Application of Spice simulator in the evaluation of crosstalk computational models

André Tosin, Willian Gerlach Dietz, Carlos Antonio França Sartori
and José Roberto Cardoso

*Departamento de Engenharia Elétrica PEA/EPUSP, Escola Politécnica da
Universidade de São Paulo, São Paulo, Brazil*

Abstract

Purpose – This paper aims to present an evaluation of the crosstalk models developed for a shareware version of Spice.

Design/methodology/approach – The transmission line model is compared with π lumped circuit models by varying the number of cells. The influence of losses is also introduced, and some constraints regarding the models are examined, showing the suitability of the new Spice libraries developed by the authors. In order to validate the theoretical results some experiments were carried out.

Findings – The paper finds that Spice is a powerful engineering tool for students and technicians involved in EMC analyses.

Originality/value – The main aspects concerning the accuracy and integrity of the experimental data are emphasized and discussed in this paper.

Keywords Electromagnetism, Compatibility methods, Simulation

Paper type Research paper

1. Introduction

In general, Crosstalk is an unintended electromagnetic coupling among conductors. As an example of electromagnetic coupling, it can be mentioned the coupling among transmission line conductors, printed circuit board tracks, coaxial cables, etc. Recently, with the constant increasing in the frequency range of electronic systems, crosstalk has become a major problem, which must be evaluated carefully. For instance, the crosstalk between two cables can induce signals on peripheral ones that might radiate to the external environment causing fails to adjacent systems; consequently, causing the latter to be out of compliance with radiated emission regulatory requirements. Thus, the prediction of eventual problems can represent a huge economy of investments, and solutions can be applied decreasing the electromagnetic emissions, and improving the immunity of the systems. The electromagnetic coupling in multiconductor systems has been a research topic since the turn of the century (Mie, 1900). Originally, signal transmission through telephone and telegraph lines (Carson and Hoyt, 1927; Rice, 1941). In the 1960s were presented several methods where the transmission line models were used to evaluate electromagnetic coupling (Bergeron, 1961; Branin, 1967; Dommel, 1969). However, it was in the 1970s and 1980s that the research about this subject flourished. For instance, at those decades, many authors presented methods for constructing

ISEF 2005 – XII International Symposium on Electromagnetic Fields, in Mechatronics, Electrical and Electronic Engineering, Baiona, Spain, 15-17 September 2005.

solutions to multiconductor transmission line equations in homogeneous and non-homogeneous media using matrix theory to simplify the analysis (Chang, 1970; Paul, 1973, 1975). In 1988, for instance, Paul presented a model for coupled transmission line using Spice (Paul, 1988, 1992). It will be used in this paper with a graphical library cell that was implemented in the Spice (Dietz *et al.*, 2004). In this paper, although, for simplification, it is presented the transmission line model developed for a simple three-conductor configuration, it shall be mentioned that its results can be extended to n-conductor configurations. The shareware version of Spice was used to implement this model, and the transmission line model was compared with the well-known π lumped circuit model; firstly, using one section and later several sections. Furthermore, some frequency-independent losses were taken into account by a model that uses resistors positioned at specific locations of the circuit. Some experiments using three-conductor flat cable transmission lines were carried out to validate the models here presented. The main aspects concerning the accuracy and integrity of the experimental data are discussed, and the constraints are emphasized in the paper.

2. Methodology

Basically, the objective in a crosstalk analysis is to determine the near-end and far-end voltages, $V_{NE}(t) = V_R(0, t)$ $e V_{FE}(t) = V_R(L, t)$, given the line cross-sectional dimensions, and termination characteristics $V_S(t), Z_S, Z_L, Z_{NE}, Z_{FE}$, like shown in Figure 1.

