

Author:	Sierra Wireless			Date:	October 02, 2019			
APN Content Level	BASIC	INTERMEDIATE	✓	ADVANCED	Confidentiality	Public	✓	Private
Hardware Compatibility	Product Line	AirPrime	Series	ALL				
Software Compatibility	ALL	Document Type	Application Note	✓	Technical Note			

## 1 Version

These documents may be updated over their lifetime. To ensure you design with the correct version, please check [the Source](#) for latest versions.

## 2 Introduction

This document defines and describes antenna fundamentals, integrated antenna design and selection, as well as common antenna implementation issues. It mainly targets integrators with partial RF and antenna expertise, hence simplified RF and antenna technical details are being presented. This document intends to describe the importance and complexity of antenna design, and emphasizes on the high necessity of involving a credible antenna expert in the antenna design and selection process.

**Warning:** *Information herein are the best available as of issuing date. Not all antenna types are being considered. Sierra Wireless does not take any responsibility for the use of such information. We recommend customers to contact credible antenna design experts directly before making antenna solution decisions.*

## 3 Glossary

Term/Initials	Definition
PCB	Printed Circuit Board
OTA	Over-the-Air
RX	Receive Mode
TX	Transmit Mode
RSE	Radiated Spurious Emission
TRP	Total Radiated Power
TIS	Total Isotropic Sensitivity
VSWR	Voltage Standing Wave Ratio
RF	Radio Frequency
PTCRB	PCS Type Certification Review Board
FCC	Federal Communications Commission
GCF	Global Certification Forum
MNO	Mobile Network Operator
BOM	Bill of Materials
SAR	Specific Absorption Rate
RHCP	Right Hand Circular Polarization

## 4 Overview

An antenna is a device used to transform RF signals into an electromagnetic wave in free space, and to transform RF electromagnetic waves into electrical signals. It is one of the most critical components for wireless communication systems.

Antenna performance and RF layout design are the most critical components in a cellular product. Much of the cellular product success or failure depends on antenna performance. A poorly designed or implemented antenna and RF layout design are the most common causes of failures with regulatory and industry certifications. Too often, product designers either add an off-the-shelf antenna to the finalized IoT product design, or implement an antenna reference design without accounting for particular product characteristics such as dimensions, enclosure material, installation requirements, etc. This kind of approach has usually proved to be failing.

## 5 Antenna Basics

An antenna is a device to transmit or receive electromagnetic waves. Most antennas are resonant devices, which operate efficiently over a relatively narrow frequency band. An antenna must be tuned to the same frequency band as the radio system to which it is connected, otherwise reception or transmission will be affected.

The choice of antenna type and configuration is very important for communication systems. The antenna must be able to radiate or receive efficiently so the power supplied to the antenna is not wasted. Regardless of the antenna type, the performance can be categorized by the same metrics: impedance matching, bandwidth, directivity, gain, efficiency, polarization, and radiation pattern.

### 5.1 Impedance Matching, VSWR, and Reflected Power

The ability of an antenna to accept power from a source is determined by the input impedance. For maximum power transfer, the input impedance should exactly match the output impedance of the source. The 50  $\Omega$  system impedance level was chosen as the standard coaxial cable impedance. The quality of antenna impedance match is most commonly characterized by either Return Loss, or Voltage Standing Wave Ratio (VSWR). These impedance parameters measure how much of the power supplied to the antenna reflects from the antenna terminals.

Ideally, all of the power supplied to the antenna is radiated with no reflection. In practice, some reflection loss will still exist. The VSWR of 2:1 or less is often considered acceptable. Most commercial antennas are specified to be 1.5:1 or less over some bandwidth. Based on a 100-watt radio, a 1.5:1 VSWR equates to a forward power of 96 watts and a reflected power of 4 watts. The reflected power in such cases is 4%. The following conversion table presents reflected power for different VSWR values. It is also important to remember that although VSWR is important, other performance parameters also need to be considered. A resistor may be perfectly matched but will not radiate any RF energy; antennas can be the same.

Table 1: Return Loss to VSWR Conversion Table

Return Loss (dB)	VSWR	Reflection Coefficient, $\Gamma$	Mismatch Loss (dB)	Reflected Power (%)	Through Power (%)
1	17.39	0.891	6.868	79.43	20.57
2	8.72	0.794	4.329	63.10	36.90
3	5.85	0.708	3.021	50.12	49.88
4	4.42	0.631	2.205	39.81	60.19
5	3.57	0.562	1.651	31.62	68.38
6	3.01	0.501	1.256	25.12	74.88
7	2.61	0.447	0.967	19.95	80.05
8	2.32	0.398	0.749	15.85	84.15
9	2.10	0.355	0.584	12.59	87.41
10	1.92	0.316	0.458	10.00	90.00
11	1.78	0.282	0.359	7.94	92.06
12	1.67	0.251	0.283	6.31	93.69
13	1.58	0.224	0.223	5.01	94.99
14	1.50	0.200	0.176	3.98	96.02
15	1.43	0.178	0.140	3.16	96.84
16	1.38	0.158	0.110	2.51	97.49
17	1.33	0.141	0.088	2.00	98.00
18	1.29	0.126	0.069	1.58	98.42
19	1.25	0.112	0.055	1.26	98.74
20	1.22	0.100	0.044	1.00	99.00
21	1.20	0.089	0.035	0.79	99.21
22	1.17	0.079	0.027	0.63	99.37
23	1.15	0.071	0.022	0.50	99.50
24	1.13	0.063	0.017	0.40	99.60
25	1.12	0.056	0.014	0.32	99.68
26	1.11	0.050	0.011	0.25	99.75
27	1.09	0.045	0.009	0.20	99.80
28	1.08	0.040	0.007	0.16	99.84
29	1.07	0.035	0.005	0.13	99.87
30	1.07	0.032	0.004	0.10	99.90
31	1.06	0.028	0.003	0.08	99.92
32	1.05	0.025	0.003	0.06	99.94
33	1.05	0.022	0.002	0.05	99.95
34	1.04	0.020	0.002	0.04	99.96
35	1.04	0.018	0.001	0.03	99.97
36	1.03	0.016	0.001	0.03	99.97
37	1.03	0.014	0.001	0.02	99.98
38	1.03	0.013	0.001	0.02	99.98
39	1.02	0.011	0.001	0.01	99.99
40	1.02	0.010	0.000	0.01	99.99

