Antenna Design Guide

Author: Tapan Pattnayak

Associated Part Family: CY8C4XX7-BL, CY8C4XX8-BL, CYBL10X6X, CYBL10X7X

Related Application Notes: None

To get the latest version of this application note and the associated Gerber file, please visit http://www.cypress.com/go/AN91445

AN91445 explains antenna design in simple terms and recommends two Cypress-tested PCB antennas that can be implemented at a very low cost for use with the Bluetooth Low Energy (BLE) solutions that are part of Cypress's PRoC™ and PSoC® families. The PRoC BLE and PSoC 4 BLE 2.4-GHz radio must be carefully matched to its antenna for optimum performance. It concludes by showing how to tune the antenna in the final product.

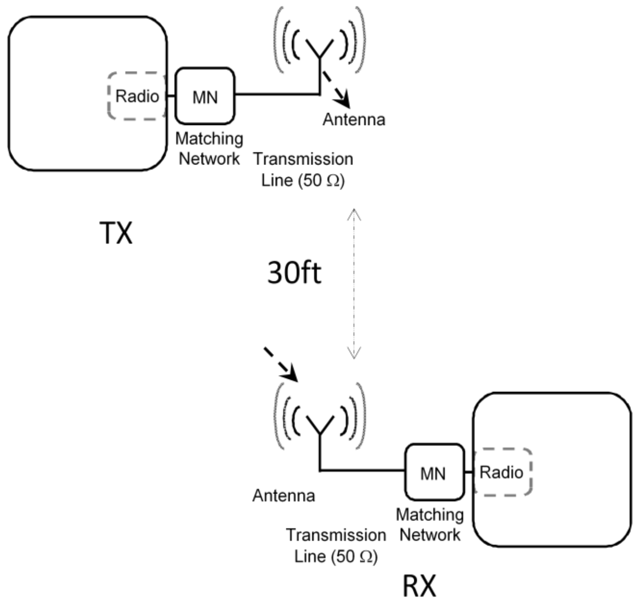
Contents

1 Introduction

An antenna is a critical component in a wireless system that transmits and receives electromagnetic radiation in free space. The wireless range that an end-customer gets out of an RF product with a current-limited power source such as a coin-cell battery depends greatly on the antenna design, the enclosure, and a good PCB layout.

It is not uncommon to have a wide variation in RF ranges using the same silicon and the same power but different layout and antenna-design practice. This application note describes the best practices, layout guidelines, and an antenna-tuning procedure to get the widest range with a given amount of power. This is an important consideration for BLE system which has to operate from a tiny power source. Below Figure shows the critical components of a wireless system both at the Transmitter (TX) and Receiver (RX).

Figure 1. Typical Short-Range Wireless System

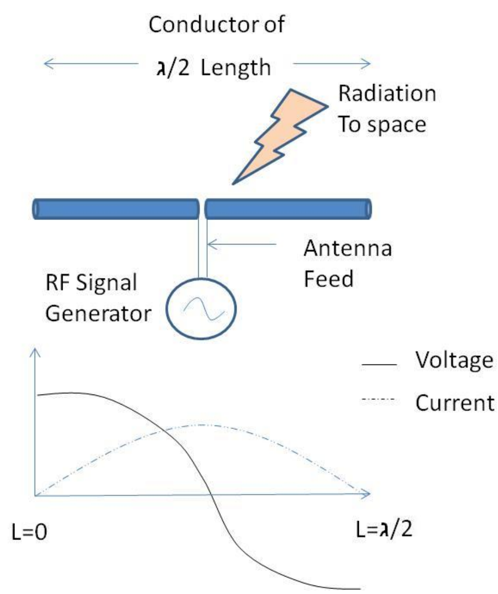


A well-designed antenna increases the operating distance of the wireless product. The more power it can transmit from the radio, the larger the distance it can cover for a given packet error rate (PER) and receiver sensitivity. Similarly, a well-tuned radio in the receiver side can work with minimal radiation incident at the antenna.

2 Antenna Basics

An antenna is basically a conductor exposed in space. If the length of the conductor is a certain ratio or multiple of the wavelength of the signal[1], it becomes an antenna. This condition is called “resonance”, as the electrical energy fed to antenna is radiated into free space.

Figure 2. Dipole Antenna Basic



[1] See “harmonic antenna operation”

In Figure 2, the conductor has a length λ/2, where λ is the wave length of the electric signal. The signal generator feeds the antenna at its center point by a transmission line known as “antenna feed”. At this length, the voltage and current standing waves are formed across the length of the conductor, as shown in Figure 2.

The electrical energy input to the antenna is radiated in the form of electromagnetic radiation of that frequency to free space. The antenna is fed by an antenna feed that has an impedance of, say, 50 Ω, and transmits to the free space, which has an impendence of 377 Ω[2].

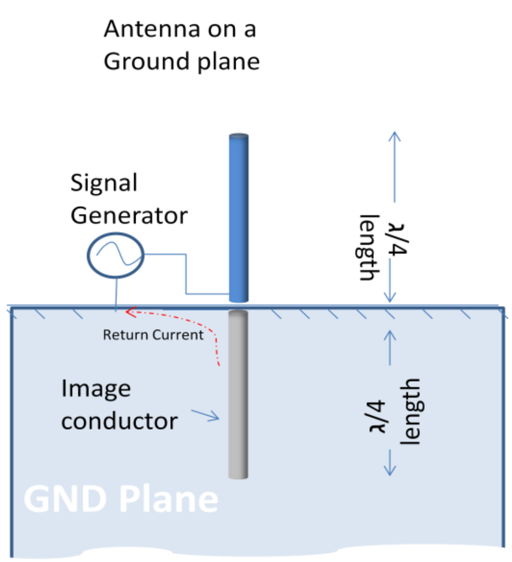
Thus, the antenna geometry has two most important considerations:

1. Antenna length
2. Antenna feed

The λ/2-length antenna shown in Figure 2 is called a dipole antenna. However, most antennas in printed circuit boards achieve the same performance by having a λ/4-length conductor in a particular way. See Figure 3.

By having a ground at some distance below the conductor, an image is created of the same length (λ/4). When combined, these legs work like a dipole antenna. This type of antenna is called the quarter-wave (λ/4) monopole antenna. Most antennas on the PCB are implemented as quarter-wave antennas on a copper ground plane. Note that the signal is now fed single-ended and that the ground plane acts as the return path.[3]

Figure 3. Quarter-Wave Antenna



For a quarter-wave antenna that is used in most PCBs, the important considerations are:

1. Antenna length
2. Antenna feed
3. Shape and size of the ground plane and the return path

[2] Impedance of Free Space if there is no material nearby

[3] We will see the effect of this return path later. This is a very important aspect in PCB layout of the antenna and the antenna feed.

3 Antenna Types

As described in the previous section, any conductor of length λ/4 exposed in free space, over a ground plane with a proper feed can be an effective antenna. Depending on the wavelength, the antenna can be as long as the FM antenna of a car or a tiny trace on a beacon. For 2.4-GHz applications, most PCB antennas fall into the following types:

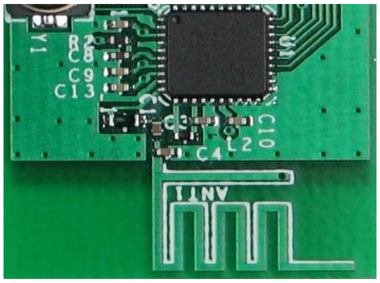
1. **Wire Antenna**: This is a piece of wire extending over the PCB in free space with its length matched to λ/4 over a ground plane. This is generally fed by a 50-Ω[4] transmission line. The wire antenna gives the best performance and RF range because of its dimensions and three-dimensional exposure. The wire can be a straight wire, helix, or loop. This is a three-dimensional (3D) structure, with the antenna over a height of 4-5 mm over the PCB plane, protruding into space.

Figure 4: Wire Antenna

2. **PCB Antenna**: This is a trace drawn on the PCB. This can be a straight trace, inverted F-type trace, meandered trace, circular trace, or a curve with wiggles depending on the antenna type and space constraints. In a PCB antenna, the antenna becomes a two-dimensional (2D) structure in the same plane of the PCB; see Figure 5.

   There are guidelines[5] that must be followed as the 3D antenna exposed in free space is brought to the PCB plane as a 2D PCB trace. A PCB antenna requires more PCB area, has a lower efficiency than the wire antenna, but is cheaper. It has easy manufacturability and has the wireless range acceptable for a BLE application.

Figure 5. PCB Antenna

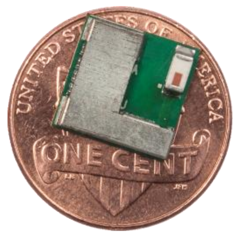


[4] The feed is generally of 50 ohm in most RF PCB catering to low-power wireless applications. However, other impedance values are possible.

[5] Please refer to the section on MIFA and IFA on page 7

3. **Chip Antenna**: This is an antenna in a small form-factor IC that has a conductor packed inside. This is useful when there is limited space to print a PCB antenna or support a 3D wire antenna. Refer to Figure 6 for a Bluetooth module containing a chip antenna. The size of the antenna and the module in comparison with a one-cent is coin is given below.

Figure 6. Cypress EZ BLE Module (10 mm × 10 mm) with Chip Antenna



4 Choosing an Antenna

The selection of an antenna depends on the application, the available board size, cost, RF range, and directivity.

Bluetooth Low energy (BLE) applications such as a wireless mouse requires an RF range of only 10 feet and a data rate of a few kbps. However, for a remote control application with voice recognition, an antenna should have a range around 20 ft in an indoor setup and a data rate of 64 kbps.

For wireless audio applications or indoor positioning, antenna diversity is required. For antenna diversity, two antennas are placed orthogonally on the same PCB such that at least one of them is always receiving some radiation while the other may be shadowed by reflection and multi-path-fading. This is required where real-time audio data is transmitted and a high throughput without packet loss is required.

5 Antenna Parameters

The following section gives some key antenna performance parameters.

- **Return loss**: The return loss of an antenna signifies how well the antenna is matched to the 50-Ω transmission line (TL), shown as a signal feed in Figure 7. The TL impedance is typically 50 Ω, although it could be a different value. The industry standard for commercial antennas and testing equipment is 50-Ω impedance, so it is most convenient to use this value.

  Return loss indicates how much of the incident power is reflected by the antenna due to mismatch (Equation 1). An ideal antenna when perfectly matched will radiate the entire energy without any reflection.

  If the return loss is infinite, the antenna is said to be perfectly matched to the TL, as shown in Figure 7. S_{11} is the negative of return loss expressed in decibels. As a rule of thumb, a return loss ≥ 10 dB (equivalently, S_{11} ≤ –10 dB) is considered sufficient. Table 1 relates the return loss (dB) to the power reflected from the antenna (percent). A return loss of 10 dB signifies that the 90% of the incident power goes into the antenna for radiation.

