Design and Layout Guidelines for Matching Network and Antenna for WirelessUSB™ LP Family

Author: Mahesh Kiwalkar

Associated Project: No

Associated Part Family: CYRF6936, CYRF6986

Software Version: None

Related Application Notes: None

Abstract

AN48610 provides design and layout guidelines for the matching network and antenna recommended for the WirelessUSB™ LP/LPstar radio. Follow these suggestions to minimize time and expenses when developing your own integrated wireless solution.

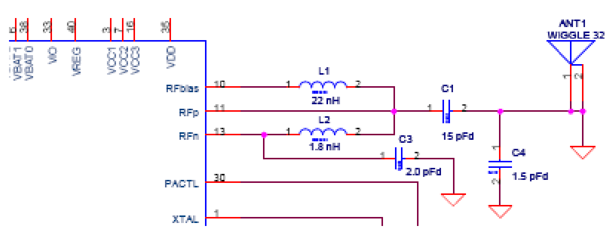
Contents

Introduction

This application note describes the matching network and antenna design recommended for the 2.4-GHz WirelessUSB LP/LPstar radio. A properly designed PCB facilitates the evaluation, characterization, and production test correlation of the WirelessUSB LP/LPstar radio system on-chip solution. These suggestions are tested and proven by Cypress to ensure optimal radio performance when combining RF analog circuitry with other low frequency analog and digital board components. This application note provides design details for the matching network, impedance measurements, and layout suggestions. The antenna design and layout suggestions and the RF performance results are also discussed.

Matching Network

Figure 1. Matching Network Schematic for CYRF6936



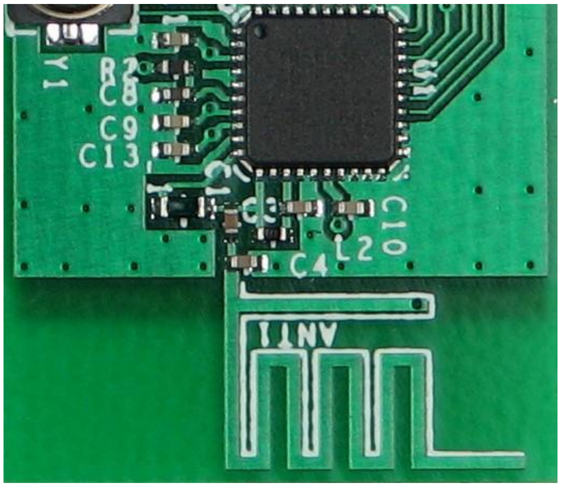
As shown in Figure 1, RFp and RFn are the differential output impedance pins on the WirelessUSB LP radio. A component matching network connects these LP radio pins to the antenna. Because the RFp and RFn pins connect to the antenna, it is important that the matching network transforms the impedances at the RFp and RFn pins to match the input impedance of the antenna. This increases the transmission and reception range.

For the CYRF6986, please refer to the application example in the datasheet or the reference design.

The primary functions of this matching network are listed.

- **50 ohm Match**: Efficiently matches radio chip output impedance to the antenna input impedance. This provides efficient TX power output to the antenna and acceptable RX sensitivity.
- **Balun**: The matching network acts as the balun, transforming the balanced radio chip output to the unbalanced antenna.
- **DC Blocking**: Blocks DC from reaching the antenna output.
- **Harmonic Suppression**: Rejects harmonics and out of band emissions to meet regulatory compliance testing.

Figure 2. Matching Network Layout on CY3630M Module

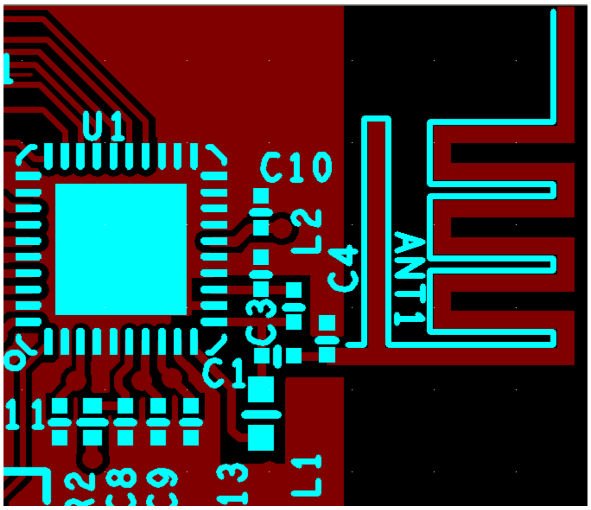


As shown in Figure 2 and Figure 3, the matching components used on the reference design PCB are C1, C3, C4, L1, and L2.

- C1 provides DC blocking.
- C3 provides impedance matching.
- C4 provides harmonic notching.
- L1 is the RF choke, self-resonant at 2.45 GHz.
- L2 provides power combining with 180 degree phase phase shifting and impedance matching.

The antenna matching network element values are achieved by optimization simulations. All the matching components, except L1, are made from the same package size to mount on 0402 pads. The placement of these components and the dimensions of the transmission lines used in this network play a significant role in its impedance. The suggested placement of these elements is shown in the following figure.

Figure 3. Matching Network Layout



The single ended impedance data measured at the RFp and RFn pins of the WirelessUSB LP/LPstar for both Transmit and Receive modes chip are shown in Table 1 and Table 2. The differential impedance measured is shown in Table 3.

The matching network on the Cypress reference design boards is designed such that the impedance looking in at the junction of C1 and C4 is approximately matched to 50 ohms at 2.4 GHz. This measurement can be made on a network analyzer using a SMA connector at the junction of C1 and C4. This also permits using an external antenna, when its input impedance is matched to 50 ohms.

Cypress offers the following suggestions based on the matching network layout to optimize RF board performance:

- Limit the number of signal vias in the matching network path, because they add unaccounted inductance to the circuit.
- In contrast, use a large number of vias to tie the front and backside ground plane regions together, especially along the antenna trace.

- Do not place the crystal under the matching network – antenna section. This could contribute to unnecessary sideband noise.
- Ensure that there are no isolated GND islands for the components that connect to ground.
- Orient the chip in the layout such that the RF input/output pins are closest to the antenna. Running longer traces affect the impedance of the network.
- Use the shortest path traces between components in the matching network.
- Sharp bends in traces must be avoided. If the component placement necessitates a bend, two 45 degree bends are better than a 90 degree bend.

