Tapping your phone on a POS terminal transmits a signal from your phone to the terminal at 13.56 MHz.

Space Exploration

NASA communicates with the Curiosity Rover using either ~400 MHz or 8-9 GHz frequencies.

Medical Devices

MRI (1-300 MHz) and X-Ray Machines (10 PHz up to 100 EHz) are used to look inside your body, while pacemakers (402 - 405 MHz) are used to keep your heart ticking at a normal pace.

RFID

From 120 KHz up to 24.125 GHz, RFID tags can be found in all sorts of things, such as key cards to unlock doors to asset tracking in large warehouses.

Over 70% of vehicles today are made with Remote Keyless Entry. For Key fob frequencies, Japan also utilizes 315 MHz while Europe uses 433.92 MHz or 868 MHz.

RF technology is becoming increasingly common but communicating with robots millions of kilometers away requires planning. Then there are medical-grade devices that must work, which requires planning on top of planning. Devices such as Insulin pumps, pacemakers, and defibrillators can make for a bad day should they fail. Children’s toys, on the other hand, don’t require as much scrutiny, since a person’s life does not require them to function (though parents may disagree). While these products have varying performance specifications, all have certain facets about their RF design that are similar.

DESIGN CONSIDERATIONS
MAKE THE WAVES WORK FOR YOU

There are various design considerations one must address when designing RF applications regardless of the purpose or performance specifications. The degree to which these considerations must be adhered to depends on the class of the product being designed (how severe an impact potential communication failure would cause). Each of these facets of RF design will play a major role in the end product success.

User Experience

While it's impossible to accommodate all use cases, it's important to try to make your product as intuitive as possible.

Implementation

Every project is different and requires a little planning beforehand to ensure the right decisions are being made.

Grounding

Grounding is one of, if not THE, most important aspect for any RF design, and is the foundation for PCB design.

Thermal Profile

Transmitting RF tends to heat things up. This can have undesirable effects if not properly mitigated.

Interference

Almost everything in the environment can and will disrupt RF signals; it's important to protect against such interference.

Keeping this in mind, let's expand upon these topics to help understand their importance.

USER EXPERIENCE

THINKING ABOUT WHO WILL USE YOUR WIDGET

Your RF design means nothing if a user can't easily interface with the end product; however, providing configurable settings can complicate the design. When determining the design specifications and requirements, consider the following:

1. Will the end user be able to change the transmit and receive frequencies?
2. Will the end user be able to modify the transmit power?
3. Will the end user be able to change squelch types?
4. Does your circuit allow for different modulation types?

If your answer is "yes" to any of these questions, then you need to design around a user being able to change these parameters.

If your design has a wide array of configurable options, then you'll likely want to accommodate for the inclusion of a Field Programmable Gate Array (FPGA). Incorporating an FPGA into your circuit will allow a user to modify the RF operating parameters through a keypad on the product, improving the user experience. This also provides the ability to upgrade your product via software updates, should that be a feature you wish to implement.

At the end of the day you have to put yourself in a user's shoes. "If I needed to use this product, would I enjoy the experience or get frustrated?". If you think you would be frustrated using a product you designed, then a user will absolutely get frustrated. While it's impossible to accommodate all use cases, it's important to try to make your product as intuitive as possible. For RF designs, one of many ways you turn first time users into next-time users is by providing easy and intuitive ways to change frequencies, squelch types, power levels, etc. But in order to get that point, it's important to create a high-functioning PCB layout.

What is a squelch?

A specialized circuit function that is used to suppress or mute weaker signals. The circuit will "unsquelch" when a desired signal is received. There are different ways to implement squelch in your design, including:

- Noise Squelch
- Tone Squelch
- Continuous Tone-Coded Squelch System (CTCSS)
- Continuous Digital-Coded Squelch System (CDCSS)

IMPLEMENTATION

THE LOGISTICS OF WIDGET MAKING

Once you have an understanding of the “what” and “why” of your project, you have to start thinking about the “how”. Will you utilize Commercial-Off-The-Shelf (COTS) products for things like antennas or filters, or will you design a trace antenna or filtering specific to your needs? Like with all projects, what you’re able to accomplish comes down to two main things: **time and money.**

Design and Prototype

If you happen to have an abundance of both time and money, then taking the time to design and prototype your own modules comes with the added benefit of knowing exactly what’s “under the hood” and how everything should interact with the various parts of your design.

