Contents

1. Introduction 2
2. P9241-G Layout Quick Checklist 2
3. Power Circuits 3
4. Bridge Routing 5
5. Bootstrap Capacitors 7
6. VCC5V BUCK and LDO Routing 8
7. VBRIDGE Step-Down Converter 9
8. PCB GND Planes 10
9. Sensitive Circuits 11
10. Current Sense Amplifier Placement and Routing 12
11. Demodulation Filter Components Placement and Routing 13
12. Flash IC Placement and Routing 15
13. External Crystal 15
14. Non-Sensitive Circuits 16
15. Thermal Management 16
16. PCB Footprint Design 16
17. Revision History 18

List of Figures

Figure 1. Power Blocks 3
Figure 2. Bridge Power Stage Schematic Snippet 4
Figure 3. Half-Bridge FET Bypass Capacitor Placement and Routing Recommendation (Top Layer Only) 5
Figure 4. Half-Bridge FET Bypass Capacitor Placement and Routing Recommendation (Top Layer Only), Color Coded by Node Name 6
Figure 5. BRG Components Placed Next to the P9241-G (DRV_VIN, VBRG_IN, BST_BRG1/2 Capacitors), GH/GL Series Resistor Placement, and Routing (Top Layer Only) 7
Figure 6. Schematic and Trace Snippet of Layout and Placement Recommendation for VCC5V Buck, LDO33, LDO18, and PREG CAP Components 8
Figure 7. Internal and Bottom Layout Recommendation for VCC5V Buck, LDO33, LDO18, and PREG CAP Components 8
Figure 8. VBRG Buck Regulator Schematic Snippet 9
Figure 9. VBRG Buck Regulator Schematic Snippet 9
Figure 10. GND Noise Areas and Primary Power Transfer Current Paths 10
Figure 11. Inner Layers and GND Planes (MID1, MID2, and Bottom Layer) With Key Guidance Points Highlighted 11
Figure 12. BRG Current Sense Resistor – Kelvin Sense Connections (Top Layer) 12
Figure 13. Voltage and Current Demodulation Filter Schematic 13
Figure 14. Voltage Demodulation Diodes to Demodulation Filter Layout 14
Figure 15. Flash IC Placement and Layout 15
Figure 16. External Crystal Placement 15
Figure 17. PCB Footprint Design Guidance (All Dimensions in mm) 17

© 2019 Renesas Electronics Corporation

1. Introduction

The P9241-G wireless power transmitter is an Integrated Circuit (IC) consisting of four N-Channel MOSFET gate drivers, an integrated Buck switching regulator, multiple Low Drop Out linear regulators (LDOs), wireless in-band communication demodulator, an ARM Cortex™ µProcessor, and General Purpose Inputs/Outputs (GPIOs). These devices use advanced signal processing allowing it to be fully compliant with the Wireless Power Consortium Qi (WPC) Baseline Power Profile (BPP), iOS 7.5W wireless charging for iPhone device, and 10W wireless charging for Android proprietary modes. Considering the integration of power, analog, and digital circuit blocks, the device can achieve optimal operation by following the placement and layout guidance outlined in this document. During placement and routing, there are multiple considerations and tradeoffs associated with managing the critical current paths while implementing the circuit onto a PCB.

To optimize the design, place components based on its circuit function to guarantee best performance when the schematic is implemented into a functional PCB layout. By following the instructions and guidelines in this document, efficient operation can be optimized for each circuit function simultaneously while using minimal amount of PCB area required for safe operation and power transfer. Furthermore, the thermal management of the application is an important aspect of the product's performance and must be optimized while designing the PCB and routing the connections needed for device operation.

This layout guide is intended to inform the PCB designer to place components in order of functionality and performance priority as well as critical routing connections related to its operation. There are three main categories of circuitry: Power Circuits, Noise Sensitive Circuits, and Non-Sensitive Circuits. The P9241-G PCB is recommended to be routed using at least four copper layers.

2. P9241-G Layout Quick Checklist

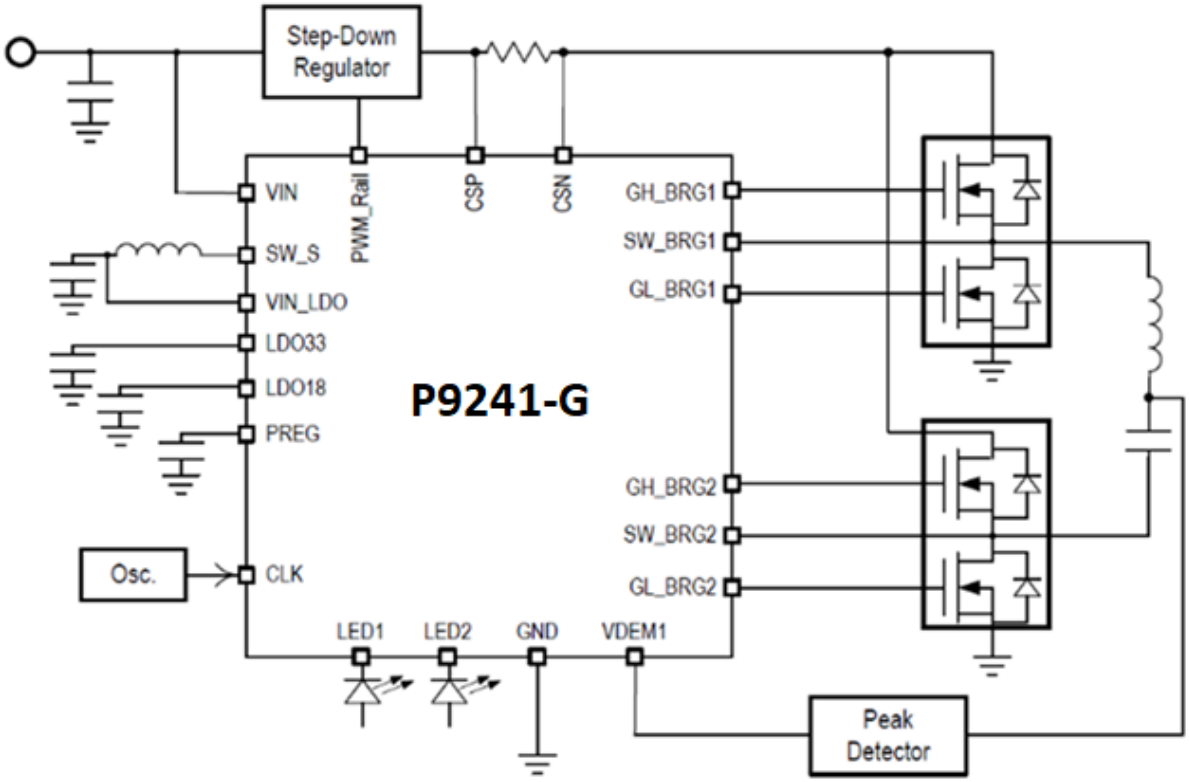
- Full Bridge FETs are placed near resonance components with bypass capacitors from HS FET drain to LS FET source of each Half-Bridge loop area and layer transitions minimized.
- Wide routing for power paths (VIN, VBRIDGE, SW_BRG1, SW_BRG2, LC Node, and GND) (50mils, or wider depending on trace length). The ratio of width to length should be ≥ 1 width unit 8 unit lengths with 1:1 being ideal.
- Bridge GND: FET source GND connections are directly connected to local bridge FET bypass capacitors, EMI filters, ZVS caps with connection to the thermal heat sinking plane at the current sense resistor followed by direct connection on the PGND plane to BUCK.
- External Voltage Regulator CIN capacitors are placed first. Use wide VIN, LX, VBRIDGE, and GND routing (75mils, or wider depending on length).
- PREG cap GND must not be shared with the VIN pin bypass cap GND. Place next to IC, routed on component side straddling pins 3 and 2.
- VIN input bypass capacitors are placed next to the IC, routed on component side straddling pins 4 and 6.
- LDO bypass capacitors require direct GND connections to the EPAD using inner or bottom layer.
- Use current sense resistor with "kelvin" connections low side current sense amplifier inputs (via appropriate resistors).
- Demodulation (Envelope Detect) filter components must be located close to the P9241-G device.
- FLASH IC is placed close to the P9241-G, and is an approved part.
- The critical components are specified with minimum parameters to comply with the Reference BOM.

© 2019 Renesas Electronics Corporation

3. Power Circuits

The main power circuits of the P9241-G wireless power transmitting solution are the step-down converter for VBRIDGE, Bridge Drivers, the external power FETs, resonance tank(s), coil select FETs, and the integrated BUCK Regulator. Secondary power circuits are the LDOs (LDO33, LDO18, PREG), and VDDIO.