Equation 1 $$Return\ Loss\ (dB) = 10\log\left(\frac{P_{incident}}{P_{reflected}}\right)$$

Figure 7. Return Loss

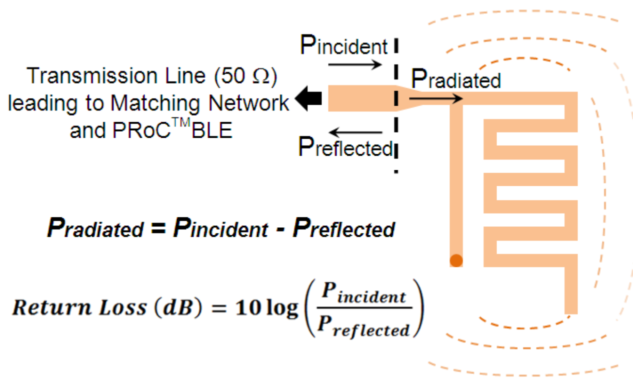
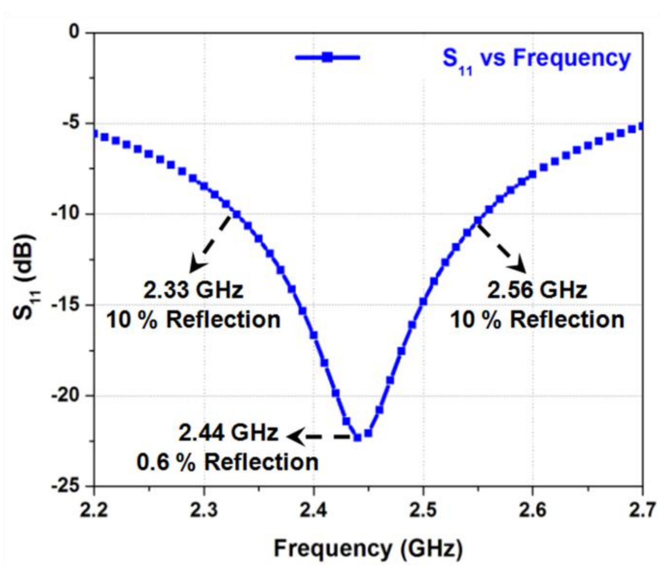


Table 1. Return Loss and Power Reflected from Antenna

S_{11} (dB)	Return Loss (dB)	$P_{reflected}$ / $P_{incident}$ (%)	$P_{radiated}$ / $P_{incident}$ (%)
–20	20	1	99
–10	10	10	90
–3	3	50	50
–1	1	79	21

- **Bandwidth**: Bandwidth indicates the frequency response of an antenna. It signifies how well the antenna is matched to the 50-Ω transmission line over the entire band of interest, that is, between 2.40 GHz and 2.48 GHz for BLE applications.

Figure 8. Bandwidth



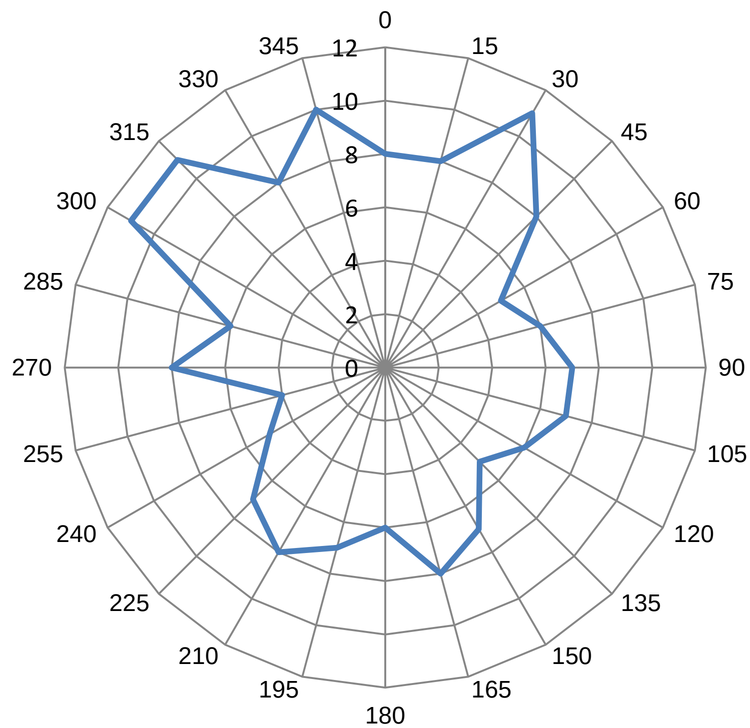
As Figure 8 shows, the return loss is greater than 10 dB from 2.33 GHz to 2.55 GHz. Therefore, the bandwidth of interest is around 200 MHz.

- **Radiation efficiency**: A portion of the non-reflected power (see Figure 7) gets dissipated as heat or as thermal loss in the antenna. Thermal loss is due to the dielectric loss in the FR4 substrate and the conductor loss in the copper trace. This information is characterized as radiation efficiency. A radiation efficiency of 100 percent indicates that all non-reflected power is radiated to free space. For a small-form-factor PCB, the heat loss is minimal.

- **Radiation pattern**: Radiation pattern indicates the directional property of radiation, that is, which directions have more radiation and which have less. This information helps to orient the antenna properly in an application.

An isotropic dipole antenna radiates equally in all directions in the plane perpendicular to the antenna axis. However, most antennas deviate from this ideal behavior. See the radiation pattern of a PCB antenna shown in Figure 9 as an illustration. Each data point represents RF field strength, measured by the received signal strength indicator (RSSI) in the receiver. As expected, the contours are not exactly circle, as the antenna is not isotropic.

Figure 9. Radiation Pattern



- **Gain**: Gain indicates the radiation in the direction of interest compared to the isotropic antenna, which radiates uniformly in all directions. This is expressed in terms of dBi—how strong the radiation field is compared to an ideal isotropic antenna.

6 Antennas for Cypress PRoC/PSoC BLE

One of the product objective for Cypress BLE is to have an antenna design within the tight area that requires no more than two external components for tuning. Tuning is the process that ensures that near-maximum power is sent to the antenna while transmitting over the working band of frequencies. This is ensured by making the return loss in the band of interest greater than 10 dB. When the impedance seen looking into the antenna and the chip output impedance are the same, maximum power is transferred to the antenna; the same rule holds true for receiving too. Antenna tuning ensures that the antenna impedance is matched to 50 Ω looking towards the antenna. Radio tuning ensures that the impedance looks 50 Ω, looking towards the chip, when the chip is in the receive mode.

The integrated balun inside PRoC/PSoC BLE is not exactly 50-Ω impedance and may require two components for tuning. For a low-data-rate and low-RF-range application, the PCB antenna Cypress recommends does not require any component for antenna tuning.

For high-data-rate applications like voice recognition over remote control, at least four components for the matching network are recommended. Two of these will be used for radio tuning and two will be used for antenna tuning. It may be possible to do the tuning with two components if the resulting bandwidth is acceptable. Having an [6]extra component footprint is a wise design choice for future mitigation of [7]EMI radiation in a new product. Filters can be implemented for out-of-band operation using those components.

Cypress PRoC/PSoC devices can also be employed in applications such as indoor positioning, smart home, smart appliances, and sensor hub. Because these applications may not have space constraints, you can employ an antenna with a better RF range and radiation pattern. The wire antenna can be a perfect fit for such an application where the ID (Industrial Design) can have some height to fit a wire.

In some application like wearable ultra small form factor is required. The chip antenna usually takes less space compared to a PCB antenna, The chip antenna is more popular in this application category. Cypress recommends a few guideline for using the ultracompact chip antennas.

There are many applications that directly embed a Cypress module in the host PCB for wireless connectivity. For such applications, a very-low-cost, FCC-passed, tiny module is desired. Cypress has come up with EZ-BLE module for such application. The Cypress EZ-BLE module uses Johansson chip antenna 2450AT18B100E.

Though there are multiple antennas for the 2.4-GHz band, most BLE applications are catered by two Cypress-Proprietary PCB Antennas. Cypress recommends using two proprietary PCB antennas, meandered inverted-F antenna (MIFA) and inverted-F antenna (IFA), which are characterized and simulated extensively for BLE applications. MIFA in particular is useful to most of the applications.

However, you can choose any antenna described in this document to suit your application requirements.

7 Cypress-Proprietary PCB Antennas

Cypress recommends IFA and MIFA types of PCB antennas. The low data rate and typical range requirement in a BLE application make these antennas extremely useful. These antennas are inexpensive and easy to design, because they are a part of the PCB, and provide good performance in the 150-250 MHz bandwidth range.

MIFA is recommended for applications that require a minimum PCB area such as a wireless mouse and presenter. IFA is recommended for applications where one of the antenna dimensions is required to be much shorter than the other such as a heart-rate monitor. Most BLE applications are catered by MIFA antennas.

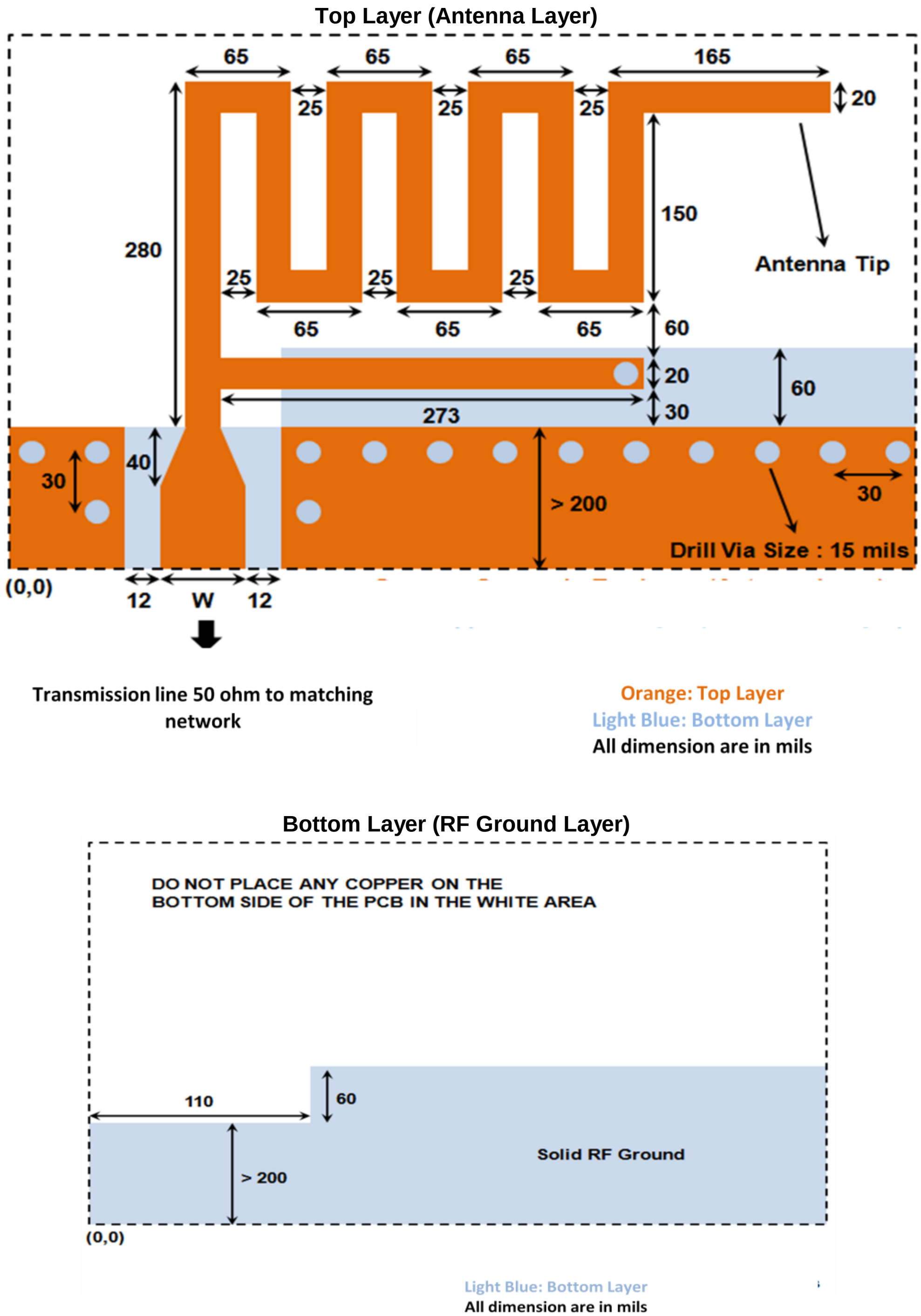
7.1 Meandered Inverted-F Antenna (MIFA)

