Another EMC resource
from EMC Standards

EMC design of Switching Power Converters Part 7 (continued 5)

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 4) — Important wiring and PCB layout issues for suppression devices

First published in The EMC Journal, Issue 102, September 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 101 [99]. 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.

Please remember what I repeatedly said in earlier articles – it is much better to use a more “EMC-benign” switcher topology (e.g. resonant mode) than it is to use an ordinary one and then try to suppress its horrible noise emissions.

7.7 Improving capacitor suppression with wiring and PCB layout

As made clear in the previous issue of the EMC Journal [99], the filtering achieved by DC capacitors connected in shunt across a power rail depends on the impedance they achieve. Good attenuation (suppression) of RF noise requires the achievement of low impedance, and in [99] we discussed issues relating to the filter capacitors themselves.

But the way in which we electrically connect to the capacitors, using wires, cables, busbars and/or PCBs is also very important indeed for achieving low impedance and getting good filtering. Although the focus of this article is on DC filter capacitors, the same issues arise for filter capacitors on AC power supply lines, and any DC or AC filter capacitors used in shunt across the switch-mode (or “chopped”) AC or DC outputs from the converter/chopper.

Way back in the mid-1980s I was leading a project to create a new bench-top microwave network analyser for a well-known instrumentation company based in the UK. We wanted to use the internal power distribution technique that Tektronix had recently used in their 70,000 Series oscilloscopes and made the design details public (we were not copying their intellectual property without permission).

If my memory is correct, Tektronix were routing their power between modules as a 25Vrms sinewave at about 25kHz, transforming it in each module to provide the voltage rails they needed. These days we would route around 12Vdc and use DC/DC converters in each module, but that technology was not very well developed in the mid-80s and distributing AC with local voltage transformation was low-cost and thermally efficient.

The technical challenge was the switch-mode AC-AC inverter that took the 50 or 60Hz mains supply and generated 300W or so of 25kHz sinewave for distribution to the various modules. While we were developing our version of this, we had a very large DC-Link capacitor charged up by the AC rectifier to around 340Vdc, connected to the PowerFET chopper circuit by 150mm long 6mm copper-diameter round wires.

When first testing the first "breadboard" prototype we got all sorts of weird and wonderful waveforms that didn't seem to correspond with what we expected (and yes, we were using differential 'scope probes that were galvanically isolated and rated for safe operation up to 500Vdc – we never, ever "float" our 'scopes by removing the safety earth lead from the mains plug, because it creates lethal risks for electric shock and fire).

Trying to figure out why the chopper voltages were so weird, and after we had all run out of ideas to test, I put the differential probes across one of the 150mm long 6mm diameter copper wires from the DC-Link capacitor to the chopper FETs – and measured 150V peak-to-peak along it!

150V peak-to-peak being dropped along just 150mm of a 6mm diameter copper wire seemed impossible to us at that time. We had assumed that using such an over-the-top wire (our power inverter was intended to output about 300W, so the average current from the DC-Link would only be around 1A) would be an effective short-circuit, but of course we then realised that we had not considered its 100nH or so of series inductance and the fact that our peak-to-peak currents contained some very fast edges. (A good reason for using resonant mode converters, see section 2.2. in [42].)

So for good noise suppression we must minimise the additional series resistance and inductance added to the filter capacitor by the connections, which degrades the attenuation it achieves in practice, by using one or more of the following techniques:

a) As far as possible "star connecting" the power conductors directly to/from the terminals of the filter capacitor(s) to minimise the series R and L they have in common.

b) Using conductors with large cross-sectional areas (CSAs), to reduce their resistance at low frequencies.

   For example by using busbars with large CSAs; using large-diameter round wires; by plating wide PCB copper traces up to "8 ounce" thickness (some PCB manufacturers can do 12 or more ounces); laminating copper or brass plates as power planes (and heat spreaders) in PCBs, etc.

c) Using conductors with a large surface area, to help achieve low resistance and low series inductance at high and radio frequencies.

   For example by using rectangular rather than round busbars; using tape instead of round wire; using wide PCB traces instead of narrow, etc.

d) Using copper rather than aluminium conductors, for its lower resistivity hence lower resistance for a given CSA.

e) Routing opposing power currents (i.e. "send and return") very close together, using their mutual inductance to cancel out most of their series inductance.

   For example by using laminated wide flat busbars or PCB traces/planes on adjacent layers in the stack-up (thinner interposed dielectric layers, interleaved send/return/send/return/ etc. busbars, traces or planes, both improve performance); twisting send/return wires together (twisted "star-quad" is better than twisted pair).

f) Using parallel copper traces on adjacent layers, or laminated flat busbars, to provide a distributed capacitance in parallel with the filter capacitor.

g) Reduce the areas of all current loops, to reduce their series inductances and "Helmholtz Coil" coupling (see below).

So let's see how we might apply these techniques to our example SMPSU circuit, first introduced by Figures 3B, 3C and 3D in [64]. This is an isolating AC/DC converter using a half-bridge switching topology, because I had to draw something and the actual topology is not important for this discussion. Figure 7.7-1 shows its original single-layer (red traces) PCB layout, and is a copy of Figure 3C in [64].

Note that the DC-Link and DC output filter capacitors are connected to the rest of the circuit by short lengths of trace at each of their terminals, which could easily add series impedances greater than those of the capacitors themselves and so degrade their noise suppression by 6dB or more.

