Contents

1. Introduction ........................................................................................................................................... 2
  1.1 Key Points for Optimal Layout ........................................................................................................... 2

2. Power Circuits ......................................................................................................................................... 3
  2.1 LC Tank ........................................................................................................................................... 4
  2.2 Communication Capacitors ............................................................................................................... 5
  2.3 VRECT Capacitors ............................................................................................................................. 6
  2.4 OUT Pin Capacitor ............................................................................................................................ 6
  2.5 VDD5V and VDD18 Capacitors ......................................................................................................... 6
  2.6 SINK Trace ...................................................................................................................................... 7
  2.7 Programming Resistors ..................................................................................................................... 7
  2.8 EEPROM Circuits and Programming Interface .................................................................................. 7
  2.9 Thermal Management ....................................................................................................................... 8

3. PCB Footprint Design ............................................................................................................................ 9

4. Audible Noise Suppression ................................................................................................................... 10

5. Schematic ............................................................................................................................................ 11

6. Board Layout ....................................................................................................................................... 12

7. Revision History ................................................................................................................................ 15

List of Figures

Figure 1. P9225-R Reference Layout ........................................................................................................ 2
Figure 2. P9225-R Power Block Diagram ................................................................................................. 3
Figure 3. Recommended Orientation for the P9225-R Layout .................................................................. 4
Figure 4. P9225-R Reference Layout (LC Tank) ....................................................................................... 4
Figure 5. COMx Capacitors Layout .......................................................................................................... 5
Figure 6. VRECT Capacitor Layout – Top Layer ..................................................................................... 6
Figure 7. SINK Net Layout ....................................................................................................................... 7
Figure 8. P9225-R Layout Heat Flow ....................................................................................................... 8
Figure 9. P9225-R Recommended PCB Footprint Dimensions in mm .................................................... 9
Figure 10. P9225-R Schematic ................................................................................................................ 11
Figure 11. P9225-R Top Silkscreen (left) and Bottom Silkscreen (right) ................................................ 12
Figure 12. P9225-R Top Layer (left) and Second Layer (right). ............................................................. 13
Figure 13. P9225-R Third Layer (left) and Bottom Layer (right) ............................................................ 14
1. Introduction

The P9225-R Wireless Power Receiver (Rx) is a highly integrated solution for 5W applications. The reference layout has been optimized to minimize the printed circuit board (PCB) area (9mm × 9mm), and it can be easily copied and pasted into a design for a compact system, such as mobile phone applications.

In the layout, there are high current AC signals. It is critical that the current path is handled well in the printed circuit board (PCB) design to achieve optimum performance in electromagnetic compatibility (EMC) tests. The thermal management is also very important to the P9225-R performance, and the PCB layout should be optimized and balanced between proper design of the layout area and heat dissipation.

In the reference layout, the V-cut slots are designed on the bottom layer so that the area of the PCB with the core function circuits (green shaded area in Figure 1) can be easily detached and built into prototype products. Contact IDT for further technical information.

Figure 1. P9225-R Reference Layout

1.1 Key Points for Optimal Layout

- Route the power connections wide and keep them on the same side of the PCB as the P9225-R (≥ 70mils) for the LC, AC1, AC2, OUT, VRECT, and GND nets.
- Use the bottom layer of the board as a solid ground plane, and also place additional ground plane on the second layer.
- Connect all 8 GND pins to the ground plane(s) using via-in-pads. Add a thermal pad for the J-row GND pins.
- Avoid unnecessary layer transitions of the AC power connections (LC node and the VRECT, AC1, AC2, and GND pins).
- Place the P9225-R as close as possible to the center of the board. Avoid placing it along the PCB edge.
- Connect as much copper as possible to every pin of the P9225-R, including pins that do not carry high current.
- Place components in the following order: resonance capacitors, VRECT pin capacitors, BST pin capacitors, OUT pin capacitors, VDD18 pin capacitors, and VDD5V pin capacitors.
- Follow the placement and routing suggestions outlined in the remainder of this document for the specific types of circuits. Refer to the schematics in section 5 for the location of components.
2. Power Circuits

Figure 2 shows the main circuits and components involved in power circuits. The main power current flows from the Rx coil to the AC1 pin via the CS capacitors and returns to other end of the Rx coil from the AC2 pin. Loop 1 identified in Figure 2 is the AC current, and the polarity changes in every driving cycle. In Loop 2, high frequency harmonic current in the LC tank returns to GND via CRECT. In Loop 3, COUT buffers the output of the LDO on the OUT pin. Details of the layout guidelines for each of the current loops will be covered in subsequent sections.

