



CYPRESS

WirelessUSB™ LS Printed Circuit Board Layout Guidelines

The PCB Is a Component of Your Wireless Communication Product Design

This application note provides guidelines for printed circuit board (PCB) design using the WirelessUSB™ LS 2.4-GHz DSSS Radio SoC IC and its application as an integrated wireless communication solution. A properly designed PCB facilitates the evaluation, characterization, and production test correlation of the WirelessUSB LS radio. 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. This application note provides design details for a WirelessUSB LS radio module to aid the customer with RF PCB layout guidelines.

Reference Radio Module Description

Described in this document is a radio module that is based on the WirelessUSB LS radio reference design evaluation board. The objective is to provide a base design for applications requiring a radio module. To avoid RF design issues in PCB design, portions of the reference radio module, labeled “copy exact,” should be used as-is.

The radio module is designed for short-range wireless connectivity applications in the worldwide ISM Band 2.400- to 2.483-GHz frequency band. The user can achieve excellent data integrity at over a 10-meter range with proper RF PCB design, antenna implementation, and product packaging. This application note outlines the basic RF PCB design practices and transceiver-to-antenna matching elements without the use of a transmit/receive switch, and facilitates the application of the WirelessUSB LS radio into communication products. Additionally, the included reference design example provides guidelines that allow the customer to reduce cost and simplify the manufacturing process.

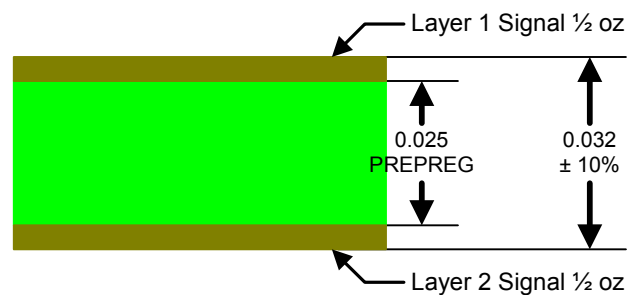
PCB Manufacturing Specifications

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 LS radio.

- Material
 - Type FR-4 epoxy glass laminate and prepreg
 - HTE Copper ½ oz copper foil external layers
 - Overall metal-to-metal thickness 0.0064 inches \pm 10%

- Drilling
 - Diameters in the drill table are finished hole sizes \pm 0.003-inch tolerance, unless otherwise specified in drill table
 - Teardrop allowed on entry of via on every trace layer
- Copper plating in through-holes is 0.001 inches min.
- Silkscreen in white non-conductive epoxy ink on both sides of board, if applicable.
- Solder mask primary and secondary side of board using liquid photoimage mask material over bare copper per IPC-SM-840
- Copper finish shall be tin- or gold-plated(50 μ -inch min.)
- 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



All dimensions are in inches

Figure 1. PCB Stackup Details

IC Pad Layout Design

The WirelessUSB LS radio employs a 28-pin exposed paddle SOIC package shown in *Figure 2* and *Figure 3*. The SOIC package has a Ground Pad (EPAD) of size 190 mils \times 225 mils (width \times length) on the bottom of the IC to improve RF grounding and RF performance of the small package. Cypress recommends adding a 125-mil hole underneath the ground pad of the OIC package to allow a solid soldering joint ground connection. This allows excess solder to bleed through the hole, permits visual inspection of the ground connection, and simplifies rework on the bench, if necessary.

- Refer to the WirelessUSB LS IC PCB PAD Layout Design Details in *Figure 4*.
- For the RFIC (28-pin IC) note the paddle soldering and through-hole structure which appear in the reference layout design. These are highly important for ensuring maximum RF performance.

Figure 4 shows a typical cross-sectional view of the reference PCB layout and WirelessUSB LS IC mounting details. The planar inverted-F antenna (PIFA) and RF grounding details

implemented on the reference radio module are shown in Figure 5.