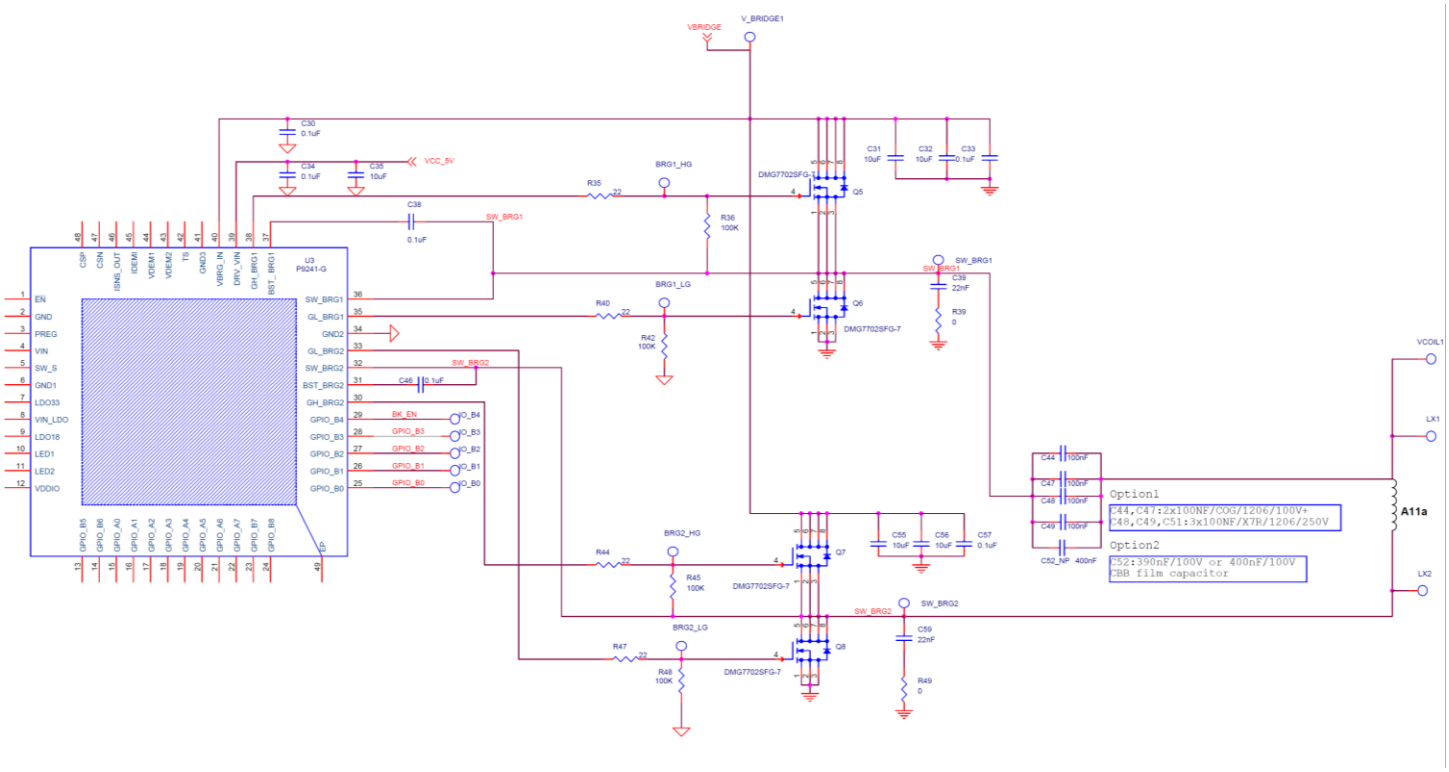
Figure 1. Power Blocks



First, based on the physical requirements of the system under design, the initial shape of the application PCB must be determined as well as the connection points for the input power supply and transmitter coils. Second, the resonance capacitors and power transfer FETs are placed in close proximity to the coils and near the center of the PCB as possible for heat sinking purposes. Third, with the coils and input power locations determined, the step-down converter power stage relative placement is made and finally, placement of the P9241-G. Orientation of the P9241-G is determined based on the ability to route connections from the Bridge driver pins to the Bridge FETs in a symmetric manner such that the trace lengths from Driver 1 and Driver 2 are similar. Referring to the schematic snippet in Figure 2 from the *P9241-G Tx Reference Design Evaluation Kit User Manual*, the placement of the components is in the following order of priority:

1. FETs (Q5, Q6, Q7, Q8), VBRIDGE Input capacitors (C31, C32, C33, C55, C56, C57)
2. Bootstrap Capacitors (C38, C46)
3. Zero-Voltage Switching (ZVS) Capacitors and respective resistors (C39, R39, C59, R49)
4. Low-Pass filter components (C30, C34)
5. DRV_VIN bypass capacitors (C34, C35), Signal Resistors (R35, R40, R44, R47).

Figure 2. Bridge Power Stage Schematic Snippet



4. Bridge Routing

Focus the key components and layout placement on proper placement and routing of the step-down converter (VBRIDGE, U1) and the ability to route with wide copper traces the connections from the Voltage Regulator output node (V_BRIDGE) to Q5, Q7 (High-side and Low-side Drains) to the current sense resistor and back to the step-down converter. The next priority is for routing the SW_BRG1/SW_BRG2 nodes to the Low-Pass filter, to the resonance tank(s), and coil select circuitry as needed to the current sense resistor (see PCB GND Planes section). Use multiple vias (at least 6 vias with 0.3mm diameter hole size, or larger) for any layer transitions on these nodes to decrease impedance on these paths. These widths will increase heat transfer as well as current carrying capacity while minimizing voltage drops due to conduction. Next, place the V_BRIDGE input capacitors so that they are centered and have minimum trace lengths from each respective capacitor plate to the connecting FET pins.

In the Figure 3, C31, C32, and C33 are placed in close proximity to Q5; while C55, C56, and C57 are placed right next to Q7. The loop area (VBRIDGE_to_drain through decoupling capacitor to BRIDGE_GND_to_source) is minimized on each half-bridge. As an alternative design choice to the P9241-G, the resonant capacitors (C44, C47, C48, and C49) can be replaced with a film capacitor (C52), which is a through-hole component. Both planes beneath the film capacitor should be expanded to allow its placement as shown in Figure 3. Do not place components beneath the film capacitor.

Figure 3. Half-Bridge FET Bypass Capacitor Placement and Routing Recommendation (Top Layer Only)

