A Practical Guide to RF for Embedded Designers

Small, wireless devices are rapidly proliferating, driven by a vast and expanding spectrum of industrial and consumer applications. Their designers must engineer greater functionality and capabilities into smaller enclosures. Many designers, however, may lack expertise or experience in RF solutions. Moreover, they confront a variety of RF technologies to choose from, such as Wi-Fi, Bluetooth, and cellular.

Overviews and specs for wireless technologies are readily available on the Web and elsewhere. Instead, this paper offers practical advice and best practices for embedding RF technologies that leverage many years of experience. It seeks to help designers make informed choices to meet the challenges effectively and economically.

We begin with the simple fact that designing wireless systems, as with most technologies, invariably demands tradeoffs.

Data Rate/Throughput Versus Range
A primary consideration is the impact that data rate, or throughput, has on range. As the data rate increases, the range for a wireless link decreases. One reason for this is that when the data rate increases, a higher signal to noise ratio is required for successful demodulation. To achieve higher signal noise ratio, more signal power is needed at the receiver, assuming a constant receiver noise floor. Therefore, when throughput grows—everything else being constant—the distance between the two antennas must decrease.

A second reason that increased data rate can lead to decreased range is that higher signal to noise ratio requires increased linearity in the transmitter power amplifier. Increased power amplifier linearity, with all else constant, is typically achieved by decreasing transmitter power, which further decreases wireless range.

Range Versus Antenna Size
This is another key tradeoff. As the antenna size decreases, the performance and radiation efficiency of the antenna decreases as well. Consequently, the range of the system decreases. For greater range, you generally need a larger antenna on both sides. Thus, squeezing an antenna into a smaller enclosure will reduce the device's range.

Power Consumption Versus Data Rate/Throughput & Range
Another tradeoff is that power consumption increases as data rates and/or range increase. To obtain longer range, your device will need more power. If you need to minimize power consumption, you may need to compromise data rates and/or range.

Unit & Service Costs Versus Infrastructure & Maintenance costs
There are many wireless and RF technologies and each has its own costs. Suppose, for example, you want to pull data from gas pumps and to do so, you need to deploy sensors at every gas station. You could use a Wi-Fi solution, which considering the number of

devices you require, would entail less hardware costs than putting a cellular modem inside each device. But while the hardware costs per unit would be less, consider the infrastructure expenses. How can you ensure reliability? If a Wi-Fi gateway fails, would the gas station attendant know how to fix it? With a cellular solution, a provider like AT&T or Verizon maintains the infrastructure. Your system would always work without requiring maintenance. Herein lies the tradeoff. A Wi-Fi system has more infrastructure and maintenance costs whereas a cellular system requires greater materials costs and monthly fees.

Pre-Certified Modules Versus Chipsets

Once you select your technology, you need to choose your hardware. There are two choices—pre-certified modules or chipsets.

Manufacturers offer pre-certified modules that contain the RF chip, microcontroller, and usually a simple communication interface back to the host. They often are already FCC certified for intentional radiator and radiated spurious emissions (RSE). They also frequently include the software stack and sometimes the antenna. Pre-certified modules can greatly reduce the time and costs needed for development. They also reduce design verification testing and certification expenses.

Chipsets are less expensive than modules and, theoretically, can consume less board space, but they are not pre-certified. You, rather than the manufacturer, must do the development work such as selecting the components, packaging them into a design, and then testing the product. All this takes time and money and increases certification risk. With their lower per unit costs, chipsets make economic sense when devices are needed in high volumes (>200k EAU).

Predicting Range

When designing a wireless system, one of the most important metrics is range. Range is simply the maximum distance between the two antennas in which communications can still exist. The calculation used to predict range is called a link budget. The equation below, called the Friis equation, is used to determine your range at minimum receive power. It is applicable for a free space environment in which there are no obstacles, reflections, or anything else between the two antennas. Margin is required for any communication link due to obstacles, reflections, etc., reducing the signal power at the receiver.

$$Pr(dBm) = Pt(dBm) + Gt(dB) + Gr(dB) + 20\log10(\lambda/(4*\pi*R)) - Margin(dB)$$

Where
Pr(dBm) = power incident at receiving antenna, in dBm
Pt(dBm) = power incident from the transmitting antenna, in dBm
Gt(dB) = gain of transmitting antenna, in dB
Gr(dB) = gain of receiving antenna, in dB
λ = wavelength in free space, in meters

R = distance, or range, between transmitting and receiving antennas, in meters
Margin(dB) = link margin, in dB

If you use radios for your system, your radio will have a receiver sensitivity specification. Pr is the power measured at the receiver. Pr must be greater than the receiver sensitivity for the radio to decipher the message over the noise. Transmitter output power is Pt. Power levels are often described in terms of dB, which is a logarithmic scale, due to the large variations of magnitude with wireless communications. dBW or dBm is an absolute power measurement. dBW is decibels relative to one watt and dBm is decibels relative to 1 milliwatt. You also need to factor in the gain of both antennas (Gr is the receive antenna gain and Gt is the transmit antenna gain). See Figure 1.