28 Lead (300 Mil) SOIC - EPAD

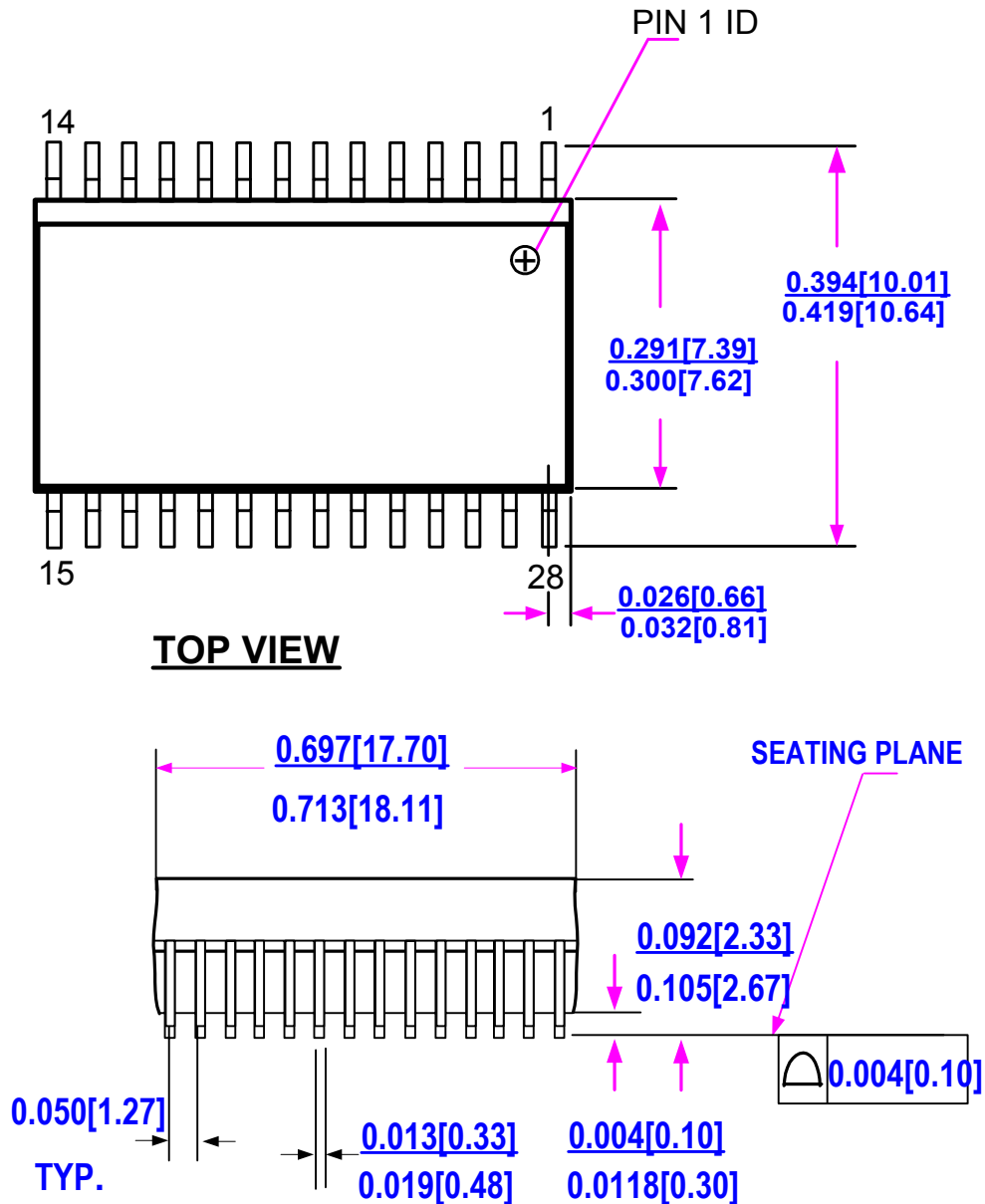
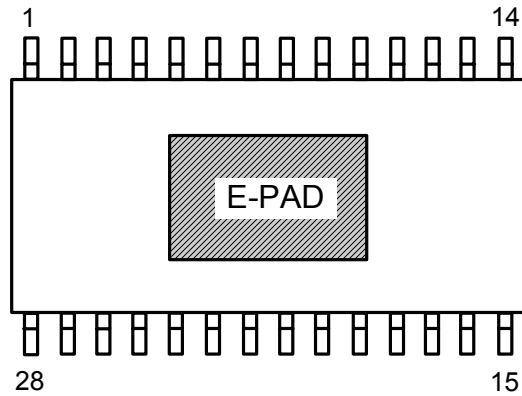


