A FREQUENCY-DOMAIN APPROACH TO INTERCONNECT CROSSTALK SIMULATION AND MINIMIZATION

José Ernesto Rayas-Sánchez

Department of Electronics, Systems and Informatics, ITESO University,
Tlaquepaque, Jalisco, 45090 Mexico. Email: erayas@iteso.mx, Web: http://iteso.mx/~erayas

ABSTRACT

A frequency-domain approach to efficiently simulate and minimize the crosstalk between high speed interconnects is proposed in this paper. Several methods for modeling coupled microstrip transmission lines are discussed. Several possible simulation strategies are also considered. A straightforward yet rigorous frequency domain approach is followed. This approach can be used for linearly and non-linearly terminated microstrip coupled lines, since it exploits the Harmonic Balance technique. A typical example of microstrip interconnects is simulated and the results are compared with those obtained in previous work by other authors, using time-domain methods. The simulation method proposed in this work yields good accuracy. A crosstalk minimization problem is formulated and resolved following the method proposed.

RESUMEN

En este artículo se propone un método en el dominio de la frecuencia para simular y minimizar la interferencia entre pistas de interconexión de alta velocidad. Se analizan varios métodos para modelar líneas de transmisión acopladas microcinta. También se consideran varias posibles estrategias de simulación. En este trabajo se emplea un método en el dominio de la frecuencia simple pero riguroso. Este método puede aplicarse para líneas de transmisión acopladas que estén terminadas con componentes lineales o no lineales, debido a que utiliza la técnica de Balance de Armónicos. Un ejemplo típico de interconexiones microcinta es simulado y los resultados son comparados con los obtenidos por otros autores usando métodos en el dominio del tiempo. El método de simulación propuesto en este trabajo arroja buena precisión. Un problema de minimización de la interferencia entre conexiones es formulado y resuelto siguiendo el método propuesto.

A FREQUENCY-DOMAIN APPROACH TO INTERCONNECT CROSSTALK SIMULATION AND MINIMIZATION

José Ernesto Rayas-Sánchez

Department of Electronics, Systems and Informatics, ITESO University,
Tlaquepaque, Jalisco, 45090 Mexico. Email: erayas@iteso.mx, Web: http://iteso.mx/~erayas

ABSTRACT

A frequency-domain approach to efficiently simulate and minimize the crosstalk between high speed interconnects is proposed in this paper. Several methods for modeling coupled microstrip transmission lines are discussed. Several possible simulation strategies are also considered. A straightforward yet rigorous frequency domain approach is followed. This approach can be used for linearly and non-linearly terminated microstrip coupled lines, since it exploits the Harmonic Balance technique. A typical example of microstrip interconnects is simulated and the results are compared with those obtained in previous work by other authors, using time-domain methods. The simulation method proposed in this work yields good accuracy. A crosstalk minimization problem is formulated and resolved following the method proposed.

1. INTRODUCTION

The performance of high speed analog and digital electronic circuits critically depends on the quality of the transmitted signals, that should be undisturbed, undistorted, and with the desired speed. As the general speed of electronic circuits and devices increases, more attention should be paid to the design of interconnects.

The crosstalk minimization problem, or more generally, the signal integrity analysis associated to the design of interconnects has gained great importance due to:

(a) the recent advances on integrated circuits technologies (GaAs MESFET, HEMT, SiGe MOSFET, etc.) has reduced the single device switching time to tens of picoseconds or less,

(b) the development of VLSI technologies and packaging techniques are yielding smaller, denser chips,

(c) the use of high density buses, at both the printed circuit board (PCB) and the multi-chip module (MCM) levels, has increased the proximity of interconnects.

When the physical length of the interconnects becomes comparable to the wavelength of the highest frequency being transmitted, lumped impedance models can no longer be used for accurate simulation. Instead, a distributed transmission line model for the interconnect should be used. Further, the planar geometry used in integrated circuit technology allows that on-chip and inter-chip interconnections (PCBs, ASICs, ICs, MCMs) can be modeled as microstrip lines [1].

Much research has been accomplished on modeling and simulating microstrip lines as high speed interconnects. A time domain approach has been the most popular approach, especially in digital systems, to simulate crosstalk between interconnects, measuring the transient waveform of the undesired signal given a square or trapezoidal excitation pulse. A weakness of this method is that crosstalk may vary extremely with frequency, so that the crosstalk simulated can increase very significantly with small changes in the transient input waveform. An alternative method to efficiently simulate and minimize the crosstalk between interconnects is proposed in this paper, following a straightforward yet rigorous frequency domain approach. This approach can be used for linearly and non-linearly terminated microstrip coupled lines, since it is based on the Harmonic Balance technique. The accuracy of the proposed method is validated by directly comparing with previous work from other authors. Minimization of crosstalk between interconnects is also illustrated following the same frequency-domain approach.

2. CROSSTALK BETWEEN INTERCONNECTS

The crosstalk between channels A and B is defined as the ratio of the output of channel A, with no input signal, divided by the output of channel B excited by an input signal (see Fig. 1). In dB the crosstalk, *XTK*, from B to A is defined as

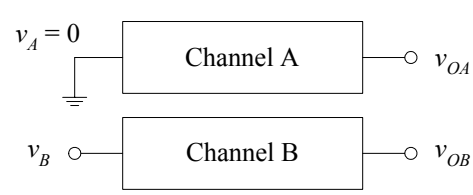


Fig. 1 A general two-channel system.

$$XTK = 20\log\frac{|v_{OA}|}{|v_{OB}|} \quad \text{dB} \tag{1}$$

Ideally, the crosstalk between channels that are supposed to be electrically unconnected should be zero (or minus infinite in dB). This is not the case when channels behave like coupled transmission lines, which in turns depends on their physical dimensions, proximities and materials, and operating frequencies.

