HAL
open science

REDUCTION OF ELETROMAGNETIC PERTURBATIONS BY OPTIMIZING THE PRINTED CIRCUIT BOARD

Jad Taki, M Bensetti, Daniel Sadarnac

To cite this version:

Jad Taki, M Bensetti, Daniel Sadarnac. REDUCTION OF ELETROMAGNETIC PERTURBATIONS BY OPTIMIZING THE PRINTED CIRCUIT BOARD. XVII International Symposium on Electromagnetic Fields in Mechatronics, Electrical and Electronic Engineering (ISEF 2015), 10-12 Sep 2015, Valencia, Spain. ISBN 978-84-606-9102-0, Sep 2015, Valencia, Spain. hal-01215835

HAL Id: hal-01215835

https://centralesupelec.hal.science/hal-01215835v1

Submitted on 15 Oct 2015

HAL is a multi-disciplinary open access archive for the deposit and dissemination of scientific research documents, whether they are published or not. The documents may come from teaching and research institutions in France or abroad, or from public or private research centers.

L'archive ouverte pluridisciplinaire **HAL**, est destinée au dépôt et à la diffusion de documents scientifiques de niveau recherche, publiés ou non, émanant des établissements d'enseignement et de recherche français ou étrangers, des laboratoires publics ou privés.

ISEF 2015 - XVII International Symposium on Electromagnetic Fields
in Mechatronics, Electrical and Electronic Engineering
Valencia, Spain, September 10-12, 2015

REDUCTION OF ELETROMAGNETIC PERTURBATIONS BY OPTIMIZING THE PRINTED CIRCUIT BOARD

J. TAKI[1], M. BENSETTI[1], and D. SADARNAC[1]

[1] GeePs | Group of electrical engineering – Paris, UMR CNRS 8507, CentraleSupelec, Univ Paris-Sud, Sorbonne Universités, UPMC Univ Paris 06,
11, rue Joliot-Curie, Plateau de Moulon, F91192 Gif-sur-Yvette, France
jad.taki@centralesupelec.fr mohamed.bensetti@centralesupelec.fr daniel.sadarnac@centralesupelec.fr

Abstract ***–EMC(Electromagnetic Compatibility) studies require the modeling of the printed circuit board (PCB). Such models need to be compatible with the simulation software usually used in the industry. An analysis for modeling and optimizing the PCB is presented in this study. The method of moments is used to determine a transmission line electrical model (RLGC) for traces. This model is based on the physical geometry of traces and the material characteristics of the copper and the dielectric. Given the PCB model, an optimization can be performed to minimize the conduction emissions of the board.***

Introduction

Technological advances in power electronics tend to raise the switching frequency. As a result, the conducted emissions we measure at the battery may also be raised. These perturbations may interact with all components in a power electronics board. The source of these emissions includes active components, but PCB (Power Circuit Board) traces conduct these perturbations and tend to raise them at high frequencies.

EMC standards are becoming more restrictive, driving an increased interest in modeling the PCB and reducing its effect on emissions. These models should be fast to generate and compatible with electric circuit simulators suck as SPICE.

Previous studies have used a number of different methods to model the PCB at high frequencies. Some are based on simplified analytical equations [1][2] which sometimes neglect the mutual inductances and capacitances between traces. Others are based on the finite element method (FEM) which consumes a lot of hardware and time, and is not ideal for an optimization process.

The purpose of this study is first to optimize a PCB such that the electromagnetic perturbations are reduced. In order to do this, a PCB model is needed. An analytical method has been chosen to generate a transmission line model (RLCG) that is frequency-dependent.

This paper will introduce the method used to model the PCB, and give a general idea of the electrical models that we will obtain. After that, it will discuss how these models will be integrated in the electrical circuit simulator and the different types of simulations that we can use in order to determine the electromagnetic perturbations. The section that follows introduces the optimization procedure that is used in this work in order to reduce the conducted emissions. This method will be used on a DC-DC buck converter to validate the procedure.