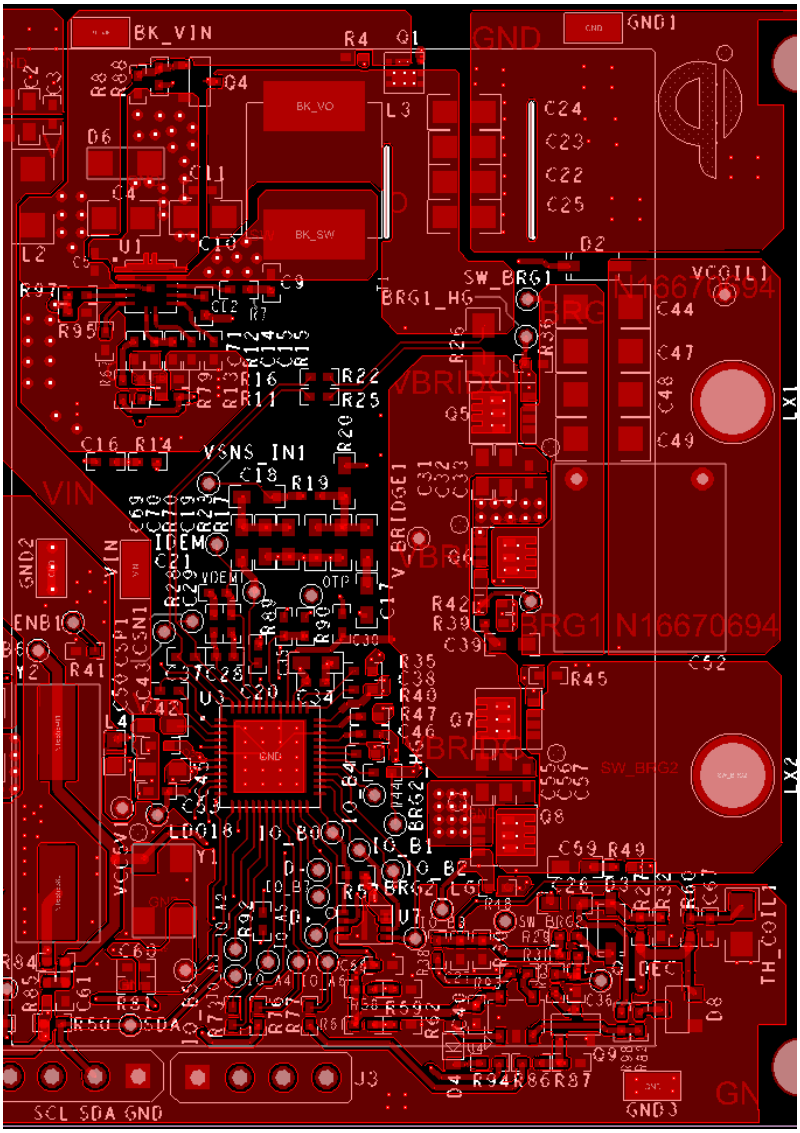
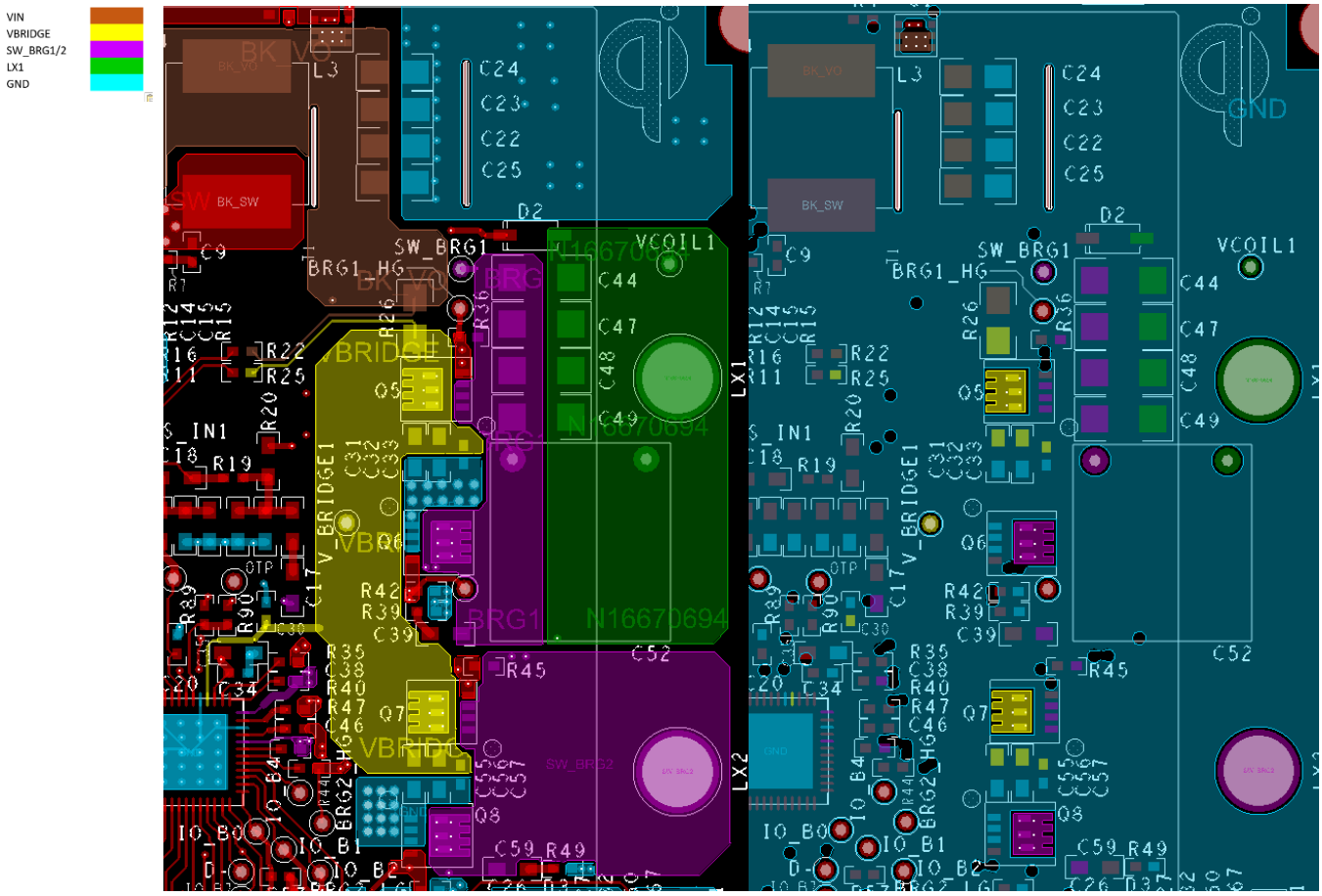


Figure 4 provides color coding and node names to help provide reference for each node to help highlight proper placement and routing.

Figure 4. Half-Bridge FET Bypass Capacitor Placement and Routing Recommendation (Top Layer Only), Color Coded by Node Name

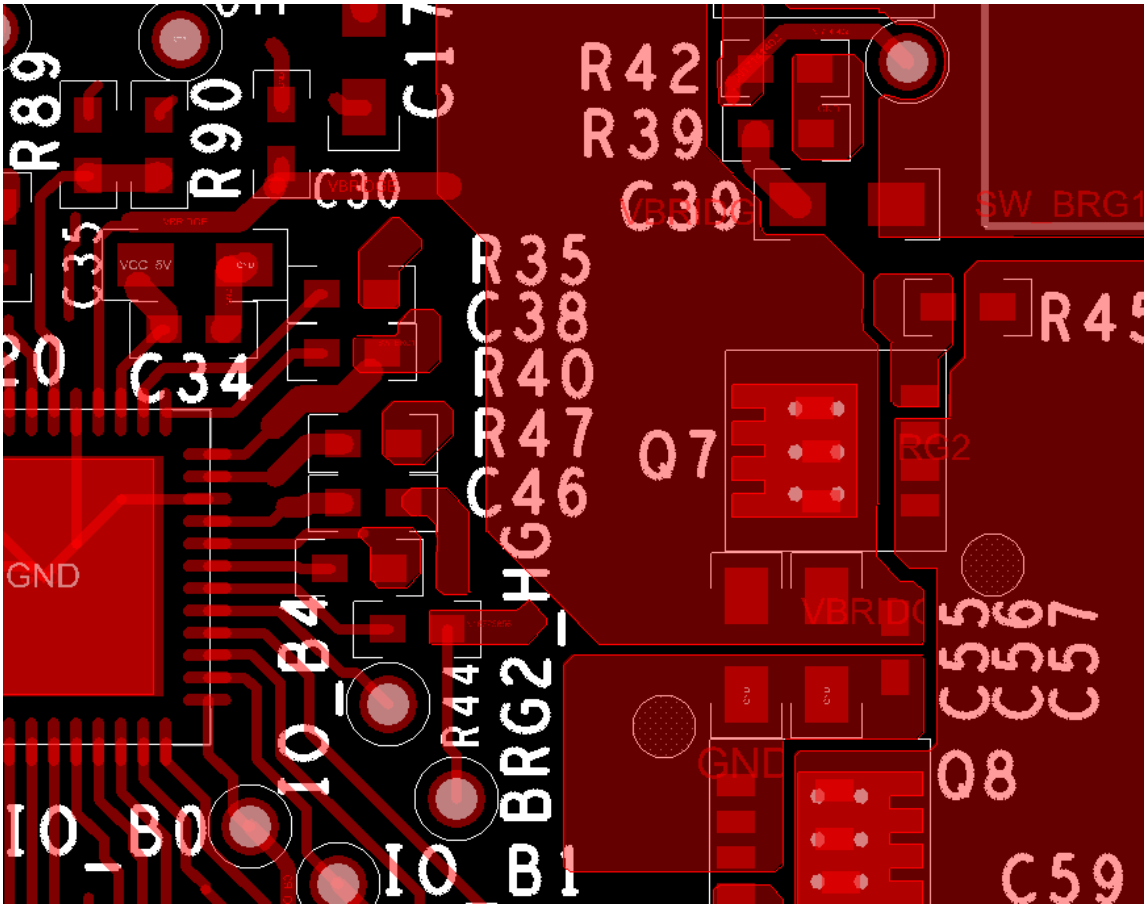


5. Bootstrap Capacitors

Place Bootstrap Capacitors (C38, C46) such that they are near the P9241-G and placed between BST_BRG1 to SW_BRG1 and BST_BRG2 to SW_BRG2, respectively. To optimize performance and continuing to proceed through placement and layout of the PCB, the next step is completing the routing of the BRG switching node (SW_BRG1/2) to the LC tank. The critical components for this portion of the layout is to maximize the copper surface area used for routing to decrease conduction losses (AC losses from skin effect and DC losses from copper foil sheet resistance); therefore, use of copper planes is highly recommended. Figure 4 highlights the recommended routing guidelines by PCB layer.

Another important aspect of the P9241-G solution involves the Zero-Voltage Switching (ZVS) Capacitors and its respective components (C39, R39, C59, and R49). These are used to control the rate of change of the BRIDGE switching nodes (SW_BRG1 and SW_BRG2) during the dead-time (time when both FETs are off and the software state is being changed, such as high-to-low or low-to-high). The dead-time is imperative to prevent shoot-through currents and the ZVS capacitors keep the voltage on those nodes from hard-switching or changing too quickly, which helps reduces EMI to radiate above the CISPR22's permissible limits. These components should be located next to the corresponding BRIDGE FETs, and have a solid connection to the low-side FET source of each switching node (see ZVS1/2 (colored-coded purple) in Figure 4 found next to BRG FETS Q5 and Q8). The final BRG components that must be placed and routed are all located next to the P9241-G device. These components are: the VBRG_IN bypass capacitors (C30), the Gate driver High and Low (GH_BRG1/2, GL_BRG1/2) series signal resistors (R35, R40, R44, and R47), and the Bootstrap high-side gate driver capacitors (C38 and C46). All of these components should be placed near the P9241-G respective pins as shown in Figure 5.

