June 1996

Application Note 42007

by Daryl Sugasawara
and Jens Paetau

Safety, EMI and RFI Considerations

INTRODUCTION

The filtering of conducted and radiated noise is an intricate part of the design of a power supply or DC to DC converter. This Application Note addresses the origin of generated EMI and RFI and the best ways to control them. The topics discussed will be:

- Background on Safety Agencies and Documents
- EMI filter design for higher power Switch Mode Power Supplies
- EMI sources and ways to control them
- PCB layout considerations
- EMI and RFI Filter considerations for low power boost converters; ML4861 design example

SAFETY AGENCIES AND BACKGROUND

The main objective of the national and international safety regulatory agencies is to provide the user with a safe and quality product which is not going to interfere with other electronic equipment. These safety agencies and standards are different depending on the country they originated. Equipment manufacturers that would like to sell their product in these countries first need to get their product approved by the corresponding safety agencies. Most power supply manufacturers use the IEC (International Electro Technical Commission), VDE (Verband Deustcher Electrotechnik), or UL (Underwriters Laboratories) and CSA (Canadian Standards Association) as their base to the majority of the world's safety requirements. Regarding the radiated and conducted interference levels acceptable in the United States and internationally are FCC's Docket 20780 and VDE's 0806. One point to consider is that both, FCC and VDE standards exclude sub assemblies from compliance to these rules. This is understandable since power supply radiation and conduction characteristics can vary depending on different system level loading. Therefore the final product, where the switching power supply is to be used, must also comply to EMI and RFI specifications. Both agencies require manufacturers to minimize the radiated and conducted interference of their equipment which is connected to the AC mains and employs high frequency digital circuitry.

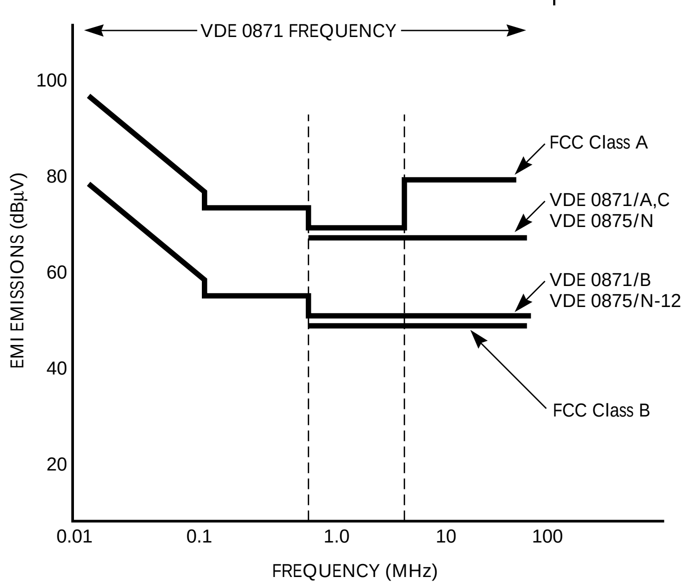


Figure 1. FCC and VDE Compliance Curves

VDE has subdivided its RFI regulations into two categories

a) 0-10KHz unintentional high frequency generation (VDE 0875, VDE 0879)

b) 10KHz-30MHz intentional high frequency generation (VDE 0871, VDE 0872)

The FCC includes all electronic devices which generate signals at a rate greater than 10KHz. The FCC and VDE regulations closely follow each other. The FCC class A specification covers business, commercial and industrial environments, while FCC B covers residential environments only. The main difference as can be seen in figure 1 is the frequency span covered by both agencies. The VDE frequency range for EMI and RFI emissions covers a spectrum from 10KHz to 30MHz, while FCC's frequency span covers only the range of 450KHz to 30MHz.

EMI FILTER DESIGN IN SWITCH MODE POWER SUPPLIES

How is EMI generated? Technically, EMI is generated by a varying electric or magnetic field and transmitting them by means of conductive, inductive or capacitive coupling, through free space or a combination of these means. Switching power supplies are one of the worst sources for EMI and RFI generation because of their inherent current voltage waveforms and very fast switching times. Switching transistors, MOSFETs, diodes, transformers, and inductors are the main source of RFI generation. The common mode noise generated by switching is a problem in large computer systems and can be controlled with an input filter between line, neutral and chassis. The differential noise, like transient response, is a function of the output filter capacitors and filter chokes.

REV. 1.0 10/25/2000

To properly contain the conducted noise in power supplies, ceramic capacitors help reduce the high frequency noise and should be located as close to the connector (input/output) as physically possible. Note that some power supply topologies are better than others when it comes to the conducted noise. For example, a flyback power supply has a triangular current waveform which inherently generates less RFI noise than other converters such as a forward or a bridge converter that switch rectangular current waveforms.