It is important to note that both Return Loss (also called S11) and VSWR are simply different formats of the same impedance data. For example, VSWR of 2:1 correlates with logarithmic S11 of -9.5 dB.

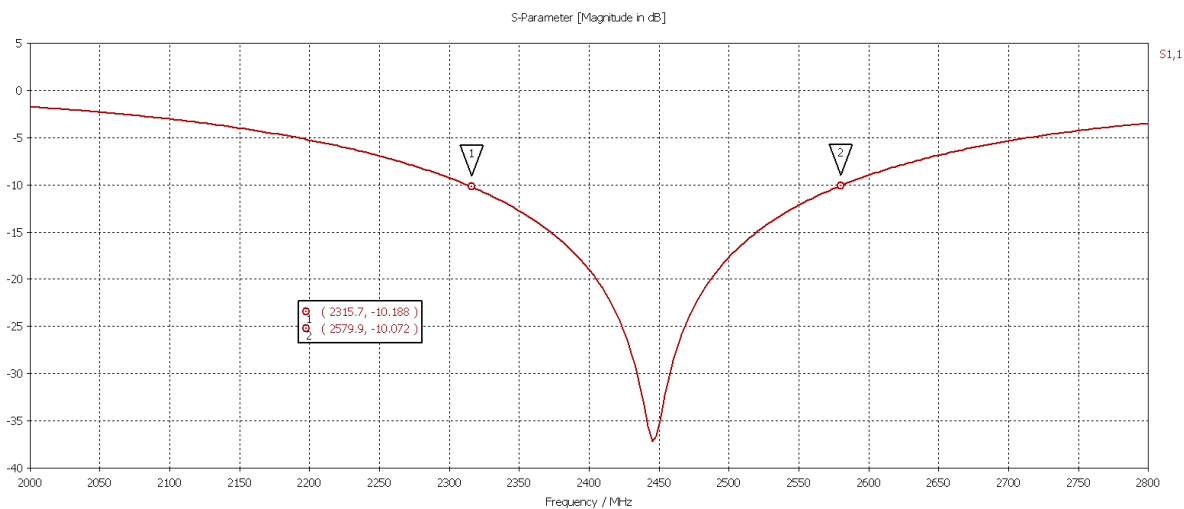


Figure 1. Antenna Return Loss (S11) vs. Frequency

## 5.2 Bandwidth

The bandwidth of an antenna is defined as the range of frequencies the antenna can support efficiently. Bandwidth specifications are set in each case to meet the needs of the particular application.

It is important to remember that frequency response is affected by environmental factors such as near objects and enclosures, as well as high volume production assembly margins, which results in degraded VSWR. Hence, it is very important to design an excess of bandwidth at the beginning of the product design.

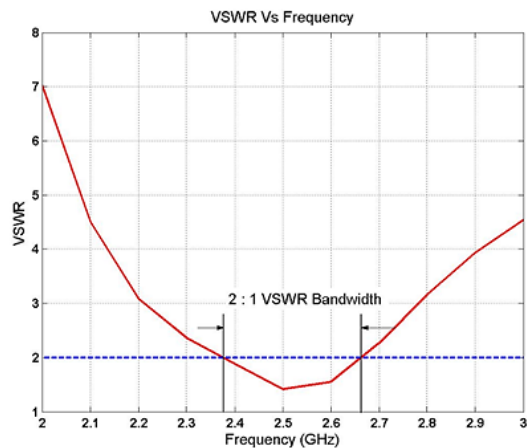


Figure 2. Usable Antenna Bandwidth – VSWR vs. Frequency

## 5.3 Gain and Directivity

Gain is a parameter which measures the degree of directivity of the antenna's radiation pattern. A high-gain antenna will radiate most of its power in a particular direction, while a low-gain antenna will radiate over a wider angle. High-gain antennas have the advantage of longer range and better signal quality, but must be aimed carefully at the other antenna. Low-gain antennas have shorter range, but the orientation of the antenna is relatively unimportant. Antenna gain should not be confused with amplifier gain, which is a separate parameter measuring the increase in signal power due to an amplifying device (power amplifier). In the USA, the FCC puts limits on the maximum antenna gain wireless systems can deploy to ensure that they don't cause interference. Sierra Wireless provides this maximum antenna gain information in its Product Technical Specification documentation. These limits must not be exceeded when integrating our modules.

