



CYPRESS

WirelessUSB™ Dual Antenna Design Layout Guidelines

The Antenna as a Component of Your Wireless Communication Product Design

Introduction

This application note describes the guidelines for the antenna design and in particular, addresses the design of an integrated printed trace wiggle antenna implementation on a reference printed circuit board. The development of small integrated antenna facilitates the application of the Cypress WirelessUSB™ chip as an integrated wireless communication solution. WirelessUSB is the code name for a technology specification (Cypress Semiconductor's proprietary) for small form factor, low cost, short range radio links between mobile, desktop PCs, and other portable devices. The antenna is an integral part of any wireless communication system. It is the interface between a radio module and atmosphere.

A properly designed antenna facilitates the evaluation, characterization, and production test correlation of the WirelessUSB LS radio. In designing a short-range radio data communication system, the product/system designer faces one of the most important tasks, the antenna design. The key parameters are the antenna size, cost of implementation, radiation effectiveness, ease of manufacturability, and range performance. These recommendations have been 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 purpose of an antenna is to transmit/receive electromagnetic waves to and from the atmosphere. The selection of the antenna for a WirelessUSB solution can have a big impact on wireless communication system performance, system form factor, and cost. 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).

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 affects the performance of wireless communication system utilizing the Cypress WirelessUSB system radio chip. This application note describes a printed trace wiggle antenna design considerations and implementation guidelines for incorporating the WirelessUSB radio system chip into product applications in the ISM frequency band 2.4–2.5 GHz.

Antenna Design Guidelines for WirelessUSB System Radio Chip Applications

The most challenging parameter in designing the antenna in consumer, industrial, scientific, and medical applications is

the environmental impact. The operating environment parameters include indoor, outdoor, building materials, high-rise buildings, factories, and major highways. The physical environment parameters are related to the immediate physical structure surrounding the antenna, such as the placement of the antenna within a product package, the material of the enclosure, the human body, and the surrounding electronic/mechanical components. Therefore a significant amount of characterization, testing, validation and measurement have to be completed to optimize the antenna design.

We have to consider several important antenna characteristics to achieve required antenna performance before deciding on the type of antenna and its implementation. In today's wireless communication product design the choice of antenna selection is more critical and challenging depending upon the application demands in terms of cost, physical constraints and product packaging.

Antenna Types

There are many different types of antennas. Antennas most relevant to 2.4–2.5 GHz designs are:

- Dipole Antennas
- Sleeve Dipole Antennas
- Loop Antennas
- Helical Antennas
- Whip (Monopole) Antennas
- Ceramic Chip Antennas
- Slotted Antennas
- Printed/Planar Inverted –F Antennas
- Printed Trace Wiggle Antennas
- Microstrip Patch Antennas

Each type of antenna has its own advantages and disadvantages depending upon the application. The optimal antenna solution is one that is part of mechanical structure or physical housing for achieving low-cost and consistency in performance.

PCB Materials—Choose the Right One for Your Application

A variety of different materials are used to fabricate PCBs. These materials can also be assembled in a variety of different ways potentially using multiple laminates, different materials and different plated through via structures. A range of finishes can also be used making use of materials such as gold, nickel, tin, and lead. The board material of choice for radio circuit application is FR4.

Printed circuit board materials are available in various grades as defined by the National Electrical Manufacturers Assoc-

ation (NEMA). It would be convenient for designers if NEMA was closely allied with the electronics industry—controlling parameters such as resistivity and dielectric constant of the material. Unfortunately, that is not the case. NEMA is an electrical safety organization, and the different PCB grades primarily describe the flammability, high temperature stability, and moisture absorption of the board. Therefore, specifying a given NEMA grade does not guarantee electrical parameters of the material. If this becomes critical, consult the manufacturer of the raw board stock.

Raw PCB stock is graded in flammability ratings (FR) from 1 to 5, with 1 being the most flammable and 5 the least. FR-4 is commonly used in industrial quality equipment, while FR-2 is used in high volume consumer applications. Although there are no set rules for this, it appears to be an industry “standard”. Deviating from it without good reason can limit the number of suppliers for raw board material and PCB houses that can fabricate the board, as their tooling is already set up for these materials.

In selecting a board material, pay careful attention to moisture absorption. Just about every desirable performance characteristic of the board will be negatively impacted by moisture. This includes surface resistance of the board, dielectric leakage, high voltage breakdown and arcing, and mechanical stability. Also pay attention to operating temperature. High operating temperatures can occur in unexpected places, such as in proximity to large digital chips that are switching at high speeds. Be aware that heat rises, so if one of the fast switching ICs is located directly under a sensitive analog circuit—both the PCB and circuit characteristics may vary with the temperature.

