EMC Design Guidelines for Electrical Architectures

Marcel van Doorn

Philips Innovation Services – EMC consultancy
High Tech Campus 26, 5656 AE Eindhoven, The Netherlands
marcel.van.doorn@philips.com

Abstract— **This paper presents new quantitative EMC (ElectroMagnetic Compatibility) design guidelines at board, cable and enclosure level to reduce radiated emission in the frequency range from 30 MHz to 6 GHz. These new design insights are the outcome of extensive 3D electromagnetic simulations on a general modelling framework for electrical architectures. They can be used during the early phases of system design when choices have to be made on printed circuit board technology, cable and connector technology, and shielding design.**

I. Introduction

Many EMC design guidelines have been published in books, technical papers and application notes. Electronic companies often have their own EMC design guides. Over the past 20 years the University of Missouri-Rolla and the Clemson University collected many EMC design guidelines [1]. As is indicated on their website a few guidelines are very important and can nearly always be applied. Others apply only to specific situations and are not appropriate in general. It is important to comply with those guidelines that have the highest impact in your particular application.

Some general design guidelines often referred to in literature are:

- Minimize the loop areas associated with high-frequency power and signal currents.
- Don't split, gap or cut the signal return plane.
- Control signal transition times.

These guidelines have a generic, qualitative character. This means that by applying all these guidelines there is no guarantee that your design will comply with the legal emission requirements. In this paper we go one step further. We want to develop quantitative design guidelines that are directly related to the maximum radiated field. The designable parameters that have the highest impact on the radiated emission performance will be quantified. When the signal characteristics of critical high-speed interfaces are known, this approach will help to predict the radiated emission performance during the early phases of the product creation process. During the architecture design phase the right technology choices can then be made. Further, we assume that the basic qualitative design rules from literature are known and applied by the design team, e.g. no high-speed signals crossing slots in return plane, matched transmission lines, etc.

In Section II a short introduction of the modelling framework is given. Sections III, IV, and V present design guidelines at printed circuit board, cable, and enclosure level respectively.

II. Modelling Framework

In Ref. [2] the general modelling framework for an electronic system has been described in detail. It consists of two printed circuit boards (PCBs) connected via a cable. The boards can be mounted on a metal plate and a long wire can be attached to the PCB ground plane (Fig. 1). This general modelling framework has been incorporated in a tool for 3D electromagnetic simulations CST MWS [3].

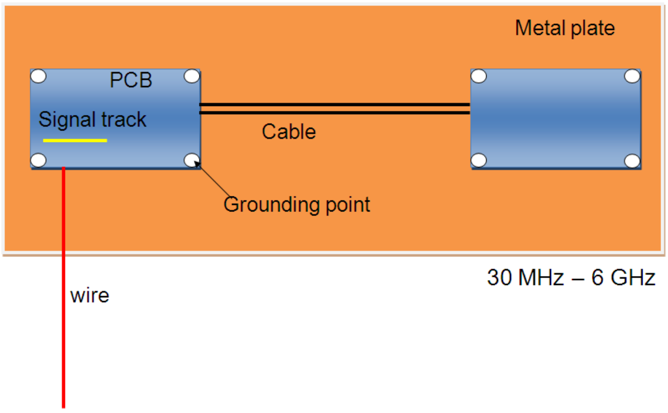

Fig. 1 Modelling framework

The model has been fully parameterized. This makes it possible to study the radiated emission of a PCB, cable, and shielding enclosure as a function of its designable parameters: PCB dimensions, track length, width, and height above the ground plane, track position on the board, cable length, type (co-planar, microstrip), and height above the metal plate, amount of grounding connections, length external ground wire, etc.

The functional signal source is a sine wave voltage source between 30 MHz and 6 GHz with an output impedance of 50 Ω (tracking generator). This voltage source is connected to a 50 Ω microstrip transmission line on the board or to a 50 Ω cable (co-planar or microstrip) between the boards. Both track and cable are terminated with 50 Ω. For all frequencies between 30 MHz and 6 GHz the voltage on the line is 0.5 V rms (line current is 10 mA).

Output of the simulations is the maximum radiated electric field at the surface of a sphere with radius 10 m for all frequencies between 30 MHz and 6 GHz.