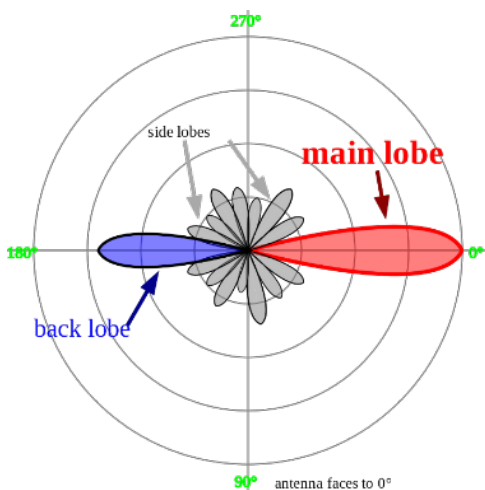


Figure 3. Antenna Directivity and Gain Example

Directivity of an antenna is defined as the ratio of the radiation intensity in a given direction from the antenna to the radiation intensity averaged over all directions. In other words, directivity is the ability of an antenna to focus energy in a particular direction when transmitting, and to receive energy better from a particular direction when receiving.

Omni-Directional antennas receive signals from all directions and are often called whip antennas. Omni-Directional antennas usually have a low gain. They are useful in situations where the mobile device is constantly moving such as on a vehicle, where the location of the nearest mobile tower is not known or when there is no clear line of sight to the tower and the signal is bouncing off nearby objects.

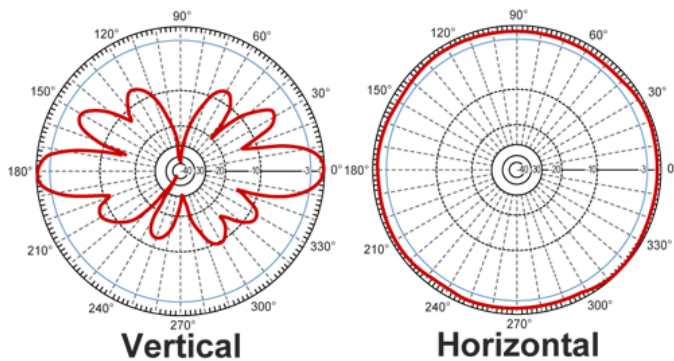


Figure 4. Omni-Directional Antenna Directivity Example

Directional antennas on the other hand, are designed to receive signals from a particular direction. Sometimes called a beam antenna, they often have a higher gain. They are possible in low to medium signal areas where there is line of sight or near line of sight to the tower, or where the antenna is placed on or near a wall or barrier that the RF signal cannot pass through.

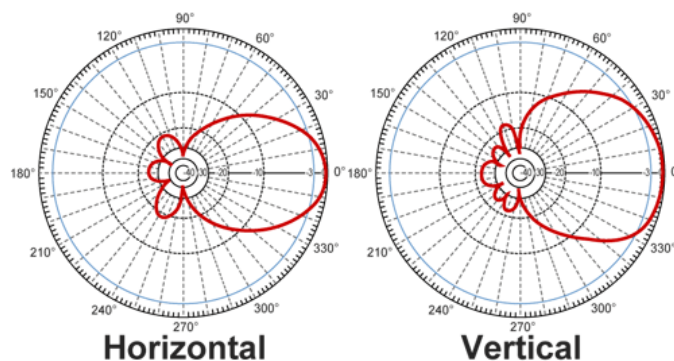


Figure 5. Directional Antenna Directivity Example

Most 2G, 3G and 4G systems are generally designed to be used with low gain omni directional antennas, high gain antennas can interfere with network operation.

## 5.4 Efficiency

Efficiency of a transmitting antenna is the ratio of power actually radiated in all directions to the power absorbed by the antenna terminals. The power supplied to the antenna terminals which is not radiated is converted into heat. This is usually through loss resistance in the antenna's conductors, but can also be due to dielectric or magnetic core losses in antennas or antenna systems using such components. Such loss effectively takes power from the transmitter, requiring a stronger transmitter in order to transmit a signal of a given strength.

Efficiency is one of the most important antenna parameters. Efficiency of small antennas that are tightly integrated into a small product form factor could be largely affected. Grounded conductors and dielectric materials nearby, like typical plastic housing, will constrain and absorb the near-fields of the antenna and cause significant losses. Common plastic materials can become significant absorbers of RF energy when very close to an antenna and even the plastic color can at times make a significant difference.

Antenna vendors sometimes only provide antenna gain information in their data sheets and leave out the efficiency measurements. It is important to remember that antenna gain only tells you about how directional the antenna is. Antenna efficiency information needs to be considered as well and should be provided by the vendor.

PCB mounted antennas normally also need some amount of tuning when placed into a product. Antenna vendors can often help with this tuning and can verify antenna efficiency with the antenna placed in your product.

The efficiency of a typical embedded antenna can range from about 40 to 75%. Higher than 75% efficiency is very difficult to achieve with an embedded antenna, and lower than 40% efficiency will typically cause certification failures. In most cases, the efficiency goal should be 60%, with 40% as an absolute minimum.

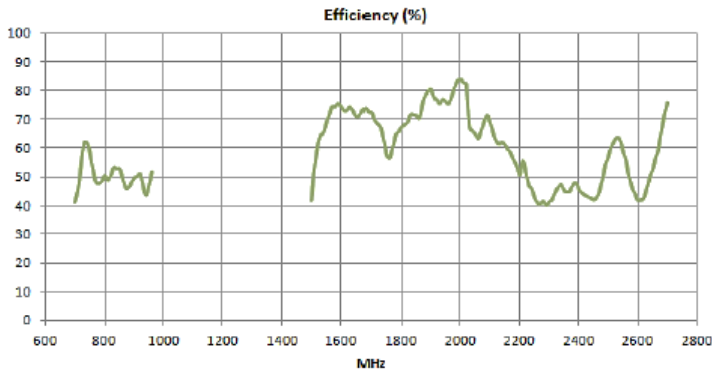


Figure 6. Antenna Efficiency Example

## 5.5 Radiation Patterns and Polarization

Radiation pattern describes the relative strength of the radiated field in various directions from the antenna at a constant distance; that is, which directions have more radiation and which have less. This information helps to orient the antenna properly in an application. Even an omni-directional antenna will have one or two narrow directions where there is a reduction in radiated energy.

