# Practical Power Planes for EMC

**T.P.Jarvis, RadioCAD Ltd, t.jarvis@radiocad.com**

**Abstract:** *The ability of the ground-plane to provide a ubiquitous return path for signal traces in Printed Circuit Boards (PCBs) led to its adoption to preserve signal integrity as clock speeds increased. Additional power-planes were added as multi-layer PCB technology became commercialised. Although primarily planes were added in pairs for manufacturing reasons, it was quickly realised that a plane-pair could provide an effective method of powering Integrated Circuits.*

*Plane-pairs provide the dual advantages of 'free' power-supply decoupling capacitance and reduced noise coupling between IC power pins. They have become the de-facto solution for multi-layer digital designs.*

*The physical resonance of such structures are often neglected in the design process, thereby introducing serious undesirable side-effects. This paper investigates these side-effects both in theory and by practical example. The investigation shows that plane-pairs are not always the best solution to IC power problems and should not be employed simply by default. The author concludes by deriving best practice guidelines for the appropriate use of power and ground planes.*

## Introduction

Two historical factors led to the now common-place use of power and ground planes in digital PCB design. Namely: the need to preserve signal integrity as processor speeds increased and the advent of commercial multi-layer PCB technology.

A ground-plane provides a ubiquitous return path for signal traces in a PCB provided that it covers the same board area as the signal traces and is uncut. The transmission-line impedance of signal traces (on a given PCB layer) is maintained constant with respect to the ground-plane. A ground-plane also provides some shielding for PCB signals traces. All these effects improve signal integrity and reduce unwanted noise emissions. This is achieved without placing any restrictions on the signal trace routing path.

In multi-layer PCBs a single ground-plane presented a copper-balance manufacturing problem. Historically this was overcome by adding a second plane on an opposing layer. The additional plane is normally used to carry DC power to semiconductor devices, hence the terms 'power-plane' and 'plane-pairs'.

In this paper the ground-plane is defined as the common reference plane for all the signals interconnecting devices on a PCB. Although strictly speaking the ground-plane is also a power-plane, the term 'power-plane' in this paper excludes the ground-plane and refers solely to a plane carrying a DC voltage used to power devices on the PCB.

The combination of a power and ground plane sandwich (plane-pair) within a PCB provides 'free' power-supply decoupling capacitance and current spreading within these planes improves the noise isolation between IC power pins. In addition to these desirable effects power and ground planes have undesirable side-effects arising from their physical resonance.

Until fairly recently these resonant effects had been ignored by designers. Resonant effects undermine the effectiveness of much current practice in power-plane design for EMC. Resonant effects in power-planes lead to excessive emissions from digital circuits, whereas the same resonant effects in the ground-plane can reduce the immunity of sensitive analogue circuits.

## Ground-Plane Resonance

Copper planes act like patch antennas. Resonance occurs at frequencies where an integer multiple of half the wavelength is equal to one of the physical dimensions of the ground-plane (figure 1). Note however that cables and other large devices attached to the PCB will alter the resonant frequencies.

Because the ground-plane needs to be physically large (compared to traces and other PCB structures), it forms a good antenna at relatively low frequencies. This can lead to problems with both EMC emissions and immunity.

Good EMC immunity is particularly important in critical automotive applications. The Automotive Directive[1] requires radiated immunity to 30 V/M and most car manufactures stipulate higher field strengths (emissions are considered in detail in the following section on plane-pair resonance).

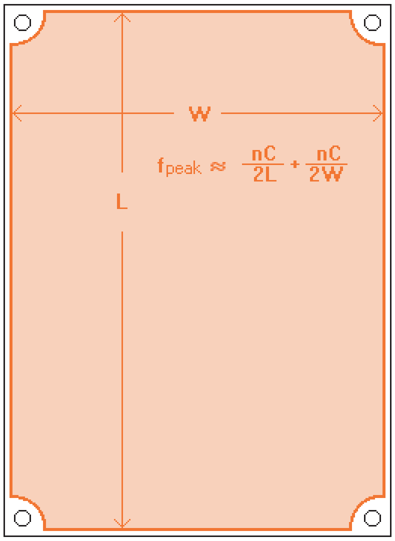


**Figure 1: Ground-plane example**

The ground-plane acts as the major antenna during immunity testing (neglecting cables). Significant currents are induced into the ground-plane. At resonance significant voltages develop across the plane from point to point. These disturbance voltages are not perfectly mirrored in the signal traces. As a result semiconductor devices see RF disturbance voltages develop differentially between various pins (particularly signal-ground). These devices then demodulate the RF envelope producing well documented susceptibility problems[2][3].