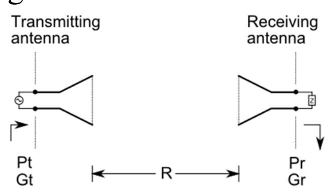


Figure 1.

$$Path\ Loss = 20\log10\left(\frac{\lambda}{4R}\right)$$

Path loss represents the propagation loss between the two antennas, which is a function of the distance between them. R represents the distance between the two antennas. Wavelength (λ) is a function of frequency.

The last parameter to consider in this equation is the link margin. You need to add a link margin factor to ensure your system functions properly at the maximum range you require. So if your system must operate at a minimum of 100 meters and you design your system so that your receive power matches receiver sensitivity, you will have spotty performance if anyone walks between the antennas, obstacles arise or if the environment changes (fading). You need to build in a lot of margin at 100 meters. Add in a link margin of 10–20dB to ensure you will have reliable communications at that range. A 10dB margin will provide 90% reliability; a 20dB margin will provide 99% reliability.

Antennas & Their Parameters
An antenna is any material used to radiate or receive electromagnetic waves and we all use them in our day-to-day lives. There are antennas in our smart phones, automobiles, homes, watches, and even pets (RFID).

Why is the antenna important? It is the first component in the receive chain. It sets the minimum noise floor for your device, which means how well your device can listen to other devices. It is also the last component in the transmit chain, impacting the maximum wireless range of your device as well as battery life. Here are some key antenna parameters every wireless engineer should know.

Impedance

Antenna impedance is the ratio of the voltage over the current at the antenna input terminals. Most people are familiar with impedance from their speakers. Your speaker might be 4 ohms or 8 ohms. Antennas have impedance as well. It should be matched to the system impedance, which is typically 50 ohms. When that happens, you have maximum power transfer and no reflections. Often, however, the antenna is not perfectly matched to the system impedance, which may not be 50 ohms. As a result, some of the power is reflected back and thus lost.

A few terms that define mismatch loss or reflection is reflection coefficient, VSWR (voltage to standing wave ratio), and return loss. An ideal antenna will have a reflection coefficient of zero, a VSWR of one, and return loss of zero.

Typically, good industry design practice is having a VSWR of two to one or a return loss of -10 dB. This means 90 percent of the energy actually gets to the antenna and ten percent is reflected back toward the power amplifier.

Directivity

Directivity is a measure of how well the antenna can concentrate or focus energy in any given direction. Figure 2 is a far-field radiation pattern where most of the energy is focused in the Z direction or the zenith. This is a high directivity, single beam antenna solution. A good application for such a solution would be a satellite TV antenna. Other antennas have an omni-directional pattern like the cellular antenna in your handset. Here, the energy is spread out more evenly throughout all angles from the antenna. Without an omni-directional pattern, you would have to point your cell phone at the nearest cell tower if you want reception, assuming you know where it is located. This is why you want an omni-directional, low directivity solution; the orientation of the device is trivial.

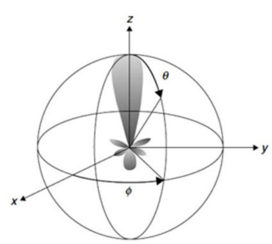


Figure 2.

Efficiency

Antenna efficiency is by far the most important parameter for small wireless embedded devices. This is defined by the radiated power over the input power to the antenna. Typically, small wireless devices will have an antenna efficiency in the 20 to 60 percent range. It is very difficult to approach 100 percent efficiency but it is possible; you just need enough volume to let the antenna radiate effectively. There is a direct tradeoff between antenna size and antenna performance. There is a constant struggle between industrial designers and antenna designers. You cannot fight with physics. Marketing and industrial designers want to make devices smaller and smaller, which means the mechanical engineer can provide less volume for the antenna. The struggle for the antenna designer is to get enough volume allocated for the antenna element so it can meet performance requirements such as efficiency.

Gain

Gain is the efficiency multiplied by the directivity. This is a good measure of how much power is radiated in any given direction. It is a logarithmic scale expressed in units of dBi. The "i" stands for isotropic. A gain of zero means the same amount of power is radiated from your antenna as that of an isotropic radiator.