The MIFA is a popular antenna widely used in human interface devices (HIDs) because it occupies a small PCB area. Cypress has designed a robust MIFA that offers an excellent performance with a small form factor. The antenna size is 7.2 mm × 11.1 mm (284 mils × 437 mils), making it suitable for HID applications such as a wireless mouse, keyboard, or presenter. Figure 10 shows the layout details of the recommended MIFA, both top layer and bottom layer in a two-layer PCB. The antenna trace-width is 20 mils throughout. The main parameter that would change, depending on the PCB stack spacing, is the value of "W," the RF trace (transmission line) width.

[6] Extra components before the antenna is a recommended practice that helps in implementing filters for EMI reduction in future.

[7] EMI is electro-magnetic interference regulation that sets limit for radiated power for public health.

Figure 10. MIFA Layout



Note: The Gerber and *.brd* files of MIFA for a FR4 PCB with 1.6-mm thickness are provided in the *AN91445.zip* file at www.cypress.com/go/AN91445.

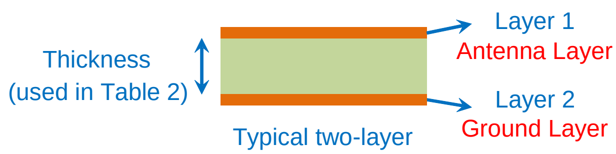
7.2 Antenna Feed Consideration

Table 2 provides the "W" value for different PCB thicknesses between the top and bottom layers for a two-layer FR4 substrate (relative dielectric constant = 4.3). The top layer contains the antenna trace; the bottom layer is the immediate next layer containing the solid RF ground plane. The remaining PCB area of the bottom layer can be used as a signal ground plane (for the PRoC/ PSoC and other circuitry). Figure 11 relates the PCB thickness to "W" for a typical two-layer PCB.[1]

Table 2. Value of "W" for FR4 PCB: Thickness Between Antenna Layer and Adjacent RF Ground Layer

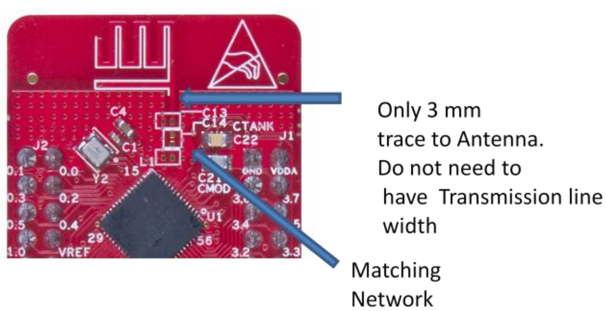
Thickness (mils)	W (mils)
60	65
50	59
40	52
30	44
20	33

Figure 11. Clarification of PCB Thickness



For the small length of PCB trace that feeds the antenna, the width requirement can be relaxed. Ensure that the antenna trace width and the antenna feed connection have the same width. Figure 12 shows one such case where the trace width feeding the antenna is not as wide as recommended in Table 2.

Figure 12. Antenna Feed Width for Short Trace



However, if it is a long transmission line approximately 1 cm from the matching network to antenna or back to the ANT pin of the PRoC/PSoC device, Cypress recommends a transmission line (TLine) type of layout, having a specific width "W" over a bottom ground plane for the feed.

Note: See the coplanar wave guide calculator in Appendix 2 for the calculation of width for Coplanar transmission line.

Figure 13 plots S11 of the MIFA. The MIFA has a bandwidth (S11 ≤ –10 dB) of 230 MHz around 2.44 GHz.

Figure 13. S_{11} of the MIFA (Return Loss = $-S_{11}$)

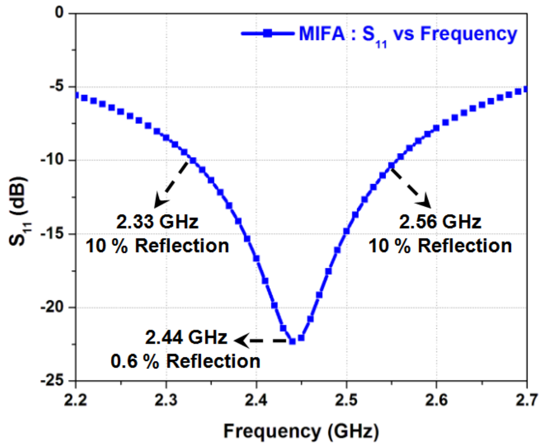
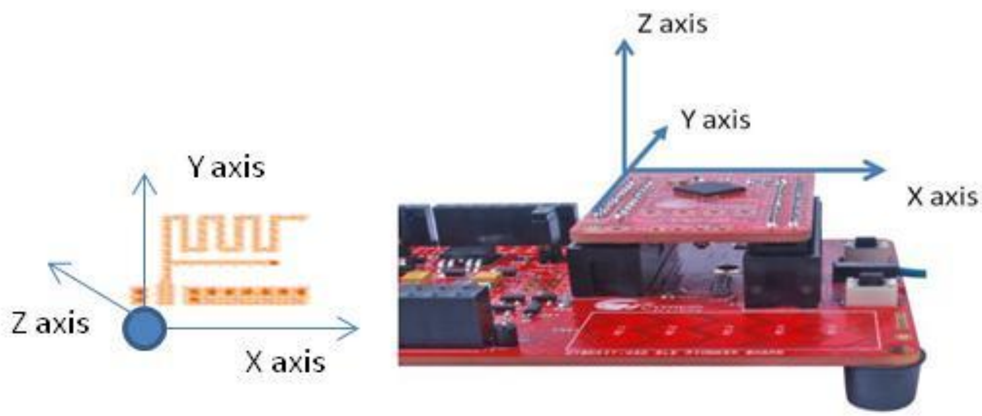
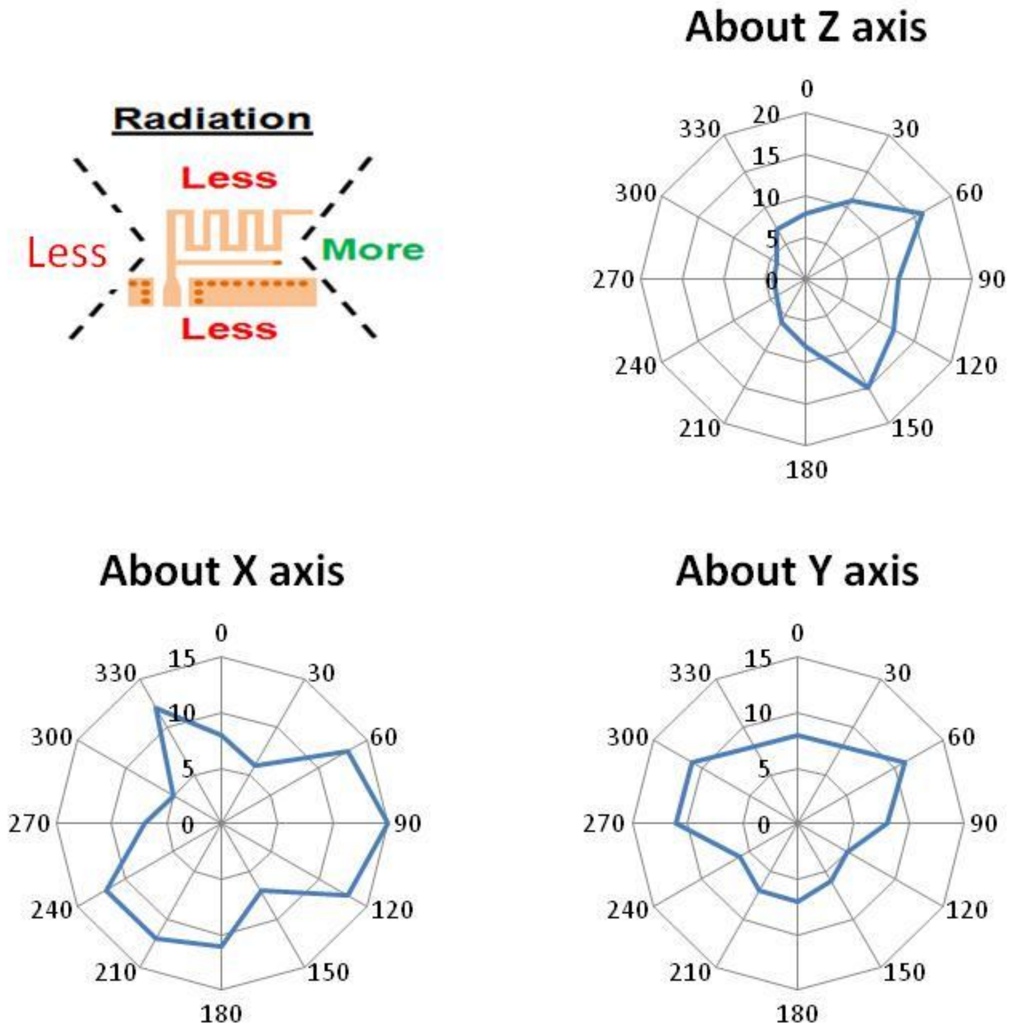


Figure 14 shows the complete 3D radiation-gain pattern of the MIFA at 2.44 GHz. This information is helpful in placing the MIFA for custom applications to maximize the radiation in the desired direction. In this diagram, the antenna is in the XY plane; the Z-axis is vertical to it.

Figure 14. 3D Radiation-Gain Pattern for MIFA





From the radiation patterns, we can infer that the maximum radiation happens across a 30-degree cone around the X-axis. This can be explained as the MIFA is no longer a strictly horizontal or a vertical antenna in the XY plane. Both the vertical legs and the tip contribute to the radiation and result in a slanted radiation pattern.

7.3 Antenna Length Considerations

Depending on the PCB thickness, the MIFA antenna should be length-adjusted to adjust the antenna radiation impedance and frequency selectivity. Cypress recommends the values listed in Table 3 for antenna lengths for various board thicknesses.

