A Universal Model for Lossy and Dispersive Transmission Lines for Time Domain CAD of Circuits

José I. Alonso, *Member, IEEE,* José Borja, and Félix Pérez, *Member, IEEE*

Abstract—**In this paper, the Branin's method of characteristics has been extended to obtain a universal equivalent model for lossy and dispersive transmission lines. Existing CAD software packages, such as SPICE, can be used for this implementation. The starting point for obtaining the model are the analog filters that approximate the characteristic impedance $Z_0(s)$ and the propagation function $F(s) = \exp(-\gamma l)$ of the transmission line. The circuits are synthesized using conventional network synthesis techniques. An examination of the validity of the model is carried out analyzing an example of RLCG lines driven by bipolar logic gates and the distortion of a Gaussian dc pulse as it propagates along a microstrip line.**

I. Introduction

The DEVELOPMENT of high-speed circuits is currently of increasing technological importance given the rising demand for greater velocities in the computation, signal processing and data link fields. Increasing velocities will require that more attention be paid to the design of interconnections, whose length may become significantly large in proportion to the wavelength. In addition to propagation delay, other wave shape related parameters such as rise and drop times, settling time, overshoot, preshoot, attenuation and coupling are key to the success of high velocity digital or wideband analog circuits. Consequently, the transmission lines models used in SPICE-type [1] commercial programs, which generally assume ideal lines, are unsuitable for studying the above mentioned phenomena.

It is thus necessary to find more complete and efficient transmission line models that allow inclusion of the effects of varying line parameters with frequency on a time domain analysis of the circuit. A SPICE compatible model for dispersive and lossy transmission lines is proposed in [2]. This model, however, requires a large number of sections of lumped elements and is thus inappropriate for many practical cases.

This article proposes a technique for obtaining transmission line models that uses conventional elements—resistors, capacitors, ideal transmission lines and dependent generators—and can thus be used in any SPICE-type commercial program. The model is based on an extension of Branin's model [3], as well as on the use of transfer functions that approximate frequency domain line behavior. The appropriate circuits are then synthesized using conventional network synthesis techniques.

Manuscript received July 18, 1991; revised December 30,1991. This work was supported by the project TIC-0023/14 of the National Programme on Information and Communication Technologies (PRONTIC).

J. I. Alonso and F. Pérez are with the Dpto. de Señales, Sistemas y Radiocomunicaciones, ETSI Telecomunicación, Universidad Politecnica de Madrid, 28040 Madrid, Spain.

J. Borja is with Telettra Espana, S. A.

IEEE Log Number 9106777.

II. Deriving the Transmission Line Model

A. Basic Principles

The transmission lines used in circuit interconnections can be uniquely characterized in the frequency domain by the following matrix equation:

$$\begin{bmatrix} V_1 \\ I_1 \end{bmatrix} = \begin{bmatrix} \cosh \gamma l & Z_0 \sinh \gamma l \\ \frac{1}{Z_0} \sinh \gamma l & \cosh \gamma l \end{bmatrix} \begin{bmatrix} V_2 \\ -I_2 \end{bmatrix} \tag{1}$$

where (V_1, I_1), and (V_2, I_2) are the terminal voltages and currents at the near end and the far end of the transmission line, as shown in Fig. 1, $Z_0(s)$ is the line's characteristic impedance, $\gamma(s)$ is the propagation constant and l is the line length.

In an ideal transmission line, the characteristic impedance is independent of the frequency, and the propagation constant is imaginary and varies linearly with the frequency, giving rise to a constant delay in the signals propagated. In real lines, however, owing to diverse effects (such as air-dielectric interfaces or conductor resistivity), the dependence of the characteristic impedance and propagation constant on the frequency is more complex and demands that the lines be simulated appropriately.

The method normally followed consists of obtaining a generic two-port network that characterizes the transmission line. The set of parameters chosen for the characterization can be any of those used for defining a two-port: *ABCD* parameters [2], *Z* parameters [4], etc. Consequently, it becomes a matter of fitting the chosen parameters to functions that are dependent on the characteristic impedance and on the propagation constant of the line. From this point of view, careful attention must be paid to the topology chosen, since, as is shown in the appendix, an unsuitable choice could limit the fit's validity.

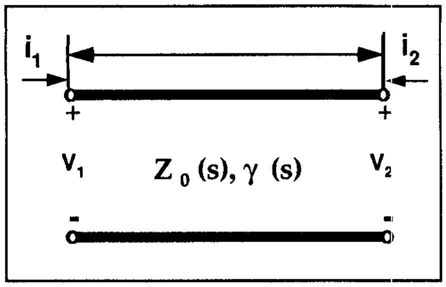


Fig. 1. Transmission line relationships.

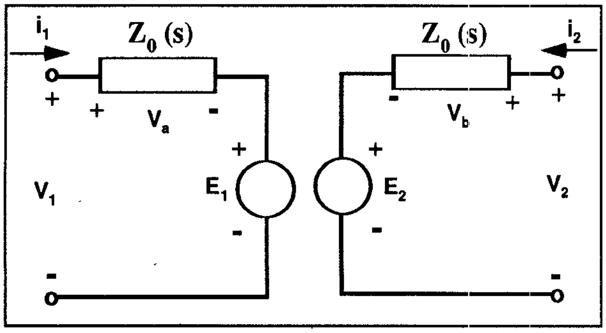


Fig. 2. The characteristic model of a transmission line.

The proposed technique to obtain a circuit model compatible with SPICE uses a topology derived from the generalization of the method of characteristics, very similar to that used in the works of F. Y. Chang [5] and Gruodis *et al.* [6]. Nevertheless, the aim of the Gruodis work is not to find equivalent circuits for their implementation in SPICE, although his method has been implemented in ASTAP [7] for low-frequency power lines. On the other hand, our models are more efficient in the sense of requiring less elements, and an efficient way of implementing the model in SPICE is presented as well.

B. Generalization of the Method of Characteristics

Equation (1) can be converted into the following equivalent form:

$$V_1 = Z_0 I_1 + [\exp(-\gamma l)](V_2 + Z_0 I_2) \quad (2a)$$

$$V_2 = Z_0 I_2 + [\exp(-\gamma l)](V_1 + Z_0 I_1) \quad (2b)$$

which suggest using the transmission line model shown in Fig. 2.