Directivity and gain are a function of the angle from the antenna. Gain and directivity will change as a function of angle from the antenna. For example, with a single beam far-field radiation pattern, there will be a lot of gain in one direction, but the gain will drop considerably as you move off the main beam. Again refer to Figure 2.

Bandwidth

All of these antenna parameters are good only across certain frequency ranges. Antennas have finite bandwidth, a finite frequency over which they can operate. You cannot use a Bluetooth antenna for GPS or a GPS antenna for cellular. You can design a very broadband antenna that will cover GPS, Bluetooth, Wi-Fi, cellular, etc., but there will be a penalty in volume and size. The more bandwidth required or multi-band operation (cellular), the larger the antenna will need to be.

Antenna Types

The most commonly used antennas today are the half-wave dipole and quarter-wave monopole. The former is a balanced structure with two quarter-wave arms (see Figure 3). Quarter-wave monopoles have a one quarter-wave arm extended over a ground plane (see Figure 4). The ground plane is actually part of the antenna and its size and shape are critical to antenna performance.

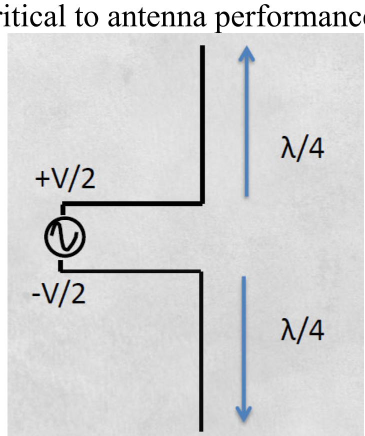


Figure 3

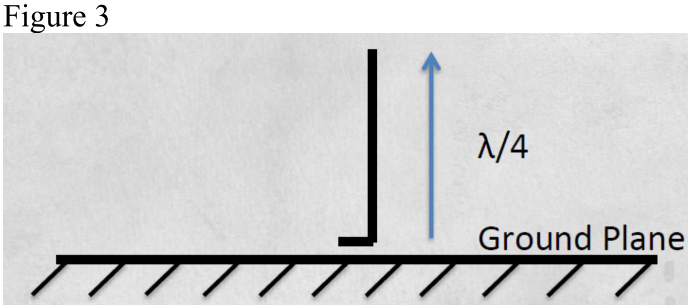


Figure 4.

Both monopoles and dipoles have an impedance close to 50 Ω (ohms). Half-wave dipoles are about 72 Ω; the quarter-wave monopole is half that at 36 ohms. This proximity to 50 Ω simplifies connecting your system with a minimum amount of loss or converting it to 50 Ω.

When to use dipoles and when to use monopoles

If a device already has a ground plane, space has been saved because it does not need quarter-wave arms. You can use one quarter-wave arm and the device's ground plane as the other half of the antenna. This is why most small, off-the-shelf devices that have internal antennas like PIFAs and chip antennas are of the monopole type. They use the ground plane as part of the radiator. On the other hand, the drawback is that monopoles are dependent on the ground plane or cable routing. If you have available space, use a dipole because it will keep currents off the cable or the ground system and has more consistent performance. When you can avoid concerns like cable routing, you will obtain more repeatable performance from device to device. But you do pay for it in size.

Frequency versus Wavelength

The size of a quarter wave depends on frequency. Wavelength (λ) is equal to the speed of light over frequency ($\lambda = c/f$). The takeaway is that as frequency goes up, wavelength goes down. An example is the lowest frequency used for LTE in the US is about 700 megahertz, which corresponds to about a 100 mm quarter wave or four inches. Notice that your smartphone is probably about four to five inches in length. Moreover, many 4G phones are larger than 3G phones. The larger phone size may be due to the market's desire for larger LCDs, but it also aids in antenna performance on lower frequency bands and provides more isolation between the ever-growing number of antennas in handsets today.

Does the Size of the Ground Plane Matter?

The ground plane size of the printed circuit board (PCB) can have a dramatic effect on antenna efficiency for monopole type antennas that utilize the ground plane for radiation. This includes PCB-mounted chip antennas. For optimal efficiency and bandwidth, the ground plane length should be a quarter wave or longer for the lowest frequency of operation. Reducing the long dimension of a PCB ground plane by 50% could reduce the antenna efficiency by as much as 50% for some antennas. Consequently, preserving the ground plane size that is recommended by the antenna vendor or designer is a key consideration for any monopole type antenna.

Embedded Device Antenna Choices

Chip antennas are off-the-shelf (OTS) antennas that are soldered directly to the PCB. They have a rapid development time, are relatively inexpensive, and consume the least amount of PCB real estate. They use ceramic material to load the antenna and reduce its effective size. If you have very tight size constraints on the board, you may need to use a chip antenna.

