



REALTEK

NOT FOR PUBLIC RELEASE

**RTL8211E-CG
RTL8211EG-CG**

INTEGRATED 10/100/1000 GIGABIT ETHERNET TRANSCEIVER

LAYOUT GUIDE

(CONFIDENTIAL: Development Partners Only)

Rev. 1.0

13 August 2010

Track ID: JATR-2265-11



REALTEK

Realtek Semiconductor Corp.

No. 2, Innovation Road II, Hsinchu Science Park, Hsinchu 300, Taiwan

Tel.: +886-3-578-0211. Fax: +886-3-577-6047

www.realtek.com

COPYRIGHT

©2010 Realtek Semiconductor Corp. All rights reserved. No part of this document may be reproduced, transmitted, transcribed, stored in a retrieval system, or translated into any language in any form or by any means without the written permission of Realtek Semiconductor Corp.

DISCLAIMER

Realtek provides this document “as is”, without warranty of any kind. Realtek may make improvements and/or changes in this document or in the product described in this document at any time. This document could include technical inaccuracies or typographical errors.

TRADEMARKS

Realtek is a trademark of Realtek Semiconductor Corporation. Other names mentioned in this document are trademarks/registered trademarks of their respective owners.

USING THIS DOCUMENT

This document is intended for the software engineer’s reference and provides detailed programming information.

Though every effort has been made to ensure that this document is current and accurate, more information may have become available subsequent to the production of this guide.

REVISION HISTORY

Revision	Release Date	Summary
1.0	2010/08/13	First release.

Table of Contents

1.	INTRODUCTION	1
2.	DESIGN AND LAYOUT GUIDE	2
2.1.	PLACEMENT	2
2.2.	MAGNETICS	3
2.3.	CRYSTAL/OSC.....	3
2.4.	FERRITE BEADS AND DE-COUPLING CAPACITORS	3
3.	SIGNAL AND TRACE ROUTING	4
3.1.	RX CLOCK (RXC)	4
3.2.	MDI SIGNALS	5
3.3.	GMII (MII)/RGMII SIGNALS.....	6
3.4.	POWER SIGNALS	8
4.	PCB STACK-UP.....	9
4.1.	OVERVIEW	9
4.2.	FOUR-LAYER STACK-UP (MOTHERBOARD APPLICATION)	9
4.3.	SIX-LAYER STACK-UP (NOTEBOOK APPLICATION)	10
4.4.	EIGHT-LAYER STACK-UP (NOTEBOOK APPLICATION).....	11
5.	GROUND PLANE LAYOUT	12
5.1.	FOUR-LAYER BOARD GROUND PLANE LAYOUT (TYPICAL APPLICATION).....	13
5.1.1.	Ground Plane Layer 1 Layout.....	14
5.1.2.	Ground Plane Layer 2 Layout.....	14
5.1.3.	Ground Plane Layer 3 Layout.....	15
5.1.4.	Ground Plane Layer 4 Layout.....	15
6.	POWER PLANE LAYOUT.....	16
6.1.	POWER PLANE LAYER 1 LAYOUT.....	18
6.2.	POWER PLANE LAYER 4 LAYOUT.....	18
7.	CENTER-TAPPING	19
8.	LED PINS.....	20
9.	SWITCHING REGULATOR.....	21
9.1.	PCB LAYOUT.....	21
10.	PARTS RECOMMENDATIONS.....	22
10.1.	10/100/1000M MAGNETIC	22
10.2.	RESISTORS	22
10.3.	CAPACITORS	22
10.4.	POWER INDUCTOR.....	23
10.5.	RJ-45 JACK	23
11.	SPECIAL NOTES.....	23

List of Figures

FIGURE 1. GMII/RGMII AND MDI PLACEMENT	2
FIGURE 2. RX CLOCK	4
FIGURE 3. MDI SIGNALS	5
FIGURE 4. SIGNAL TRACE	5
FIGURE 5. SIGNAL TRACE ANGLES	6
FIGURE 6. R1C1/R2C2 PLACEMENT	7
FIGURE 7. CLOCK AND HEAT SINK	7
FIGURE 8. CLOCK AND I/O TRACE	8
FIGURE 9. RESERVE 2.5V POWER FOR GMII/RGMII USE	8
FIGURE 10. FOUR-LAYER STACK-UP	9
FIGURE 11. SIX-LAYER STACK-UP (A)	10
FIGURE 12. SIX-LAYER STACK-UP (B)	10
FIGURE 13. SIX-LAYER STACK-UP (C)	11
FIGURE 14. EIGHT-LAYER STACK-UP	11
FIGURE 15. GROUND PLANE LAYOUT-1	12
FIGURE 16. GROUND PLANE LAYOUT-2	12
FIGURE 17. GROUND PLANE SEPARATION	13
FIGURE 18. GROUND PLANE LAYER 1 LAYOUT	14
FIGURE 19. GROUND PLANE LAYER 2 LAYOUT	14
FIGURE 20. GROUND PLANE LAYER 3 LAYOUT	15
FIGURE 21. GROUND PLANE LAYER 4 LAYOUT	15
FIGURE 22. POWER PLANE	16
FIGURE 23. DECOUPLED CAPACITOR EXAMPLE	16
FIGURE 24. POWER SOURCE DISTRIBUTION	17
FIGURE 25. POWER PLANE LAYER 1 LAYOUT	18
FIGURE 26. POWER PLANE LAYER 4 LAYOUT	18
FIGURE 27. CENTER-TAPPING	19
FIGURE 28. LED PINS	20
FIGURE 29. SWITCHING REGULATOR	21

1. Introduction

The Realtek RTL8211E-CG/RTL8211EG-CG is a highly integrated Ethernet transceiver that complies with 10Base-T, 100Base-TX, and 1000Base-T IEEE 802.3 standards. It provides all the necessary physical layer functions to transmit and receive Ethernet packets over CAT 5 UTP cable or CAT 3 UTP (10Mbps only) cable.

The RTL8211E/RTL8211EG uses state-of-the-art DSP technology and an Analog Front End (AFE) to enable high-speed data transmission and reception over UTP cable. Functions such as Crossover Detection & Auto-Correction, polarity correction, adaptive equalization, cross-talk cancellation, echo cancellation, timing recovery, and error correction are implemented in the RTL8211E/RTL8211EG to provide robust transmission and reception capabilities at 10Mbps, 100Mbps, or 1000Mbps.

Data transfer between MAC and PHY is via the Reduced Gigabit Media Independent Interface (RGMI) for 1000Base-T, 10Base-T, and 100Base-TX.

2. Design and Layout Guide

System designers should follow basic rules in layout and placement, general termination, power supply filtering, plane partitioning, and EMI reduction in order to optimize designs that use the RTL8211E/RTL8211EG. Following these rules will greatly contribute to a properly functioning hardware system.

This guide has the following goals:

- (1) Create a low-noise, power-stable environment
- (2) Reduce the degree of EMI/EMC and their influence on the RTL8211E/RTL8211EG
- (3) Simplify the task of routing signal traces

In order to achieve maximum performance using the RTL8211E/RTL8211EG, good design practices are required throughout the process. The following are some recommendations for implementing a high performance system.

2.1. Placement

- The RTL8211E/RTL8211EG must be placed as close as possible to the MAC (less than 2.5 inches)
- The resistor connected to the RSET pin should be placed close to the RTL8211E/RTL8211EG (less than 800mils), and as far away as possible from signal traces (e.g., VRRREG, REG_OUT, MDI0+/-, MDI1+/-, etc.) and clock signals (50mils min.)
- For good EMI performance, the PHY device must be placed as close as possible to the MAC (less than 2.5 inches). If the MAC is placed on the top layer, then the PHY should be put on the bottom layer to avoid heat sink coupling

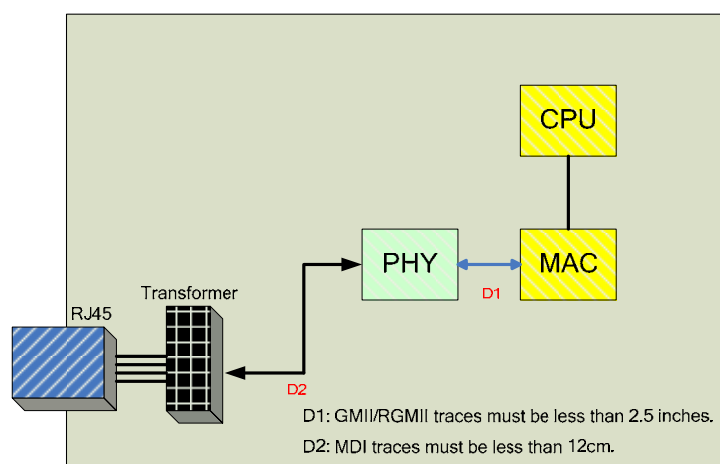


Figure 1. GMII/RGMII and MDI Placement

2.2. Magnetics

- The 10/100/1000M magnetics should be placed as close as possible to the RJ-45 connector. The MDI traces must be less than 12cm.
- The magnetics device, or devices with magnetic fields, should be separated and mounted at 90 degrees to each other.

