



# WirelessUSB™ LS Printed Circuit Board Layout Guidelines - AN5030

## 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 0.032 inches ± 10%

- Drilling
  - Diameters in the drill table are finished hole sizes ±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(10 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.

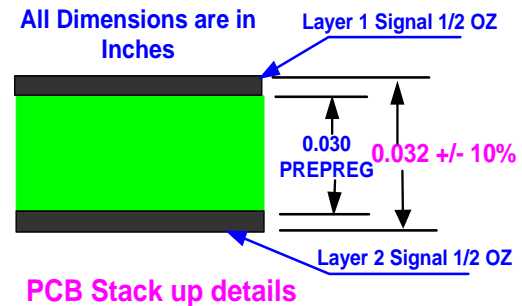


Figure 1. PCB Stackup Details

## PDC-9207: Single Wiggle Reference Radio Module

WirelessUSB LS QFN package layout design details: refer to the application note “Optimum PCB and Stencil Layout for WirelessUSB QFN Package.” The PDC-9207 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

- 12-pin 2-mm header interface connector
- Antenna matching circuitry
- PCB trace antenna (Wiggle)
- 13-MHz SMT crystal

The following dimensioning data is used to design the PCB.

- Critical RF trace width = 10 mils
- Power supply trace width = 25 mils
- Control signal trace width = 7 mils
- Via hole size = 10 mils (the pad size = 25 mils)

#### Copy Exact Areas and Components:

- Antenna: The antenna is implemented as a PCB trace on the component side of the PCB. All of the ground plane underneath the wiggle 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, C3, L2 and C4. All of the matching components are made from the same package size to mount on 0402 pads. The placement of these elements and the antenna should match precisely to that shown on *Figure 3* and *Figure 6*.
- $V_{CC}$  decoupling and routing traces: The capacitors C5, C6, C7, C8, C9, C10 and C11 mounting pads should only have vias on or next to the capacitor mounting pad which connects to the  $V_{CC}$  bus on the bottom side of the PCB. The  $V_{CC}$  pin trace should be routed to the decoupling capacitor on the component side of the board.
- Power supply isolation: It is important to provide separate traces and decoupling capacitors for both pin 44 and pin 45 of the radio. Provide separate power traces and decoupling capacitors to each pin. Avoid using the same trace to connect pin 44 to pin 45 to  $V_{CC}$ .
- RF ground is the “surface ground” on the topside of the PCB.
- A 100-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.

The radio module dimensional details are given in *Figure 2*. *Figure 3* and *Figure 4* give details of the PDC-9207 layout areas to be copied exactly to achieve the specified RF performance. The component placement details are shown in *Figure 6* and *Figure 7*.

