Another EMC resource
from EMC Standards

EMC design of Switching Power Converters Part 7 (continued 6)

Helping you solve your EMC problems

9 Bracken View, Brocton, Stafford ST17 0TF T:+44 (0) 1785 660247 E:info@emcstandards.co.uk

EMC design of Switching Power Converters

Part 7 (continued 5) — More PCB issues, plus high-performance filtering

First published in The EMC Journal, Issue 103, November 2012

Keith Armstrong, Cherry Clough Consultants Ltd, www.cherryclough.com

Contents

Issues 93-101 of The EMC Journal carried the preceding parts of this "Stand Alone" series – my attempt to cover the entire field including DC/DC and AC/DC converters, DC/AC and AC/AC inverters, from milliwatts (mW) to tens of Megawatts (MW), covering ***all*** power converter applications, including: consumer, household, commercial, computer, telecommunication, radiocommunication, aerospace, automotive, marine, medical, military, industrial, power generation and distribution, in products, systems or installations. Hybrid & electric automobiles, electric propulsion/traction; "green power" (e.g. LED lighting); and power converters for solar (PV), wind, deep-ocean thermal, tidal, etc., are also covered.

Issues 93-95 used a different Figure numbering scheme from the rest, for which I apologise. I generally won't repeat material I have already published, instead providing appropriate references to the EMC Journal [14] and my recently-published books based on those articles [15], so that you don't get bored by repetition.

7 Suppressing RF emissions from inputs and outputs

I began Section 7 in Issue 98 [72] and so far it has continued up to Issue 102 [104]. Despite my aim to only publish 'stand-alone' articles, each covering a single topic, the issue of suppression is so large that it is impossible to publish it all in a single issue. However, I have – mostly – succeeded in compartmentalising individual suppression topics in each article.

7.7 PCB planes for small non-isolating DC/DC converters

Some DC-DC converter manufacturers recommend using a separate small component-side 0V plane for the 0V reference for the DC-DC components only, connected to the main 0V plane at one point by a large-diameter via hole.

The idea is to try to restrict the switcher/chopper's intense circulating return currents to the small plane, single-point bonded to the main plane, so they don't cause much noise in the main 0V plane. But this is merely a modern restatement of the traditional "single-point earthing/grounding" method – actually a bad electronic design (and bad EMC engineering) practice that only has any merit when used to compensate for other bad electronic design (or bad EMC engineering) practices, on the basis that two wrongs (sometimes) equals a right.

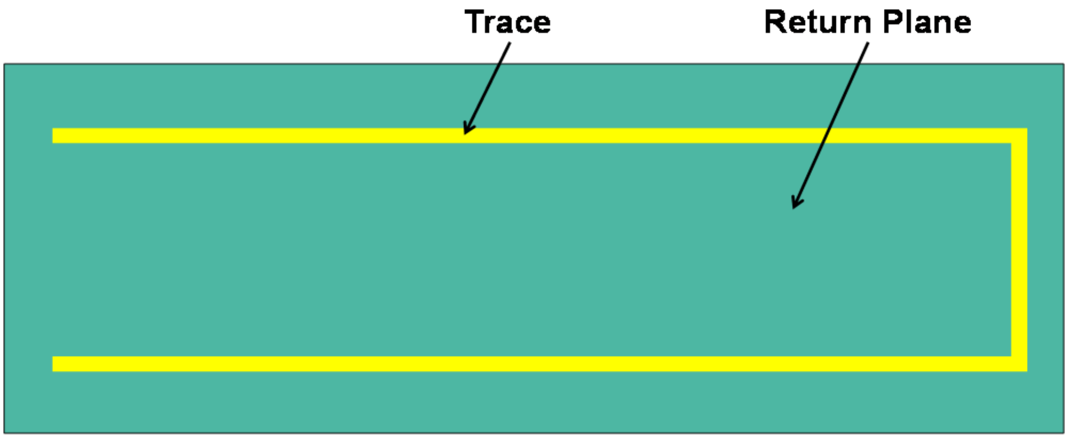
For more on why single-point earthing/grounding is now generally considered to be bad practice for controlling any frequencies above a few hundred Hertz, read Chapter 5.1.2 in [4] (repeated as Chapter 2.7.1.2 in [5]), Chapter 4.6.8 in [5], 3.1.4 and 4.3 in [37].

The plain fact is that when all of the good EMC practices are employed in circuit design and PCB layout, as described in [5] and [37] especially as regards good modern power decoupling practices, return currents in planes circulate very close indeed to the devices that caused them, *and don't travel outside that area into other areas of circuitry on the PCB to interfere with them*.

The reasons for this are described in general in Chapter 5 of [4] (repeated in Chapter 2.7 of [5]) and – more specifically for PCBs – in [85]. Figures 7 and 8 in [85], which I copied from [105], are especially powerful. They show that for a bent conductor 20mm above a metal sheet that is used as its return path, at frequencies above 1kHz all the return current flows underneath the conductor – following its bends – and doesn't flow through the rest of the metal sheet.

Figures 7 and 8 in [85] concern an electrical assembly, but I have often seen the exact same result demonstrated for PCB traces over planes, at frequencies up to 10GHz. However, until I read [105] I had never found a set of good publicly-available graphics for this effect in PCBs, and so the best I could offer when I wrote [4] and [5] was the graph of Figure 36 in [4] (repeated in [5] as Figure 2AM).

But just recently, I read [106], a wonderful series of articles on good grounding techniques in PCBs, which includes some current distribution plots of PCB planes created by Bruce Archambeault of IBM, which I now present as Figures 7.10-1 through 7.10-5.



The physical geometry for this example:
Trace total length = 22" (i.e. 559 mm)
Trace is 10 mils wide, 1 mil thick, 10 mils above return plane
(i.e. 0.25 mm wide, 2.5 microns thick, 0.25 mm above ground plane)
(Drawing courtesy of Dr. Bruce Archambeault)

From: "Successful PCB grounding with mixed-signal chips - Part 1: Principles of current flow", by Mark Fortunato of Maxim Integrated Products, in EDN Analogue Network, August 28, 2012

Figure 7.10-1 The physical geometry of the example

The distribution of current for these higher frequencies is given by the following equation.[4]

$$J(x) = \frac{I}{w\pi}\left[\tan^{-1}\left(\frac{2x-w}{2h}\right) - \tan^{-1}\left(\frac{2x+w}{2h}\right)\right] \qquad \text{(Eq. 1)}$$

Where:

J(x) is the current density;

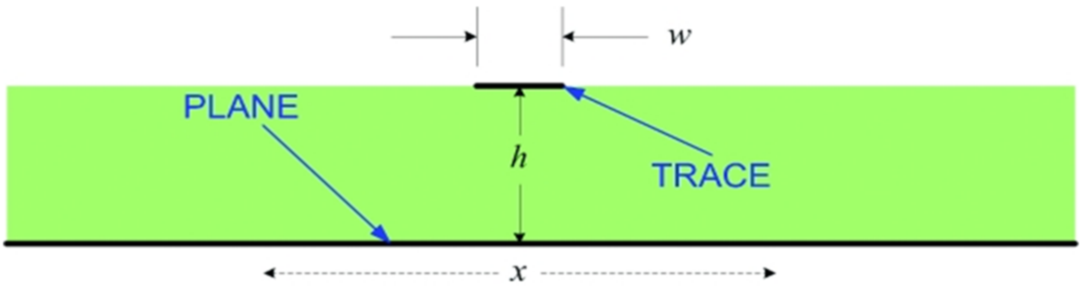
I is the total current;

w is the trace width;

h is the board layer thickness (the height the trace is above the plane);

x is how far from directly under the trace we measure the current, as shown in **Figure 13**.

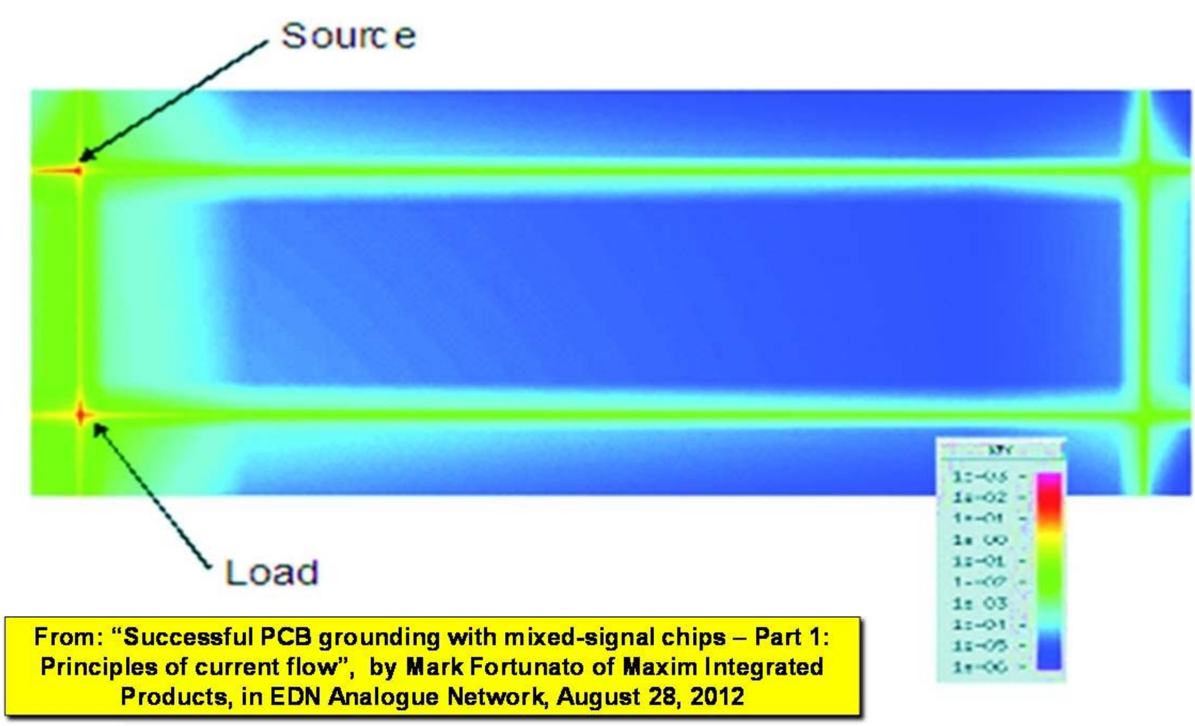
From: "Successful PCB grounding with mixed-signal chips – Part 1: Principles of current flow", by Mark Fortunato of Maxim Integrated Products, in EDN Analogue Network, August 28, 2012



Cross section of board.