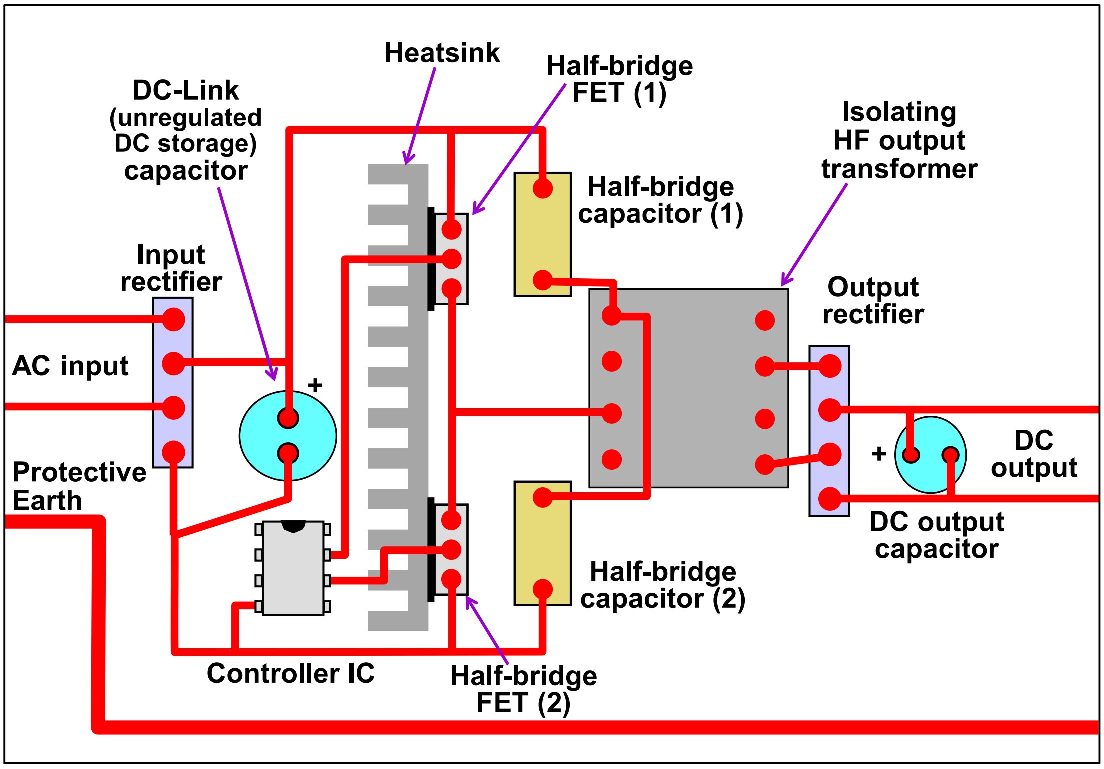

Figure 7.7-1 The original 1-layer PCB layout for the example converter

Figure 7.7-2 shows a slightly changed layout, with the input and output traces for both the DC-Link and DC output capacitors "star connected" at the capacitors themselves. This is an application of method a) in the above list.

Neither capacitor has any additional impedance added in series with the capacitor by traces (as there was before) so the filter attenuation is set entirely by the capacitor's impedance as a fraction of the source impedance of the noise. (Or so we shall assume for now! This is not the end of the story, as we will see in section 7.9.)

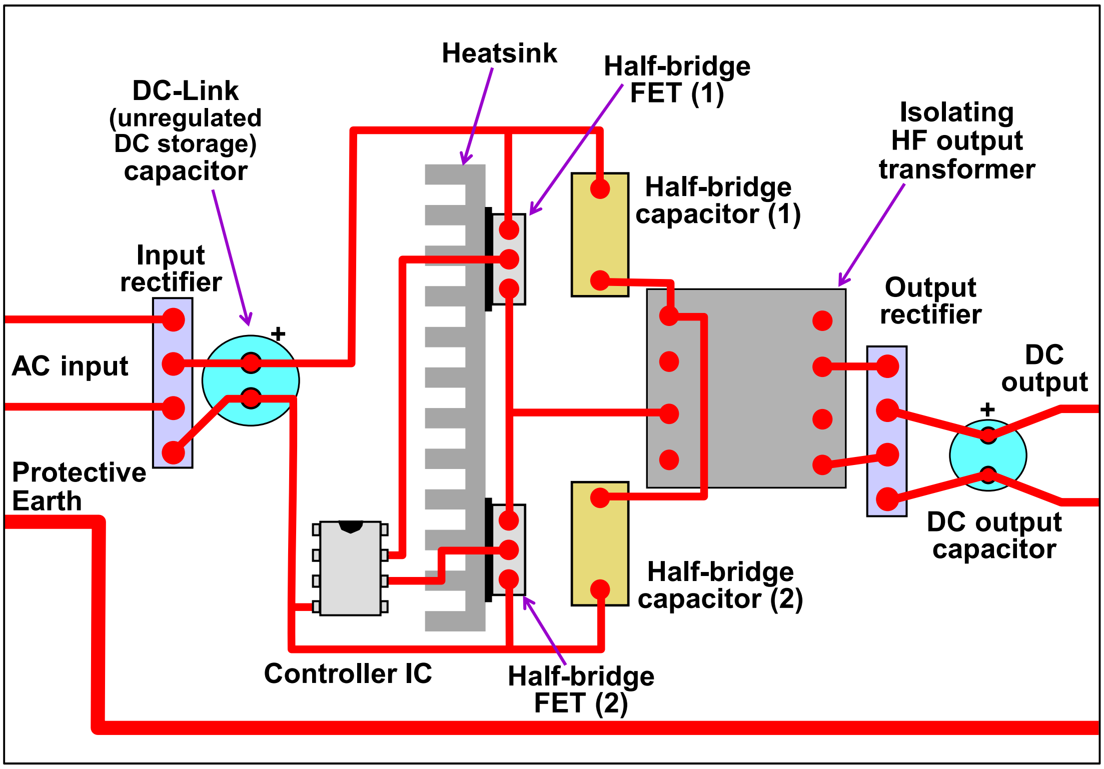

Figure 7.7-2 Modified layout that avoids adding impedance in series with shunt capacitors

Unfortunately, the simple solution of Figure 7.7-2 is not always practical, especially when several capacitors must be connected in parallel. Then we have to use some or all of the other techniques b) – g) listed above.

Using wires or busbars tends to result in complex assemblies that cost a lot to assemble, and using PCBs limits the cross-sectional area of the copper according to the greatest thickness that the manufacturer can plate-up the traces (usually no more than what is known as "8 ounce copper").