Generators E_1 and E_2 can be expressed as

$$E_1 = [\exp(-\gamma l)] \cdot V_P \quad (3a)$$

$$E_2 = [\exp(-\gamma l)] \cdot V_Q \quad (3b)$$

where

$$V_P = V_2 + V_b \quad (4a)$$

$$V_Q = V_1 + V_a. \quad (4b)$$

In this way, $E_1(s)$ and $E_2(s)$ are two voltage controlled voltage sources (VCVS's) whose gain and phase are dependent on the frequency. The problem consists of finding networks that implement the circuit's impedance $Z_0(s)$ and generators $E_1(s)$ and $E_2(s)$. These networks can now be found independently, although it must be borne in mind that in the physical problem the related phenomena are not independent.

III. Implementing the Model

The problem of synthesizing the impedance is solved by using conventional network methods [8], [9]. With respect to the voltage source $E_1(s)$, it is obtained by filtering $V_P(s)$ in a filter for which the transfer function is $F(s) = \exp(-\gamma l)$. As shown in Fig. 3, this filtering can be performed using a two-port network that has propagation function $F(s)$ and characteristic impedance any R_n, and is loaded with the said characteristic impedance.

The voltage source with value $2V_P$ may be obtained by means of two voltage sources in series, dependent on the voltages V_b and V_2 of the circuit in Fig. 2. The voltage E_1 of the circuit in Fig. 3 can be used as a control voltage of a voltage dependent source of unity gain, in order to obtain the generator E_1 of Fig. 2. With this technique a frequency dependent voltage controlled voltage source can be easily implemented using a two-port loaded with resistances and frequency independent voltage controlled voltage sources. The generator E_2 of Fig. 2 can be obtained in a similar way with the second circuit of Fig. 3.

It is possible, however, to reduce the complexity of the resultant circuit. Since the line is symmetric, the circuits to obtain E_1 and E_2 are identical and the superposition principle can be applied, leading to the results shown in Fig. 4.

As can be seen, V_{ta} and V_{tb} are now the voltages obtained on the loads of the two port, instead of E_1 and E_2, with

$$V_{ta} = E_1 - V_Q \quad (5a)$$

$$V_{tb} = E_2 - V_P \quad (5b)$$

Therefore, it is now necessary to place two voltage sources dependent on V_2 and V_b in series with the V_{tb} dependent generator (right part of the circuit in Fig. 4) in order to add V_Q to it, and in a similar way, sources dependent on V_1 and V_a must be placed in series with V_{ta} to add V_P.

IV. Synthesizing the Characteristic Impedance and the Propagation Function

The utility of the circuit in Fig. 4 will depend on the availability of simple circuit models for implementing the impedance $Z_0(s)$ and the two-port network with transfer function $F(s)$. Although this problem can be attempted by several different approaches, we will present a technique that can be used to obtain circuits with few elements and whose approximation error is small.

The starting point in the proposed technique are the rational functions in s that approximate the functions $F(s)$

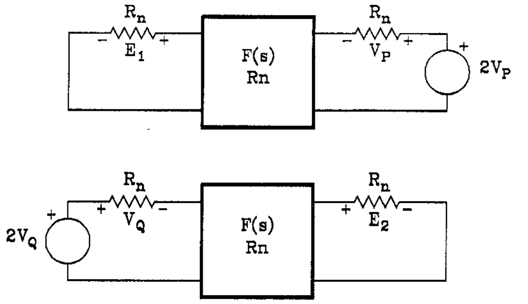


Fig. 3. Circuits to obtain the voltages E_1 and E_2.

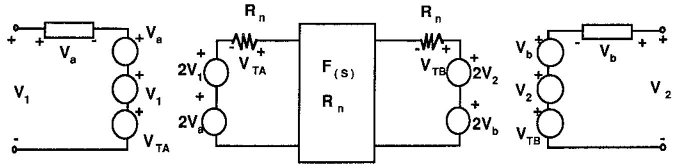


Fig. 4. Universal transmission line model.

and $Z_0(s)$. As it is described in [10], the following filters can be used to approximate the functions $Z_0(s)$ and $F(s)$:

$$Z_0(s) = k_1 \sum_{k=1}^{m} \frac{s^2 + a_k s + b_k}{s^2 + a_k' s + b_k'} \tag{6}$$

$$F(s) = k_2 \prod_{k=1}^{n} \frac{(s + c_k)}{(s + c_k')} \prod_{k=1}^{r} \frac{(s^2 - \alpha_k s + \beta_k)}{(s^2 + \alpha_k s + \beta_k)} \exp(-s\tau). \tag{7}$$

The coefficient values for the former expressions are calculated by means of a least square matching procedure, with restrictions that ensure the realizability of the functions obtained, using any of the several available expressions for the dielectric effective constant [11] and the characteristic impedance [12], in the case of the microstrip line, or using the expressions of $Z_0(s)$ and $\gamma(s)$, in the case of a RLCG line [13]. The frequency range for the fitting procedure must be consistent with the bandwidth of the signals present in the circuit.

Synthesis of the Characteristic Impedance

For a RLCG transmission line, the fact that the reactive component of $Z_0(s)$ is capacitive when $C > GL/R$ and inductive when $L > RC/G$, leads to the synthesis of an RC immittance or an RL immittance. The design procedures [8] of RC or RL immittances using the Foster or Cauer expansions result in a variety of circuits with a minimum number of elements, once the order of the function that achieves the approximation is determined. This order is related with the selected frequency range and with the grade of the adjustment.

For the case of the microstrip line, it should be noted that its characteristic impedance is often formulated as a power-to-current ratio [14], hence it is real; therefore, some phase must be admitted when it varies with frequency in order for the circuit used to synthesize the characteristic impedance to be realizable. Under this hypothesis, we compel the function to be synthesized to be real positive [9] in the adjustment procedure. In this case, the function Z_0 corresponding to an RLC immittance, can be synthesized by any of the canonical realizations.

Synthesis of the Propagation Function

The rational function in s proposed to simulate the exponential propagation function $F(s)$,—(7)—can be separated in the product of the following three functions:

$$F_1(s) = k_2 \prod_{k=1}^{n} \frac{s + c_k}{s + c_k'} \tag{8}$$

$$F_2(s) = \prod_{k=1}^{r} \frac{s^2 - \alpha_k s + \beta_k}{s^2 + \alpha_k s + \beta_k} \tag{9}$$

and

$$F_3(s) = \exp(-s\tau). \tag{10}$$

Filter $F_1(s)$ is a minimum phase filter whose modulus will approximate the attenuation constant $\alpha(\omega)$. Filter $F_2(s)$, on the contrary, corresponds to an all pass filter, and $F_3(s)$ is a constant delay. The physical line delay, associated to the phase constant $\beta(\omega)$, is approximated using both filters: $F_2(s)$, the delay of which is frequency dependent, and $F_3(s)$. The approximation procedure must be such that the sum of the delays of these two filters, added to the additional delay introduced by filter $F_1(s)$ must give a good approximation to the physical line delay.