**Figure 2. P9225-R Power Block Diagram**

When the PCB dimensions have been determined based on mechanical constraints, the connection points for the receiver coil LS should be determined. Next, the P9225-R should be placed in the center of the allocated PCB area. The orientation of the chip should be determined based on the route connections to key components. The key components should be placed in the following order of priority: resonance capacitors (CS), rectifier capacitors (CRECT), output capacitor (COUT), and VDD5V and VDD18 capacitors.
2.1 LC Tank

The LC tank circuit consists of the receiver coil (L_s) and the C_s resonant capacitance (C1, C2, C3, and C5) as illustrated in Figure 4 and Figure 10. The current runs through Loop 1 as indicated in the Figure 4, and it contains high frequency harmonics. The loop area must be minimized to reduce radiated emission noise.

Recommendation: Route the loop on the top layer and place components compactly. Along the current loop, copper width should be maximized if space allows. Using a large copper plane on the C_s capacitors and AC2 net will help to reduce copper loss caused by large currents, and it also helps to spread the heat away from C_s capacitors and the P9225-R.

Figure 4. P9225-R Reference Layout (LC Tank)
2.2 Communication Capacitors

Communication capacitors C6, C14 (for WPC), C7, and C15 (for PMA) should be placed on the top layer in order to have a direct connection to the AC1 or AC2 net without using vias as indicated in Figure 5 by the blue rectangles. The traces for the terminals on the other side of the capacitors must be routed via the third layer to connect to the respective pins on the P9225-R, and the trace width should be at least 20mil.

When communication is on-going, these capacitors function as a load to the system; therefore it is necessary to keep the trace wide enough to handle the current. To minimize the layout area, there are vias on the solder pad of the capacitors. The vias must be plugged at the bottom layer to avoid poor soldering caused by paste leakage.

Figure 5. COMx Capacitors Layout
2.3 VRECT Capacitors

High frequency harmonic currents in the LC tank are rectified and filtered by the capacitors on the VRECT pin. High frequency radiated emission noise could be induced if the loop is not minimized.

Recommendation: Place the VRECT capacitors (C23, C22, and C21) on the same layer as the P9225-R and place them as close as possible as illustrated by the green-shaded area in Figure 6.

The trace width should also be maximized. C23 (100nF) should be placed closer to the VRECT pin because it filters higher frequencies. To minimize the layout area, C21 and C22 have a low-profile 0805 package. If the space allowed or cost is a concern, C21 and C22 could be replaced by three 10µF capacitors with a 0603 package and the same voltage rating. Vias should be plugged on the bottom layer due to the via-on-pads design. At least two GND vias (10mil) should be placed to have a good connection to a solid GND plane on the bottom layer.

Figure 6. VRECT Capacitor Layout – Top Layer

2.4 OUT Pin Capacitor

Output capacitor should be placed close to the P9225-R OUT pin as well. Using long traces between the OUT pin and the output capacitor would introduce inductance, which could degrade the stability of the output LDO. The width of the trace between the OUT pin and capacitor should be maximized. At least two GND vias (10mil) should be placed to have a good connection to GND plane. To minimize the layout area, the output capacitor has a low-profile 0805 package. It could be replaced by two 10µF capacitors with a 0603 package and the same voltage rating if space allowed or cost is concern. The VOUT trace to the system load should be maximized to prevent copper losses and a rise in temperature on the board.

2.5 VDD5V and VDD18 Capacitors

The capacitors on VDD5V (C18) and VDD18 (C20) are used to stabilize the P9225-R’s internal 5V and 1.8V supplies respectively. On each pin, a single 1µF capacitor is sufficient since the load on each rail is quite small. Each capacitor should be placed close to the respective P9225-R pin. An 8mil trace is wide enough to handle the current.
2.6 SINK Trace

The SINK pin is directly connected to VRECT net. There will be current flow if there is an over-voltage condition on the VRECT net. A trace that is at least 12mil should be used on the third layer to connect the VRECT net and SINK pin, and two 10mil vias should be placed.

Figure 7. SINK Net Layout

2.7 Programming Resistors

The programming resistors on the VOSET, ILIM, RPPO, RPPG, EOC, and INT pins are in a less sensitive circuit, and they can be placed away from the P9225 if the layout is crowded. The 0201 package is sufficient for these resistors.

Recommendation: Place C19 (100nF) close to the VPP18 net that is used to pull up programming resistors.