Fig. 2 illustrates the physical structure of a coupled microstrip interconnect, consisting of two horizontal flat conductors near a ground plane. Notice that it is assumed that the conductor height is neglectable). Both conductors have the same length l and width w, and are mounted on a printed circuit wiring board with dielectric constant ε_r and thickness h. The conductors are separated a distance d. This physical representation is useful for PCB and MCM technologies.

The symbol shown in Fig. 3 will be used to represent the later coupled microstrip interconnect as a circuit component.

3. MODELING THE INTERCONNECTS

The physical structure of the couple microstrip interconnects can be modeled by full-wave electromagnetic analysis. However, a circuit approximation can be used following [2], in which case the coupled lossless transmission line equations are

$$-\frac{d}{dz}\begin{bmatrix}\boldsymbol{V}\\ \boldsymbol{I}\end{bmatrix}=\begin{bmatrix}\boldsymbol{0} & \boldsymbol{Z}\\ \boldsymbol{Y} & \boldsymbol{0}\end{bmatrix}\begin{bmatrix}\boldsymbol{V}\\ \boldsymbol{I}\end{bmatrix} \tag{2}$$

where $\boldsymbol{V} = [V_1 \quad V_2]^T$ and $\boldsymbol{I} = [I_1 \quad I_2]^T$ are the voltages and currents along each line, and

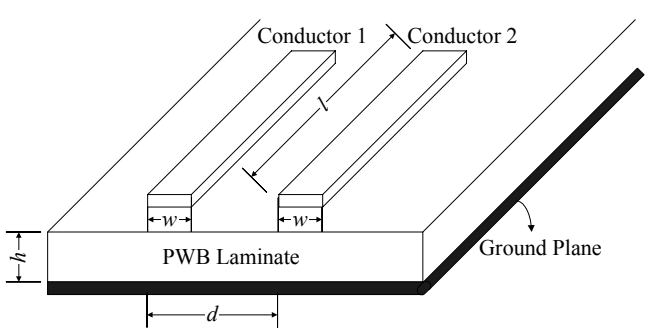


Fig. 2 Physical structure of a coupled microstrip interconnect.

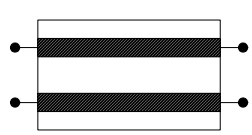


Fig. 3 Symbol of a coupled microstrip interconnect.

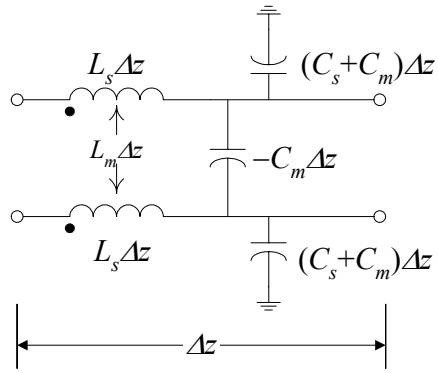


Fig. 4 A distribuited equivalent circuit of the coupled microstrip interconnect (lossless).

$$\boldsymbol{Z} = j\omega\boldsymbol{L} = j\omega\begin{bmatrix}L_s & L_m\\ L_m & L_s\end{bmatrix} \tag{3}$$

$$\boldsymbol{Y} = j\omega\boldsymbol{C} = j\omega\begin{bmatrix}C_s & -C_m\\ -C_m & C_s\end{bmatrix} \tag{4}$$

are the self and mutual inductance (impedance) and capacitance (admittance) matrices. It can be verified that an equivalent circuit to these equations is as illustrated in Fig. 4, where Δz represents a small increment along the transmission lines, so that the circuit components are distributed elements. Several possible approaches can be followed to model the coupled microstrip interconnect.

3.1 Walker's Formulas

Following [3], the empirical formulas for the LC parameters of the above equivalent circuit are shown in Appendix A. However, the per-unit length LC parameters of coupled microstrip interconnects obtained from Walker's formulas can deviate from the corresponding values obtained by full-wave electromagnetic simulation by a very significant amount [4].

3.2 SPICE Model

Standard Berkeley SPICE implementations are provided with built-in models for lossless and lossy uncoupled transmission lines. However, they do not have a model for coupled transmission lines, so that Walker's formulas can not be exploited directly in SPICE. According to [5], the coupled microstrip interconnect can be modeled by a circuit with two uncoupled transmission lines and eight polynomial controlled sources. The $\boldsymbol{L}$ and $\boldsymbol{C}$ matrices (obtained from Walker's formulas or from EM simulations) and the length of the coupled microstrip interconnect can be used to obtain the circuit components, which are compatible with most CAD programs including SPICE.

An approximate SPICE model of the coupled

transmission lines can be obtained by selecting an adequate number of elementary cells of the coupled lumped model (see Fig. 4), as recommended in [2].

3.3 Frequency-Domain Model (Kirschning-Jansen)

The Kirschning and Jansen frequency-domain model of the microstrip interconnect is based on expressions that have been derived by successive computer matching to converged numerical results originated from a rigorous spectral-domain hybrid-mode approach. These analytical expressions describe the effective dielectric constants, the power-current characteristic impedances, and the equivalent open-end lengths of coupled microstrip lines [6]. This model is accurate for the range of parameters

$$0.1 \le \frac{w}{h} \le 10 \tag{5}$$

$$1 \le \varepsilon_r \le 18 \tag{6}$$

$$f(GHz) \le \frac{30}{h(\text{mm})} \tag{7}$$

If the frequency domain model is used, it is necessary to employ the system voltage gain expression to derive the crosstalk information from the calculated scattering parameters. The linear circuit shown in Fig. 5 shows a generic network characterized by its S parameters with respect to a reference impedance Z_0. It can be shown [7] that the system voltage gain of the circuit in Fig. 5 is given by

$$A_V = \frac{v_o}{v_s} = \frac{S_{21}(1+\Gamma_l)(1-\Gamma_s)}{2[(1-S_{11}\Gamma_s)(1-S_{22}\Gamma_l)-S_{12}S_{21}\Gamma_l\Gamma_s]} \tag{8}$$

where

$$\Gamma_l = \frac{Z_L - Z_0}{Z_L + Z_0} \text{ and } \Gamma_s = \frac{R_S - Z_0}{R_S + Z_0} \tag{9}$$

are the reflection coefficients at the load and at the source, respectively.