2.1 General formulation

In this item it is presented the main aspects concerning the formulation of the transmission line crosstalk model. Assuming a lossless homogeneous transmission line, we have:

$$\frac{\partial}{\partial z} V(z, t) = -L \frac{\partial}{\partial t} I(z, t) \tag{1}$$

$$\frac{\partial}{\partial z} I(z, t) = -C \frac{\partial}{\partial t} V(z, t) \tag{2}$$

L and C are the inductances and capacitances matrices ($n \times n$), V and I are the voltage and current vectors ($n \times 1$), when it is considered a $n + 1$-conductor configuration.
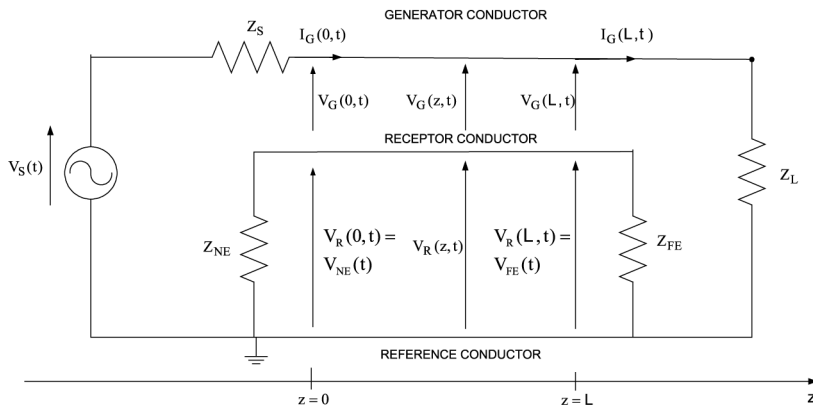

Figure 1.
Illustration of a
three-conductor
transmission line crosstalk
model

The sub-indexes G, R and M corresponding to generator, receptor and mutual characteristics of the variables and electrical parameters are usually applied in the aforementioned equations. The definitions, and the complete relationship among them can be founded in Paul (1992). The key to solving these equations in the time domain is to reduce the coupled pairs of line to a set of non-interactive two-conductor lines. In order to do this, transformations can be defined that converts the line voltages and currents to the so-called mode voltages V_m and currents I_m, as follow:

$$\mathbf{V}(z, t) = \mathbf{T}_V \mathbf{V}_m(z, t) \tag{3}$$

$$\mathbf{I}(z, t) = \mathbf{T}_I \mathbf{I}_m(z, t) \tag{4}$$

Thus, applying these transformations in equations (1) and (2), it will result:

$$\frac{\partial \mathbf{V}_m(z, t)}{\partial z} = -\mathbf{T}_V^{-1} \mathbf{L} \mathbf{T}_I \frac{\partial \mathbf{I}_m(z, t)}{\partial t} \tag{5}$$

$$\frac{\partial \mathbf{I}_m(z, t)}{\partial z} = -\mathbf{T}_I^{-1} \mathbf{C} \mathbf{T}_V \frac{\partial \mathbf{V}_m(z, t)}{\partial t} \tag{6}$$

It should be mentioned that the Spice simulator has in its library a model that gives an exact solution for lossless two-conductor lines. Hence, the Spice model can be used to solve the set of two-conductor transmission line mode voltages and currents, along the line. The mode currents and voltages at the endpoints of the line, $z = 0$ and $z = L$, can be converted to the actual line currents and voltages by implementing the transformations given by equations (3) and (4), using controlled source models. As an example, equation (3) can be rewritten as:

$$v_G = T_{VGG} V_{mG} + T_{VGR} V_{mR} \tag{7}$$

$$v_R = T_{VRG} V_{mG} + T_{VRR} V_{mR} \tag{8}$$

For simplification, the details of the definitions and relationship among the electrical parameters are not shown here: it has already been developed by other authors, and presented in the actual literature (Paul, 1992).