2.8 EEPROM Circuits and Programming Interface

Recommendation for the product development phase: Add the EEPROM circuit on the board if space allows. It is helpful for quick firmware updates. The programming interface should have test points implemented for the PS (external power supply), I2C SDA, I2C SCL, and GND net, and the test points should be grouped together in a location with easy access to the programming dongle.
2.9 Thermal Management

On the top layer, a GND plane should be placed with a strong connection to the P9225-R GND pins to improve the heat dissipation. At least six 10mil vias should be placed on the top layer thermal pad to transition heat to other layers. If space allows, place more vias and a larger thermal pad on the top layer.

The second layer is directly under the P9225-R, and more GND copper in this layer could also help to dissipate the heat from the P9225-R. The bottom layer should be solid GND plane, as it is the most effective layer to directly exchange heat with the ambient environment. In addition to the PCB assembly, the Rx receiver coil is another heat source. When designing the system mechanical structure, it is necessary to avoid placing the Rx coil close to the top of the PCB assembly because even without physical contact, both the PCBA and coil will heat each other due to heat radiation. The NTC sensor (RTS) can be placed close to the P9225-R, which is the only heat source on the board.

Figure 8. P9225-R Layout Heat Flow
3. PCB Footprint Design

The P9225-R package is a fine-pitch WLCSP package. Improper footprint design can lead to solder shorts or open circuits. Poor PCB footprint design can also cause the performance to be degraded by limiting the robustness and diameter of the pin-to-board connections. In order to minimize the risk of such events, it is recommended that the PCB pin pads and via-in-pads be designed using the following guidance.

Non-solder mask defined pins are recommended, and solder paste should be applied with stencil openings of 0.127mm to 0.268mm (recommendation: 0.19mm typical) based on stencil thickness and the solder paste selected. The pin diameter should be set to 0.268mm, the solder mask should be 0.3315mm, and via-in-pads should be 0.127mm diameter holes.

The current version of the WLCSP package outline drawing and land pattern are accessible via this link:
www.idt.com/document/psc/ahg52-package-outline-2640-x-3940mm-body-04mm-pitch-dsbga

Figure 9.  P9225-R Recommended PCB Footprint Dimensions in mm
4. Audible Noise Suppression

Wireless power receiver solutions have been observed to produce audible noise. If sound is detected, there are several steps that can be taken to reduce or eliminate the noise. Some of the sources of the audible noise have been identified to be the rectifier capacitors, the Rx coil ferrite, and communication capacitors. Typically, the rectifier capacitors are the most significant source of audible noise. This is due to the WPC communication signals being generated in the audible frequency range and the use of small-form factor ceramic capacitors. The noise occurs due to the piezoelectric effect of ceramic capacitors. The capacitors constrict and expand while providing the communication pulses, and this noise is amplified as it flexes the PCB. The primary solution to this issue is to use low-acoustic noise capacitors. Alternatively, higher voltage rated components can have superior piezoelectric properties that can reduce the audible noise. Placing the capacitors on both sides of the PCB (directly above and below each other) counters the piezoelectric forces applied to the PCB (it cancels the force by each capacitor). Another method is to add slots through the PCB on both outer sides of the capacitors or directly under each capacitor. One additional approach is to place additional lower capacitance value components in parallel to reduce the mechanical force of the piezoelectric effect per component. For any additional questions, contact IDT technical support (see last page for contact information).
5. Schematic

Figure 10. P9225-R Schematic
6. Board Layout

Figure 11. P9225-R Top Silkscreen (left) and Bottom Silkscreen (right)
Figure 12. P9225-R Top Layer (left) and Second Layer (right)
Figure 13. P9225-R Third Layer (left) and Bottom Layer (right)
### 7. Revision History

<table>
<thead>
<tr>
<th>Revision Date</th>
<th>Description of Change</th>
</tr>
</thead>
<tbody>
<tr>
<td>May 5, 2018</td>
<td>Initial release.</td>
</tr>
</tbody>
</table>
Notice

1. Descriptions of circuits, software and other related information in this document are provided only to illustrate the operation of semiconductor products and application examples. You are fully responsible for the incorporation or any other use of the circuits, software, and information in the design of your product or system. Renesas Electronics disclaims any and all liability for any losses and damages incurred by you or third parties arising from the use of these circuits, software, or information.

2. Renesas Electronics hereby expressly disclaims any warranties against and liability for infringement or any other claims involving patents, copyrights, or other intellectual property rights of third parties, by or arising from the use of Renesas Electronics products or technical information described in this document, including but not limited to, the product drawings, charts, programs, algorithms, and application examples.

3. No license, express, implied or otherwise, is granted hereby under any patents, copyrights or other intellectual property rights of Renesas Electronics or others.

