POWER SUPPLY LAYOUT
LAYOUT CONSIDERATIONS FOR LINE-POWER SUPPLIES

25 June 2026

Observe precautions! Electrostatic sensitive devices!

Patent protected:
WO98/36395, DE 100 25 561, DE 101 50 128,
WO 2004/051591, DE 103 01 678 A1, DE 10309334,
WO 04/109236, WO 05/096482, WO 02/095707,
US 6,747,573, US 7,019,241

REVISION HISTORY

The following major modifications and improvements have been made to this document:

Version	Author	Reviewer	Date	Major Changes
1.0	CB		2011	Original version
2.0	AZ	SB, JB	2026	Document maintenance

Published by EnOcean GmbH, St.-Martin-Straße 76, 81541 Munich, Germany
www.enocean.com, info@enocean.com, phone +49 (89) 6734 689-0

TABLE OF CONTENTS

LIST OF FIGURES

LIST OF TABLES

1 Introduction

Power supplies, especially switch-mode power supplies, can generate electrical noise that may disturb nearby circuits and radio systems. This noise can travel through wires or spread through the air, which may reduce system performance. Good PCB layout and proper power supply design are therefore very important to avoid interference problems.

2 Harmful Noise from Power Supplies

Power supplies, especially switch-mode power supplies, are a major source of electrical and magnetic interference. This noise, also called electromagnetic interference (EMI), appears in both conducted and radiated form. Conducted EMI is usually found in the low frequency range from a few kHz up to 30 MHz, while radiated EMI is typically found in the frequency range from 30 MHz to 10 GHz. Electrical noise can enter a system in several ways: through galvanic coupling (direct electrical contact), electrostatic coupling, electromagnetic induction, or radio frequency interference (RFI).
Unfortunately, the ground symbol used in schematics often creates the false impression that a fixed and stable reference ground always exists. In reality, there are only signal sources and their return paths. Reliable PCB design therefore requires, above all, a careful layout. Applying proper PCB layout techniques during the design stage helps prevent later "unexplained" or "random" system failures. One of the most common signs of poor electronic design or layout is a significant reduction in system performance, for example reduced range or "phantom telegrams" in radio receivers.
In a standard application, the low supply voltage is usually generated with a mains-isolated, step-down transformer, followed by rectification, filtering, and regulation. However, in many small and low-cost applications, the transformer cost becomes a major factor. In such cases, alternative solutions are often used to reduce both the cost (typically by a factor of 4 to 5) and the size of the power supply.

3 Basic Types of Power Supplies

Power supplies can be divided into two main groups: galvanically isolated and non-isolated types. Isolated converters use a transformer to provide electrical separation between the mains and the low-voltage side.
Non-isolated converters, especially RC-type designs, are smaller and more cost-effective, but they offer lower safety and lower overall performance.

Table 1 Basic Types of Converters

Converter Type	Isolated (Transformer)	Non-Isolated (especially RC-Type)
Advantages	• Galvanic isolation • Higher safety	• Low cost • Smaller size, low-power applications
Disadvantages	• Large and heavy • Higher cost	• Low system efficiency • Reduced safety

4 Basic Types of Regulated Power Supplies

Regulated power supplies are generally divided into two main types: linear and switch-mode. Linear regulators are simple and provide very clean output voltage, while switch-mode regulators offer higher efficiency and smaller size. The choice depends on the application requirements, such as noise level, efficiency, size, and cost.

Table 2 Basic Types of Regulated Power Supplies – Linear and Switch-Mode

Regulator	Linear	Switch-Mode
Advantages	• Very low output noise and ripple voltage • Very low common-mode noise current • Low cost in low-power applications	• High efficiency (typically >80%) • Small and lightweight, especially at higher power levels • Output voltage can be lower than, equal to, or higher than the input voltage
Disadvantages	• Low efficiency (typically <60%) • Output voltage is always lower than the input voltage • High heat dissipation	• Higher output noise and ripple voltage • More complex design

5 PCB Layout and Supply Voltage Requirements for RF Receiver Modules

For RF receiver modules to achieve good operating range, they require a stable and low-noise supply voltage. In addition to selecting the right power supply type and components, the PCB layout is also very important for controlling both conducted and radiated emissions. The same layout recommendations also apply to additional electronics, such as digital circuits. Always follow the layout and power supply recommendations and requirements specified by the manufacturers of the devices used in the design. This helps ensure proper operation, reduces the risk of interference problems, and improves the overall reliability of the system.
Because parasitic impedances increase with frequency, a simple PCB trace may no longer behave like a low-resistance connection. At higher frequencies, it can act as a complex path or even as an unintended antenna. Since these parasitic effects include both capacitive and inductive components, careful PCB layout is critical for the performance of the entire system.