Figure 2. WirelessUSB LS IC Package Details

DIMENSIONS IN INCHES[MM] MIN.
MAX.
 REFERENCE JEDEC MS-013



BOTTOM VIEW

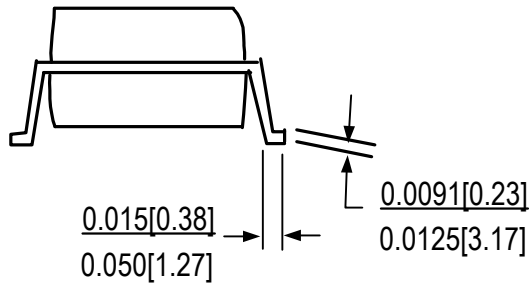
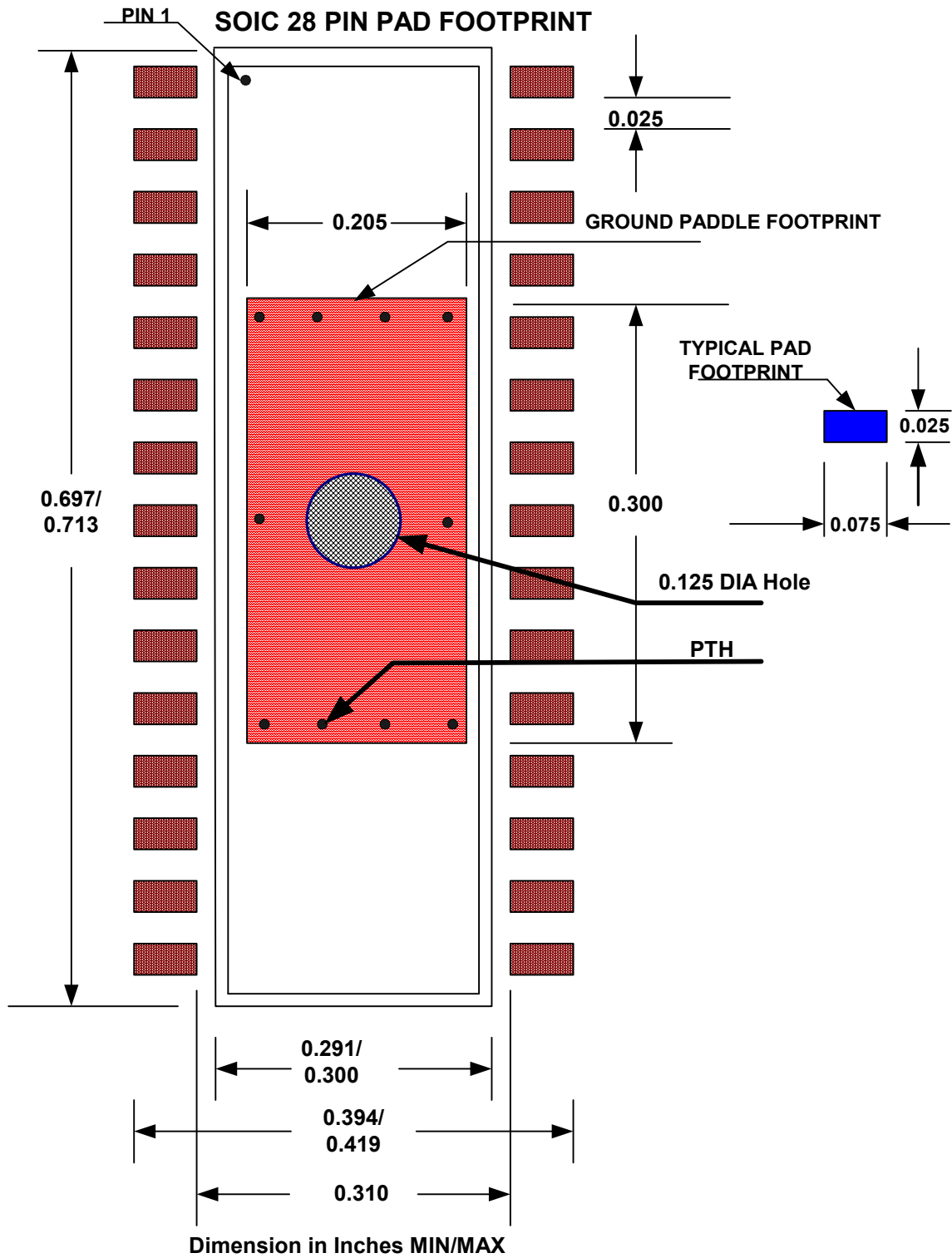


Figure 3. Dimensions



Note: 0.125 inch drill hole is not to scale (as shown in this figure).