2.2 Spice implementation
As it was previously described, to get the actual voltages and currents at the endpoints of the line, it is necessary to implement transformations that convert the "mode" domain to the "actual" domain, using controlled sources. Nevertheless, it must be mentioned that the shareware version of Spice library, have only the simple models that are not suitable for the evaluation of multiconductor transmission lines. As an example, it can be mentioned the cell Epoly, for voltages, and Fpoly, for currents, available at Spice library: they show limitations related to the number of entries, i.e. the control variables. Thus, in order to minimize these constraints, new library cells were developed and added to the graphical library of Spice. Figure 2 shows an example of crosstalk implementation model for a three-conductor configuration concerning the functional cell Wpoly that has two entries $V1$ and $V2$. These entries are multiplied by the coefficients given by the transformation matrix T_V, and added. It should be mentioned that, in the case of a lossless line and homogeneous medium, $T_V = T_I = T$.
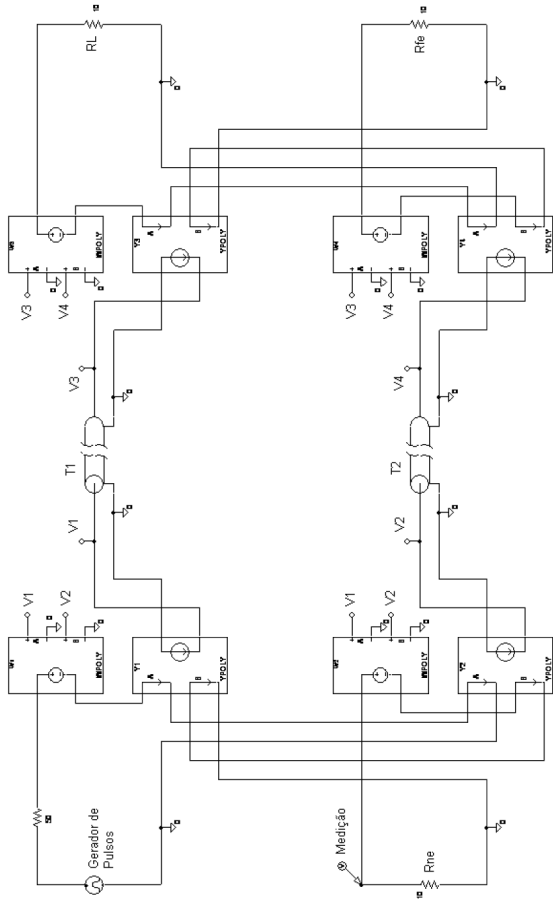
Historically, the first line models were cascade connections of Π-circuits: in fact, for lines of "moderate" electrical length (typically $L < \lambda/10$), nominal Π-circuits are often accurate enough, and they are probably the best models to use for steady-state solutions. If the steady-state solution is followed by a transient simulation, or if they are requested over a wide frequency range, then the nominal Π-circuit must be either replaced by a cascade connection of shorter nominal Π-circuits, derived from the distributed parameters, or by the transmission line model Dommel, 1996. Sometimes, less accurate representation are used to match the model to the one used in the transient simulation (e.g. lumping R concentrated in some places, rather than distributing it evenly along the line, or using approximate real transformation matrices, instead of exact complex matrices). As a brief comment, it should be mentioned that some models regarding losses, like the one developed by Dommel, assumes such kind of simplifications. In these model accurate results can be obtained by lumping $R = R'L$, in a few places, as long as $R < Z_{\text{surge}}$: For instance $R/2$ is lumped in the middle and $R/4$ at both ends of an otherwise lossless line Dommel, 1996.
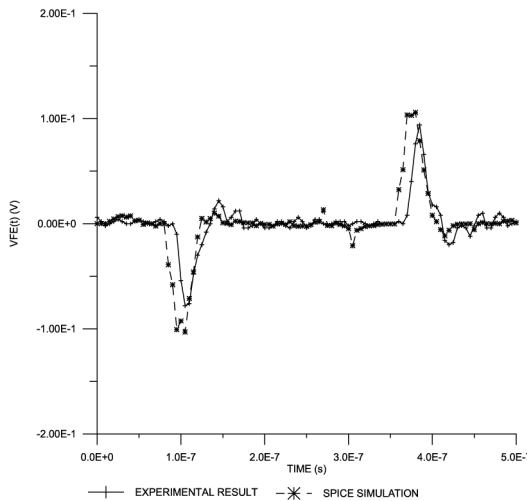
3. Results