In order to use simpler expressions, the network characterized by the S parameters can be conceptually expanded, as illustrated in Fig. 6. That is, if $\Gamma_l = 1$ ($Z_L \to \infty$) and $\Gamma_s = -1$ ($R_S = 0$), then

$$A_V = \frac{v_o}{v_s} = \frac{2S_{21}}{(1+S_{11})(1-S_{22})+S_{12}S_{21}} \tag{10}$$

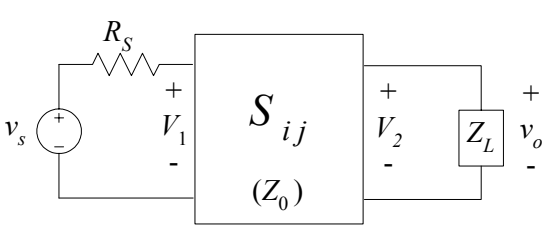


Fig. 5 A general two-port network.

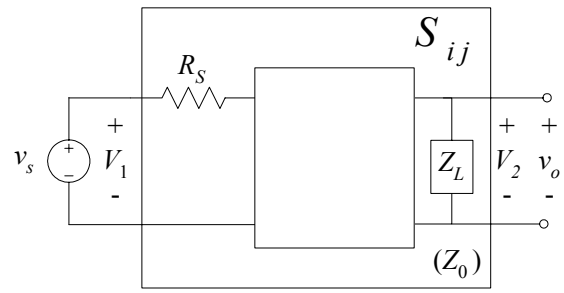


Fig. 6 Expanding the general two-port network.

4. SIMULATION OF A CROSSTALK PROBLEM

4.1 Problem Definition

A classical problem that has been studied by several authors is shown in Fig. 7. The circuit has three interconnects, several lumped passive components, one input signal, v_s, and four output voltages v_a, v_b, v_c, and v_d. The simulator must be able to calculate the voltage at any output, as well as the crosstalk between any pair of outputs.

The lumped components values are as follows: R_1 = 50 Ω, R_2 = 75 Ω, R_3 = 100 Ω, R_4 = 25 Ω, R_5 = 25 Ω, R_6 = 50 Ω, R_7 = 100 Ω, R_8 = 100 Ω, R_9 = 50 Ω, R_{10} = 100 Ω, L = 10 nH, C_1 = 1 pF, C_2 = 2 pF, C_3 = 1 pF.

In order to compare the simulation results with those obtained in previous work by other authors, the physical parameters for the microstrip lines were chosen as in [4] and [8], with the values d = 2.49 mm, h = 1.17 mm, w = 0.58 mm, μ_r = 1, ε_r = 5.182, and the length of each interconnect as L_1 = 5 cm, L_2 = 4 cm, and L_3 = 3 cm. These values correspond to the following LC parameters obtained from Walker's formulas: L_s = 494.5 nH/m, L_m = 63.29 nH/m, C_s = 69.97 pF/m, C_m = 7.94 pF/m.

In this particular case, the Kirschning-Jansen frequency domain model of the interconnects should yield good accuracy for frequencies as high as 25.64 GHz, according to (7).

4.2 Simulation Strategy

The simulation process, as well as the software tools to be used, inherently depends on the model chosen for the interconnect. The basic input data for any model of an interconnect are its physical parameters: h, ε_r, d, w. Once these parameters are determined, any of the following approaches could be followed.

A first approach consists of using Walker's formulas (see appendix A) to calculate the corresponding LC parameters and build up a SPICE model of the interconnect (e.g., Tripathi's model), and then use any circuit-level-time-domain SPICE compatible simulator,

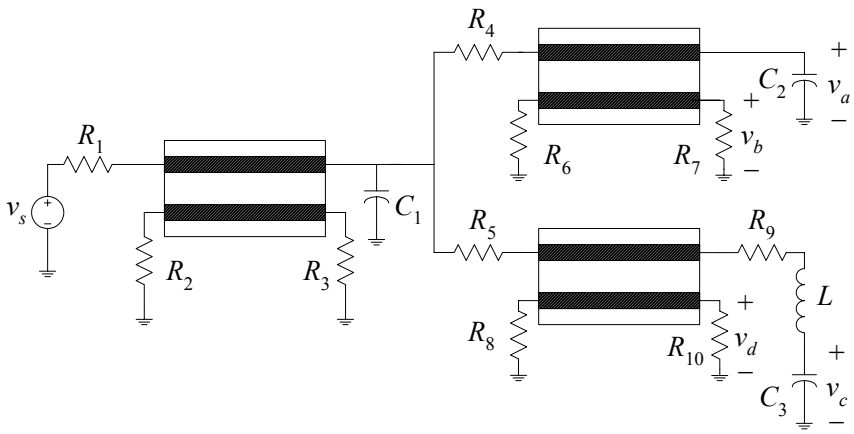


Fig. 7 A classical problem with three coupled interconnects terminated with lumped components.

such as *Orcad Pspice*™ [9].