PCB trace antennas are a favorite of ours. They are mostly custom designs that are etched copper on the PCB. Figure 5 shows a printed quad-band cellular antenna (GSM850/GSM900/GSM1800/GSM1900). You have a few weeks of design, prototyping, and testing before you integrate the design into the customer's PCB CAD. The return on investment for a trace antenna is typically less than a year so they make a lot of sense for most customers. Once volumes exceed 10,000 pieces, custom designs become a logical choice.

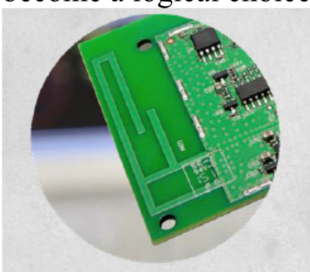


Figure 5.

Cabled antennas that are external to the device, on the other hand, offer the best performance, but consume the most space. A half-wave dipole will perform very well, but they are expensive. Additionally, the cable requires more manufacturing tolerances because there is always some kind of current on the cable. This is not much of a problem with dipoles, but in monopoles, the cable's route can impact performance. Cabled antennas internal to the device can also be advantageous when PCB space is limited and an external antenna is not feasible, but performance can be sensitive to cable routing, antenna proximity to noisy device electronics, and mechanical features that can reduce radiation efficiency.

Some Antenna Best Practices

Design the mechanicals and the enclosure around the antenna, not vice-versa.

Antennas are paramount for radio performance. Therefore, the mechanical design of wireless embedded devices must serve the needs of the antenna. Not the other way around.

Empty space is good space for the antenna.

Many mechanical engineers and industrial designers want to build products in which every single square centimeter within each device is used. You need volume around the antenna for it to radiate, however, so your designs must have enough space, especially considering your ground plane requirements. Ensure your ground plane is large enough to meet your efficiency and bandwidth performance targets.

Everything Affects Antenna Performance.

Nearly everything in small, wireless, embedded devices will conduct some RF current, which means the entire system is part of the antenna. This is a vital consideration. When designing custom antennas, we run electromagnetic simulations to project the RF currents on everything in the model—the ground plane, the battery, the connectors, the cables, LCD cans, shields, etc. They propagate to every piece of metal or any conductor within small devices. With everything part of the antenna, the key takeaway is you must tune the antenna in situ—in the final system configuration—to ensure the best performance.

Antenna Placement is Essential.
Keep antennas away from batteries, LCDs, cables, and EMI generators. Avoid conductors, absorbers, and dielectrics, which will detune the antenna, affect radiation patterns, and reduce efficiency. If the device is intended to be worn on the body, keep the antenna as far away from human tissue as possible. Placing switching power supplies close to the antenna will severely degrade receiver performance. Locating even a great antenna next to a noise generator will degrade system performance. Often, when customers complain about antenna issues, the problem is not the antenna but an electronics issue. There is simply too much EMI (noise) within the device that is radiating into the antenna and raising the noise floor at the receiver.

Antenna Testing

Tuning & Matching

A good antenna match results in maximum power transfer. You need to tune the antenna for the appropriate frequency bands for your device's application. You can measure impedance with a vector network analyzer and match it to the system impedance (typically 50Ω) for the most power transfer. You can also use the Smith chart or software tools (Atyune, AnTune, RFdude, ADS, HFSS) to calculate capacitor/inductor impedance matching networks. The point is to ensure your production level device has a tuned antenna to maximize the power transfer for the frequency bands that you want.

Over-the-Air Testing (OTA)

The last step of antenna development is OTA testing. Many customers lack the facilities for such testing and will turn to certified test labs. The device will sit on a turntable in the center of a ring of 23 antennas. As the turntable makes half a revolution, a full, 3D far-field pattern will result. You measure far-field radiation pattern, gain, efficiency, directivity, and polarization at the antenna range. Figure 6 shows a doughnut pattern that is typical of dipole or monopole antennas.