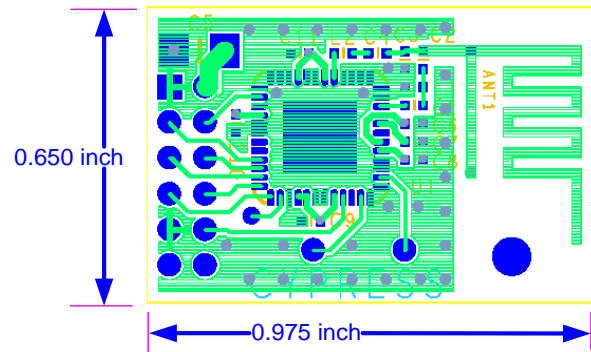


Figure 2. PDC-9207 Radio Module Dimensions

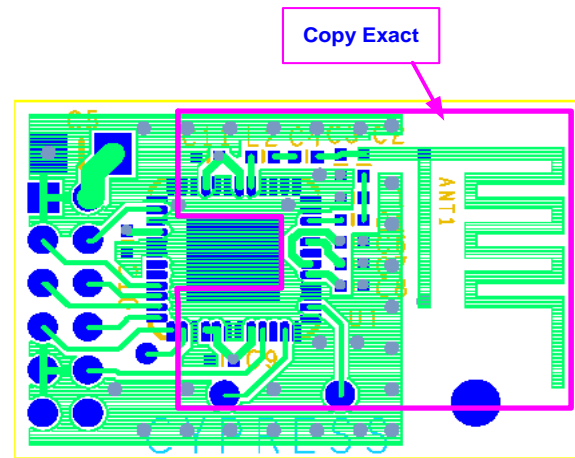


Figure 3. Example of PDC-9207 Top Side Trace Routing “Copy Exact” Layout Areas

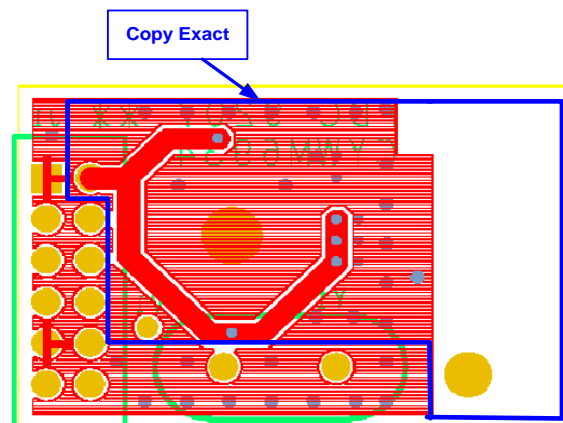


Figure 4. Example of PDC-9207 Bottom Side Trace Routing “Copy Exact” Layout Areas

The radio module schematic is shown in *Figure 5*.

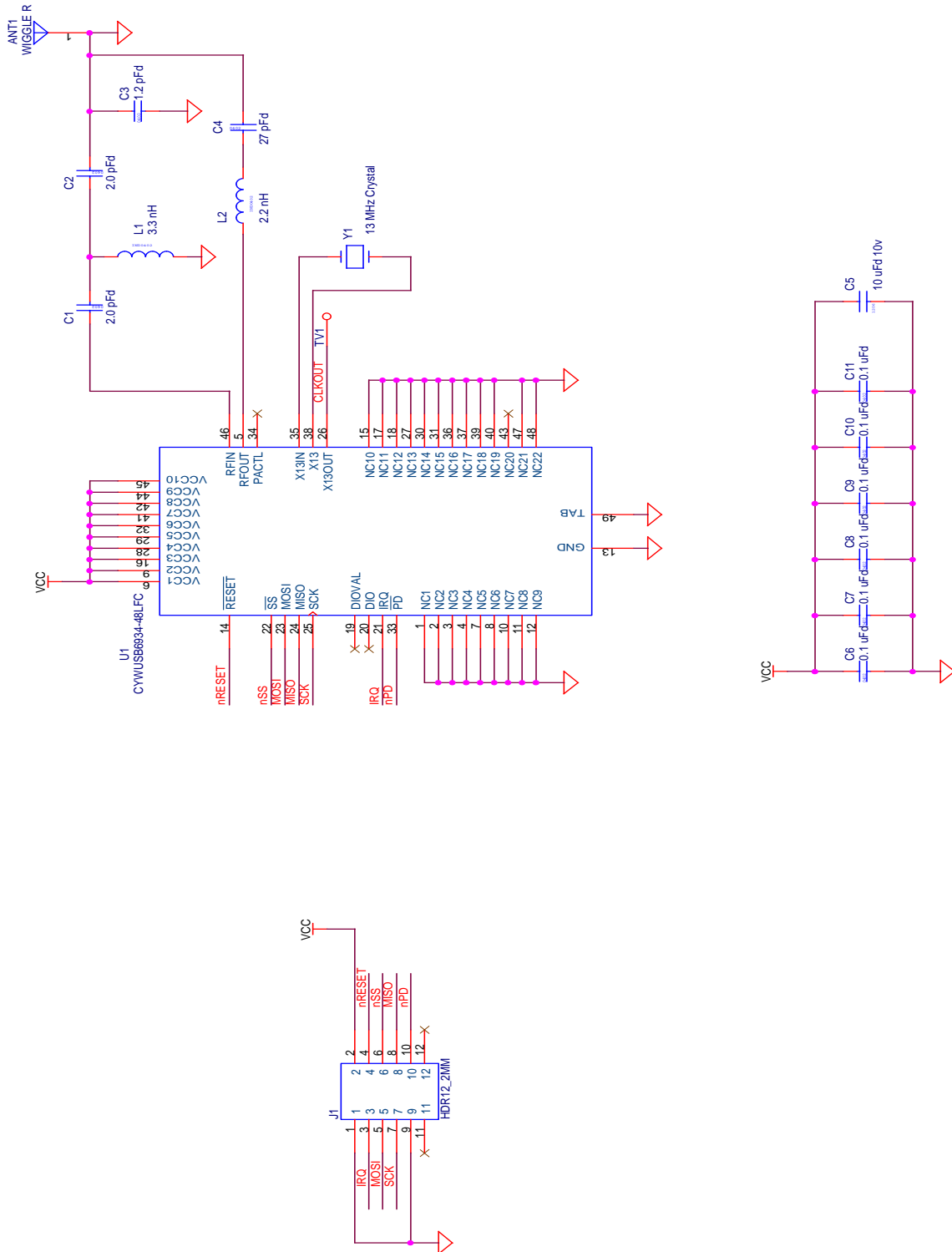
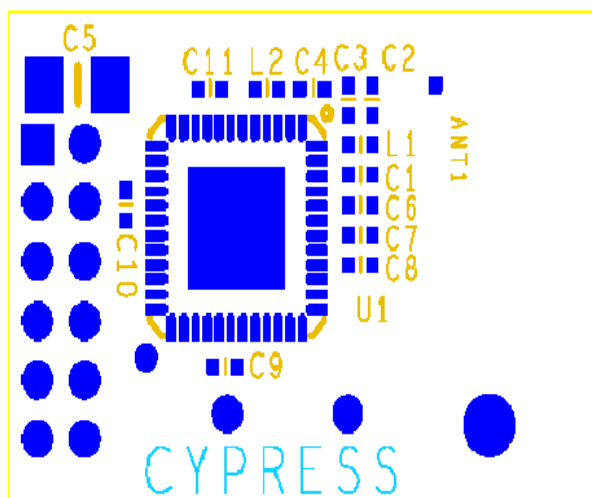