Pros

- ✔ You know everything there is to know about the circuitry
- ✔ You fully own the design for licensing purposes
- ✔ You can fully optimize the implementation for your specific design needs (cost, space, performance, etc)

Cons

- ✖ Designing every aspect takes time

Commercial-Off-The-Shelf

On the other hand, there may be no reason to reinvent the wheel if there’s an aspect of the design that may have a COTS variant which could be easily integrate. Several COTS options are available for antennas, filtering, power supplies, SOIC, and more.

Pros

- ✔ Integrating readily available components/ modules saves time
- ✔ There may be several options available (depending on the part)

Cons

- ✖ The component/ module may come with licensing fees, which is an added cost
- ✖ There may be several options available, increasing time to research for compatible parts
- ✖ May have to sacrifice performance or size to use existing parts

No choice is the wrong choice, whether you choose to design from the ground up or integrate readily available modules. Every project is different and requires a little planning beforehand to ensure the right decisions are being made.

GROUNDING
YOUR ELECTRICAL FOUNDATION

Grounding is one of, if not *THE*, most important aspect for any RF design. A poorly grounded PCB will not transmit or receive RF signals well, if at all, so it should be one of the first pieces of the design addressed. Fortunately, grounding a PCB is fairly simple and can be completed with the following steps:

STEP 1
Make one layer of your PCB a sheet of copper

STEP 2
Utilize vias to connect the various parts of your design to that copper sheet

STEP 3
Connect that copper sheet to the negative terminal on your power supply/battery

Following these steps will create a basic ground plane for the design; however, the ground plane can be made more efficient depending on certain design requirements:

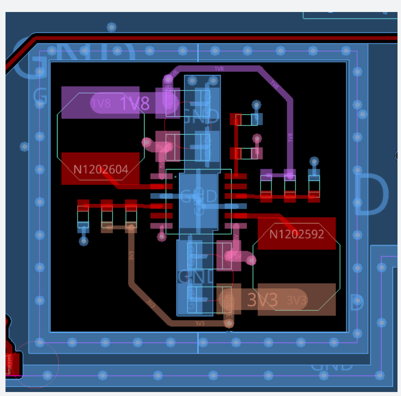
Analog and Digital Circuitry

If the PCB contains both Analog and Digital circuitry, consider creating two separate ground planes for each, connected with a thin “bridge” of copper. This helps prevent crosstalk between the two sections.

Note: Adding multiple bridges to your design may actually cause more harm than good due to increased inductance. In such cases, you may be better off keeping your ground plane solid or creating separate ground plane layers for each type of circuit. Simulation software can help determine which technique provides the most benefits for your design.

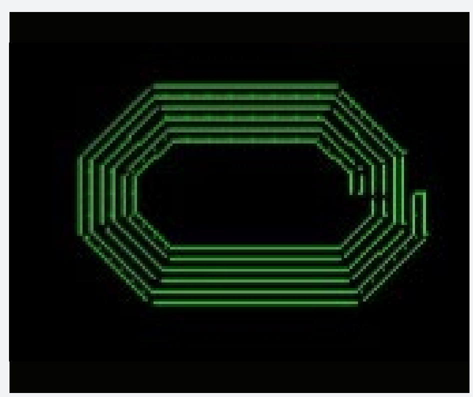
RF Circuitry

Consider surrounding the RF circuitry with vias connected to the ground plane. This is referred to as via fencing and is one of many ways to “shield” this sensitive circuitry from EMI and/or crosstalk from other sections of your PCB.



Trace Antennas

If you plan on incorporating a trace antenna into your design, do NOT have a ground plane under this trace. This will prevent the incoming or outgoing signal from being absorbed by the ground plane.