We will now describe how the filters may be synthesized using conventional circuit elements.

The transfer function $F_1(s)$ can be realized with a single bridged-T network, such as shown in Fig. 5, or a cascade of such networks [6], [15]. The input-output relation in the frequency domain for the network in Fig. 5 is given by

$$F_1(s) = \frac{V_2}{V_1} = \frac{R_n}{R_n + Z(s)} = \frac{1}{1 + \frac{Z(s)}{R_n}} \tag{11}$$

and so

$$\frac{Z(s)}{R_n} = \frac{1}{k_2} \prod_{k=1}^{n} \left(\frac{s + c_k}{s + c_k'} \right) - 1 \tag{12}$$

and it is demonstrated [8] that if $F_1(s)$ is an approximation with alternating negative real poles and zeros, the impedance $Z(s)/R_n$ will always be realizable.

The transfer function $F_2(s)$ can be realized as a lattice network [8], [9]. In that case, both generator and load can't be simultaneously grounded, and so a transformer must be included. To avoid this trouble, we propose the circuits shown in Fig. 6, for second order all pass networks [16], [17].

It should be noted that $F_1(s)$ and $F_2(s)$ must be realized as two-port networks with the same characteristic impedance R_n, a constant for further simplicity. The importance of adding a lossless ideal transmission line to the model

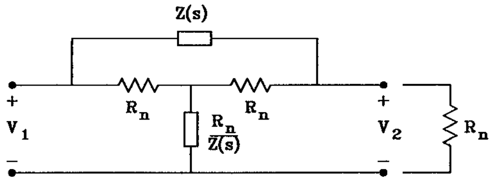


Fig. 5. Bridged T network (Characteristic Impedance R_n).

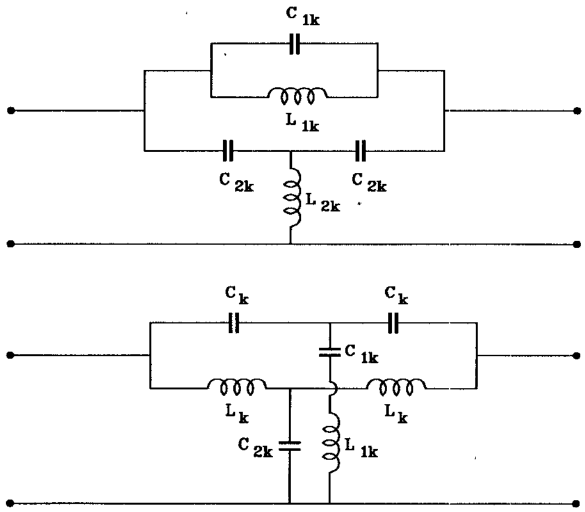


Fig. 6. Grounded second order all pass circuits (Characteristic Impedance R_n).

should be stressed. Other models obtain all the delay using only lumped elements. In our model, however, the major part of the physical delay is obtained with this lossless transmission line, whereas functions $F_1(s)$ and $F_2(s)$ implement the dispersive effects due to attenuation and frequency dependent delay. Thus, the use of the ideal transmission line simplifies the circuit and makes it more efficient. On the other hand, the model allows the use of any other kind of circuit to simulate the characteristic impedance and the transfer function.

V. Examples

Various examples with different types of transmission lines have been analyzed to check the validity of the proposed model. Some of the results obtained are described below. They give an idea of the universality, simplicity and versatility of the technique presented.

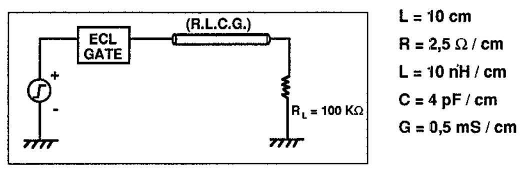
Example 1

This example analyzes a lossy transmission line with a 100 kΩ resistor load, and its near-end terminal is connected to the output of an emitter-coupled-logic (ECL) gate as shown in Fig. 7(a). This example is similar to that proposed in [5]. The lossy transmission line is 10 cm long, and its parameters are $R = 2.5\ \Omega/\text{cm}$, $L = 10\ \text{nH/cm}$, $C = 4\ \text{pF/cm}$ and $G = 0.5\ \text{mS/cm}$. The ECL gate, shown in Fig. 7(b) is excited by a step from -1.6 to -0.75 volts, with a 0.5 ns risetime, and the values of the parameters of the transistors are SPICE default except for $Tf = 0.065$ ns, $Cje = 0.2$ pF, and $Cjc = 0.3$ pF.

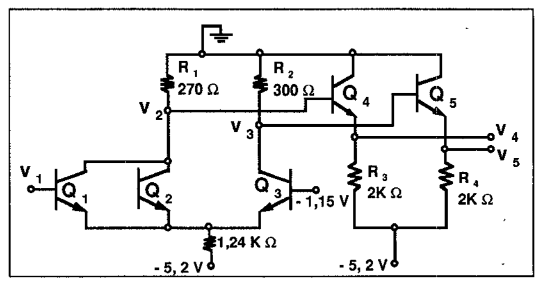
EXAMPLE 1
Element Values of Networks for the Synthesis of Characteristic Impedance and (b) $\exp(-\gamma(s)l)$

	Element Values
(a)	$R_{11} = 49.9999\ \Omega$
	$R_{21} = 5.70189\ \Omega$
	$R_{31} = 15.00903\ \Omega$
	$C_{21} = 0.89221$ nF
	$C_{31} = 0.49900$ nF

	Element Values
(b)	$R_n = 50.0\ \Omega$
	$R_{s0} = 21.2060\ \Omega$
	$R_{s1} = 0.86521\ \Omega$
	$R_{s2} = 0.6785\ \Omega$
	$R_{p0} = 109.8920\ \Omega$
	$R_{p1} = 3.6635\ \Omega$
	$R_{p2} = 4.3355\ \Omega$
	$L_1 = 4.0952$ nH
	$L_2 = 4.5434$ nH
	$C_1 = 1.3392$ nF
	$C_2 = 1.5987$ nF



(a)



(b)

Fig. 7. (a) Analyzed circuit. (b) ECL inverter circuit configuration used in (a).