4. You shall not alter, modify, copy, or reverse engineer any Renesas Electronics product, whether in whole or in part. Renesas Electronics disclaims any and all liability for any losses or damages incurred by you or third parties arising from such alteration, modification, copying or reverse engineering.

5. Renesas Electronics products are classified according to the following two quality grades. "Standard" and "High Quality". The intended applications for each Renesas Electronics product depends on the product's quality grade, as indicated below.

   - **Standard**: Computers; office equipment; communications equipment; test and measurement equipment; audio and visual equipment; home electronic appliances; machine tools; personal electronic equipment; industrial robots; etc.

   - **High Quality**: Transportation equipment (automobiles, trains, ships, etc.); traffic control (traffic lights); large-scale communication equipment; key financial terminal systems; safety control equipment; etc.

   Unless expressly designated as a high reliability product or a product for harsh environments in a Renesas Electronics data sheet or other Renesas Electronics document, Renesas Electronics products are not intended or authorized for use in products or systems that may pose a direct threat to human life or bodily injury (artificial life support devices or systems; surgical implantations; etc.), or may cause serious property damage (space system; undersea repeaters; nuclear power control systems; aircraft control systems; key plant systems; military equipment; etc.). Renesas Electronics disclaims any and all liability for any damages or losses incurred by you or any third parties arising from the use of any Renesas Electronics product that is inconsistent with any Renesas Electronics data sheet, user's manual or other Renesas Electronics document.

6. When using Renesas Electronics products, refer to the latest product information (data sheets, user's manuals, application notes, "General Notes for Handling and Using Semiconductor Devices" in the reliability handbook, etc.), and ensure that usage conditions are within the ranges specified by Renesas Electronics with respect to maximum ratings, operating power supply voltage range, heat dissipation characteristics, installation, etc. Renesas Electronics disclaims any and all liability for any malfunctions, failure or accident arising out of the use of Renesas Electronics products outside of such specified ranges.

7. Although Renesas Electronics endeavors to improve the quality and reliability of Renesas Electronics products, semiconductor products have specific characteristics, such as the occurrence of failure at a certain rate and malfunctions under certain use conditions. Unless designated as a high reliability product or a product for harsh environments in a Renesas Electronics data sheet or other Renesas Electronics document, Renesas Electronics products are not subject to radiation resistance design. You are responsible for implementing safety measures to guard against the possibility of bodily injury, injury or damage caused by fire, and/or danger to the public in the event of a failure or malfunction of Renesas Electronics products, such as safety design for hardware and software, including but not limited to redundancy, fire control and malfunction prevention, appropriate treatment for aging degradation or any other appropriate measures.

8. Please contact a Renesas Electronics sales office for details as to environmental matters such as the environmental compatibility of each Renesas Electronics product. You are responsible for carefully and sufficiently investigating applicable laws and regulations that regulate the inclusion or use of controlled substances, including without limitation, the EU RoHS Directive, and using Renesas Electronics products in compliance with all these applicable laws and regulations. Renesas Electronics disclaims any and all liability for damages or losses occurring as a result of your noncompliance with applicable laws and regulations.

9. Renesas Electronics products and technologies shall not be used for or incorporated into any products or systems whose manufacture, use, or sale is prohibited under any applicable domestic or foreign laws or regulations. You shall comply with any applicable export control laws and regulations promulgated and administered by the governments of any countries asserting jurisdiction over the parties or transactions.

10. It is the responsibility of the buyer or distributor of Renesas Electronics products, or any other party who distributes, disposes of, or otherwise sells or transfers the product to a third party, to notify such third party in advance of the contents and conditions set forth in this document.

11. This document shall not be reprinted, reproduced or duplicated in any form, in whole or in part, without prior written consent of Renesas Electronics.

12. Please contact a Renesas Electronics sales office if you have any questions regarding the information contained in this document or Renesas Electronics products.

(Note1) "Renesas Electronics" as used in this document means Renesas Electronics Corporation and also includes its directly or indirectly controlled subsidiaries.

(Note2) "Renesas Electronics product(s)" means any product developed or manufactured by or for Renesas Electronics.

Corporate Headquarters
TOYOSU FORESIA, 3-2-24 Toyosu, Koto-ku, Tokyo 135-0061, Japan
www.renesas.com

Contact Information
For further information on a product, technology, the most up-to-date version of a document, or your nearest sales office, please visit:
www.renesas.com/contact/

Trademarks
Renesas and the Renesas logo are trademarks of Renesas Electronics Corporation. All trademarks and registered trademarks are the property of their respective owners.

© 2019 Renesas Electronics Corporation. All rights reserved.