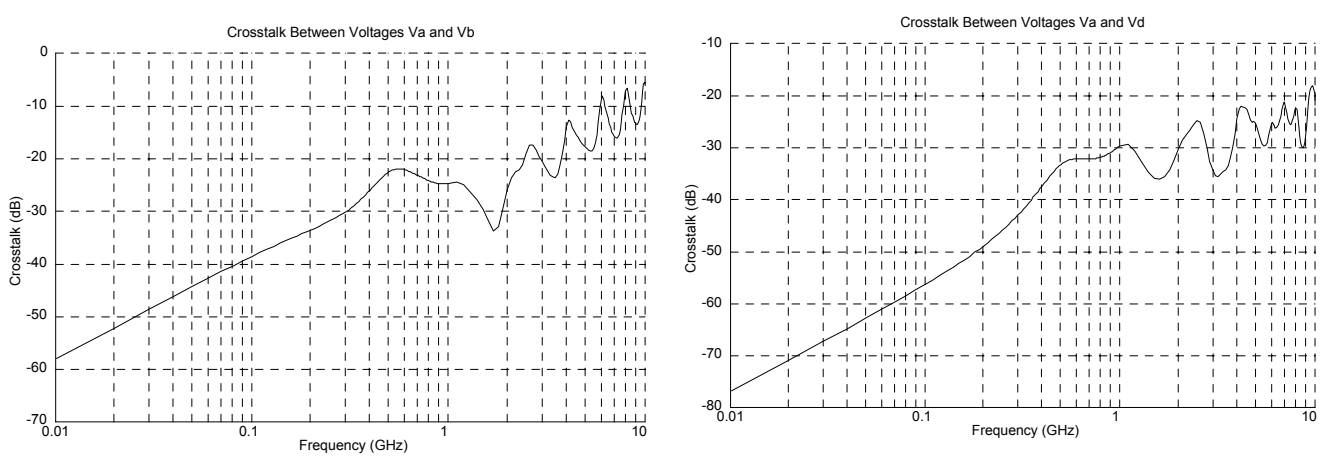
A second approach can be developed by using a full-wave electromagnetic simulator to obtain the LC matrices, such as Sonnet™ [10]. The LC values can then be used in a SPICE model as described above, or they can be incorporated into a special purpose interconnect simulator, such us *COFFEE2* [11] developed at Carleton University.

A third approach may be realized by using the frequency domain characteristics of the interconnects and a frequency domain simulator such as *OSA90/hope*™ [12]. The time domain steady state response of the circuit can be obtained from the frequency domain information by Fourier transformations.

Most of the researchers have followed one of the first two approaches, since they simulate the crosstalk effect in the time domain, transient response. The third approach was chosen in this work due to the following factors: (a) the accuracy of Kirschning-Jansen model of the interconnect (which is one of the built-in models available in OSA90/hope), is comparable with that one of an electromagnetic simulator within the model parameter and frequency regions of validity; (b) crosstalk can vary sharply within a given frequency range of operation; (c) the time domain steady state response of the circuit can be obtained by performing Fourier transformation; (d) using the built-in optimizers available in *OSA90/hope*™, minimizing crosstalk can be performed immediately after simulation.

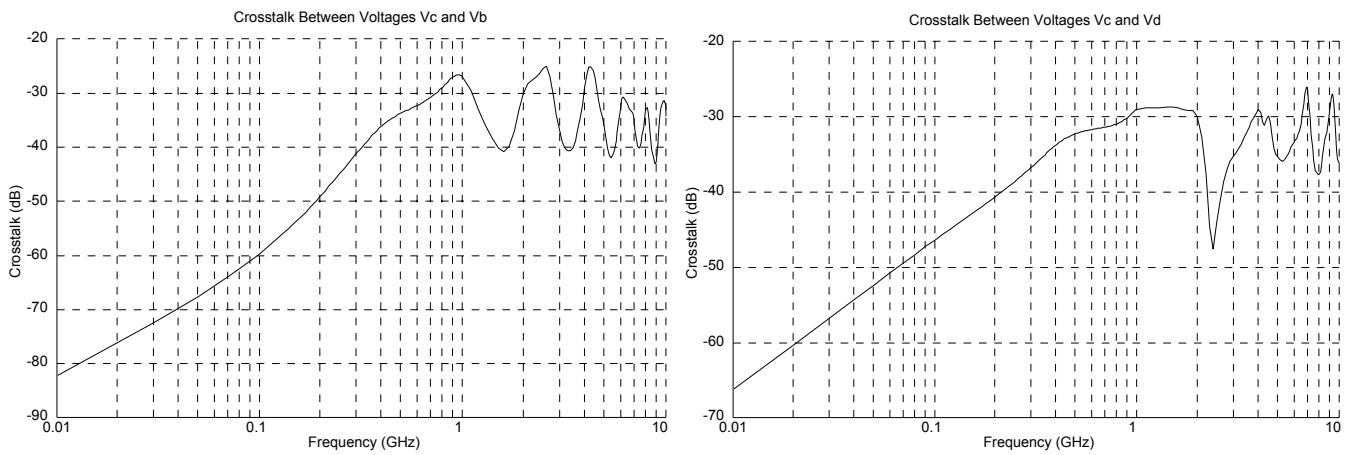
4.3 Frequency-Domain Results

An *OSA90/hope*™ input file was designed for the frequency domain simulation of the circuit shown in Fig. 7, including (10) as AC postprocessing. The Kirschning-Jansen frequency domain model was employed using the built-in linear elements MSCL (two-conductor symmetrical coupled microstrip lines) and MSUB (microstrip substrate definition) directly available in



(a) (b)

Fig. 8 Frequency-domain results: Crosstalk in dB from output *a* to (a) output *b*, (b) output *d*.



(a) (b)

Fig. 9 Frequency-domain results: Crosstalk in dB from output *c* to (a) output *b*, (b) output *d*.

OSA90/hope™. Figs. 8 and 9 shows the crosstalk obtained between all circuit outputs.