The characteristic impedance and the propagation function [13] are approximated by ratios of polynomials in the complex s-plane for their circuit synthesis. Figs. 8 and 9 illustrate the adjustment of the real and imaginary parts of the characteristic impedance and the attenuation and phase due to the propagation constant, respectively. Only one section is needed for the adjustment of each one of both functions. The circuit elements of the Foster 1 realization of the characteristic impedance and the bridged T network for the attenuation constant are shown in Fig. 10. The model includes as well an ideal transmission line of $\tau = 2$ ns. As it can be seen in Figs. 11 and 12, the errors are very small, even when very low order circuits are used.

Comparing our method with the work of F. Y. Chang [5], 13 elements (4 capacitors and 9 resistors) are needed

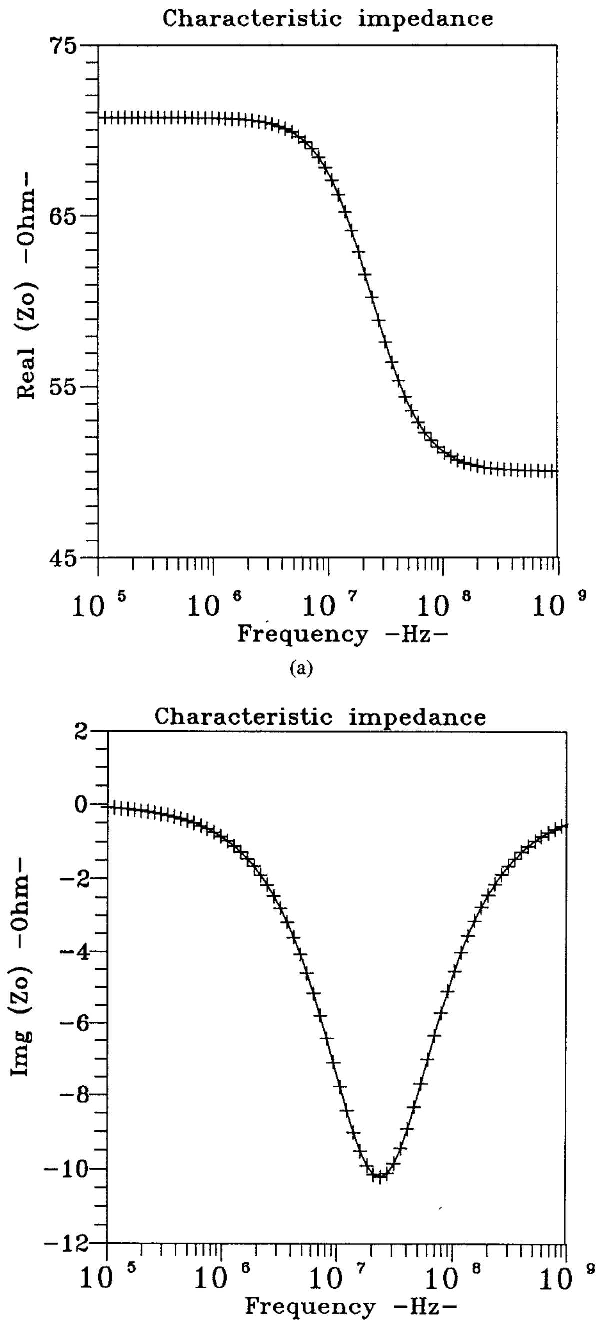


Fig. 8. Characteristic impedance of an RLCG line. (a) Real $[Z_0(s)]$. (b) Imag $[Z_0(s)]$. (———) Computed values for the line (+ + + +) Values calculated for the model.

to approximate $Z_0(s)$ with a 0.01% error in magnitude and a 0.01° error in phase. With our model, Z_0 is approximated to within 0.0015% magnitude error and 0.001° phase shift using a circuit of only 5 elements. On the other hand, the adjustment of the propagation function needs at best 80 lumped elements (40 capacitors, 20 inductors and 20 capacitors) for a correct simulation, while our approach needs only 12 lumped elements and a 2 ns. ideal transmission line to obtain similar results. The time domain analysis of our circuit was performed with SPICE giving the results shown in Fig. 13, which are very close to those presented in Fig. 11 of [5].

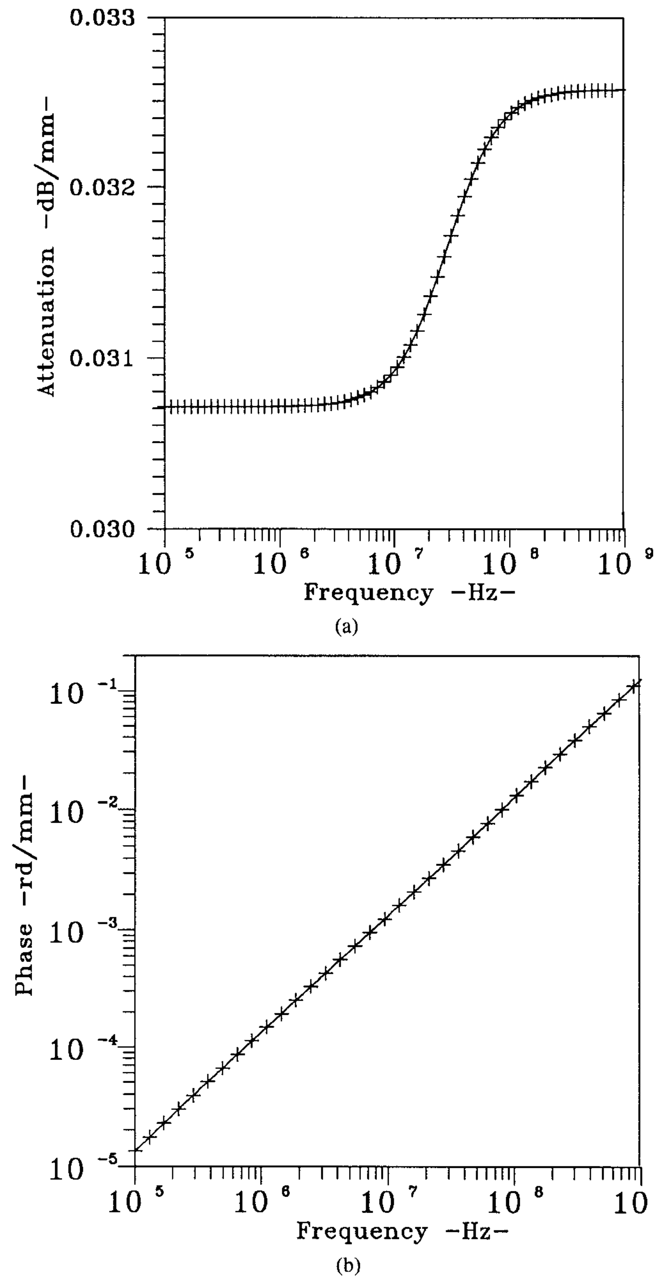


Fig. 9. Propagation constant of an RLCG line. (a) Attenuation -dB/mm-, (b) Phase-rad/mm-. (———) Computed values for the line. (+ + + +) Values calculated for the model.