Figure 4. WirelessUSB LS IC PCB PAD Layout Design Details

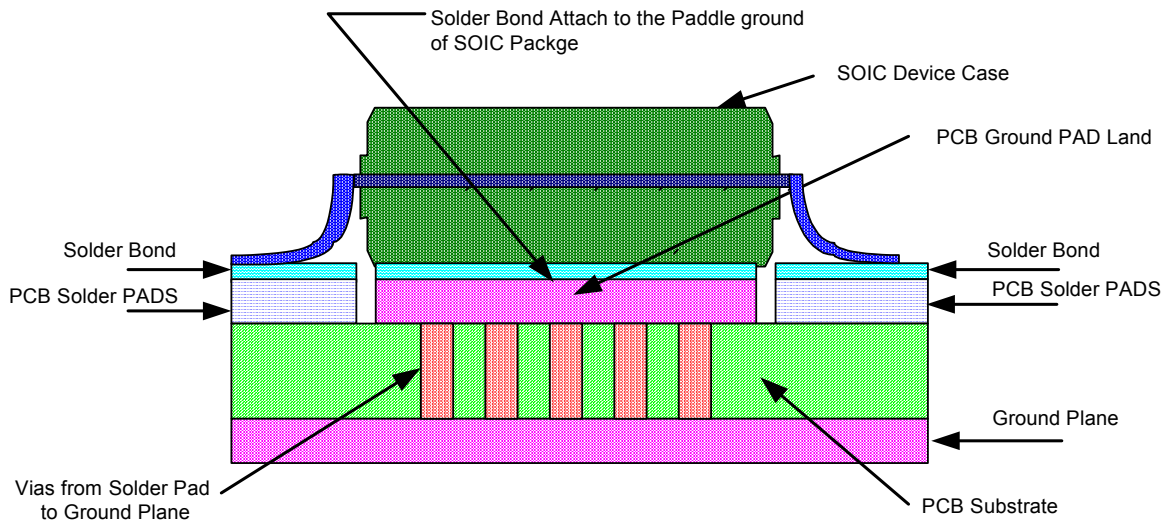


Figure 5. Typical Cross-Sectional Views of the PCB Layout and WirelessUSB LS IC Mounting Details