The worst case, that is, the maximum crosstalk in the circuit, is the one between the output voltages v_a and v_b. As mentioned before, the crosstalk phenomenon varies significantly with the operating frequency.

4.4 Time-Domain Results

An *OSA90/hope*™ input file was devised for the time domain simulation of the circuit shown in Fig. 7, using again the MSCL and MSUB built-in elements. The trapezoidal input signal shown in Fig. 10 is used as in [4]. Following [13], the Fourier exponential representation of a symmetrical ($T_r = T_f$) trapezoid waveform is given by

$$V_S(t)=\sum_n a_n e^{+j\omega_n t} \tag{11}$$

where

$$a_n=\frac{2a_v \sin(n\pi a_1)}{n\pi a_1}\frac{\sin(n\pi a_V)}{n\pi a_V} \tag{10a}$$

$$a_V=\frac{1.25T_r+T_w}{T_p}\ ,\ a_1=\frac{1.25T_r}{T_p}\ ,\ \omega_n=\frac{2n\pi}{T_p} \tag{10b-d}$$

A symmetrical trapezoid provides a reasonable representation of a digital pulse and, unlike a square wave, has a finite rise and fall times. This permits study of rise/fall time dependent effects.

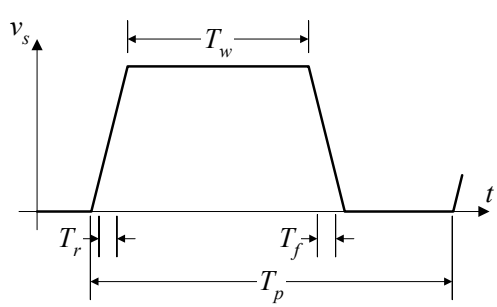


Fig. 10 A periodic trapezoidal input signal.

In order to compare the results with those obtained in [4], a rise time of 1.2 ns and a pulse width of 4.5 ns are chosen. The Harmonic Balance simulation technique is used. Sixteen harmonics were used to represent the input signal. Fig. 11 shows the time domain circuit output voltages, v_a, v_b, v_c and v_d, as well as the input trapezoid signal. The higher crosstalk obtained is from v_a to v_b outputs, as expected.

As mentioned before, the same circuit with the same parameter values was simulated in [4] using two different electromagnetic simulators (Sonnet and Zeland) to extract the ***L*** and ***C*** matrices, as well as using Walker's formulas. A comparison between the results obtained here using a frequency-domain model (Kirschning-Jansen) and those obtained in [4] is illustrated in Fig. 12. It can be seen that the accuracy of the approach followed in this work is quite acceptable, since the results agree more with the ones obtained using the electromagnetic simulators than those corresponding to Walker's formulas. Notice that the frequency domain model waveforms were shifted to the left, because they correspond to a periodic input trapezoid signal that does not start at zero seconds.

5. CROSSTALK MINIMIZATION

Minimizing crosstalk for the circuit shown in Fig. 7 can be formulated as follows. Assuming that

a) all the lumped components values are fixed
b) ε_r and μ_r are fixed
c) w, h, ε_r and μ_r are the same for the three interconnects
d) w, h, d and l are the design parameters,

