Power Tracks instead of Planes to Reduce Radiated Electromagnetic Fields

Frank B.J. Leferink — Wim C. van Etten

Thales Netherlands
Hengelo, The Netherlands
frank.leferink@nl.thalesgroup.com

University of Twente
Enschede, The Netherlands
frank.leferink@utwente.nl

University of Twente
Enschede, The Netherlands
w.c.vanetten@el.utwente.nl

Abstract

The noise voltage in the reference or ground of a printed circuit board is often the cause of unwanted radiated emission. Power supply planes attribute to the noise voltage. By replacing the power supply planes by tracks, the noise voltage in the reference or ground can be reduced, which leads to a considerable reduction of the radiated electromagnetic fields. A potential disadvantage of using power tracks is a decreased power quality, due to a higher ripple voltage. Actual circuits, fed via power planes or power tracks have been designed and measured showing the impact on radiated electromagnetic fields with constant power quality.

Keywords

Power plane, inductance, radiated emission, noise voltage

Introduction

Unwanted radiated electromagnetic fields are often created by unwanted antenna currents, generated by a noise voltage in the reference (ground) conductor. The focus has been on reducing the net partial inductance of the reference so that the noise voltage is reduced [1],[2],[3],[4],[5]. This conventional model is not valid for high frequencies where the longitudinal dimensions of a product are in the order of magnitude of a wavelength, as shown in Figure 1. The conclusion drawn, using this HF model, is that a (unbalanced) transmission line should be, geometrically seen, asymmetrical [6], [7].

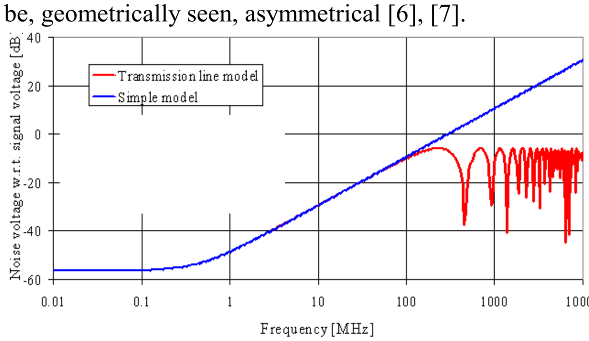

Figure 1: Noise voltage with respect to the signal voltage as function of the frequency

This conclusion can be extended to power planes too, leading to the statement that power planes should be omitted in order to reduce radiated emission.

The impact of removing power planes, and replace them by power tracks, on the radiated electromagnetic field strength of actual circuits is discussed in this paper. A possible disadvantage of power tracks is a signal integrity problem due to power variation, i.e. ripple, of the devices. This has been investigated by measuring the functional signals for the several PCB stack-ups.

Noise voltage

The noise voltage between two longitudinal points in a PCB is the source of unwanted radiated electromagnetic fields such as shown in Figure 2.

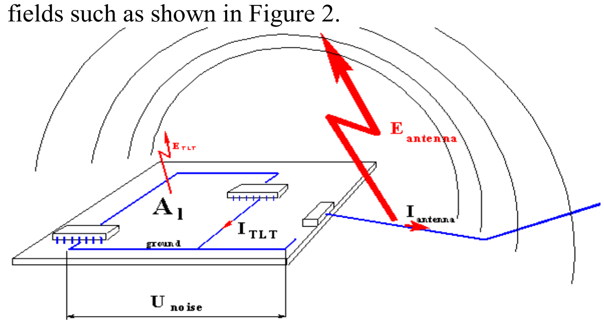

Figure 2: Radiated emission due to antenna currents, generated by noise voltages in the reference (ground plane)

In [6] and [7] a model has been developed via transmission line theory, called Signal to Noise Transformation (SNT), resulting in:

$$U_{noise} = \frac{Z_g U_s}{Z_s + Z_c} \frac{1}{\gamma_s} \frac{\left[(e^{-\gamma_s p} - 1) + \Gamma_l e^{-2\gamma_s p}(e^{-\gamma_s p} - 1)\right]}{1 - \Gamma_l \Gamma_s e^{-2\gamma_s p}} \qquad (1)$$

