

## POWER SUPPLY LAYOUT – Layout considerations for Line-Power Supplies

### Harmful noise from Power Supplies

Power supplies, especially switching-mode power supplies, are a serious source of electric and magnetic interference fields. Noise (EMI, electromagnetic interference) appears in both conducted and radiated form. While conducted EMI is mostly found in the low frequency range of several kHz to 30 MHz, the radiated EMI is mostly found in the frequency range from 30 MHz to several GHz. Electrical noise is transmitted into a system through the following ways: galvanic (direct electrical contact), electrostatic coupling, electromagnetic induction, or radio frequency interference (RFI).

Unfortunately, the ground symbol used in schematics generates the illusion that an unvarying reference ground actually exists. In real world, there are only signal sources and their returns. A reliable PCB design requires first careful layout. Applying proper board-layout techniques at design time prevents from “unexplainable” or “random” system failures later. One of the most common symptoms of an inappropriate electronic design or layout is a drastically reduced system performance (i.e. range wide reduction and “phantom telegrams” in case of radio receivers).

In classic applications, the desired low supply voltage is obtained by using a step-down mains isolated transformer, which is then rectified, filtered and regulated. However, in many smaller low-cost applications, cost and dimensions of the transformer becomes critical in the system. Under these circumstances, alternative solutions might be required in order to reduce the cost as well as the size of the power supplies.

### Basic types of power supplies

#### Basic types of voltage converters - galvanic isolated and not isolated:

Converter type	Isolated (Transformer)	Non isolated (particularly RC-Type)
<b>Advantages</b>	<ul style="list-style-type: none"> <li>Isolated</li> <li>Safer</li> </ul>	<ul style="list-style-type: none"> <li>Cost effective, smaller especially at low power</li> </ul>
<b>Disadvantages</b>	<ul style="list-style-type: none"> <li>Large and heavy</li> <li>Expensive</li> </ul>	<ul style="list-style-type: none"> <li>Poor system efficiency and safety</li> </ul>

#### Basic types of regulated power supplies - linear and switching technology:

Regulator	Linear	Switch-mode
<b>Advantages</b>	<ul style="list-style-type: none"> <li>Very low output noise and ripple voltage (clean)</li> <li>Very low common-mode noise current</li> <li>Low cost at low output power</li> </ul>	<ul style="list-style-type: none"> <li>Highest efficiency (typ. &gt; 80%)</li> <li>Small and light weight, especially at higher power</li> <li>Output voltage can be either lower, equal or higher as the input voltage</li> </ul>
<b>Disadvantages</b>	<ul style="list-style-type: none"> <li>Low efficiency (typ. &lt; 60%)</li> <li>Output voltage is always lower as the input voltage</li> <li>Heat dissipation</li> </ul>	<ul style="list-style-type: none"> <li>Higher output noise and ripple</li> <li>Demanding design!</li> </ul>

## POWER SUPPLY LAYOUT – Layout considerations for Line-Power Supplies

---

The most important requirement for the radio receiver to perform a wide range is a very stable and low noise supply voltage. As important as the choice of power supply type and components is the design of the full system regarding control of both radiated and conducted emissions. The following basic recommendations apply also for designing the layout for additional electronics like digital circuits.

Since parasitic impedances increase with frequency, a simple PCB signal trace might become a complex path (antenna!), rather than just a low resistance with respect to DC measurements. Due to these parasitic impedances – both capacitive and inductive in nature, the layout of the PCB is very critical for the entire HF system.

### Notes for Components Placement

1. Keep always ground return paths short and wide. Provide a return path that creates the smallest loop for the current to return.
2. When routing the circuitry the analog small signal ground and the digital and power ground for switching currents must be kept separate.
3. It is suggested to isolate the analog circuitry on a local ground island, which can then be connected to the rest of the system ground at one single point. This helps to keep the analog ground clean and quiet. All connections between the analogue and digital circuitry should cross nearby single point.

### Some general board layout and component design guidelines

1. Use a multilayer PCB with separate GND and VCC planes. Keep connections between each supply pin and the corresponding power/ground plane as short as possible.
2. Try to make signal ground connections through Vias to the ground plane rather than through PCB traces. Any PCB trace acts as a transmit antenna.
3. Try to keep power ground, digital ground and analog ground separately. Tie the different grounds together (if they are electrically connected) at one single point near DC output return. Star grounding should be used whenever possible, as opposed to “daisy chain” grounding.
4. Minimize all areas and lengths of loops, which conduct high frequency switching currents (see Fig. 1). Magnetic coupling is a strong function of the loop area and is difficult to counteract, because a magnetic shield is usually required instead of simple copper shielding. Since PCBs use copper conductors, the traces and ground planes may be ineffective as shields against magnetic coupling.
5. Reducing loop areas by shortening the trace lengths and by routing in/out signal traces next to their return paths, parallel to each other on adjacent layers. Loop areas can also be reduced by placing bypass capacitors as close to the noise source as possible.
6. Very important, applicable especially to digital circuits: Choose the lowest clock rates your system can tolerate, and make sure that neither the clock nor its divided frequencies fall on or very near ( $\pm 2$  MHz) to your radio frequency band. These harmonics use the physical copper paths or wire connections (e.g. serial data and power supply connections) to the receiver as antenna to come finally over the air as “in band noise” into receiver. Use damp termination resistors to “soften” very sharp clock edges on the line. Provide a shield or at least a guard ring tied to

## POWER SUPPLY LAYOUT – Layout considerations for Line-Power Supplies

---

correspondingly ground for oscillator. Avoid routing below oscillator area, around and between oscillator and tracks filled with copper, provide Vias to appropriate ground and line filters.