The characteristics of the real functional signal can be compared to the constant reference source of 0.5 V. In this way a quick assessment of the radiated emission performance of the functional design can be made.

In the next sections we will present a few design guidelines at printed circuit board, cable and enclosure level that have been developed with the modelling framework. The new design insights have been verified by measurements on real demonstration boards. In this way confidence in the model and simulation results could be gained. The simulated results have proven their value in practice, particularly during the early phases of architecture design when choices on PCB technology, layer stack-up, placement of critical components and signals, and cable/connector technology have to be made.

III. Printed Circuit Board Design Guidelines

In Ref. [2] we investigated the impact of the height of a signal track above its return plane on the radiated emission performance. The simulations learned that the maximum radiated field is proportional with track height. In the next two sections we will investigate how the maximum radiated field is influenced by the distance of the track to the edge of the board, and by attaching a long wire to the ground plane of the board.

A. Edge distance

Starting point is a PCB with dimensions 160 × 100 mm. The signal track has a length of 100 mm, a width of 0.16 mm and is located 0.1 mm above the PCB ground plane (50 Ω microstrip transmission line). Fig. 2 shows the maximum radiated electric field strength at 10 m distance from the board for three different values of the distance of the track to the board edge d: 0 mm, 2 mm, and 10 mm. As a reference the CISPR 22 emission limit at 10 m distance is plotted in the figure [4]. This CISPR 22 radiated emission limit for information technology equipment corresponds with the new CISPR 32 emission limit for multimedia equipment [5].

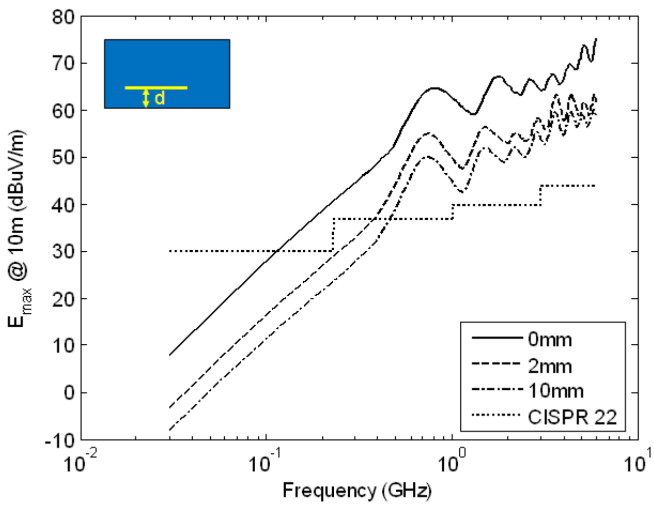

Fig. 2 Maximum radiated field of a track on a PCB for 3 different board edge distances

For low frequencies the radiation of the PCB track is proportional with f^2 (40 dB/decade). Above the first resonance frequency of the 100 mm track (around 750 MHz) the radiation is proportional with $f^{½}$ (10 dB/decade). In case of a digital pulse-train signal, instead of the constant voltage source, the voltage level on the line will decrease with f^{-1} after the first corner frequency ($f_1 = 1/\pi\tau$, with τ = pulse duration), and with f^{-2} after the second corner frequency ($f_2 = 1/\pi\tau_r$ = bandwidth, with τ_r = rise/fall time). This will bring the radiation of digital signals down as the frequencies go up (above bandwidth).

The radiated field of a track at 2 mm from the board edge (20h rule) is approximately 10 dB lower than that of a track at the board edge over the complete frequency range from 30 MHz to 6 GHz. Changing the edge distance from 2 to 10 mm (100h) gives an additional radiation reduction of 4 dB. Changing the edge distance from 0 to 50 mm (track in the middle of the board) gives a total radiation reduction of 22 dB (not plotted). This simulation result learns that the most critical signals should be placed in the middle of the board and not close to the edge.

B. Ground wire length

Fig. 3 shows the maximum radiated electric field of the PCB with matched transmission line (length = 100 mm, edge distance = 10 mm) for three different lengths of the ground wire attached to the ground plane of the board: 100 mm, 500 mm, and 1000 mm.