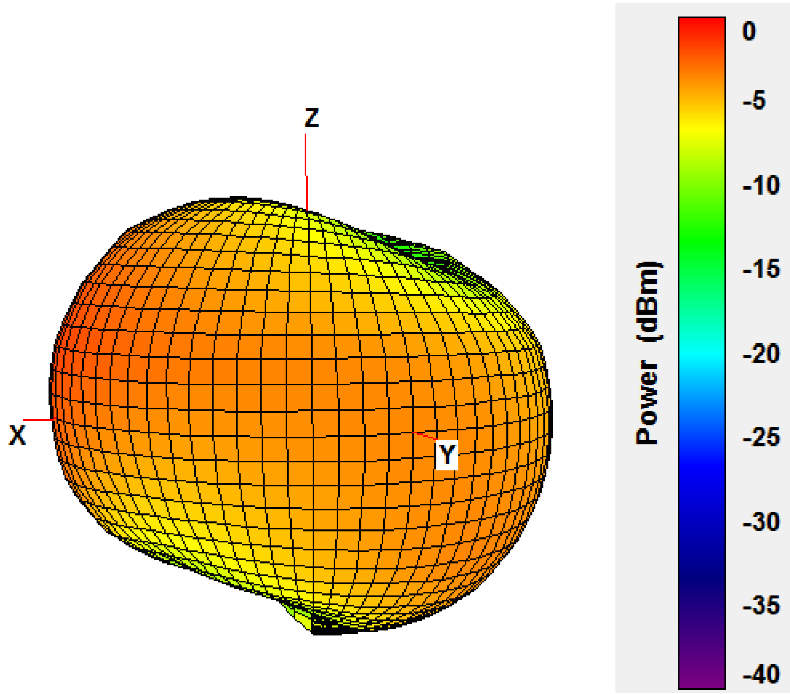


Figure 6.

Verification: What Needs to Be Tested

You have chosen your wireless technology, hardware, and antenna, and your RF product is developed. Now it is time to test your final design. But what should you test?

Conducted Output Power

What you need to test might vary depending on whether you designed with a module or a chipset. For both, however, you definitely want to test conducted output power. This measurement offers a lot of test coverage and can rule out many things in your build. It will identify major transmission line design errors and provide decent verification of your overall RF design around your chips or your radio. It can also provide some verification that your firmware has set your RF registers properly.

The measurement is made by cabling the output on your PCB to a spectrum analyzer, callbox emulator (e.g., CMW500) or a power meter. Keep in mind that your antenna is not part of this test.

Radiated Output Power

Once you measure conducted output power, you can test radiated power to ensure the antenna is connected correctly and has good performance. You may be able to test your antenna design with this measurement if you have a good frame of reference. Your available testing capabilities may vary, but making radiated output power measurements in a calibrated test lab is best. If you get a poor Effective Isotropic Radiated Power (EIRP), which is a single orientation measurement, or a poor Total Radiated Power (TRP), which is an integrated measurement of all angles from the antenna, you will know it is due to the device's RF design or antenna performance.

Receiver Sensitivity

Receiver sensitivity should be tested to ensure your device electronics (MCU, power supply, etc.) are not desensitizing the radio performance. Start out with a conducted receiver sensitivity test to ensure the chipset or module is close to the datasheet specs. Then, perform a radiated test like Total Isotropic Sensitivity (TIS), which is an integrated measurement of sensitivity at all angles from the antenna, to ensure EMI is not compromising your sensitivity.

Antenna efficiency/gain

If you bought a module with an antenna already on it, then you do not need to do antenna efficiency and gain measurements. But if you designed the antenna, this testing is a necessity, and is even recommended when integrating OTS antennas. It is highly recommended that you use a calibrated test lab for these measurements. They are a good way to verify antenna performance. If efficiency is too low, you may fail network operator OTA requirements. If gain is too high, you may fail FCC testing.

Wireless range

Testing for wireless range is a reasonable measurement to validate a final system design. Yet, this test is tricky. In RF engineering, we prefer to verify each component of a system. We measure signal quality at the transmitter output, output power, antenna efficiency, and receiver sensitivity, all component by component to verify a system. Testing for wireless range certainly is appealing, but the problem is that these tests are not easily repeatable or fully controlled. A range test in one environment will not necessarily predict how the system performs in another environment. This is why wireless range tests are subjective and not directly comparable to competitors' wireless range tests. With these caveats in mind, it is can still be worthwhile to conduct wireless range tests for high-level validation of product performance.

System-level Throughput

It is important to verify that the throughput and latency of your system is adequate for your use case. This is particularly true if you need to run time-sensitive data like video over Wi-Fi. You must ensure there are no bottlenecks in your RF link, in the interface between your RF chip and your host processor, or in your host processor when all of your application code is running.

Modulation Parameters

Modulation parameters are the real detailed test of your radio. Generally, module manufacturers perform this test for you, but if you rely on a chipset design, you will have to test modulation parameters such as Error Vector Magnitude (EVM).

RF Verification Test: Spectrum Analyzer

Transmitter Output Power & Spurious Emissions/Harmonics

Spectrum analyzers are very common in RF labs and are used to measure transmitter output power and frequency. They can also be used to test for spurious emissions, harmonics, phase noise, bandwidth, noise, and other RF parameters.