Modeling

As the purpose of this work is optimizing the PCB, the selected model should be fast to generate and one which can be electrically simulated in SPICE tools. Fig.1 presents a transmission line model (RLCG) that will represent a PCB trace where *dz* is an elementary length.

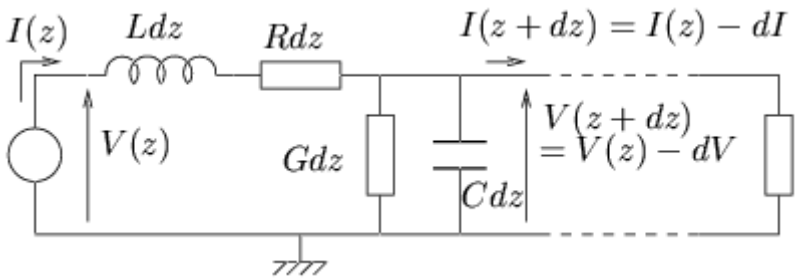

Fig. 1: Transmission line model

R represents the resistance of the trace per unit length which is frequency dependent due to the skin effect. *L* is the inductance per unit length of the loop formed by the trace and the ground plane. This inductance varies with frequency and depends on the distance between the trace and the ground plane. *C* is the capacitance per unit length between the trace and the ground plane. This element depends on the permittivity of the dielectric which can also be frequency-dependent.
G is the conductance per unit length of the trace which represents the dielectric losses. Usually this term is neglected compared to the capacitance of the trace.
The line transmission model presented in fig.1 is for a single trace on top on a ground plane. In a PCB circuit, traces will interact with each other's and will create a capacitance and a mutual inductance between them.

Resistance

The resistance per unit length of the trace has been determined using analytical equations [3]. It is divided into two sections, the resistance per unit length of the trace on low frequencies and the resistance per unit length on high frequencies.

$$R = \frac{\rho}{S}(1 + \sqrt{\frac{f}{f_{skin}}}) \tag{1}$$

where ρ is the copper resistivity, S the area of a section of the trace, f is the frequency and f_{skin} is the frequency at which the skin depth is of the same order of the trace thickness. This frequency can be determined using (2) :

$$f_{skin} = \frac{4\rho}{\pi\mu_0 t^2} \tag{2}$$

where μ_0 is the magnetic permeability of the air and t the thickness of the trace. Using (1) and (2), we can get the frequency dependence of the trace resistance. Since we mentioned that the model should be SPICE-compatible, the resistance should be presented as a function of frequency too in SPICE.
There are two ways in order to model this effect. Either the frequency is presented using the s parameter where the relation between s and the frequency is given in (3):

$$s = j * 2\pi f \tag{3}$$

or the resistance will be modeled by a lumped parameter circuit that will take into consideration the frequency-dependence as shown in fig.2.

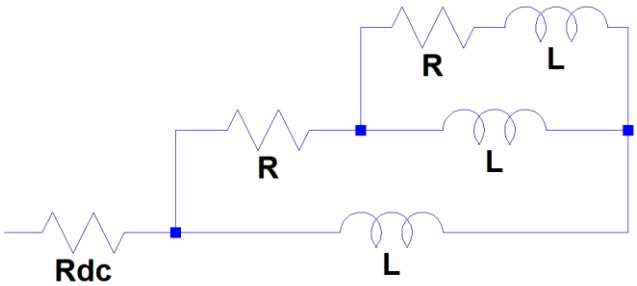

Fig. 2: Lumped parameter model of a frequency-dependent resistance

R_{dc} is resistance of the trace on low frequencies. As for the rest of the elements, it represents the variation of the resistance as a function of the frequency. The number of branches used depends on the precision needed in order to fit the model to the analytical calculation.

Inductance and Capacitance

A PCB circuit creates some inductances and capacitances in the system:

- Proper inductance of the trace which represents the inductance of the loop formed by the trace and the ground plane.
- Proper capacitance of the trace which represents the capacitance between the trace and the ground plane.
- Mutual inductances between traces on the same plane.
- Capacitances between traces on the same plane.