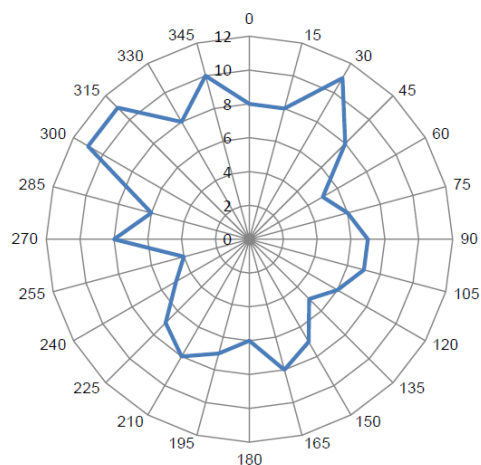


Figure 7. Antenna Radiation Pattern Example

The energy leaving an antenna is also polarized – meaning that the electric field is in a particular orientation. The polarization of an antenna refers to the orientation of the radio wave electric field with respect to the Earth's surface and is determined by the physical structure of the antenna and by its orientation. Thus, a straight wire antenna will have one polarization when mounted vertically, and a different polarization when mounted horizontally.

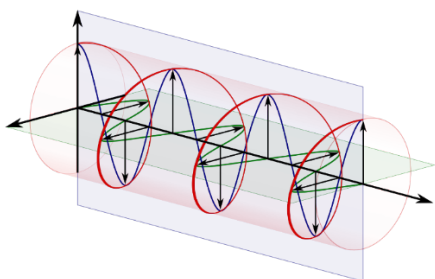


Figure 8. Antenna Polarization Example

Choice of polarization is one of the design choices available to the RF system designer. For example, low frequency (below 1 MHz) vertically polarized radio waves propagate more successfully near the earth than horizontally polarized radio waves. Therefore, mobile radio system waves are generally vertically polarized. This choice was made to maximize signal-to-noise ratios. At frequencies above 1 GHz, there is less dependence on the polarization type, although in specific applications there may be some advantage in one or the other.

While terrestrial wireless systems are linearly polarized, this is not the case with GNSS satellite positioning systems. GNSS systems generally use right hand circular polarization (RHCP). Circular polarization involves the plane of polarization rotating in a corkscrew pattern. The reason this type of polarization is used for satellite positioning systems is that the signal is less susceptible to atmospheric effects and the antenna polarization does not need to be lined up in the same way as a linearly polarized antenna. For optimum performance of a GNSS system, it is recommended that an RHCP antenna be used. Linearly polarized receive antennas can also be used but with an approximate 3dB hit in efficiency. LHCP antennas should never be used.

## 5.6 Antenna Diversity, MIMO and Envelope Correlation Coefficient

### 5.6.1 Diversity

In a real-world environment, the radio signal path between the base station radio tower and the wireless device is not always line of sight. Radio signals from the base station often bounce off a variety of different objects and structures creating multipath environments with different RF path lengths. This effect can cause the radio signal to increase or drop away when it reaches the antenna due to interference between the potentially out of phase multipath signals. To overcome this effect, a second diversity antenna can be used such that when one antenna is in a deep fade, the other can be separated in space so that it is outside the fade and still receive signal.

Diversity antennas make the radio link more robust but don't directly increase the maximum data throughput in themselves.

### 5.6.2 MIMO

MIMO (multiple input, multiple output) is a wireless communications technique in which more than one antenna is used at both the transmit and receive sides of the radio link. With this technique two data streams can be sent on the same frequency and due to the special separation between the antennas, the two data streams can be separated and decoded by the receiver increasing the data throughput capability of the radio link. MIMO works best in good radio conditions with line of sight between the transmitters and receivers. At least two receive antennas are needed on the wireless device for MIMO operation.

Due to the way LTE radios process the received signal and noise at the antennas, disparities in the antenna efficiency and gain between the antennas will reduce the throughput and should be kept to a minimum. Ideally, the antenna gain in balance should be kept to less than 3dB with a worst case of 6dB.

### 5.6.3 Envelope Correlation Coefficient

There is no magic separation distance for diversity antenna due to the complexity of the multipath fading distribution that can occur around a radio. However, when placed too close together in each other's near field antennas, they can couple together electrically and can effectively become one antenna such that the diversity benefit is lost.

Diversity antennas should be placed far enough apart electrically so that they don't combine in their near field and effectively become one antenna. In compact designs, it is usually best to separate the antennas as much as possible; antenna orientation can also be varied to reduce the antenna coupling effect.

Similar issues can occur with MIMO systems when the receive antennas are also too closely coupled such that the two data streams cannot be separated.

The Envelope Correlation Coefficient between the antennas is the cross-correlation value of the complex patterns of the primary and secondary antennas and essentially provides a guide to how similar and how closely coupled the two antennas are.

To ensure good diversity and MIMO performance, the Envelope Correlation Coefficient parameter should be kept lower than 0.5dB.

In an LTE system, the isolation from the primary antenna to the secondary receive antenna should be better than 8dB.