Figure 5. BRG Components Placed Next to the P9241-G (DRV_VIN, VBRG_IN, BST_BRG1/2 Capacitors), GH/GL Series Resistor Placement, and Routing (Top Layer Only)



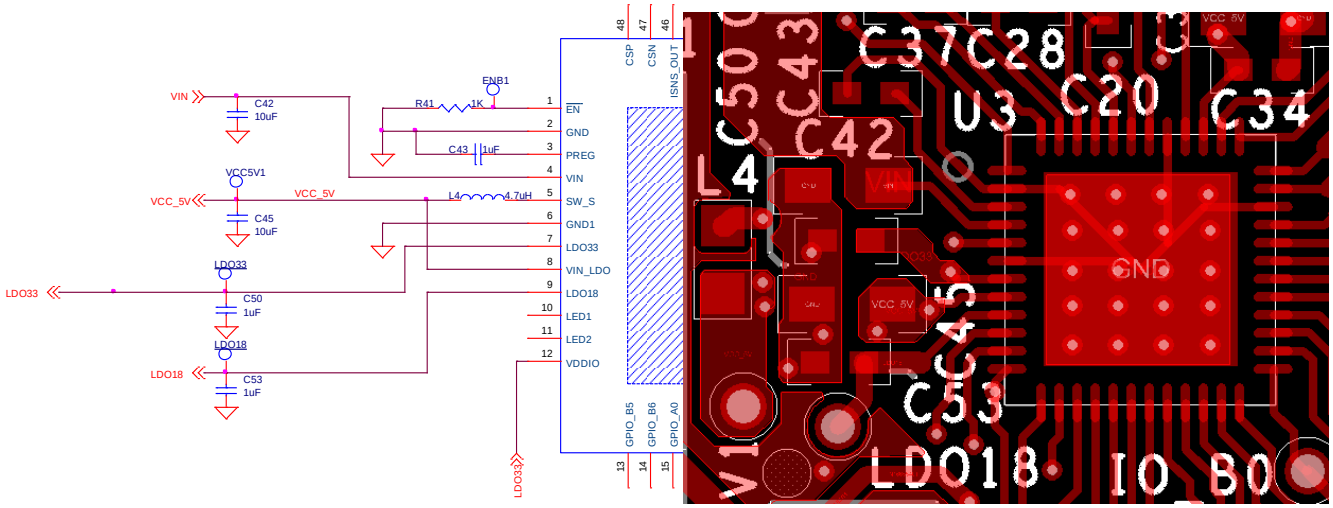
The gate drive signals can be routed with 5-8 mils trace width and should avoid being run in parallel with power switching connections (SW_BRG1/2 and BST_BRG1/2), or should have GND plane isolation from those signals to eliminate any coupling or noise issues.

© 2019 Renesas Electronics Corporation

6. VCC5V BUCK and LDO Routing

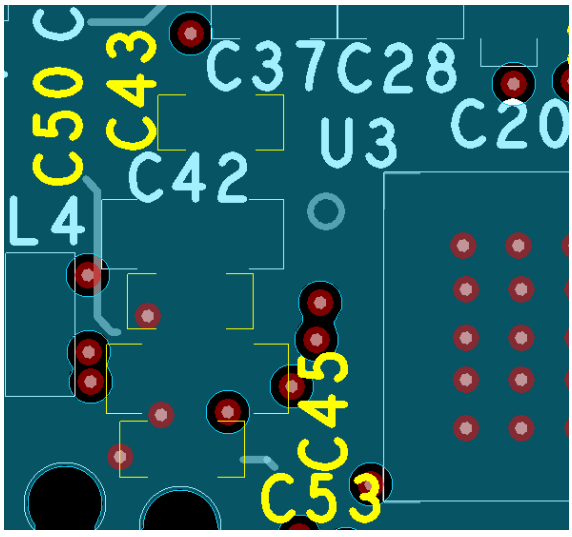
The internal VCC5V BUCK and LDO components routing are recommended to follow the guidance in placement and routing in Figure 6. Although the current flow can be small, solid wide connections are necessary for the connections. Any heat developed within the IC or on the PCB will be rapidly dissipated.

Figure 6. Schematic and Trace Snippet of Layout and Placement Recommendation for VCC5V Buck, LDO33, LDO18, and PREG CAP Components



The components listed in Figure 6 are used by the P9241-G for controlling internal bias, digital logic, the micro-processor, and power transfer. Place and route these components as shown in Figure 6. Route the VIN capacitors (C42) and VCC5V output capacitors (C45) GND connections directly to the GND pin. The LDO18 and LDO33 bypass capacitors must have a direct connection from the GND plates of these capacitors to the EPAD. The EPAD is the GND return for the LDOs and the internal micro-processor. Route the SW_S trace as short as possible to the inductor L4 on the next available internal layer adjacent to the P9241-G. This trace can be extended to make room for the more critical VIN capacitors to be placed closer to the respective P9241-G pins. The connections to PREG CAP (C43) should be short and not to be shared with the GND return used for the VIN and VCC5V capacitors. The connections to the PREG CAP (C43) can be fully routed on the P9241-G side of the PCB and should have a good connection to GND (pin 2) pin.

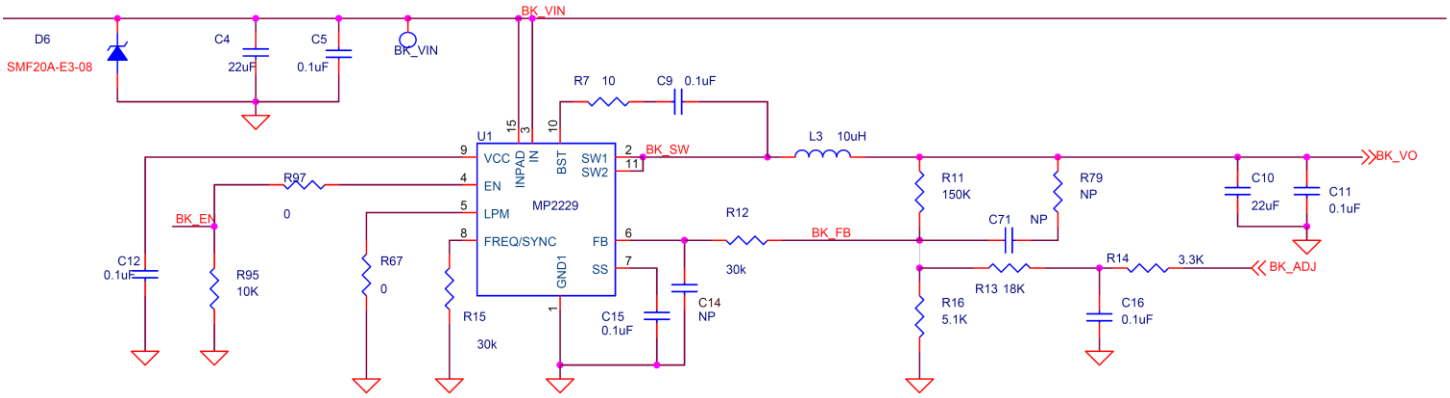
Figure 7. Internal and Bottom Layout Recommendation for VCC5V Buck, LDO33, LDO18, and PREG CAP Components