6 Notes for Component Placement

Proper component placement is essential for good system performance and low interference. The layout should be designed to minimize noise coupling and to keep sensitive analogue signals separate from noisy digital and power circuits. The following recommendations are especially important for RF and mixed-signal designs.

1. Keep ground return paths short and wide. Always provide a return path that creates the smallest possible current loop.

2. Separate analogue ground from digital and power ground. When routing the circuit, keep the analogue small-signal ground separate from the digital ground and from the power ground that carries switching currents.

3. Place the analogue circuitry in a separate analogue ground area and connect it to the main system ground at a single point. This helps keep the analogue ground clean and quiet. All connections between analogue and digital circuits should cross close to this single connection point.

7 General PCB Layout and Component Design Guidelines

Good PCB layout is essential for stable operation, low noise, and reliable RF performance. The following guidelines help reduce conducted and radiated interference and improve the overall behaviour of the system.

1. Use a multilayer PCB with separate GND and VCC planes. Keep the connections between each supply pin and the related power or ground plane as short as possible.

2. Connect signal ground to the ground plane through vias whenever possible. Avoid long PCB traces for ground connections, because any trace can act as an unwanted antenna.

3. Keep power ground, digital ground, and analogue ground separate. If these grounds must be electrically connected, connect them at a single point near the DC output return. Use star grounding whenever possible instead of daisy-chain grounding.

4. Minimise the size and length of loops that carry high-frequency switching currents. Magnetic coupling increases strongly with loop area and is difficult to suppress. Copper traces and ground planes are often not effective shields against magnetic coupling. (See Figure 1)

5. Reduce loop areas by using short traces and proper return paths. Route signal traces close to their return paths and, if possible, on adjacent layers. Place bypass capacitors as close as possible to the noise source.

6. Pay special attention to digital clock signals. Use the lowest clock frequency that the system can tolerate. Make sure that the clock frequency, its harmonics, sub-harmonics, or divided frequencies do not fall within or close to the RF band used by the receiver. Otherwise, these signals may create in-band interference and reduce receiver performance. For example, in the 315 MHz band, frequencies such as 5.00 MHz, 10.00 MHz, and 15.00 MHz can cause interference because of their harmonic relationship to the RF frequency. Slightly different frequencies, such as 4.92 MHz, 5.06 MHz, 9.83 MHz, 10.75 MHz, or 14.74 MHz, may help avoid this problem. To reduce clock-related interference:
    - use termination resistors to slow very sharp clock edges,
    - provide shielding or at least a grounded guard ring around the oscillator,
    - avoid routing traces below or close to the oscillator,
    - use copper fill connected to ground with vias where appropriate,
    - and add line filters if needed.

7. Be careful when using switch-mode power supplies. Switch-mode converters generate high-frequency switching currents. If a switch-mode supply must be used, a

constant-frequency design is preferred over a variable-frequency design. If more than one switch-mode converter is used, they should preferably operate at the same switching frequency to avoid random interference effects. Many designs provide an external synchronisation input for this purpose. Always check the component manufacturer's datasheet for detailed recommendations. Proper filtering and PCB layout are especially important in such designs.

8. Place filter capacitors directly in the main current path. Their terminals should connect directly to the PCB traces carrying the current to be filtered, without unnecessary extra trace length.

9. Keep the distance between the converter and the filter capacitors as short as possible. This reduces parasitic effects and improves transient current handling.

10. Use the smallest practical capacitor package for a given capacitance value. Smaller capacitors usually have lower parasitic inductance than larger ones. Use capacitor shapes with low inductance. Short capacitors usually have lower inductance than long ones, and higher-profile capacitors may have lower inductance than very flat ones. For decoupling, use at least two capacitors with values differing by about a factor of 100. This improves filtering over a wider frequency range.

11. Do not rely on one large capacitor alone for high-frequency filtering. A large capacitor has significant inductive behaviour at high frequencies and is therefore less effective against fast noise. Better results are achieved by connecting capacitors in parallel, for example:
    - a tantalum capacitor of at least 22 µF, and
    - a ceramic capacitor of 0.1 µF.

    Place the capacitor closest to the device pin first.

12. Provide space for additional noise suppression components on the PCB.
    Add footprints for capacitors or other filter elements even if they may not be needed at first. This reduces risk during debugging and allows easy optimisation later. If the parts are not required, they can simply be left unassembled.

13. Be careful when using high-value ceramic SMD capacitors instead of tantalum capacitors. The actual capacitance of ceramic capacitors can decrease significantly depending on the dielectric material and rated voltage. Therefore, as a general rule, choose at least twice the nominal capacitance that is actually required. For example, if 10 µF / 6.3 V is needed, use a ceramic capacitor of at least 22 µF / 6.3 V. A higher value is often better.