Table 1. Single Ended Impedance RX Mode

Port Output	Impedance (Ohms)
RFp	7.68 - j6.87
RFn	10.2 - j6.9

Table 2. Single Ended Impedance TX Mode

Port Output	PA Setting	Impedance (Ohms)
RFp	PA7	13.20 – j4.75
RFp	PA6	13.30 – j5.33
RFp	PA5	13.60 – j5.60
RFp	PA4	13.80 – j6.35
RFp	PA3	13.80 – j7.12
RFp	PA2	13.80 – j8.35
RFp	PA1	12.80 – j9.72
RFp	PA0	12.10 – j10.6
RFn	PA7	13.90 + j6.50
RFn	PA6	13.60 + j2.00
RFn	PA5	14.70 – j0.38
RFn	PA4	15.47 – j2.09
RFn	PA3	15.80 – j3.28
RFn	PA2	15.80 – j4.89
RFn	PA1	15.50 – j6.60
RFn	PA0	14.94 – j7.75

Table 3. Differential Impedance RX Mode

Port Output	Impedance (Ohms)
RFp / RFn	8 – j64

Table 4. Differential Impedance TX Mode

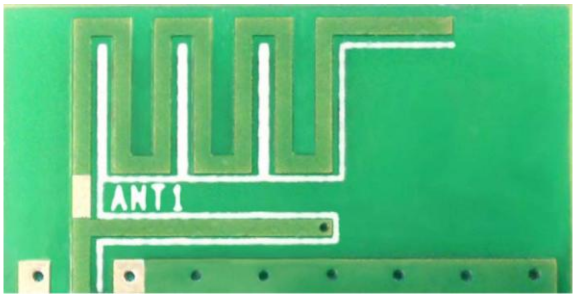
Port Output	PA setting	Impedance (Ohms)
RFp/RFn	PA7	31 – j26
RFp/RFn	PA6	27 – j39
RFp/RFn	PA5	28 – j48
RFp/RFn	PA4	26 – j56
RFp/RFn	PA3	25 – j60
RFp/RFn	PA2	21 – j65
RFp/RFn	PA1	18 – j68
RFp/RFn	PA0	15 – j71

Antenna

In designing short-range radio data communication systems, the system designer faces one of the most important tasks, which is the antenna design. The key parameters for antenna design are the antenna size, cost of implementation, radiation effectiveness, ease of manufacturability, and range performance. A properly designed antenna facilitates the evaluation, characterization, and production test correlation of the WirelessUSB LP/LPstar radio.

The primary functions of an antenna are to provide the transfer of electromagnetic energy to and from the atmosphere, and match the impedance of the transmission line feed (typically 50 ohms) and the impedance of free space (377 ohms). The selection of the antenna for a WirelessUSB solution can have a big impact on wireless communication system performance, system form factor, and cost.

Figure 4. PCB Wiggle Antenna



An antenna essentially provides a means of converting electrical energy into electromagnetic waves for transmission and reconverting the electromagnetic waves into electrical energy for reception. There are several properties of the antenna that affect the performance of wireless communication systems using the Cypress WirelessUSB system radio chip.

This application note describes design considerations and implementation guidelines pertaining to a printed trace wiggle antenna for incorporating the WirelessUSB radio system chip into product applications in the ISM frequency band
2.4–2.5 GHz. These suggestions are tested and proven by Cypress Semiconductor to ensure optimal radio performance when combining RF analog circuitry with other low frequency analog and digital board components.

The radio module printed circuit board is implemented on a two-layer board using low cost FR-4 material. The picture of the wiggle antenna as implemented on Cypress reference radio module is shown in Figure 4.

The antenna is implemented as a wiggle PCB trace on the top component side of the PCB. The ground plane underneath the wiggle trace (along the entire length of the antenna) must be removed from the backside of the PCB. The suggested antenna design requires no more than 435 X 280 mils of space. The antenna design is shown in Figure 7.

Antenna Tip Length versus Board Thickness

The recommended length of the tip of the antenna varies with the board thickness. Note that the tip length of 165 mils shown in Figure 7 is for a board thickness of 31 mils. The length dimension shown in Figure 5 and Table 5 work best for different board thickness.

Figure 5. Antenna Tip Length

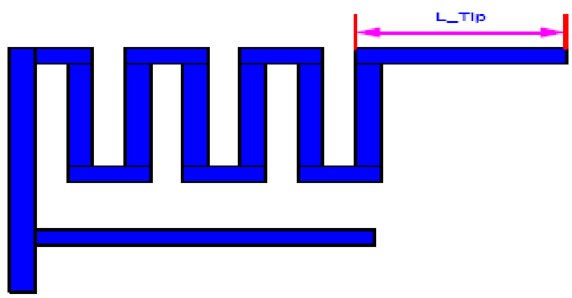


Table 5. Antenna Tip Length versus PCB Thickness

Antenna Tip (L_Tip)	PCB Thickness
L_Tip = 353 mils	16 mils
L_Tip = 165 mils	31 mils
L_Tip = 125 mils	47 mils
L_Tip = 45 mils	62 mils

Layout Recommendations

Cypress offers the following suggestions based on the antenna layout to optimize RF performance:

- Do not place components, mounting screws, or ground plane near the tip of the antenna, because the coupling of the antenna radiation to ground reduces the effective range of operation.
- In case of multilayer board, there must not be any GND plane near the tip of the antenna on any of the layers.
- The horizontal stub which runs to GND with the GND via is instrumental in increasing the return loss of the antenna.
- The antenna tip length may need to be compensated to optimize the antenna design for different board thickness.
- Avoid in-circuit test pins on RF nodes, because they create small antennas and may result in possible FCC issues and degrade RF performance.
- Large, continuous ground plane surfaces provide better radiation performance than small surfaces.