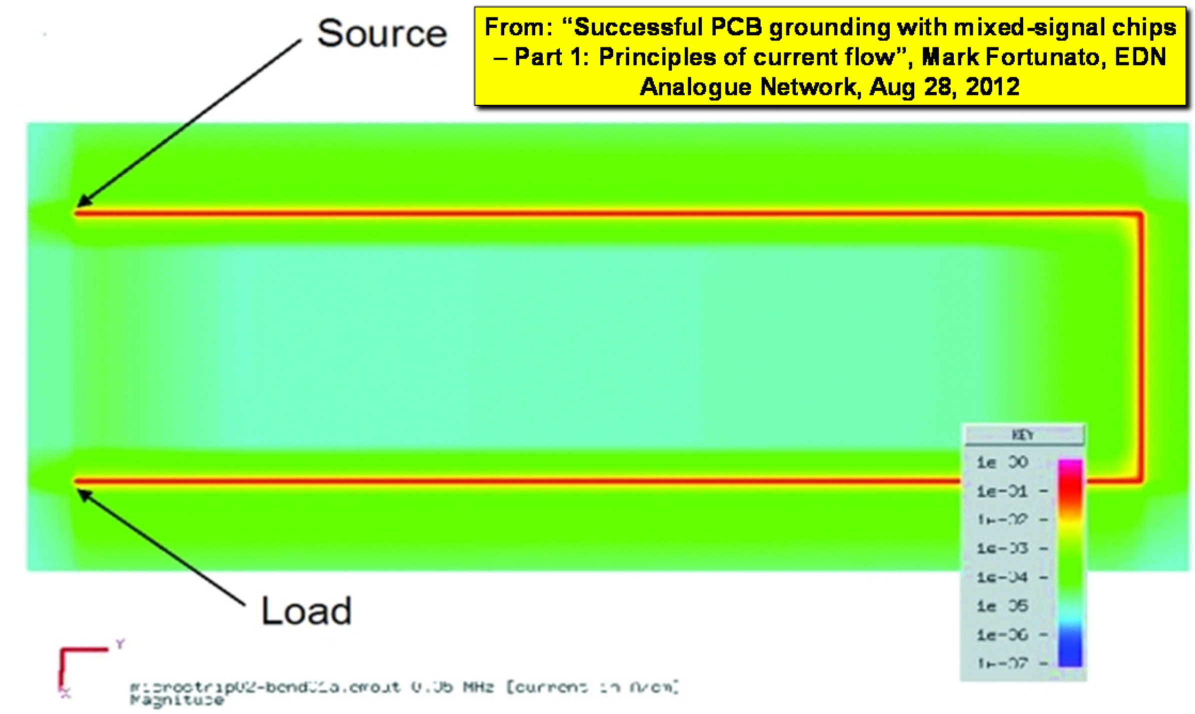
[4] Ott, Henry W., Electromagnetic Compatibility Engineering, John Wiley and Sons, 2009. ISBN 978-0-470-18930-6, p 392

Figure 7.10-2 The maths behind the return current's distribution in the plane



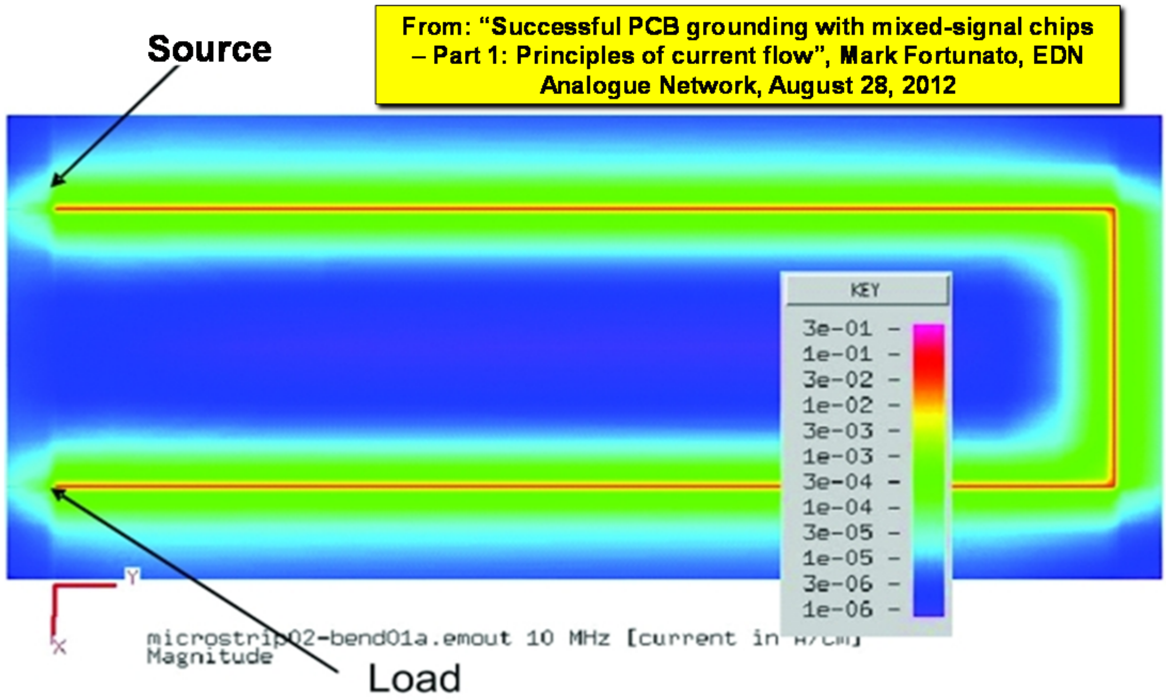
1kHz ground current flows from load to source in a straight line. (Drawing courtesy of Dr. Bruce Archambeault.)

Figure 7.10-3 Send current 1kHz, return current mostly doesn't follow the trace's route



50kHz ground current flows everywhere. (Drawing courtesy of Dr. Bruce Archambeault.)

Figure 7.10-4 Send current 50kHz, return current mostly follows the trace's route



1MHz ground current follows the signal trace. (Drawing courtesy of Dr. Bruce Archambeault.)

Figure 7.10-5 Send current 1MHz, all return current follows the trace's route

Note that where there is a metal plane close to a conductor, the RF return current flows in it. Most simulations (like the above figures) assume a 0V or ground plane, but the exact same effect occurs for the return currents that flow in power planes, which is why the modern understanding of power rail decoupling as described in Chapter 5 of [37] is so very important for the good EMC design of PCBs carrying modern and future ICs.

Dr Archambeault's graphics, as reproduced in [106] only go up to 1MHz, but the equation in Figure 7.10-2 and all the simulations and real-life tests with surface-current probes that have ever been done, show that – up to a point – the higher the frequency, the closer the return current path "sticks" to the route taken by the trace carrying the send current.

So plots with 10MHz, 100MHz, 1GHz etc. send currents would show the width of the return current in Figure 7.10-5 narrowing until it was about 0.75mm (30 mils) wide, centred under the trace.

Now let's be quite clear about this, so there can be no mistake or misunderstanding for the currents flowing in PCB planes due to the traces routed on that PCB:

- Above some frequency, almost all of the trace's return current in the plane follows the route taken by the trace, bends and all.

  (The actual frequency depends on the geometry of the trace and its return plane.)
- Above that frequency, at a small distance away from the route taken by any trace, its return current has a negligible current density, so has a negligible effect on any other circuits.
- Appropriate segregation of circuit areas on the PCB, together with good modern power rail decoupling and other good EMC design and layout practices, means that one circuit area will not interfere with another even though they share the same PCB plane.
- So there is no need for any plane splits. The bad EMC practice of splitting PCB ground planes has no place in a good EMC design and PCB layout.

The relevant Maxwell's Equation is Faraday's Law of Electromagnetic Induction, which tells us that all current loops (even if they are due to stray coupling) always take the path of least overall impedance. Above some frequency that depends on the geometry of the trace and its return plane, the loop having the least impedance is dominated by mutual inductance and capacitance, and not by resistance, which is why the return current sticks close to the route taken by the trace as shown by the above figures.

Notice that the mathematical expression in Figure 7.10-2 assumes a return plane of infinite size, and so Dr Archambeault's current density plots imply zero RF current in all plane areas that are not close to the send trace. But no-one makes electronic products with infinite ground planes in them (because they couldn't afford the shipping costs, amongst other things).

Dr Archambeault knows this to be true, because in [107] he describes the effect of limiting the return plane dimensions. Long story short – only about 95% of the return current in a plane flows under the trace, the remaining 5% being spread all over the PCB with an exponentially reducing current density as the distance from the trace's route increases.

The effect of limiting the size of the ground (or 0V) plane is that all of the small amount of current that would have flowed off to infinity is concentrated at the edge of the PCB, where it is a cause of emissions. This is the reason why it is good EMC practice to extend 0V (or ground) planes at least 3mm beyond any traces, and beyond any components on the PCB by at least twice their height – effectively a moat of 0V or ground plane surrounding the area containing the components and their traces.

It is also a reason for several of the other good PCB layout practices described in [37] and my PCB EMC design and layout training courses.

So – returning to the recommendations by some DC-DC manufacturers to use a separate small component-side 0V plane for the 0V reference for the DC-DC components only, connected to the main 0V plane at one point by a large-diameter via hole – what can we now say about this?