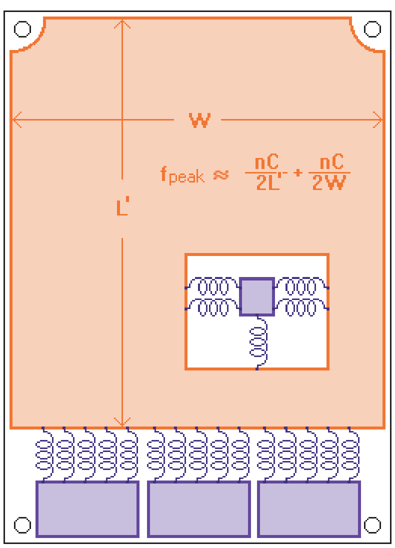


**Figure 2: Isolating sensitive analogue**

One solution involves isolating sensitive (susceptible) circuits in their own small areas off the ground-plane (figure 2).

An isolated circuit will either amplify sensitive signals to a higher, less sensitive level or convert them to digital signals before they are passed into the main PCB area. All the signals interfacing isolated areas (blue in figure 2) must be appropriately filtered at the barrier with the main ground-plane (red figure 2). This is an essential and oft overlooked requirement. Routing signal traces over ground-plane cuts or routing them from one area to another without proper filtering usually produces worse effects than not segregating at all. An appropriate filter is one that has a high impedance at high frequencies and yet does not affect the low frequency wanted signal. In the case of sensor inputs (such as the bottom row, figure 2) filtering of the cable inputs is also required as the cables also act as unwanted antenna. Some suitable filters for immunity to 1 GHz are shown in figure 3.

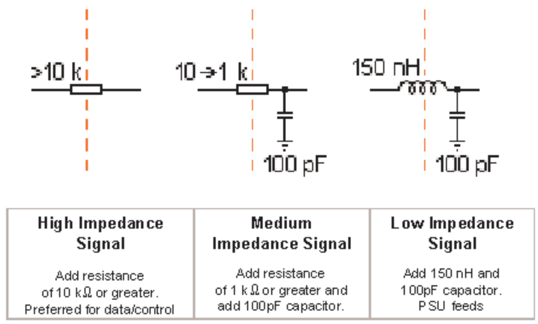


**Figure 3: Barrier filter examples**

Resistors work well in high impedance circuits because at RF frequencies signal trace transmission line impedance is low (typically between 100 and 200Ω). Therefore the high impedance of the resistor acts to reflect most of the RF energy.

## Plane-Pair Resonance

Plane-pairs are generally used to power ICs on a PCB. The supply noise generated by the ICs on a PCB is therefore driven directly into the plane-pair. Energy at the resonant frequencies of the plane-pair will tend to be radiated by the planes acting as a single patch antenna. To understand the mechanism it is necessary to understand not only the resonant effects of planes but also the noise characteristics of IC power feeds.

Figure 4 shows a typical output stage of a CMOS IC. In the stable state no current flows and no power is consumed. For a short period during logic transition both transistors conduct. A current impulse is drawn from the supply to make the transition happen. Transitions are happening all the time throughout the CMOS ICs on the PCB resulting in noise voltages due to the impedance of the power feed.

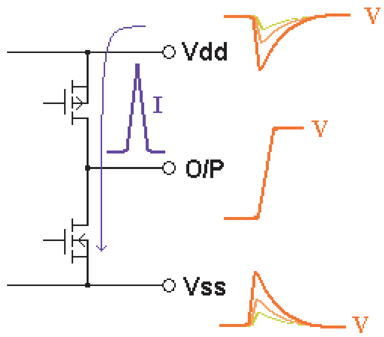


**Figure 4: CMOS totem-pole output**

Assuming that the ICs are clocked the ideal spectrum of the power noise would appear as in figure 5.