Figure 15. Length of MIFA

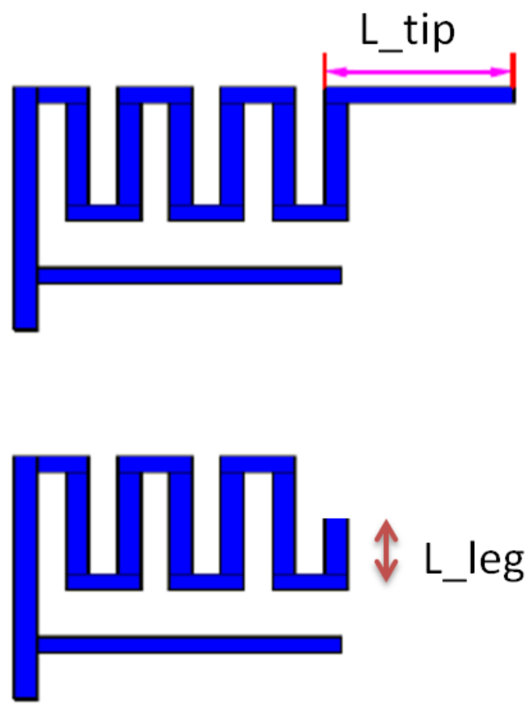


Table 3. Leg and Tip length

PCB Thickness	Antenna L_Tip / L_leg
16 mils	L_tip= 353 Mils
31 mils	L_tip= 165 Mils
47 mils	L_tip= 125 Mils
62 mils	L_leg= 215 Mils

Figure 15 shows two MIFA antennas for two different board thicknesses. Antenna designers should refer to Table 3 for adjusting the length of the MIFA antennas for a specific board thickness.

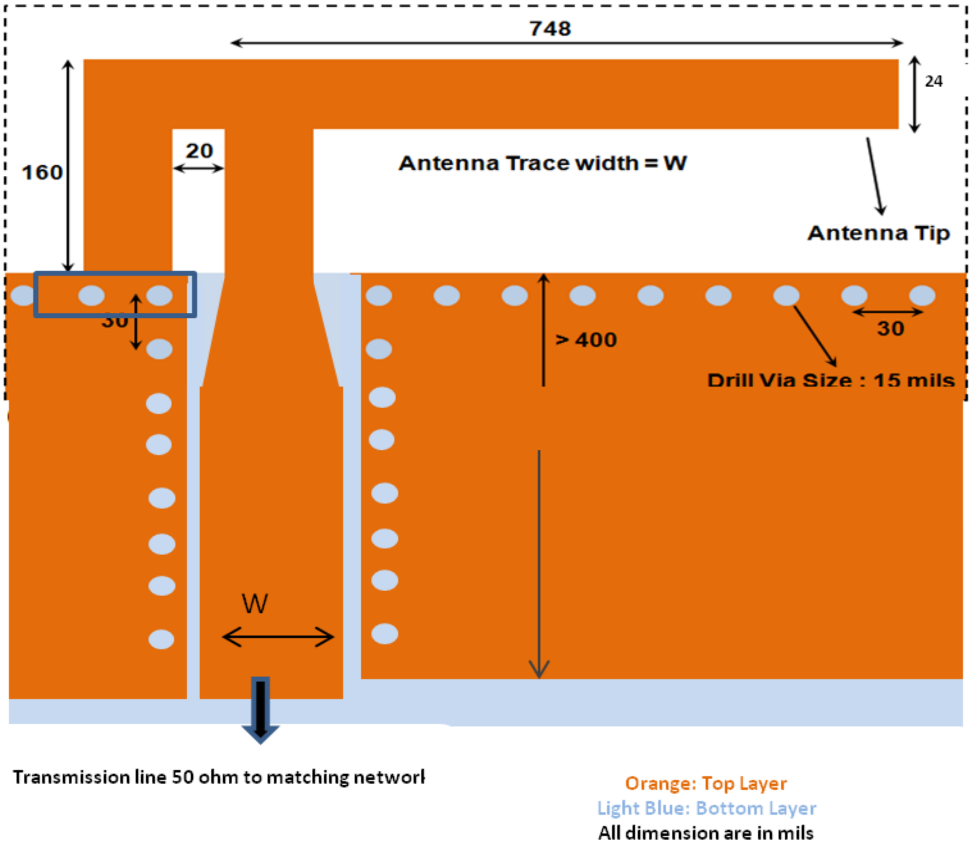
7.4 Inverted-F Antenna (IFA)

The IFA is recommended for applications in which one of the antenna dimensions is constrained, such as in a heart rate monitor. Figure 16 shows the layout details of the recommended IFA, both top layer and bottom layer, in a two-layer PCB. The trace width is 24 mils.

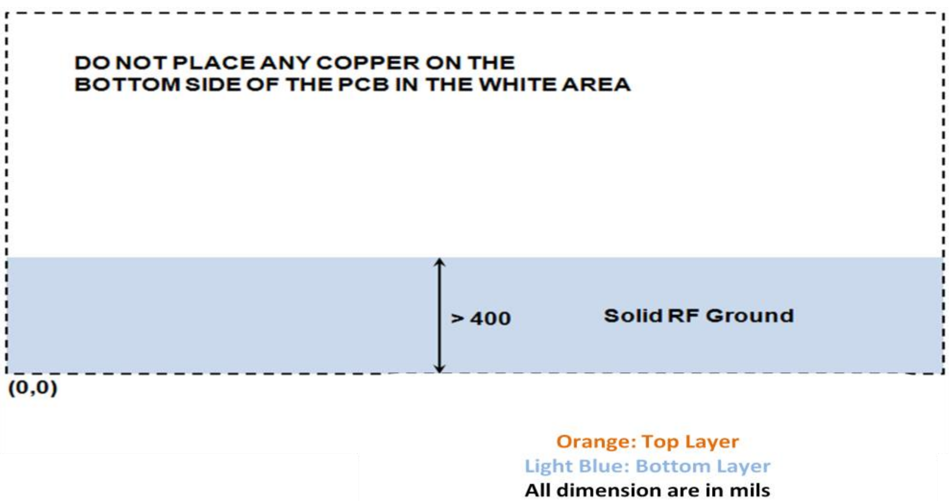
The IFA is designed with a size of 4 mm × 20.5 mm (157.5 mils × 807 mils) for an FR4 PCB with a 1.6-mm thickness. The IFA has a larger aspect ratio (width to height) than the MIFA.

Figure 16. IFA Layout

Top Layer (Antenna Layer)



Bottom Layer (RF Ground Layer)



Note: The Gerber file (as well as the *.brd* file) for an FR4 PCB with 1.6 mm thickness is provided in the *AN91445.zip* file at www.cypress.com/go/AN91445.

As explained for the MIFA antenna, the feed trace width "W" is dependent on the PCB stack of the product. Table 4 provides the "W" value for different PCB thicknesses between the top layer (antenna layer) and bottom layer (adjacent RF ground layer) for an FR4 substrate (relative dielectric constant = 4.3).

Table 4. Value of "W" for FR4 PCB: Thickness Between Antenna Layer and Adjacent RF Ground Layer for 50-ohm Impedance

Thickness (mils)	W (mils)
60	65
50	59
40	52
30	44
20	33

For short traces less than 3 mm, the width of the trace for antenna feed can be relaxed. The antenna feed can be of the same width as the antenna trace; see Figure 12. Please Refer to coplanar wave guide calculator in Appendix 2 for the calculation of width for Coplanar transmission line.

The bandwidth ($S_{11} \leq -10$ dB) of the IFA is 220 MHz around 2.44 GHz, as shown in Figure 17.

Figure 17. S_{11} of the IFA (Return Loss = $-S_{11}$)

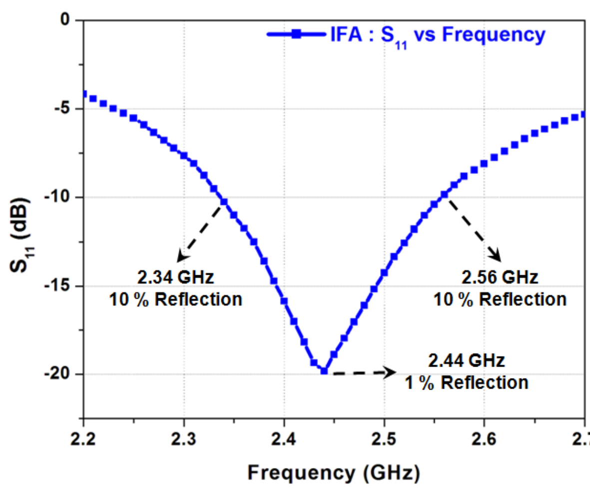
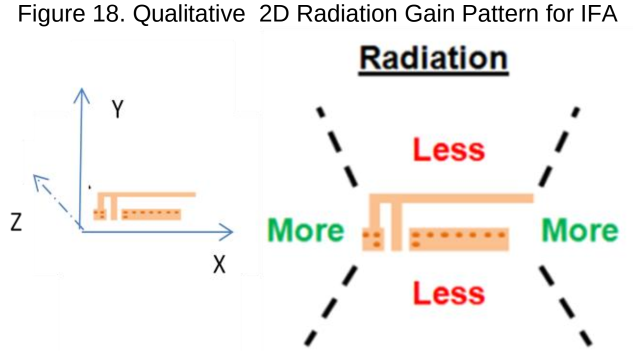


Figure 18 shows the qualitative radiation pattern of an IFA in the XY plane. This information is helpful in placing the IFA suitably for custom applications to maximize the radiation in the desired direction. For the sake of brevity, only a qualitative radiation direction is shown. For detailed radiation patterns in all XY, YZ, and ZX planes, contact Cypress Technical Support.

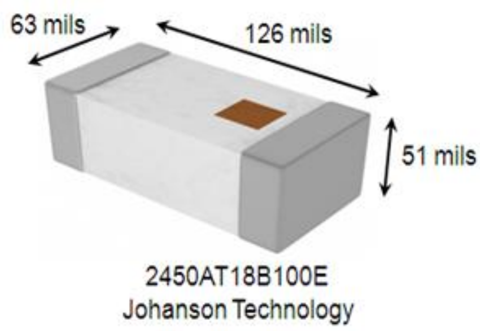
Figure 18. Qualitative 2D Radiation Gain Pattern for IFA



8 Chip Antennas

For applications where the PCB size is extremely small, chip antennas are a good solution (Figure 19). They are off-the-shelf antennas that take up minimal PCB area and offer reasonable performance. However, chip antennas increase the BOM and assembly expense, as they are external components that need to be purchased and assembled. Typically, the price of chip antennas ranges from 10 to 50 cents, depending on the dimension and performance.

Figure 19. Chip Antenna



Another important factor to consider when using chip antennas is that they are sensitive to RF ground size. The manufacturer's recommendations must be followed for ground-size considerations. Unlike PCB antennas, chip antennas cannot be tuned by changing the antenna length. They require an additional matching network for antenna tuning, increasing the BOM expense even further.

Cypress suggests chip antennas only for specialized applications that demand an extremely small PCB area. For such applications, Cypress recommends the Johansson Technology antennas mentioned below.

1. 2450AT18B100E
2. 2450AT42B100E .