7. Switch mode power supplies particularly carry high frequency switching currents. When a switch mode must be used, a constant frequency type should be used in preference to a variable frequency design. If more than one switch-mode is present, in order to avoid randomly interference fading effect, arrange for them all to run at the same frequency (many designs offer external frequency synchronization inputs). For more information, please check individual manufacturer datasheets when using such components. The use of the right filter elements and optimal layout is here crucial.
8. Place the filter capacitors so that their terminals are directly connected to the PCB traces that carry main current to be filtered (no extra trace length).
9. Keep the distance between the converter and the filter capacitors as short as possible to reduce parasitic effects and transient current flow.
10. Pick the smallest package for a given capacitance. Physically small capacitors tend to have lower parasitic inductance than physically large ones.
11. A short capacitor has less inductance than a long one, and a high profile capacitor has less inductance than a low profile capacitor. Use at least two capacitors which differ by a factor 100 in value to decouple. The reactance of large capacitors has a significant inductive component at higher frequencies.
12. Because of its inherent inductive component, a single large capacitor is not very effective against high frequency noise. Using paralleled capacitors like tantalum (e.g. at least 22  $\mu$ F) in combination with ceramic capacitors (e.g. 0.1  $\mu$ F) reduces filter impedance across a wide frequency band (see Fig. 2). Among the paralleled capacitors: Place the capacitors closest to device.
13. When laying out the PCB, always make provisions for as many noise suppression elements like i.e. capacitors as possible. Consider this as risk reduction for the debug phase of your design. If i.e. the capacitors are not needed, just leave them out when the board is assembled.
14. When you intend to use high value ceramic SMD capacitors instead of tantalum as filter elements at output of i.e. a DC/DC converter please note that their nominal capacity depends very strong on the specific dielectric and rated voltage, see below for different usual materials in comparison to low value capacitors material (COG). Therefore, as general rule you shall choose at least their double nominal value, so i.e. if you need 10  $\mu$ F/6.3 V you shall rather use 22  $\mu$ F/6.3 V ceramic capacitor, more is better.

Capacitance of Ceramic Capacitors at 10%, 50% and 100% of Rated Voltage (DC), "Typical".

Dielectric:	COG	X7R	Z5U
%C at 10%:	100%	105%	120%
%C at 50%:	100%	95%	50%
%C at 100%:	100%	70%	30%

15. For heavy environments, you may also consider the use of shield planes and discrete inductors e.g. ferrite beads or common mode chokes inserted into the signal or power line paths. Their handling requires however some experience.

POWER SUPPLY LAYOUT –  
Layout considerations for Line-Power Supplies

16. If possible, use a Spectral analyzer and a near field probe set to identify the interferences and their sources on PCB.

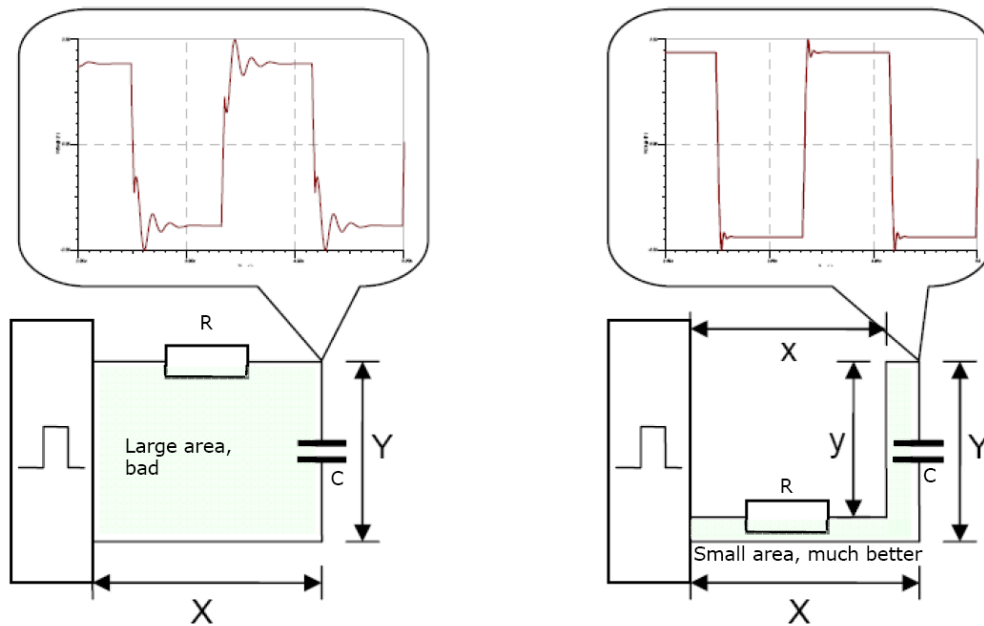
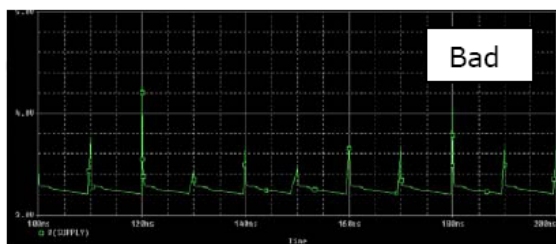
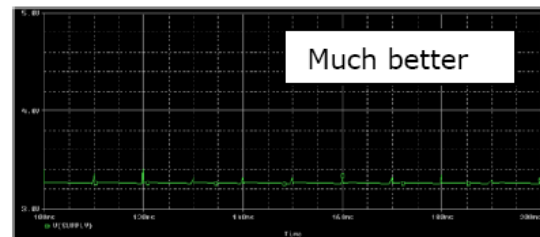