Near-field probing
It is very common to use a spectrum analyzer and near-field probe to look for noise and emissions on a PCB. You move the probe around the top of the board to locate noise. Often, however, you need to insert an amplifier between the probe and the spectrum analyzer to bolster the signal above the noise floor so it is visible on your spectrum analyzer screen. Without an amplifier, the noise you pick up with your near-field probe might be below the noise floor of the spectrum analyzer. An amplifier can get the signal above that noise floor.

RF Verification Test: Vector Signal Analyzer
Another common piece of test equipment, especially for chipset designs, is the vector signal analyzer or VSA. This is usually software installed on the spectrum analyzer that enables you to demodulate your incoming RF signal and measure EVM, spectral flatness, I/Q offset, and other modulation parameters. It can display the modulation constellation, providing a lot of important data.

RF Verification Test: "Call Box"
Another consequential piece of test equipment is the "call box," or wireless network emulator. It emulates a cellular base station for M2M products with cellular technology. The base station emulator box calls a mobile device and then controls it. It will turn the output power up and down and perform the full suite of cellular protocol and RF tests. The devices are common in the factories and design centers of cell phone manufacturers. They are costly, however, starting at about $80,000 per unit, and an array of optional features can elevate the price to over $500,000.

Regulatory Concerns: FCC
The Federal Communications Commission (FCC) is the government agency that controls emissions from electronic devices. Here are the basics of several critical aspects of FCC certification.

EMI: FCC Part 15B: Unintentional Radiator
Any electronic device that has anything within it that switches, has a clock, or operates with a frequency above 9kHz, even if there is no radio or RF, must have FCC Unintentional Radiator certification. When a device is tested, its radio transmissions are turned off to detect any unintentional emissions. If the radio transmissions cannot be turned off in a particular device, its RF output and harmonics are ignored in the testing.

Passing FCC Unintentional is much more difficult today than it was in the past. The problem is that clock speeds are increasing all the time and faster frequencies mean shorter wavelengths. Consequently, smaller structures on the PCB become better antennas for radiating the noise produced by these fast clock edges. Today, many designs that are well devised and function effectively often fail FCC Unintentional, whereas ten years ago, they would have passed. This is a growing challenge for designers.

FCC Intentional Radiator

FCC Intentional Radiator evaluates the characteristics of transmitter output for products that contains radios. It also evaluates unwanted emissions with the radio transmitting. The most common failure occurs when the harmonics of the transmitter are too high. One reason for this is a nonlinear power amplifier is delivering high harmonics to the antenna and the antenna has reasonable gain at the harmonic frequencies. In those cases, a filter between the radio and the antenna is a potential solution. Or if you have linearity problems in your power amp, make it more linear by turning down the power.

Another cause is when the structures on the PCB near the antenna pick up a strong fundamental frequency that the antenna radiates. These non-linear components—diodes, LNAs or switching power supplies—square the fundamental, producing harmonics. One tip is to keep radiated RF out of the power system.

FCC Modular Approvals
The third element is FCC Modular Approvals, which can be confusing. What does it mean when a cellular or RF module is pre-certified? FCC Modular Approval covers FCC Intentional Radiator only. That means if you buy a pre-certified module, you should be able to avoid the intentional radiator testing, but you will still need to submit your product to unintentional radiator FCC testing. When you deploy a certified radio module in your product that has an FCC grant that the manufacturer obtained after testing, you can leverage the grant if the grant conditions are met. Some typical FCC grant conditions are listed here.

First, the end product can contain no other radios within 20 cm of each other that can transmit at the same time. If you have other radiators within 20 cm of your certified module and those radios can transmit at the same time as yours, then you probably will need to do a Class II Permissive change for the highest power radio. Class II Permissive change will probably include some spot testing or at least some additional paperwork.

Second, the end device cannot be used with 20 cm of the human body. If it is, you may have to do specific absorption rate (SAR) testing (discussed below).

The third is to use an antenna of similar type and of equal or lower gain. When the manufacturer got its module approved, it needed to use some kind of antenna to do the testing. What you cannot do is put a high gain antenna on the module that could put a lot more power in one direction. You must use an antenna that is either similar to one that used on the modular grants or lower performing so you do not put out too much power in any given direction.

FCC: SAR (Specific Absorption Rate)
SAR testing determines the degree to which RF energy heats human tissue. SAR applies to any portable electronic devices where the transmitter could be within 20 cm of the human body. This is a well-publicized concern with cell phones, but it applies to all devices that might be used within close proximity to the body, such as wearables and Wi-Fi-equipped tablets and laptops that sit on our laps.