In order to illustrate and to validate the proposed methodology, some experiments were carried out. Basically, it was used a three-conductor configuration with resistive loads. In fact, a three-conductor flat cable was used, which length is equal to 1.5 m. The central conductor was grounded, and adopted as the reference. In order to avoid external interference, the experiments were carried out inside a Faraday cage. The experiment set-up or the configuration that is exemplified can be better explained assuming the configuration given by Figure 1, and renaming Z_S, Z_L; Z_{NE}, and Z_{FE} by $R_G(0)$, $R_G(L)$, $R_R(0)$, and $R_G(L)$, respectively. The values between parentheses are related to the position on the line, 0 or L. It should be mentioned that, although many configurations were simulated, taking into account the transmission line model, the lumped circuit models by varying the number of cells, as well as losses, the results here presented were chosen to be considered canonical one and a suitable example for analysis of the phenomenon.

First, for a "load configuration" given by $R_G(0) = 0$ and $R_R(0) = R_R(L) = R_G(L) = 100\,\Omega$ the source or the pulse generator was adjusted with amplitude 2.25 V; the pulse-width to 300 ns, and the rise time and the fall time to 30 ns (Figure 3(a)). Secondly, the amplitude was changed to 3 V, and the loads to $R_R(0) = R_R(L) = R_G(L) = 10\,\text{K}\Omega$ and $R_G(0) = 0$ (Figure 4(a)). In order to use the signal of the generator in the simulations; the data were acquired as "txt" file using the software available by the oscilloscope. These were implemented as input files to the VPWL_FILE at Spice, which simulates the input pulse waveshape by linear piecewise forms. The voltages $V_{FE}(t)$ for both configurations were, then, obtained (Figures 3(b) and 4(b)). It should be mentioned that the per-unit-length parameters for the configuration were calculated using the methodology described in Paul (1992). These resulting values are: $l_G = l_R = 0.7485\,\mu\text{H/m}$, $l_M = 0.2408\,\mu\text{H/m}$, $c_G = c_R = 24.982\,\text{pF/m}$ e $c_M = 6.2661\,\text{pF/m}$. Moreover, the "Transformation Matrices" T_V and T_I, which for a lossless line and homogeneous medium are equals, were obtained by diagonalizing the **CL** or **LC** matrix. The so-called "characteristic impedance" and the velocity of propagation, assuming a lossless line in homogeneous medium, and the delay time $T_D = 5\,\text{ns}$, were calculated. The calculating values Z_{CmG}, Z_{CmR}, and $V_{mG} = V_{mR}$ are 296.63 Ω; 152.19 Ω; and 3×10^8, respectively.
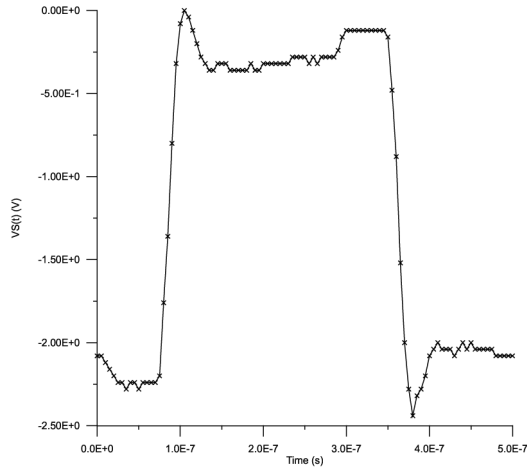
(a) The 2.25V Input pulse V_s (t)