design the three interconnects so that the crosstalk from v_a

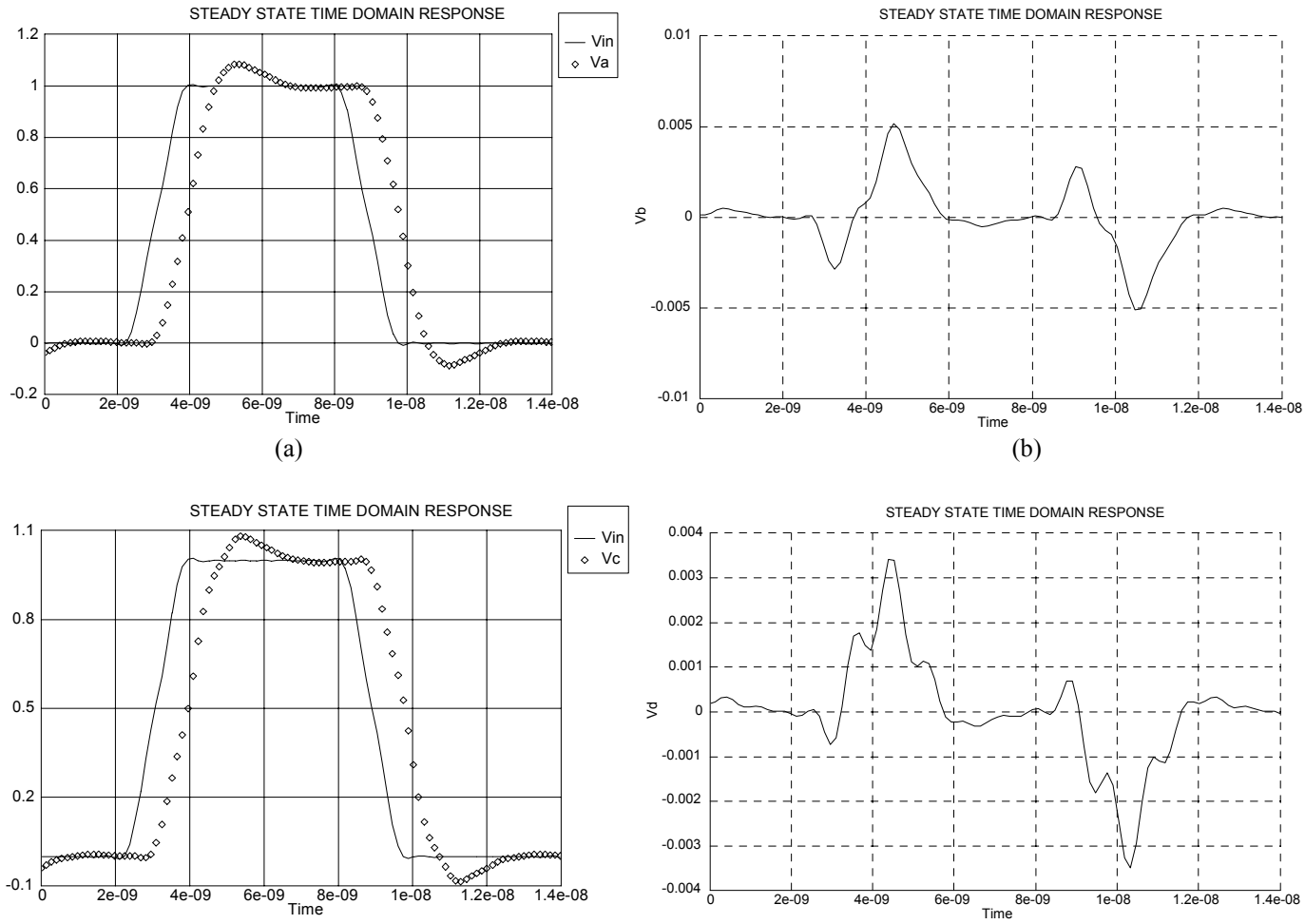


Fig. 11 Time-domain results: (a) input signal v_s and output voltage v_a, (b) output voltage v_b, (c) input signal v_s and output voltage v_c, (b) output voltage v_d.

to v_b is less than 0.02 (-34 dB) within an operating frequency range from 500MHz to 5 GHz, and the following constraints are satisfied:

$$0.25 \text{ mm} < w < 1\text{mm}$$
$$0.5 \text{ mm} < h < 2 \text{ mm}$$
$$0.5 \text{ mm} < d < 10 \text{ mm}$$
$$L_1 = 0.5\, L_2$$
$$L_2 = L_3$$
$$2 \text{ cm} < L_2 < 20 \text{ cm}$$

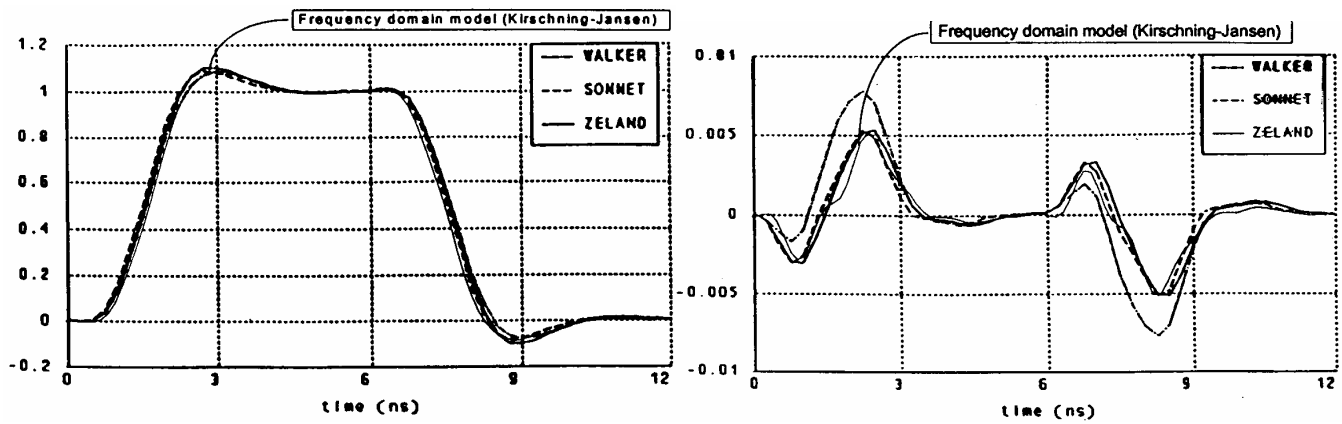
Before optimization, the simulation results obtained are shown in Fig. 13a, using the following physical parameter values for the interconnect: $w = 0.7435$ mm, $h = 1.5$ mm, $d = 3.1925$ mm, $L_2 = 5.5$ cm, $\varepsilon_r = 5.1825$ and $\mu_r = 1$.

Using the l_1 optimizer, the crosstalk specification is satisfied as shown in Fig. 13b. The following solution was found after 12 iterations: $w = 0.7284$ mm, $h = 0.5$ mm, $d = 2.185$ mm, $L_2 = 5.33$ cm.

Finally, the time domain simulation of the crosstalk voltage using the same trapezoid signal described previously is illustrated in Fig. 14, showing the results before and after optimization.