With:

	p	length transmission line
	γ_s	propagation constant
	Γ_l	reflection coefficent load
	Γ_s	reflection coefficent signal source
	U_s	voltage of source
	Z_c	characteristic impedance
	Z_g	impedance ground (return) plane
	Z_s	internal impedance signal source

For high frequencies, and if $Z_l = Z_s = Z_c$, then (1) reduces to

$$\frac{U_{noise}}{U_s} = \frac{Z_g}{2Z_c} \frac{1}{\gamma_s}\left[e^{-\gamma_s q} - 1\right] \qquad (2)$$

The amplitude is maximal if $\gamma_s q = (2n-1)\pi$ for n=0, 1, 2…., i.e. if $q = (n - 1/2)\upsilon / f$, resulting in a maximal noise voltage of

$$\frac{U_{noise}}{U_s} = \frac{L_{g_{net}}}{L_{loop}} \qquad (3)$$

In other words, the longitudinal noise voltage developed in the ground conductor is determined by the net partial inductance of that ground conductor with respect to the total loop inductance of the transmission line. This means that a (unbalanced) transmission line should be, geometrically seen, asymmetrical to reduce the noise voltage [6], [7].

Conventional power distribution

Power is generally provided via low impedance conductors (planes) so that the longitudinal voltage drop between two units during a high-frequency current pull is minimal. A cross-section of two typical 4-layer multi-layer PCB is drawn in Figure 3. VCC denotes the power plane, Sig1 and Sig2 the signal track planes and GND the ground plane. The instantaneous signal current generated by the device in the signal line has to be supplied by the decoupling capacitor C_{dec} and this current will generate a noise voltage in the ground system through the net partial inductance of the ground plane. In case of a lack of a nearby placed decoupling capacitor the instantaneous signal will be supplied by the power planes.

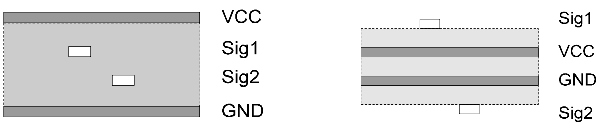

Figure 3: Cross-section of typical 4-layer PCB

The noise voltage $U_{noise,\ power}$ is, as a first order assumption, created by the instantaneous supply current I_{is} and the impedance of the reference (ground) plane Z_{noise}:

$$U_{noise,power} = Z_{noise} I_{is} \tag{4}$$

This simple, conventional, model leads to the same limited conclusions as drawn in the last two decade for the noise voltage due to signal line currents: reduce the impedance of the reference to reduce the noise voltage. We have shown that the noise voltage at high frequencies is not determined by the absolute impedance of the reference plane, but by the geometrical asymmetry of the transmission line system. This observation would imply the use of power supply <u>tracks</u> instead of supply planes. This has been drawn in Figure 4.

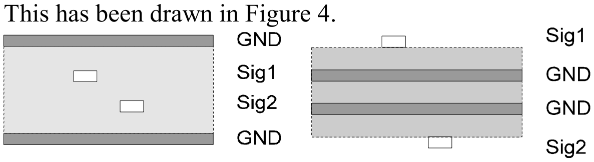

Figure 4: Cross-section of improved multilayer PCBs using the power isolation concept

Measurement data using PCB transmission lines

In [19] the measurement results obtained by using a set of printed circuit board transmission lines has been discussed. The data has been summarised in Figure 5.

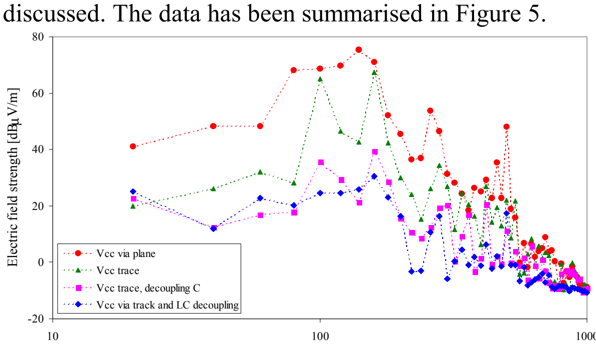

Figure 5: Summarised measurement data

The effect of preventing the power supply current to flow through the ground plane is obvious when comparing the measured data; A reduction of the field strength of approximately 50 dB can be observed. The influence of resonances, peaks around 100 and 200 MHz, can be seen.