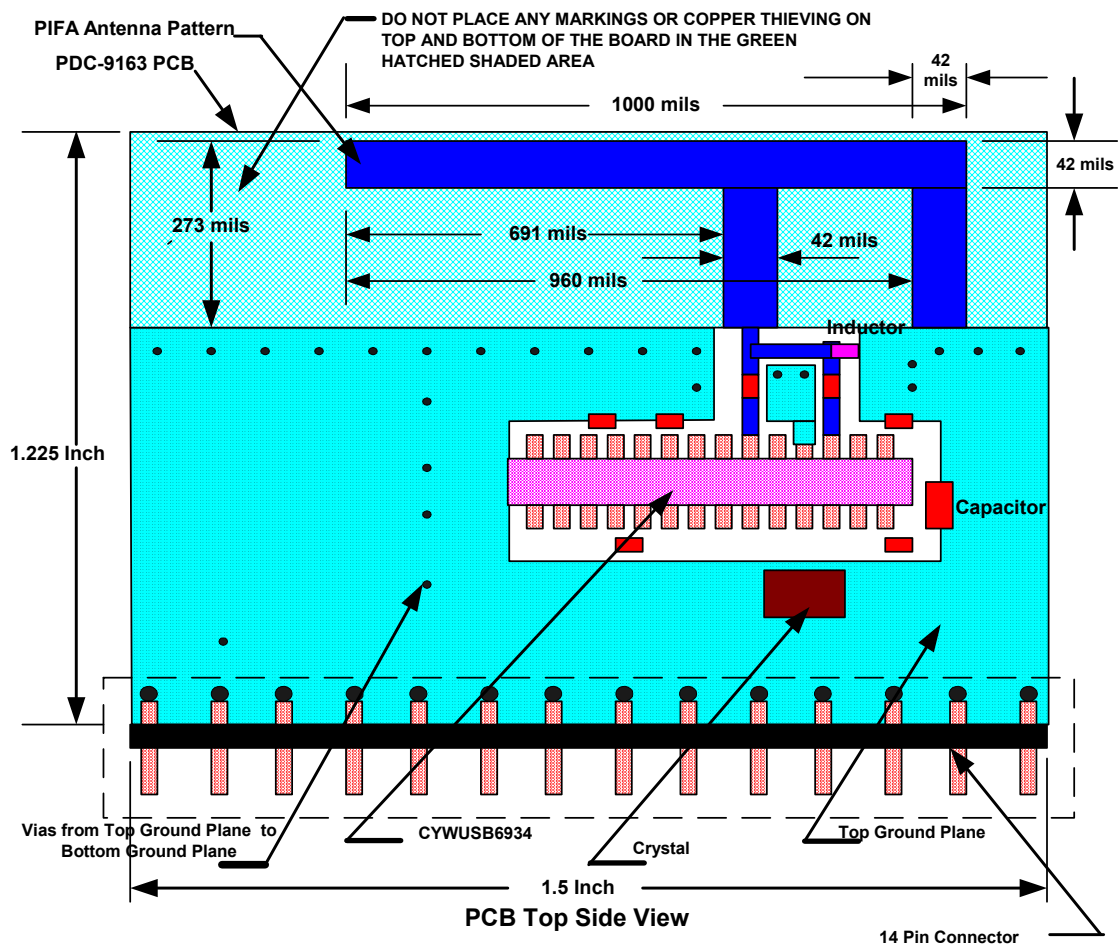


Figure 6. Typical Top View of the PIFA Antenna Layout

PDC-9163: Reference Radio Module

The PDC-9163 printed circuit board is implemented on a two-layer board using low-cost FR-4 material. The bottom of the board under the IC is used for power routing and non-critical signal lines. However, most of the bottom layer is a ground plane with the exception of the region under the antenna which must be clear of all copper. The ground plane on the top layer of the circuit board is connected to bottom layer ground plane using vias (i.e., plated through-holes).

This reference board design consists of:

- WirelessUSB LS radio
- Power supply filtering circuitry
- 14-pin header interface connector
- Antenna matching circuitry
- Planar inverted-F antenna (PIFA)
- 13-MHz SMT crystal

Copy Exact Areas and Components:

- **Antenna:** The antenna is implemented as a PIFA with the PCB trace on the component side of the PCB. All of the ground plane underneath the PIFA trace (along the entire length of the antenna) must be removed from the backside of the PCB. Do not place components or ground plane near the tip of the antenna, as the coupling of the antenna radiation to ground will reduce the effective range of operation.
- **Antenna matching components:** The antenna matching components are C1, L1, C2, and C4. All of the matching components are made from the same package size to mount on 0603 pads. The placement of these elements and the antenna should match precisely to that shown on *Figure 7* and *Figure 8*.
- **VCC decoupling and routing traces:** The capacitors C5, C7, C8, C9, C10 and C11 mounting pads should only have vias on or next to the capacitor mounting pad which connects to the VCC bus on the bottom side of the PCB. The VCC pin trace should be routed to the decoupling capacitor on the component side of the board.
- **RF ground** is the “surface ground” on the topside of the PCB. So it is important that the antenna ground connection point on the top surface ground plane remains wide to ensure a low inductance path back to the SOIC paddle ground.
- A 125-mil hole should be added at the center of the paddle ground to facilitate solder reflow and possible rework should the reflow be insufficient.
- The crystal component placement area should have a ground plane underneath the crystal. No other signals should be routed underneath the crystal.
- If X13OUT signal is used for your application, the signal X13OUT should be surrounded by ground traces on either side. It is recommended that a continuous ground plane be maintained under the X13OUT trace. X13OUT will corrupt the oscillator performance if it is allowed to couple to X13 and/or X13IN.

Figure 7 and *Figure 8* give details of the PDC-9163 layout areas to be copied exactly to achieve the specified RF performance.