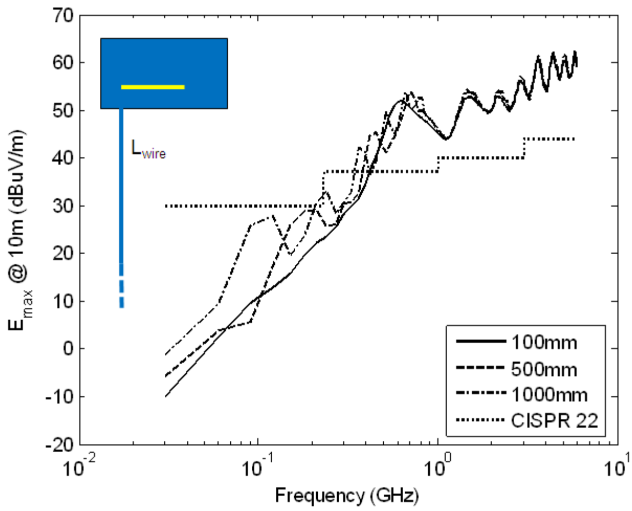

Fig. 3 Maximum radiated field of a PCB with 3 different ground wire lengths

If a long wire of 1000 mm is attached to the ground plane of the PCB wire resonances occur with a maximum radiated field that is 16 dB higher than that of the PCB stand alone. A wire of 1 m length with some top capacitance (PCB ground plane) has its half wavelength resonance frequency just below 150 MHz. The shorter the attached wire the higher the first resonance frequency. Above 1 GHz the attached wire gives no additional contribution to the maximum radiated field of the PCB itself anymore. The emission is than mainly determined by the resonant track on the board. The well known mitigation techniques of applying common-mode chokes around cables, or connecting the cable ground to a metal plate are not effective anymore above 1 GHz. The direct radiation of the board itself has to be reduced, for instance by placing the track in the middle of the board or by applying another layer stack-up with a shorter distance between signal track and return plane.

According the International Technology Roadmap for Semiconductors [6] the chip to board frequencies will increase and the rise/fall times will decrease. A rise time of 50 picoseconds corresponds with a signal bandwidth of 6 GHz. This will result in more and more harmonics of digital signals well above 1 GHz. Consequently board design has to get more attention to make it right first time.

IV. Cable Design Guidelines

In Ref. [2] we compared the radiation of a co-planar strip with a microstrip flex foil cable. Both cables had a length of 300 mm and a characteristic impedance of 50 Ω. The microstrip cable radiated 50 dB less than the co-planar strip cable over the complete frequency range from 30 MHz to 6 GHz. Now we will investigate how the radiated emission of a microstrip cable increases if a pigtail is involved.

A. Pigtail size

Fig. 4 shows the model of the 50 Ω microstrip flex foil cable with pigtail. It has a length of 300 mm and the ground plane of the cable is connected to the ground planes of the PCBs at both ends. The dimensions of the PCBs are 160 × 100 mm. The default parameters of the cable are:

h = height dielectric = 0.05 mm
ε_r = relative dielectric constant foil material = 3.1
W_s = width copper strip = 0.1 mm (= W_{pig})
W_g = width ground plane = width cable = 20 mm
L_{cable} = length cable = 300 mm
L_{pig} = length pigtail (varies)
d_{pig} = distance pigtail to strip = 9.95 mm

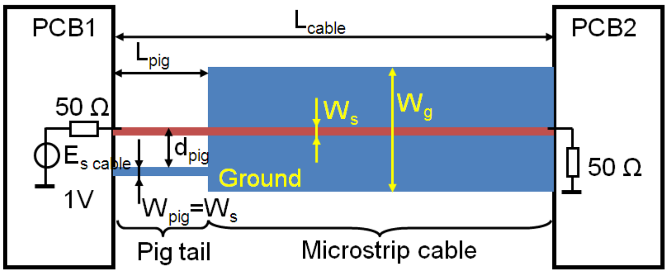

Fig. 4 Pigtail model microstrip cable

Fig. 5 shows the maximum radiated field of the microstrip cable with different pigtail lengths: 0 mm, 1 mm, and 10 mm. From the simulation results we learn that the shorter the pigtail the lower the radiated field. By reducing the pigtail size from 10 mm to 1 mm a radiation reduction of approximately 10 dB can be realized between 30 MHz and 1 GHz. By reducing the pigtail size from 1 mm to 0 mm, a radiation reduction of **30 dB** takes place over the frequency range from 30 MHz to 6 GHz. So, when selecting the right cable type for a high-speed interface, it is also extremely important to select the optimum cable connector, i.e. a connector that smoothly connects the cable ground plane to the board ground planes over its complete width (no pigtail; flex-rigid technology).