Power quality

The disadvantage of power tracks is that the decoupling of a devices should be adequate in order to supply the circuit. We already suggested that in high-speed systems the dimensions of the power plane is large with respect to the propagation distance. Local decoupling, or using a better term, local stabilisation, is needed anyway. We should therefore focus on a mimum stabilisation for power quality and a maximum decoupling for reduced radiated emission. This concept has been discussed by other researchers before [8-18]. We will concentrate on the asymmetry of the (power supply) transmission line.

Experiments

A power supply (LM340-5 and capacitors), a clock generator, an octal driver (74ACT541) and an actual load (8 SMD resistors) are placed on a printed circuit board (PCB), as shown in Figure 6 and 7. The power supply was via a ground plane (Figure 6) and a power track (Figure 7). For reference, a PCB with signal traces above a ground plane has been measured too.

The radiated emission level has been measured with an absorbing clamp. To reduce external influences the PCBs were placed in metallic box. Note: in july the EMC laboratory was destroyed due to a fire, and shielded rooms were not available during the experiments.

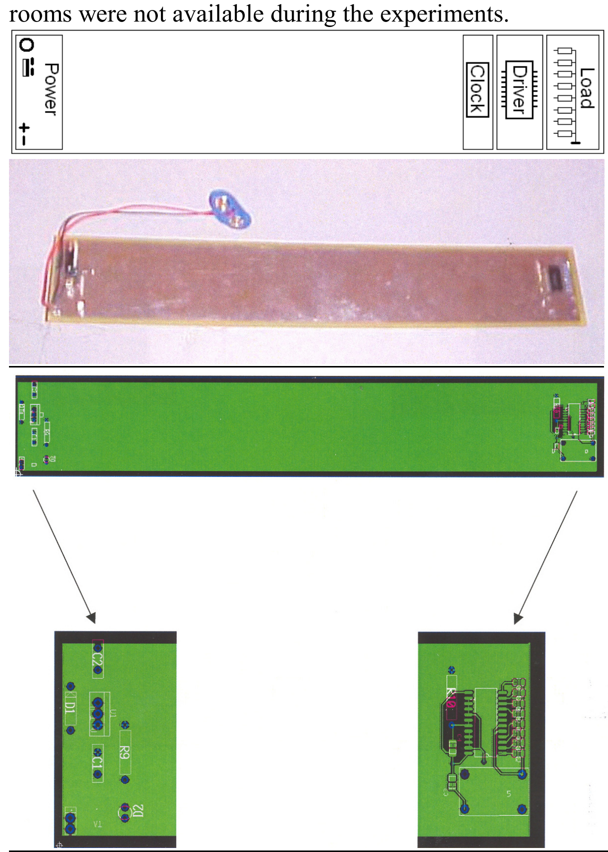

Figure 6: Printed circuit board 1, power plane

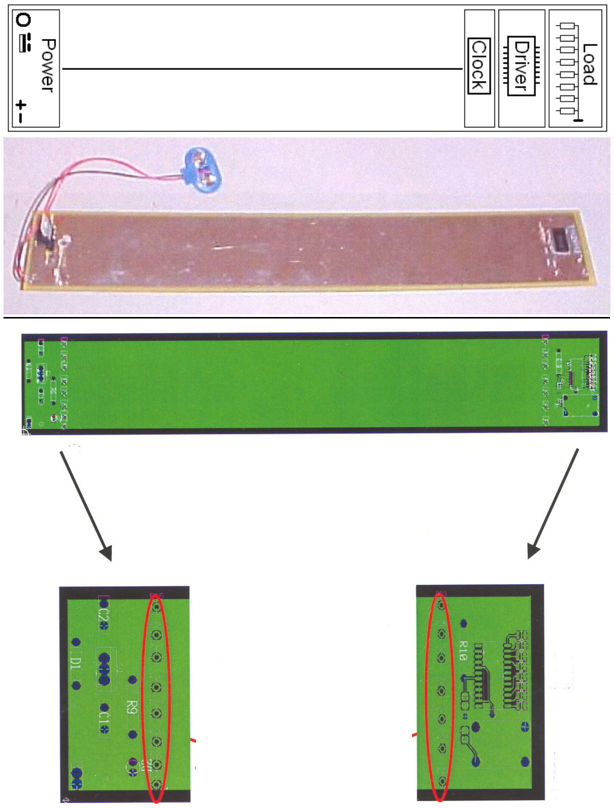

Figure 7: Printed circuit board 2, power track

A clock oscillator operating at 10 MHz and with fast rising edges (<1 ns) has been used as driver. Note that not the fundamental clock frequency but the risetime of the clock and driver is important. The clock oscillator and the driver where decoupled with respect to the bulk supply using a (SMD) series resistance and a (SMD) capacitor.
The output signal of the driver integrated circuit is shown in Figure 8, which proves that the signal is not distorted.