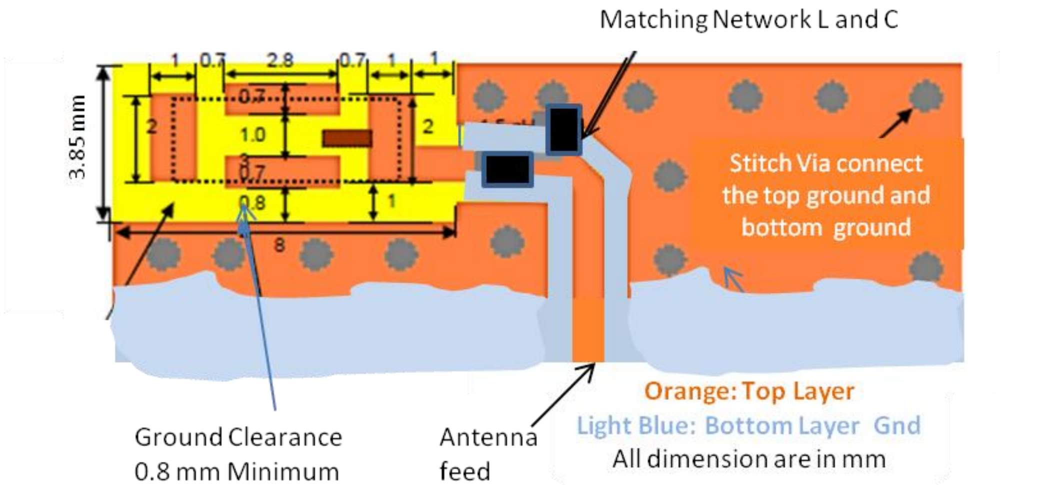
The 2450AT18B100E has dimensions of 63 mils × 126 mils; the 2450AT42B100E has bigger dimensions of 118 mils × 196 mils but provides a better RF performance.

The Cypress BLE module CYBLE-022001-00 uses the 2450AT18B100E antenna and has gone through extensive characterization for RF performance and pre-compliance testing. Both the chip antenna requires a few layout guidelines for an optimal RF performance. The following are the major considerations for a chip antenna placement, layout, and RF performance:

1. Ground clearance around the antenna
2. Antenna placement for optimal radiation
3. Antenna feed consideration
4. Antenna matching network for bandwidth extension

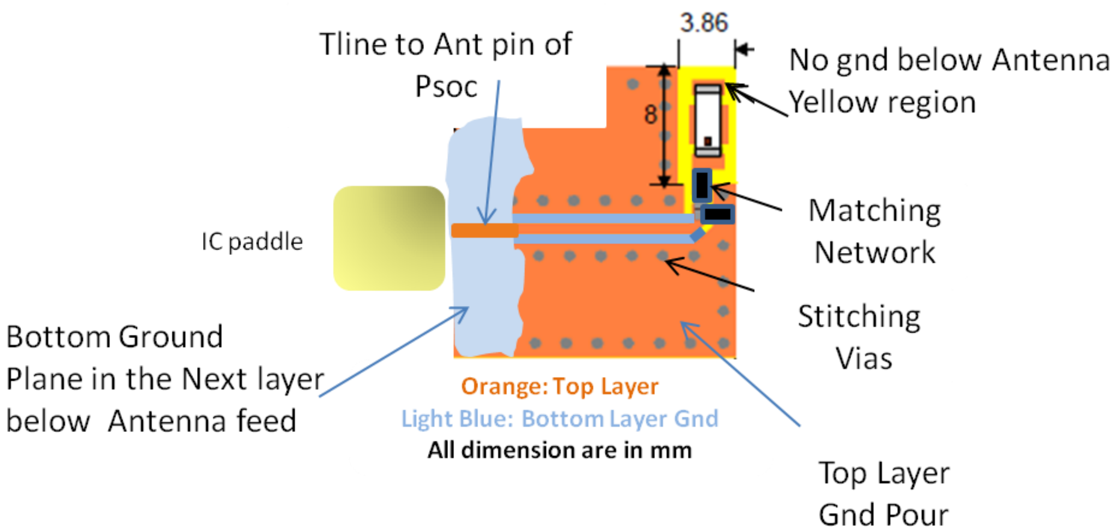
Figure 20 and Figure 21 show the layout guidelines for the chip antenna from Johanson Technology 2450AT42B100E. See their website for detailed guidelines for these antennas.

Figure 20. Layout Guideline for Johanson 2450AT42B100E Chip Antenna



This layout also shows the 50-Ω transmission-line feed and the matching components. The width of the transmission-line feed depends on the board thickness. The exact width is determined from Table 4.

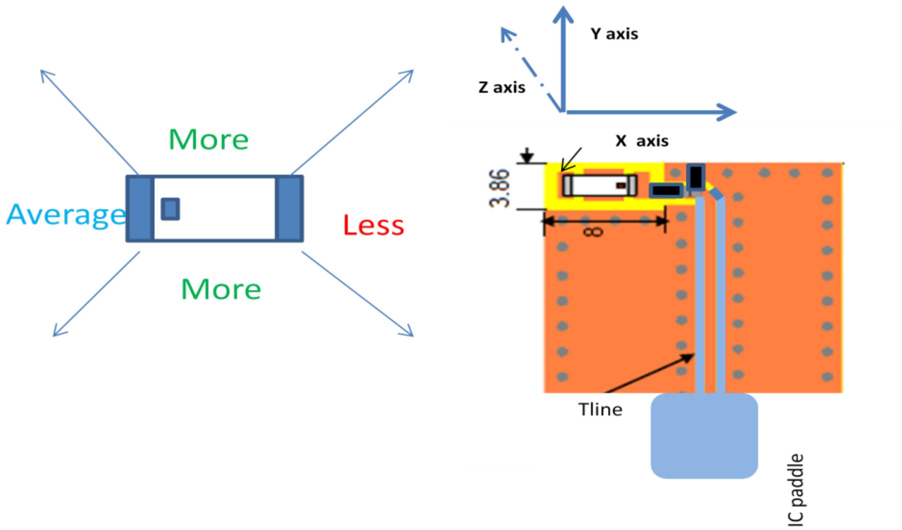
Figure 21. Johanson Antenna Layout Guideline for 24AT42B100E



The chip antenna performance depends on the ground plane. Generally, these antennas require much bigger ground plane and larger spacing. The minimum ground clearance shown in Figure 21 is 0.8 mm from the antenna edge to the ground edge for the 2450AT42B100E part. A better return loss is observed if the clearance is of the order of 2-3 mm.

The chip antenna is not exactly isotropic. There is some preferred direction of radiation. The direction of maximum radiation varies with the ground clearance and plastic assembly. See Figure 22 for the general directivity of the Johanson chip antenna (2450AT42B100E).

Figure 22. Radiation Pattern from Chip Antenna



Cypress suggests chip antennas only for specialized applications that demand an extremely small PCB area such as a nano Bluetooth dongle or an ultra-small module. The Johansson antenna is characterized for RF performance and pre-compliance in Cypress for cypress EZ-BLE module You can use other chip antennas from vendors such as Murata, Vishay, Pulse, and Taoglas.[8]

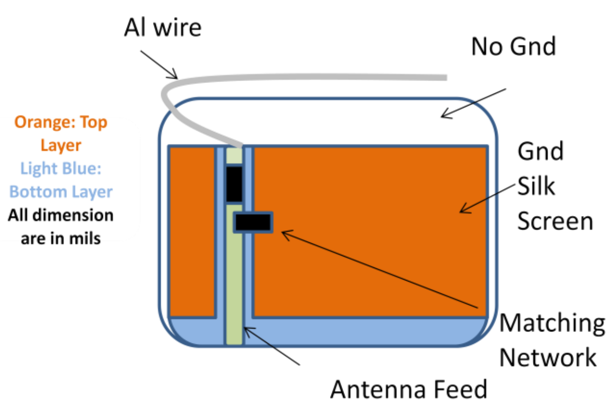
9 Wire Antennas

Wire Antennas are the classical antennas that are conductors of quarter-wave length. They 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 have the best range and have the most isotropic radiation pattern.

For BLE applications requiring a small form factor, they are not preferred as they take a lot of space and vertical height. However, if space is not a constraint, they can be the best antenna to use in terms of RF range, directivity and radiation pattern. In general applications such as a smart home controller that plugs into a wall can use this type of antenna. The wire shape and size need to be optimized for a particular industrial design (ID). The wire can be bent according to the enclosure. Special care should be taken for manufacturing of the wire antenna as they can be of various shapes according to the enclosure.

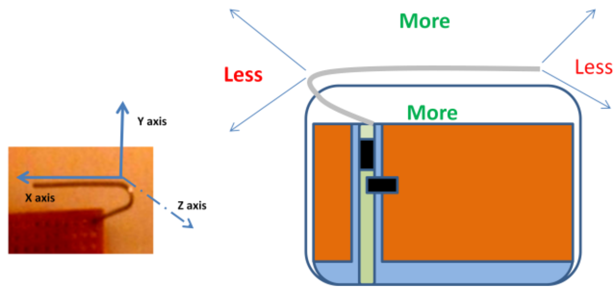
Figure 23. Wire Antenna Layout



8 Only Johansson antenna is characterized; others are not.

A wire antenna is the best in RF performance. They have the best antenna efficiency and directivity compared to other antennas. See Figure 24 for the qualitative radiation pattern out of wire antenna.

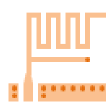
Figure 24. Qualitative Radiation Pattern Out of Wire Antenna



10 Antenna Comparison

Use Table 5 as a quick reference to select the appropriate antenna for your application.

Table 5. Comparison of MIFA, IFA, Chip, and Wire Antennas

Properties at 2.44 GHz	MIFA	IFA	Chip Antenna	Wire Antenna
Appearance				
Recommended Applications	Less Area (Mouse, Keyboard, Presenter)	Height Constrain (Heart Rate Monitor)	Small Area (Nano Dongle, BLE Module)	More Height (6 mm) (3D) (Sensor Hub)
Dimensions (mm)	7.2 × 11.1	4 × 20.5	3.2 × 1.6	6 × 30
Dimensions (mils)	284 × 437	157.5 × 807	126 × 63	250 × 1200
Gerber File	Web	Web	Refer to datasheet	
Cost (US$)	Minimal	Minimal	0.1–0.5	0.1
Bandwidth (MHz) ($S_{11} \leq -10$ dB)	230	220	230	200
Gain (dBi)	1.6	1.1	0.5	2

11 Effect of Enclosure and Ground Plane on Antenna Performance

Antennas used in consumer products are sensitive to PCB RF ground size and the product's plastic casing. The antenna can be modeled as an LC resonator whose resonant frequency decreases when either L (inductance) or C (capacitance) increases. A larger RF ground plane and plastic casing increase the effective capacitance and thus reduce the resonant frequency.