Copper or brass metal plates can be laminated in PCBs and be used as power planes (as well as heatsinks and/or heat spreaders, see [35] and/or Chapter 13 of [5]), and Figure 7.7-3 shows some examples of interconnection technologies that have been developed to provide the lowest practical degradation of capacitive suppression whilst minimising assembly time – by using a wide variety of laminated flat send/return busbars.

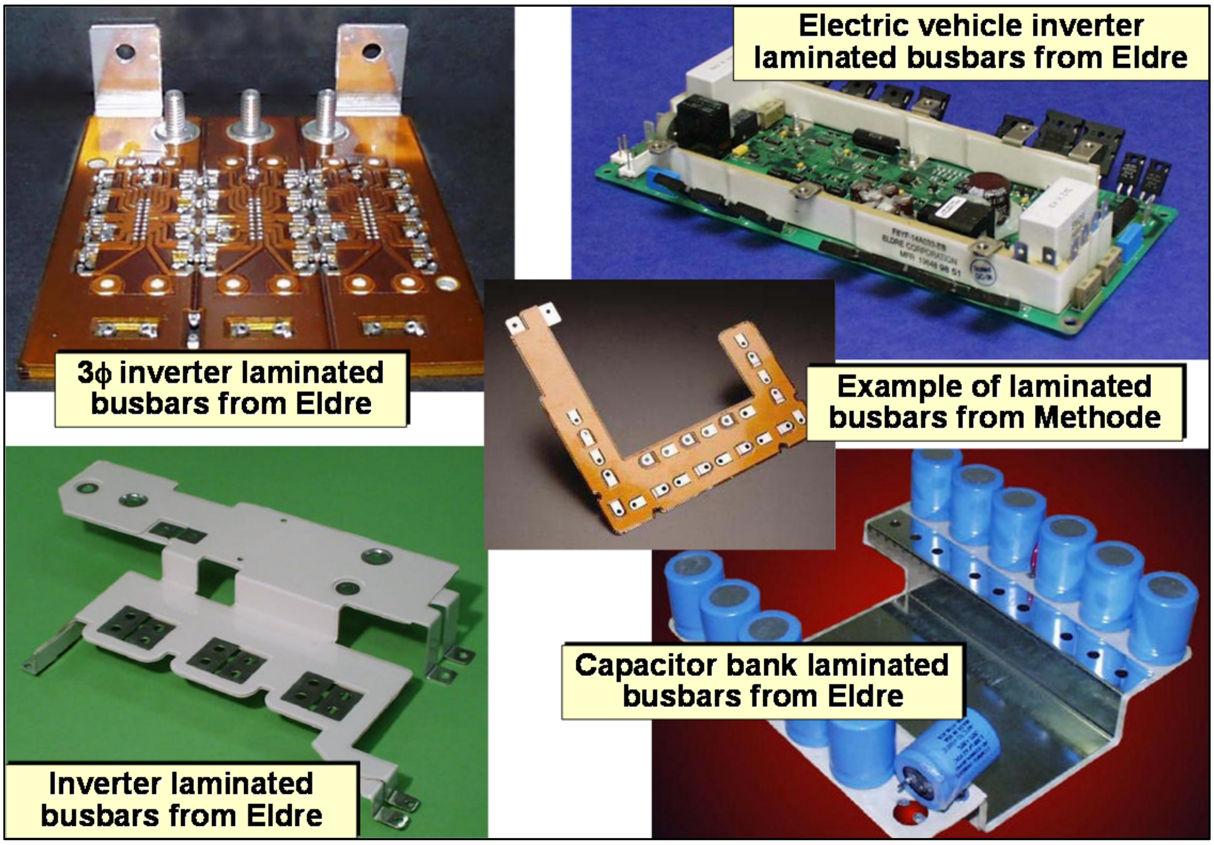

Figure 7.7-3 Some examples of laminated busbars used in power converters

These laminated busbars do not have to lie flat in one plane, like PCB traces do, they can (generally) be shaped and folded to create three-dimensional shapes to help provide mechanical support and fit within the overall enclosure.

The electric vehicle motor drive inverter example in the top right-hand corner shows laminated busbars soldered onto the PCB like any other component.

7.8 Suppressing the DC-Link conductors

Previous articles in this series have dealt with suppressing the noise in the power input and power output conductors (see 7.1 through 7.6 in [72] [84] [92] and [99]) – but the DC-Link can cause high levels of emissions too and may need to be suppressed.

As shown in [72], a DC Link carries powerful DM and CM switching noise currents and voltages, so all of the methods b) – g) in the list in 7.7 above should be applied to the DC-Link conductors, where practical.

These methods have the effect of minimising the DC-Link conductors' 'accidental antenna' emissions, both in the near-field and in the far-field, see Chapter 4.3 of [4], or Chapter 2.6.3 of [5] (they use the same text and figures).

It is very important to make the DC-Link conductors very short indeed, ideally no more than a few tens of millimetres.

In the example power converter used in Figures 7.7-1 and 7.7-2, the DC-Link is just a few short traces on a PCB and is usually adequately suppressed by providing appropriate values and types of shunt capacitance, see section 7.6 in [99] and taking a little care with the layout (see later).

But where the DC-Link cannot be very short, for example when a single AC/DC rectifier provides a DC-Link that must be shared by two or more DC/DC or DC/AC switching power converters, we have to treat it as a very noisy conductor indeed – just like the PWM output from a chopper power converter.

All the filtering techniques discussed in sections 7.3.15 (in [84]) and/or the shielding techniques discussed in section 7.5 (in [92]), for suppressing the PWM outputs of choppers (whether DC or AC), should be applied to suppressing DC-Link cables.

I generally find that the lowest-cost but still effective solution is to combine filtering with shielding, so that the shortcomings of each technique when done in a low-cost manner, are compensated for by the other.