## 6 Antenna Selection

Sierra Wireless, a wireless module manufacturer, is an excellent information source for antenna selection and implementation guidance due to its comprehensive experience in the IoT space. Furthermore, obtaining antenna design and implementation assistance from credible antenna solution experts such as [Taoglas](#), [Pulse Electronics](#), and [Ethertronics](#), would potentially save very high costs associated with product re-design after failing FCC, PTCRB, GCF, or MNO RF certification requirements.

### 6.1 External vs. Embedded

Antennas for wireless devices can be completely embedded inside the product or can be mounted external to the product housing. While a properly designed external antenna will provide a better RF performance in most practical scenarios, modern product requirements usually require an embedded antenna.

External antennas are typically less expensive, less complex, and tend toward greater efficiencies. They are also easier to certify due to having less interference with other device components. Internal antennas come in a variety of sizes, bands, and efficiencies to provide design flexibility since there are many factors which can influence antenna efficiency and generate RF interference. For instance, conductive parts of the device will have the largest influence on antenna performance. Non-conductive components, such as housing materials, and PCB trace paths and lengths will also affect antenna design and performance.

### 6.2 Antenna Types

#### 6.2.1 PCB Trace Antenna

The PCB trace antenna is a relatively inexpensive solution. However, it requires a large footprint which has implications for the overall device size. It can be formed in a multitude of ways, has excellent multi-band capability, and can be made highly efficient. At the same time, PCB trace antennas are sensitive to detuning caused by housing materials. Printed antennas are strongly influenced by the PCB dielectric constant, which results in a reduction of the usable bandwidth, as well as reduced antenna efficiency.

The size and shape of the local ground plane is often an integrated part of the antenna design. The PCB trace antenna should be placed on the edge of a ground plane, i.e. on the edge of the PCB. The PCB trace antenna must not be surrounded by ground planes and other conducting surfaces.

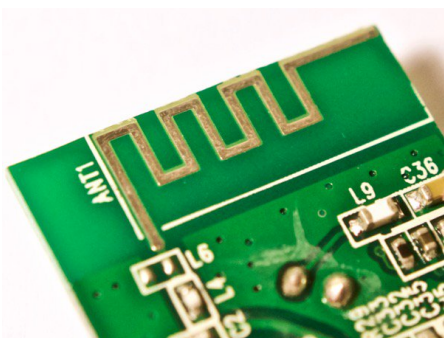


Figure 9. PCB Trace Antenna Example

### 6.2.2 Chip Antenna

For applications where the PCB size is extremely small, chip antenna could be a good solution. These are off-the-shelf antennas that offer reasonable performance, but increases BOM and assembly expense. Chip antennas require a large ground plane and sufficient spacing, and cannot be tuned by changing the antenna length. Tuning requires an additional antenna matching network, which increases the total BOM cost. It is important to mention that chip antennas are not fully isotropic – there is some preferred direction of radiation, which is also affected by ground clearance and housing assembly. It is generally recommended to only use chip antennas for applications that require an extremely small PCB area. Isotropic means that the antenna radiates equally in all directions.

When producing size estimations, IoT device designers must remember to include the antenna and its necessary clearance areas, as well as leave sufficient distance at the edge of the housing.

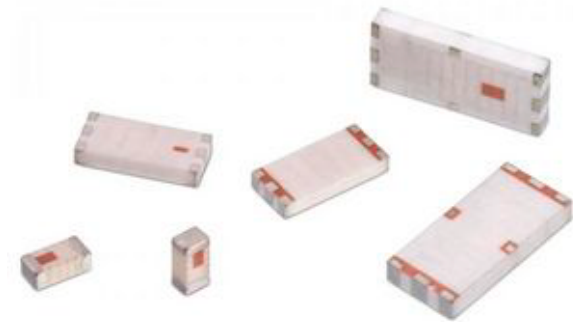


Figure 10. *Chip Antenna Example*

### 6.2.3 Microstrip Patch Antenna

The microstrip patch antenna is a low-profile antenna, which requires a ground plane beneath it. Microstrip patch antennas could be made multi-band, but not as easy as PCB trace antennas. Patch antennas are usually printed on the top surface of a 2-layer PCB. Patch antennas are considered as directional antennas, and usually considered for a fixed-mount device on the side of a building where the radiation from an omni-directional antenna would be wasted in the direction of the building.

Major operational disadvantages of the microstrip antennas are their low efficiency, low power, poor polarization, poor scan performance, spurious feed radiation and very narrow frequency bandwidth.

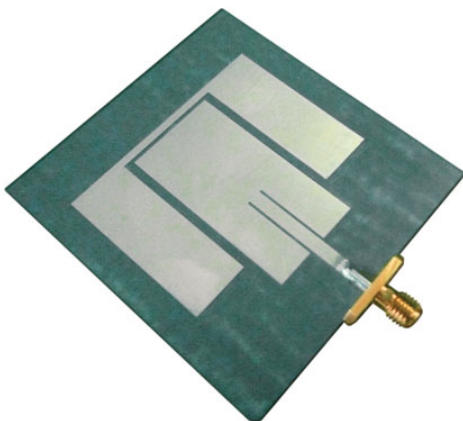


Figure 11. *Microstrip Patch Antenna Example*

## 6.2.4 Wire Antenna

Wire antennas are the classical antennas that are fixed on the PCB but rise from the PCB plane and protrude to free space over a ground plane. They have excellent RF performance as they are exposed to space as a 3D antenna. They also have the best range and have the most isotropic radiation pattern. The major disadvantage of a wire antenna is its required size, particularly in the lower RF bands. Even when the ground plane is utilized for half of the antenna design, the overall size can be large and difficult to put onto a PCB.