SAR testing is fairly expensive and can be avoided in some cases, such as when your output power, duty cycle or antenna gain is very low. To determine if this is the case, use the Maximum Permissible Exposure calculation, or MPE, which considers the frequency of your radio, the closest distance the device can be used to the body, and the radio's output power. (For more information, go to https://apps.fcc.gov/oetcf/kdb/forms/FTSSearchResultPage.cfm?switch=P&id=20676.) For example, the MPE calculation may be in your favor if your device has an extremely low-powered radio, even it if it is a wearable. The takeaway is if your device is within 20 cm of the body, you should do an MPE calculation to determine if SAR testing is needed.

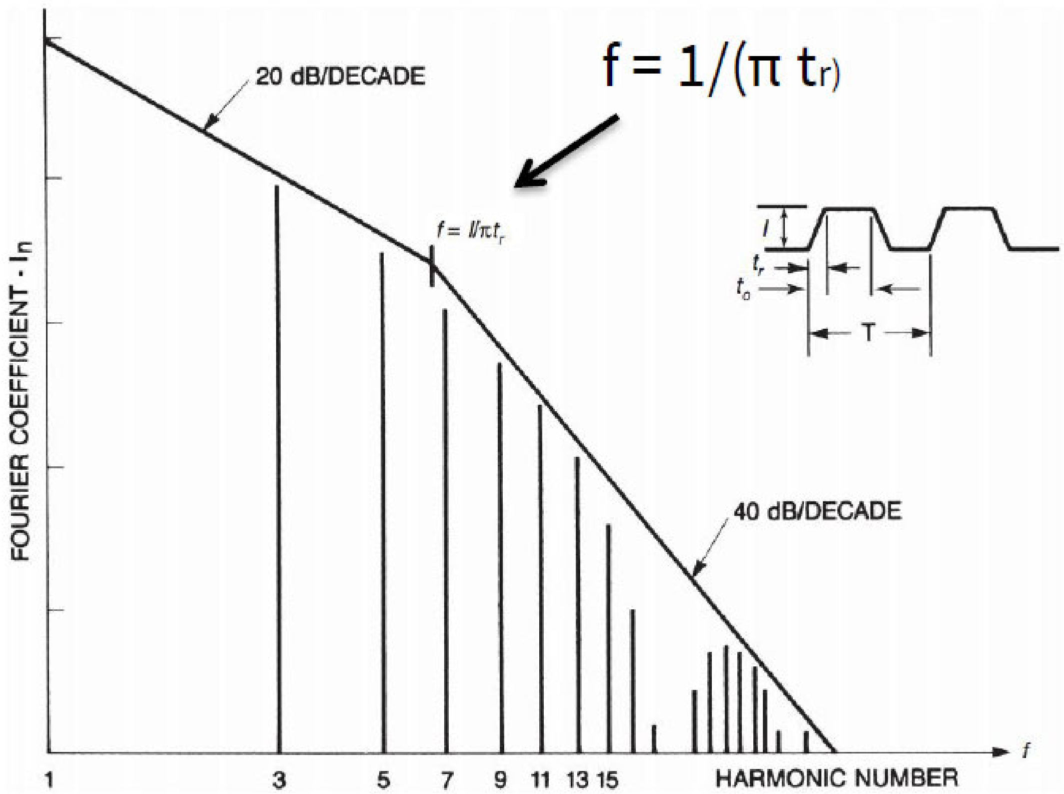
EMI & PCB Layout

Our final topic is EMI and PCB layout. Like it or not, high-speed PCB layout is RF engineering. There are many reasons for designing your PCB for low EMI. We addressed two—passing Part 15B Unintentional Radiator and Intentional Radiator for the FCC. The third is receiver "desense," or self-quieting.

EMI: Receiver "Desense" or Self-quieting

Receiver "desense" is when narrowband or broadband noise from the PCB and electronics in your device radiates from the board and is picked up in your antenna. Your antenna then sends that noise into your receiver, degrading your receiver's sensitivity and decreasing the range of your wireless link.

Figure 7 is the Fourier transform of a trapezoidal wave, which represents the noise radiated from common noise sources such as clocks and switching regulators. As the frequency moves out on the right, the noise power drops off. This means that the noise power is highest at lower frequencies. If you have a lower frequency radio like a 900MHz radio or especially a 300–400 MHz radio, the interfering noise power is higher than for higher frequency radios such as those that operate at 2.4GHz. That is why desense is such a problem for radios like LTE Bands 13 and 17 (700MHz), UMTS band V (850MHz), and ISM radios at 433MHz. The lower the frequency of your radio, the worse your desense becomes.



Source: Ott, Henry.
Electromagnetic Compatibility Engineering, p. 469

Figure 7. Envelope of Fourier spectrum of a 50% duty cycle trapezoidal wave.