How Many Layers are Best?

Many times, the number of board layers already has been determined by system constraints. If the designer has a choice, however, there are some guidelines. Very simple consumer electronics are sometimes fabricated on single-sided PCBs, keeping the raw board material inexpensive (FR-1 or FR-2) with thin copper cladding. These designs frequently include many jumper wires—simulating the circuit routing on a double-layer board. This technique is only recommended for low frequency circuitry. For reasons described below this type of design is extremely susceptible to radiated noise. Therefore, it is actually more complex to design a board of this type due to the many things that can go wrong.

The next level of complexity is double layer. Initially, this type of board would seem to lend itself to easier routing, because now there are two layers of foil, and it is possible to route signals by crossing traces on different layers. While that is certainly possible, it is not recommended for analog circuitry. Wherever possible the bottom layer should be devoted to a continuous ground plane and all other signals routed on the top layer. A ground plane provides several benefits.

Ground is frequently the most common connection in the circuit. Having it continuous on the bottom layer usually makes the most sense for circuit routing. It increases mechanical strength of the board. It lowers the impedance of all ground connections in the circuit which reduces undesirable conducted noise. It adds a distributed capacitance to every net in the circuit, helping to suppress radiated

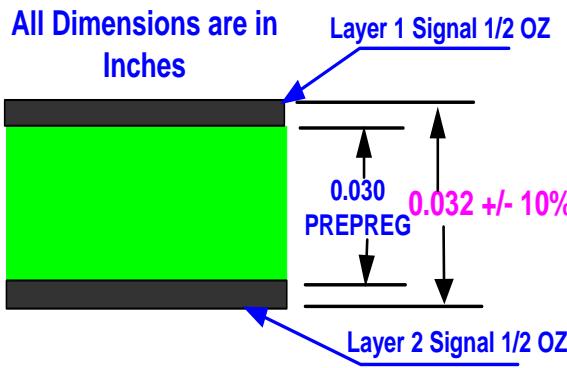
noise. It acts as a shield to radiated noise coming from underneath the board.

PCB Manufacturing Specifications

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

Below are some specifications Cypress uses in specifying a two-layer printed circuit board for use with the WirelessUSB radio.

- Material
 - Type FR-4 epoxy glass laminate and prepreg
 - HTE Copper $\frac{1}{2}$ oz copper foil external layers
 - Overall metal-to-metal thickness 0.0032 inches \pm 10%
- Drilling
 - Diameters in the drill table are finished hole sizes \pm 0.003-inch tolerance, unless otherwise 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
 - Shall be 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 per IPC-600-A Class-2
- Maximum wrap or twist
 - Shall not exceed 0.01 in/in



PCB Stack up details

Figure 1. PCB Stack-Up Details

General guidelines are given, which will aid in designing a printed trace Wiggle antenna on a PCB. These suggestions should be evaluated and optimized for each individual process and design. Many factors affect the overall RF characteristics of a design and can be examined and verified with PCB simulation and analysis tools.

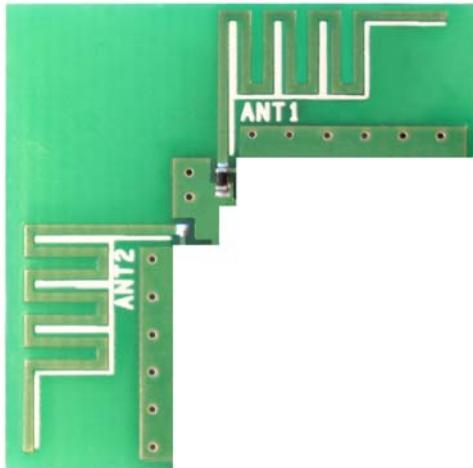


Figure 2. Dual Wiggle Antenna Top Side as Implemented on Cypress Reference Radio Module

Radio Module with Wiggle Antenna

The radio module printed circuit board is implemented on a two-layer board using low-cost FR-4 material. The picture of dual wiggle antenna as implemented on Cypress reference radio module is shown in *Figure 2* and *Figure 3*. The Wiggle trace antenna and RF grounding details implemented on the reference radio module are shown in *Figure 4* and *Figure 5*.