Example 2

This example studies the propagation of a 45-ps-wide (at the half magnitude points) Gaussian pulse along a 122.14 mm transmission line realized on an alumina substrate ($\epsilon_r = 10.2$). As shown in Fig. 14 the line physical parameters, are: $H = 0.635$ mm, $W = 0.508$ mm, $t = 0.017$ mm, tan $\delta = 0.0015$ and $R_s = 8.63 \times 10^{-3}$ $\sqrt{f(\mathrm{GHz})}$. The line is loaded at both ends with a 53.7 Ω impedance.

The line is the same used by V. K. Tripathi in [2], except that the length is 26 times shorter. In [2] for that length (4.7 mm), and using a first order network, 16 lumped elements sections (more than 175 lumped elements in total) and 15 ideal transmission line sections are

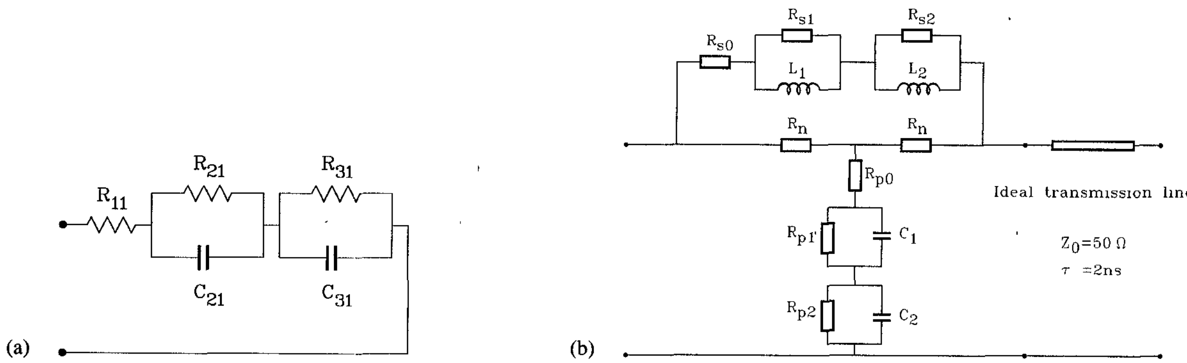


Fig. 10. Networks for the synthesis of (a) The characteristic impedance and (b) exp $(-\gamma(s)l)$.

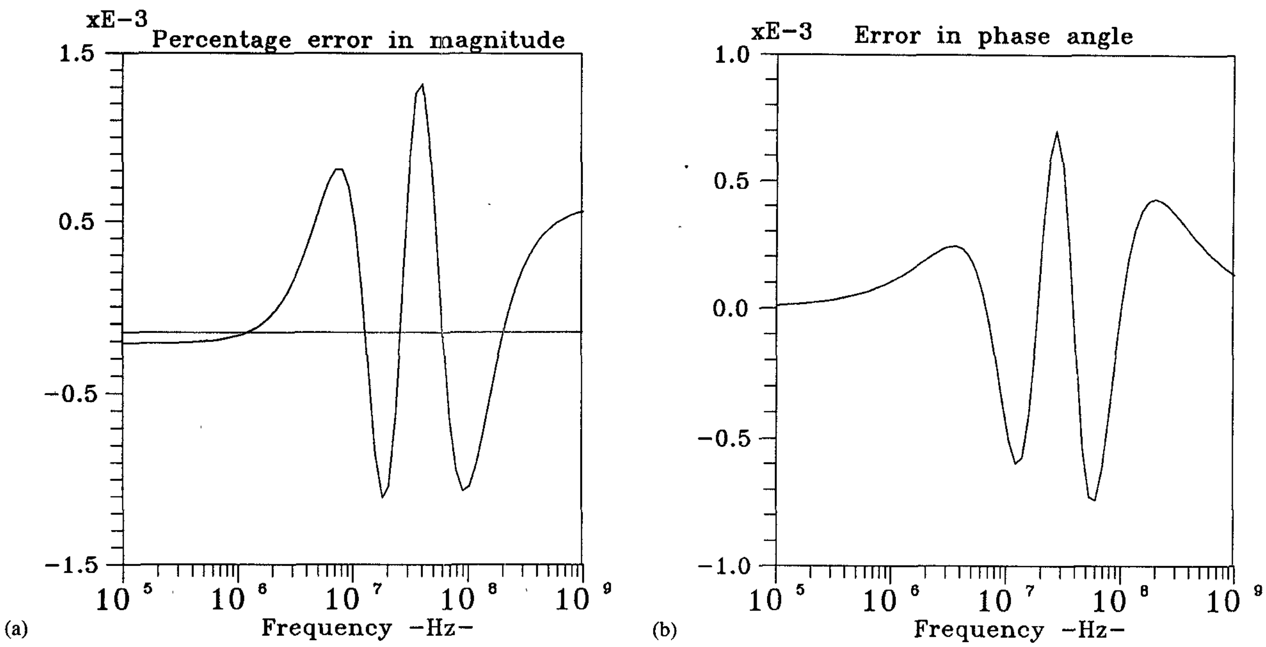


Fig. 11. Errors of approximating $Z_0(s)$ by the circuits of Fig. 10. (a) Percentage error in magnitude $Z_0(s)$. (b) Error in phase angle.