We can say that it is most probably an unnecessary practice, when good EMC design and layout practices are used. If used, it can be expected to increase the emissions from the PCB at frequencies at which the DC-DC converter components with their dedicated 0V plane resonate.

For designers who are uncomfortable in going against the board layout recommendations of their suppliers, I always recommend that they follow supplier recommendations but add details that allow them to add components to the PCB that bring it more into line with good modern EMC design practices.

Then, if failing to meet EMC requirements, they have a quick way to experiment with PCB modifications that should help get them out of trouble, without the delay of a 'respin' of the board layout.

In the case of the DC-DC converter advice being discussed here, I recommend surrounding the converter's component-side 0V plane area with a closely-spaced ring of trace that is via'd to the main 0V plane at least every $1.5/f_{max}$ metres along its entire length (i.e. its entire circumference), where f_{max} is the highest frequency that it is necessary to control in MHz, assuming an FR4 board dielectric.

For example, where f_{max} is 300MHz, the spacing between the capacitors or zero-Ohm links spaced all around the converter's component side 0V plane, must not be greater than 5mm (preferably much less).

Then, if the converter appears to be contributing significantly to the board's emissions, capacitors or zero-Ohm links can be manually soldered all around the perimeter, between its component-side 0V plane and the closely-spaced trace ring, see Figure 7.10-6.

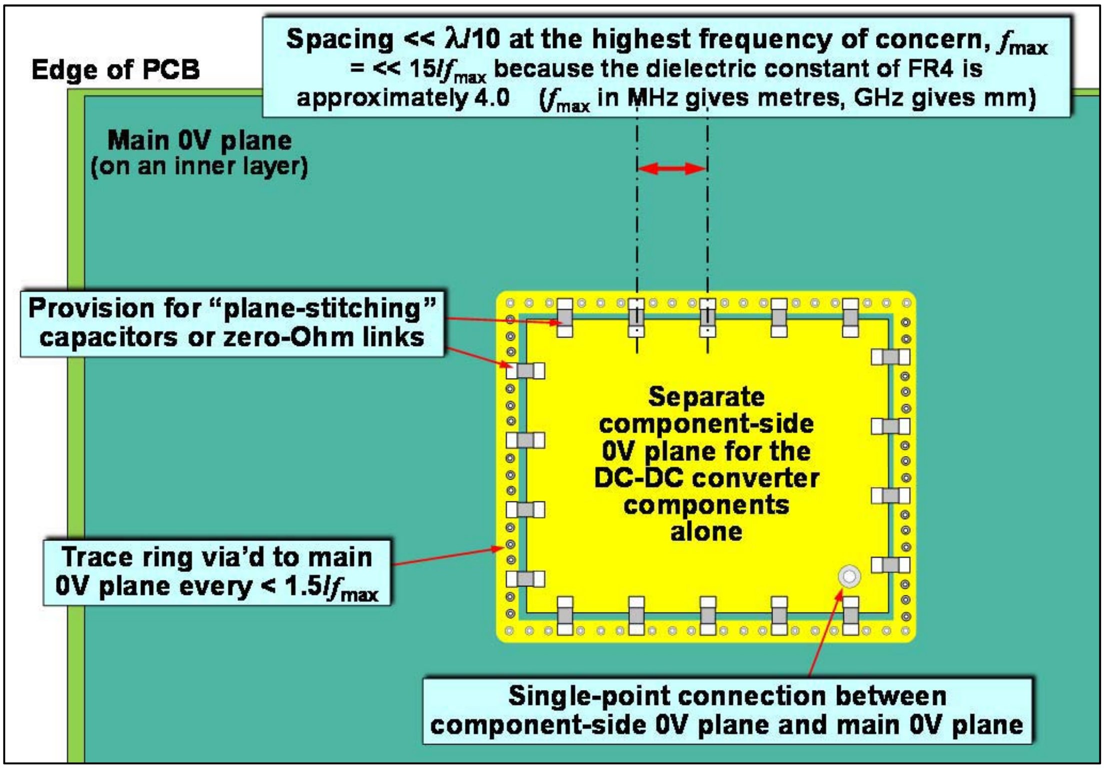


Figure 7.10-6 Adding a trace ring to an isolated component-side 0V plane

The spacing between these “plane-stitching” components must not be greater than 15/f_{max} metres. For example, where f_{max} is 300MHz, the spacing between the capacitors or zero-Ohm links spaced all around the converter’s component side 0V plane, must not be greater than 50mm.

Larger spacings than 15/f_{max} run the risk of *increasing* emissions by reducing their slot resonant frequencies by too much, so they occur within the frequency range to be controlled. Where this technique proves necessary, much smaller spacings than 15/f_{max} will generally give much better suppression.

7.10 High-performance filtering

It is always tempting to assume that the filter suppliers’ shortform data tells us what attenuation we can expect in real life, but instead they only tell us what we can expect to get if we repeat the tests the filter manufacturer made – which do not reflect real-life applications. I explained what the major problems were in an earlier article in this series, in Issue 99 of the EMC Journal:

- Input and output impedances that differ from the 50 Ohms used for testing (see 7.3.8 in [84])
- RF References that are not perfect shields over the tested frequency range (see 7.3.10 and 7.3.12 in [84]),
- RF Bonds that are not vanishingly small impedances over the tested frequency range (see 7.3.10 and 7.3.12 in [84]).

To put some rough and ready numbers to the last two points, assuming 50 Ohm input and output impedances, the combined impedance of the RF Reference and the filter’s RF-bonding to it must be below 5 milliOhms for any filters achieving 80dB attenuation.

When the source or load impedance of a filter is less than 50 Ohms (as it often is), achieving 80dB attenuation (say) needs proportionally less RF-bonding and Reference plane impedance than 5 milliOhms.

To achieve 5 milliOhms in real life at, say, 10MHz, we need to be aware that at 10MHz an inductance of only 80pH is sufficient to create an impedance of 5 milliHenries, and 80pH corresponds to a connection length of about 80 microns. For 80dB attenuation at 100MHz, the connection length that creates 80pH is only 8 microns.

Clearly, no practical length of wire or braid strap, or even fixing screw, can reliably achieve such short connection lengths, and what is needed for RF-bonding at such high levels of filter performance is multiple direct connections between the metal surfaces of the body of the filter and the RF Reference of the equipment, each having a surface finish that ensures high conductivity over the life of the equipment despite the various kinds of corrosion that can occur.

Figure 7.11-1 shows that the attenuation of a filter is compromised by the stray mutual inductance between its input and output current loops – one of the many reasons why simple calculations and simple SPICE™ simulations are always wrong, and predict much better filter attenuation than will occur in real life with the same component values.

Adding a shield between the input and output current loops, as shown in Figure 7.11-2, reduces the magnetic field coupling between them, and so improves the filter’s performance in real life.

There is also stray capacitance between the input and output components and conductors, not shown in Figure 7.11-2, which is also reduced by the shield at the location shown in that figure.

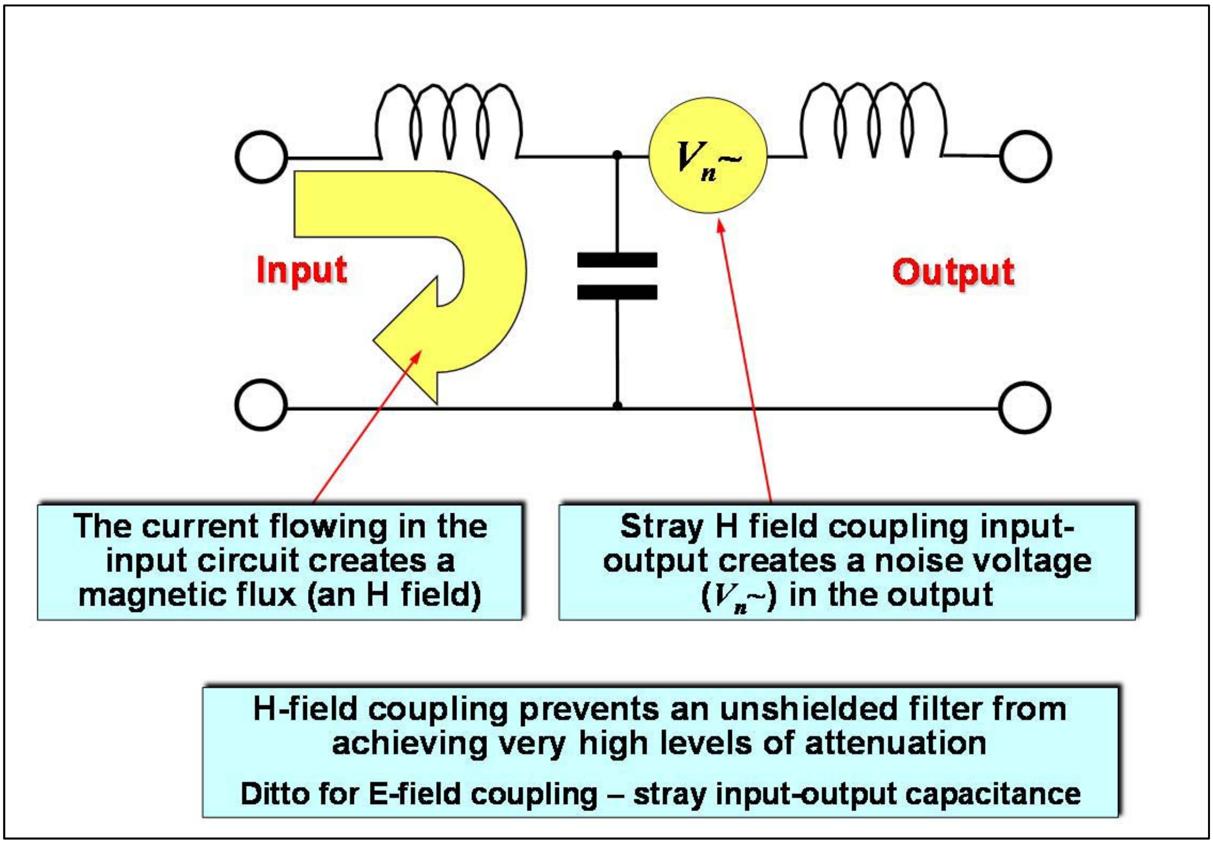


Figure 7.11-1 Stray magnetic field coupling degrades real-life filter performance