Some practical design guidelines for the antenna choice, selection, and implementation follow:

- It is disadvantageous to use any form of EMI/RFI shield coatings on plastic housing to solve EMI problems, without considering the effect of shielding on antenna placement and location.
- Eliminate connectors and interconnect transmission lines to avoid insertion loses on transmit power and receive sensitivity on the receiver.
- Product applications using keypads, LCD or other types of displays, battery packs, and other metallic surfaces affect and degrade the symmetry of the radiation pattern, reflections, and multipath. As a result, the location of the antenna placement is critical. Place the antenna for best balance of the distribution of these objects.
- If you are using an external antenna and connecting the antenna with a coaxial cable assembly, the cable routing needs to be designed in such a manner to keep it away from motors and battery packs.
- The orientation of the device and the product usage model during the operation must be considered in mounting the antenna inside the device.

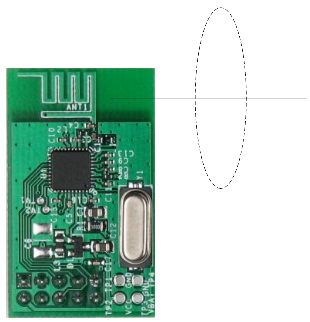
- Note that the performance of the antenna is dependent on its immediate surroundings, packaging, and proximity to the ground plane. The placement of antenna position must be identified early in the design process.
- The effects of human body and the operator's hand must be examined and validated away during the product operation. By keeping the antenna away, the Specific Absorption Rate (SAR) is reduced and pattern symmetry is improved.

Antenna Test Results

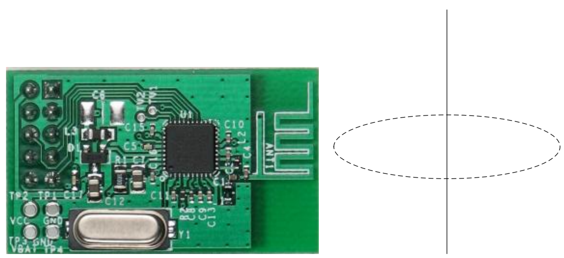
Polar plots for the radiation pattern (Y and Z axes) measured on the PCB wiggle antenna are shown in Figure 8 to Figure 15. Measurements are taken at 2440 MHz. The orientation of wiggle antenna with respect to the different axes is shown in Figure 6.

Figure 6. Antenna Orientation: Radiation Pattern

X Axis Measurement



Y Axis Measurement



Z Axis Measurement

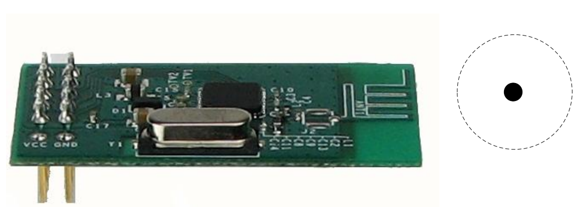


Figure 7. Wiggle Antenna Layout Information

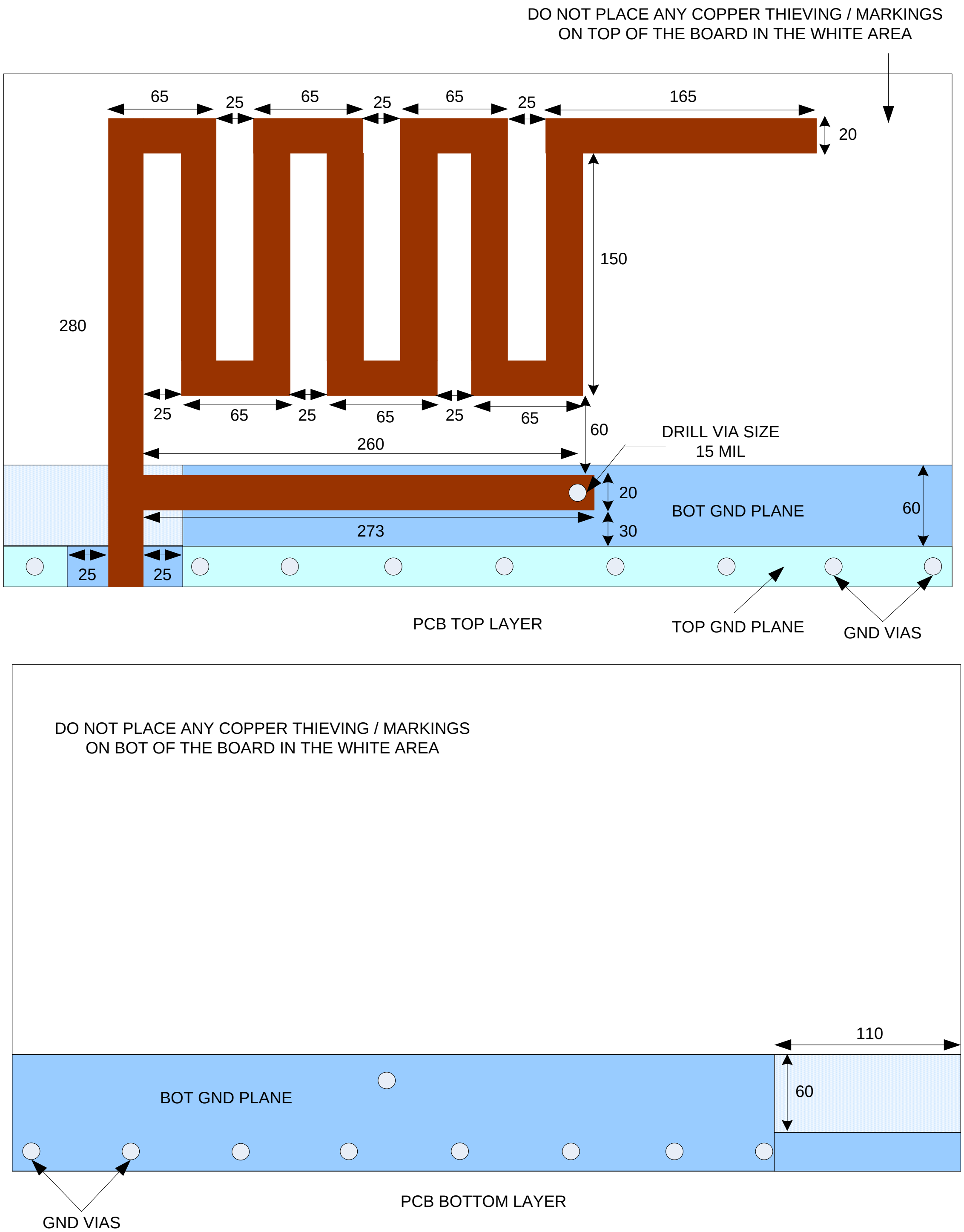


Figure 8. Wiggle Antenna Polar Plot (Relative Gain) for Z Axis with Measurement Antenna Vertical