RF circuitry is greatly affected by noise and outside influences. Establishing proper grounding in the beginning of the design process provides the foundation for successful RF design and will help to protect this sensitive circuitry. Now, that we have defined our user requirements, implementation plan, and grounding is in place, next we will look at how to manage all the energy your RF circuit will start generating when communicating.

THERMAL PROFILE

EXPLORING CIRCUIT PERFORMANCE UNDER TEMPERATURE

Electricity tends to warm things up (excitable electrons will do that, I suppose). The amount of heat produced is dependent on what that electricity is doing in your circuit. When RF is incorporated into the design, it's more likely your circuit will produce quite a bit of heat. This will have undesirable effects on other parts of your circuit if left unmitigated. To determine if heat mitigation is required in your design, ask yourself some questions:

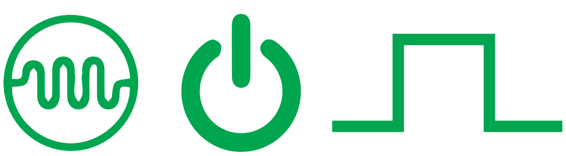
- **How often will your circuit transmit a signal (duty cycle)?**
  While the circuit is "on", it will be generating heat. If the circuit is "on" more often than "off", you may need to consider heat mitigation. Heat Mitigation could be in the form of heat sinks or active cooling depending on how much heat the circuit generates.

- **What frequencies are you transmitting?**
  HF frequencies are typically used to communicate thousands of kilometers away and tend to have beefy transmit power requirements to be successfully received at such distances. On the other hand, UHF and L-band frequencies tend to be used for Line Of Sight (LOS) applications which don't require as much power to transmit.

- **How much power (in watts) will your circuit transmit this signal?**
  Tied to the point above, if the intent is to communicate thousands of kilometers away, you'll likely have to transmit these signals with a significant amount of power (watts). More power means more heat to dissipate.

Let's utilize these questions to analyze design scenarios and determine if heat mitigation should be implemented.

Scenario 1: Cell Phone



Design Specifications: Your cell phone operates between 700 MHz up and 2500 MHz and will typically transmit from 0.1 to 2 watts (depending on distance from a cell tower). The only time your cell phone actively transmits signals is during phone calls or continuous data streaming (video chats or VoIP). A cell phone otherwise briefly transmits when fetching data from the internet or "pinging" cell towers to maintain connectivity to the most ideal cell tower.