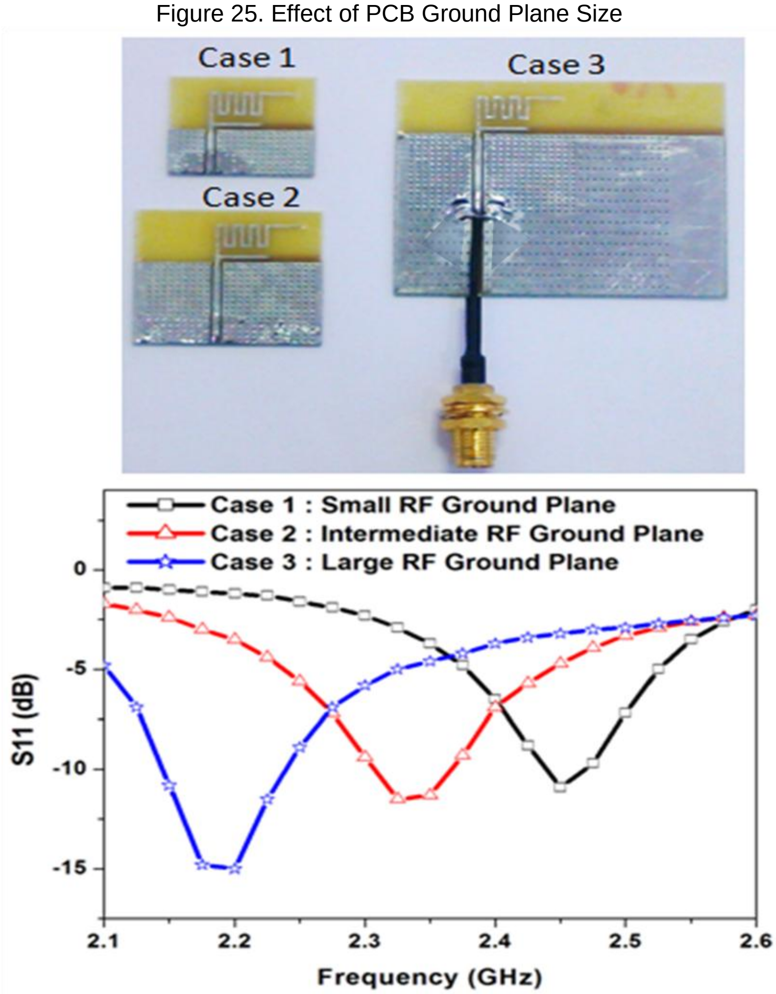
11.1 Effect of Ground Plane

As explained before, a monopole PCB antenna requires a ground plane for proper operation.

Figure 25 shows an example where a MIFA is placed on a PCB with a different ground plane size. The PCB size varies from 20 mm × 20 mm to 50 mm × 50 mm.

The curves show that larger RF ground planes decrease the resonant frequency and better grounding provides better return loss. This is the key for a good PCB layout. The better the ground provided for the quarter-wave antenna, the better it will correlate with the theoretical behavior. This is a key concept in antenna design for small modules where there is hardly enough space for ground clearance.

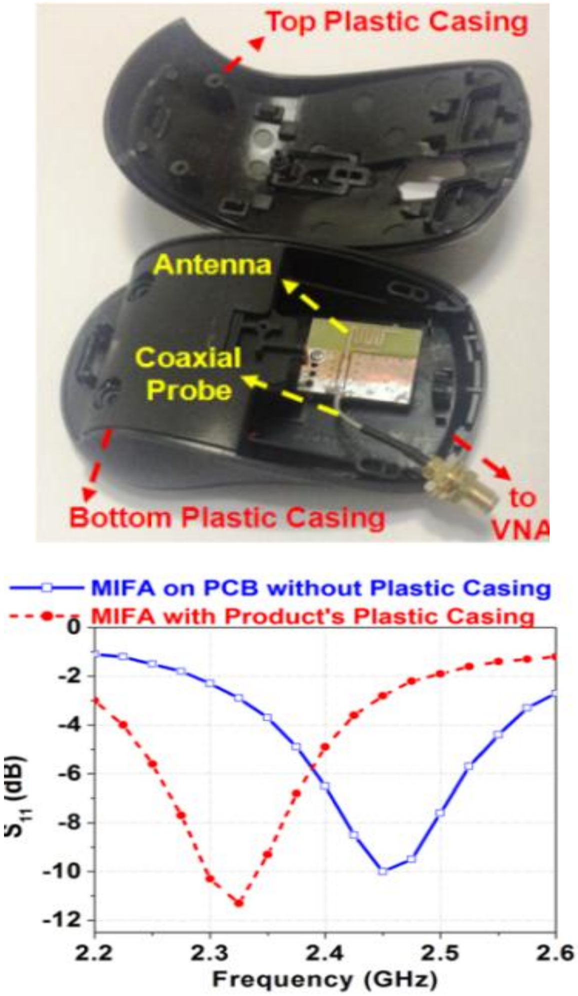
Figure 25. Effect of PCB Ground Plane Size



11.2 Effect of Enclosure

Similar to the effect of the ground plane, to quantify antenna sensitivity to the product's plastic casing, experiments were performed on a wireless mouse as shown in Figure 26. The Cypress MIFA is placed inside the plastic casing of the wireless mouse, and then measurements are made for radiation pattern and return loss.

Figure 26. Effect of Plastic Casing



Both Figure 25 and Figure 26 reveal some important observations:

- The resonant frequency shifts to a lower frequency when the antenna is placed near the plastic casing.
- The shift in resonant frequency is observed to be about 100 MHz to 200 MHz. The antenna must be tuned again to bring it to the desired band. For antenna tuning, see Guidelines for Enclosure and Ground Plane.

In conclusion, increasing the ground plane size and plastic casing tends to decrease the resonant frequency of the antenna by approximately 100 MHz to 200 MHz.

12 Guidelines for Enclosure and Ground Plane

- Ensure that there is no component, mounting screw, or ground plane near the tip of the antenna or length of the antenna.
- The battery cable or mic cable should not cross the antenna trace on the PCB on the same side of the antenna.
- The antenna should not be covered by a metallic enclosure completely. If the product has a metallic casing or a shield, the casing should not cover the antenna. No metal is allowed in the antenna near-field.
- The orientation of the antenna should be in line with the final product orientation so that the radiation is maximized in the desired direction.

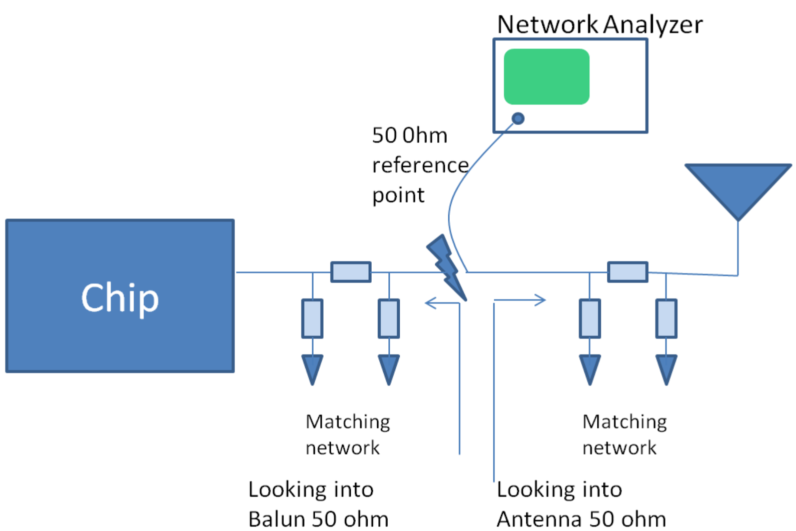
- There should be enough ground: the more the ground, the better is the S_{11} of the MIFA, IFA, chip antenna, and wire antenna.
- There should not be any ground directly below the antenna. See Figure 14. This applies for all antennas.
- There should be enough ground at a distance (ground clearance) from the antenna and this ground plane should have a minimum width. See Figure 10, Figure 15, and Figure 20.

13 Antenna Tuning

Antenna tuning is the process of ensuring that the return loss is greater than 10 dB for the antenna, when looking from the chip output towards antenna, in the desired frequency band. The same tuning procedure should be followed when looking into the radio and making sure that impedance is 50 Ω in the receive mode. A return loss greater than 10db, ensures that 90% of the power output of the chip is transmitted to the antenna. Similarly in receive mode it is ensured that 90% of the received power is provided to Radio. Both antenna tuning and radio tuning are referred to as antenna tuning. Radio tuning is also called balun tuning as the balun is the first component on the receiver that receives the received signal from antenna.

The power transfer is maximized by ensuring the output impedance of the radio is complex conjugate of the antenna impedance. In most antennas tuning this is achieved by transforming the antenna impedance to 50 ohm and balun to 50 ohm, by passive components known as matching network components. This requires that the antenna need to be transferred to 50 ohm and Radio needs to be transferred to 50 ohm, at the reference point separately using network analyzer. Please refer to Appendix B for matching network design reference.

Figure 27. Reference for Tuning and Matching Network



The 50-Ω reference point is connected to the network analyzer port. When tuning the antenna, the chip side is disconnected by removing the balun-matching components. When performing tuning of the radio the antenna-matching components are disconnected. Having a 50 ohm reference point is a convenience as most standard instruments are suited for 50 ohm port impedance.

In Figure 27 even though six components are shown, you can tune the antenna using only two components. The antenna tuned by PCB length design does not need any component. The balun side requires only two components to attain 50-ohm impedance. In most applications using Cypress MIFA the antenna is made 50 ohm by correct length. The balun side uses 2 components at most for getting to 50 ohm in receive mode. For application using chip antenna 2 components are required for the antenna to get to 50 ohm and 2 components are required for the radio or balun to get to 50 ohm.

The following section describes the step-by-step procedure for antenna tuning by using a network analyzer. For antenna tuning we need to look towards the antenna

13.1 Tuning Procedure

As explained in Effect of Enclosure and Ground Plane on Antenna Performance, the effect of enclosure and ground detunes the antenna from the desired band and affects the return loss. Thus, antenna tuning is a two-step process where the bare PCB is tuned for the desired band first, and then in the second phase after the industrial design is finalized, the tuning is checked with the plastic enclosure and human body contact.

A basic familiarity with Smith Chart is required for antenna tuning by Network Analyzer. Without loss of generality the readers are encouraged to read about Smith chart. The antenna tuning is checked with a network analyzer. A network analyzer is an instrument which characterizes the s parameter, such as S11 and S21. The S11 is a indication of return loss and S21 is the forward transmission ratio. Interested readers are encouraged to refer any of the link provided below.

As the first step, the network analyzer is calibrated, and then the antenna is tuned by adjusting the matching network components and verifying the tuning in the Smith chart.