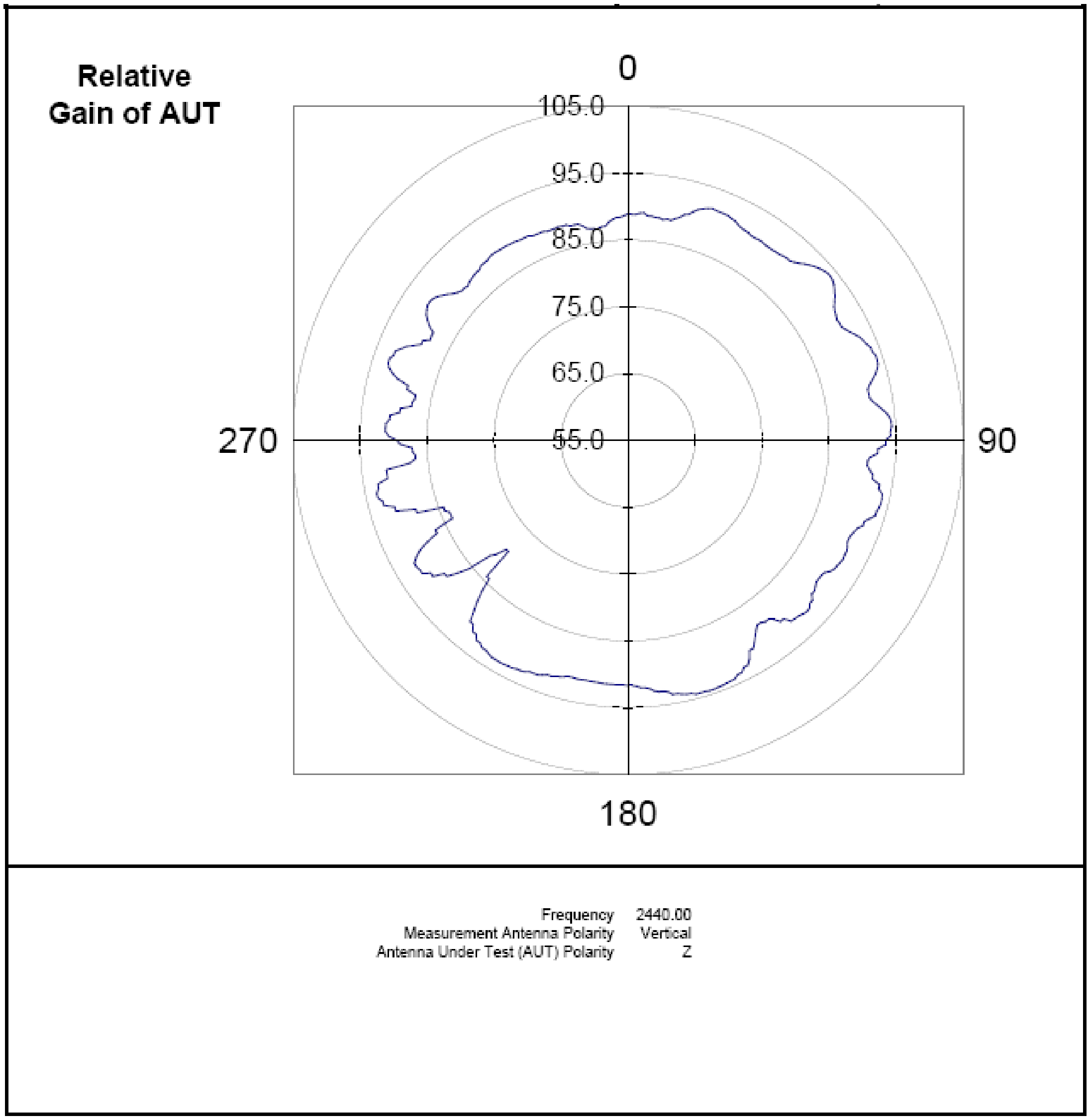
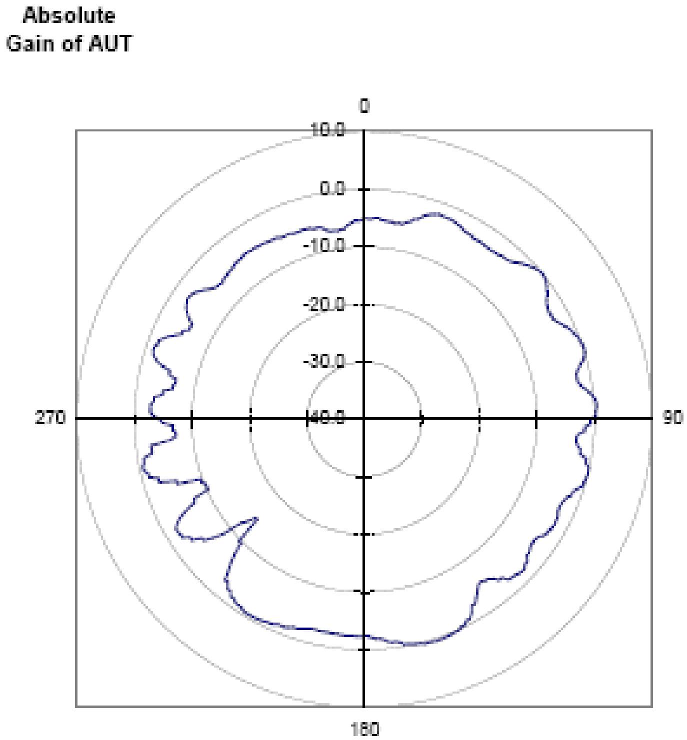
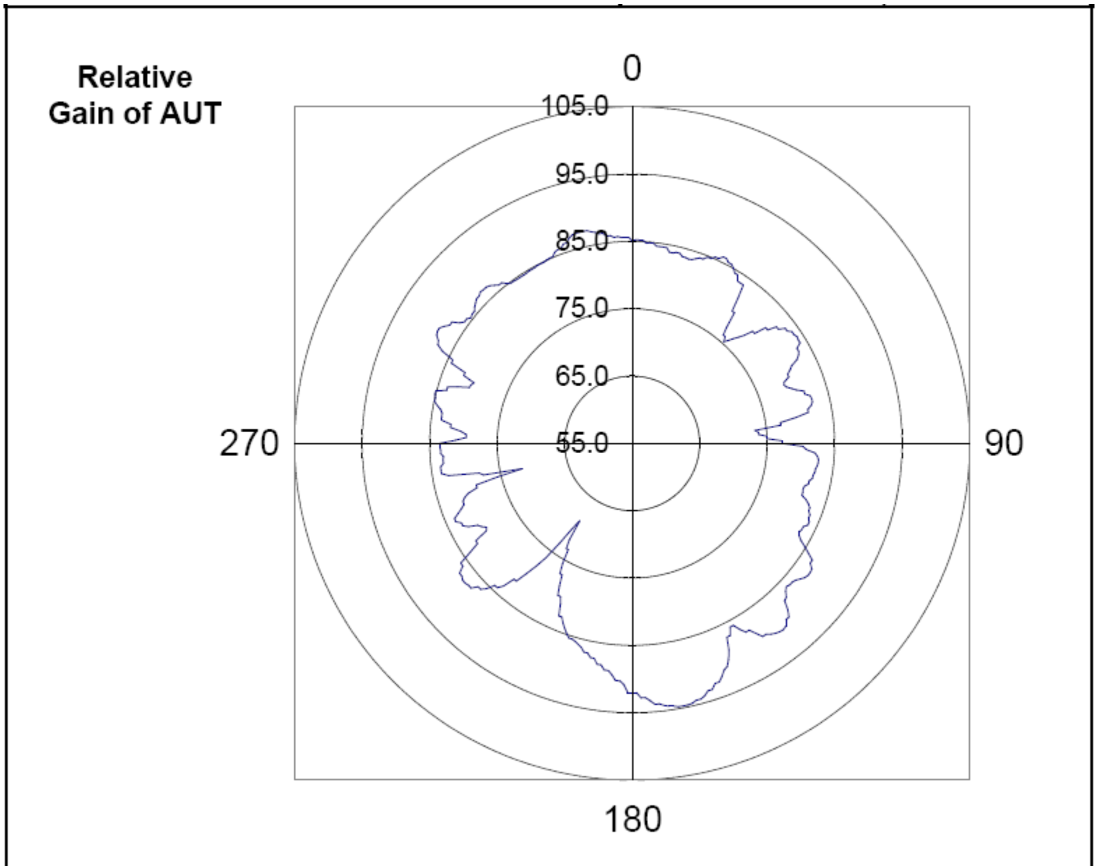