7. VBRIDGE Step-Down Converter

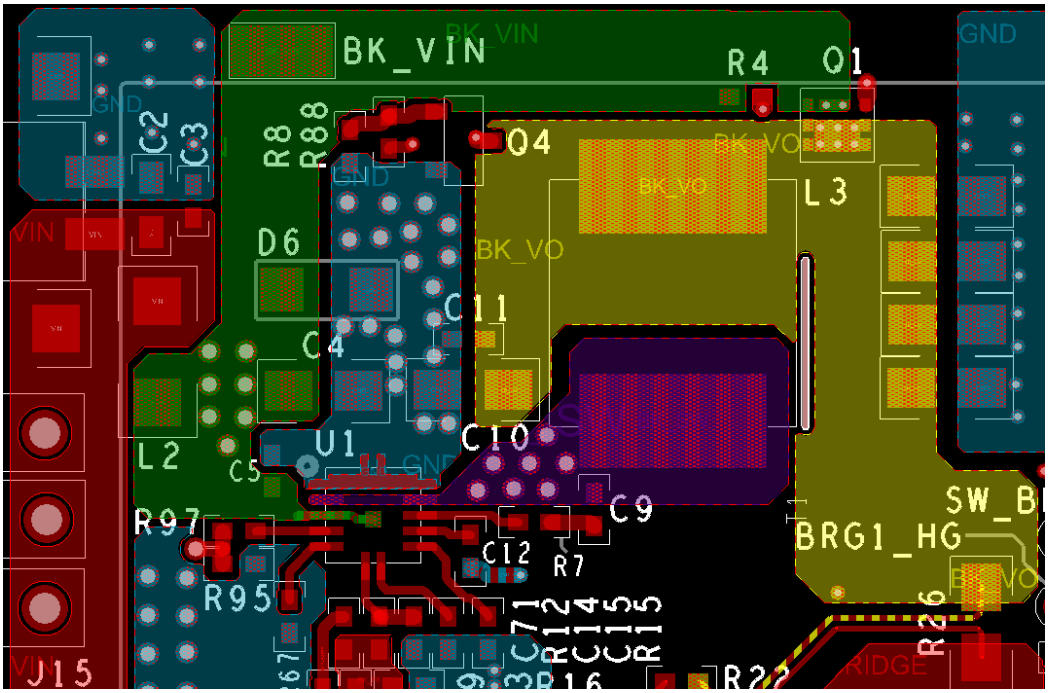
The VBRG regulator (U1) is the primary power control for wireless power and the voltage level is dynamically controlled by the P9241-G using the GPIO_B7 (pin 23) connection to the feedback node using a low-pass filtered adjustable Pulse Width Modulated (PWM) square wave signal. After connecting to a receiver (Rx), the P9241-G adjusts the transmitted power level by changing VBRIDGE according to the Control Error Packets (CEPs) received by the Rx. The typical step-down converter used as the VBRIDGE control with the P9241-G is the MP2229 as shown in Figure 8.

Figure 8. VBRG Buck Regulator Schematic Snippet



Typically with step-down converter designs, the most important capacitors are the input bypass capacitors (C4 and C5) due to the high (di/dt) change in current pulses. These bypass components must be placed close to MP2229 (U1) to help filter and dampen unwanted AC noise entering into VBRIDGE, which provides the power line for the Switching Nodes. The GND plane is shared in the center for input bypass capacitors and output capacitors. Under the IC layers, L1 and L2 are both kept primarily as solid GND planes to maximize heat dissipation and current condition at minimal impedance, and multiple GND vias are placed next to the MP2229 GND pins. L1 (common mode choke) and L2 (bead) components improve EMI performance.

Figure 9. VBRG Buck Regulator Schematic Snippet



8. PCB GND Planes

It is important to properly allow current flow to and from each circuit component of the solution. The primary conduction path is from the main power connector positive terminal, to the VBRIDGE voltage regulator, to the resonance tank and back through GND planes, to each successive current source (GND is referred to as the return current path) when completing each current loop.

It is highly recommended to allocate one complete layer of every PCB design to be a solid GND plane so current flow return to its source is unobstructed from mechanical blocks, such as cut planes in the form of routed traces or slots in the physical PCB. There are two primary GND return loops that exist in the P9241-G:

- Tx: VBRG regulator and the WPC resonance tank
- Flash IC to P9241-G GND, and other sub-circuits in the Quiet GND area

Figure 10. GND Noise Areas and Primary Power Transfer Current Paths

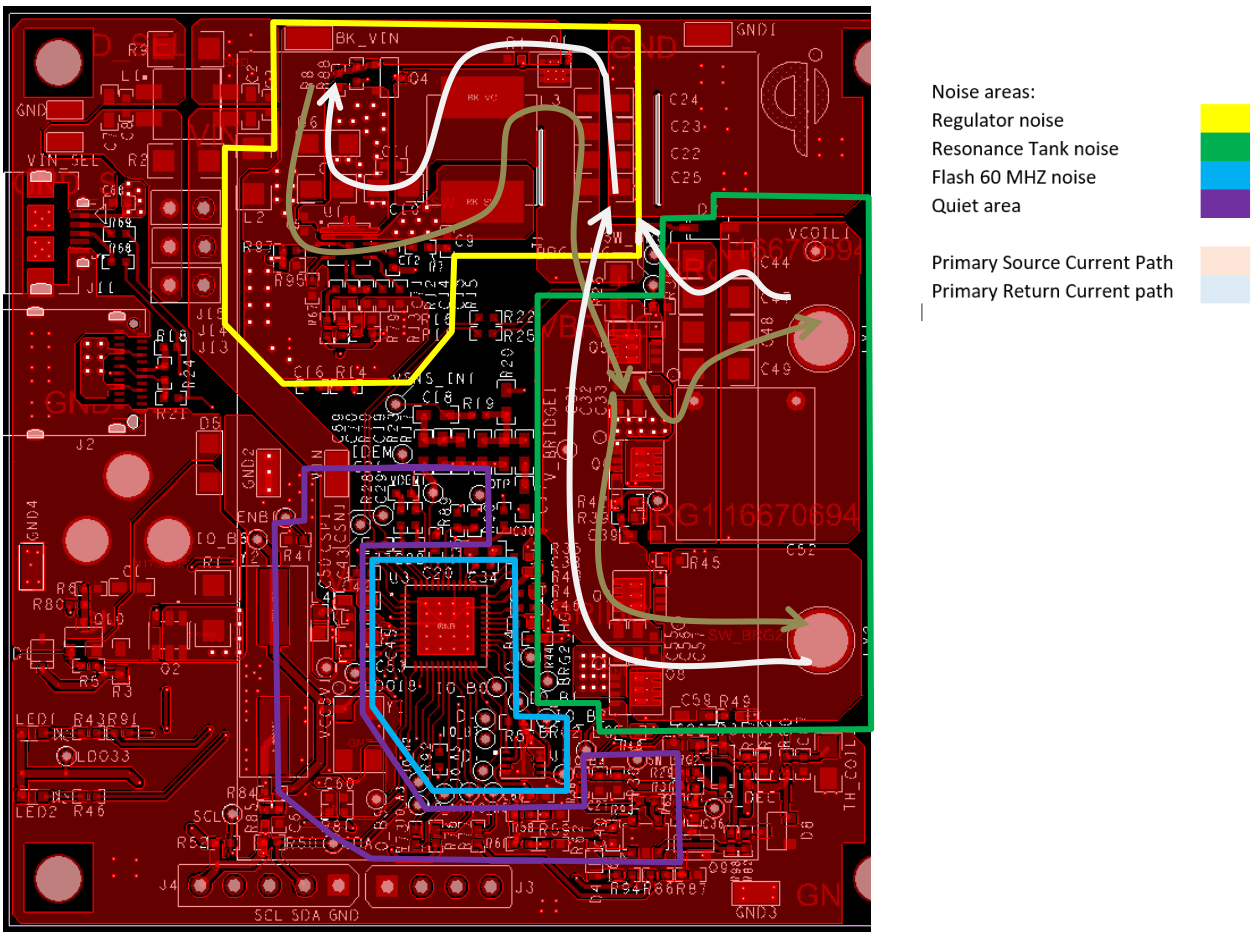
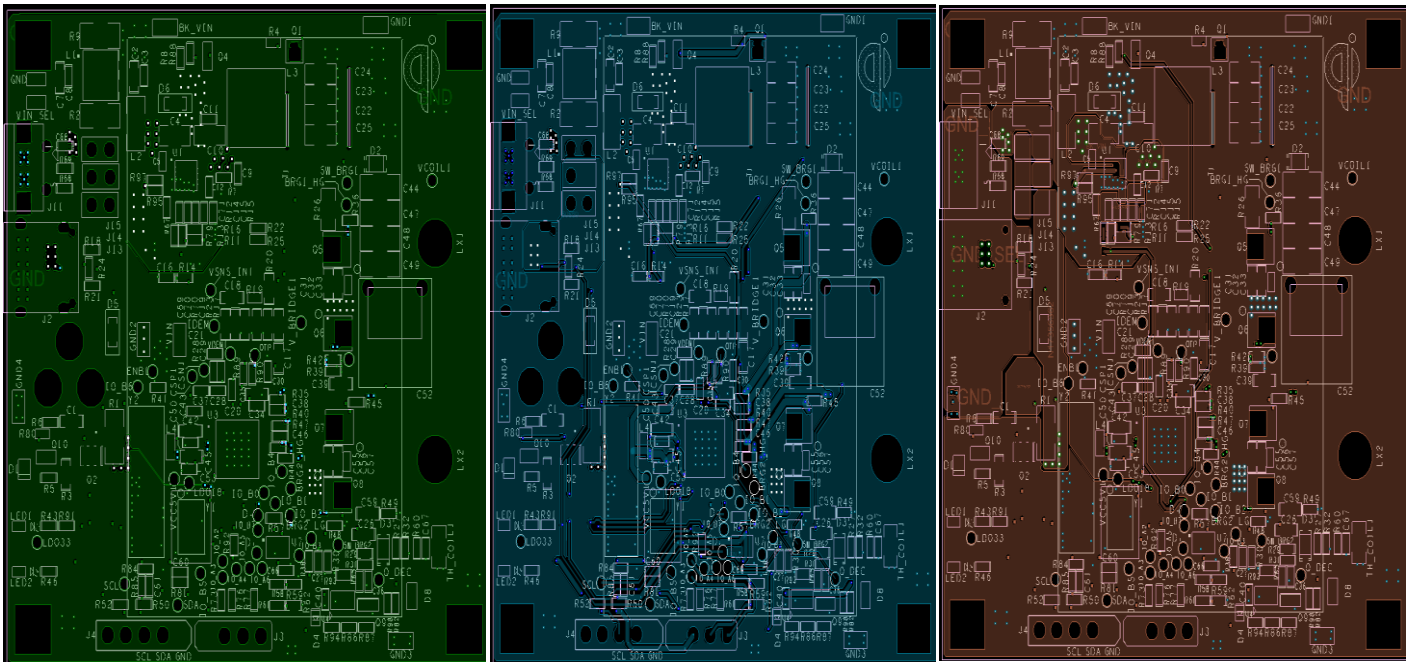