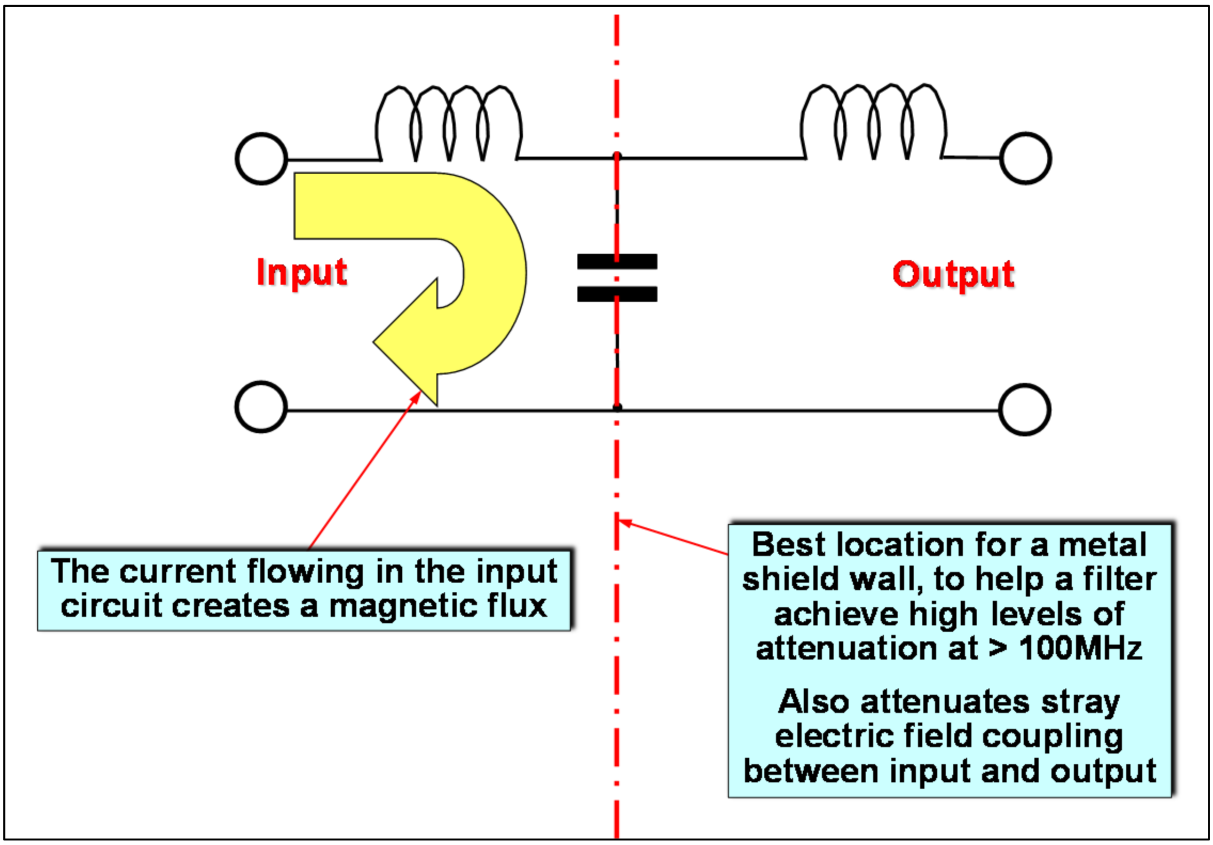


Figure 7.11-2 A shield reduces magnetic (and electric) field stray coupling

High levels of filtering attenuation, especially at higher frequencies will not be achieved without a shield between everything associated with a filter's input, and everything associated with its output. Higher levels of filter attenuation and/or higher frequencies, requires higher levels of shielding effectiveness at the frequencies being filtered.

The key component here is the feedthrough filter, also called (especially in larger sizes) the through-bulkhead filter. These obtain their shielding from the metal walls they are RF-bonded to as they passes through them.

Figures 7.11-3 and 7.11-4 show a small variety of the many feedthrough filters available. Because the military often have to deal with very harsh EM environments, they often have to use such high-performance filters, which is why many of the commercially-available feedthrough filters have military specifications.

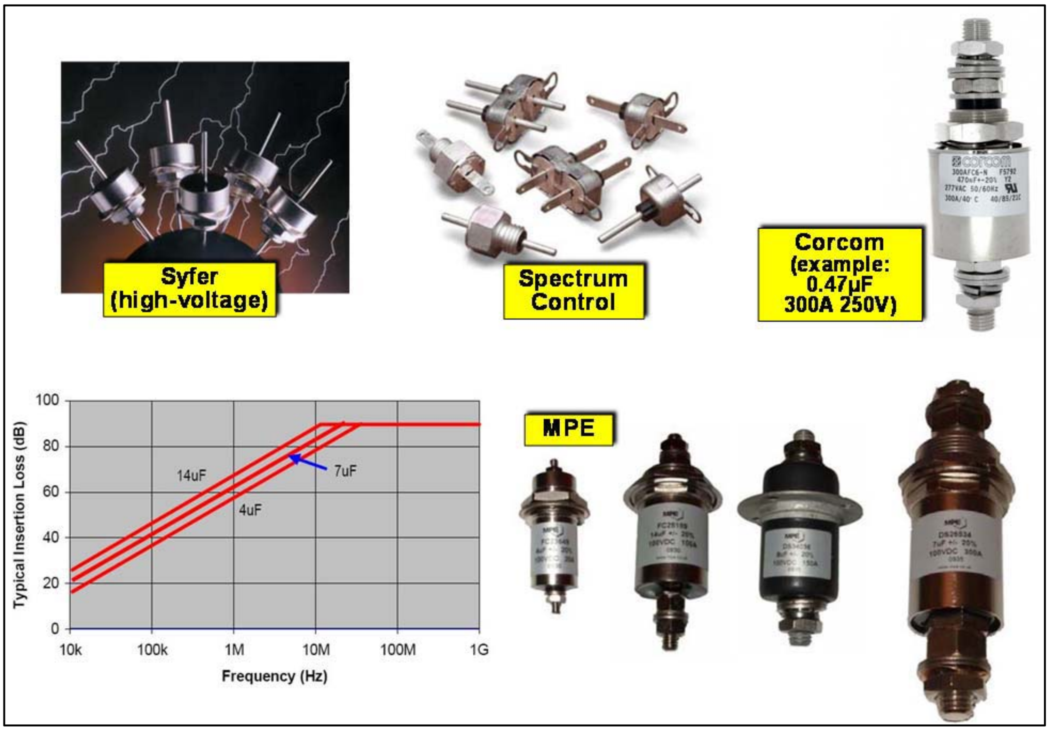


Figure 7.11-3 Some examples of feedthrough filters

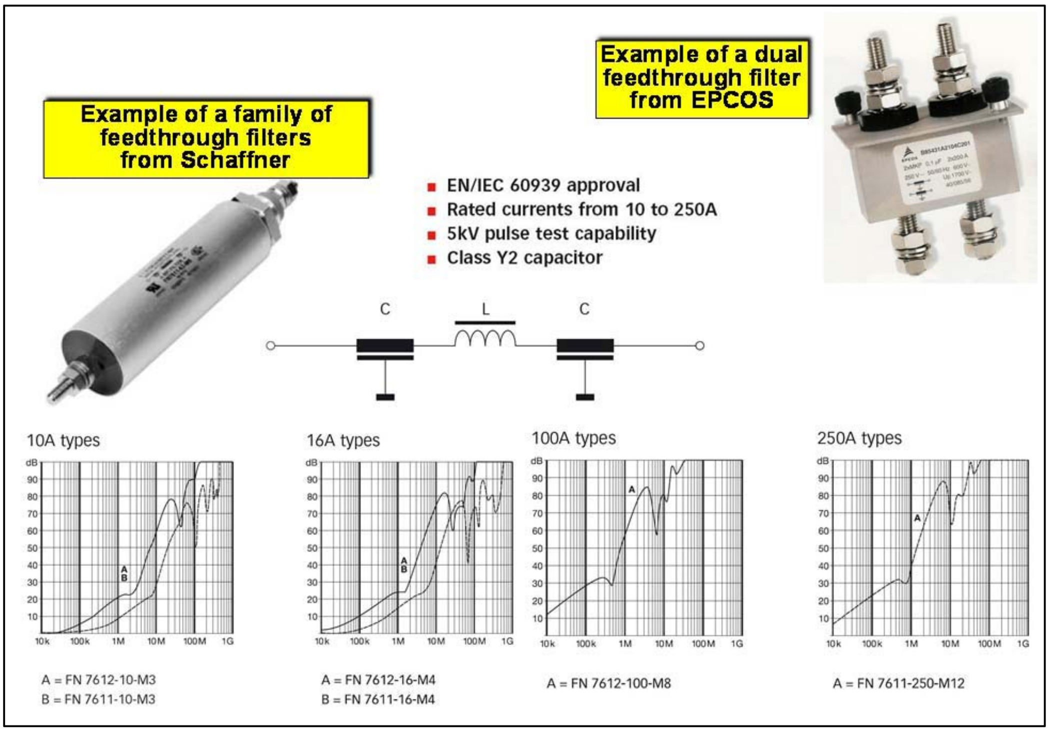


Figure 7.11-4 Some more examples of feedthrough filters

Shielded connectors are also available with feedthrough filter pins, as shown in Figure 7.11-5. Figure 7.11-6 shows the internal construction of the discoidal ceramic capacitors often used in such connectors, while 7.11-7 shows a shielded connector with high-performance feedthrough π-filter pins using two planar capacitor arrays, either side of ferrite beads.