2.3. Crystal/OSC

- Following the 3W rule, the Crystal should be placed at least three times its own width from I/O ports, important or high frequency signal traces (TX, RX, power), and magnetics.
- The outer shield of the Crystal requires good grounding to avoid induction of EMC/EMI. The retaining straps of the OSC, if any, need good grounding as well.

2.4. Ferrite Beads and De-Coupling Capacitors

Every PCB design has its unique noise coupling behavior. Ferrite beads are used to suppress power noise. System designers are suggested to provide the option to replace the ferrite beads with 0Ω resistors. Decoupling capacitors must be placed close to the power pins, such that the distance from the IC power pin to the capacitor is less than 200mils.

3. Signal and Trace Routing

Noise, ringing, and data lines must be controlled with proper termination. Power supply pins should be protected by proper filtering techniques. Good routing of traces can reduce propagation delay, crosstalk, and high frequency noise. It will also improve the signal quality to the receiver and reduce transmit signal losses.

3.1. *RX Clock (RXC)*

- Clock signal traces should be as short and wide as possible.
- Route the clock traces adjacent to an unbroken ground or power plane. Minimize vias and layer changes.
- Place the filter network (Figure 2) as close to the driving source (RXC pin) as possible. The RXC filter network minimizes EMI effects.

Figure 2 Resistor Value: 22ohm

Figure 2 Capacitor Value: <22pF (Only necessary for countering excessive EMI problems. Default is not connected (NC))

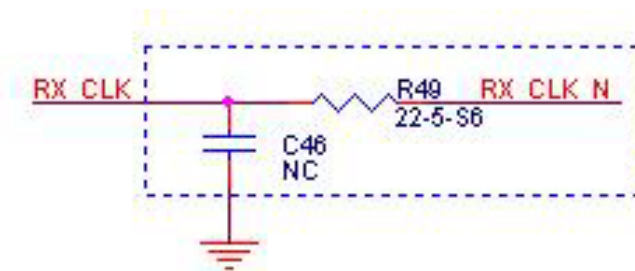


Figure 2. RX Clock

3.2. MDI Signals

- Traces routed from the RTL8211E/RTL8211EG to the 10/100/1000M magnetics, and to the RJ-45 connector, should be as short as possible. The 12cm maximum length between the RTL8211E/RTL8211EG and magnetics is achievable only when there is no interference. It is also very important to keep all four differential pair signal traces (MDI0+/-, MDI1+/-, etc.) at matching lengths (within 800mil). MDI impedance is 50ohm common mode, 100ohm differential mode)
- The two traces of each pair should be placed close to each other (D1) since they are differential pair signals to each other and provide a strong cancelling effect on noise. D1 can be the width of each of the two differential traces. E.g., if the width of the trace is 8mil, then D1 can be 8mil wide (Figure 3).

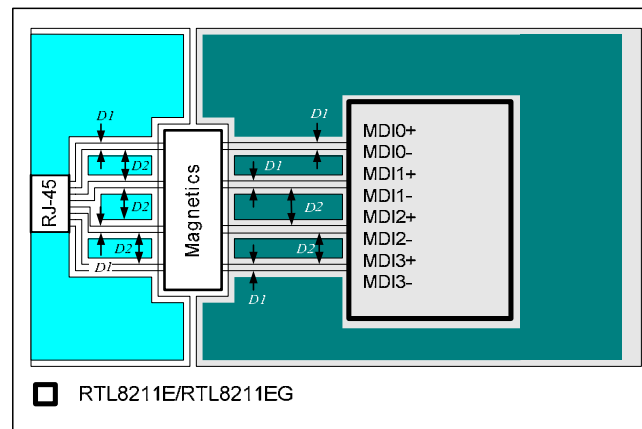


Figure 3. MDI Signals

- We suggest that there should be more than 30mil spacing between different differential pairs to minimize crosstalk coupled from other pairs (D2 in Figure 3). In addition, Ground Plane shielding can be used to separate all four signal pairs. However, a good layout should avoid the following situations:
 - Intersection of any two pairs of signal traces.
 - Intersection of the two signal traces of the same differential pair.
- To minimize impedance mismatch, we recommend not using vias on the four differential pairs.
- Signals crossing a plane split (see Figure 4) may cause unpredictable return path currents and would likely result in signal quality failure, as well as creating EMI problems.

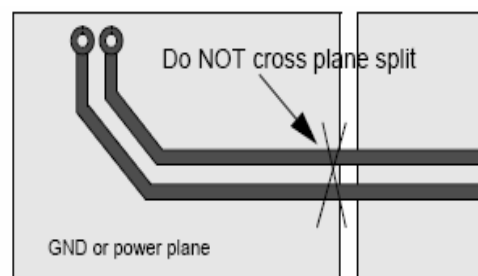


Figure 4. Signal Trace

3.3. GMII (MII)/RGMII Signals

- Ninety-degree trace turns must be avoided. We recommend that the traces turn at 45° angles as shown in Figure 5. Sharp edges may add unexpected parasitic effects into the circuitry. Reducing the trace length will reduce trace inductance during quick energy bursts.

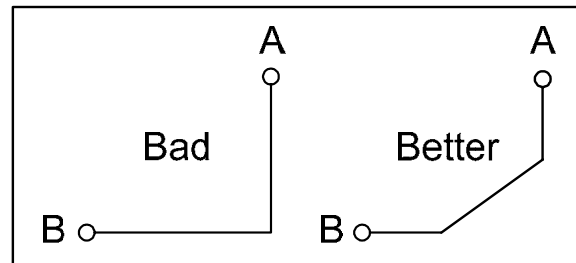


Figure 5. Signal Trace Angles

- The trace length and the ratio of trace width to trace height above the ground planes should be carefully considered. Clock and other high-speed signal traces must be as short as possible (less than 2.5 inches). It is better to have a ground plane under these traces. Using a GND plane to surround them is necessary.
- RXC and TXC are high-speed (125MHz) signals; keep a 20mils space between clock and data signals.
- Match each GMII (MII)/RGMII TX and RX (RXC/RXD/RXCTL) group trace length to within 100mils.
- Route the GMII (MII)/RGMII traces at 50ohm impedance, and route via an inner layer to reduce radiation.
- Keep all GMII (MII)/RGMII traces as short as possible (less than 2.5 inches).
- All GMII (MII)/RGMII traces must be referenced to an unbroken ground plane.

- Place R1/C1 close to the RTL8211E/RTL8211EG, and R2/C2 close to the MAC (must be less than 500mils).