Figure 11. Inner Layers and GND Planes (MID1, MID2, and Bottom Layer) With Key Guidance Points Highlighted



Inner Layer (L1): Solid GND Plane

Inner Layer (L2):
- BRG FET PGND vias
- P9241-G EPAD thermal vias
- GND plane poured after routing

Bottom Layer:
- BRG FET thermal plane
- P9241-G EPAD thermal vias
- GND plane poured after routing

Some key points regarding the full-board screen captures shown above include:

- Use of small internal planes for heat dissipation for connections to the BRG FETs and power inductors (it is recommended to use multiple vias to improve thermal connection and to lower impedance)
- Verify that the primary source and return currents have solid copper planes for conduction
- Add GND planes on every layer after routing and placement to improve thermal performance of the entire PCB

9. Sensitive Circuits

The sensitive circuits refer to noise sensitive circuits that are placed in an area of the PCB that is not expected to have circulating current flow (VBRG Voltage Regulator, WPC resonance tank noise) and are preferred to be placed away from the primary current paths (Source and Return) in the “Quiet” GND area Figure 10. The following functional circuit blocks should be placed near the P9241-G or in the quiet GND area as possible or have special routing requirements:

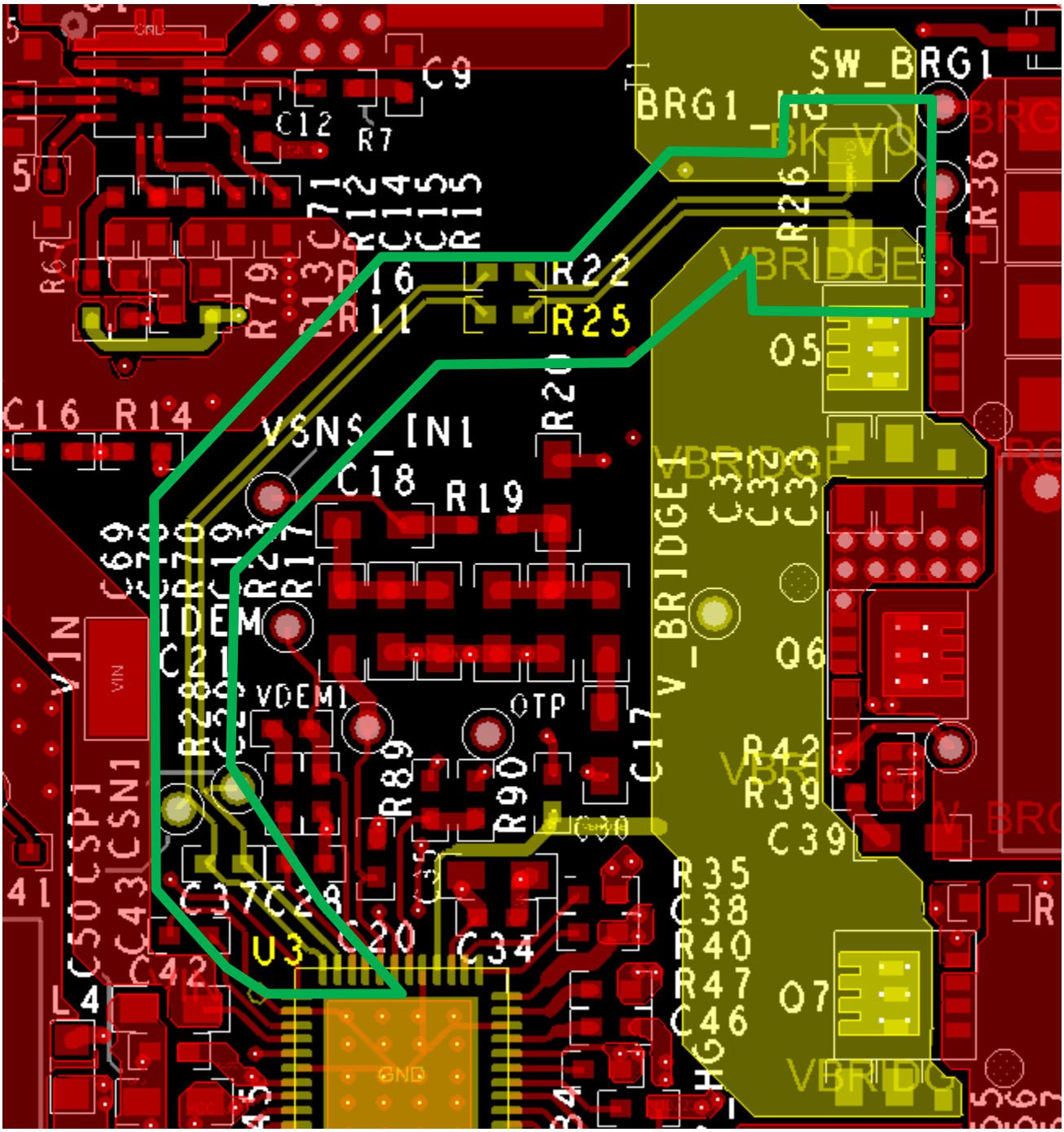
- Current Sense with kelvin sense connections (4-wire) voltage sensing at current sense resistor (R26)
  - Used for BRG current sensing for Power loss FOD, OCP, and Demodulation based on coil current variation
- Demodulation filter components
- VCOIL peak voltage sensing circuitry
  - Optional: Used for Q measurement, Tx coil OVP, maximum power limiting
- Flash IC (must be placed next to P9241-G)
- SMD Crystal (Y1)

© 2019 Renesas Electronics Corporation

10. Current Sense Amplifier Placement and Routing

Place Current Sense Resistor (R26) in close proximity to the BK_VO to VBRIDGE planes. Proceed with the “kelvin” connections by following in equal and parallel distance in its route to CSP (pin 48) and CSN (pin 47). Keep the trace length to minimal to ensure optimal sense points and performance to the amplifier inputs (see Figure 12).

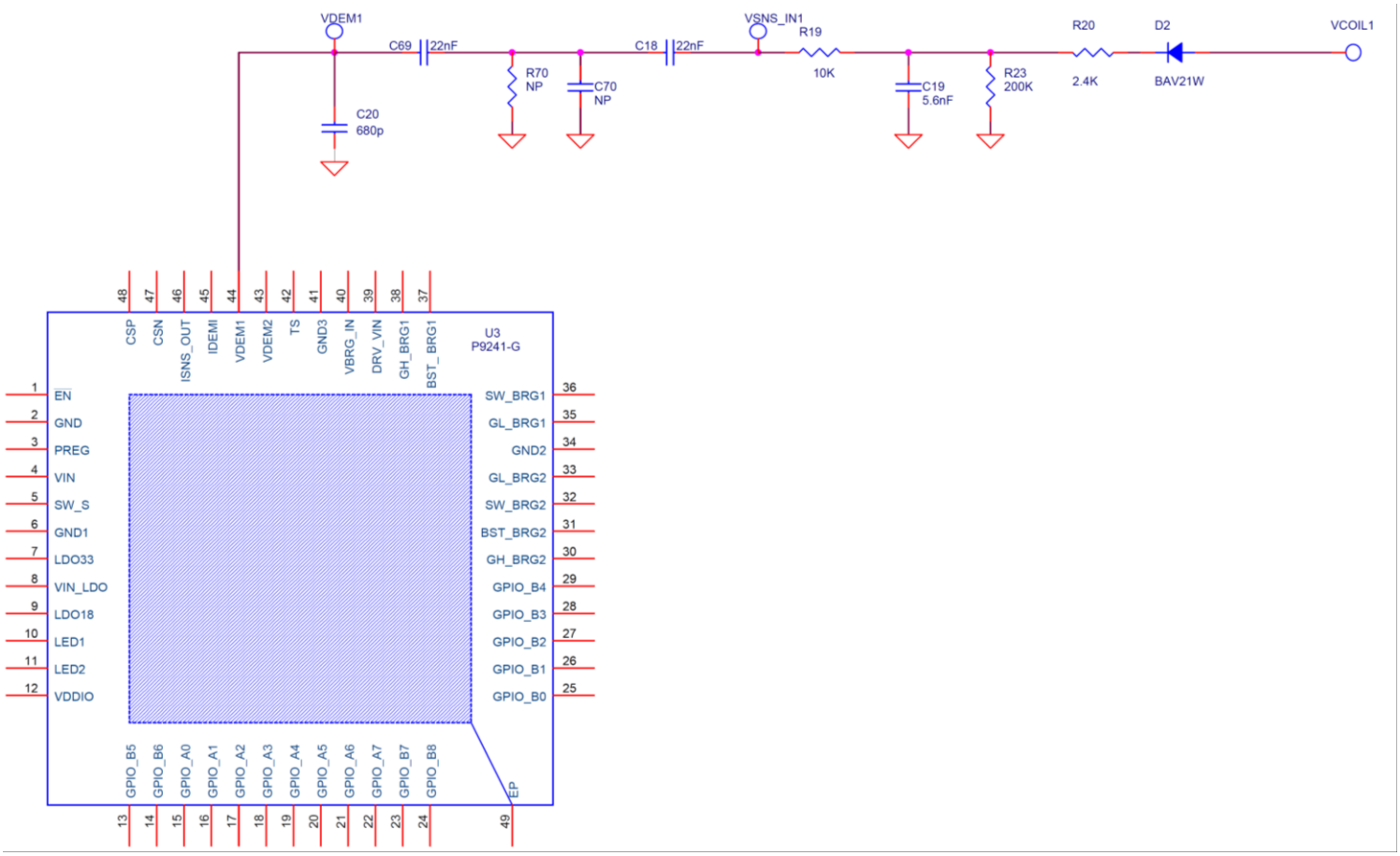
Figure 12. BRG Current Sense Resistor – Kelvin Sense Connections (Top Layer)