Suppressing noise in switching power supplies is a very tricky business. Ideally EMI and RFI should be contained with a minimum increase in weight, circuit complexity, cost and efficiency. The idea is to block, or bypass the interference noise. This can be achieved by introducing a high impedance into the path of the interfering currents and bypassing them to ground through a low impedance path. One way to do this is to place filters on the input and output leads, but this is only helpful in cases where the system ground is very close to chassis ground.

For the AC lines, a coupled inductor with very low stray capacitance, two safety approved capacitors (x type) between the lines, and small capacitors (y type) between each line and ground should suppress switching noise to acceptable levels.

These capacitors and inductors are typically within the following values:

Cx = 0.1µF to 2µF

Cy = 2200pF to 33000pF

L = 1.8mH at 25A to 47mH at 0.3A

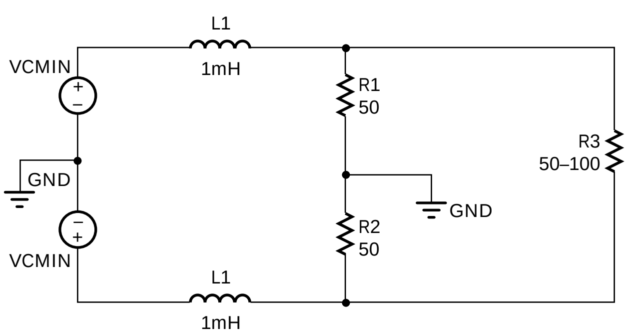


Figure 2A. First Order Filter

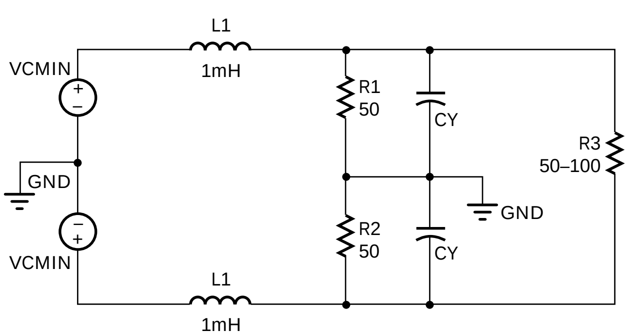


Figure 2B. Second Order Filter

The value of the common mode inductor for a first order filter (Figure 2A) is simply the load in ohms divided by the radian frequency at and above which the signal is to be attenuated. For example, attenuation above 4000Hz into a 50 ohm load would result in a 2mH inductor (50/2π4000). This would imply that the attenuation at 4000Hz would be 3dB, increasing at 6dB per octave.

The second order filter (Figure 2B) has two advantages in that it provides 12dB per octave attenuation and it provides greater attenuation at frequencies above the inductor self resonance. The typical common mode transfer function can be expressed as follows:

$$\frac{VCM_{OUT(S)}}{VCM_{IN(S)}} = \frac{1}{\frac{1+L}{R_L s + LCs^2}}$$

The damping factor which corresponds to the gain of the filter at the 3dB point, should be chosen to be anywhere between 1 and 4. In this example a value of 3 chosen and the following can be deduced:

$$\zeta = 3 = \frac{L\omega_N}{2R_L}$$

If we choose a cutoff frequency of 15KHz and a load resistance at the cutoff frequency of 50Ω, we can calculate the inductance as:

$$L = \frac{3 \times 2 \times R_L}{2\pi \times f_N} = \frac{3 \times 2 \times 50}{2\pi \times 15000} = 3.2\text{mH}$$

The filter capacitance (y-capacitors) can be calculated as:

$$C = \frac{1}{(2 \times \pi \times f_N)^2\ L} = 0.035\mu\text{F}$$

If necessary, adjustments can be made to these components in order to reduce leakage current requirements set by the safety agencies. Another important point is to make sure that the resonant frequency of the input filter is lower than the switching frequency of the power supply. Higher frequency power supplies are usually easier to filter than lower frequency supplies.

The discharge resistor as recommended by VDE 0806 and IEC 380 is calculated as follows:

$$R = \frac{1}{2.21 \times C_X} \approx 1\text{M}\Omega$$

Where t = 1sec and C is the sum of x capacitors.