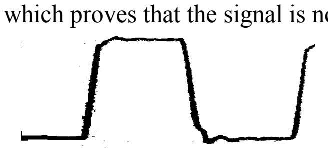

Figure 8: Signal on load resistors

The power ripple on the PCBs is shown in Figure 9, left for the plane, and right for the track.. The left is appr. 200 mV peak, and the right appr. 300 mV peak.

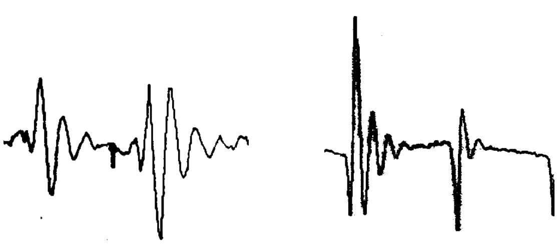

Figure 9: Voltage ripple on integrated circuit power pin, left for power plane, right for power track

Measurements have been performed at every 10 MHz harmonic between 10 MHz and 3000 MHz. The measured power is shown in Figure 10.

It shows

- PCB with signal track above a ground plane, so that the ground plane conducts the momentary return current, which causes a noise voltage, and the noise volatge causing an antenna current in the attached wire.
- PCB with power plane, wich conducts only a limited amount of the momentary current current due to the local decoupling capacitor+resistor.
- PCB with power track, wich again conducts only a limited amount of the momentary current current due to the local decoupling capacitor+resistor.

The curves show the measured power, with the absorbing clamp, at 10 MHz intervals (thin line). A 2-point moving average curve (thick line) has been added for readability. A logarithmic trend line is added, calculated using Excel, which shown even more the effect of the power track instead of the power plane.
Note that in this measurement the devices have been decoupled properly, so that only a limited amount of momentary (high-frequency) current is flowing through the power plane/track.

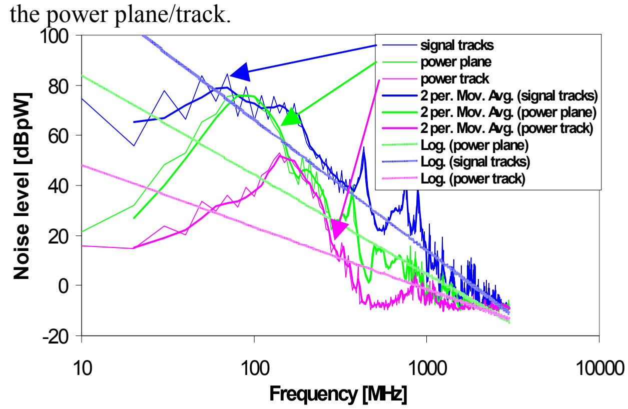

Figure 10: PCB with signal tracks above a ground plane, a PCB with power plane and a PCB with power track

Conclusion

The noise voltage, the main cause of unwanted radiated electromagnetic fields, can be reduced by creating geometrically asymmetric transmission lines. This asymmetry should also be used for power supply systems. Radiated electric field measurements have been performed using actual circuits, showing a reduction of the radiated emission level without compromising the integrity of the functional signals..