In order to determine all these inductances and capacitances, the Method of Moments [4][5][6] has been used. It is a numerical computational method of solving linear partial differential equations which have been formulated as integral equations:

$$\emptyset(x,y) = \frac{1}{4\pi\varepsilon}\iint_{s}\frac{\rho(x',y')}{r}ds \tag{4}$$

(4) is the integral form of Poisson's equation in the transverse x-y plane where a charge distribution in the x-y plane, ρ, is distributed over some surface s. We know how to determine the potentials at every point in the region and we wish to determine the charge distribution that follows it since the capacitance is related to the potential as well as the charge distribution as we can see in (5) where Q is the charge distribution and V is the differential potential.

$$Q = C * V \tag{5}$$

In order to determine the capacitance of the system, (4) and (5) are used with relative permittivity of the dielectric between the traces and the ground plane. As for the inductances, (6) gives a relation between the inductance L, the permittivity of the air ε_0, the electromagnetic permeability of the air μ_0 and the capacitance of the traces if we replace the dielectric in the system by air C_0:

$$LC_0 = \mu_0\varepsilon_0 \tag{6}$$

If the Method of Moment is used to determine the capacitances in the air of the system, (6) will give us all the inductances needed to have a proper PCB model. An important note to mention is that the Method of Moments gives us inductances and capacitances per unit length and in order to have a model that is valid and precise on high frequencies, it is important to choose the length of each section of the trace and so then number of sections of the trace.

Optimization Procedure

In order to reduce the electromagnetic perturbations, it is important to generate a PCB that does not increase these perturbations. To apply this procedure to a power electronics circuit, it is necessary to have models of every element. Passive components are modeled by lumped circuits that are valid at high frequencies using measurement from a network analyzer. After placing the components, the routing must be developed. Initially, the routing of the PCB must follow some general rules such as maintaining a distance between the traces and the edges of the board, separating the power section from the command and avoid right angles.
In the optimization procedure, it is considered at first that any routing between two pins of a component is composed of two traces as shown in fig. 3.

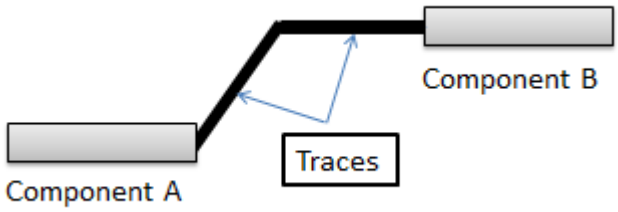

Fig. 3: Routing technique

By changing the position of the intersection of the two traces, the routing changes and thus its electromagnetic effect on the conductive perturbations. This would be the main variable to optimize in the procedure. The optimization algorithm used in this work is the genetic algorithm [7]. This algorithm should have three parts:

- Selecting the right generation of variables
- Producing the PCB model for the chosen individual at the current iteration
- The objective function in order to find the best individual that responds to the problem which is reducing the electromagnetics conductive emisssions.

For the selection, the algortihm should at first a fixed number of individuals forming a generation. The choice of the individuals (its values) should follow the limitations giving in the description of the algorithm. For example, a limitation can be the intervals that will limit the choice of the x and y position of the intersection point. In the first iteration, the algorithm will calculate the objective function for each individual of the generation and at the end of the iteration it will choose the best individual (the lowest objective function) with other mutated ones to participate in the next iteration. The algorithm will continue throught iterations until it attends a certain precision or obtain the same lowest objective function for several iterations consecutivally. As a result, the algorithm will give the best individual as well as the value of the objective function corresponding to this individual.
For each selected individual, a PCB model should be produced. The board will be meshed and the routine will begin using the technique mentioned before with only one intersection between 2 pins. After the routing is finished, the program start searching for traces that could group to form one model using the Method of Moments. The traces that the program searches for are traces who are in the same colomn and have the same angle between each others as shown in fig. 4.