The tuning procedure uses the following:

- Agilent 8714ES network analyzer (calibrated)
- Cypress CY5682 kit mouse as DUT
- A semi-rigid cable with 50 ohm characteristic impedance up to 5GHz
- A high-Q RF component (this example uses Johanson kit P/N: L402DC)

The following major steps are required to tune the antenna:

1. Prepare the ID
2. Set up and Calibrate Network Analyzer
3. Tune the Bare PCB Antenna
4. Adjust Tuning with Plastic and Human Body Contact for Antenna
5. Tune the Radio Side by Putting the Chip in Receive Mode

13.1.1 Prepare the ID

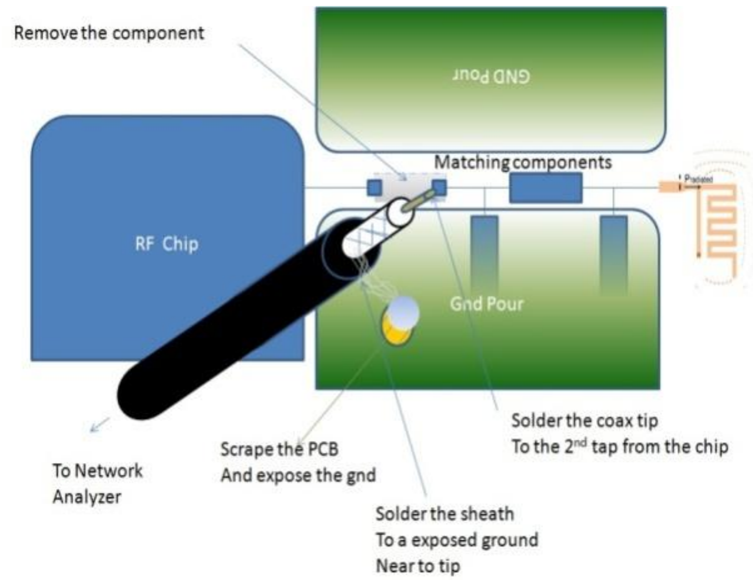
This is a very important step as the placement of the coaxial cable can show variations in S_{11} by up to 3 dB. The ground connection of the coaxial cable shield should be as close to the transmission line return path as possible. The basic steps of ID preparation are given below.

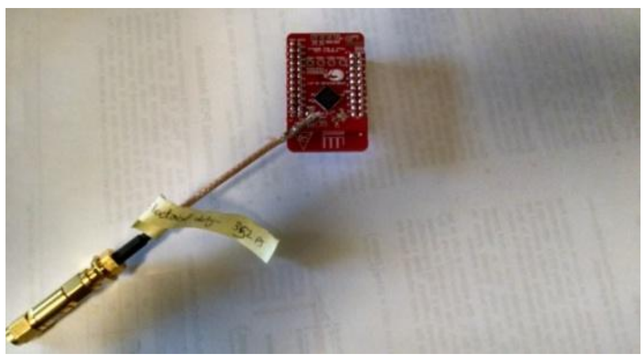
1. Open the plastic casing and remove the batteries or power supplies.
2. Connect the coaxial cable close to the RF out pin from the chip. Remove the connection from the chip. If not, the balun will load the coaxial cable in addition to the antenna. See Figure 28.
3. Ensure that there is an exposed ground near to the tip of the coaxial cable. Connect the sheath or the shield of the cable to ground.

   While connecting the shield/sheath to ground, ensure that it is as short as possible. The shorter the distance, the better the tuning accuracy. There can be 3-dB differences in return-loss measurement depending on where the coaxial cable is connected to ground.
4. Connect a 10-pF capacitor from the first pad going from the 50-Ω reference point to the antenna tip.

   There should always be a capacitor between the coaxial cable and the antenna. This blocks the DC to and from the network analyzer.

Figure 28. Coax Connection Point





13.1.2 Set up and Calibrate Network Analyzer

1. Connect the 3.5-mm calibration kit for calibration and then press the ‘cal’ button on the Agilent 8714ES network analyzer after setting the calibration kit option from the network analyzer to 3.5 mm. You can use any other calibration kit such as a type N calibration kit.
2. Press the frequency button and set the start and stop frequency to 2 GHz and 3 GHz respectively, and then set the format to Smith chart.
3. Press the marker button and set markers to 2.402 GHz, 2.44 GHz and 2.48 GHz.
4. Press the cal button, select S_{11} on the network analyzer. and then set it to ‘user 1 port calibration’.
5. When prompted to connect the ‘open’ load, connect the “Open fixture” to the VNA and press ‘measure standard’.
6. Connect the “Short Fixture” and press measure standard.
7. Connect the “Broadband load” fixture and press ‘measure standard’. After this, the network analyzer will calculate the coefficient and display the 50-Ω load as a point on the Smith chart exactly marked “50,0.”
8. Connect the tuning coaxial cable and set the electrical delay by pressing the ‘scale’ button and setting the electrical delay correctly.

13.1.3 Tune the Bare PCB Antenna

There are two methods to tune the antenna to bring it near 50 ohm.

1. Length adjustment of the antenna if it is a PCB trace or a wire antenna, by cutting off the extra length
2. Use of a matching network (recommended practice)

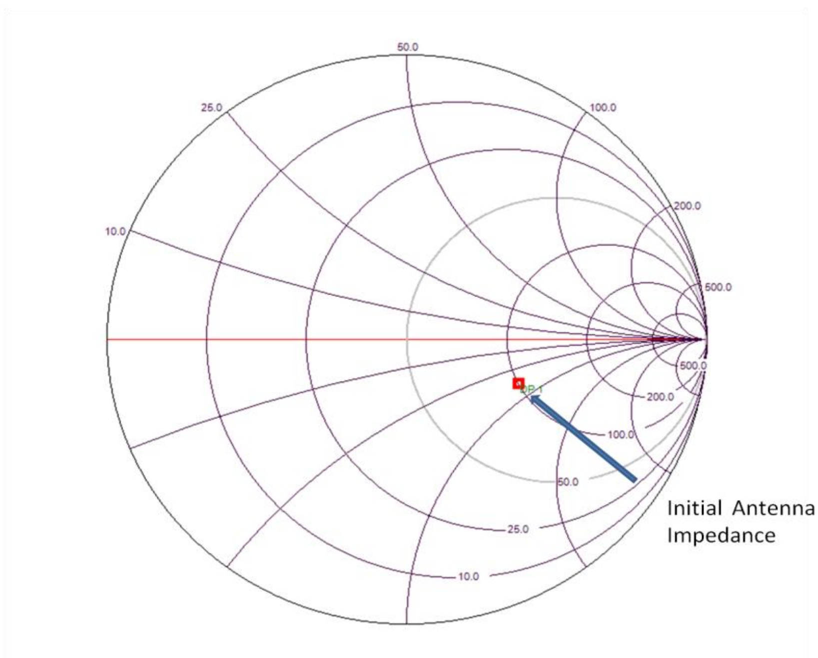
For PCB trace antennas or wire antennas, it is often easier to adjust the length of the PCB trace antenna by scrapping off the extra length at the end of the antenna trace. For this, it is advised to keep the length of the antenna a little longer than the Cypress-recommended length and later cut the length to get the resonance around 2.4 GHz. This is a crude method and does not require any additional components.

However, the matching network method is the most widely used method as it gives the flexibility in future to implement additional filtering for passing EMI/EMC and has a better repeatability. However, the matching network method requires expertise. Contact Cypress Technical support for tuning support for high-volume manufacturing. Use the following procedure to tune the bare PCB using matching network method.

This section below describes the steps required to tune the antenna or the radio using matching network components. The reader is assumed to have some familiarity with Smith Chart.

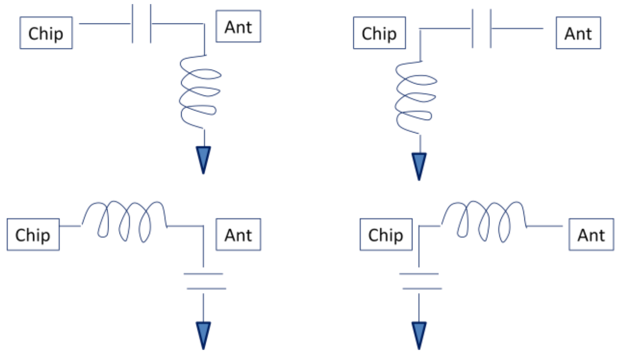
1. Connect an 8.2-pF or 10-pF capacitor in series with the antenna. In the band of interest, it acts as 0 Ω. This gives the antenna impedance. The impedance of antenna is at (100.36 –j34.82), shown as a dot in the Smith chart.

Figure 29: Smith Chart of Antenna Only



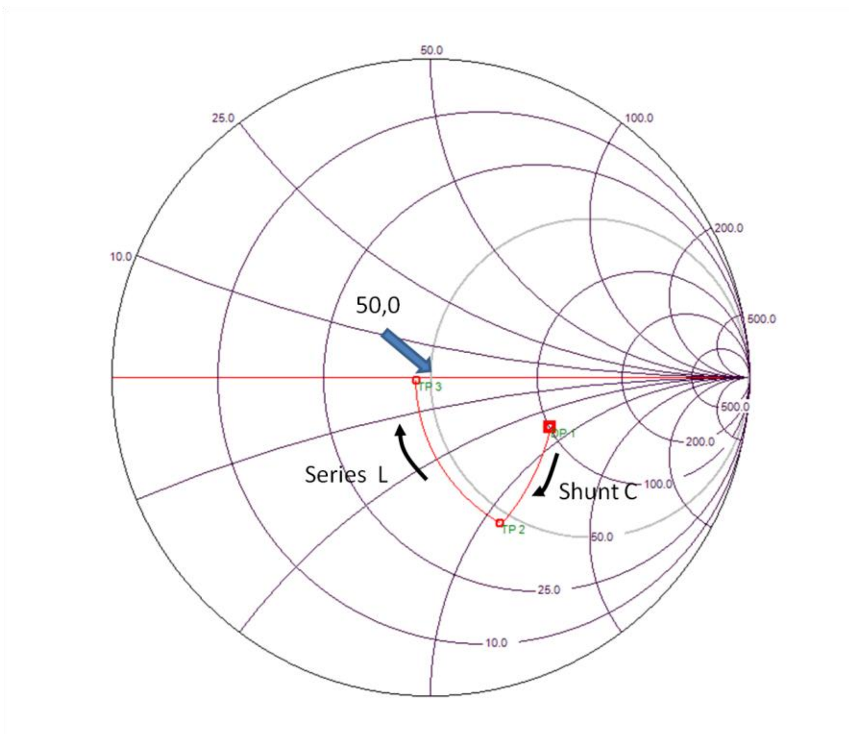
2. After determining the antenna impedance, use L-C components to bring it to 50-Ω impedance by performing an impedance transformation.
3. Impedance transformation networks are networks that transform one impedance to the required impedance without consuming any power. Refer to the impedance transformation property of the L and C resonating networks. Without going to the detail of the matching networks, we can state that most of the matching networks (Figure 30) for cypress MIFA or IFA can be met by two components.

Figure 30. Matching Network



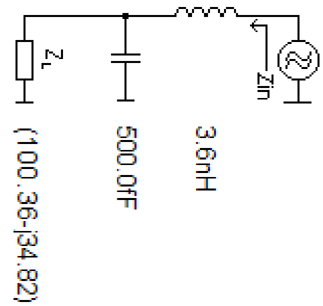
The matching network components can be simulated in a standard open-source tool like Smith V3.10 from Bern Institute. By putting a shunt capacitor of 0.45 pF with the antenna and a series inductor of 3.6 nH, the impedance is transformed to 50 Ω, and the imaginary part is cancelled out mostly in the band of interest. Because the exact values are not available, we chose a 0.5-pF shunt capacitor and a 3.6-nH series inductance.

Figure 31. Moving to 50 Ω in Smith chart



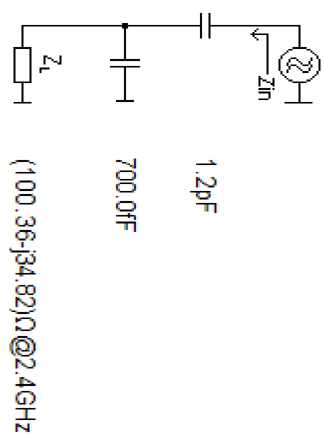
The final schematic of the matching network is shown below. Z_L represents the impedance of the antenna alone when seen through a 0 ohm resistor. Z_{in} is the impedance seen by a network analyzer with a 50-ohm output impedance.