References

[1] H.W. Ott, "Controlling EMI by Proper Printed Wiring Board Layout", EMC Zurich 1985, 6th Int. Zurich Symp. and Techn. Exh. on EMC, Zurich, Switzerland, pp. 127-132, 1985

[2] C.R. Paul, "Modeling Electromagnetic Properties of Printed Circuit Boards", IBM Journal Research and Development, vol, vol. 33, nr. 1, pp. 33-49, 1989

[3] F.B.J Leferink, M.J.C.M. van Doorn, "Inductance of Printed Circuit Board Ground Planes", IEEE Int. Symp. on EMC, Dalles, TX, pp. 327-329, 1993

[4] J.L. Drewniak, T.H. Hubing, T.P. VanDoren, "Investigation of Fundamental Mechanisms of Common-Mode Radiation from Printed Circuit Boards with Attached Cables", IEEE Int. Symp. on EMC, Chicago, IL, pp. 110-115, 1994

[5] D.M. Hockanson, J.L. Drewniak, T.H. Hubing, T.P. VanDoren, S. Fei, W.L. Cheung, "Quantifying EMI Resulting From Finite-Impedance Reference Planes", IEEE Trans. on EMC, vol. 39, nr. 4, Nov. 1997, pp. 286-97, 1997

[6] F.B.J. Leferink, "Preventing Electromagnetic Interference From Integrated Circuits and Printed Circuit Boards Using Computer Simulation", Report EL BSC 92N138, Twente University of Technology, 1992

[7] F.B.J. Leferink, "Reduction of Radiated Electromagnetic Fields by Creation of Geometrical Asymmetry, PhD. Thesis University of Twente, 2001, ISBN 90-365-1689-7

[8] B. Danker, "The Decoupling of DC Supply Lines", EMC'94 Roma, pp. 177-183

[9] M. Coenen, "Optimising IC Decoupling, for Performance and EMI Levels", *Electronic Product Design*, vol. 17, nr. 1, Jan 1995, pp. 26-34, 1995

[10] J. Fan, Y. Ren, J. Chen, D.M. Hockanson, H. Shi, J.L. Drewniak, T.H. Hubing, Th. P. van Doren, R.E. DuBroff, "RF Isolation Using Power Islands in DC Power Bus Design ", *IEEE Int. Symp. on EMC*, Seattle, WA, pp. 838-843, 1999

[11] Y. Fukumoto, S. Nakamura, O. Wada, R. Koga, "A Design Method of Decoupling Circuits for a Digital PCB to Reduce High Frequency Current on Power and Ground Planes", *EMC'99 Tokyo, Int. Symp. on EMC*, Tokyo, Japan, pp. 9-12, 1999

[12] J. Held, Th. Wolf, "Optimized Decoupling Concepts for Digital VLSI Circuits", *IEEE Int. Symp. on EMC*, Montreal, Canada, pp. 904-909, 2001

[13] T.H. Hubing, J.L. Drewniak, T.P. VanDoren, D.M. Hockanson, "Power Bus Decoupling on Multilayer Printed Circuit Boards", *IEEE Trans. on EMC*, vol. 37, nr. 2, May 1995, pp. 155-66, 1995

[14] T. Hubing, J. Chen, J. Drewniak, T. van Doren, Y. Ren, J. Fan, R. DuBroff, "Power Bus Noise Reduction Using Power Islands in Printed Circuit Board Designs", *EMC'99 Tokyo, Int. Symp. on EMC*, Tokyo, Japan, pp. 1-8, 1999

[15] L.P. Janssen, "Reducing the Emission of Multi-Layer PCBs by Removing the Supply Plane", *EMC Zurich 1999, 13th Int. Zurich Symp. and Techn. Exh. on EMC*, Zurich, Switzerland, pp. 639-644

[16] S. Radu, D. Hockansen, "An Investigation of PCB Radiated Emissions from Simultaneous Switching Noise", *IEEE Int. Symp. on EMC*, Seattle, WA, pp. 893-898, 1999

[17] H. Sasaki, T. Harada, T. Kuriyama, "A New Decoupling Technique for Suppressing Radiated Emissions Arising from Power Bus Resonance of Multilayer PCBs", *EMC'99 Tokyo, Int. Symp. on EMC*, Tokyo, Japan, pp. 17-20, 1999

[18] H. Sasaki, T. Harada, T. Kuriyama, "A New Decoupling Circuit for Suppressing Radiated Emissions due to Power Plane Resonance', *IEICE Trans. Comm.*, vol. E85-B, no. 5, May 2002, pp. 1031-1037

[19] Frank B.J. Leferink, Wim C. van Etten, Reduction of Radiated Electromagnetic Fields by Removing Power Planes, IEEE International Symposium on EMC, Santa Clara, 2004, pp. 226-230