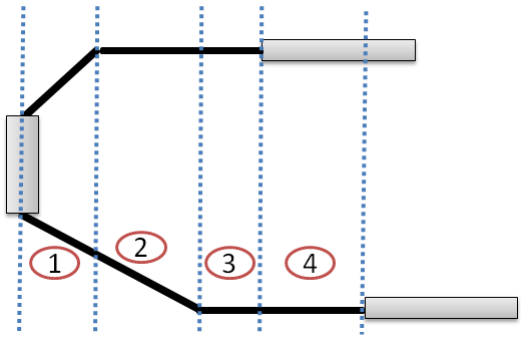

Fig. 4: Traces Separation for modeling

We can notice that there is four different regions in this example. Each region have traces with same angle between them and consist a unique PCB model. These models will be linked to either component pins or other parts of traces.
As mentioned before, the objective function is to find the best individual that reduces the electromagnetics conductive emisssions. The function chosen to get the best result is a relative comparison between the sum of the spectrum of perturbations over frequency between the case for the actual indivual and the one for the current best indivudual. If the comparaison is negative, the current individual will be the current best individual. The algorithm will keep going all over the avaible individuals until the best one is found.

Study case: A DC/DC converter

The optimization technique is being validated in this section on a DC/DC converter as shown in fig. 5.

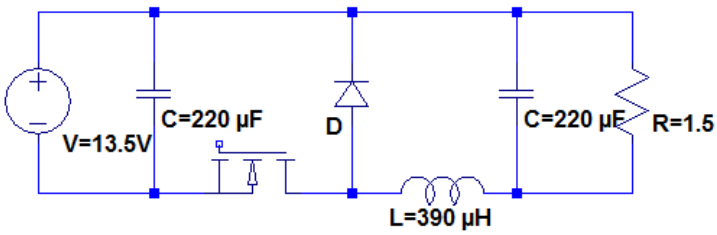

Fig. 5: DC/DC converter

Each component in this converter has been modeled using a Vector Network Analyser (VNA) and a fitting technique to be efficient up to 1 GHz and the switching frequency is 200 KHz. The PCB optimization procedure is used to get the most efficient routing regarding the EMC conducted emissions. These perturbations are being measured by two LISN (Line Impedance Stabilization Network) that are placed between the battery and the decoupling capacitor. Fig. 6 shows an electrical model of the LISN used in this study. Fig. 7 shows a geometry comparaison between a random PCB and an optimized one created by the algorithm described in this article. The board has been meshed into 50 units through x and y axis with 5 mm distance between 2 units. It is clear that the more the number of meshing units is higher, the more choices of routing there is and a better optimized one. Components of the converter are presented by blue lines (excluding the charge) and the traces between them are in red.