Figure 9. Wiggle Antenna Polar Plot (Absolute Gain) for Z Axis with Measurement Antenna Vertical



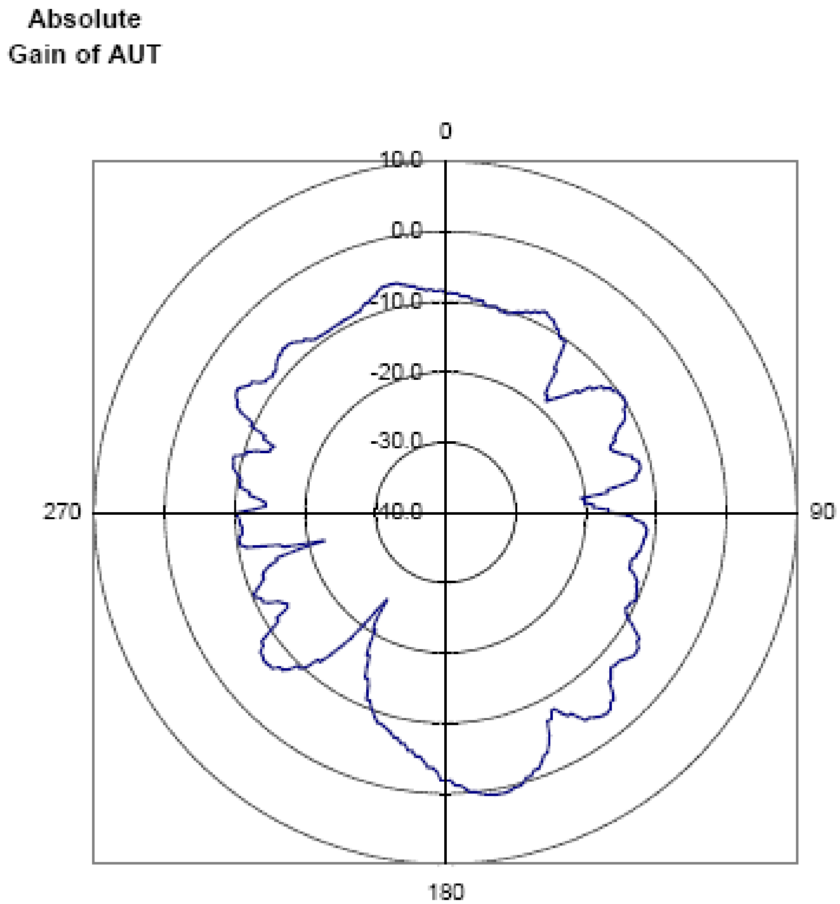
Frequency	2440.00
Absolute Gain of Reference Antenna (dBi)	8.86
Reference Antenna Relative Gain Max (dBuV/m)	102.80
AUT Relative Gain Max (dBuV/m)	94.40
Difference (Reference Antenna - AUT) (dB)	8.40
AUT Setup Loss (dB)	0.00
Maximum Absolute Gain of AUT (dBi)	**0.46**
Correction Factor (Convert From Relative to Absolute Gain) (dB)	93.94
Measurement Antenna Polarity	Vertical
Antenna Under Test (AUT) Polarity	Z