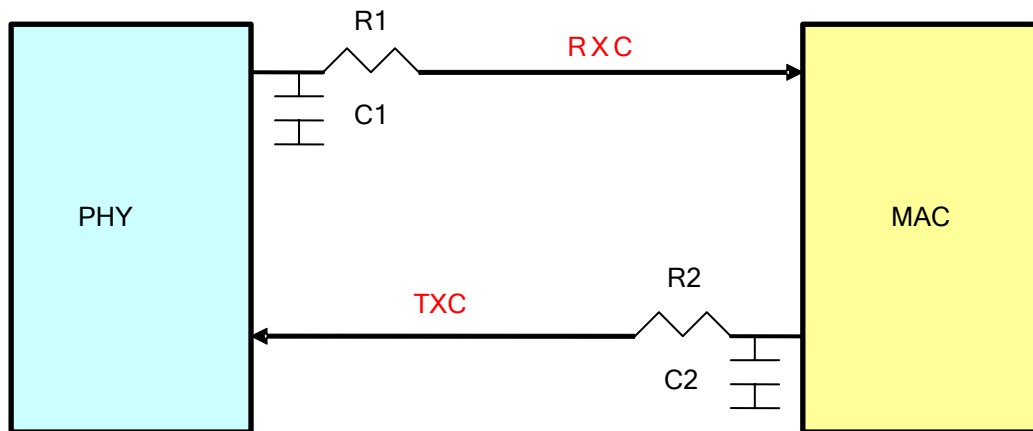


Figure 6. R1C1/R2C2 Placement

- Some return current paths, e.g., clock buffer and clock trace, can couple with the heat sink via parasitic capacitance, then radiate to air from the heat sink. To avoid this, RGMII traces must be routed away from the heat sink.

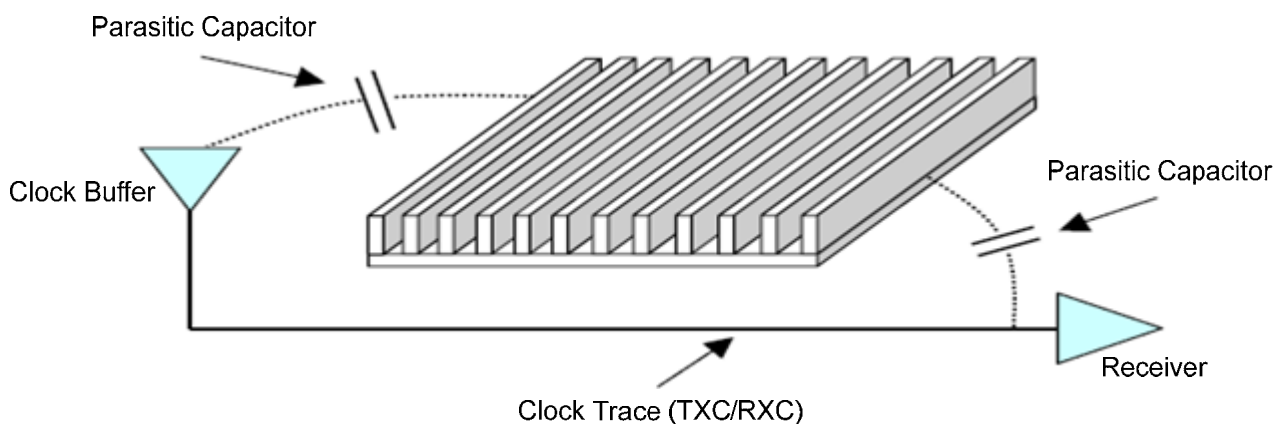


Figure 7. Clock and Heat Sink

- Route the GMII (MII)/RGMII traces away from I/O traces to avoid crosstalk (>20mils).

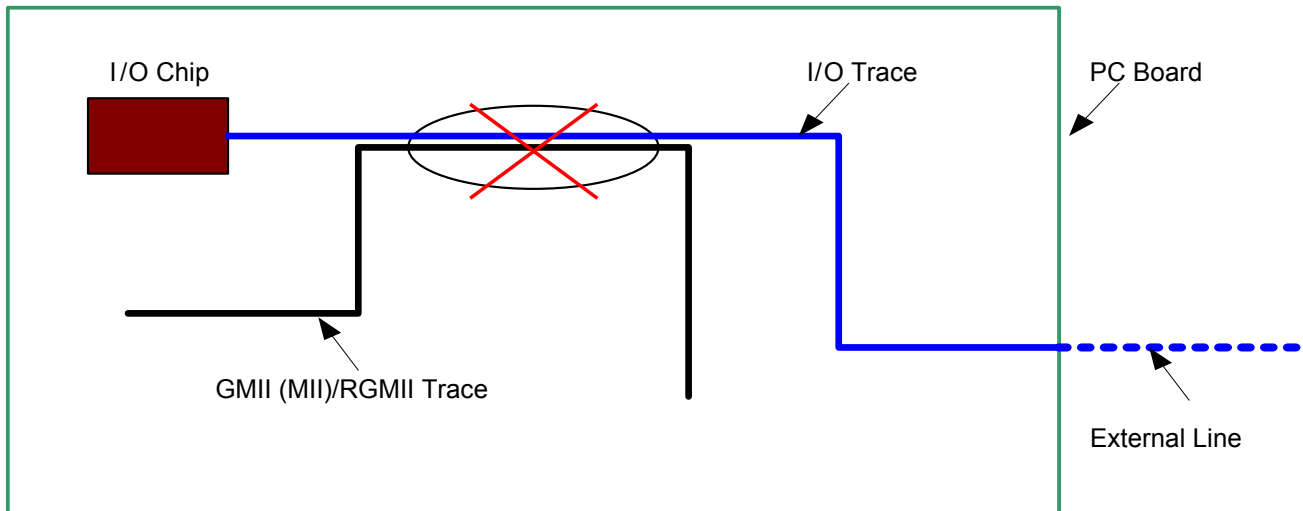


Figure 8. Clock and I/O Trace

- Reserve 2.5V power for GMII/RGMII use.

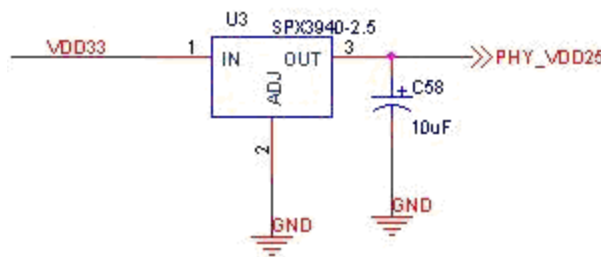


Figure 9. Reserve 2.5V Power for GMII/RGMII Use

- The RTL8211E (48-Pin QFN) uses Pin15 and Pin21 for 2.5V RGMII.
- The RTL8211EG (64-Pin QFN) uses Pin20 and Pin26 for 2.5V GMII.

3.4. Power Signals

The power supply into the RTL8211E/RTL8211EG digital power pins can be improved with de-coupling capacitors. The power signal traces (de-coupling cap traces, power traces, grounding traces) should be as short and wide as possible. The vias of the de-coupling capacitor should be large enough in diameter. All analog power pins on the RTL8211E/RTL8211EG need to be de-coupled with a capacitor. The de-coupling capacitors must be placed close to the RTL8211E/RTL8211EG (<200mils), and the traces should be kept short.

4. PCB Stack-Up

4.1. Overview

PCB stack-up is a major factor affecting the EMC performance of a product. A good stack-up can be very effective in reducing radiation from the loops on the PCB (differential-mode emissions), as well as the cables attached to the board (common-mode emissions). On the other hand, a poor stack-up can increase the radiation from both of these mechanisms considerably.

When using a multi-layer board, there are four main objectives to achieve:

1. Power and Ground planes should be coupled as closely as possible.
2. A signal layer should always be adjacent to a plane, and should be tightly coupled (close) to the adjacent plane.
3. High-speed signals (GMII/RGMII traces) must be routed on buried layers between planes so that the planes can act as shields and contain any radiation from the high-speed traces.
4. Multiple ground planes are very advantageous, since they will lower the ground (reference plane) impedance of the board and reduce the common-mode radiation.

The following are some recommendations to improve EMI performance.