11. Demodulation Filter Components Placement and Routing

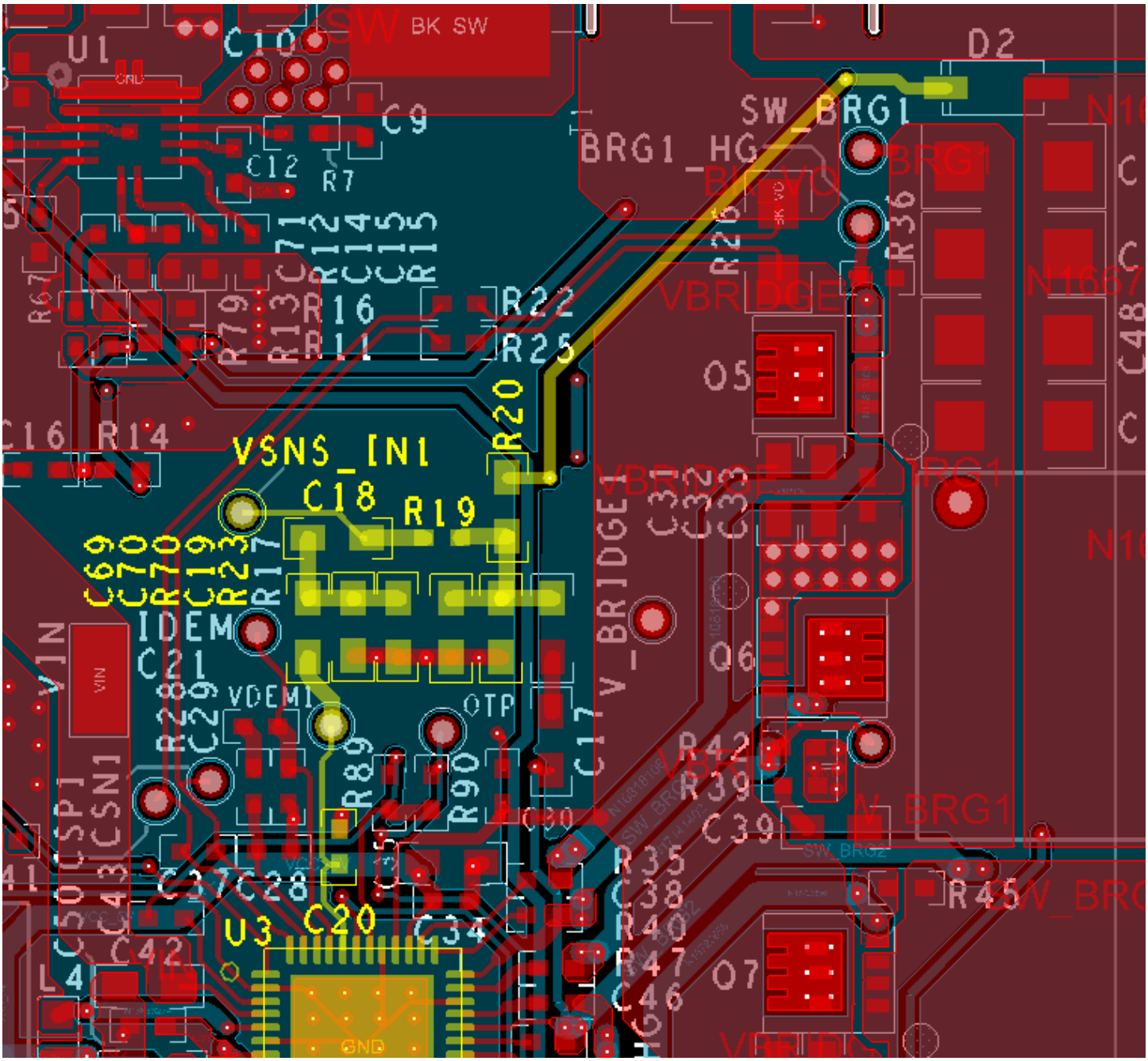
When routing the voltage demodulation filter components, it is recommended to place the diode peak detectors (D2) near the LC nodes and route the “DC” peak voltage signals (cathodes from diodes D2) to the remaining DEMOD filter components, which should be placed close to the P9241-G. The following schematic shows the demodulation voltage filter components.

Figure 13. Voltage and Current Demodulation Filter Schematic



Components C20, C69, C70, and R70 should be located near the associated P9241-G pins and be in the “Quiet” GND area.

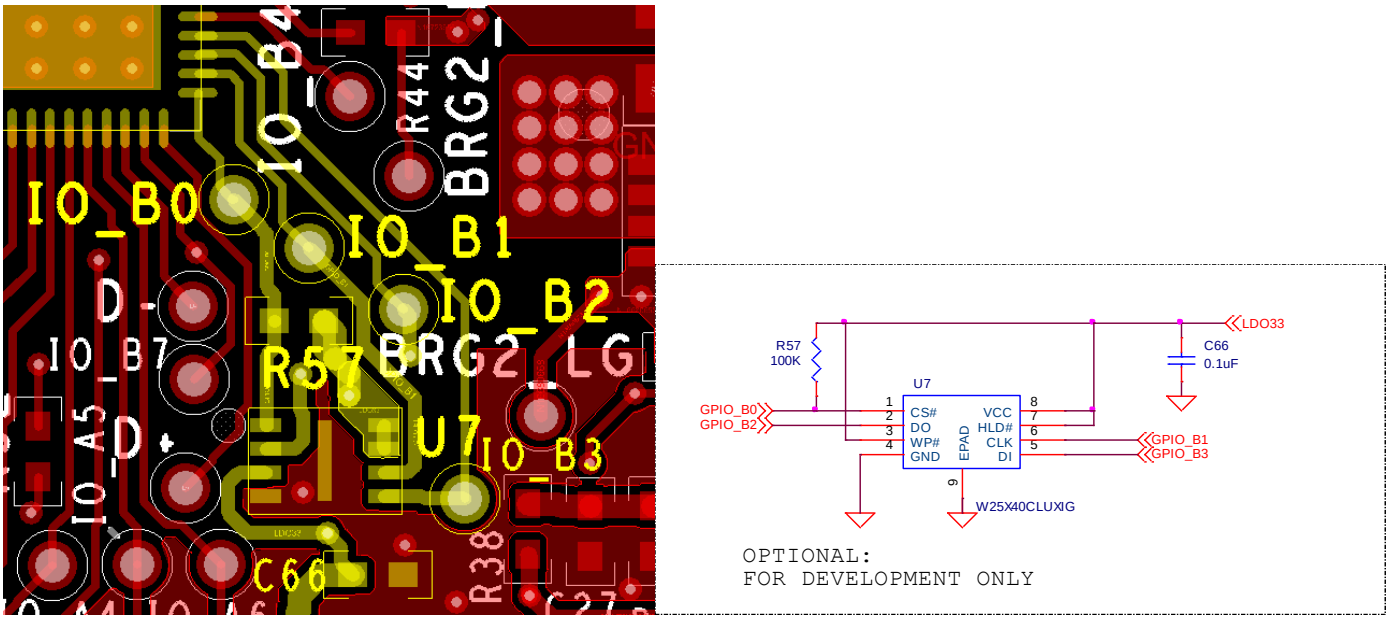
Figure 14. Voltage Demodulation Diodes to Demodulation Filter Layout



12. Flash IC Placement and Routing

Due the high data speed of the Flash IC (U7), this component is recommended to be placed next to the P9241-G. Be sure the FLASH IC GND pins to P9241-G EPAD connections are directly connected to each other.