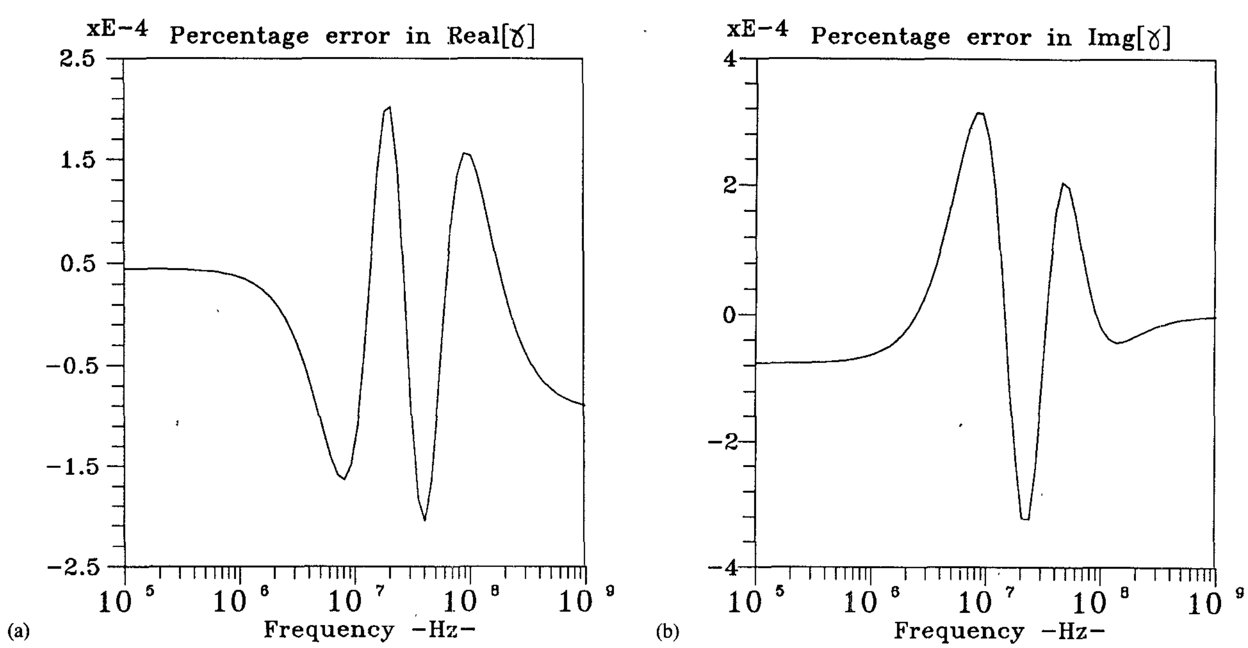
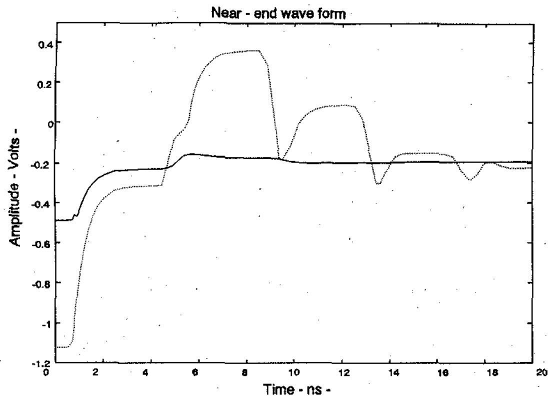
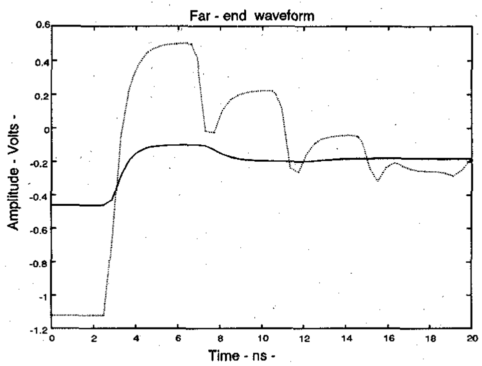


Fig. 12. Errors of approximating $\gamma(s)$ by the circuits of Fig. 13. (a) Percentage error in Re $[\gamma(s)]$. (b) Percentage error in Imag $[\gamma(s)]$. The time delay of the lossless transmission line is 2 ns.



(a)



(b)

Fig. 13. SPICE analysis of an RLCG line excited by an ECL gate. (a) Near-end waveform (b) Far-end waveforms. (----) Lossless line. (________) Lossy dispersive line.

EXAMPLE 2
ELEMENT VALUES OF NETWORKS FOR THE SYNTHESIS OF (a) NORMALIZED PHASE CONSTANT (ONE SECTION IS NEEDED FOR A 12.21 mm LINE LENGTH) (b) ATTENUATION (ONE SECTION IS NEEDED FOR A 40.71 mm LINE LENGTH) (c) CHARACTERISTIC IMPEDANCE

	Element Values
(a)	$L_1 = 0.35007$ nH
	$C_1 = 92.21530$ fF
	$L_{11} = 0.55310$ nH
	$C_{11} = 85.38790$ fF
	$C_{21} = 0.18443$ fF

	Element Values
(b)	$R_n = 53.7\ \Omega$
	$R_{a1} = 4.76728\ \Omega$
	$R_{b1} = 0.87580\ \Omega$
	$L_a = 53.19857$ pH
	$L_b = 93.09022$ pH
	$R_{a2} = 604.90395\ \Omega$
	$R_{b2} = 3.29262$ KΩ
	$C_a = 18.44809$ fF
	$C_b = 32.28163$ fF

	Element Values
(c)	$R_{11} = 53.6514\ \Omega$
	$R_{21} = 16.5538\ \Omega$
	$R_{31} = 0.0788\ \Omega$
	$L_{21} = 80.6861$ pH
	$C_{31} = 85.4848$ pF

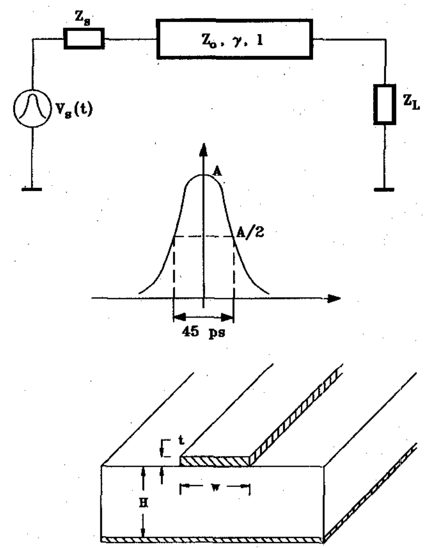
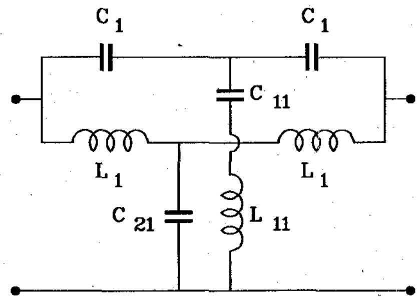
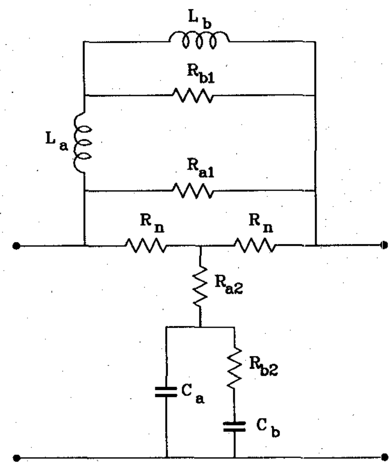


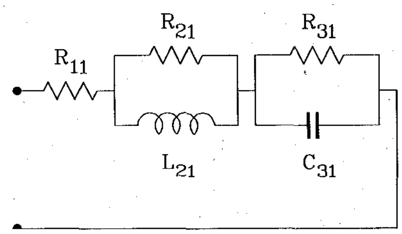
Fig. 14. A microstrip transmission line excited by a Gaussian pulse. Parameters and dimensions of the line.