4.2. Four-Layer Stack-Up (Motherboard Application)

- MDI and RGMII signals are routed on layer 4 and reference layer 3 (GND plane)

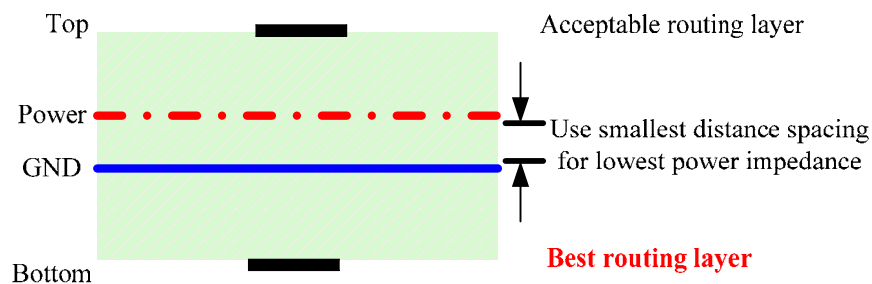


Figure 10. Four-Layer Stack-Up

4.3. *Six-Layer Stack-Up (Notebook Application)*

- GMII/RGMII signals are routed on layer 4 (IN2) and reference layer 5 (GND plane)

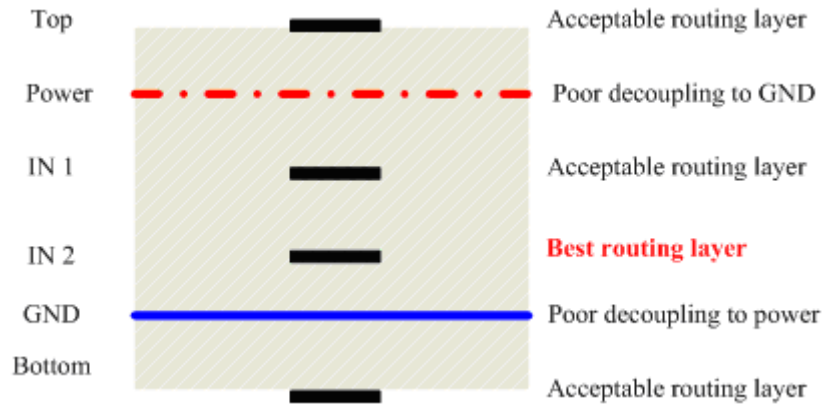


Figure 11. Six-Layer Stack-Up (A)

- GMII/RGMII signals are routed on layer 4 (IN1) and reference layer 3 (GND plane)

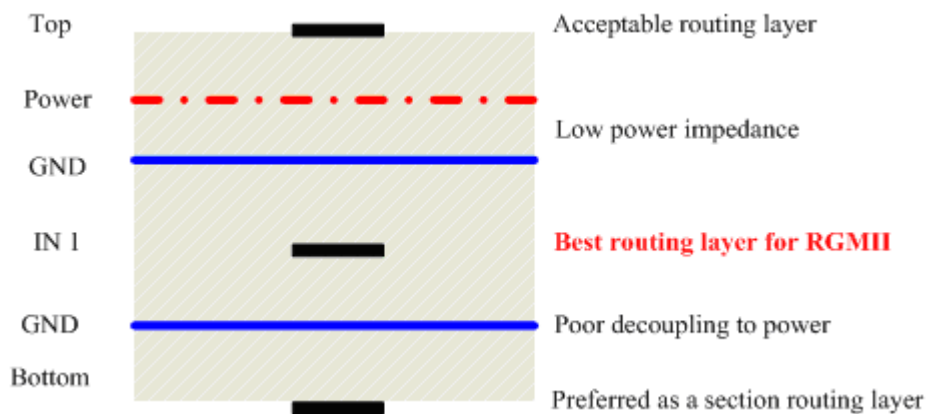


Figure 12. Six-Layer Stack-Up (B)

- GMII/RGMII signals are routed on layer 3 (IN1) and reference layer 2 (GND plane)

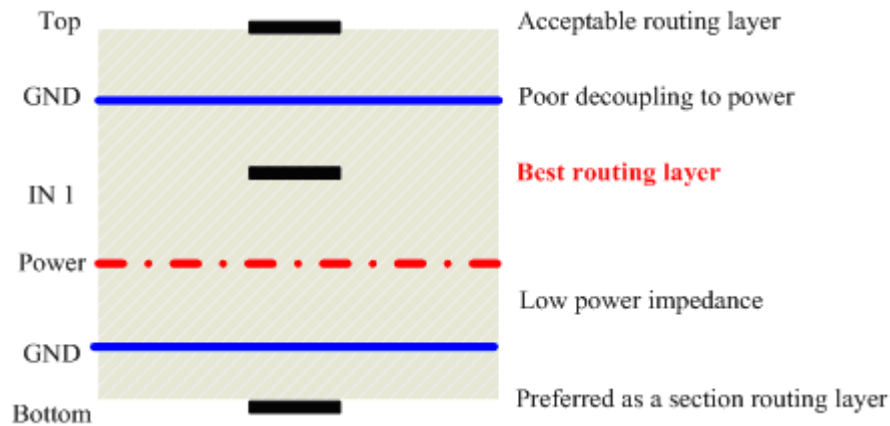


Figure 13. Six-Layer Stack-Up (C)

4.4. Eight-Layer Stack-Up (Notebook Application)

- GMII/RGMII signals are routed on layer 3 (IN1) or layer 6 (IN3), and reference layer 2 (GND plane) or layer 7 (GND plane)

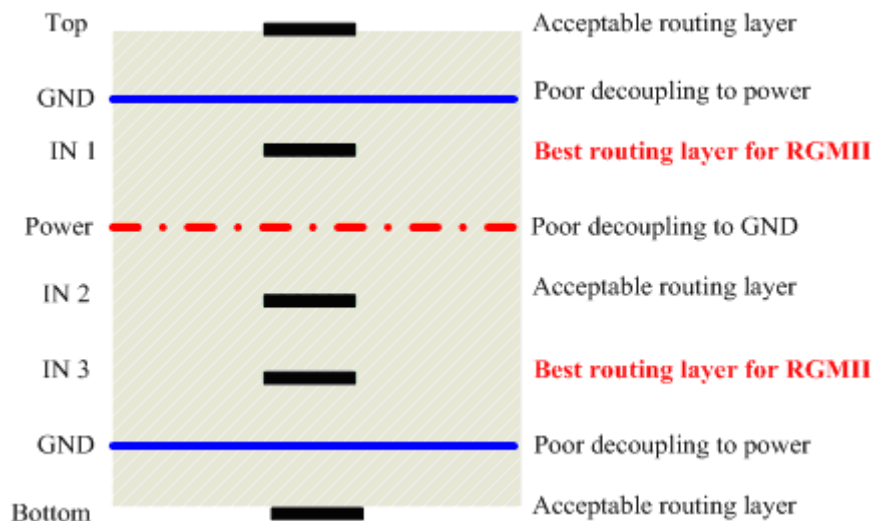


Figure 14. Eight-Layer Stack-Up

5. Ground Plane Layout

Isolated separation between Analog and Digital Ground domains is not recommended, as bad ground plane partitioning could cause serious EMI emissions and degrade analog performance due to bouncing noise.

The RTL8211E/RTL8211EG has only a single ground plane for analog power (AVDD33 and AVDD12) and digital power (DVDD33, DVDD12). In the center of the IC, there is an Exposed Pad (EPAD) ground. The PCB layout requires 9 vias to connect the EPAD to the lower layer ground plane (see Figure 15).

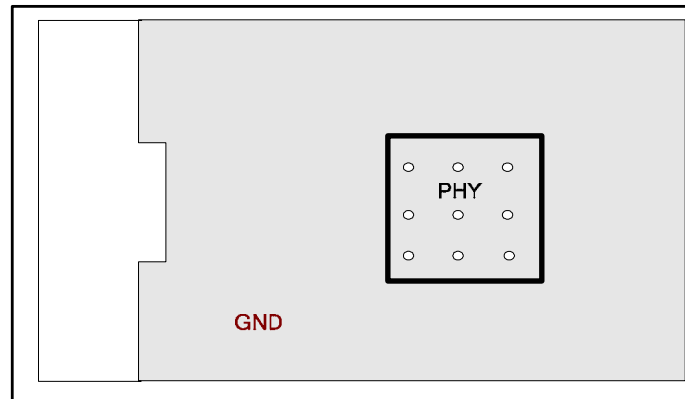


Figure 15. Ground Plane Layout-1

To achieve better ground plane performance, it is recommended to keep the plane as large and uniform as possible. Figure 16 illustrates a not so good (left) and a good ground plane layout (right).