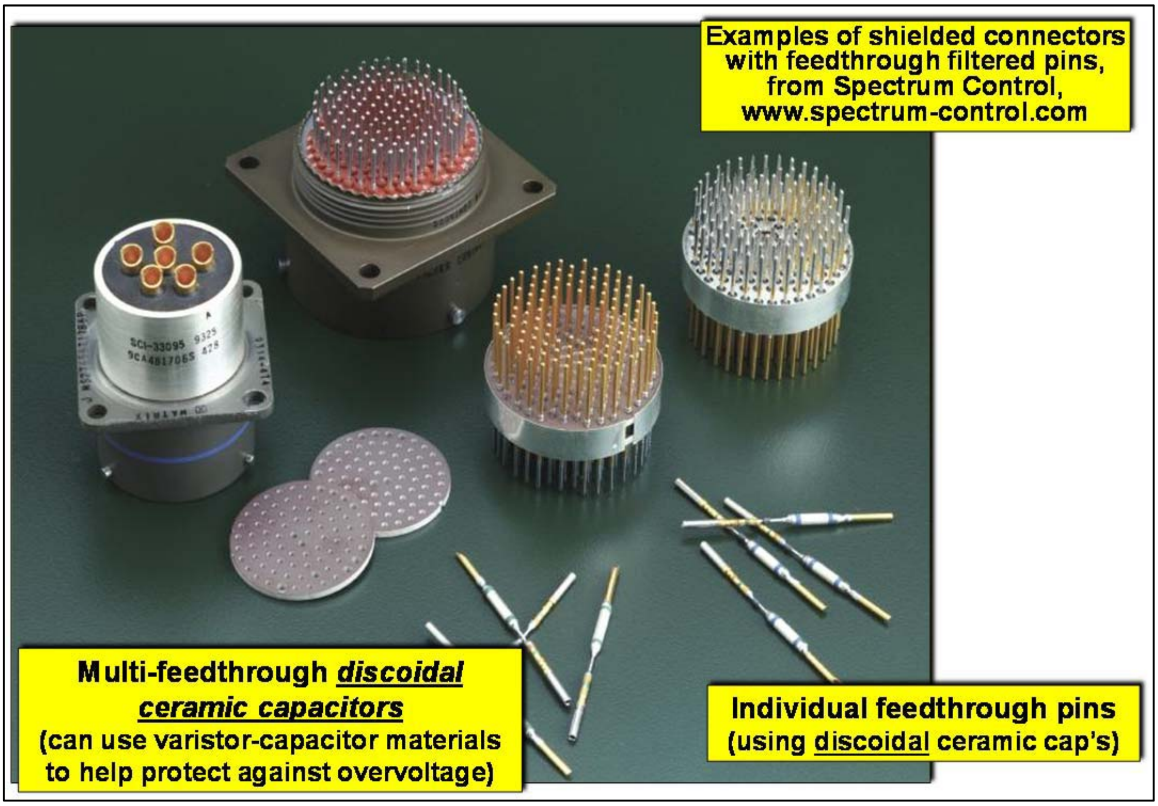


Figure 7.11-5 Some examples of connectors with feedthrough filters

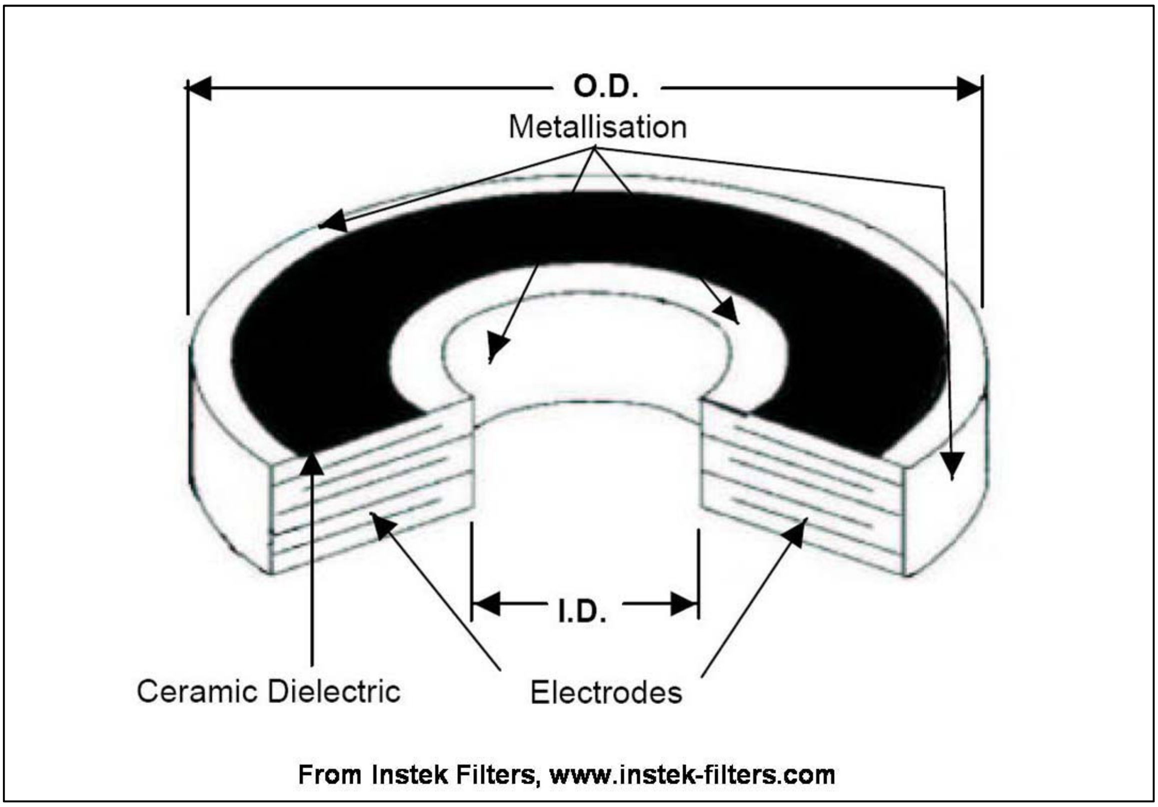


Figure 7.11-6 The construction of ceramic discoidal capacitors

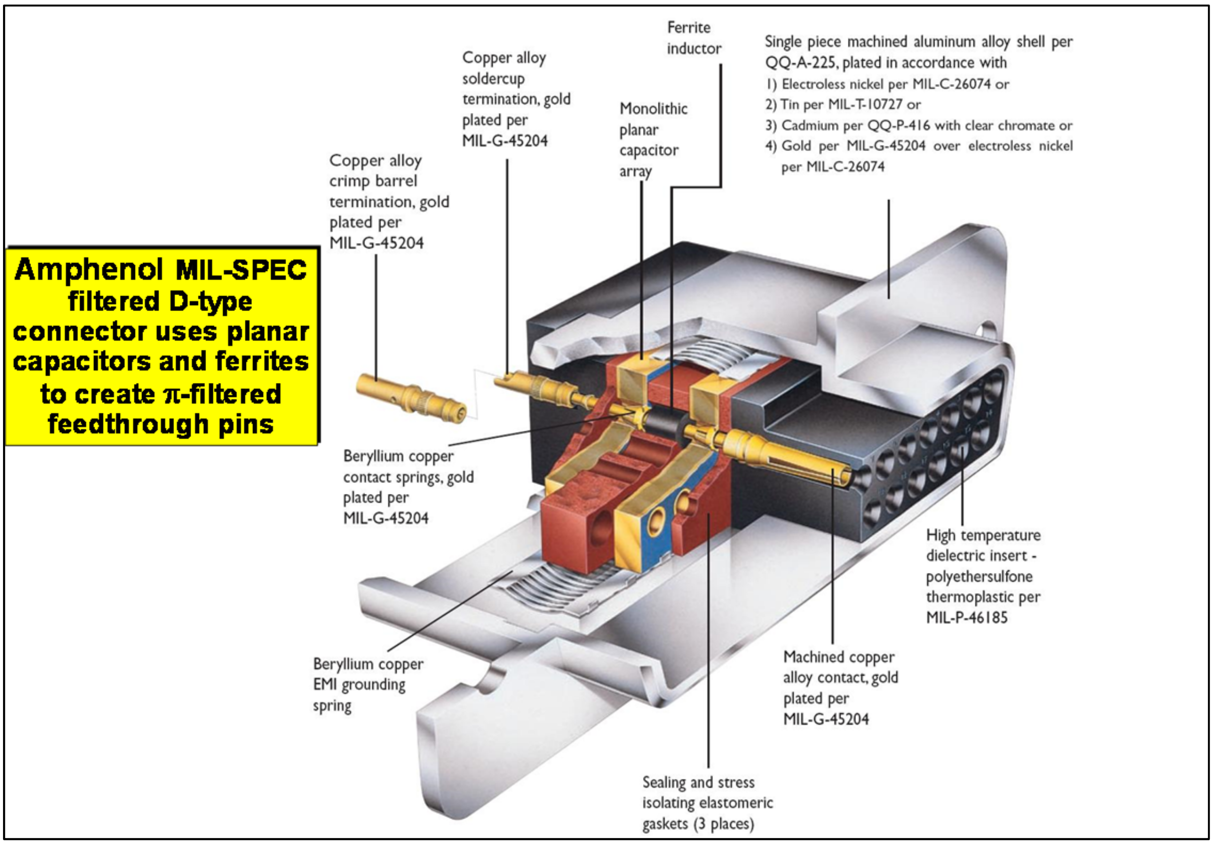


Figure 7.11-7 An example of a connector with feedthrough filters using planar capacitors

Figure 7.11-8 shows an example of so-called "Room Filters" mounted on the wall of an EMC shielded test chamber, although of course they can be used anywhere high-performance filtering is required.