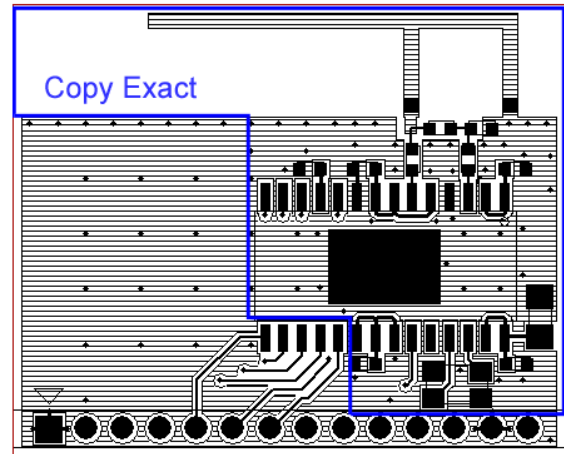


Figure 7. Example of PDC-9163 Top Side Trace Routing “Copy Exact” Layout Areas

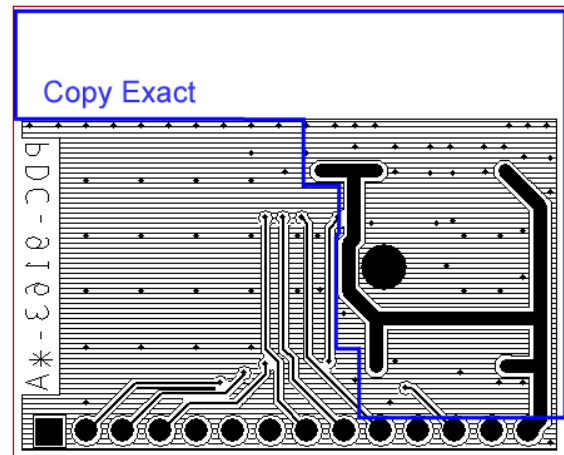
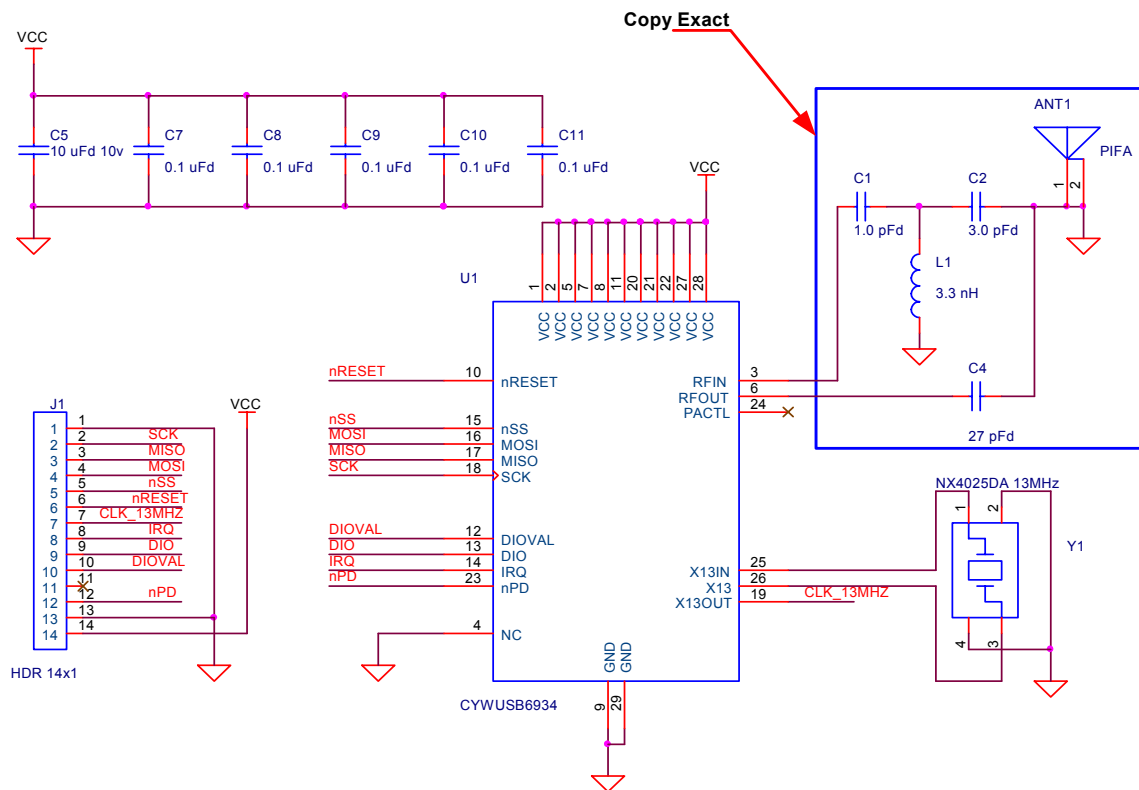