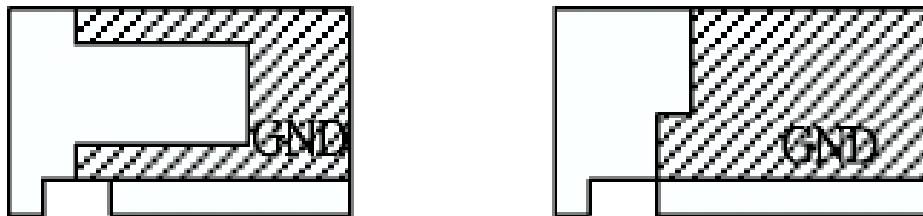


Figure 16. Ground Plane Layout-2

The plane area beneath the magnetics should be left void. The void area is to keep transformer-induced noise away from the power and system ground planes (Figure 17).

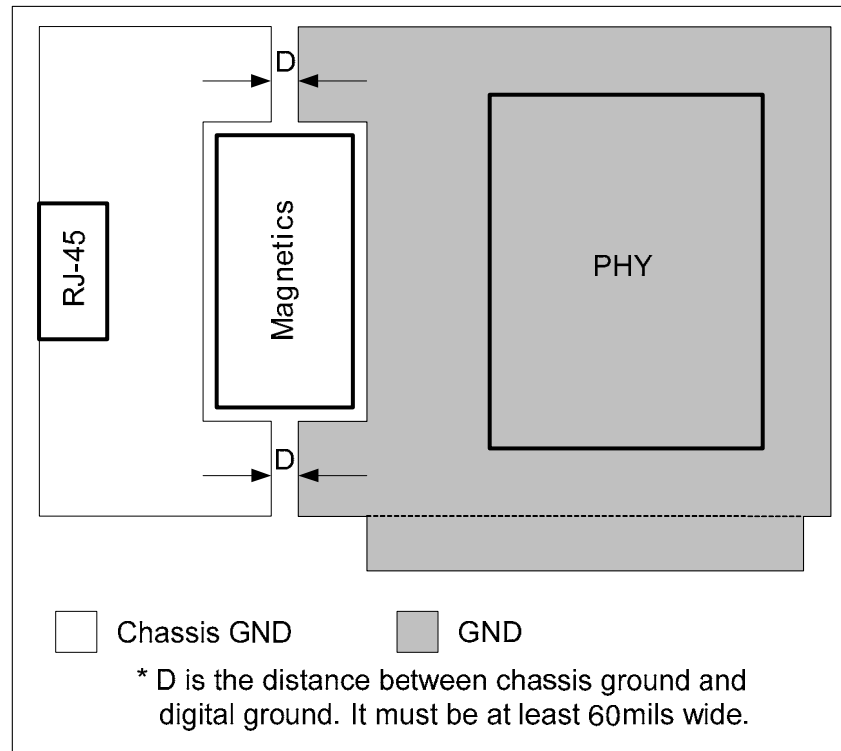


Figure 17. Ground Plane Separation

The Chassis Ground as shown in Figure 17 is known as an ‘Isolated Ground’. It connects directly to the RJ-45 connector (fully shielded is recommended). In addition, a 2kV (3kV recommended) high voltage capability capacitor is needed to connect to this chassis ground for ESD protection.

It is important to keep the gap (D in Figure 17) between Chassis GND and System GND wider than 60mils for better isolation.