Fig. 1: Effect of large and small loop areas



6.8  $\mu$ F, Ripple: 1 Vpp



6.8  $\mu$ F + 0.1  $\mu$ F + 0.01  $\mu$ F,  
Ripple: 0.1 Vpp

Fig. 2: Ripple voltage at power supply output,  
without and with small capacitors connected in parallel

## POWER SUPPLY LAYOUT –

Layout considerations for Line-Power Supplies

---

**Additional, specific power supply design hints regarding EnOcean based devices (please always consult Dolphin Core Description and current user manuals)**

- **Important:** at start-up and after wake-up from sleep a current peak of up to 600 mA will be drawn for up to 3  $\mu$ s, see Dolphin Core Description. This must be taken into account for the design of power supplies, respectively energy budget calculations. Thereafter the current consumption of the TCM module alone is up to 40 [mA@3.3 V](#), minimal required start-up voltage 2.7 V. According Dolphin Core Description tolerated supply voltage slope (valid for rising and falling) is max. 5 V/ms.
- In order to provide a good radio performance, great attention must be paid to the power supply and a correct layout and shielding. It is recommended to place a 22  $\mu$ F ceramic capacitor between VDD and GND, close to the device (material: X5R, X7R, min 6.3 V to avoid de-rating effects).
- In addition, an HF SMD EMI Suppression Ferrite Bead such as the Würth WE-CBF HF SMD EMI Suppression Ferrite Bead (Würth order number 742863160) with DC resistance < 1 Ohm shall be inserted in the power supply line.
- For a good performance, the ripple on the power supply rail should be below 10 mVpp. All GND device pins must be connected to GND.
- The length of lines connected to I/Os should not exceed 5 cm.
- It is recommended to have a complete GND layer in the application PCB, at least in the area below the module and directly connected components (e.g. mid-layer of PCB).

**Take care to the following advices to prevent interference with your application:**

- It is recommended to avoid any copper structure in the area directly underneath the TCM module (top-layer layout of PCB). If this is not possible, coating on top of the PCB should be provided in order to prevent short circuits to the module. All bare metal surfaces including Vias have to be covered (e.g. adequate layout of solder resist).
- It is mandatory that the area marked by the circle in the foot print figure is kept clear of any conductive structures in the top layer and 0.3 mm below. Otherwise, RF performance will be degraded!
- Furthermore, any distortive signals (e.g. bus signals or power lines) should not be routed underneath the module. If such signals are present in your design, we suggest separating them by using a ground plane between module and these signal lines.
- Avoid if possible to use in your design switch mode power supply working with a frequency around 125 kHz or one of their first (sub)harmonics, knowing that the transceiver use a 125 kBaud data rate (corresponding to 8  $\mu$ s/bit). Therefore, switch power supplies working at frequencies corresponding to first sub-harmonics and harmonics of 125 kHz (i.e. around 63 kHz, 125 kHz and 250 kHz) should be avoided. These interferences can be coupled direct into receiver and could affect the signal integrity by reducing the receiver sensitivity and/or producing Rx and/or Tx interrupts (hang-ups) because such interferences could be interpreted as possible "preambles" of permanently incoming telegrams).

POWER SUPPLY LAYOUT –  
Layout considerations for Line-Power Supplies

---

**Related documents:**

**EnOcean Dolphin Core Description:** <https://www.enocean.com/dolphin-core-description/>

**EnOcean TCM xxx user manuals** <https://www.enocean.com/en/products/>

**EnOcean Appnotes:** <https://www.enocean.com/en/support/application-notes/>

**Disclaimer**

The information provided in this document is subject to modifications and intended for reference purposes only. EnOcean assumes no liability either for violation of industrial property or other rights of third parties that stem from this information, nor for errors and / or omissions.

We reserve the right to make changes without prior notice. Make sure that this is the latest version of the document before use. For the latest documentation, visit the EnOcean website at [www.enocean.com](http://www.enocean.com)