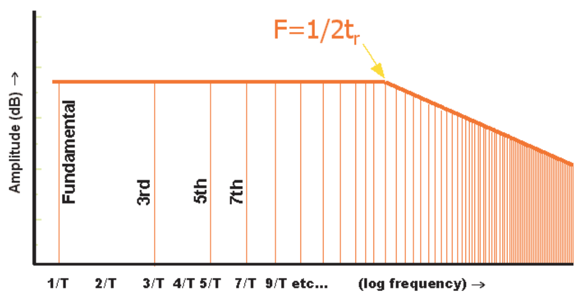


**Figure 5: Power feed noise spectrum**

Harmonics carry on at the same intensity up to the transition frequency, which is dependent on the rise time $t_r$ of the current impulse. For modern CMOS the transition frequency is often more than 1 GHz. To determine actual emissions one must multiply the noise spectra with the frequency response of the power feed system.

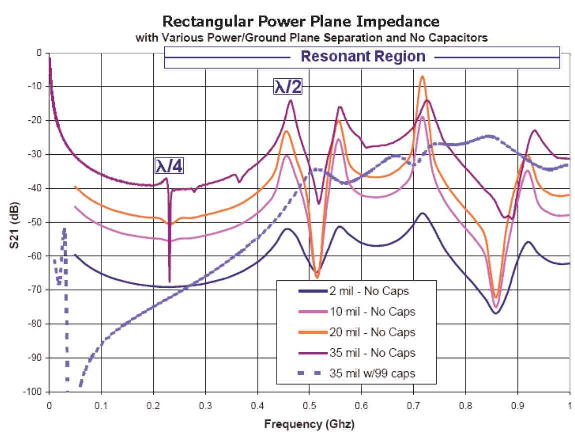


**Figure 6: Power-plane response -v- plane separation**

Since the noise spectrum is flat up to the transition frequency, the emission profile is dependent on the frequency response of the power feed system.

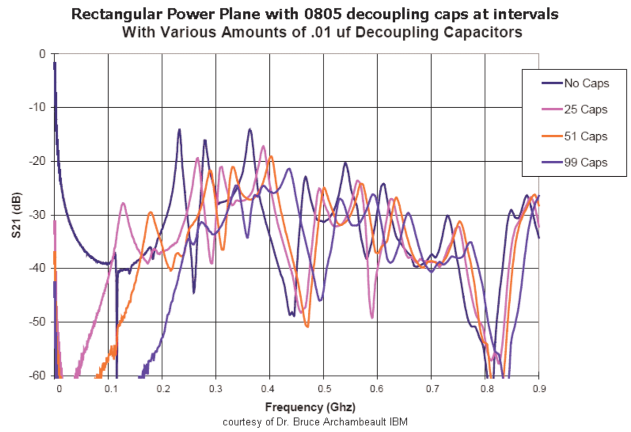


**Figure 7: Power-plane response -v- number of decoupling capacitors**

Frequency responses for a 300 x 250 mm rectangular plane-pair are shown for varying plate separation (figure 6) and varying number of decoupling capacitors (figure 7)[4]. It can quickly be seen that plane resonance dominates the emission profile. Simply adding more decoupling capacitors has little effect in the resonant region. Reducing the plate separation yields a significant reduction in emissions in the resonant region.

## Plane-Pair Emissions Tests

To further investigate power-plane emissions a PCB was designed where plate separation and other features could be varied easily (figure 8).

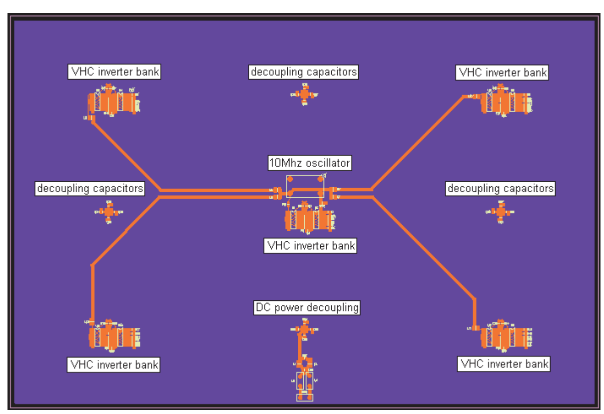


**Figure 8: Power-plane emissions test-jig**

This comprised a 300 x 200 mm 2 layer PCB with five banks of inverters: one central bank and four in the board corners. Each bank was made of twelve 74VHC04 inverters, each inverter driving a 120Ω load. All signal traces were routed on the top layer with the bottom layer a solid ground-plane covering the whole PCB (figure 9).