Filtering and/or shielding DC-Link conductors becomes especially problematic when the DC-Link has to be longer than one-tenth of the wavelength at the highest frequency of concern – because it is approaching its first resonant length of one-quarter of a wavelength at which its accidental antenna behaviour is a maximum.

For example, a 10m long cable has its first, lowest, resonant frequency at about 5MHz (assuming it is PVC insulated with a velocity factor around 0.7) so we should take special care to suppress any noise in it at frequencies above about 2MHz. A 3m long cable requires special care with suppression at frequencies above about 6MHz.

In a system or installation, we should always take care to route noisy cables such as DC-Links and PWM power converter outputs close to the designated RF Reference at all times, to further reduce its remaining accidental antenna emissions by providing a nearby path for its CM (also known as "antenna mode") noise currents. See [100] [101] and [102] for more details on these good EMC practices.

Where a long DC-Link consists of traces or planes in a PCB rather than conductors in a cable, they should be routed as send/return traces/planes on adjacent layers in the stack-up to reduce DM and CM emissions.

Also, because CM emissions are usually the main problem, we need to reduce them further by RF-bonding both the send and return traces or planes to the chassis or enclosure metalwork with spacings no more than one-tenth of the wavelength at the highest frequency of concern. See Chapter 7.4.4 of [5] or Chapter 3 of [37] for details of this very powerful CM suppression technique.

7.9 Reducing stray noise coupling

If a shunt suppression capacitor had perfectly zero impedance at all frequencies (which is of course not possible), and if the conductors connecting to it were star connected as shown in Figure 7.7-2 so that they added no series R or L to the capacitor at all, the previous discussions in this series of articles imply that the noise suppression achieved by the shunt capacitor would be infinite.

In fact, if we use SPICE™ or most other types of basic circuit simulator to model a filter with a shunt capacitor, setting the capacitor value to 99,999.99F (I *do* mean Farads) to simulate a perfect zero-Ohms impedance across the whole frequency range indicates that we can expect infinite attenuation from the capacitor with any finite source impedance.

But, as I hinted towards the end of section 7.7, this is not the whole story, and in fact stray magnetic field coupling (i.e. stray mutual inductance) between current loops on either side of the capacitors always occurs. The electricity that we think of as flowing in conductors is not actually confined to the conductor.

Taking the DC-Link capacitor in Figure 7.7-2 as an example, stray mutual inductance couples noise from the switcher's current loops (paths) into the loop comprising the Power supply, allowing the noise to bypass the DC-Link capacitor and increasing emissions.

Also, stray mutual inductance coupling from the switcher's current loops into the DC output circuit allows noise to bypass the DC output capacitor, also increasing emissions.

Figure 7.9-1 shows the major current loops associated with the mains input – the area filled in with a dark blue colour, and those associated with the switcher circuit – area filled in with a red colour.

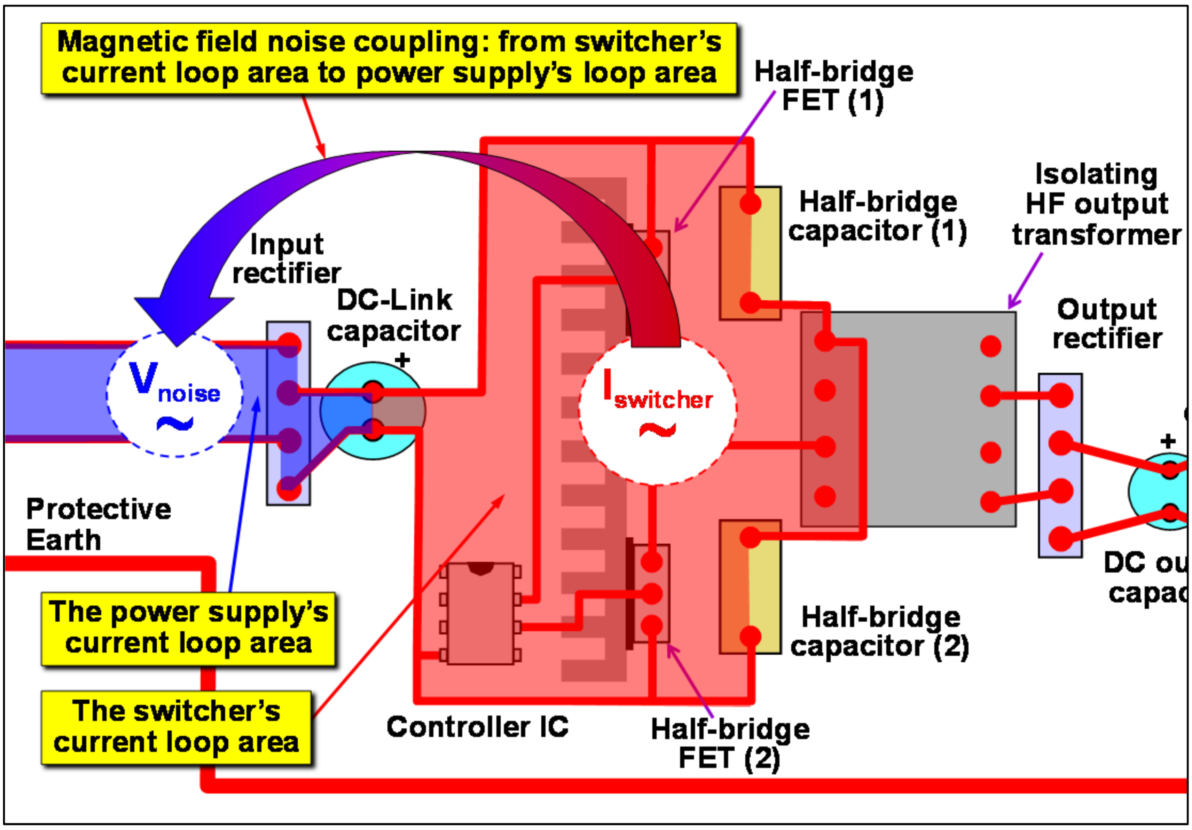

Figure 7.9-1 Identifying current loops in the example half-bridge switcher

Figure 7.9-2 shows the same PCB assembly in elevation view rather than in plan, with the magnetic field pattern lines associated with the switcher current loops shown. Where these field lines pass through the Power supply loop, they couple switcher noise into it by mutual inductance.