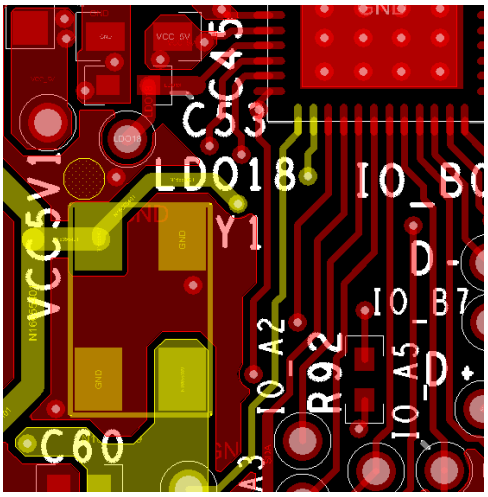
Figure 15. Flash IC Placement and Layout



13. External Crystal

Trace connecting the crystal of the IC (Y1) should be as short and wide as possible to help reduce parasitic inductance. This component is recommended to be placed next to P9241-G and in the “quiet area” as shown in Figure 10.

Figure 16. External Crystal Placement



14. Non-Sensitive Circuits

The remaining components and pins that have not been mentioned are regarded as non-sensitive circuits and the placement and routing of these nodes is not considered critical to performance but can be required for functionality. All remaining connections and components do not have placement demands (can be placed anywhere) and they simply need to be connected. In most cases 0.127mm (5mils) wide traces and any method of connectivity will suffice if the circuit is completed according to the schematic.

15. Thermal Management

The thermal management of the P9241-G is of critical importance for successful deployment of the product to reduce heat generation of the total design and to prevent localized heating of any components. To optimize the design, thermally and electrically wide continuous copper planes are highly recommended on every power node. Furthermore, filling all layers of the PCB with copper planes for current conduction and heat dissipation will dramatically improve the thermal performance. Use of multiple vias and EPADs is an effective way to spread localized heating through the PCB. An optimally designed PCB will have even temperature across the board surface. To promote higher heat dissipation, it is recommended to use low DCR, RDSon, and NP0/C0G type capacitors for all components listed in the “Power Circuits” section of this document (following the P9241-G reference components is highly recommended).

For the P9241-G PCB footprint design, an array of GND vias is recommended in the EPAD 4x6 or 6x6 arrays are preferred with 0.3mm (12mil) diameter hole sizes and connections to all PCB layers. It is not recommended to use less than four layers when designing the PCB. Conduct thermal inspection of each design and add additional heat spreading copper planes to spread heat away from higher temperature components to reduce localized heating or prevent components from overheating when used without air conditioning on hot summer days. When designed properly, the PCB solution can operate in up to 105°C ambient air without violating the thermal limits of any components called out in the reference design.

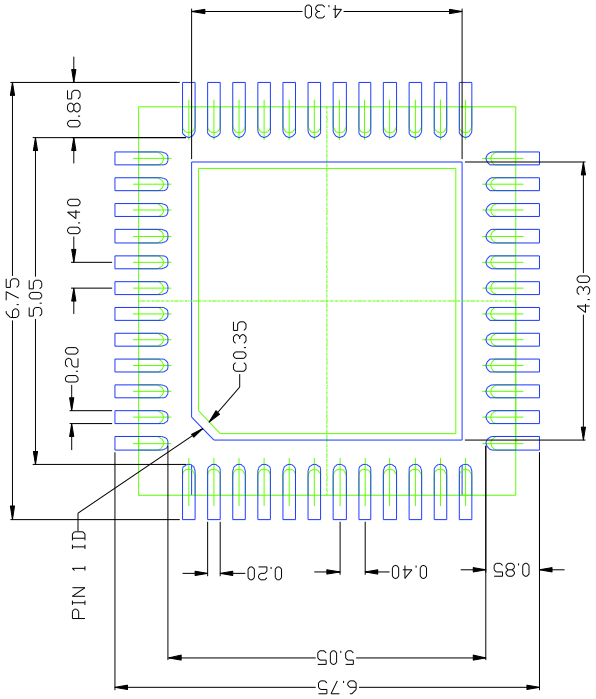
16. PCB Footprint Design

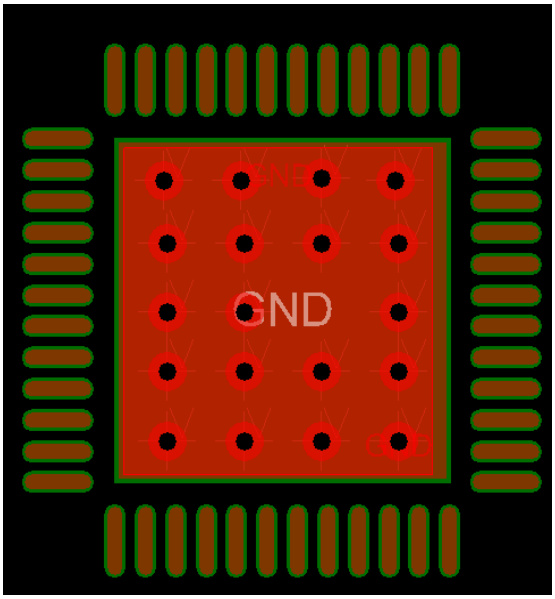
The PCB footprint can have a substantial impact on reliability and production yield. To avoid placement and soldering issues during assembly and product deployment, it is recommended to allow sufficient copper landing pads to be exposed, implement solder mask (2.5 to 3 mils applied between every pin with 1 to 2 mils clearance beyond the exposed copper), and match the pastemask openings to the exposed copper of each pin. To apply a proper amount of solder paste to the IC, fine pitch application lead-free no clean or water soluble solder pastes are recommended and use of stencil thicknesses of approximately 4mils. Furthermore, PCB cleaning is highly recommended after the reflow process is completed.

Reference the P9241-G datasheet during footprint design; Figure 17 can be used for details pertaining to landing pad and solder masking recommendations. The exposed pin (etch) used to solder the IC to the PCB must be at least as large as the physical pin of the package. It is highly recommended to add some additional length extending away from the IC from each pin (~5mils) to promote proper soldering. It is imperative to include soldermask around each pin to eliminate potential solder shorts during assembly and reflow. It is also not suggested to pour continuous copper planes under the pin area, use a short (up to 0.127mm or 5mils) trace matching the pin width to connect pins to the planes beyond the perimeter of the package. The EPAD must contain solderpaste openings for proper electrical and mechanical connection to the PCB and they should be four separated squares approximately 1.5 × 1.5 mm per side with 2.26mm pitch centered within the EPAD.

© 2019 Renesas Electronics Corporation

Figure 17. PCB Footprint Design Guidance (All Dimensions in mm)





17. Revision History

Revision Date	Description of Change
February 15, 2019	Initial release.

IMPORTANT NOTICE AND DISCLAIMER

RENESAS ELECTRONICS CORPORATION AND ITS SUBSIDIARIES (“RENESAS”) PROVIDES TECHNICAL SPECIFICATIONS AND RELIABILITY DATA (INCLUDING DATASHEETS), DESIGN RESOURCES (INCLUDING REFERENCE DESIGNS), APPLICATION OR OTHER DESIGN ADVICE, WEB TOOLS, SAFETY INFORMATION, AND OTHER RESOURCES “AS IS” AND WITH ALL FAULTS, AND DISCLAIMS ALL WARRANTIES, EXPRESS OR IMPLIED, INCLUDING, WITHOUT LIMITATION, ANY IMPLIED WARRANTIES OF MERCHANTABILITY, FITNESS FOR A PARTICULAR PURPOSE, OR NON-INFRINGEMENT OF THIRD-PARTY INTELLECTUAL PROPERTY RIGHTS.

These resources are intended for developers who are designing with Renesas products. You are solely responsible for (1) selecting the appropriate products for your application, (2) designing, validating, and testing your application, and (3) ensuring your application meets applicable standards, and any other safety, security, or other requirements. These resources are subject to change without notice. Renesas grants you permission to use these resources only to develop an application that uses Renesas products. Other reproduction or use of these resources is strictly prohibited. No license is granted to any other Renesas intellectual property or to any third-party intellectual property. Renesas disclaims responsibility for, and you will fully indemnify Renesas and its representatives against, any claims, damages, costs, losses, or liabilities arising from your use of these resources. Renesas' products are provided only subject to Renesas' Terms and Conditions of Sale or other applicable terms agreed to in writing. No use of any Renesas resources expands or otherwise alters any applicable warranties or warranty disclaimers for these products.

(Disclaimer Rev.1.01 Jan 2024)

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-us/.

Trademarks

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

© 2024 Renesas Electronics Corporation. All rights reserved.