5.1. Four-Layer Board Ground Plane Layout (Typical Application)

1. Signal 1 (top layer)
2. Power (Keep GND area for RTL8211E/RTL8211EG)
3. GND
4. Signal 2 (bottom)

5.1.1. Ground Plane Layer 1 Layout

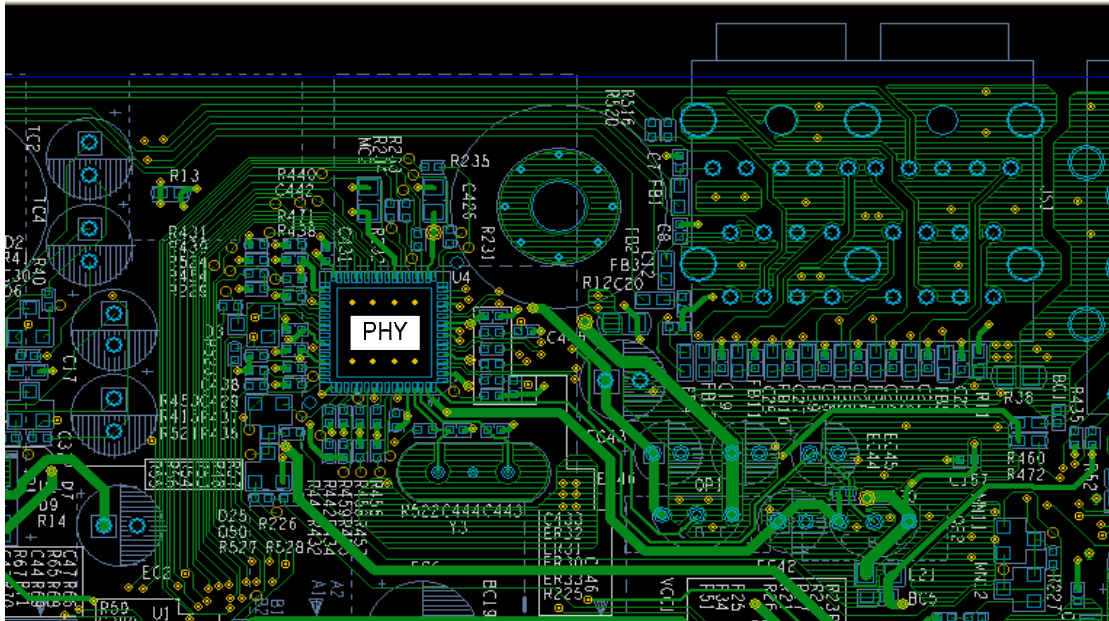


Figure 18. Ground Plane Layer 1 Layout

5.1.2. Ground Plane Layer 2 Layout

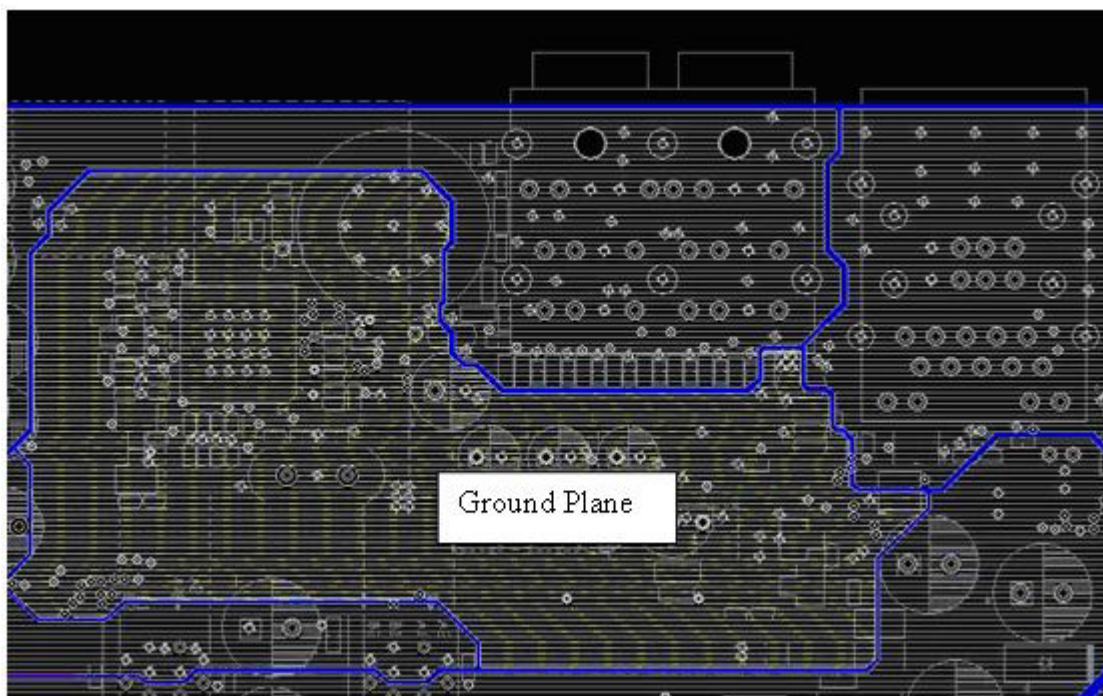


Figure 19. Ground Plane Layer 2 Layout

5.1.3. Ground Plane Layer 3 Layout

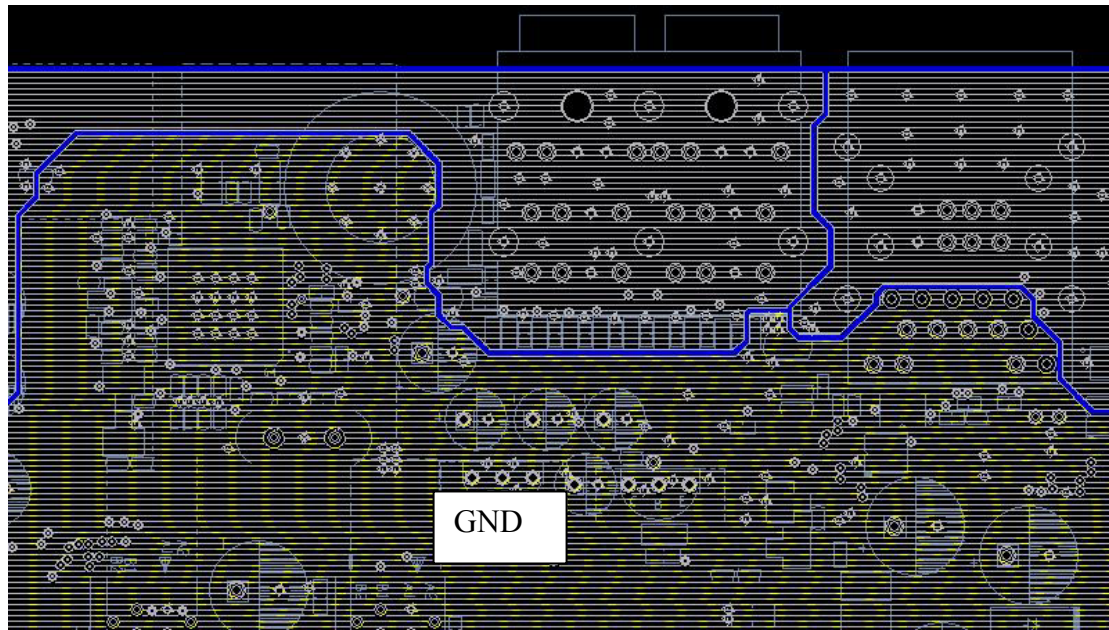


Figure 20. Ground Plane Layer 3 Layout

5.1.4. Ground Plane Layer 4 Layout

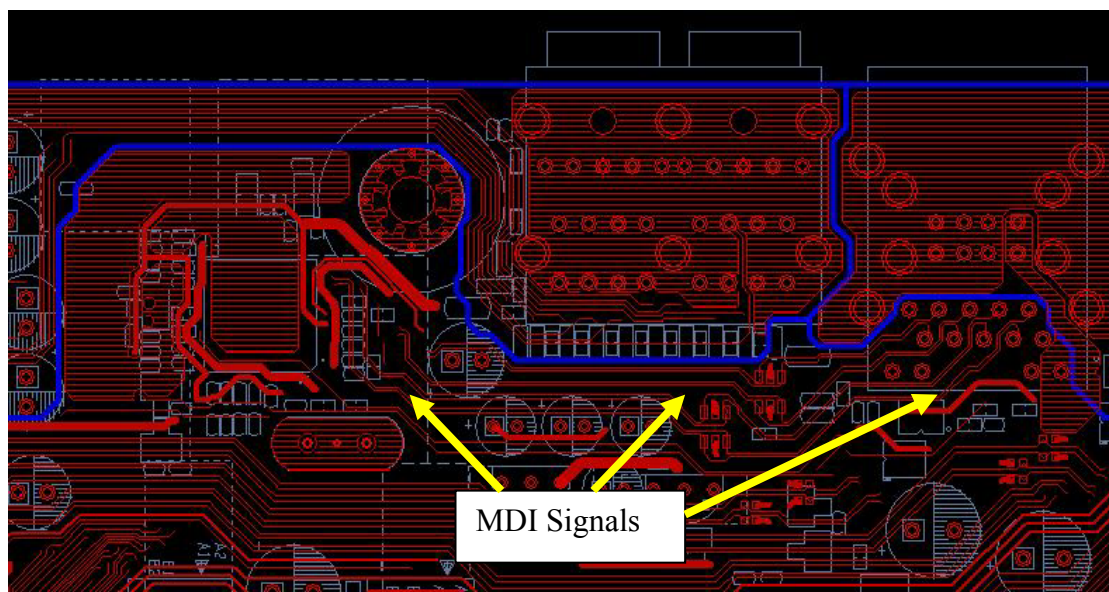


Figure 21. Ground Plane Layer 4 Layout

6. Power Plane Layout

It is recommended to use at least a 4-layer PCB. The digital power plane should be separated from analog areas, which are extremely sensitive to noise.

Any analog circuitry on the same plane as the digital power will experience an energy fluctuation due to the fast switching time of digital components. This could improperly bias transistors, and cause the circuits to malfunction. A low-pass filter combination of a ferrite bead and capacitors should be used to provide a clean, filtered power plane for analog consideration (Figure 22). Keep power traces to the RTL8211E/RTL8211EG as short and wide as possible and make good use of vias.

(a) Keep the Digital Power Plane as a whole, and leave some space for the Analog Power Plane

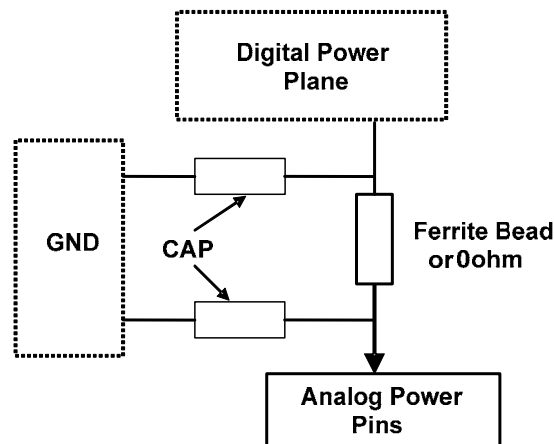


Figure 22. Power Plane

(b) Decoupled Capacitor Example

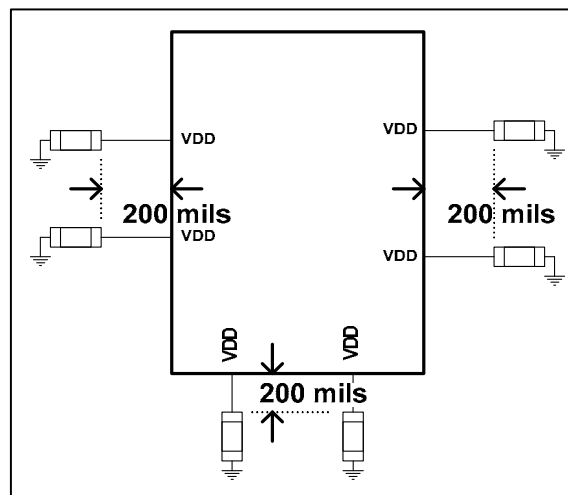


Figure 23. Decoupled Capacitor Example

To improve the performance of the power plane, try to keep the contact area between the RTL8211E/RTL8211EG VDD pins and power plane as large as possible rather than using small narrow traces (Figure 24).

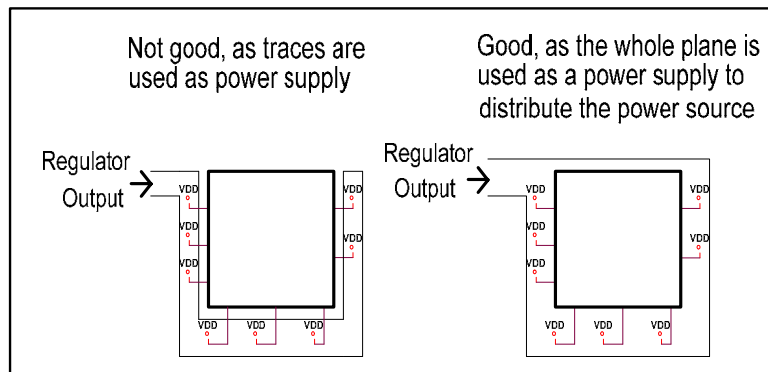


Figure 24. Power Source Distribution

- The 3.3V and 1.05V power noise levels must be kept below 100mV in gigabit mode.
- All 3.3V/1.05V decoupling capacitors shown in the reference schematic, available from Realtek, should be used in all designs.
- The DVDD33 power (3.3V) plane must be kept as whole and as large as possible.