Table 3 Typical Ceramic Capacitor Capacitance at 10, 50, and 100% of Rated DC Voltage

Dielectric	**C0G**	**X7R**	**Z5U**
%C at 10%	100%	105%	120%
%C at 50%	100%	95%	50%
%C at 100%	100%	70%	30%

14. For harsh environments, consider additional suppression methods. Shield planes, ferrite beads, and common-mode chokes in signal or power lines may help reduce interference. However, these components should be used with care and require some design experience.

15. If possible, use a spectrum analyser and a near-field probe set during development. These tools help identify interference sources and problem areas on the PCB.

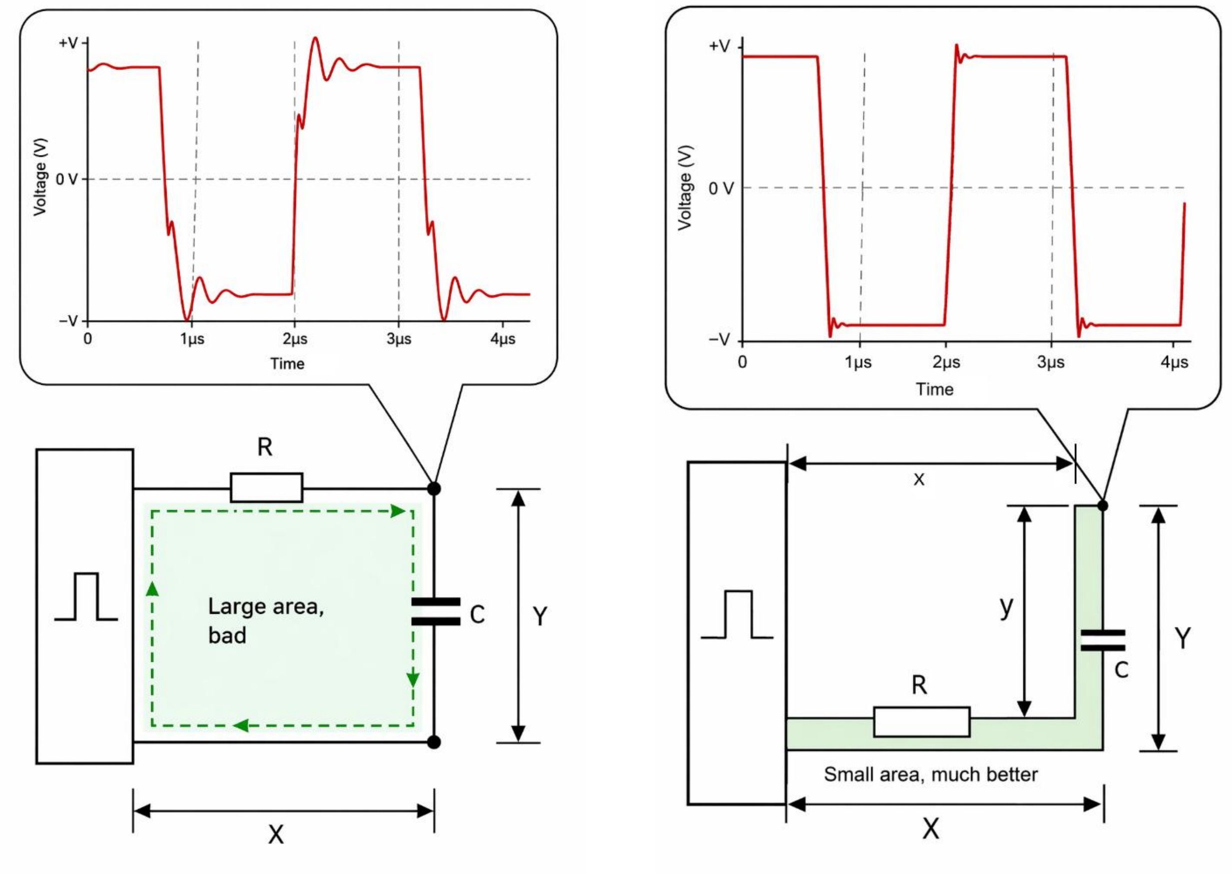

Figure 1 Effect of Loop Area Size on Signal Behaviour

DISCLAIMER

The information provided in this document describes typical features of the EnOcean radio transmission system and should not be misunderstood as specified operating characteristics. No liability is assumed for errors and / or omissions. We reserve the right to make changes without prior notice. For the latest documentation visit the EnOcean website at **www.enocean.com**.