Figure 8. Example of PDC-9163 Bottom Trace Routing “Copy Exact” Layout Areas


Figure 9. WirelessUSB LS Radio Module Schematic
Table 1. WirelessUSB LS Radio Module Bill of Materials

TITLE WirelessUSB LS Radio Module
PCA # 121-16302 Revision **
SCH # REF-11569 Revision *A
DATE 09/25/2003

| Item | Qty | Reference | Description | Manufacturer | Mfg. Part Number |
|------|-----|----------------------|-----------------------------------|-----------------------|------------------|
| 1 | 1 | C1 | CAP 1.0PF 50V CERAMIC NPO 0603 | Kemet | C0603C109C5GACTU |
| 2 | 1 | C2 | CAP 3.0PF 50V CERAMIC NPO 0603 | Kemet | C0603C309C5GACTU |
| 3 | 1 | C4 | CAP 27PF 50V CERAMIC NPO 0603 | Kemet | C0603C270J5GACTU |
| 4 | 1 | C5 | CAP 10UF 10V CERAMIC Y5V 1206 | TDK | C3216Y5V1A106Z |
| 5 | 5 | C7, C8, C9, C10, C11 | CAP 0.10UF 16V CERAMIC X7R 0603 | Kemet | C0603C104J4RACTU |
| 6 | 1 | J1 | CONN HDR RT ANG 14POS 0.100" | MOLEX | 22-28-8142 |
| 7 | 1 | L1 | INDUCTOR 3.3NH ± 0.3NH FIXED 0402 | Panasonic - ECG | ELJ-RF3N3DF |
| 8 | 1 | U1 | IC, 24-PIN LS RADIO TRANSCEIVER | Cypress Semiconductor | CYWUSB6934-28SEC |
| 9 | 1 | Y1 | Crystal 13-MHz SMT | NDK | NX4025DA |
| 10 | 1 | PCB | PRINTED CIRCUIT BOARD | Cypress Semiconductor | PDC-9163 REV *A |