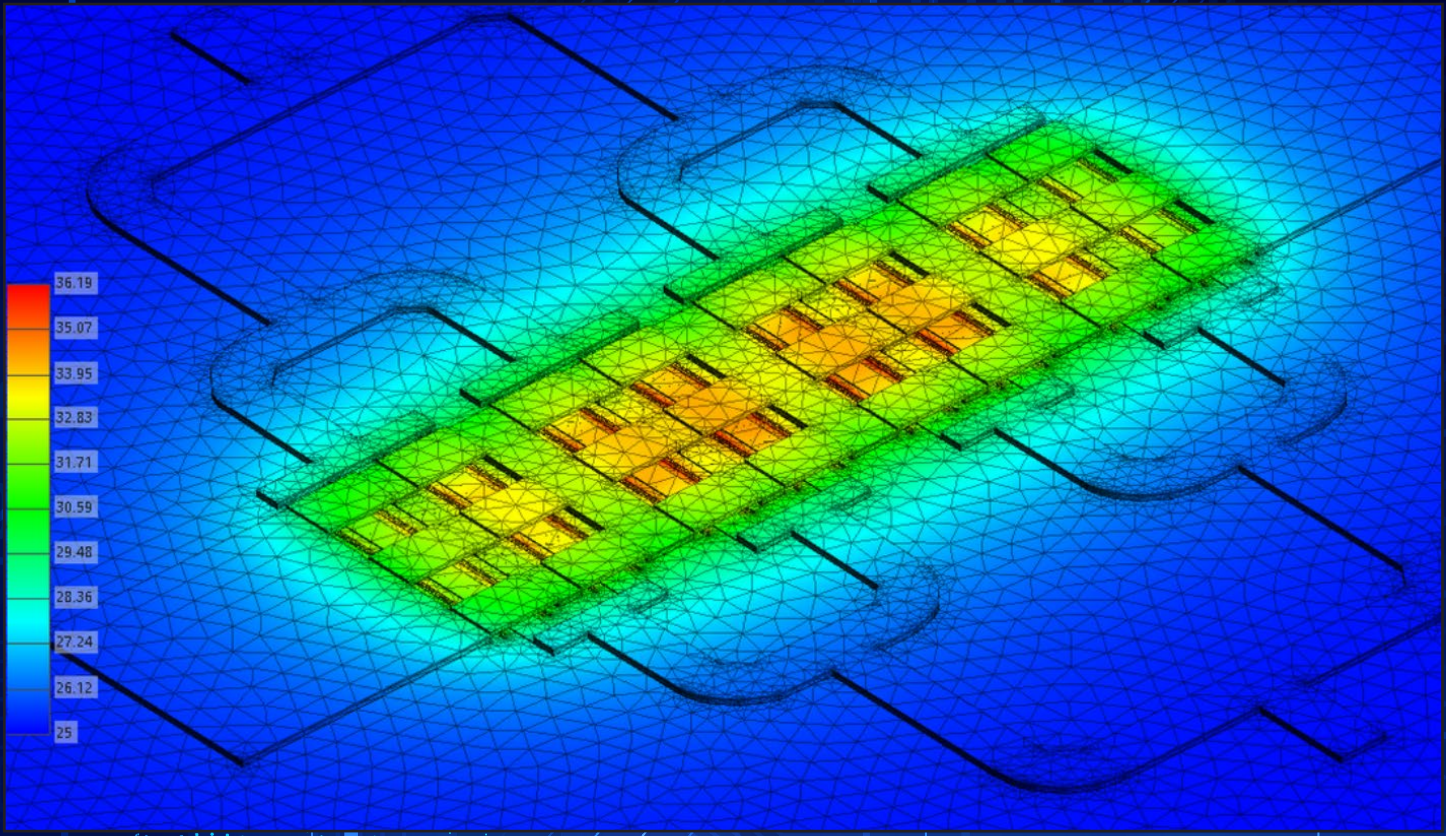
Answer: Typically, cell phones require little, if any, means of heat dissipation. Your cell phone may get warm while actively calling someone, but otherwise doesn't heat up to the point where you can't touch it with your bare hands.

Scenario 2: Military Grade Radios that Utilize MANET Waveforms

Design Specifications: Radios that utilize Mobile Ad-Hoc Networking (MANET) networks "know" about all the other radios in that network. This is accomplished by constantly transmitting beacons or short bursts of metadata to all other radios in the network. Depending on how far away these radios are from each other, they could be transmitting this data at 10+ watts. All radios in this network are constantly transmitting (high duty cycle). Military applications utilize VHF through Ku band frequencies.

Answer: These radios will get quite warm and require some form of heat dissipation. This is typically accomplished with a heat sink chassis.

THERMAL PROFILE
DETERMINING YOUR HEAT PROFILE



Asking yourself these questions won't provide a definitive answer if heat mitigation is required but they will point you in the right direction. If heat mitigation is a possible requirement, the thermal profile should be examined in greater detail. This can be achieved by calculating the amount of heat (in joules) that parts of your circuit will generate (across operating conditions if possible) to determine if heat mitigation is necessary.

Want to learn more about Electrical Thermal Co-Design?