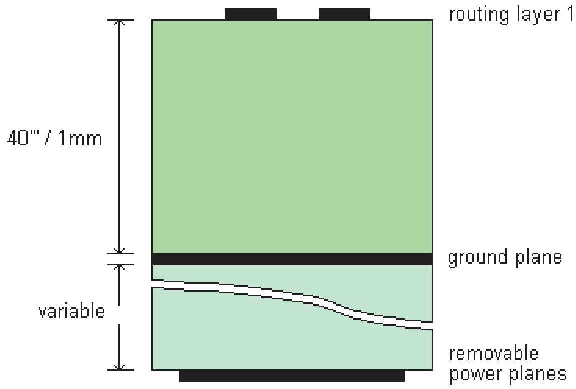


**Figure 9: Test-jig PCB stack**

A variety of removable power-plane PCBs were attached to the underside of the test-jig PCB. Vias in both PCBs were aligned, threaded with 0.4mm dia copper wire and soldered. This mechanism providing good RF connection to the power feed circuit. A single, central clock was employed to feed all five inverter blocks. To eliminate clock trace emissions from the results, each clock trace was filtered to progressively remove harmonics above 80 MHz. Radiated emissions were then investigated from 200 MHz to 1 GHz in RadioCAD's 3M semi-anechoic chamber (figure 10).

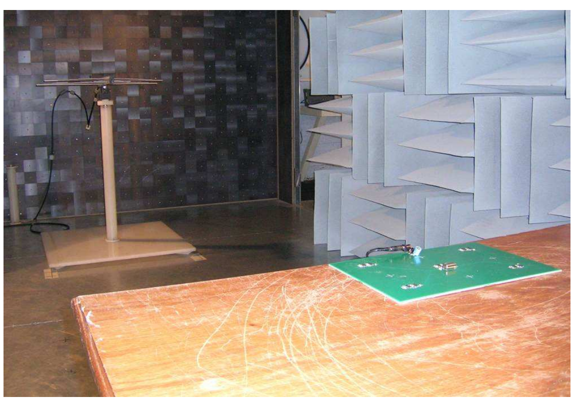

**Figure 10: Emissions test set-up**

For all of these investigations the test-jig was powered by a linear DC supply placed on the chamber floor. To ensure that the DC power lead wasn't contributing any significant antenna effect, selected tests were repeated using only battery power (figure 11). The 10 MHz clock used for tests didn't produce significant emissions above 700 MHz, so some tests were repeated using a 50 MHz clock. The 50 MHz results didn't add any additional information and are therefore not included here.

**Test results:** firstly emissions for a plane-pair of the same size as the ground-plane (200 x 300 mm) were compared for plane separations of 1.6, 0.2 and 0.05 mm (figures 12 to 15).

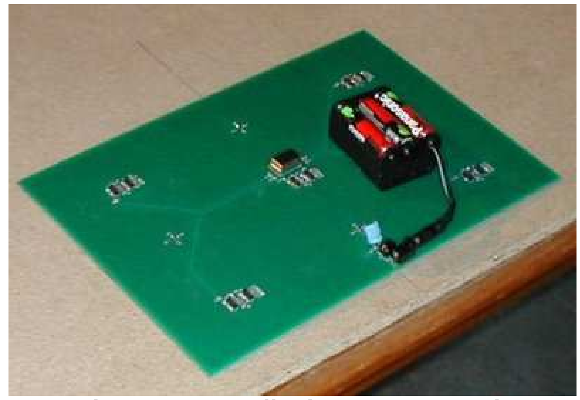

**Figure 11: Test-jig, battery powered.**

It can be seen that, as predicted, emissions reduce as the plane separation reduces.