Another concept to understand is that your receiver noise floor is very low compared to the FCC emissions specifications. This means that your product might pass FCC certification, yet may have very poor range due to receiver desense. To reduce noise, and therefore improve both signal to noise ratio (SNR) and range/throughput, as well as emissions that could impact regulatory compliance, it is important to design the PCB for lower EMI. Here are some tips.

Avoid Split Ground Planes
Start by avoiding split ground planes. Use a single ground system. You should almost never use separate analog and digital grounds. Many people think this is a good idea, but it can cause RF problems. What is ideal is having a continuous ground plane reference for every layer. Also, avoid any slots or large breaks in your ground plane. Return currents for signals will loop around slots or breaks, increasing their radiation. The goal is to keep the return current as close as possible to the direct current path, minimizing the loop area and the antenna efficiency of the trace.

Filter & Shield Cables
Filter and shield your cables because they are notorious for emitting noise. Display cables are especially problematic. It is wise to use a multi-layer flex PCB type of cable with two ground planes that sandwich the signal layer and noise and prevent the cables from radiating. Also, keep your cables as short as possible.

Shields
Do not create a product without putting shield footprints on the PCB. Plan for the shields upfront, knowing you will not have to populate them if found unnecessary.

Also plan to battle marketing and manufacturing. The marketing people will protest that shields might make the board a little bigger and costlier. Manufacturing will not want shields because they can be difficult to solder down and place. Indeed, testing the circuits underneath shields is difficult, but they are necessary and will help ensure compliance. One can use a two-piece shield to facilitate testing. Under this scenario, a wall is always populated on the board and the lid can be placed or removed easily. Visit a website that does tear downs of cell phones and tablets or a tear down such a device yourself and you will see that everything is underneath the shield. The message is simple. Do not design without shields; they could be your saviors.

Filter & Bury Clocks
Use crystals and not oscillators whenever possible! Place a resistor or maybe even a ferrite bead in series with the clocks running on your PCB. Put that series component as close to the output driver pin as possible.

Bury clock traces between the power and ground planes on your board. Do not have anything on the top or bottom layer or on the surface layers that does not have to be there.

Finally, flood the top and bottom of your PCB with ground pours. Flood all layers with ground pours and stitch with vias heavily (5mm grid).

Contain Power Supply Noise
It is very important to contain your power supply noise. Filter the inputs and outputs of all your regulators.

Switching regulators: Minimize the size of input and output current loops in the layout. Use smaller passives to help minimize current loops. Do not oversize the inductor or input/output capacitors. Always add a provision for a shield over switching power supplies unless you know the solution will not desense the radio. Use linear regulators in place of switching regulators if possible. The extra power loss may well be worth the increased sensitivity of your radio.

Route power as wide traces, not as planes. It is common practice to route power in large, low impedance power planes. Power is inherently noisy at RF frequencies, so we like to rout our power as very wide traces that have low impedance instead of using power planes. Power planes offer more opportunity for the noise to couple to other things and radiate.

Use Decoupling Capacitors
Use decoupling capacitors with high self-resonant frequencies on your board. Place them as close to the IC power pins as possible. It is important to use some small value caps, which are better at RF frequencies, because real capacitors in the real world act like inductors at high RF frequencies. Therefore, the lower your capacitor value, the higher the self-resonant frequency. Lower value capacitors perform better at high RF frequencies.

Watch out if you use a lot a different decoupling cap values on the same net. Different values can cause what are called anti-resonances, which can result in bad responses at certain frequencies. One very good practice is instead of using a lot of different values of decoupling caps, use multiples of the same values to avoid producing anti-resonances.

Postscript
This paper illustrates the point we made upfront. Commercial technology design always requires compromises. We provided some best practices, insights, and hints to help you make the most informed choices when designing solutions. Demand for small wireless devices will certainly increase as industrial and Internet of Things applications proliferate. We at Digi are committed to innovating the design and, consequently, the state-of-the-art of these technologies.

Key Takeaways

- ✓ Designing small wireless devices demands compromises.
- ✓ Data rates, range, antenna size, power consumption, and cost are key variables.
- ✓ Chipsets are less expensive than modules, but require greater certification.
- ✓ Design the mechanicals and the enclosure around the antenna, not vice-versa.
- ✓ Ensure antennas are tuned in final enclosures to maximize performance in the desired frequency bands.
- ✓ Test the final design for key parameters such as output power, receiver sensitivity, antenna gain, and range.
- ✓ If a wireless device is used within 20 cm of the human body, do an MPE calculation to determine if SAR testing is needed.
- ✓ It is important to design the printed circuit board for low EMI.
- ✓ Bottom line: effective solutions require informed choices.