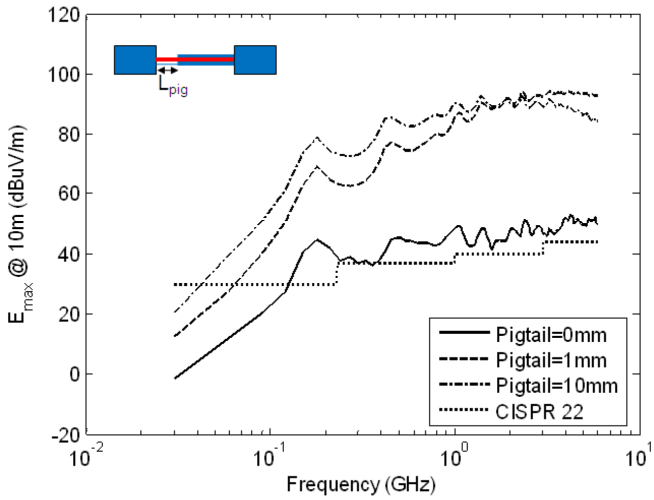

Fig. 5 Maximum radiated field of a microstrip cable with 3 different pigtail sizes

V. Enclosure Design Guidelines

In this section we will investigate the effect of a metal plate underneath a PCB on the radiated emission performance. The PCB with dimensions 160 × 100 mm can be placed at different heights above the metal plate with dimensions 500 × 500 mm. The transmission line on the PCB (length 100 mm, characteristic impedance 50 Ω) can be placed on the top or bottom side of the board.

A. Height above metal plate

Fig. 6 shows the maximum radiated electric field of the PCB above a metal plate at 10 m distance from the source for three different PCB heights h_{PCB}: 100 mm, 10 mm, and 1 mm. The signal track is located at the bottom of the board between the PCB ground plane and the metal plate. There are no grounding connections between the PCB ground plane and the metal plate. For frequencies below the first cavity resonance frequency of 0.9 GHz the height of the PCB above the metal plate has not much influence on the radiated emission performance. For high frequencies and a small height of the PCB above the metal plate (< 10 mm), strong cavity resonances dominate the radiated emission behaviour. The cavity resonances are determined by the dimensions of the PCB ground plane. At a PCB height of 1 mm the maximum radiated field is **24 dB** higher compared to a PCB height of 100 mm, and 8 dB higher compared to a PCB height of 10 mm. The strong cavity resonances were confirmed by practical field measurements on a PCB-plate test set-up. At a PCB height of 100 mm above the metal plate the maximum radiated field is almost equal to that of the PCB stand alone in free space.

When the signal is located at the top side of the board, the cavity resonances do not occur. Therefore it can be wise to place critical signals at the top side of the board instead of at the bottom side where resonances occur in the cavity formed by the PCB ground plane and metal plate [2].

In the next section we will investigate how the signal track at the bottom side of the board can be shielded by many grounding points between the PCB ground plane and the metal plate.