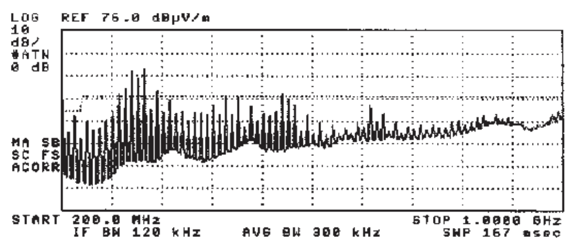


**Figure 12: 300x200mm solid plane-pair, 1.6mm separation, ten 100nF caps, Horizontal Pol.**

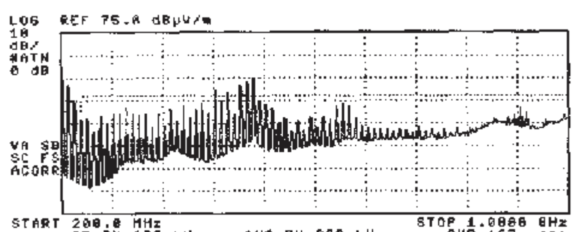


**Figure 13: 300x200mm solid plane-pair, 0.2mm separation, ten 100nF caps, Horizontal Pol.**

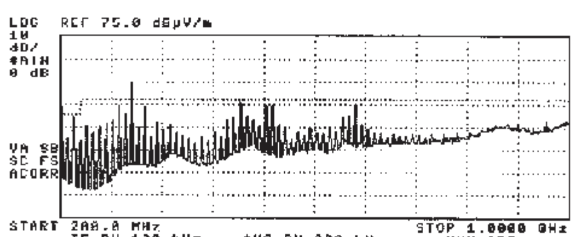


**Figure 14: 300x200mm solid plane-pair, 0.2mm separation, 36 100nF caps, Horizontal Pol.**

From the PCB dimensions the resonant region was expect to start at 250 MHz. Figure 13 shows significant and unexpected emissions at 200 MHz (below the resonant region). By increasing the number of power supply decoupling capacitors from 10 to 36 these were removed (figure 14). Again as predicted, this increase had little effect in the resonant region.

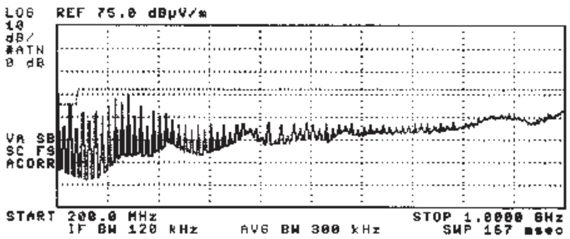


**Figure 15: 300x200mm solid plane-pair, 50µm separation, 36 100nF caps, Horizontal Pol.**

Another test was made without a power-plane (only a ground-plane). The inverter blocks were powered by 0.4 mm hook-up wire held close to the ground-plane thereby simulating narrow PCB traces (figure 16).

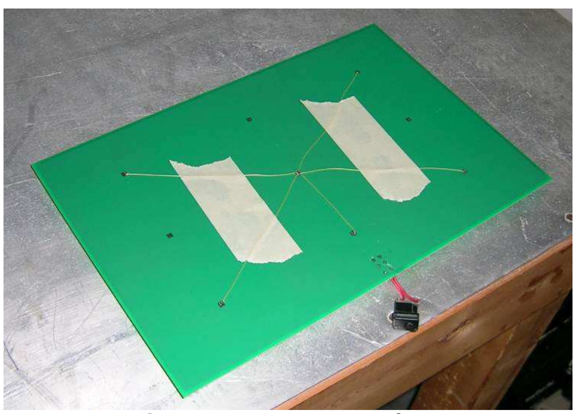

**Figure 16: No power-plane**

Interestingly this set-up, with only 10 decoupling capacitors (2 per block), has remarkably similar radiated emissions to a plane-pair at 50 µm with 36 decoupling capacitors (compare figures 15 and 17).