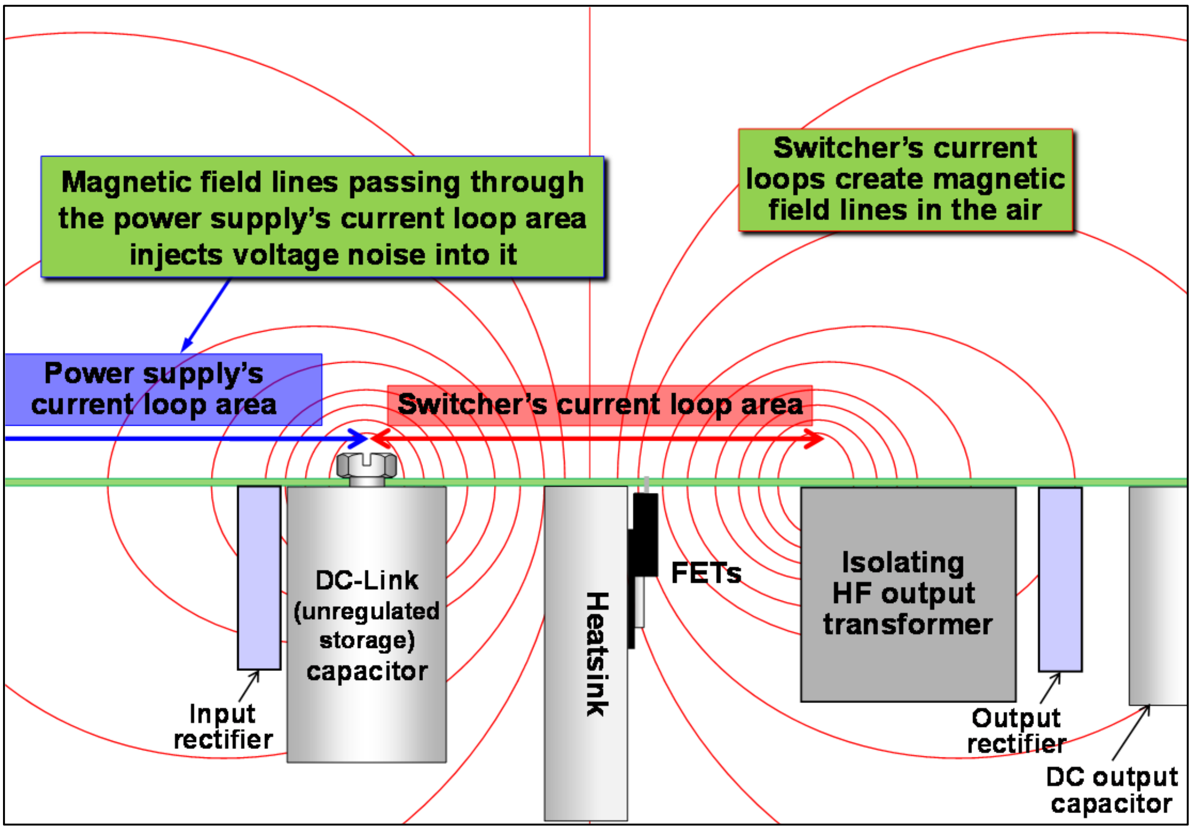

Figure 7.9-2 Stray magnetic field coupling between the current loops in the example

Although a PCB has been used as the example of mutual inductance noise coupling, the exact same problem arises with any type of physical construction, for example high-voltage stacks of switching devices connected to DC-Links by busbars.

What we might call "mutual capacitance" coupling also occurs between conductors in a circuit, creating stray capacitive noise coupling that also bypasses DC-Link and other suppression capacitors or other filter components.

The general expression for mutual inductance coupling of this sort is:

$$V_{COUPLEDNOISE} = -M_{STRAY} \cdot (dI_{SWITCHER}/dt)$$

— where M_{STRAY} is the stray (accidental, parasitic) coupling between two current loop areas in Henries. This simple formula assumes that the victim circuit's impedance is much larger than $1/(2\pi f \cdot M_{STRAY})$ where f is the frequency in Hertz.

It is important to understand that mutual inductance coupling takes noisy *currents* in the source circuit's current loops (in this example, those which flow in the power switching circuits) and creates noise *voltages* in series with the victim circuit's current loop (in this example, the Power supply).

Even if the mains supply had a zero impedance (impossible, of course) and so the above equation did not apply, there would still be some noise voltage injected into it by the stray mutual inductance coupling.

The above equation also assumes that the size of the victim circuit is such that it is not resonant at the frequency f – which is only true at low frequencies, say below 1MHz (but this depends on many factors, not least – for a mains power cable – the design of any mains filter that is fitted). Things get much more complex than this simple equation can cope with if these conditions are not met!

Simple equations are almost always to be distrusted in EMC engineering! For any kind of accuracy in assessing a real-life design prior to the very costly and time-consuming steps of building the design and testing it, we need field solvers that can pass parameters into circuit simulators.

Basic circuit simulators don't reveal mutual inductance/capacitance noise coupling because they assume that currents and voltages do not have associated magnetic (H) and electric (E) fields. But even if they did, they would be unable to perform correct simulations without all the coupling coefficients between all the different parts of the circuit being specified by the user.

These days we can purchase field solvers which extract the necessary M_{STRAY} (mutual inductance) and C_{STRAY} (what we might call "mutual capacitance") coupling coefficients from the physical structure of the conductors and insulators, and can automatically feed them back into the SPICE™ (or other) circuit simulator to get an accurate simulation of the power converter's electrical behaviour *as constructed*.

These coupling coefficients are part of what is often called the "hidden schematic" – the stray (some people say "parasitic") elements of the circuit that we get for free (even though we don't want them at all).