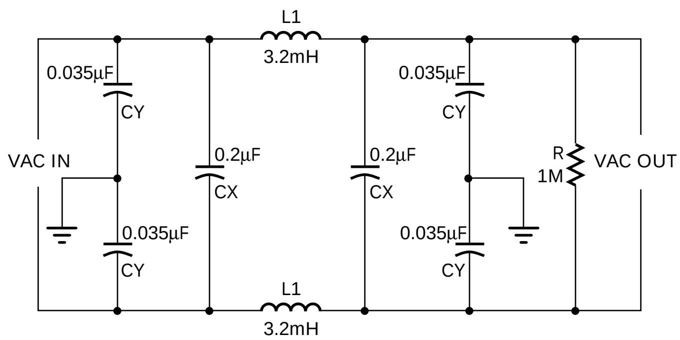


Figure 3. Second Order Filter

The final EMI filter for a power supply operating at power levels below 200W is shown in figure 3.

Further improvements to the frequency response of this pi filter can be achieved by inserting several such sections in series, but this would increase cost, weight and space. Other solutions which help the filtering of switching noise is converting the storage capacitors into a pi filter section, and connecting an extra line choke between the storage capacitors. Note that there are several manufacturers of EMI filter modules or components which can provide the power supply designer a ready solution for EMI and RFI suppression.

EMI SOURCES AND WAYS TO CONTROL THEM

Design considerations should be taken in the following areas, which are the typical RF generating sources:

- DC isolation
- Circuit grounding
- Susceptibility to audio frequency and RF noise conducted to the supply on power lines
- Interference generated in the supply and conducted to the other parts of the system on power lines
- Radiated interference and susceptibility
- Turn-on, turn-off transients

All of these problems can be solved by placing a component in the right place such as to avoid interference. This makes the layout of the power supply one of the most crucial steps when developing a switcher. Here are some hints on what to keep in mind when developing and laying out a power converter:

Care should be taken in the generation of fast rise current spikes resulting from the sudden reverse biasing of diodes used in the transformer rectifier circuitry.

Ringing in the transformer can be clamped out by placing snubbers across the windings. Snubbers are discrete circuits which reduce the switching transients dissipatively as heat, or non-dissipative, by returning the energy back to the input.

A small ferrite bead on the leads of the main MOSFET switching devices will suppress the generation of high frequency ringing at turn-on and turn-off periods.

A faraday shield, returned to DC ground, and placed between the primary and the secondary windings will prevent capacitive coupling of voltage transients into the power supply and output leads.

In order to minimize the lower and higher frequency harmonics generated in the transformer, a high permeability material must be chosen. This results in low exciting volt amperes and low core losses, which means better efficiency, smaller size and reduced weight. Also, decoupling RF noise at the source by using good high frequency capacitors with low inductance and low ESR is helpful.

Lead lengths needs to be kept to a minimum to avoid RF energy radiating into free space.

Decoupling outgoing leads with small capacitors will localize conducted energy before spreading to the rest of the circuitry.

PCB LAYOUT CONSIDERATIONS

It is always possible to contain or fix EMI and RFI to a certain degree, but the real trick is at the layout stage of the PC board. Consider switching waveforms as the ones generated from a DC to DC converter and the traces which carry these signals as the antennas. These traces are related in a complex way with the radiated fields from the converter. The prediction of radiation from a circuit requires the identification of the frequency content and their absolute magnitudes of the time domain waveform. Of particular importance is that there is no energy contained below the fundamental frequency, and 99% of the energy is contained below $1/(\pi\ tr)$ (where tr is the rise time of the waveform). Note that the edge rate is a key driver (or limiter) for the frequencies of concern. Power supply and DC to DC converter designers try to achieve fast rise and fall times to minimize the switching losses, but this high di/dt conditions will generate ground and trace bounce voltages which can excite components and cables to radiate in the common mode manner. Multi-layered boards which use good ground and power planes provide the low impedance power distribution necessary for good power supply decoupling. The following are some of the techniques used to minimize EMI and RFI generation:

Enclosing signals between power and ground planes achieves a locally shielded enclosure that reduces radiation by 30 to 40dB and also reduces radiated susceptibility and ESD susceptibility. Think of traces as transmission lines. A good rule is to keep signals with 1nsec rise or fall times shorter than 9cm (3.5in).

Traces which carry high frequency currents should be surrounded by a coplanar ground trace in order to reduce both radiation and crosstalk to other traces.

Another good design practice is to provide a chassis ground ring around the periphery of the board. This ring provides a formidable shield or field interceptor to prevent radiation at the circuit boundaries. This layout practice is also recommended as an interceptor for Electrostatic discharges, and thus provides a more robust design.