Figure 10. Wiggle Antenna Polar Plot for Z Axis with Measurement Antenna Horizontal



Frequency	2440.00
Measurement Antenna Polarity	Horizontal
Antenna Under Test (AUT) Polarity	Z

Figure 11. Wiggle Antenna Polar Plot (Absolute Gain) for Z Axis with Measurement Antenna Horizontal



Frequency	2440.00
Absolute Gain of Reference Antenna (dBi)	8.86
Reference Antenna Relative Gain Max (dBuV/m)	102.80
AUT Relative Gain Max (dBuV/m)	94.70
Difference (Reference Antenna - AUT) (dB)	8.10
AUT Setup Loss (dB)	0.00
Maximum Absolute Gain of AUT (dBi)	**0.76**
Correction Factor (Convert From Relative to Absolute Gain) (dB)	93.94
Measurement Antenna Polarity	Horizontal
Antenna Under Test (AUT) Polarity	Z

Figure 12. Wiggle Antenna Polar Plot (Relative Gain) for Y Axis with Measurement Antenna Vertical

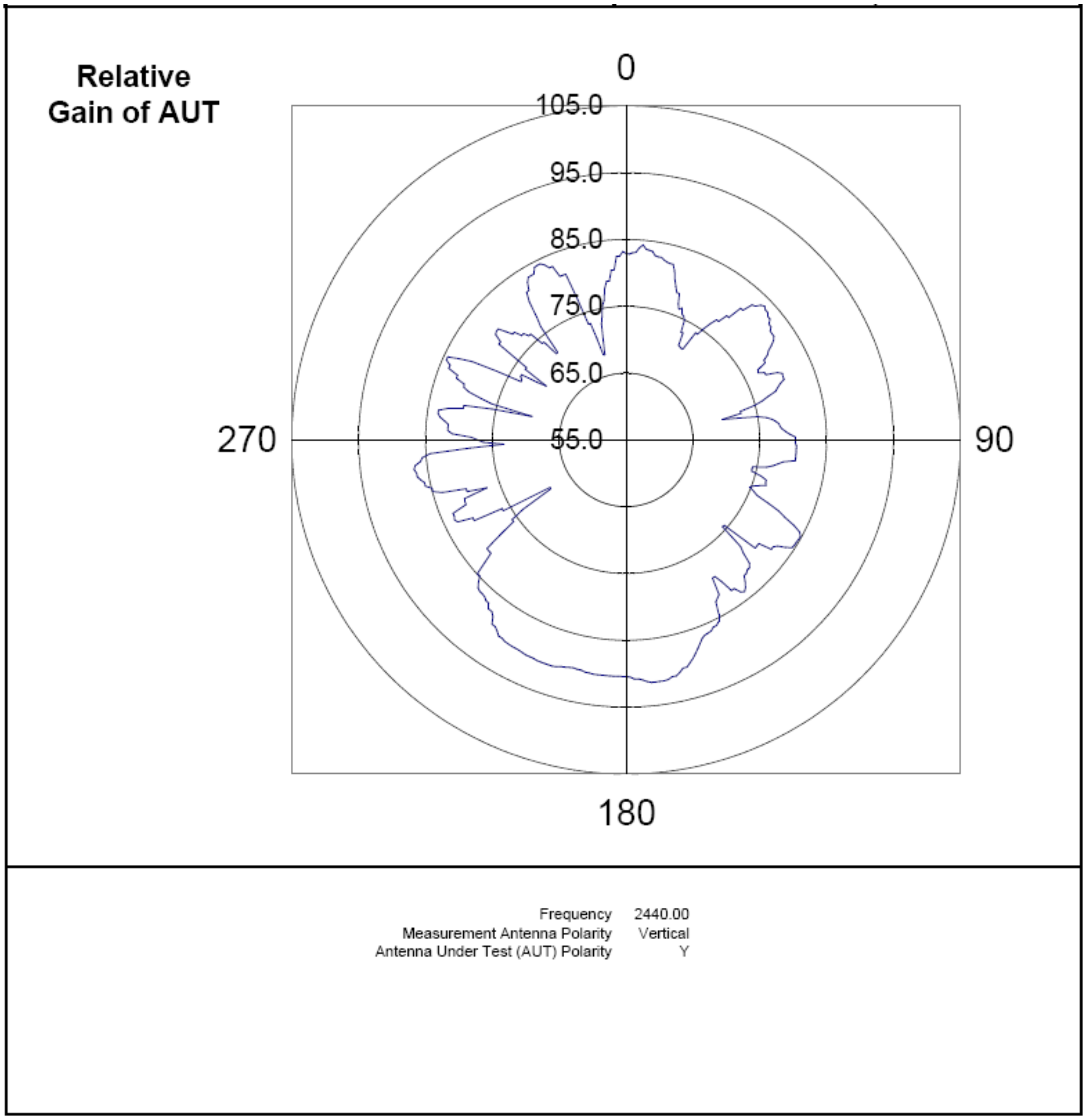
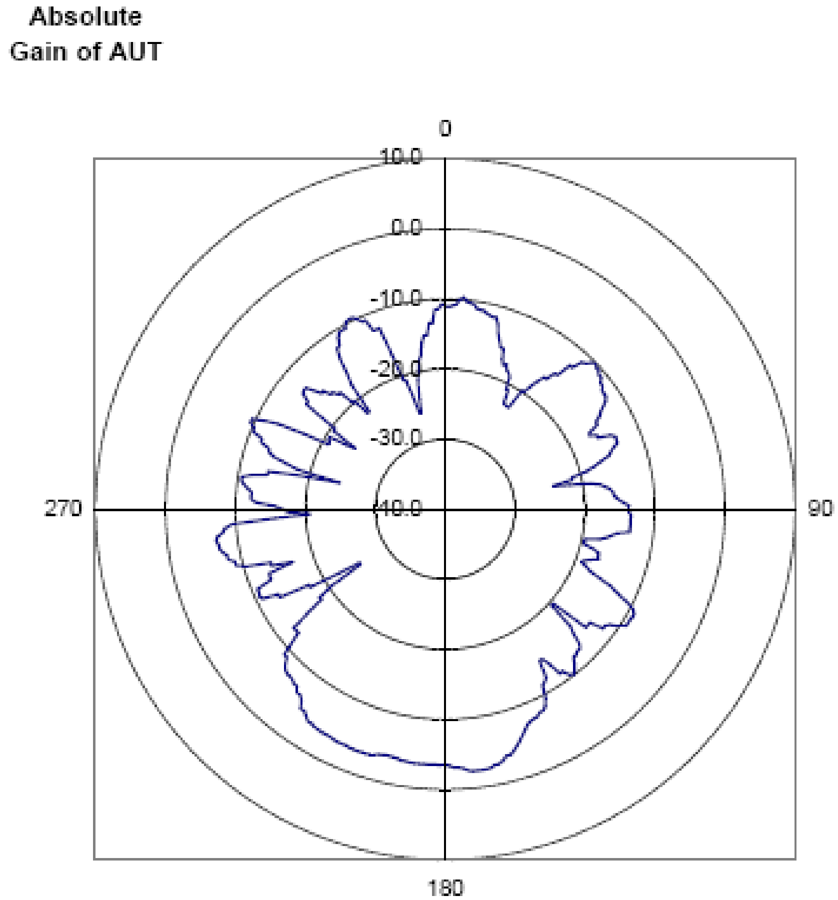