By using field solvers to extract the entire hidden schematic and feeding it back into circuit simulators, we can design and verify our product designs in the virtual world, improving them until we can guarantee the Signal Integrity (SI), Power Integrity (PI) and *almost* guarantee the EMC, of any electronic design before the first prototype is constructed, saving a great deal of time and money and reducing financial risks.

The field solvers and circuit simulators that can do this aren't cheap, but when their costs and benefits are analysed they are usually found to pay back their total investment within a year – making them "no brainer" decisions for Financial Directors (money is always available for such sure-fire investments). But most engineers don't appear to know how to talk to the financial people who really control their companies and so huge financial benefits go begging, leading to uncompetitiveness and possible failure in the long term.

Read [103] if you are interested in overcoming this problem of engineers communicating with financial managers, which I think is the single most important issue in most electronic engineering companies today.

Anyway, getting back to the mutual inductance noise coupling problems associated with our example half-bridge switcher's board assembly, we can obviously route the traces on the PCB so that the current loops on both sides of the DC-Link capacitor have the smallest areas, and so that they are as far apart from each other as possible given that both loops share the same capacitor connections.

Especially, in any PCB layouts, we absolutely *must* avoid *any* overlaps between the current loop areas associated with the circuits on either side of the DC-Link capacitor.

When using cables or busbars instead of PCB traces or planes, we follow the same sort of guidance as listed in section 7.7 and either twist the send/return conductors associated with each current loop, or use laminated busbars or PCB traces/planes on adjacent layers in the stack-up, to keep all current loop areas as small as possible, and as far apart as possible.

We can apply this approach to our example half-bridge switcher layout by changing to a 2-layer PCB and making sure that all send/return current paths are routed broadside to each other (i.e. in parallel with each other on adjacent layers). Figure 7.9-3 shows how this might be done for the example switcher in Figure 7.9-1, moving the components around as required to make it easier.

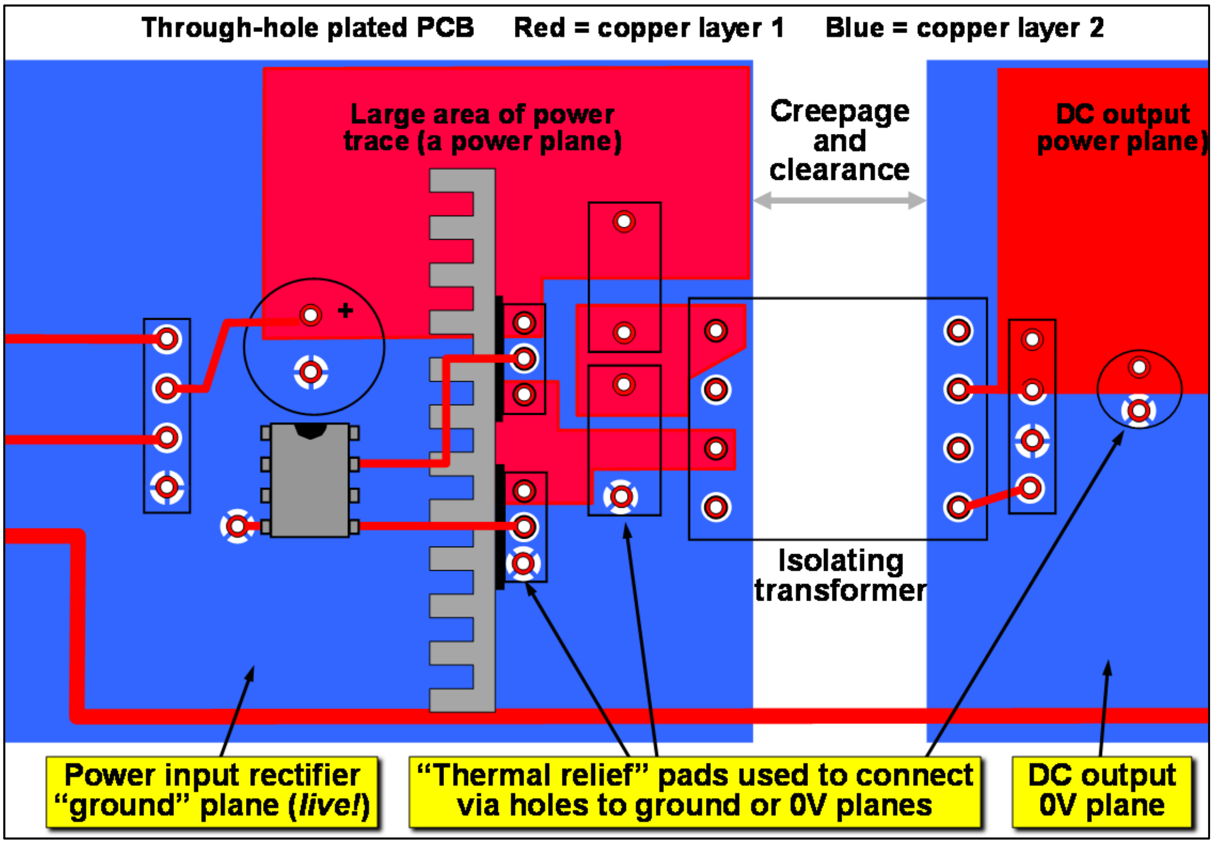

Figure 7.9-3 Using a two-layer PCB to reduce current loop areas dramatically

This approach considerably reduces the current loop areas because their maximum width reduces to the spacing between the send/return trace layers. On a two-layer PCB this, of course, is the thickness of the board itself (typically 1.5mm).

This approach works because all currents (including strays) naturally prefer to flow in loops that have the lowest overall impedance, as discussed in [33], [85], and either Chapter 5 of [4] or Chapter 2.7 of [5] (they use the same text and figures).

When send/return currents flow in very close proximity on adjacent parallel metal surfaces they experience maximum mutual inductance and maximum capacitance between them, creating the lowest overall loop impedance above a few kHz. This is also, of course, why we get good EMC by using laminated send/return busbars and twisted conductors.