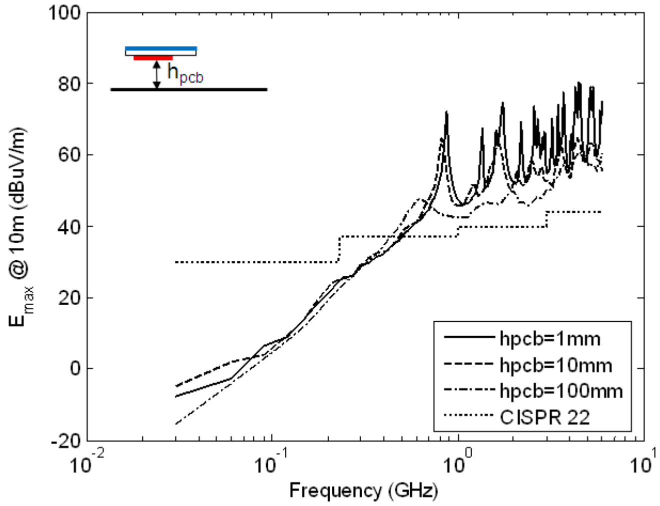

Fig. 6 Maximum radiated field for 3 different PCB heights above metal plate

B. *Number of grounding connections*

Fig. 7 shows the maximum radiation of the PCB with different numbers of grounding connections between the PCB ground plane and metal plate: 0, 4 (at the corners), and 52 (every 10 mm around the PCB circumference). The height of the PCB above the metal plate h_{PCB} = 1 mm. The signal track is located at the bottom side of the board between PCB ground plane and metal plate.

The more grounding connections the lower the radiated field. This is true until the distance between the grounding connections reaches a half wavelength. The cavity resonances determine the maximum radiated field. Below the resonance frequencies 4 grounding points shield 11 dB better than no grounding points, and 52 grounding points shield even 57 dB better.

In case of few grounding connections between PCB ground and nearby metal plate, critical high-speed signal tracks with harmonics above 1 GHz should preferably be located at the top side of or inside the PCB. In multilayer boards with components at the top side a ground plane could be placed at the bottom side of the board to prevent cavity resonances excited by signal tracks.

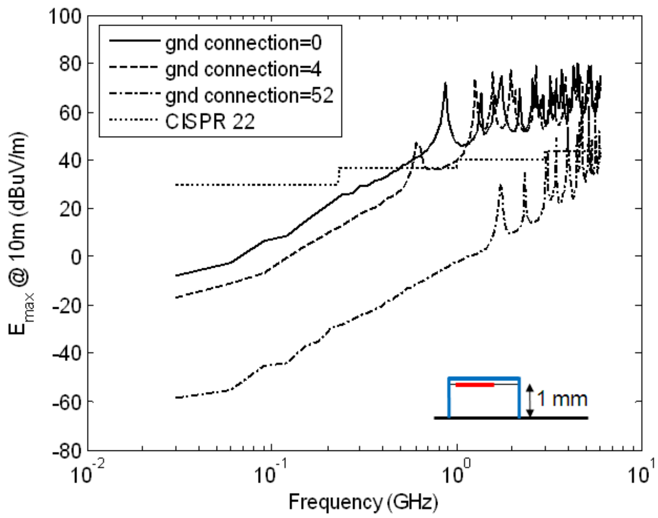

Fig. 7 Maximum radiated field for different number of ground connections to a metal plate

VI. Conclusions

New quantitative EMC design guidelines at board, cable, and enclosure level have been presented. They are based on 3D electromagnetic simulations on a general modeling framework for an electronic system and have a direct relation to the maximum radiated emission between 30 MHz and 6 GHz. With these guidelines EMC experts can make the right technology choices during the architecture phase of a product: PCB layer stack-up, cable/connector types, and shielding design. The new guidelines are verified by measurements on demonstration boards and have proven their value in practice.

In the future the modeling framework will be extended to study the impact of vias and plane resonances on radiated emission when critical signals cross power planes.

Acknowledgment

The author would like to thank Saoer Sinaga and Stef Worm from Philips Innovation Services for executing the modelling and simulation process with CST MICROWAVE STUDIO®.

References

[1] Todd Hubing, *EMC Design Guide Collection*, Learn EMC, http://www.learnemc.com/tutorials/guidelines.html

[2] M.J.C.M. van Doorn, *EMC Expert System for Architecture Design*, Proceedings of the 2011 Asia Pacific EMC Symposium, Jeju Island, Korea, May 2011.

[3] CST MICROWAVE STUDIO®, a specialist tool for 3D EM simulation, http://www.cst.com/.

[4] CISPR 22, *Information Technology Equipment – Radio disturbance characteristics – Limits and methods of measurement*, International Standard Edition 6.0, 2008-09.

[5] CISPR 32, *Electromagnetic compatibility of multimedia equipment – Emission requirements*, International Standard Edition 1.0, 2012-01.

[6] ITRS, *International Technology Roadmap for Semiconductors 2010*, http://www.itrs.net/.