Figure 32. Theoretical Matching Network



The simulation software gives us an idea of what the component values should be. However, the real component values differ significantly from the simulated values. This is because at 2.4 GHz, the lead inductance of the capacitance, the parasitic loading of pads and the ground return path create extra parasitic that completely change the Smith chart. For this application, we have to choose 0.7-pF capacitor and a series 1.2-pF capacitor to attain resonance. This is very common in the 2.4-GHz RF tuning with standard components.

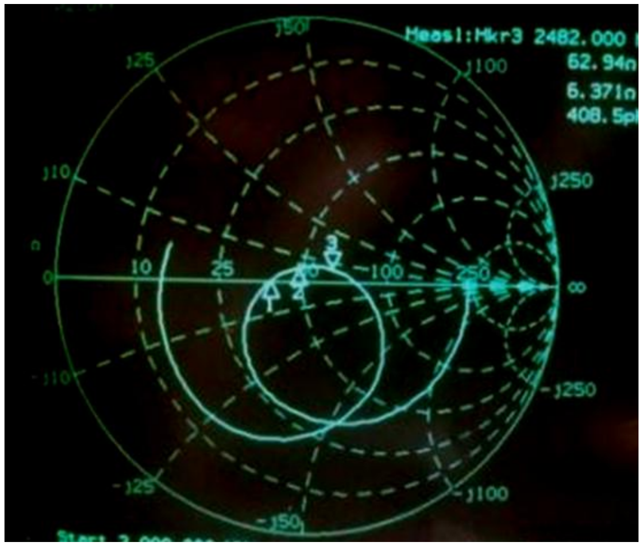
Figure 33. Real Matching Network



An explanation of this behavior follows:

The antenna impedance was seen through an 8.2-pF capacitor that was assumed to be 0 ohm. However, the parasitic of the lead inductance at 2.4 GHz is added to this number. In addition, the ground return was immediate after the antenna. However, with the matching component populated, the ground return path adds an extra parasitic. Therefore, the antenna already sees enough inductance. To tune it, we added some capacitance. This is a classical problem in antenna tuning where the theory and practice differ: you add a capacitance but it results in the Smith chart moving in the direction as if adding an inductance. Figure 34 shows the final Smith chart with the real components.

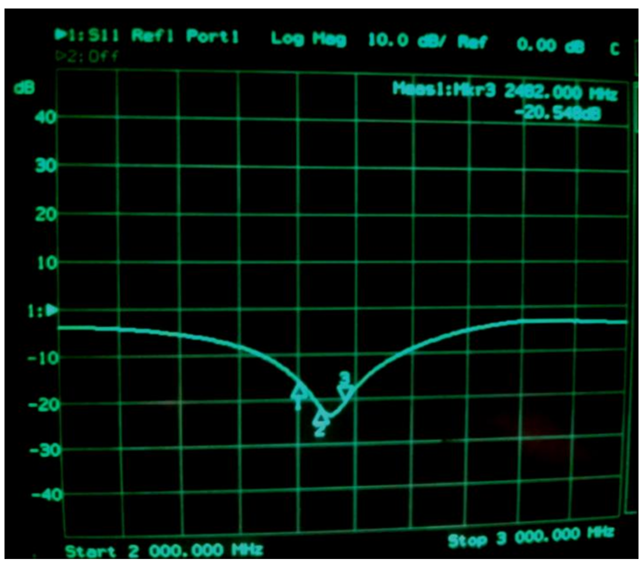
Figure 34. Smith Chart with Real Components



From Figure 34, it is clear that all the marker point 1, 2, 3 representing 2402 MHz, 2440 MHz, and 2480 MHz are close to the (50,0) point on the Smith chart. This shows a good match.

The return loss is plotted for the following component values. A return loss greater than 15 dB is good enough for our application.

Figure 35. Return Loss with Real Components



As seen Figure 35, the return loss is greater than 15 dB for the marker 1, 2 and 3.

13.1.4 Adjust Tuning with Plastic and Human Body Contact for Antenna

A plastic casing on the PCB changes the antenna tuning. Any antenna can be affected because of objects in near field. The near field is the region close to the antenna where the fields have not formed yet. The magnetic field and the electric fields are not orthogonal to each other. It takes up to 4-mm distance from the antenna to have radiated electric and magnetic fields formed properly. After this distance, the far field starts. In the far field region, the electric and magnetic fields are orthogonal to each other. The radiation pattern in a far field region remains same with respect to angular position. Near-field obstructions detune the antenna an may kill the antenna radiation. If it is a narrow-band antenna, there are very high chances of objects in its near field disturbing the antenna.

A plastic casing or a battery cable running nearby can completely detune the antenna and in the band of interest from 2.402 GHz to 2.482 GHz, it can exhibit a return loss less than 10 dB. Therefore, after the bare PCB is tuned, it is essential that the PCB is kept in the plastic casing and checked for the tuning again with a hand on the device. This is cumbersome to do, especially with the coaxial cable coming out of the plastic assembly. The coaxial can be brought out by drilling a small hole in the ID. Finally, the tuning is checked with plastic and also a by placing a hand on the plastic, simulating a user's operation of the device. The effect on return loss was observed to be minimal.

Figure 36. Smith Chart with Plastic Assembly, Illustration of Connecting with ID

13.1.5 Tune the Radio Side by Putting the Chip in Receive Mode

The radio tuning is similar to bare PCB tuning as explained in section 13.1.3. For radio tuning the antenna side is disconnected and network analyzer is connected to the 50 ohm reference point. The chip is powered and put in continuous receive mode. The matching component is adjusted to get a 50 ohm looking into the chip from network Analyzer by the use of Smith Chart.

At the end we have a 50 ohm looking towards the antenna at the reference point and 50 ohm looking into the chip from the reference point. Thus we ensure the maximum power transfer by ensuring the 2 sides are complex conjugate of each other.

14 Summary

This application note described how one can easily design an optimal antenna for a custom product using PRoC BLE/PSoC BLE. The application note also provides guidelines for antenna layout for various types of antennas.

About the Author

Name: Tapan Pattnayak

Title: Sr. Staff Systems Engineer

Background: Tapan received his B. Tech degree in Electrical Engineering from Indian Institute of Technology Kharagpur (IIT Kharagpur) in 2002. Currently, he is working at Cypress Semiconductor Technology, San Jose, USA.

15 Appendix A: Checklist

You can use the checklist in Table 6 while designing the antenna to track your progress.

Table 6. Checklist for Optimal Antenna Design

Check	Step
	Decide on the PCB antenna type based on the application at hand: MIFA, IFA, wire antenna, or chip antenna. See Table 5. Comparison of MIFA, IFA, Chip, and Wire Antenna.
	Note the chosen antenna layout (dimension). Download the Gerber files from www.cypress.com/go/AN91445.
	Orient the antenna suitably for maximum radiation in the desired direction. For MIFA, see Figure 14. 3D Radiation-Gain Pattern for MIFA. For IFA, see Figure 18. Qualitative 2D Radiation Gain Pattern for IFA.
	Determine the "W" value to be used in the antenna layout, based on the PCB thickness (stack). See Table 2 and Table 4.
	Select the antenna tip length or leg length for MIFA, Figure 15.
	Check Ground! This is the Key. . Check the Ground clearance for MIFA, IFA or Chip antenna. Check the bottom layer minimum Ground width for better s11. Please look at the layout pictures.
	Make sure that Antenna feed has a solid Gnd plane below it. Make sure that the RF output of the chip is routed like a Tline.
	Do the ID preparation steps for antenna
	Calibrate the VNA (one-port calibration is sufficient).
	Measure S_{11} (dB) with the complete product casing present. See Figure 35. Return Loss with Real Components.
	Tune by matching network S_{11} (dip) shifts to the desired 2.44 GHz with the bare PCB and with complete product casing present. See Figure 36. Smith Chart with Plastic Assembly.
	Note the final matching network components of the antenna and use them for volume production.

16 Appendix B: References

The following references provide further detailed information.

16.1.1 Antenna Basics

- Constantine A. Balanis, *Antenna Theory: Analysis and Design*, 3rd edition. Wiley - Interscience, 2005 (Chapters 2 and 5).
- Antenna with multiple fold, Philip Pak-Lin Kwan, Paul Beard, US Patent 7936318 B2
- AN48610, Cypress Semiconductor, Design and layout guideline for matching network and antenna for wireless USB

16.1.2 Smith Chart Basics

- David M. Pozar, “Microwave Engineering,” 4th edition, Wiley, 2011 (Chapters 2, 4, and 5).
- Christopher Bowick, John Blyler, Cheryl Ajluni, “RF Circuit Design,” 2nd edition, Newnes, 2007 (Chapter 4).
- Smith v3.10, Bern Institute

16.1.3 Useful Free Online Software

- Transmission line calculator: Grounded CPW (air gap = 12 mil, ε_r = 4.3 for FR4):
  www1.sphere.ne.jp/i-lab/ilab/tool/cpw_g_e.htm
- Smith Chart based matching: L or Pi matching:
  http://cgi.www.telestrian.co.uk/cgi-bin/www.telestrian.co.uk/smiths.pl
- Smith Chart Bern Institute
  http://www.fritz.dellsperger.net/

16.1.4 Chip Antenna Layout

http://www.johansontechnology.com/datasheets/antennas/2450AT42B100.pdf

Document History

Document Title: AN91445 – Antenna Design Guide

Document Number: 001-91445

Revision	ECN	Orig. of Change	Submission Date	Description of Change
**	4468573	GOWB	08/07/2014	New Spec
*A	4565905	TAPI	11/10/2014	Updated all figures and sections. Corrected sections. De-prioritized length cutting. Added Chip antenna layout guideline
*B	4768767	TAPI	06/18/2015	Module characterization results with chip antenna referred. Added the following sections: Chip antenna layout, wire antenna layout, antenna length cutting for a quick churn, description about far field and near field Edits throughout the document Sunset review Updated template

Worldwide Sales and Design Support

Cypress maintains a worldwide network of offices, solution centers, manufacturer's representatives, and distributors. To find the office closest to you, visit us at Cypress Locations.

Products

Automotive	cypress.com/go/automotive
Clocks & Buffers	cypress.com/go/clocks
Interface	cypress.com/go/interface
Lighting & Power Control	cypress.com/go/powerpsoc cypress.com/go/plc
Memory	cypress.com/go/memory
PSoC	cypress.com/go/psoc
Touch Sensing	cypress.com/go/touch
USB Controllers	cypress.com/go/usb
Wireless/RF	cypress.com/go/wireless

PSoC® Solutions

psoc.cypress.com/solutions

PSoC 1 | PSoC 3 | PSoC 4 | PSoC 5LP

Cypress Developer Community

Community | Forums | Blogs | Video | Training

Technical Support

cypress.com/go/support

Cypress Semiconductor
198 Champion Court
San Jose, CA 95134-1709

Phone : 408-943-2600
Fax : 408-943-4730
Website : www.cypress.com