Figure 13. Wiggle Antenna Polar Plot (Absolute Gain) for Y Axis with Measurement Antenna Vertical



Frequency	2440.00
Absolute Gain of Reference Antenna (dBi)	8.86
Reference Antenna Relative Gain Max (dBuV/m)	102.80
AUT Relative Gain Max (dBuV/m)	91.50
Difference (Reference Antenna - AUT) (dB)	11.30
AUT Setup Loss (dB)	0.00
Maximum Absolute Gain of AUT (dBi)	**-2.44**
Correction Factor (Convert From Relative to Absolute Gain) (dB)	93.94
Measurement Antenna Polarity	Vertical
Antenna Under Test (AUT) Polarity	Y

Figure 14. Wiggle Antenna Polar Plot (Relative Gain) for Y Axis with Measurement Antenna Horizontal

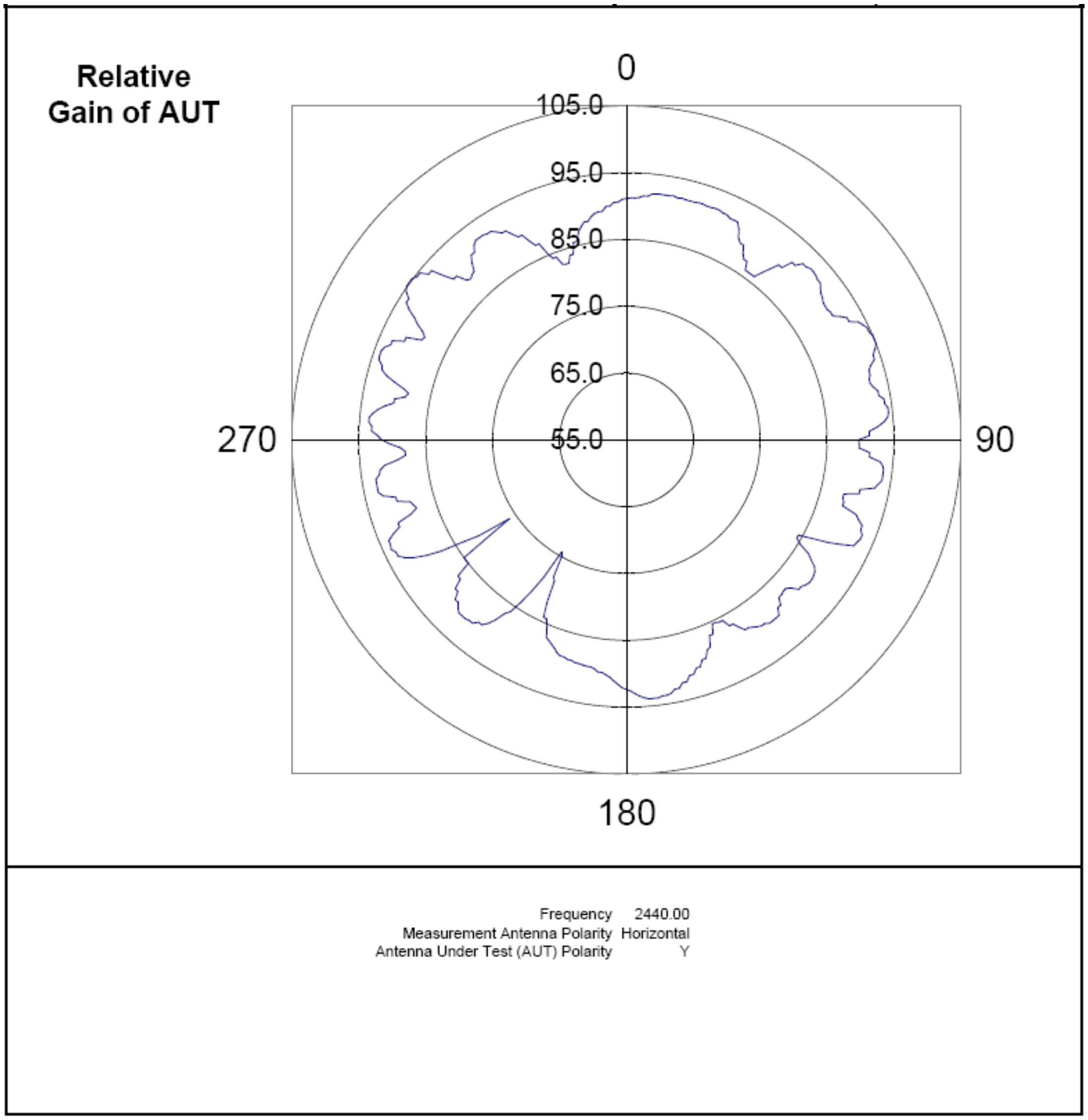
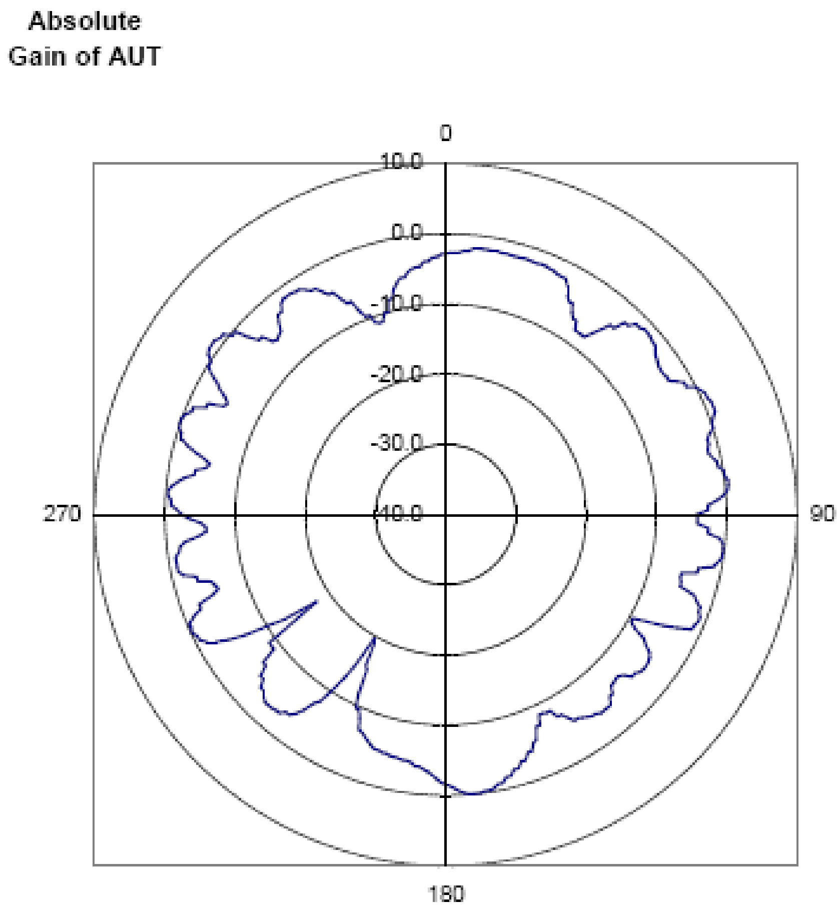