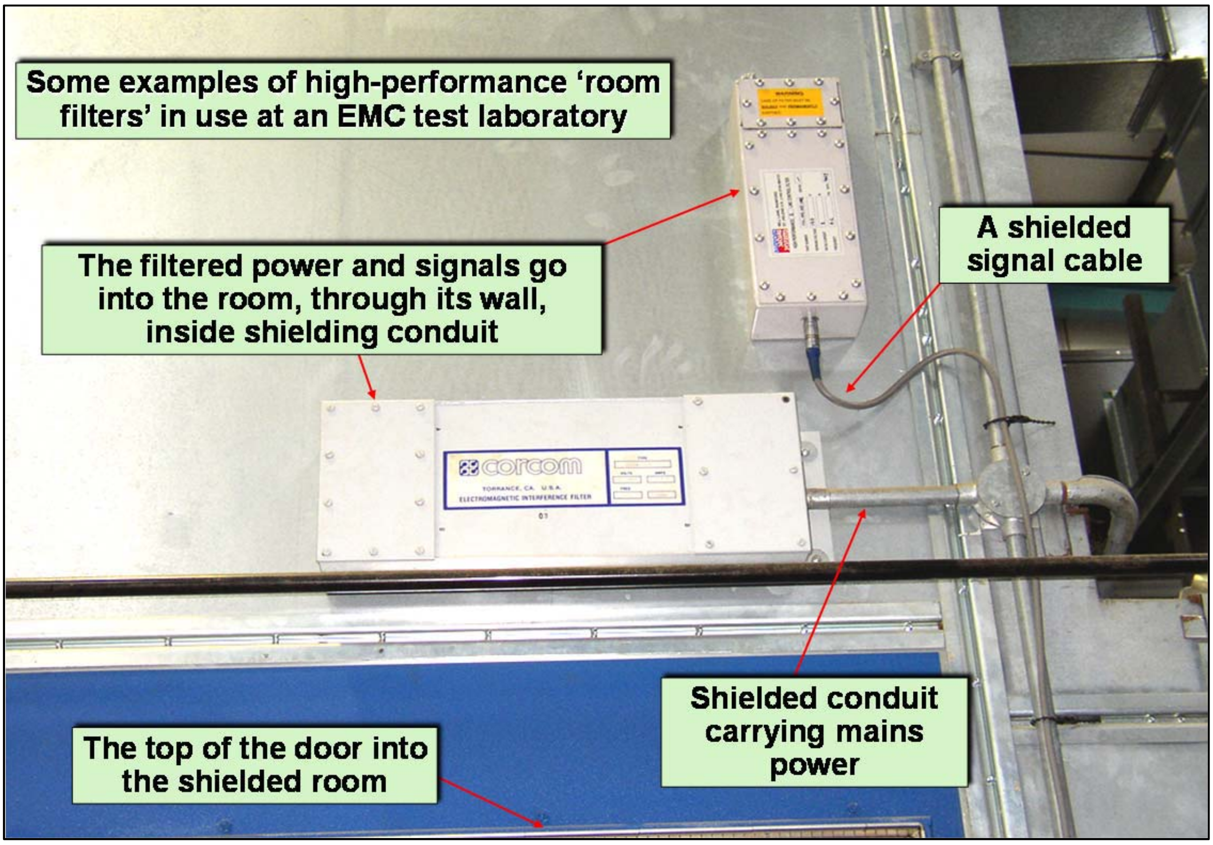


Figure 7.11-8 Examples of room filters in use

These are actually high-performance feedthrough filters that rely on a shielding barrier, although all of the filter remains on one side of the shield and the cables that connect to the other side are shielded – with their cable shield connected to the shield barrier. Figure 7.11-9 shows the general principle of mounting a room filter, using shielded conduits in this example, instead of shielded flexible cables.

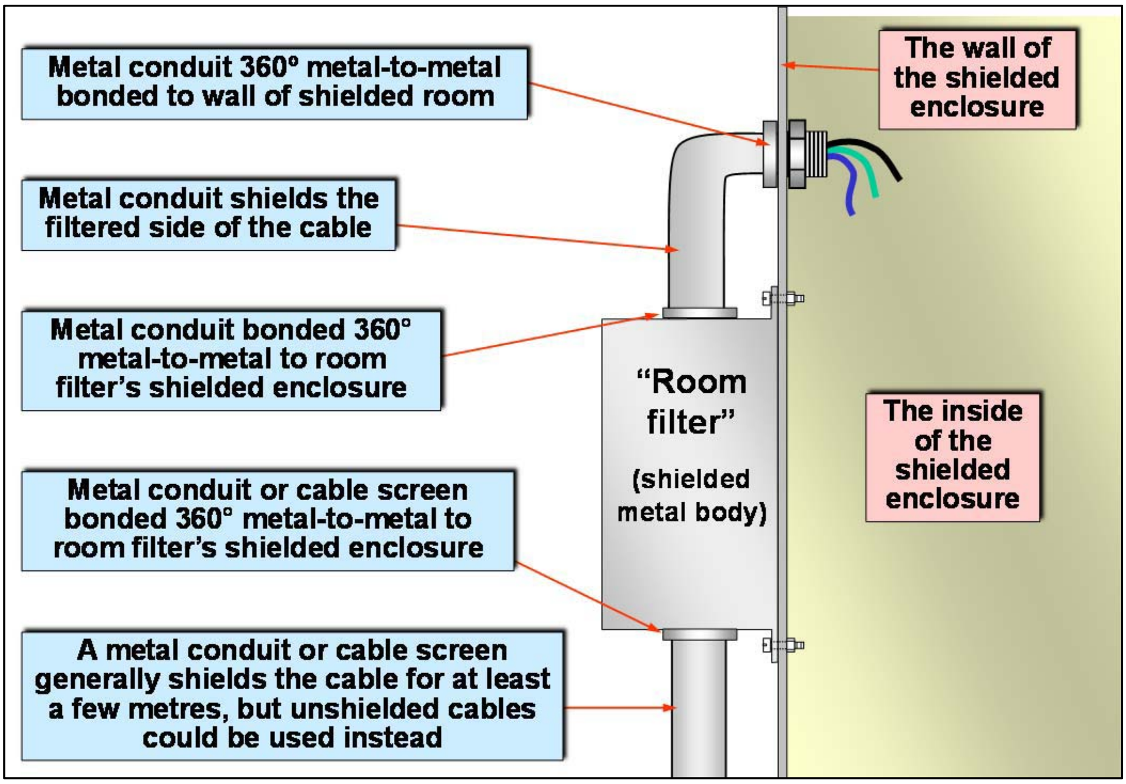


Figure 7.11-9 General principles of room filter installation

Figure 7.11-10 shows how to convert a low-cost chassis-mounted filter into a room filter, by enclosing it in a shielded box.

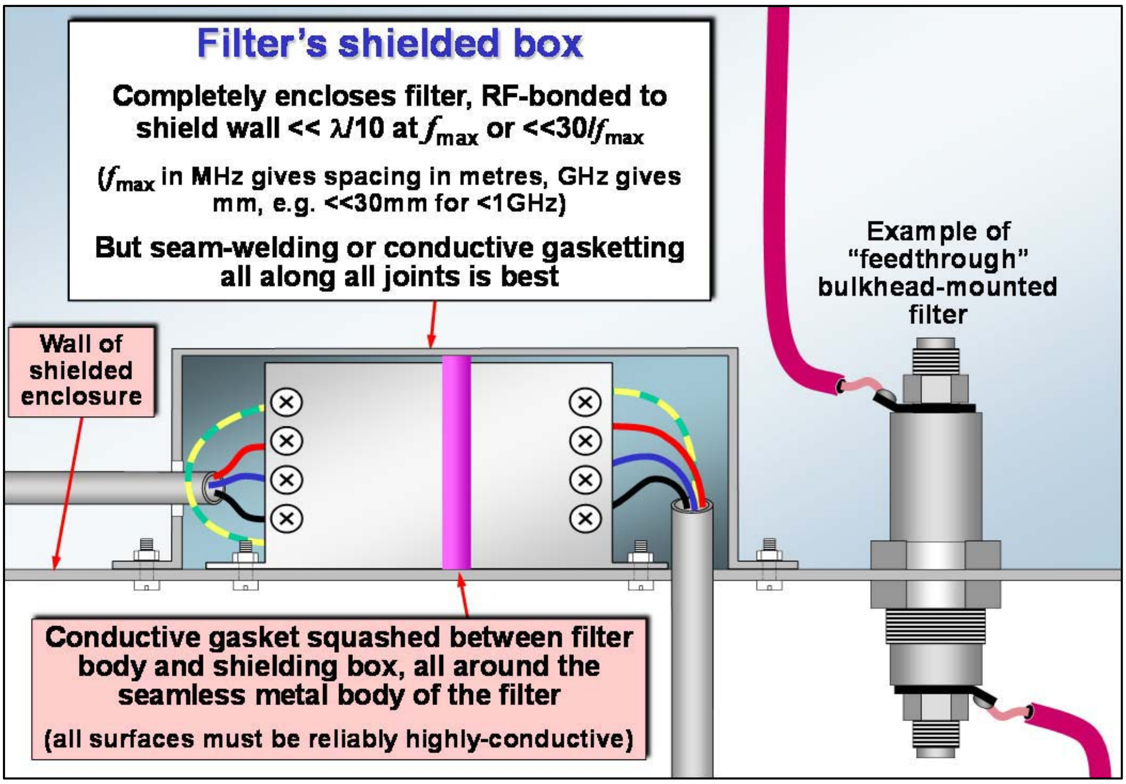


Figure 7.11-10 A D-I-Y Room Filter using a low-cost chassis-mounted filter

Figure 7.11-11 shows a clever filter design from MPE, which is supplied in a shielded box with a length of shielded cable on its input or output. It is essentially a room filter type of construction, with its cable shield RF-bonded to the shield wall between the filter's input and output circuits.