The PCB layout techniques used in Figure 7.9-3 have many other benefits for SI, PI and EMC, see Chapter 7 of [5] or [37].

If we can't reduce the stray mutual inductance coupling by enough using the techniques described so far in this section, we still have some further design options:

h) Interpose another current loop that spaces the Power input loop further away from the switcher loop, where the H fields are weaker, as shown in Figures 7.9-3 and 7.9-4. This requires an additional DC-Link capacitor to create the interposed loop, but it only needs to have a low impedance at the noise frequencies being coupled by the stray mutual inductance so might not need to be as large as the DC-Link capacitor.

   Figures 7.9-4 and 7.9-5 show the use of this technique.

i) Shield the current loops from each other by placing a large plate of aluminium between the loop areas, or by enclosing the switcher circuits in a metal box, see Chapter 6 of [5] for the design details for effective shielding.

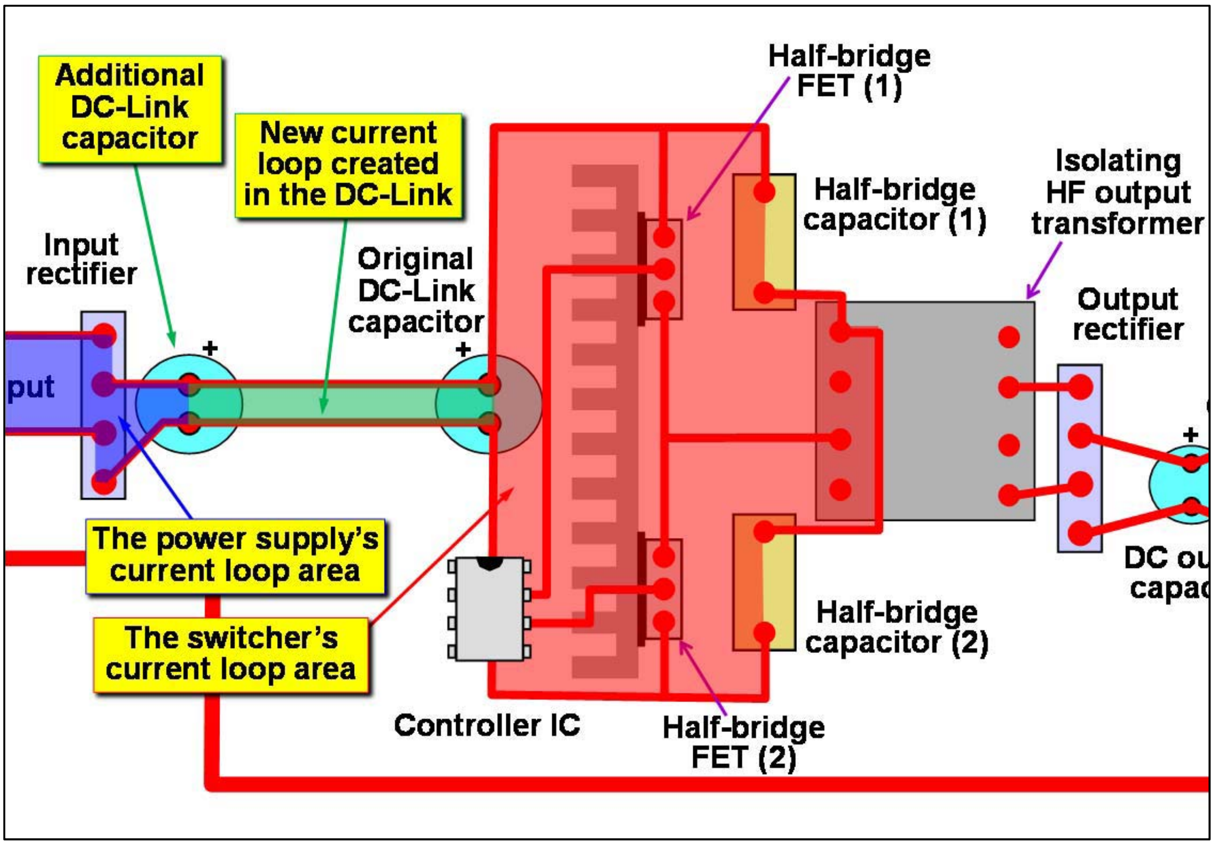

Figure 7.9-4 Adding an extra current loop into the DC-Link

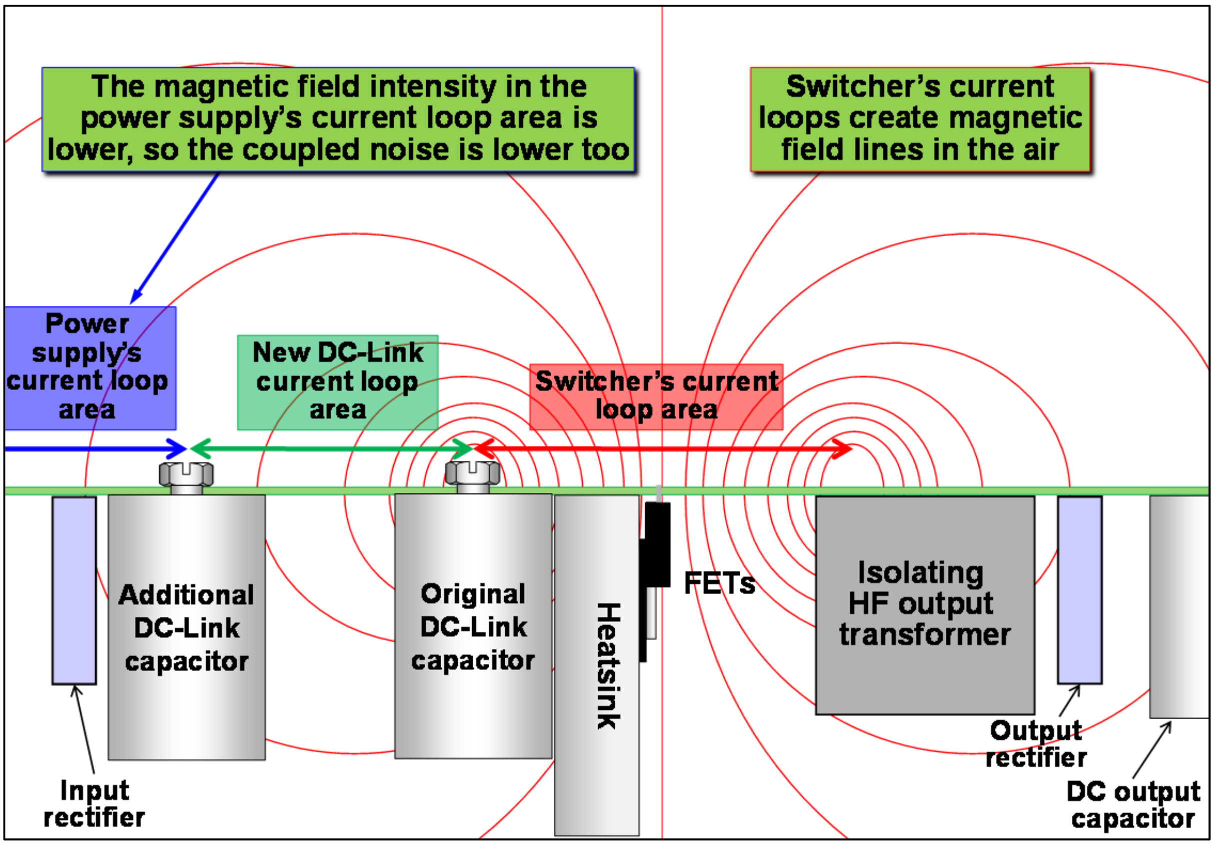

Figure 7.9-5 An extra current loop can move the power supply input loop further away, into a region with lower stray coupling from the switcher's circuits

In the above discussion I have tended to focus on suppressing the stray mutual inductance noise coupling that “bypasses” the DC-Link capacitors. However, the same design techniques work just as well for the stray “mutual capacitance” coupling that also allows noise to bypass the DC-Link capacitor.

And although the discussions in this section have focussed on the noise suppression achieved by the DC-Link capacitor, they apply equally well to any shunt capacitors or series inductors (chokes) used for noise suppression (i.e. filtering) at any frequency in any situation, on any DC or AC power, signal or data conductors.

For example, the exact same stray inductive and capacitive coupling effects allow switching noises to bypass the DC output capacitor and appear in the DC output circuit.

We might be tempted to think that noise in a DC or PWM DC or AC output circuit will not be measured by EMC tests that only measure the noise on the mains cable, but this would be incorrect. For a nice quiet mains cable, we need quiet-enough output conductors. How quiet, depends upon how the output conductors and the circuits they power are shielded (see Chapter 4 of [5] for the design issues associated with shielding cables, and Chapter 6 for designing shielded enclosures).

Also, the exact same stray coupling effects allow unwanted noises from any circuits to bypass any filter components, whether they are capacitors or inductors, and generally cause worse emissions at higher frequencies. These design issues are thoroughly dealt with in the guidance on cabling, filtering and shielding in Chapters 4, 5 and 6 of [5].