## 6.2.5 Balanced vs Unbalanced Antenna Designs

Most compact designs with integrated antennas use  $\frac{1}{4}$  wave unbalanced antenna type designs which act against the product's ground plane. Whilst this provides significant benefits in the antenna volume and proximity to the ground plane, it can result in the RF noise that is circulating in the ground being strongly coupled into the receive antenna. It also puts constraints on the ground plane dimensions and layout of the product which must be accounted for in the design. In contrast,  $\frac{1}{2}$  wave balanced dipole type wireless antennas often have better antenna performance and better noise rejection and can be easier to implement. However, they take up a larger volume and are normally placed further away from the ground plane.

# 7 Antenna Implementation

## 7.1 Ground Plane

The ground layer is extremely important in RF PCB design. The return path for the RF signal is in the ground plane beneath the RF trace. For good RF performance, the return path should be uninterrupted and as wide as possible. If the ground plane is interrupted, return currents find the next smallest path around the interruption. This forms a current loop, adding undesired inductance, affecting the impedance match between the radio and antenna, and attenuating the RF signal significantly. If the ground plane beneath the RF trace is narrow, it does not behave like a microstrip and may have more signal leakage.

## 7.2 Guidelines for Antenna Placement, Enclosure, and Ground Plane

- For board, mounted antennas, it is best to place the antenna in a corner of the PCB with sufficient clearance and as far away as possible from other components and traces.
- Always follow the antenna designer or antenna manufacturer recommended ground pattern for the antenna.
- Do not place any component, planes, mounting screws, or traces in the antenna keep-out area across all layers. A keep-out area depends on the antenna used.
- It is also not always good to place the antenna close to the plastic in the industrial design. Plastic has a higher dielectric constant than air. Proximity of the plastic to the antenna results in the antenna seeing a higher effective dielectric constant, which increases the electrical length of the antenna trace and reduces the resonant frequency. It may be possible to retune the antenna if this is unavoidable.
- The antenna must not be completely covered by a metallic enclosure. If the product has a metallic casing or a shield, the casing must not cover the antenna. The metal is not allowed in the antenna near-field.
- The antenna orientation should be in line with the final product orientation so that the radiation is maximized in the desired direction.
- Plan to have a provision for an antenna matching network since lots of parameters in the antenna's proximity (plastic, ground variation, etc.) can vary its impedance, and therefore the antenna may need retuning.
- When using the matching network values provided by the antenna manufacturer, ensure that you use the trace length from the antenna to the matching network specified in the manufacturer datasheet or reference design.
- Always verify the antenna matching network with the final plastic enclosure in place and the product placed in typical use case scenarios. Cable routing can make a significant difference to antenna performance so should be controlled.
- Allow for enough ground clearance from the antenna; this ground plane must have a minimum width.
- Keep a dedicated ground layer. Do not have traces running across the RF trace in the ground plane.
- To reduce coupling between antenna and other signals and improve EMC, the top and bottom layers should be exclusively reserved for GND – to cover as much of the whole surface on the top and bottom side by GND as possible. This ground area should be connected with many vias spaced not more than one-twentieth of the wavelength of the operating frequency.

- The application board should be designed in the way to support a plain GND connection on the whole surface located under the wireless module area. A matrix of via should be implemented to connect the top layer (in contact to the module's LGA ground pads) to other adjacent GND layers of the PCB to maximize the supporting GND plan. Such implementation aims to reduce noise coupling interference, spurious radiation and improve heat dissipation.
- Do not have split grounds unless you can ensure that no current loops are formed in the ground for the current in the return path.
- Allow a wide ground plane beneath the RF trace. Narrow ground planes increase leakage.
- The bottom ground plane, together with the top ground plane and vias between the two ground planes, ensures that all traces are well shielded. This arrangement significantly improves the EMI and EMC performance.
- Always consider the wireless radio, antenna and RF trace routing and placement as one of the first considerations in your design.
- It is recommended to cover the corners and edges of the power plane with ground and via holes connecting ground planes on either side of the power plane. This helps to capture unwanted EMI emitted from the power plane through the board edges.
- For PCB mounted antennas, keep high components away from the antenna element as most PCB mounted antennas use the PCB ground plane as the counterpoise and RF energy can be lost due to coupling to these types of components.

## 7.3 Common Antenna Implementation Mistakes

### 7.3.1 Dipole or Monopole Parallel to Ground Plane

The commercial dipole or monopole style antenna is often mounted close to and parallel to a ground plane. These antennas must be mounted perpendicular (or within 25 deg of perpendicular) to the local ground plane. Both impedance and efficiency can suffer dramatically from improper mounting.

### 7.3.2 Antenna in Conductive “Window”

Sometimes a product is housed in a metal enclosure for functional or aesthetic reasons. If the antenna is not external to the enclosure, the typical improper solution is to open up a “window” in the metal enclosure. This idea seems straightforward but usually fails. An antenna in a conductive window will present a very degraded efficiency (up to 15 dB). Thus, windows for antennas should be avoided completely.

### 7.3.3 Printed Antenna Surrounded by Ground Plane

Printed antennas should be placed on the periphery of the PCB under consideration. Ideally, the antenna should have three edges exposed (at one end of the PCB). If this is not possible, the printed antenna should be placed in the corner of the board so two sides are exposed. Surrounding the printed antenna on three or four sides will degrade performance significantly.

### 7.3.4 Antenna Impedance not Validated in Product

The commercial antenna specifications present the manufacturer measured return loss parameters. While this is useful data, antenna impedance must be validated with the antenna installed in the product, with all of the surrounding materials in place, and with all components installed on the PCB. Even the small details of a plastic enclosure can influence the impedance of the antenna.