(a)

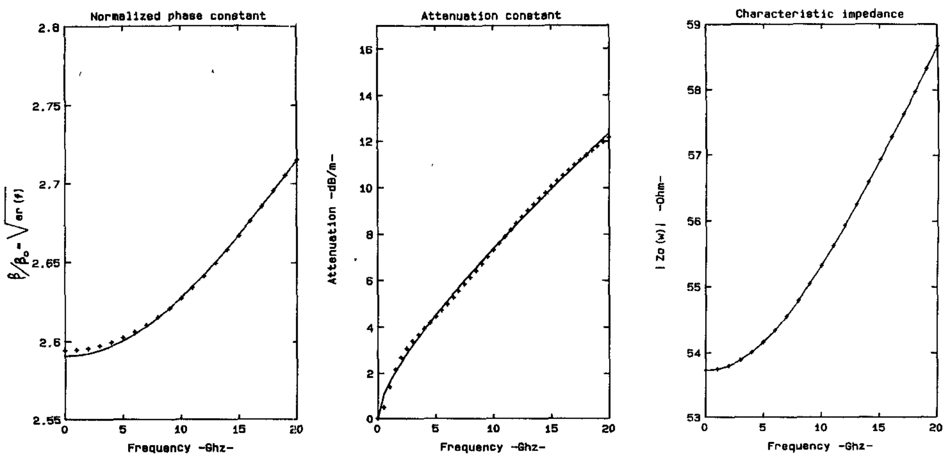


(b)



(c)

Fig. 15. Networks for the synthesis of (a) Normalized phase constant (one section is needed for a 12.21 mm line length), (b) Attenuation (one section is needed for a 40.71 mm line length), (c) Characteristic impedance.



(a) (b) (c)

Fig. 16. Propagation characteristic of the microstrip transmission line. (a) Normalized phase constant. (b) Attenuation constant. (c) Characteristic impedance. (————) Computed values for the line (+ + + +) Values calculated for the model.

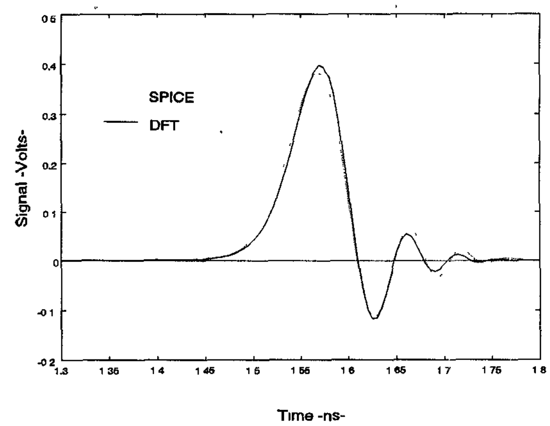


Fig. 17. Gaussian dc pulse dispersion at a distance L = 122.142 mm, (- - - -) Response of the model developed with SPICE (————). Computed by using DFT technique.

needed. For the same length and frequency adjustment range, our method needs only an ideal line of τ = 30 ps. and 32 lumped elements.

Fig. 15 shows the values of the elements of a section of those used to adjust the 122.14 mm line. In this case, for a narrower range of frequencies, ten circuit sections as those of Figure 15(a) are used for the simulation of the phase constant and three bridged-T networks are necessary to simulate the attenuation. A single ideal transmission line with τ = 858 ps is used to complete the circuit model of the line. Fig. 16 shows a good agreement between the propagation characteristics of the obtained circuit and the values computed for the line.

Finally, Fig. 17 illustrates the results of the time domain analysis of the proposed example using the SPICE and the circuit model. The results of analyzing the same circuit via DFT techniques are shown in the same figure, in which a good fit between the results obtained with both methods can be observed.

VI. Conclusion

A technique to develop an efficient and universal model for lossy and dispersive transmission lines has been presented. This technique is based on a generalization of Branin's model, which allows the optimization of the characteristic impedance $Z_0(s)$, and of the propagation function $F(s) = \exp(-\gamma l)$ to be performed independently.

The advantages of the proposed technique are related with the next facts:

a) The use of an ideal transmission line to implement part of the physical delay reduces the number of lumped elements needed.

b) The use of a topology that allows the adjustment of $Z_0(s)$ and $F(s)$ to be independent processes also reduces the number of lumped elements and complexity.

c) A novel combination of voltage sources allows its implementation in SPICE.

d) The proposed technique permits, as well, an easy implementation of frequency dependent voltage controlled voltage sources.

The application of the technique to different types of transmission lines -RLCG and microstrip- discloses its simplicity, versatility and accuracy.

Appendix

This appendix discloses the reason why it is preferable to adjust independently the characteristic impedance and the propagation constant. Let us consider the model in figure 18, proposed in [2] for simulating a lossy and dispersive transmission line. The model consists of an ideal transmission line and two RLC two-port networks connected in cascade. The conclusions reached also hold when no ideal transmission line exists; simply take $\tau = 0$ in the model.

The general topology is the following:

Keeping in mind that the two port networks must be passive and symmetric, the $[ABCD]$ parameters of two-port networks 1 and 2 are

$$A_2 = D_1 = D \tag{A1}$$

$$B_2 = B_1 = B \tag{A2}$$

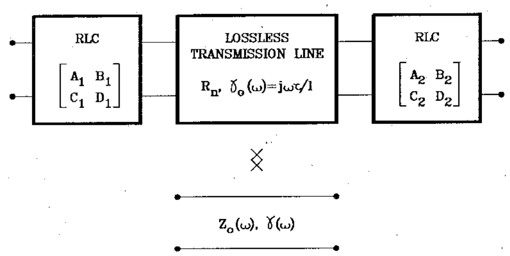


Fig. 18. General topology of a circuit model of a transmission line.