The use of ferrites is very effective to directly suppress high frequency energy by both reactive impedance and absorptive losses. Ferrite beads for example use soft magnetic materials which require little energy to alter the magnetic flux. These ferrites become more resistive at high frequencies, which effectively reduce the bandwidth and higher frequency energy content of digital signals. Ferrite chokes are also commonly used in loose wires to cancel out common mode and differential mode currents. These ferrites come in all kinds of shapes, sizes and mounting options (including surface mount).

Try and keep high frequency currents to a local area of the PCB. This practice limits the capability of those currents to excite an efficient radiator.

PCB Layout and Component Selection Summary:

1. Always use a solid ground and power plane (multi layer boards are preferred).
2. Border PCB with chassis ground trace.
3. Centrally locate high frequency clock circuits. (distribute signals symmetrically)
4. Locate line drivers and receivers close to the connectors.
5. Decouple high frequency currents locally.
6. Use shielded components.
7. Use bulk capacitors near ports.
8. Use ferrites for input/output lines.
9. Use narrow and buried traces when possible. (depends on the current it needs to handle)
10. Use high frequency ceramic (surface mount when possible) capacitors and locate close to the pin.
11. Keep high di/dt lines as short as possible.
12. Keep input and output leads away from electromagnetic noise generators.
13. Minimize capacitive coupling to chassis grounds.

EMI AND RFI CONSIDERATIONS FOR LOW POWER BOOST REGULATORS; ML4861 DESIGN EXAMPLE

The ML4861 is a small boost converter used in 1 to 3 cell battery applications. This DC to DC converter comes in three versions which are: 3.3V, 5V, 6V or adjustable V_{OUT}. One of the main features of this small boost converter is the fact that it only requires one inductor and two capacitors to form a complete battery boost converter operating from 1 to V_{OUT}–0.5V and offering very good efficiency (>90%). Some applications however, require that conducted and radiated noise be maintained to a minimum within a certain window of the frequency spectrum. In such cases, the appropriate selection of components and layout is very important. For example some pager or cellular applications may require to keep EMI and RFI noise to a minimum from 1MHz to 400MHz, in which case the following considerations should be made:

Use the above mentioned layout considerations when designing the PCB to minimize the radiated and conducted energy.

Use a shielded inductor with the appropriate inductance, maximum current and lowest possible DC Resistance value to optimize the efficiency of the converter. Use the tables shown in the data sheet of this device, and place it as close to the IC as physically possible. Note, the higher the inductance, the better the efficiency, but the lower the maximum current available. Also, the higher the input voltage, the higher the efficiency. For a given input voltage (1, 2 or 3 cells), select the highest inductance which meets the desired maximum output current, efficiency and size limitations of the design. The recommended inductor manufacturers which offer shielded power inductors are:

Coiltronics-CTX Series	Tel. (305) 781-8900
Sumida-CDRH Series	Tel. (708) 956-0666
Sumida-CDR63B, CDR74B and CDR105B Series	

Use low ESR capacitors close to the output voltage pin to minimize ripple voltage. It is also recommended to use two capacitors (one at the output of the ML4861 and one at the port) in case long VCC lines are required. The recommended capacitor manufacturers are:

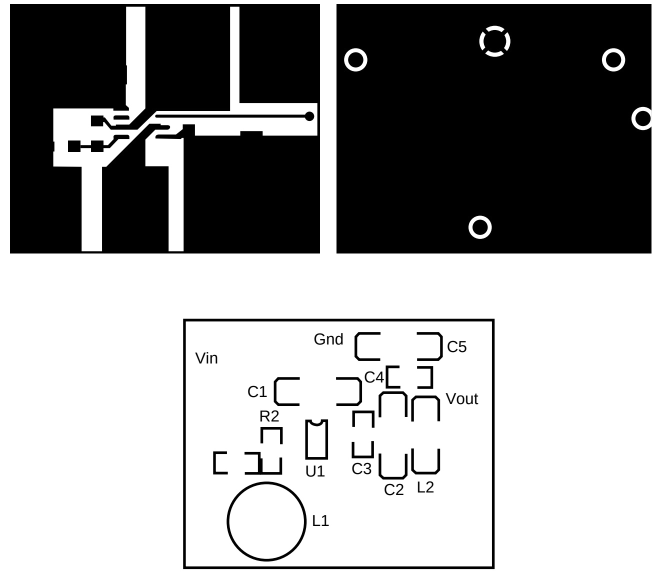
Matsuo-267M Series	Tel. (714) 969-2491
Sprague 595D Series	Tel. (603) 224-1961

To further reduce the conducted noise to the rest of the circuitry, use high frequency ceramic capacitors preceded by a small ferrite bead to shunt high frequency noise as shown in Figure 4, Option 2. Another possibility is shown as Option 3, where a small, high frequency EMI filter is connected directly to the output port. When using these small filters, it is important to have a solid ground plane connected to chassis ground to effectively shunt the high frequency energy to ground. The recommended filters are:

Murata Erie Tel. (800) 831-9172
NFM40R Series (Rated current = 200mA)
NFM41R Series (Rated current = 300mA)
NFM61R/H Series (Rated current = 2A)

The selection of these filters improves the EMI and RFI feedthrough at the expense of efficiency of the converter. It is therefore important to choose a low DC resistance device which also meets the required converter current and insertion loss characteristics.