### 7.3.5 Antenna Too Close to Digital Noise Source

Electrical noise generated by the digital circuitry of the device can enter the antenna directly and result in poor or degraded performance, and failure at the PTCRB test lab. Care should be taken to locate the antenna as far as possible from digital noise sources. It is recommended to make provision for RF shielding where proximity cannot be avoided.

### 7.3.6 Antenna Placed in Poor Location with regard to Body Effects and SAR Performance

If the distance between the human body and the wireless device / antenna is less than 20 cm, SAR requirements and limits will apply (both for CE and FCC certification). In order to pass the over the air and SAR requirements, careful antenna placement and design is required. It is recommended that you work closely with your antenna vendor or design house to ensure you meet these requirements unless you have significant experience with antenna design.

Interaction with a human body can cause de-tuning of the antenna and introduce power loss in both receive and transmit operations. Interaction with a human body significantly degrades performance and can cause SAR certification failures. Specific testing should be performed to properly quantify the effects of the human body both from an SAR and over the air performance perspective.

## 7.4 Antenna Design and Implementation Assistance

There are three primary approaches available to achieve the intended antenna implementation and performance:

- Commercial off-the-shelf antenna
- Reference antenna design
- Custom antenna design

Most of antenna experts recommend reference antenna design and custom antenna design approaches for embedded antennas. It is important to note that antenna reference designs would still require some custom design modifications in a new device.

When evaluating a reference design, it is important to know whether it presents the actual measured data from a working antenna, or just a simulation data from the antenna-design software. Simulation data is useful, but usually requires some significant adjustments once implemented in the hardware. A reference design must never be implemented without the actual measured data confirmation of its performance parameters.

When implementing a PCB mounted reference antenna design, it is imperative that you have the antenna vendor review your board layout and dimensions to make sure their reference antenna will work with your design. It is also good to have them assist with the antenna tuning and optimization process to ensure optimal antenna performance.

It is strongly recommended that you evaluate the OTA performance (TRP and TIS) to verify antenna efficiency and to ensure that any noise that might cause receiver desensitization is kept to a minimum. It is essential that this testing be performed at an early stage of the product integration. Sierra Wireless may be able to assist with these measurements if required.

## 7.5 Antenna System Issues

### 7.5.1 Connecting the Antenna

The front-end is comprised of the antenna, power amplifiers, filters, switches, and linear components. It is critical that front-end design is made with the controlled impedance structures. These structures could be a coaxial cable, PCB traces, flex circuits with controlled impedance lines, etc. Attention must be paid to the various interfaces between these structures and the antenna and circuit terminals. Circuit components and antennas that are well-matched when measured in isolation can suffer from impedance-matching issues when used in combination with poorly designed interconnects. It is important to mention that most RF filters rely on good impedance matching to provide specified filtering. A high Return Loss would not allow to properly terminate the filter component. In such cases, some tuning may be required to ensure that the filter is operating properly. Cables and PCB traces might also introduce losses and create opportunities for noise to enter the RF system.

In general, it is recommended that the path loss between the module and the antenna is kept to < 0.5dB to meet carrier over the air performance requirements.

RF traces running on outer layers of the PCB can pick up RF noise. Where possible, shielded RF transmission line topologies should be used.

Micro coaxial cables can have limited shielding properties; when routing these types of cables between, it is good to ensure that these lines are as short as possible and routed away from the PCB and RF noise generating circuits.

### 7.5.2 Self-Interference

Self-interference is created when the emissions from the device's own internal circuitry enter the RF front end and raise the receiver noise floor, degrading receiver sensitivity. Self-interference is known to be one of the biggest challenges to successful device certification. Self-interference can result in a sensitivity reduction of 10 dB or higher, which could significantly reduce the operating range and performance of the cellular device.

The primary interfering noise sources often come from digital components, such as the CPU and memory, RF oscillators and switch mode power supplies. Digital noise could also come from other sources, but CPU and memory are often the main contributors. Eliminating self-interference can be a complex task. Product designers should follow the following guidelines:

1. Shield digital and power components.
2. Place antennas far from digital components.
3. Route antenna interconnects away from digital components.
4. High speed digital signals like USB should be isolated and placed as far from RF traces and the antenna as possible.
5. All signals including RF lines should be located on inner layers and surrounded by GND as much as possible.
6. Vias should be added to link the top and bottom layers to improve grounding. A line of vias should also be added to isolate incompatible signals located on inner layers (RF vs digital signals, including but not limited to USB, UART, GPIO; SIM signals; and power supplies lines).
7. A ring of GND vias should be present all around the PCB edge.

Unfortunately, it is very difficult to completely understand the full extent of any self-interference until the product design is completed. Therefore, it is critical to design shielding and other self-interference mitigations into the device from the very beginning.

### 7.5.3 Co-existence with other Wireless Technologies

Wireless products often integrate multiple transceivers and receivers, operating in various bands and technologies, such as Cellular, WiFi, GNSS, Short Range RF, etc. Integrating multiple internal antennas in a small IoT device (such as OBD dongle) could be a very challenging task, since all these wireless technologies will transmit their own signals and generate noise, which can interfere with cellular band reception. Equally, the cellular transmitter generates large-amplitude signals which can overpower nearby receivers and cause signal distortion. The output of the typical cellular radio transmitter is high enough to interfere with a nearby WiFi, Bluetooth, and GNSS receiver, even though the frequency bands are separated by hundreds of MHz. As with self-interference, co-existence issues are difficult to measure before a product is assembled and operating. Good design decisions early in the process are important – another reason to obtain qualified expert advice as early in a project as possible.