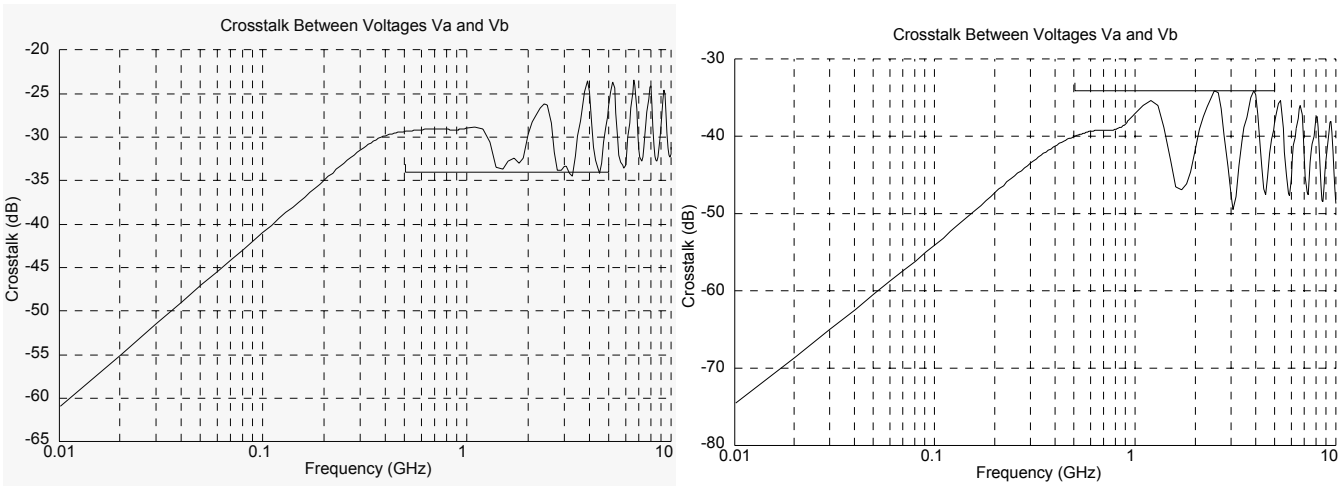
9. CONCLUSIONS

A frequency-domain approach to efficiently simulate and minimize the crosstalk between interconnects is proposed. For most practical circuits, the crosstalk between interconnects may vary extremely with frequency. Crosstalk minimization following a time-domain, transient response approach does not guarantee that crosstalk specification will be fulfilled within the whole operating frequency range of the interconnects. The method proposed permits a straightforward crosstalk simulation and minimization in the frequency range of interest, as well as steady-state time-domain calculations by Fourier transforming the frequency-domain results. The proposed technique can be applied for linearly and nonlinearly terminated interconnects, since it uses the Harmonic



(a) (b)

Fig. 12 A comparison between the results obtained with the frequency-domain method (Kirsching-Jansen model) used here and those obtained in [4] using Walker's formulas, and the electromagnetic simulators Sonnet and Zeland: (a) output voltages *a*, (b) output voltages *b*.



(a) (b)

Fig. 13 Crosstalk from output voltage *a* to output voltage *b*: (a) before minimization, (b) after minimization.

Balance technique. The accuracy of the frequency-domain model for simulating crosstalk between interconnects is validated by comparing with other results. The model used yields better accuracy than that one obtained by using Walker's formulas to calculate the LC parameters of the corresponding lossless coupled transmission lines.

APPENDIX A: WALKER'S FORMULAS

The self inductance for each conductor and the ground plane is given by

$$L_s = \frac{\mu_r \mu_o}{K_{L1}} \left(\frac{h}{w} \right) - \frac{\mu_r \mu_o}{4\pi} \ln \left[1 + \left(\frac{2h}{d} \right)^2 \right] \text{H/m} \qquad \text{(A1)}$$

The mutual inductance between the two conductors

$$L_m = \frac{\mu_r \mu_o}{4\pi} \ln \left[1 + \left(\frac{2h}{d} \right)^2 \right] \text{H/m} \qquad \text{(A2)}$$

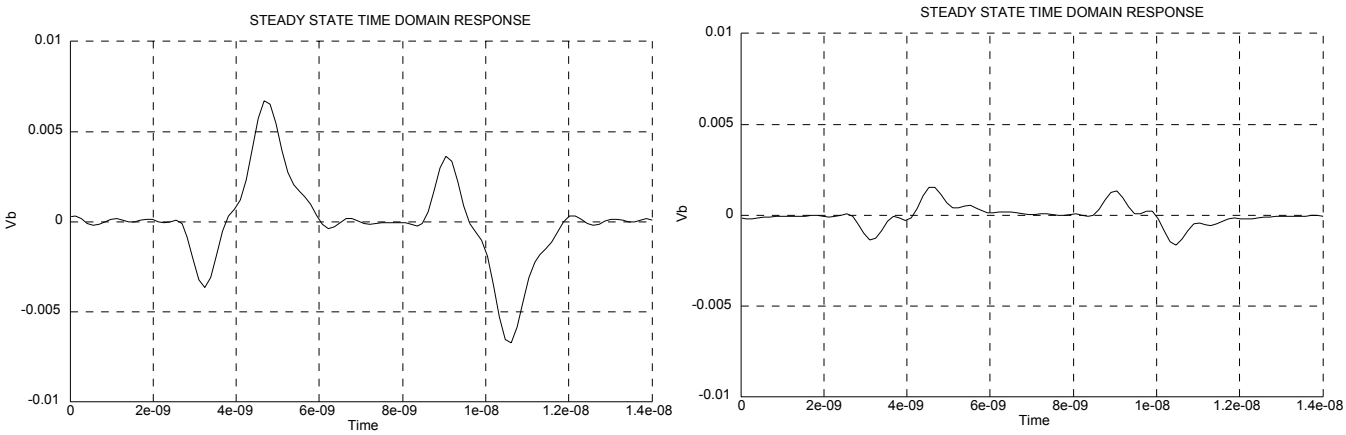
The capacitance between each conductor and the ground plane is

$$C_s = \varepsilon_r \varepsilon_o K_C \left(\frac{w}{h} \right) \text{F/m} \qquad \text{(A3)}$$

The capacitance between both conductors

$$C_m = \frac{\varepsilon_r \varepsilon_o}{4\pi} K_{C1} K_{L1} \left(\frac{w}{h} \right)^2 \ln \left[1 + \left(\frac{2h}{d} \right)^2 \right] \text{F/m} \qquad \text{(A4)}$$

where the fringing factors are



(a) (b)

Fig. 14 Time-domain output voltage *b*: (a) before crosstalk minimization, (b) after crosstalk minimization.

$$K_{L1} = \frac{120\pi}{Z_{0(\varepsilon_r=1)}}\left(\frac{h}{w}\right) \qquad \text{(A5)}$$

$$K_{C1} = \left[\frac{120\pi}{Z_{0(\varepsilon_r=1)}}\left(\frac{h}{w}\right)\sqrt{\frac{\varepsilon_e}{K_{L1}\varepsilon_r}}\right]^2 \qquad \text{(A6)}$$

and the characteristic impedance of each channel is, for $w/h \le 1$,

$$Z_{0(\varepsilon_r=1)} = 60\ln\left(\frac{8h}{w}+\frac{w}{4h}\right)\ \Omega \qquad \text{(A7)}$$

for $w/h \ge 1$,

$$Z_{0(\varepsilon_r=1)} = \frac{120\pi}{\left(\frac{w}{h}\right)+2.42-0.44\left(\frac{h}{w}\right)+\left(1-\frac{h}{w}\right)^6} \qquad \text{(A8)}$$

Some practical design considerations concerning Walker's formulas are presented below.