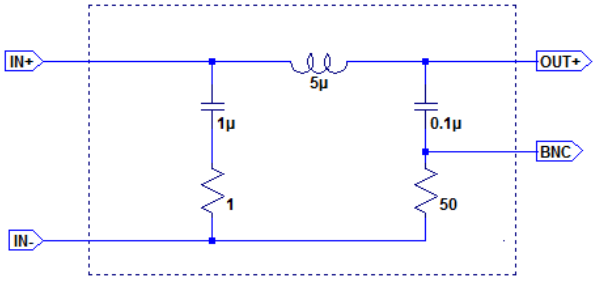

Fig. 6: LISN electrical model

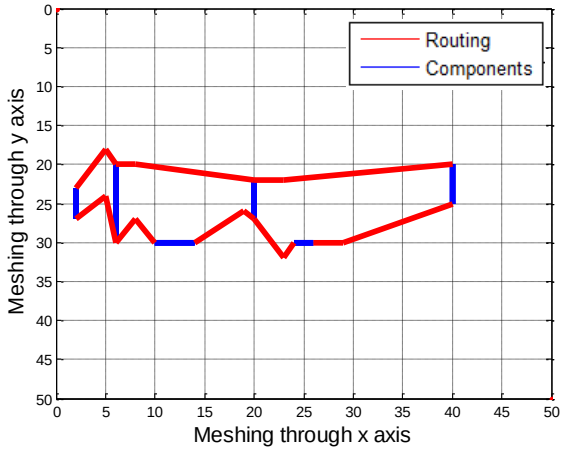

Fig. 7.a: Random PCB routing for the converter

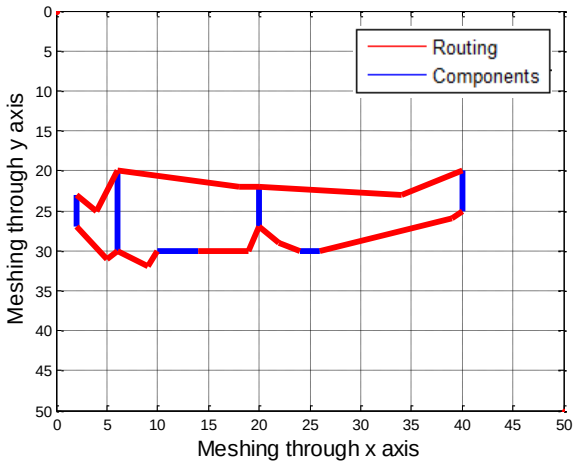

Fig. 7.b: Optimized routing for the converter

As we can notice, both routing are not regular and have no repetitive rules and it is clear that all right angles have been avoided even during the optimization results. The association of all models in the converter has been simulated in LTSpice [8] throught a time domain simulation where the results have been sent to Matlab in order to generate the perturbation spectrum throught the fast fourier transformation (FFT).

Spectrum comparison

Two types of results will be shown in this section. Fig. 8 compares the conductive emissions spectrum between a converter with or without PCB and Fig. 9 compares the spectrum between the converter with the random PCB shown in Fig. 7.a and the optimized one in Fig .7.b.

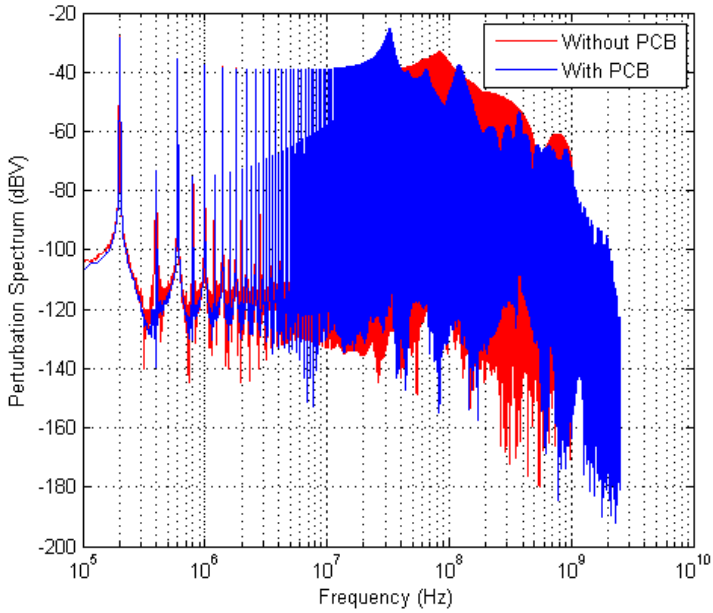

Fig. 8: Comparaison between the spectrums of the converter with or without the PCB

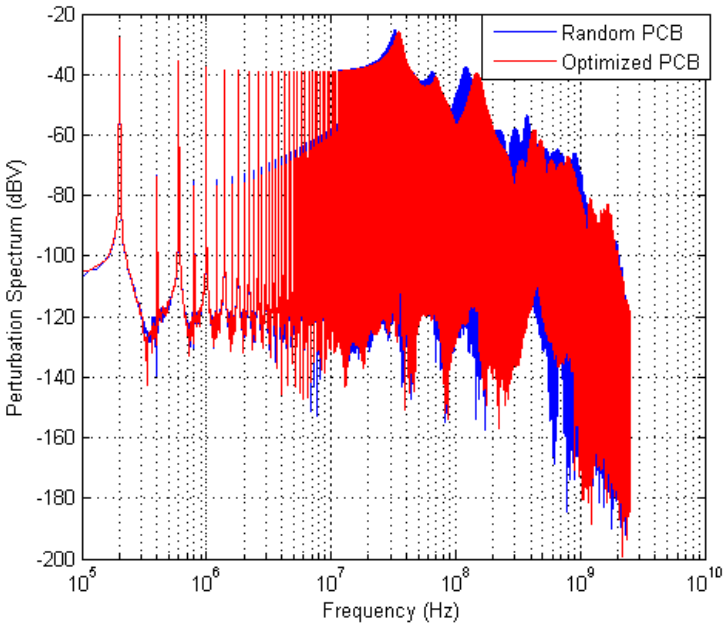

Fig. 9: Comparaison between the spectrums of the converter with a random and optimized PCB