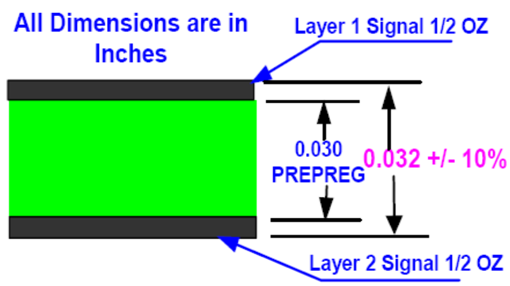
Figure 15. Wiggle Antenna Polar Plot (Absolute Gain) for Y Axis with Measurement Antenna Horizontal



Frequency	2440.00
Absolute Gain of Reference Antenna (dBi)	8.86
Reference Antenna Relative Gain Max (dBuV/m)	102.80
AUT Relative Gain Max (dBuV/m)	95.10
Difference (Reference Antenna - AUT) (dB)	7.70
AUT Setup Loss (dB)	0.00
Maximum Absolute Gain of AUT (dBi)	**1.16**
Correction Factor (Convert From Relative to Absolute Gain) (dB)	93.94
Measurement Antenna Polarity	Horizontal
Antenna Under Test (AUT) Polarity	Y

PCB Manufacturing Specifications

Figure 16. PCB Stack Up Details



Printed circuit board designers provide PCB manufacturers with detailed instructions on how a printed circuit board must be constructed including material, thickness, and international standards to be followed. These instructions are typically provided in the Fabrication Notes.

Cypress follows these specifications for using a two-layer printed circuit board with the WirelessUSB radio:

- Material
  - Type FR-4 epoxy glass laminate and prepreg
  - HTE Copper ½ oz copper foil external layers
  - Overall metal-to-metal thickness 0.0032 inches ± 10%
- Drilling
  - Diameters in the drill table are finished hole sizes ± 0.003-inch tolerance, unless specified in the drill table
  - Teardrop allowed on entry of via on every trace layer
- Copper plating
  - In through-holes 0.001 inches minimum
- Silkscreen
  - In white non-conductive epoxy ink on both sides of board, if applicable
- Solder mask
  - Primary and secondary side of board using liquid photo image mask material over bare copper per IPC-SM-840
- Copper finish
  - Is tin or gold-plated (10 µ-inch minimum)
- Manufactured boards
  - To be in accordance with performance standard IPC-6011/6012, Class-2 board to be inspected according to IPC-600-A Class-2
- Maximum wrap or twist
  - Must not exceed 0.01 in/in

Summary

This application note discusses the general guidelines that aid in the design and layout of the matching network and antenna for the WirelessUSB LP/LPstar radios. These suggestions must be evaluated and optimized for each individual process and design. Many factors affect the overall RF characteristics of a design and must be examined and verified with PCB simulation and analysis tools.

Related Application Notes

None.

About the Author

Name:	Mahesh Kiwalkar
Title:	RF Wireless Staff Applications Engineer
Background:	Mahesh Kiwalkar is an RF Wireless Staff Applications engineer at Cypress Semiconductor. He has been involved with RF wireless design, development, and applications for more than six years. He received his Masters in Electrical Engineering from Pennsylvania State University.
Contact:	mahesh.kiwalkar@cypress.com

Document History

Document Title: Design and Layout Guidelines for Matching Network and Antenna for WirelessUSB™ LP Family

Document Number: 001-48610

Revision	ECN	Orig. of Change	Submission Date	Description of Change
**	2589908	MKKI/AESA	10/15/2008	New application note.
*A	2738702	DVJA	07/15/2009	Updated antenna tip length value and removed the antenna leg length diagram.
*B	3200423	KKCN	04/11/2011	Updated the text to make it compatible with LP/LPstar.
*C	3435331	ZHC	11/10/2011	Updated template according to current CY standards.

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
Optical Navigation Sensors	cypress.com/go/ons
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 5

Cypress Developer Community

Community | Forums | Blogs | Video | Training

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

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