6.1. Power Plane Layer 1 Layout

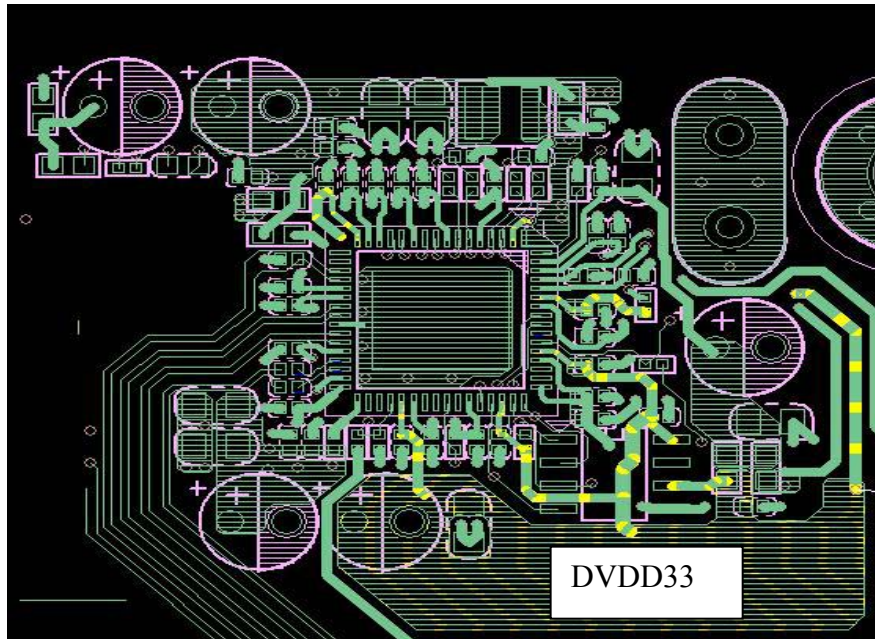


Figure 25. Power Plane Layer 1 Layout

6.2. Power Plane Layer 4 Layout

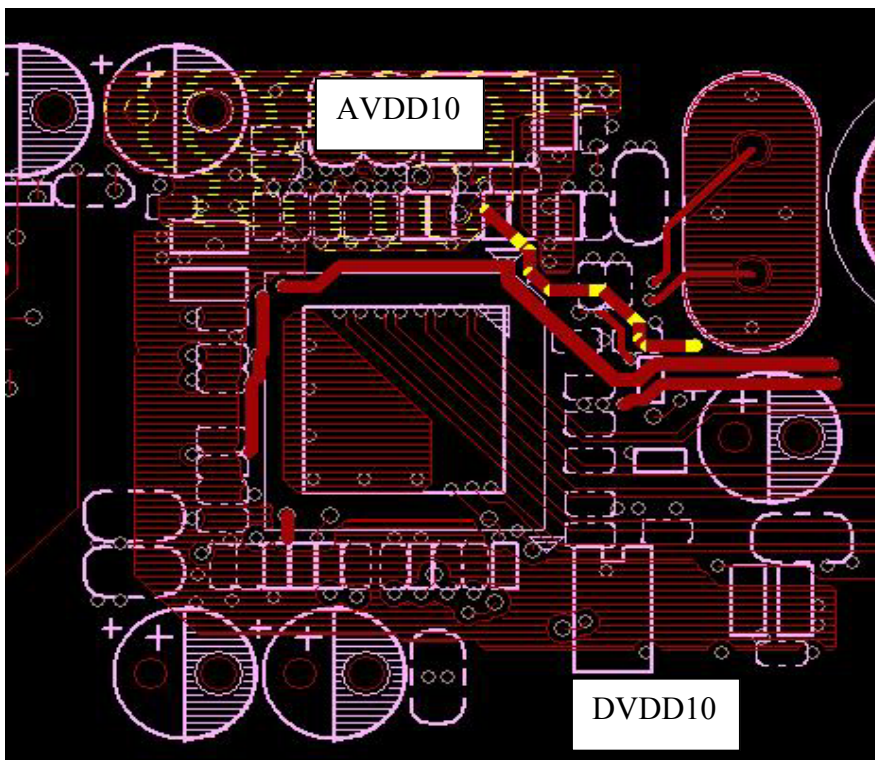


Figure 26. Power Plane Layer 4 Layout

7. Center-Tapping

- A center-tapped fine-tuned capacitor (C8 Value: 0.1 μ F~10pF) can improve EMI for single tone noise.

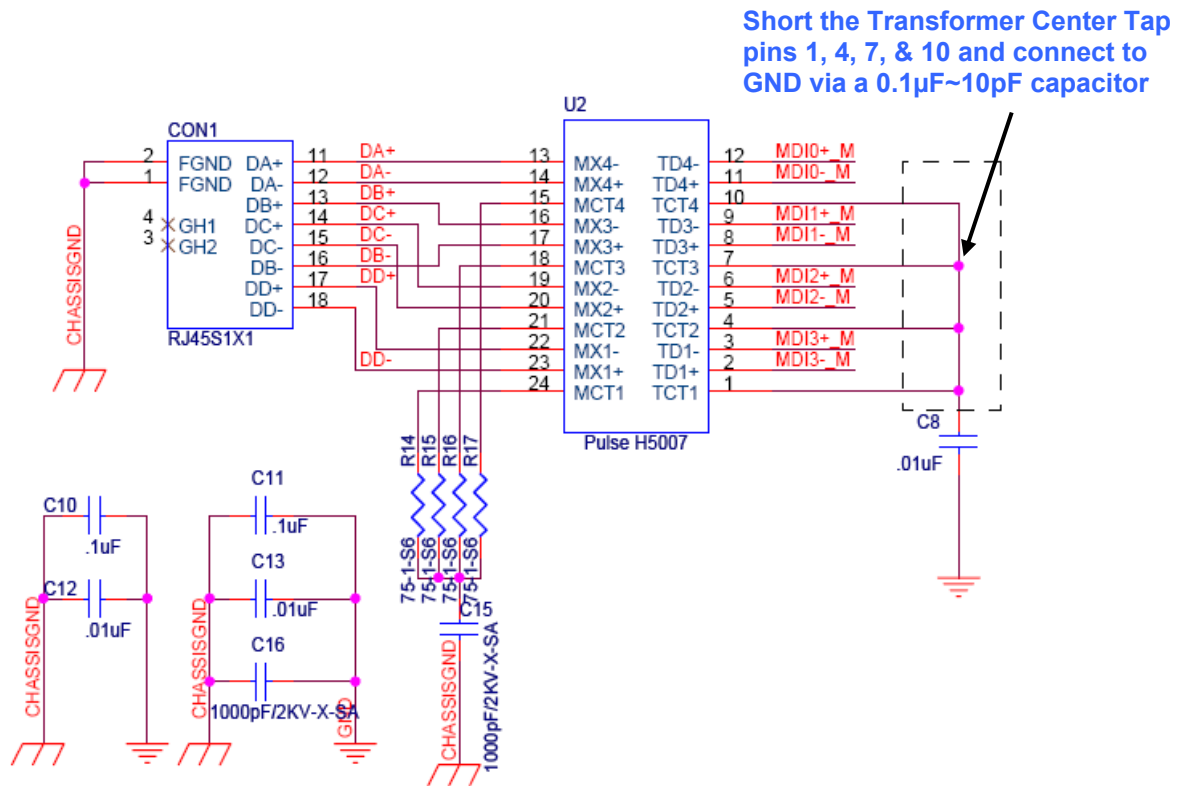


Figure 27. Center-Tapping

8. LED Pins

- Reserve a decoupling capacitor (D-CAP) for LED pins, and place close to the LAN connector (Refer to the reference schematic, available from Realtek).