Figure 3. Dual Wiggle Antenna Bottom Side as Implemented on Cypress Reference Radio Module

Dual Wiggle Antenna

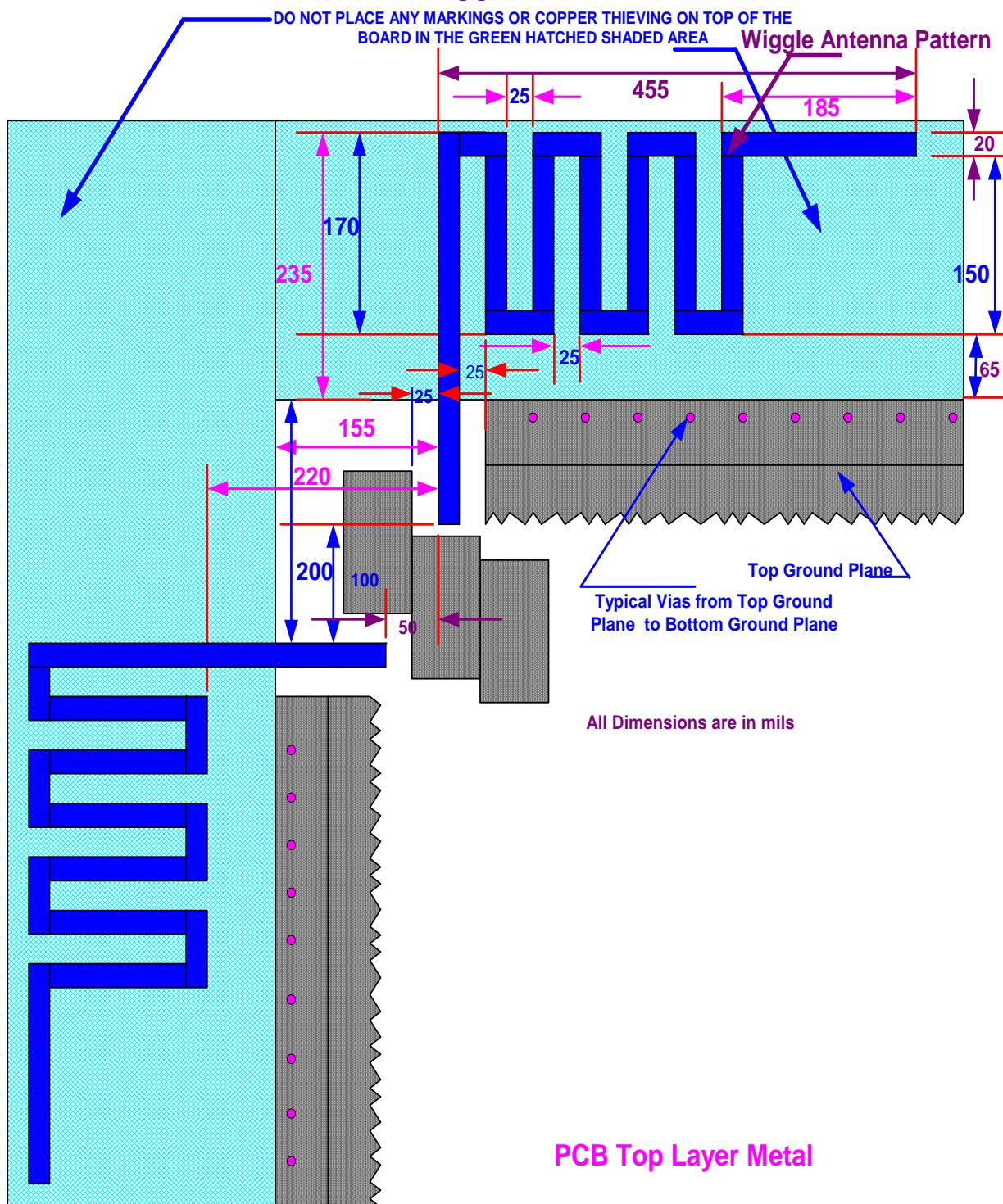


Figure 4. Dual Wiggle Antenna Design Details Top Side

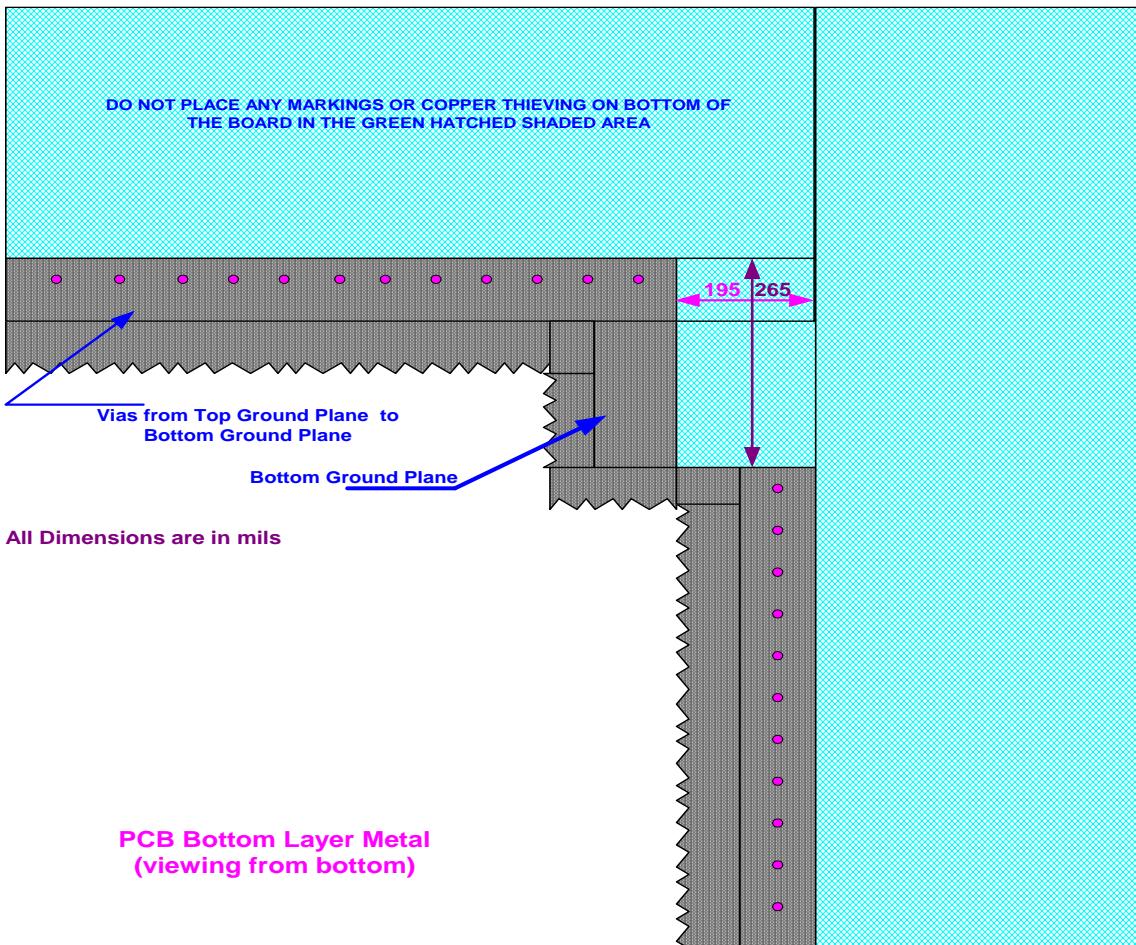


Figure 5. Dual Wiggle Antenna Design Details Bottom Side

Design Guidelines for the Antenna Choice and Implementation

The rules of thumb for the antenna choice, selection and implementation:

- The performance of the antenna is dependent on its immediate surroundings, packaging and proximity to the ground plane. The placement of antenna position should be identified early in the design process.
- The orientation of the device and the product usage model during the operation should be considered in mounting the antenna inside the device.
- If you are using an external antenna and if you are connecting the antenna with a coaxial cable assembly, the cable routing needs to be designed in such a manner as to keep it away from motors and battery packs.
- Large, continuous ground plane surfaces will provide better radiation performance than small surfaces.
- Product applications using keypads, LCD or other types of displays, battery packs and other metallic surfaces will affect and degrade the symmetry of the radiation pattern, reflections and multipath. Therefore the location of the an-

tenna placement is critical. Place the antenna for best balance of the distribution of these objects.

- The effects of human body and the operator's hand should be examined and validated away during the product operation. By keeping the antenna away, the Specific Absorption Rate (SAR) will be reduced and pattern symmetry will be improved.
- It is better to eliminate connectors and interconnect transmission lines to avoid insertion losses on transmit power and receive sensitivity on the receiver.
- 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.

Summary

This application note addresses the radio module design using WirelessUSB Radio ICs and, in particular, discusses one version of the PCB trace antenna layout details.

WirelessUSB is a trademark of Cypress Semiconductor Corporation. All product and company names mentioned in this document are the trademarks of their respective holders.