A.1 Effective dielectric constant

The effective dielectric constant, ε_e, accounts for nonhomegeneity of the region surrounding conductors. As $2h/w$ aproaches zero, the effective dielectric constant approaches the dielectric constant of the PWB laminate (ε_r), because most of the electric flux is totally in it. Conversely, as $2h/w$ becomes large, the effective dielectric constant approaches the average of the air (ε_o) and the laminate dielectric constants. In other words, the effective dielectric constant is the dielectric constant of a homogeneous medium that replaces the air and PWB laminate. Following Pozar [14]:

$$\varepsilon_e = \frac{\varepsilon_r+1}{2}+\frac{\varepsilon_r-1}{2}\frac{1}{\sqrt{1+12h/w}} \qquad \text{(A9)}$$

A.2 Fringing Factors

The fringing factor K_{C1}, takes into account the flux fringing of the electric field lines in a "parallel" plate capacitor. As the rate h/w increases, the actual capacitance increases, yielding a greater value than would be predicted from direct parallel plate equations, neglecting fringing. If the medium surrounding the flat conductor and the parallel plane were homogeneous, the capacitive fringing factor would be equal to the inductive one ($K_{C1} = K_{L1}$). However, in this case, the flat conductor is separated from the ground plane by the PBW laminate with relative dielectric constant, ε_r, and the region above the conductor is assumed to have a relative dielectric constant $\varepsilon_r = 1$ (air), so that the medium surrounding the conductor is not homogeneous.

A.3 Effects on Crosstalk

A ground plane greatly reduces the mutual capacitance and mutual inductance and hence crosstalk between two conductors. The mutual capacitance is very distance sensitive. Decreasing the spacing d by a factor of x, increases the mutual capacitance by a factor x^2. The mutual inductance per unit length has the same behavior. This is due to the fact that

$$\ln\left(1+\left(\frac{2h}{d}\right)^2\right) \approx \left(\frac{2h}{d}\right)^2 \text{ for } d >> h \qquad \text{(A10)}$$

which affects (A2) and (A4).

10. REFERENCES

[1] D.S. Gao, A.T. Yang and S.M. Kang, "Modeling and simulation of interconnection delays and crosstalks in high-speed integrated circuits," *IEEE Trans. Circuits Syst.*, vol. 37, pp. 1-8, 1990.

[2] T. Dhaene and D. De Zutter, "Selection of lumped element

models for coupled lossy transmission lines," *IEEE Trans. Computer-Aided Design*, vol. 11, pp. 805-815, 1992.

[3] C.S. Walker, *Capacitance, Inductance and Crosstalk Analysis*. Norwood, MA: Artech House, 1990.

[4] J.W. Bandler, R.M. Biernacki and S.H. Chen, "Toward direct EM optimization of VLSI interconnects: validation of coupled transmission line models," *Proc. 1995 Canadian Conf. Electrical and Computer Engineering* (Montreal, Canada), pp. 377-380, 1995.

[5] V.K. Tripathi and J.B. Retting, "A SPICE model for multiple coupled microstrips and other transmission lines," *IEEE Trans. Microwave Theory Tech.*, vol. 33, pp. 1513-1518, 1985.

[6] M. Kirschning and R. Jansen, "Accurate wide-range design equations for the frequency-dependent characteristics of parallel coupled microstrip lines," *IEEE Trans. Microwave Theory Tech.*, vol. MTT-32, pp. 83-90, 1984. Corrections: *IEEE Trans. Microwave Theory Tech*., vol. MTT-33, p. 288, 1985.

[7] J.J. Choma, *Electrical Networks: Theory and Analysis.* New York: Wiley, pp. 523-526, 1985.

[8] S. Lum, M.S. Nakhla and Q.J. Zhang, "Sensitivity analysis of lossy coupled transmission lines," *IEEE Trans. Microwave Theory Tech.*, vol. 39, pp. 2089-2099, 1991.

[9] Pspice, Cadence Design Systems, Inc., 555 River Oaks Parkway, San Jose, CA 95134.

[10] Sonnet® Suites 7.0, Sonnet Software, Inc., 1020 Seventh North Street, Suite 210, Liverpool, NY 13088, 2002.

[11] E. Chiprout and M.S. Nakhla, *Asymptotic Wave Evaluation and Moment Matching for Interconnect Analysis*. Boston, MA: Kluewer, 1994.

[12] *OSA90/hope*™*,* formerly Optimization Systems Associates Inc., P.O. Box 8083, Dundas, Ontario, Canada L9H 5E7, 1997, now Agilent Technologies, 1400 Fountaingrove Parkway, Santa Rosa, CA 95403-1799.

[13] R.A. Sainati and T.J. Moravec*,* "Estimating high speed circuit interconnect performance," *IEEE Trans. Circuits Syst.*, vol. 36, pp. 533-540, 1989.

[14] M. Pozar, *Microwave Engineering*. Amherst, MA: John Wiley and Sons, 1998.