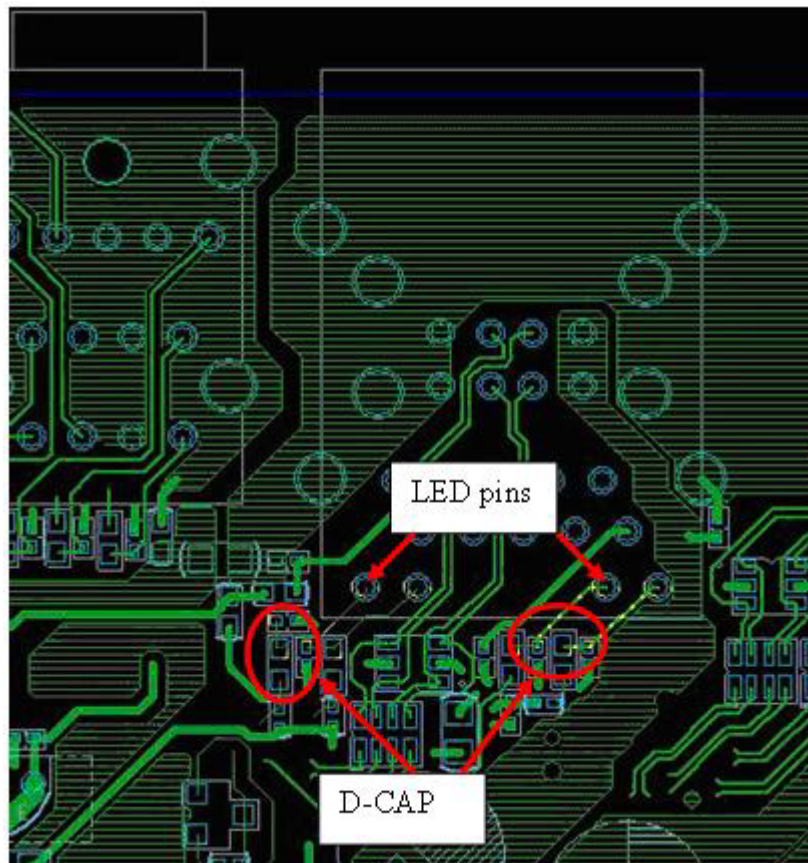
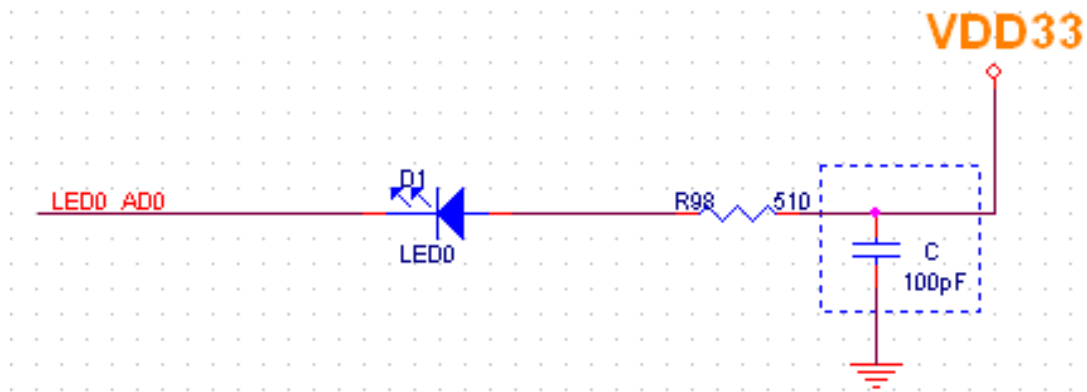


Figure 28. LED Pins

9. Switching Regulator

The RTL8211E/RTL8211EG incorporates a state-of-the-art switching regulator that requires a well-designed PCB layout in order to achieve good power efficiency and lower the output voltage ripple and input overshoot. The 1.05V switching regulator output pin (REG_OUT) should be connected only to DVDD10 and AVDD10 (do not provide this power source to other devices).

9.1. PCB Layout

- The input 3.3V power trace connected to VDDREG must be wider than 40mils.
- The bulk de-coupling capacitors (Cin1 and Cin2) must be placed within 200mils (0.5cm) of VDDREG to prevent input voltage overshoot.
- The output power trace of REG_OUT must be wider than 60mils.
- Lx (2.2 μ H) must be kept within 200mils (0.5cm) of REGOUT.
- Cout1 and Cout2 must be kept within 200mils (0.5cm) of Lx to ensure stable output power and better power efficiency.
- For switching regulator stability, the capacitor Cout1 and Cout2 must be a ceramic (X5R) capacitor. Cin1 and Cin2 are recommended to be ceramic capacitors.
- Place Lx and Cin1 on the same layer as the RTL8211E/8211EG. Do not use vias on VDDREG and REGOUT traces.

Note: Violation of the above rules will damage the IC.

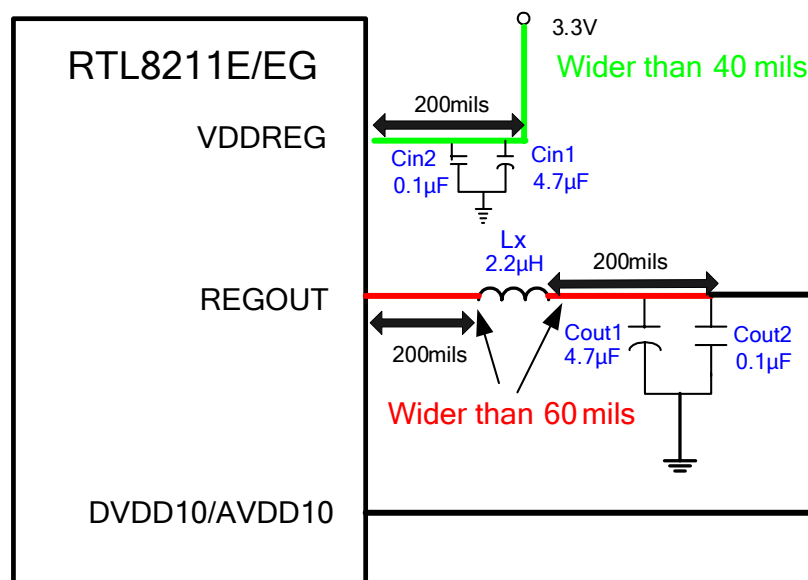


Figure 29. Switching Regulator

10. Parts Recommendations

10.1. 10/100/1000M Magnetic

Turn Ratio TX/RX: 1:1

Primary Inductance: 350 μ H OCL with 8mA bias

Insertion Loss: -1.0 dB Max, 1 ~ 100MHz

Return Loss: -18dB Min @ 100 Ω , 1 ~ 30MHz

-14dB Min @ 100 Ω , 30 ~ 60MHz

-12dB Min @ 100 Ω , 60 ~ 80MHz

Differential to Common Mode Rejection:

-40dB Min @ 1 ~ 60MHz

-30dB Min @ 60 ~ 100MHz

Hi-Pot: 1500Vrms @ 60sec

Operating Temperature: 0°C to 70°C

Recommended Magnetics: Pulse H5007 or similar

10.2. Resistors

Resistors that have tolerance requirements within 1% are strongly recommended. See the reference schematic, available from Realtek, for details.

10.3. Capacitors

For switching regulator power filtering, an X5R ceramic capacitor is recommended for the power circuit.

10.4. Power Inductor

The power inductor used by the switching regulator must be capable of handling 600mA of current, and the resistance value should be as small as possible to achieve the expected switching regulator efficiency, which must be higher than 75%.

Typically, if the power inductor's ESR at 1MHz is below 0.8Ω , the switching regulator efficiency will be above 75%.

10.5. RJ-45 Jack

A fully shielded RJ-45 connector should be used.

11. Special Notes

- Keep a void area of at least 100mils from the edge of each layer (e.g., power plane, GND plane, etc.) to the PCB edge in order to minimize fringe effect and lower EMI emissions.
- The GMII/RGMII traces are high-speed signal traces. For the best performance, be sure to follow all GMII/RGMII-specific layout guidelines.

Realtek Semiconductor Corp.**Headquarters**

No. 2, Innovation Road II

Hsinchu Science Park, Hsinchu 300, Taiwan

Tel.: +886-3-578-0211. Fax: +886-3-577-6047

www.realtek.com