┼ EXPERIMENTAL RESULT – ✳ – SPICE SIMULATION

(b) The values of V_{FE} (t) (measured and
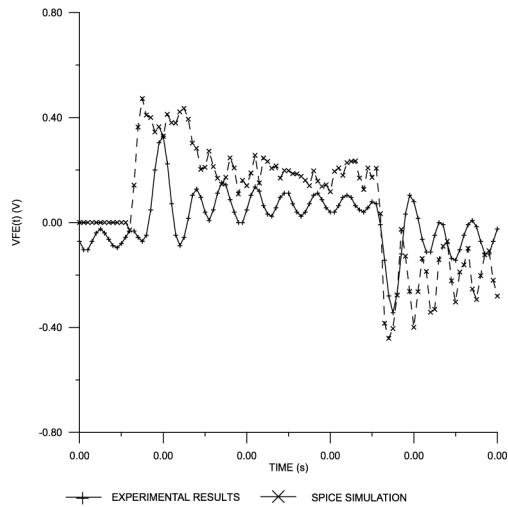simulated) for V_s (t) given by Fig. 3.a.

Figure 3.

4. Discussions

Based on the results presented previously, some comments can be made helping to
understand the differences between them. The first one is concerning an "inversion
of the phase" of the voltage V_{FE} that can be realized in Figure 3(b), (t). In order to
explain this, a simplification have to be assumed: supposing a transmission line
electrically short, i.e. $L < \lambda/10$, weakly coupled and subject to low frequency
signals, the far-end voltages ($\xi = FE$) can be calculated by the following
expression Paul (1992):

(a) The 3V Input pulse V_s (t)

EXPERIMENTAL RESULTS   SPICE SIMULATION
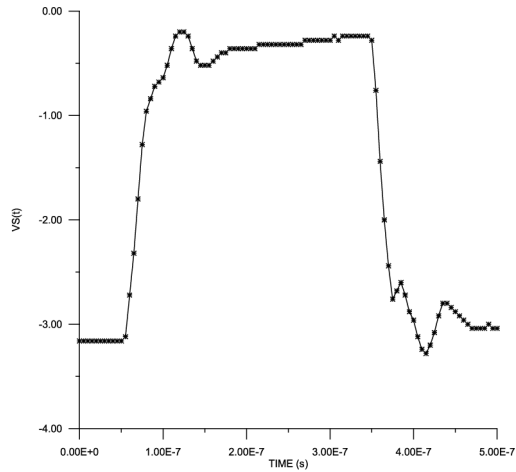
(b) The values of V_{FE} (t) (measured and
simulated) for V_s (t) given by Fig. 4.a.

Figure 4.

$$V_\xi(t) = K_\xi \frac{\mathrm{d}V_S(t)}{\mathrm{d}t} \tag{9a}$$

where

$$K_\xi = -\left(\frac{Z_\xi}{Z_{NE} + Z_\xi}\right)\frac{l_M L}{Z_S + Z_L} + \left(\frac{Z_{NE}Z_\xi}{Z_{NE} + Z_\xi}\right)\frac{Z_L c_M L}{Z_S + Z_L} \tag{9b}$$

The near-end can be evaluated ($\xi = NE$), using a expression similar to the previous one where the first parcel of the formulation is positive.

Thus, observing equation (9b), it can be noticed that K_{FE} will be negative when an inductive coupling dominates, that is the case shown in Figure 3, hence, the inversion in the $V_{FE}(t)$ voltage. On the other hand, in Figure 4 there is a higher impedance as load, and the capacitive coupling dominates, resulting a positive value for $V_{FE}(t)$. Regarding the data acquisition step, that is the one where the input pulse curves are captured by the oscilloscope and converted to txt files, it should be emphasized that the model is susceptible to fast variations in the pulse train, and to minimize the eventual noises during the input data acquisition, it is used the oscilloscope function average. Moreover, based on the fact that the oscilloscope only stores a fixed number of points for the curves shown in the display, in order to have a higher accuracy in the stored input data, it was used just one pulse period.