It is noticeable in Fig. 8 that the PCB starts taking effect around 30 MHz and that it is important to take this fact into consideration in the simulation if the prediction of the conductive emissions above this frequency is needed. Fig. 9 and 10 shows that an optimized PCB can push some resonances to higher frequencies as well as reducing the amplitude of these resonances. A difference of more than 10 dB appears around 300 MHz and few dB between 50 MHz and 100 MHz.

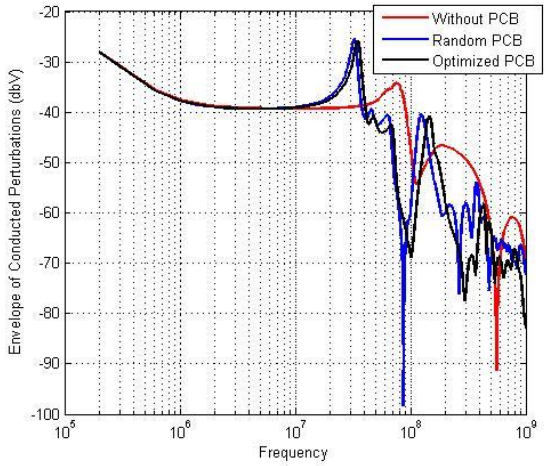

Figure 10: Envelope of the Spectrums for the converter without and with a random and optimized PCB

Conclusion and Perspective

In this study, a PCB optimization procedure has been introduced in order to reduce the electromagnetic conductive emissions. Each link between two pins has been divided into two traces and the optimization algorithm chooses the best positions for these intersections in order to satisfy the objective function. This study had the components placement as a fix variable and the only optimization was on the traces. The results shown for a DC/DC converter study case proves that optimizing the PCB reduces the perturbations on high frequencies.

In order to improve the quality of the circuit from an EMC and PCB view, multiples ameliorations can be introduced. The number of traces per link can be increase and so the number of entries to the optimization algorithm will increase. The meshing is another factor that plays a big role in the precision and choices of traces. As mentioned, the components' place are fixed in this study so adding this factor as an entry to the algorithm can improve the EMC quality of the board but can also make the study more complex and takes more time.

References

[1] S.Y. Poh, W.C. Chew and J.A. Kong, Approximate Formulas for Line Capacitance and Characteristic Impedance of Microstrip Lines, IEEE Transactions on Microwave Theory and Techniques, vol. 29, issue 2, pp. 135-142, 1981

[2] M.N.O. Sadiku, S.M. Musa, Comparison of Approximate Formulas for the Capacitance of Microstrip Line, in IEEE SoutheastCon Proceedings, pp. 427-432, 2007

[3] F. Lafon, Développement de techniques et de méthodologies pour la prise en compte des contraintes CEM dans la conception d'équipements du domaine automobile. Etude de l'immunité, du composant à l'équipement, PhD at INSA Rennes, France

[4] R.P. Clayton, Analysis of Multiconductor Transmission Lines, Manhattan: Wiley, 2008, pp 190-219

[5] R.F. Harrington, Field Computation by Moment Methods, New York: R. F. Harrington, 1968, pp 22-28

[6] N.Tidjani, J.C. Le Bunetel, A. Ouchar and Y. Raingeaud, Electromagnetic Coupling Modeling of Microstrip Lines Used in Switching Power Supplies, Journal of Control Science and Engineering, Vol.1, pp 55-59, 2013

[7] M. Mitchell, An Introduction to Genetic Algorithms, Cambridge, Massachusetts - London, England, 1999

[8] http://www.linear.com/