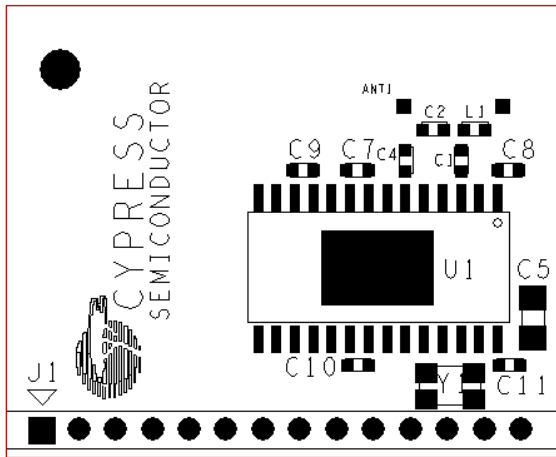


Figure 10. PDC-9163 Component Placement – Top Side

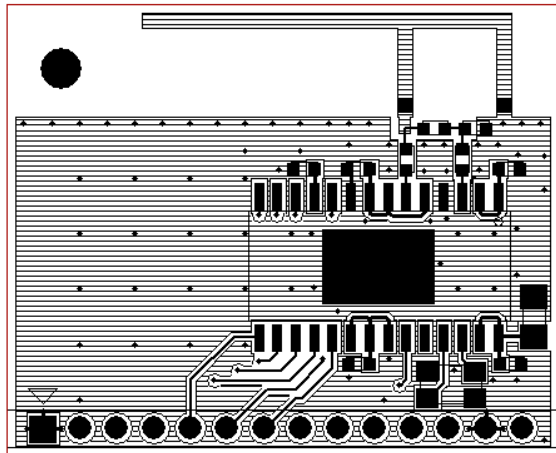


Figure 11. PDC-9163 PCB Trace Routing – Top Side

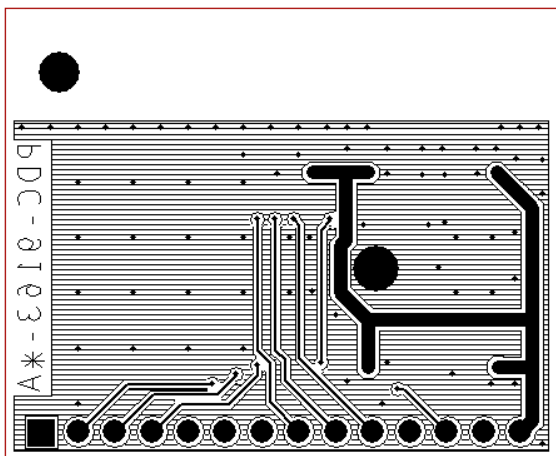


Figure 12. PDC-9163 PCB Trace Routing – Bottom Side

Antenna Matching Components

A component matching network connects the TX and RX pins of the LS radio to the antenna. Because the TX and RX pins both connect to the antenna it is important that the matching network corresponds to impedances of the TX and RX pins to increase transmission and reception range. Using the PDC-9163 radio module, impedance measurements were made under the following test criteria:

- Measure the impedance of the RF port in receive and transmit modes
- Measure the impedance of the transmitter
- Measure the antenna impedance

| Port Input/Output | Impedance Value in Ohms | Test Condition |
|-------------------|-------------------------|----------------|
| RFIN | $27.7 + j 20.2$ | RX mode |
| RFIN | $10.0 + j 0.0$ | TX mode |
| RFOUT | $15.2 + j 19.0$ | RX mode |
| Antenna Input | $35.7 + j 10.7$ | |

The antenna matching network element values are arrived by optimization simulations. The matching network connections and layout are shown in *Figure 9* and *Figure 11*.

Practical RF Printed Circuit Board Design Guidelines

In today's wireless communication products there is a demand for a new level of performance, low cost, small physical size, and more features and functionality.

The continuing breakthroughs in surface-mount-technology, miniaturization of passive components, and a higher level of integration with mixed signal integrated circuits including RF circuitry takes wireless communication to higher operating frequencies. The industry faces problems as the operating frequency increases. Use the following guidelines to design PCBs using WirelessUSB LS to eliminate common high-frequency design challenges:

- The RF ground plane is critical for achieving the required RF performance. Good performance can be achieved with careful layout of the ground plane without the need for 4 or 6 layers.
- The power supply connection to the WirelessUSB LS IC demands good decoupling with low ESR ceramic capacitors. Traces must be kept very short between WirelessUSB LS transceiver and passive components. Locate vias between decoupling capacitors and the power supply bus near (or under) the decoupling capacitors.
- Give careful attention to the 13-MHz crystal placed between X13 and X13IN. In particular, keep the X13OUT trace separated from X13 and X13IN traces.
- Signal coupling can be created due to the proximity of traces. Lengths of individual and adjacent tracks can have an inductive or capacitive effect causing signal coupling. Because traces to test points can also cause signal coupling keep test points directly on the trace.
- Limit the number of signal vias as they add inductance to the critical circuit path. In contrast, use a large number of



vias to tie the front and backside ground plane regions together.

- Avoid in-circuit test pins on RF nodes as they create small antennas and may result in possible FCC issues.

It is essential to minimize these problems early in the product development stage to reduce the number of prototypes necessary to meet the twin objectives of desired functionality and good product yield.

Summary

This application note addresses the radio module design using WirelessUSB LS and, in particular, discusses the PCB layout issues and the importance of optimized PCB design from an RF perspective. PCB designers must consider the effects of PCB stackup, the cumulative effect of parasitic capacitances, the effect of lead inductances of inductors, the component placements both surface mount and through hole, the effect of Q (passive discrete components) and traces, and vias used for power supply and ground connections.

This document is subject to change, and may be found to contain errors of omission or changes in parameters. For feedback or technical support regarding Cypress WirelessUSB products please contact Cypress at www.cypress.com. WirelessUSB is a trademark of Cypress Semiconductor. All product and company names mentioned in this document are the trademarks of their respective holders.

Approved AN4005 2/13/04 kkv