Figure 5. WirelessUSB LS Radio Module Schematic

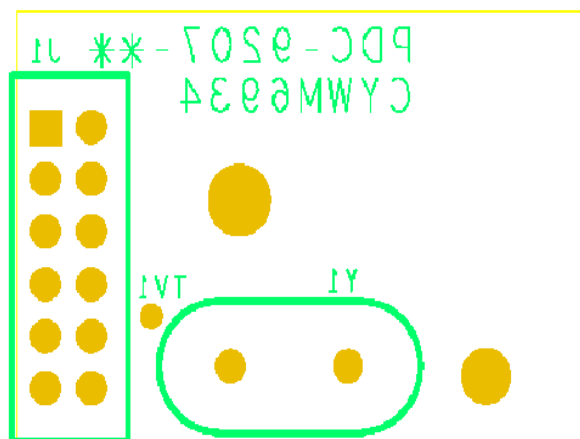
The radio module bill of materials is shown in *Table 1*.

**Table 1. PDC-9027 Bill of Materials (BOM)**

Item	Qty	CY Part Number	Reference	Description	Manufacturer	Mfr Part Number																												
<table border="1"> <tr> <td>TITLE</td> <td colspan="6">WirelessUSB LS Micro Radio Module</td> </tr> <tr> <td>PCA #</td> <td>121-20700</td> <td>Revision</td> <td>**</td> <td></td> <td></td> <td></td> </tr> <tr> <td>SCH #</td> <td>REF-13001</td> <td>Revision</td> <td>**</td> <td></td> <td></td> <td></td> </tr> <tr> <td>DATE</td> <td>10/12/2004</td> <td></td> <td></td> <td></td> <td></td> <td></td> </tr> </table>							TITLE	WirelessUSB LS Micro Radio Module						PCA #	121-20700	Revision	**				SCH #	REF-13001	Revision	**				DATE	10/12/2004					
TITLE	WirelessUSB LS Micro Radio Module																																	
PCA #	121-20700	Revision	**																															
SCH #	REF-13001	Revision	**																															
DATE	10/12/2004																																	
1	2	730-11955	C1,C2	CAP 2.0 PF 50V CERAMIC NPO 0402	Kamet	C0402C209C5GACTU																												
2	1	730-11954	C3	CAP 1.2 PF 50V CERAMIC NPO 0402	Kamet	C0402C129C5GACTU																												
3	1	720-11291	C4	CAP 27PF 50V CERAMIC NPO 0402	Kemet	C0402C270J5GACTU																												
4	1	730-10765	C5	CAP 10UF 10V CERAMIC Y5V 1206	TDK	C3216Y5V1A106Z																												
5	6	730-11952	C6,C7,C8,C9, C10,C11	CAP 0.1 uF 50V CERAMIC X5R 0402	Kamet	C0402C104K8PACTU																												
6	1	420-11976	J1	CONN HEADER 12 PIN 2MM GOLD	3M	151212-8422-TB																												
7	1	800-11277	L1	INDUCTOR 3.3NH +- .3NH FIXED SMD 0402	Panasonic - ECG	ELJ-RF3N3DF																												
8	1	800-11652	L2	INDUCTOR 2.2NH +- .3NH FIXED SMD 0402	Panasonic - ECG	ELJ-RF2N2DF																												
9	1	CYWUSB6934-48LFC	U1	IC, LS RADIO TRANSCEIVER QFN- 48	Cypress Semiconductor	CYWUSB6934-48LFC																												
10	1	800-11920	Y1	CRYSTAL 13.00MHZ HC49	eCERRA	GG-1300001																												
11	1	PDC-9207 REV**	PCB	PRINTED CIRCUIT BOARD	Cypress Semiconductor	PDC-9207 REV**																												