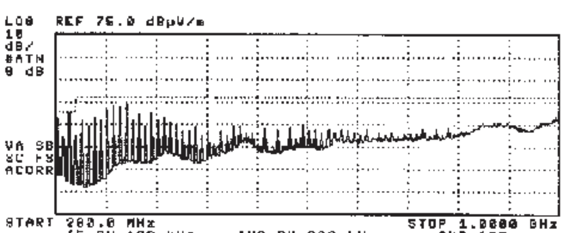


**Figure 17: 300x200mm ground-plane, no power-plane, ten 100nF caps, Horizontal Pol.**

The author expects that impedance mismatch explains the significantly lower emissions in both cases. The RF impedance of a typical power feed circuit is of the order of 1 to 10 Ω. A 0.2mm plane-pair is fairly well impedance matched to this. At 50 µm a plane-pair is not only better damped than at 0.2 mm, but has an RF impedance much less than 1 Ω. This means that the source and antenna are not well matched, resulting in lower emissions. Similarly a narrow (0.4mm) power trace is not only predominantly inductive, but has a high impedance of about 150 Ω. This is also not well matched to the noise source, again resulting in lower emissions.

The author expected the supply noise in the no-plane test set-up to be considerably more than the 50 µm plane-pair test set-up. Power noise spectra for both is given in figures 18 and 19.

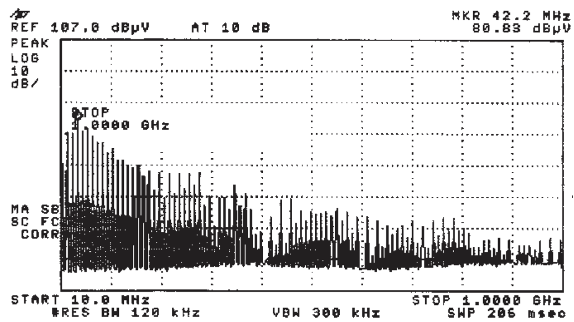


**Figure 18: 300x200mm solid plane-pair, 50µm separation, 36 100nF caps, Vcc Noise**

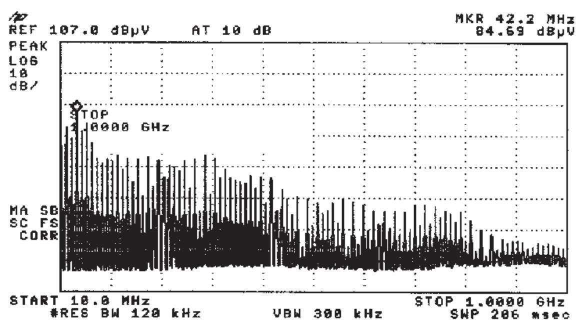


**Figure 19: 300x200mm ground-plane, no power-plane, ten 100nF caps, Vcc Noise**

The no-plane supply noise is 4.5 dB greater than the 50 µm plane-pair at the peak frequency (40 MHz). The difference was less than expected. It was thought that the smaller number of decoupling capacitors was contributing to this difference. Since it isn't practical to place more than one or two decoupling capacitors per power pin an alternative approach of using lower-impedance X2Y attenuators[5] was tried. The results however were inconclusive with further investigations needing to be undertaken.

**Regional power-planes:** Modern EMC practice is not to use a single power-plane but to use smaller regionally power-planes. Using this techniques it is possible, in theory, to push the resonant region of all the plane-pairs outside the compliance region. For example: power-planes no larger than 75 x 75 mm should be suitable for emissions up to 1 GHz. Regional power-planes tend to be interconnected using inductors or ferrite-beads (figure 20).

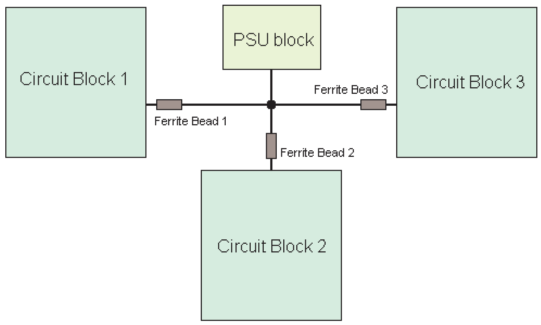


**Figure 20: Regional power-plane example**

To investigate regional power-planes below 1 GHz a test PCB of 5, 75 x 75 mm planes was made (figure 21). Each plane fed one inverter block and were interconnected using Murata BLM41PG102SN1 ferrite-beads[6].