Included is the recommended layout and schematic. Note that the schematic includes several components which are optional and may not be required for the majority of applications.



ML4861 Suggested Layout

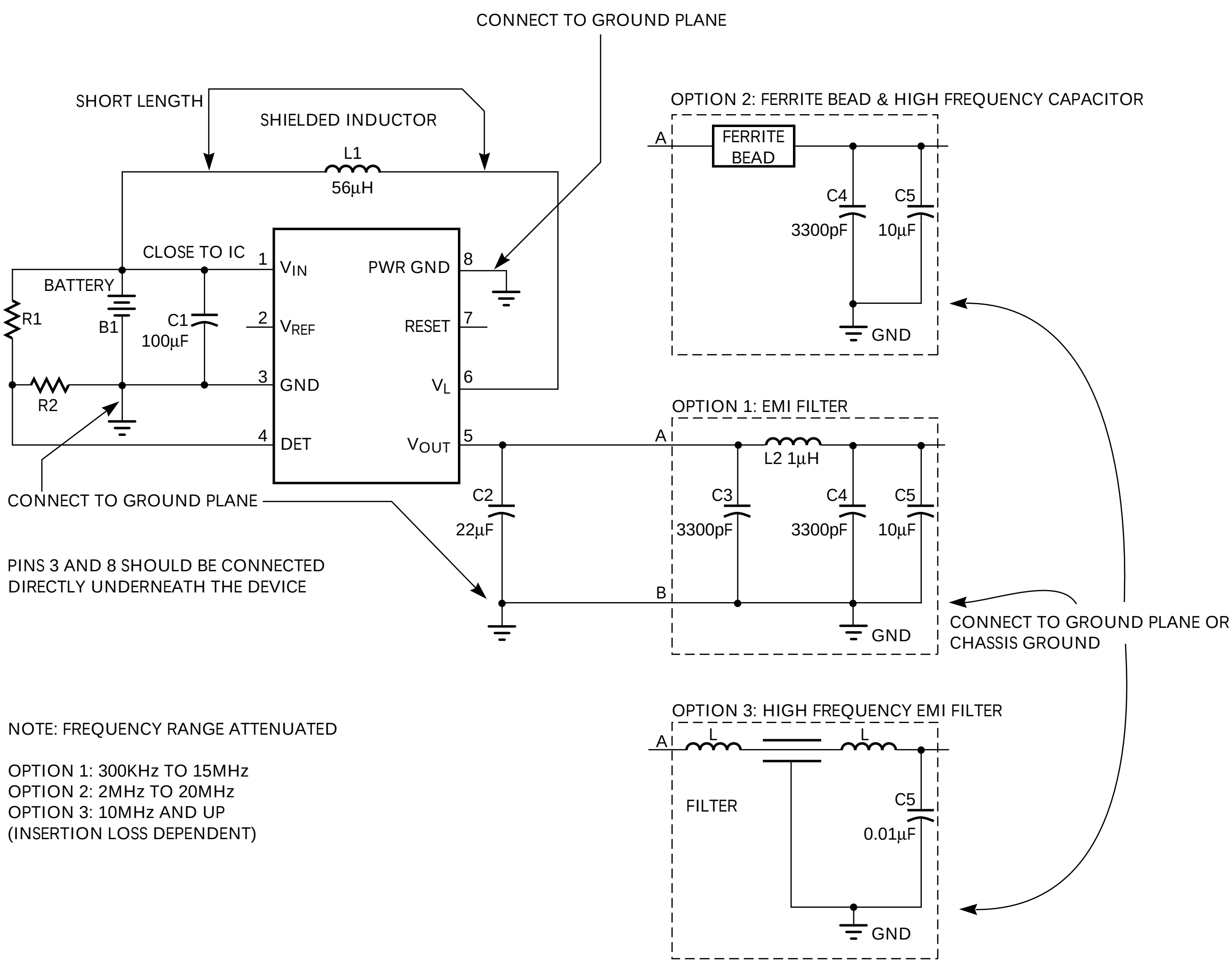


Figure 4. ML4861 EMI/RFI Circuit Suggestions