The shielded cable effectively extends the shield-barrier wall as far as is required, so that the filter can be positioned where it creates the best result (usually very close to an electrical/electronic unit that emits a lot of conducted noise).

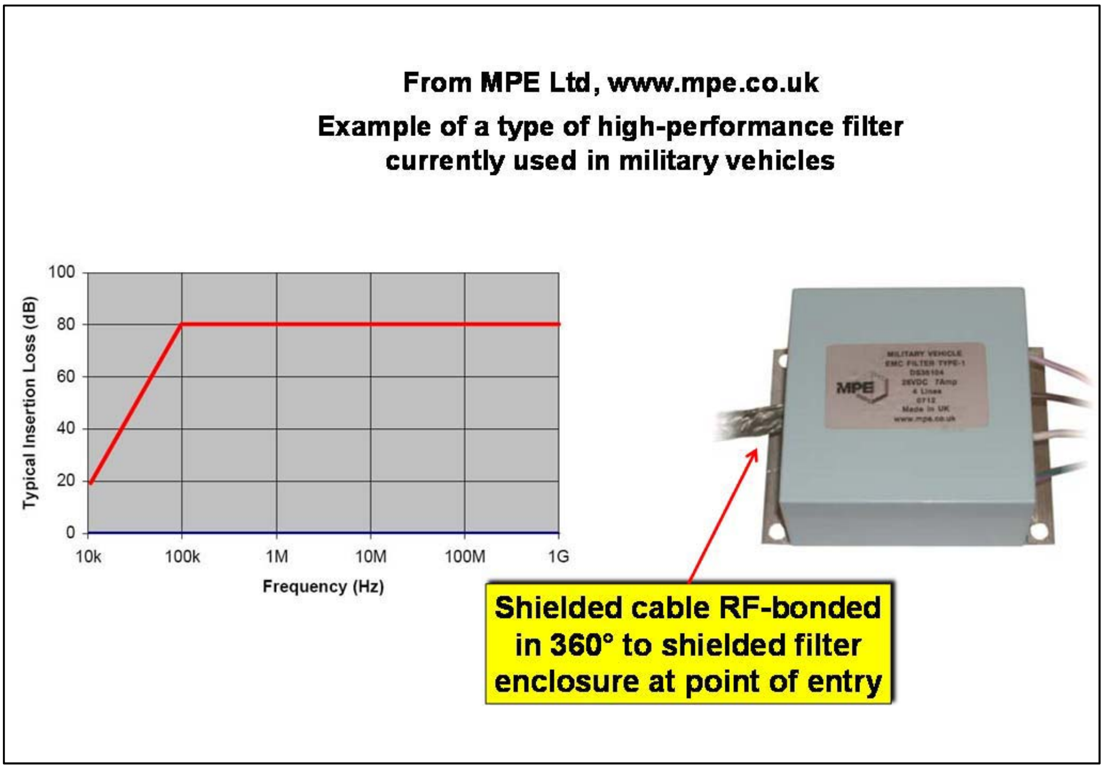


Figure 7.11-11 Mount the filter outside of the shielded enclosure, using shielded cables

Finally, Figure 7.11-12 repeats a figure used earlier in this series of articles, as a reminder that no conductors – whether they are used for electrical, electronic, mechanical, pneumatic, hydraulic, or whatever purposes, may enter or exit a shielded volume, without either being:

- RF-bonded directly to the shield wall at the point of entry/exit
- Feedthrough-filtered at the shield wall at the point of entry/exit
- Shielded, with their shield RF-bonded directly to the shield wall at the point of entry/exit.

As Figures 7.11-9, 7.11-10 and 7.11-11 show, the "shield wall at the point of entry" can be extended away from the shielded enclosure, by using cable shielding techniques. How to correctly bond cable shields to shielded enclosures is covered in Chapter 4.6 of [5].

People often assume that cables carrying analogue sensor or audio signals, or which carry low-rate data (e.g. for computer mice), or metal pipes carrying air, gas, oil or water, can pass through holes in shielded enclosures without compromising its shielding. They find out this is not good EMC practice when they fail EMC tests or suffer EMI in their real-life applications.

All conductors used for any purposes, whether electrical or not, are actually "accidental RF antennas" that operate happily up to many tens of GHz (see Chapters 3.3 – 4.5 in [4], repeated as Chapters 2.6.3 – 2.6.5 in [5]. So, if any conductors pass through the wall of a shielded enclosure without being RF-bonded by one of the three methods listed above, they will very significantly degrade its shielding effectiveness.

This issue is important in the context of the high-performance filtering that is the subject of this section, because high-performance filters always rely on shielding, so anything that degrades the effectiveness of their shielding also degrades the performance of their filtering.

Modern feedthrough filtering on PCBs uses surface-mounted three-terminal filter components and surface-mounted board-mounted metal cans, often all assembled and automatically reflow-soldered at the same time as the rest of the board's components.

The filter components cannot be assembled in the cans' walls, because this would require an additional process step (probably manual assembly) and add too much cost. Figure 7.11-13 gives an outline of what is required to make such board-mounted filters work correctly, and Figure 7.11-14 shows the sort of improvement that can be achieved by correctly combining filtering and shielding at board level.

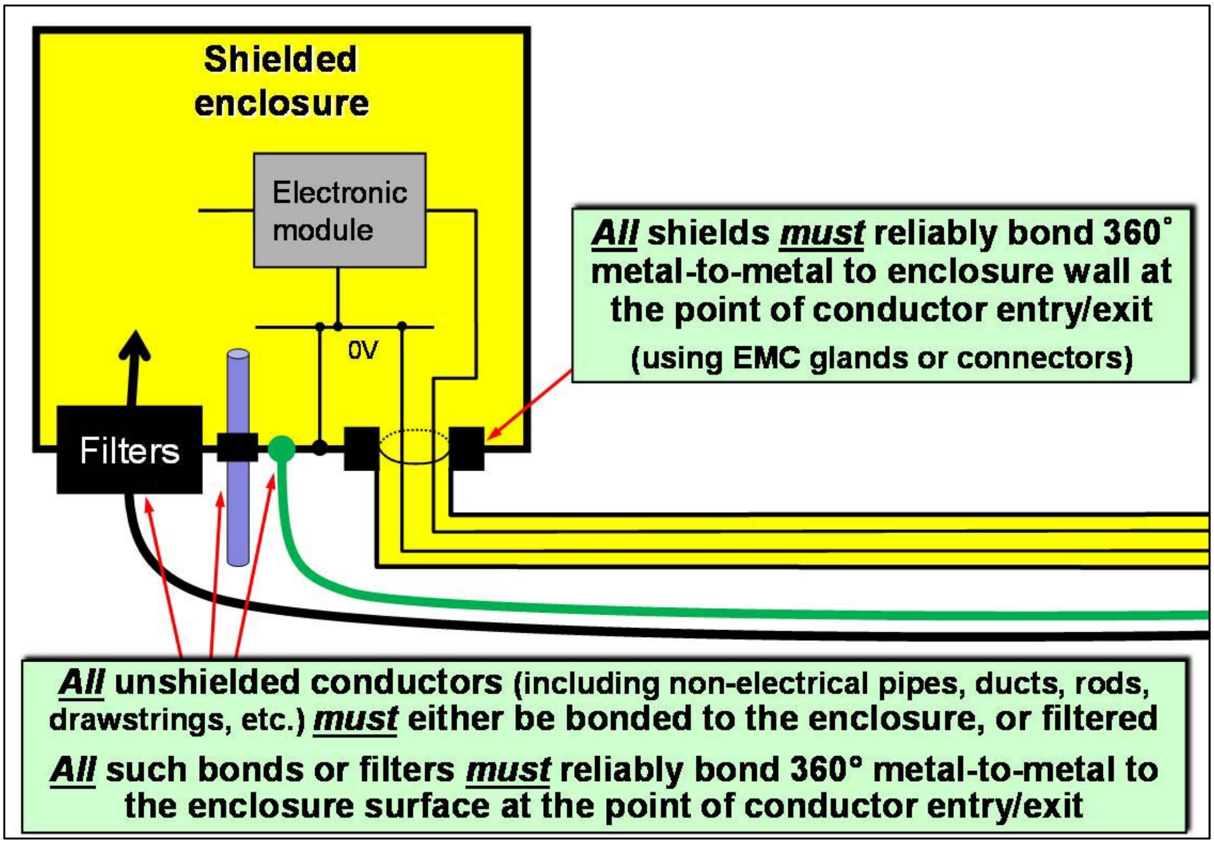


Figure 7.11-12 RF-bonding rules for all conductors entering/exiting a shielded enclosure