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

[33] “Maxwell’s equation, Quantum Electrodynamics, and good installation practices for SI, PI and EMC”, Keith Armstrong, The EMC Journal, Issue 91, November 2010, pages 32-46

[35] “EMC Techniques for Heatsinks”, Keith Armstrong, The EMC Journal, Issue 89, July 2010, pages 31-38, available from the archives at www.theemcjournal.com.

[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).

[42] “EMC Design of High-Frequency Power ‘Switchers’ and ‘Choppers’”, Keith Armstrong, The EMC Journal, Issue 94, May 2011, pp 39-50, available from the archives at www.theemcjournal.com

[64] “EMC Design of High-Frequency Power ‘Switchers’ and ‘Choppers’ – Design techniques for HF isolating transformers”, Keith Armstrong, The EMC Journal, Issue 95, July 2011, available from the archives at www.theemcjournal.com

[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

[92] “EMC Design of High-Frequency Power ‘Switchers’ and ‘Choppers’ – Shielding (screening) the power converter’s output cable”, Keith Armstrong, The EMC Journal, Issue 100, May 2012, pages 32-38, available from the archives at www.theemcjournal.com

[99] “EMC Design of High-Frequency Power ‘Switchers’ and ‘Choppers’ – Suppressing RF noise in DC inputs, outputs, and DC-Links”, Keith Armstrong, The EMC Journal, Issue 101, July 2012, pages 30-38, available from the archives at www.theemcjournal.com.

[100] “EMC for Systems and Installations”, Tim Williams and Keith Armstrong, Newnes 2000, ISBN: 0-7506-4167-3, www.bh.com/newnes, RS Components part number: 377-6463

[101] "Good EMC Engineering Practices in the Design and Construction of Industrial Cabinets", by Keith Armstrong, published by REO (UK) Ltd and downloadable for free from: www.reo.co.uk/knowledgebase. Although the title concerns industrial cabinets, the good EMC practices described apply equally well to any system comprising two or more separate electrical/electronic modules/products/units that are interconnected by any conductors (power, signals, control, data, etc.).

[102] "Good EMC Engineering Practices in the Design and Construction of Fixed Installations", by Keith Armstrong, published by REO (UK) Ltd and downloadable for free from: www.reo.co.uk/knowledgebase. This focuses on buildings on land, but most of the good EMC practices described apply equally well in any/all electrical/electronic installations, including offshore and all vehicles (air, space, cars, rail, marine, submarine etc.).

[103] "Getting what you want", Keith Armstrong, The EMC Journal, Issue 77, July 2008, pages 19-22, available from the archives at www.theemcjournal.com.