## 8 Summary

The antenna is one of the most critical components for wireless communication systems. It cannot simply be added to the product at the end of the design phase, since the entire product design influences antenna performance. Antenna implementation must be holistic in approach, involving the entire device and considered from the very beginning of the product design. Understanding and anticipating various RF issues is critical in the design approach. It is vital to integrate various mitigation options into the device from the very beginning of the design process. Resolving RF issues during the FCC, PTCRB, GCF, and MNO certification phases have been shown to be very difficult, time consuming, and expensive.

Sierra Wireless, and our credible partners such as [Taoglas](#), [Pulse Electronics](#), and [Ethertronics](#), would be excellent information sources for antenna selection and implementation guidance due to our comprehensive experience in the IoT space. The expertise provided by us or other credible vendors should be used in case the product designers possess limited RF and antenna expertise.

## 9 Support

For direct clients: contact your Sierra Wireless FAE.

For distributor clients: contact your distributor FAE.

For distributors: contact your Sierra Wireless FAE.

## 10 Document History

Level	Date	History
1.0	January 25, 2019	Creation
1.1	October 02, 2019	Added: <ul style="list-style-type: none"> <li>• 5.6 Antenna Diversity, MIMO and Envelope Correlation Coefficient</li> <li>• 6.2.5 Balanced vs Unbalanced Antenna Designs</li> </ul>
		General updates

## 11 Legal Notice

### Important Notice

Due to the nature of wireless communications, transmission and reception of data can never be guaranteed. Data may be delayed, corrupted (i.e., have errors) or be totally lost. Although significant delays or losses of data are rare when wireless devices such as the Sierra Wireless modem are used in a normal manner with a well-constructed network, the Sierra Wireless modem should not be used in situations where failure to transmit or receive data could result in damage of any kind to the user or any other party, including but not limited to personal injury, death, or loss of property. Sierra Wireless accepts no responsibility for damages of any kind resulting from delays or errors in data transmitted or received using the Sierra Wireless modem, or for failure of the Sierra Wireless modem to transmit or receive such data.

### Safety and Hazards

Do not operate the Sierra Wireless modem in areas where cellular modems are not advised without proper device certifications. These areas include environments where cellular radio can interfere such as explosive atmospheres, medical equipment, or any other equipment which may be susceptible to any form of radio interference. The Sierra Wireless modem can transmit signals that could interfere with this equipment. Do not operate the Sierra Wireless modem in any aircraft, whether the aircraft is on the ground or in flight. In aircraft, the Sierra Wireless modem **MUST BE POWERED OFF**. When operating, the Sierra Wireless modem can transmit signals that could interfere with various onboard systems.

---

*Note: Some airlines may permit the use of cellular phones while the aircraft is on the ground and the door is open. Sierra Wireless modems may be used at this time.*

---

The driver or operator of any vehicle should not operate the Sierra Wireless modem while in control of a vehicle. Doing so will detract from the driver or operator's control and operation of that vehicle. In some states and provinces, operating such communications devices while in control of a vehicle is an offence.

### Limitations of Liability

This manual is provided "as is". Sierra Wireless makes no warranties of any kind, either expressed or implied, including any implied warranties of merchantability, fitness for a particular purpose, or noninfringement. The recipient of the manual shall endorse all risks arising from its use.

The information in this manual is subject to change without notice and does not represent a commitment on the part of Sierra Wireless. SIERRA WIRELESS AND ITS AFFILIATES SPECIFICALLY DISCLAIM LIABILITY FOR ANY AND ALL DIRECT, INDIRECT, SPECIAL, GENERAL, INCIDENTAL, CONSEQUENTIAL, PUNITIVE OR EXEMPLARY DAMAGES INCLUDING, BUT NOT LIMITED TO, LOSS OF PROFITS OR REVENUE OR ANTICIPATED PROFITS OR REVENUE ARISING OUT OF THE USE OR INABILITY TO USE ANY SIERRA WIRELESS PRODUCT, EVEN IF SIERRA WIRELESS AND/OR ITS AFFILIATES HAS BEEN ADVISED OF THE POSSIBILITY OF SUCH DAMAGES OR THEY ARE FORESEEABLE OR FOR CLAIMS BY ANY THIRD PARTY.

Notwithstanding the foregoing, in no event shall Sierra Wireless and/or its affiliates aggregate liability arising under or in connection with the Sierra Wireless product, regardless of the number of events, occurrences, or claims giving rise to liability, be in excess of the price paid by the purchaser for the Sierra Wireless product.

### Patents

This product may contain technology developed by or for Sierra Wireless Inc.

This product includes technology licensed from QUALCOMM®.

This product is manufactured or sold by Sierra Wireless Inc. or its affiliates under one or more patents licensed from MMP Portfolio Licensing.

### Copyright

© 2019 Sierra Wireless. All rights reserved.

### Trademarks

Sierra Wireless®, AirPrime®, AirLink®, AirVantage®, WISMO®, ALEOS® and the Sierra Wireless and Open AT logos are registered trademarks of Sierra Wireless, Inc. or one of its subsidiaries.

Watcher® is a registered trademark of NETGEAR, Inc., used under license.

Windows® and Windows Vista® are registered trademarks of Microsoft Corporation.

Macintosh® and Mac OS X® are registered trademarks of Apple Inc., registered in the U.S. and other countries.

QUALCOMM® is a registered trademark of QUALCOMM Incorporated. Used under license.

Other trademarks are the property of their respective owners.