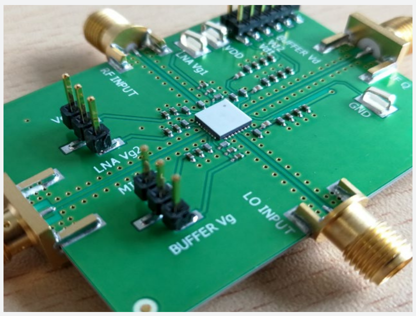
Download our e-book, Hot or Not? An Introduction to Electrical Thermal Co-Design

INTERFERENCE

DON'T LET OTHER SIGNALS GET YOU DOWN

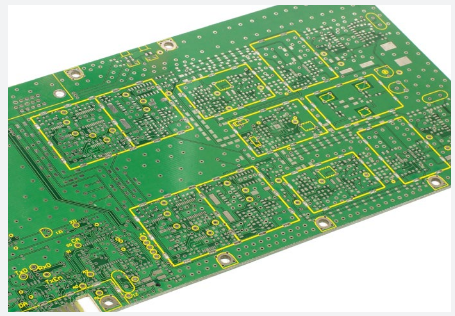
Physics is super fun to "bend to your will" (in case intent is not obvious, this is sarcasm). Almost everything in the environment can and will disrupt whatever RF signals you wish to transmit and/or receive. Even the signals generated in the PCB can negatively affect other areas of the PCB, which is why it's important to protect against such interference. This protection can be accomplished by many means including:

Via Fencing



Surrounding a section of the design with one or two rows of vias connected to ground is a very cost-effective means of shielding against EMI.

Physically Distancing

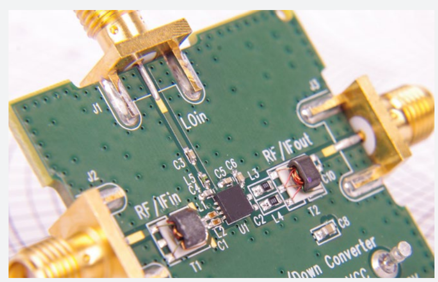


Separate certain sections or circuits on the PCB. For example, keep noisy power supply circuitry away from sensitive RF circuitry.

Physical Shielding

Affix physical metal shields over specific circuitry. When you absolutely need to keep EMI in or out, it's hard to beat metal shielding. This option must be carefully considered as metal shielding increases the weight and physical footprint of the design.

Via Stitching



If the design has large copper areas ("blank space") on multiple layers, systematically fill the copper areas with vias and "stitch" them together. This process provides shorter return loops for nets as well as maintains a lower impedance for the PCB.

Keeping the power supply circuitry on one side of the PCB and RF circuitry on the opposite side is one way to minimize interference but need not be the only way. You are encouraged to use whatever means necessary to prevent EMI from negatively affecting the circuit. Your only constraint is the physical dimensions and weight that the fully built product must conform to. Addressing interference and taking precautionary steps during the design process can protect your RF circuitry from the ill-effects of noise or otherwise unwanted signals.

Preventing interference can be very tricky. This is another area where simulation can be very useful especially if you can simulate all aspects of your design to determine the cumulative effect of product operation on performance.

PUTTING IT ALL TOGETHER

CAN YOU HEAR ME NOW?

We've come a long way from ringing bells on the other end of rooms or transmitting pulses of electricity over vast stretches of copper cable. Today, we can wirelessly access all human knowledge from our cell phones or even communicate with robots millions of kilometers from Earth. Getting to this point, however, involved much planning, as well as trial and error. The more planning you involve in the beginning, the less "trial and error" you'll have in the end.

That said, the "trial and error" step (i.e. testing your prototype) can be time and cost consuming. Being able to simulate most or all of the design before fabrication of the first PCB can help you find and correct issues beforehand. Software can help create, simulate, and validate the design:

Design

RF circuitry and additional layout considerations can be incorporated into the PCB with the advanced functionality of OrCAD PCB Designer and the Productivity Toolbox to easily implement:

- Grounding Techniques
- Via Stitching
- Via Fencing
- Shielding
- RF Signal Shielding
- Trace Antennas

Analysis

Ensure your RF circuitry functions as intended before testing with AWR. Achieve a comprehensive analysis of RF and microwave products including:

- Amplifiers
- Antennas
- RF/Wireless Communications
- Filters
- Passives
- Radars

EMA provides a closed-loop flow between design and analysis to ensure your RF structures are designed correctly the first time.

To learn more go to:

WWW
go.ema-eda.com/AWR

info@ema-eda.com

TRIAL
Get the Free Trial