**Figure 6. PDC-9207 Component Placement – Top Side**



**Figure 7. PDC-9207 Component Placement – Bottom Side**

Antenna Design dimensional/implementation details: See the application note "WirelessUSB LS Antenna Layout Guidelines".

### 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 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

Port Input/Output	Impedance Value in Ohms	Test Condition
RFIN	$24.7 + j 1.5$	RX mode
RFIN	$19.25 - j 39.3$	TX mode
RFOUT	$13.0 - j 2.25$	RX mode

The antenna matching network element values are arrived by optimization simulations. The matching network connections and layout are shown in *Figure 5* and *Figure 4*.

## 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/LR 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.

Cypress Semiconductor  
198 Champion Court  
San Jose, CA 95134-1709  
Phone: 408-943-2600  
Fax: 408-943-4730  
<http://www.cypress.com>

© Cypress Semiconductor Corporation, 2005-2007. The information contained herein is subject to change without notice. Cypress Semiconductor Corporation assumes no responsibility for the use of any circuitry other than circuitry embodied in a Cypress product. Nor does it convey or imply any license under patent or other rights. Cypress products are not warranted nor intended to be used for medical, life support, life saving, critical control or safety applications, unless pursuant to an express written agreement with Cypress. Furthermore, Cypress does not authorize its products for use as critical components in life-support systems where a malfunction or failure may reasonably be expected to result in significant injury to the user. The inclusion of Cypress products in life-support systems application implies that the manufacturer assumes all risk of such use and in doing so indemnifies Cypress against all charges.

This Source Code (software and/or firmware) is owned by Cypress Semiconductor Corporation (Cypress) and is protected by and subject to worldwide patent protection (United States and foreign), United States copyright laws and international treaty provisions. Cypress hereby grants to licensee a personal, non-exclusive, non-transferable license to copy, use, modify, create derivative works of, and compile the Cypress Source Code and derivative works for the sole purpose of creating custom software and/or firmware in support of licensee product to be used only in conjunction with a Cypress integrated circuit as specified in the applicable agreement. Any reproduction, modification, translation, compilation, or representation of this Source Code except as specified above is prohibited without the express written permission of Cypress.

Disclaimer: CYPRESS MAKES NO WARRANTY OF ANY KIND, EXPRESS OR IMPLIED, WITH REGARD TO THIS MATERIAL, INCLUDING, BUT NOT LIMITED TO, THE IMPLIED WARRANTIES OF MERCHANTABILITY AND FITNESS FOR A PARTICULAR PURPOSE. Cypress reserves the right to make changes without further notice to the materials described herein. Cypress does not assume any liability arising out of the application or use of any product or circuit described herein. Cypress does not authorize its products for use as critical components in life-support systems where a malfunction or failure may reasonably be expected to result in significant injury to the user. The inclusion of Cypress' product in a life-support systems application implies that the manufacturer assumes all risk of such use and in doing so indemnifies Cypress against all charges.

Use may be limited by and subject to the applicable Cypress software license agreement.