**Figure 21: Regional power-plane test-jig**

Emission results however show an unexpectedly significant peak at 310 MHz (figure 22).

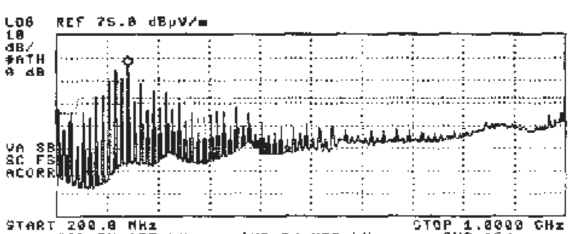


**Figure 22: 300x200mm ground-plane, regional power-planes, Horizontal Pol.**

Comparing the regional and solid power-plane results at the same separation from the ground-plane (figures 12 and 22), it can be seen that emissions at the first resonance are virtually unaffected. Higher order resonant emissions have been reduced (particularly around 500 MHz).

The most likely explanation is that the regional planes, being fed with synchronous noise (due to a common clock), are acting together much like a phased array. This effect results in resonant emissions due to the PCB dimensions rather than the dimensions of the individual planes.

## Summary - 'Best Practice' When Using PCB Planes

The case for incorporating a ground-plane into a PCB is conclusive. Resonant effects however can cause EMC immunity problems. In particular highly sensitive circuits such as analogue sensor inputs need special treatment. This always involves isolating them from the injected noise source. There are a number of techniques to achieve this. One of these is to remove sensitive circuits from the main ground-plane.

In addition to immunity problems plane resonance can cause excessive EMC emissions. The widely used technique of using small regional power planes is not guarantied to overcome emission problems. Only small plane separations of 50 μm (2''') or less reliably reduce power noise emissions. These separations however require specialist PCB manufacturing techniques.

Good emission performance can be achieved using conventional PCB manufacturing techniques without power-planes. This is achieved by using only a ground-plane and narrow, inductive traces as power feeds. The approach has limitations:

(i) DC Volts drop due to trace resistance,
(ii) Increased noise at power pins,
(iii) May require alternative techniques to restore copper balance such as hatching the ground-plane or adding dummy copper to signal layers.

As with all engineering it is a matter of determining whether the benefits outweigh the disadvantages. □

## References

[1] Commission Directive 95/54/EC adapting to technical progress Council Directives 72/245/EEC and amending Directive 70/156/EEC. OJ No L 266, 08/11/995.

[2] Improving the immunity of sensitive analogue electronics, T.P.Jarvis and I.R.Marriott UK EMC Journal, Feb 1997.

[3] No More Rat-a-Tat-Tats: Designing Telephone Equipment for Immunity, T.Jarvis, Compliance Engineering Magazine, Nov 2001.

[4] Effective Power/Ground Plane Decoupling for PCB, Dr. Bruce Archambeault, The IEEE Southeastern Michigan EMC Society September 17, 2003.
http://www.ewh.ieee.org/r4/se_michigan/emcs/DL-ARCH-decoupling3.pdf

[5] X2Y® Technology used for Decoupling, Dale L Sanders, James Muccioli, Tony Anthony and Dave Anthony, X2Y Attenuators, LLC. IEE New EMC issues in Design: Techniques, Tools and Components 28 April 2004.

[6] http://www.murata.com/cn/catalog/k99/c04_01.pdf

## Acknowledgement

The author would like to thank TopScoreUK for kindly providing the test PCBs used in the preparation of this paper.

| | |
|---|---|
| **TopScoreUK**<br>http://www.topscoreuk.co.uk | **PO Box 6551**<br>**Syston**<br>**Leicester LE7 3YR**<br>**United Kingdom** |

## Biographical Notes

**T.P.Jarvis, July 2004** – Tim Jarvis is an independent consultant. His company RadioCAD assists clients in designing electronic products for compliance with European directives and worldwide standards. Tim has worked in the electronics industry since 1983 and specialises in RF design and EMC (www.radiocad.com).