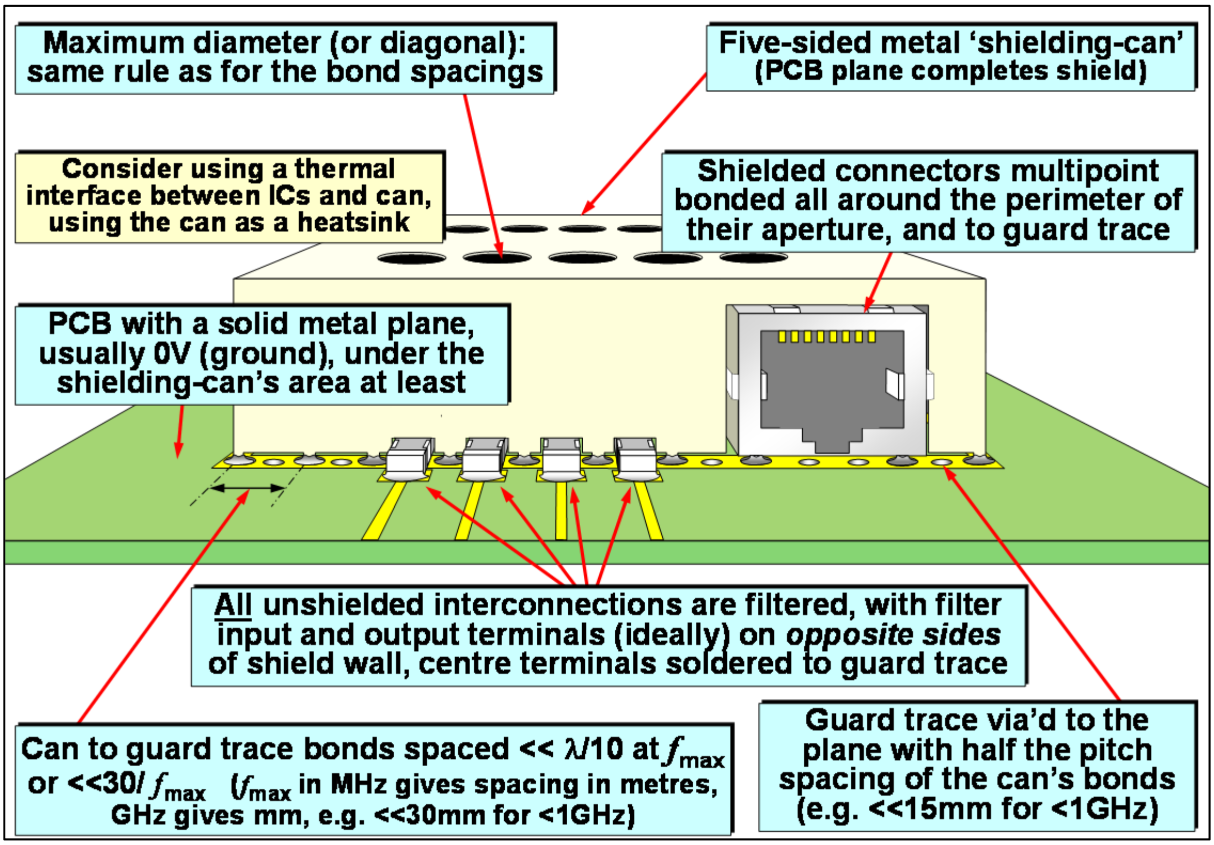


Figure 7.11-13 Overview of good practices when shielding surface-mounted PCB filters

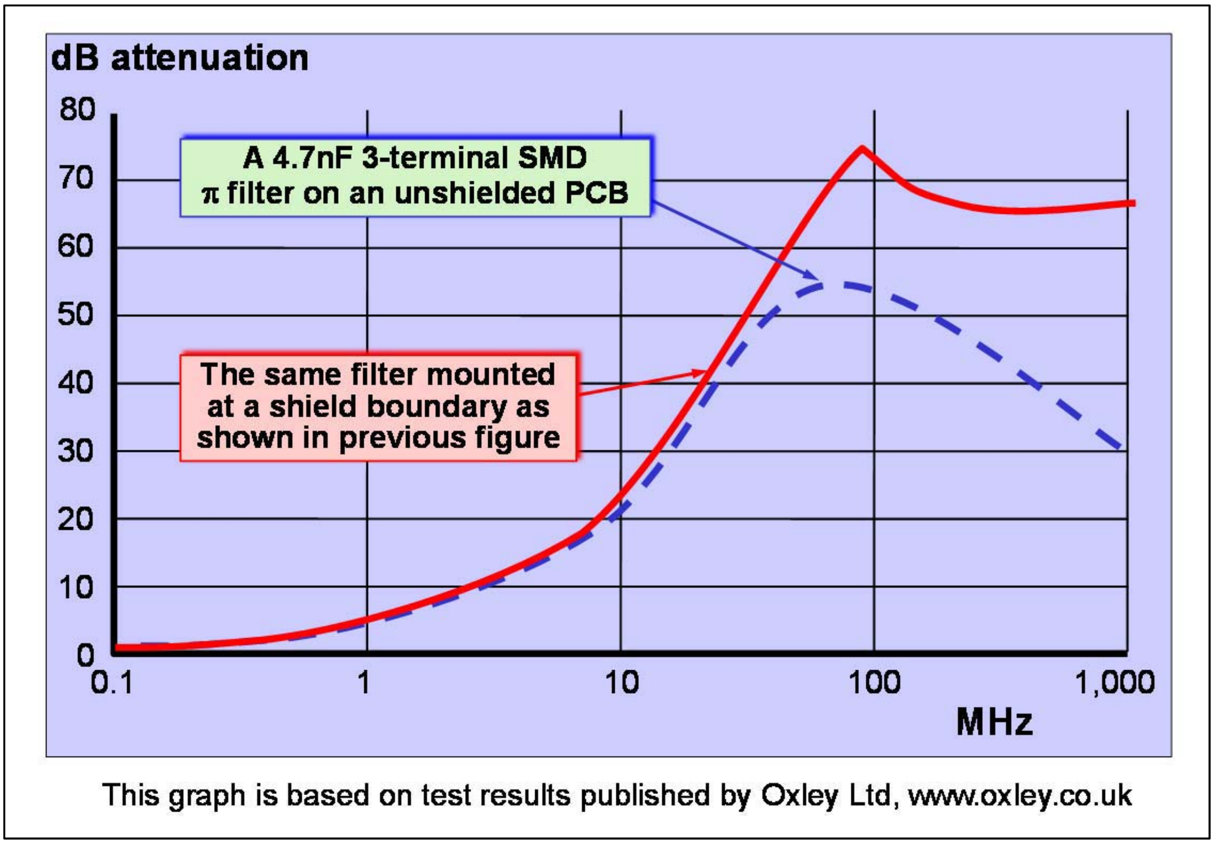


Figure 7.11-14 PCB filters benefit hugely from PCB shielding above 100MHz

All digital integrated circuits (ICs) now use silicon processes that result in switching edge rates that generate GHz noise, and there is no end to this upward trend in emissions frequencies.

The upper frequencies of radiated emissions and immunity tests are now increasing from 1GHz to at least 2.7GHz (sometimes 6GHz, or more) for almost all types of equipment, and there is no end to this upward trend in test frequencies.

Figure 7.11-14 shows that the typical surface-mounted filter component has an attenuation that gets worse as frequency increases above about 30MHz, so to achieve good filtering at GHz frequencies they must be combined with board-level shielding. For this reason, I now recommend every electronic designer using board-mounted filtering to make provision for shielding cans for every filter.

References (for this article only)

[4] "The Physical Basis of EMC", Keith Armstrong, Armstrong/Nutwood UK October 2010, ISBN: 978-0-9555118-3-7, from www.emcacademy.org/books.asp. The same material is presented as Chapter 2 in [5].

[5] "EMC Design Techniques for Electronic Engineers", Keith Armstrong, Armstrong/Nutwood UK Nov. 2010, ISBN: 978-0-9555118-4-4, from www.emcacademy.org/books.asp.

[14] Keith Armstrong's earlier articles are available from the archives at www.theemcjournal.com

[15] Textbooks by Keith Armstrong and others are available from www.emcacademy.org/books.asp

[37] "EMC for Printed Circuit Boards – Basic and advanced design and layout techniques", Keith Armstrong, Armstrong/Nutwood UK 2010, ISBN: 978-0-9555118-5-1, from www.emcacademy.org/books.asp (note, this is the same text as the 2007 edition, ISBN: 978-0-9555118-0-6 or 978-0-9555118-1-3, repackaged in a different style of book).

[72] "EMC Design of High-Frequency Power 'Switchers' and 'Choppers' – Suppressing RF emissions from converter inputs and outputs", Keith Armstrong, The EMC Journal, Issue 98, January 2012, pages 27-38, available from the archives at www.theemcjournal.com

[84] "EMC Design of High-Frequency Power 'Switchers' and 'Choppers' – Suppressing RF emissions from converter inputs and outputs", Keith Armstrong, The EMC Journal, Issue 99, March 2012, pages 30-42, available from the archives at www.theemcjournal.com

[85] “Fundamentals of EMC Design: Our Products Are Trying To Help Us”, Keith Armstrong, Interference Technology Magazine, 04/03/2012, www.interferencetechnology.com/fundamentals-of-emc-design-our-products-are-trying-to-help-us-3

[104] “EMC Design of High-Frequency Power ‘Switchers’ and ‘Choppers’ – Important wiring and PCB layout issues for suppression devices”, Keith Armstrong, The EMC Journal, Issue 102, September 2012, pages 38-43, available from the archives at www.theemcjournal.com

[105] “Modeling and Simulation of Powertrains for Electric and Hybrid Vehicles”, Marco Klinger, Workshop FR-AM-4-1, IEEE 2009 Int’l EMC Symposium, Austin, TX, Aug 17-21, 2009, CD-ROM ISBN: 978-1-4244-4285-0

[106] “Successful PCB grounding with mixed-signal chips”, by Mark Fortunato of Maxim Integrated Products, EDN Analogue Network.

- Part 1: Principles of current flow, August 28, 2012: www.edn.com/design/analog/4394761/Successful-PCB-grounding-with-mixed-signal-chips---Part-1--Principles-of-current-flow?cid=Newsletter+-+EDN+on+Analog
- Part 2: Design to minimize signal-path crosstalk, September 10, 2012: www.edn.com/design/analog/4395817/Successful-PCB-grounding-with-mixed-signal-chips---Part-2--Design-to-minimize-signal-path-crosstalk?cid=Newsletter+-+EDN+on+Analog
- Part 3: Power currents and multiple mixed-signal ICs, September 17, 2012: www.edn.com/design/analog/4396382/Successful-PCB-grounding-with-mixed-signal-chips---Part-3--Power-currents-and-multiple-mixed-signal-ICs?cid=Newsletter+-+EDN+on+Analog

[107] “Design Tips – High Speed Traces Close to Edge of Reference Plane”, Bruce Archambeault, PhD, IEEE EMC Society Newsletter, Issue 220, Winter 2009 (i.e. early 2009) pp 69-70, http://ewh.ieee.org/soc/emcs/acstrial/newsletters/winter09/index.html