Although the results presented in Figures 3 and 4 show a good agreement between the experimental and simulated results, it can be observed, in both of them, a 20 ns delay between the experimental and simulated data: this delay occurred mainly because of the cable length that interconnect the line with the equipment located out of the Faraday's cage. In these experiments, although these cables had been positioned distant enough to avoid stray couplings, their lengths are about the line length, causing the observed delay.

Regarding the amplitude differences between the measured and simulated signals, they can be explained by the fact that the line losses and interconnection cable impedances were not taken into consideration in this analysis. It can also be mentioned that in Figure 4(b) there is a greater oscillation in the measured and simulated signals that can be explained by the use of a high impedance load (10 KΩ), and the resulting a reflection coefficient Γ_L that will contribute to a greater reflected voltage peaks.

5. Conclusions
The development and implementation of the crosstalk models for a shareware version of Spice has confirmed how powerful this engineering tool can be for students and technicians involved in the EMC analyzes. That is, based in the methodology here presented, the different Spice version constraints regarding the number of components and coupling among them can be minimized. The results related to a "canonical" configuration were presented as an example to illustrate the application, and the suitability of the new Spice libraries developed by the authors. The validation of the theoretical results were carried out based on the experimental ones, and a discussion of the main aspects concerning the accuracy and integrity of the experimental data has been emphasized and discussed in detail in this paper.

References

Bergeron, L. (1961), *Water Hammer in Hydraulics and Wave Surges in Electricity*, ASME Committee, Wiley, New York, NY.

Branin, F.H. Jr (1967), "Transient analysis of lossless transmission lines", *Proceedings Letters from Proceedings of the IEEE, July*, revised in September.

Carson, J.R. and Hoyt, R.S. (1927), "Propagation of periodic currents over a system of parallel wires", *Bell Syst. Tech. J.*, Vol. 6, pp. 495-545.

Chang, F-Y. (1970), "Transient analysis of losslell coupled transmission lines in a non-homogeneous dielectric medium", *IEEE Transaction on Microwave Theory and Techniques*, Vol. MTT-18 No. 9, pp. 616-26.

Dietz, W.G., Tosin, A. and Sartori, C.A.F. (2004), "Desenvolvimento de Modelos para Análise de Crosstalk utilizando-se o aplicativo Spice", MOMAG 2004 – (11° SBMO – Simpósio Brasileiro de Microondas e Optoeletrônica e o 6° CBMag – Congresso Brasileiro de Eletromagnetismo), August, São Paulo, Brazil (in Portuguese).

Dommel, H.W. (1969), "Digital computer solution of electromagnetic transients in single- and multiphase networks", *IEEE Transactions on Power Apparatus and Systems*, Vol. pas-88 No. 4.

Dommel, H.W. (1996), *EMTP Theory Book*, 2nd ed., Chapter 4, Microtran Powersystem analysis Corporation, pp. 4-1 to 4-108.

Mie, G. (1900), "Elektrisch Wellen an zwei parallelen Drahten", *Ann. Phys.*, Vol. 2, pp. 201-49.

Paul, C.R. (1973), "On uniform multimode transmission lines", *IEEE Transactions on Microwave and Techniques*, Short Papers, pp. 556-8, August.

Paul, C.R. (1975), "Useful matrix chain parameter identities for the analysis of multiconductor transmission lines", *IEEE Transactions on Microwave Theory and Techniques*, September, pp. 756-60.

Paul, C.R. (1988), "A simple Spice model for coupled transmission lines", *IEEE Transactions on Electromagnetic Compatibility*, Symposia Records 1955 to 1995.

Paul, C.R. (1992), *Introduction to Electromagnetic Compatibility*, Wiley, New York, NY, Chapter 4, pp. 120-76: "Transmission Lines": "Crosstalk" and Chapter 10, pp. 491-631: .

Rice, S.O. (1941), "Steady-state solutions of transmission line equations", *Bell Syst. Tech. J*, Vol. 20, pp. 131-78.

Corresponding author
Carlos Antonio França Sartori can be contacted at: sartori@pucsp.br