$$C_2 = C_1 = C \tag{A3}$$

$$D_2 = A_1 = A. \tag{A4}$$

After normalizing the impedances with respect to R_n, the following equation must be satisfied:

$$\begin{bmatrix} A & B \\ C & D \end{bmatrix} \begin{bmatrix} \cosh(j\omega\tau) & \sinh(j\omega\tau) \\ \sinh(j\omega\tau) & \cosh(j\omega\tau) \end{bmatrix} \begin{bmatrix} D & B \\ C & A \end{bmatrix} = \begin{bmatrix} \cosh\gamma l & Z_n \sinh\gamma l \\ \frac{1}{Z_n}\sinh\gamma l & \cosh\gamma l \end{bmatrix} \tag{A5}$$

where

$$Z_n = \frac{Z_0}{R_n}. \tag{A6}$$

In this system of equations, the $[ABCD]$ parameters may be calculated, and, hence the $[Z]$ parameters, that, normalized with respect to R_n, are given by

$$Z_{11} = \frac{A}{C} = Z_n \frac{1 + \exp[-(\gamma l - j\omega\tau)]}{1 - \exp[-(\gamma l - j\omega\tau)]} \tag{A7}$$

$$Z_{22} = \frac{D}{C} = \frac{1 + \exp[-(\gamma l - j\omega\tau)]}{1 - \exp[-(\gamma l - j\omega\tau)]} \tag{A8}$$

$$Z_{21} = Z_{12} = -\frac{1}{C} = 2 \cdot \sqrt{Z_n}\, \frac{\exp\left(\dfrac{-\gamma l - j\omega\tau}{2}\right)}{1 - \exp[-(\gamma l - j\omega\tau)]}. \tag{A9}$$

These results explain the difficulty in finding a suitable topology for networks, as the variation of the $[Z]$ parameters is not easy to carry out. In an L network such as that used in [2], $Z_{22} = Z_{12}$, and that is incompatible with (A8) and (A9). In that case, the model works because a very short ideal transmission line is used in comparison with the wavelength, and therefore l and τ are very small, and

$$Z_{21} \approx \sqrt{Z_n} \cdot Z_{22} \tag{A10}$$

and since the variation of $Z_0(s)$, and therefore, of Z_n with the frequency is very small, Z_{21} and Z_{22} can be equal to a value somewhere between the values given by (A8) and (A9) giving acceptable results. For another point of view, if one analog filter was associated to $\exp[-(\gamma l - j\omega\tau)/2]$ and another one to $\sqrt{Z_n}$, the two-port network that we should have to synthesize would not be realizable as Z_{21} has poles that Z_{22} does not, that is, those of $\sqrt{Z_n}$.

References

[1] L. W. Nagel, "SPICE2: A computer program to simulate semiconductor circuits," Memo. M520 Elect. Res. Lab., College of Engineering, Univ. of California, Berkeley, CA, 1975.

[2] V. K. Tripathi and A. Hill, "Equivalent circuit modelling of losses and dispersion in single and coupled lines for microwave and millimeter-wave integrated circuits," *IEEE Trans. Microwave Theory Tech.*, vol. 36, no. 2, pp. 256–262, Feb. 1988.

[3] F. H. Branin, Jr., "Transient analysis of lossless transmission lines," *Proc. IEEE*, vol. 55, pp. 2012–2013, Nov. 1967.

[4] T. J. Giovanni, "Time-domain simulation of linear, frequency-dependent distributed networks using SPICE2," in *1989 Proc. Int. Symp. Circuits and Systems*, vol. 1119, pp. 1122.

[5] F. Y. Chang, "Waveform relaxation analysis of RLCG transmission lines," *IEEE Trans. Circuits Syst.*, vol. 37, no. 11, pp. 1394–1415, Nov. 1990.

[6] A. J. Gruodis and C. S. Chang, "Coupled lossy transmission line characterization and simulation," *IBM J. Res. Develop.*, vol. 25, pp. 25–41, Jan. 1981.

[7] W. Weeks, A. Jiménez, G. Mahoney, D. Metha, H. Qassemzadeh, and T. Scott, "Algorithms for ASTAP-A network analysis package," *IEEE Trans. Circuit Theory*, vol. CT-20, pp. 628–634, 1973.

[8] G. C. Temes and J. W. LaPatra, *Introduction to Circuit Synthesis and Design*. New York: McGraw-Hill, 1977.

[9] N. Balabanian and T. A. Bickert, *Electrical Network Theory*. Malabar, FL: Krieger, 1985.

[10] Jose I. Alonso, "Análisis en el dominio del tiempo decircuitos de microondas mediante la Transformada Z. Aplicación al estudio de efectos dispersivos en líneas de transmisión," M.S. thesis. UPM, Sept. 1989.

[11] H. A. Atwater, "Test of microstrip dispersion formulas," *IEEE Trans. Microwave Theory Tech.*, vol. 36, no. 2, pp. 619–621, Mar. 1988.

[12] E. Hammerstad and O. Jensen, "Accurate models for microstrip computer-aided design," in *1980 IEEE MTT-S Int. Microwave Symp. Dig.*, pp. 407–409.

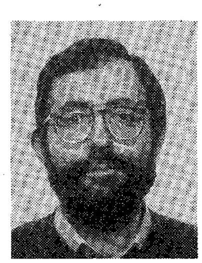
[13] R. E. Collin, *Foundations for Microwave Engineering*. New York: McGraw-Hill, 1966, pp. 87–89.

[14] R. H. Jansen and M. Kirsching, "Arguments and an accurate model for the power-current formulation of microstrip characteristic impedance," *Arch. Elek. Übertragung (AEÜ)*, vol. 37, pp. 102–108, 1983.

[15] F. Brglez, "Design of RL networks beyond 100 MHz," *IEEE Trans. Circuits Syst.*, vol. 23, no. 1, pp. 650–654, Nov. 1976.

[16] E. A. Guillemin, "Synthesis of RC networks," *J. Math. Phys.*, vol. 28, 1949.

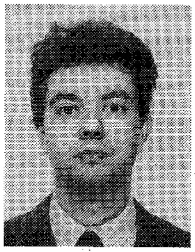
[17] V. R. Cunningham, "Design single-section delay equalizers," *Electron. Des.*, vol. 19, pp. 82–87, Sept. 1973.



José I. Alonso (M'90) was born in Villacañas (Toledo), Spain. He received the Ingeniero de Telecomunicación degree in 1982 and the Ph.D. degree in 1989, both from the Universidad Politécnica de Madrid, Spain.

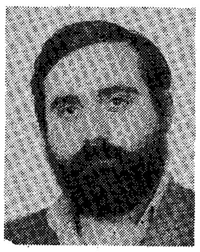
From 1982 to 1985 he worked as a microwave design engineer at Telettra Española S.A. In 1985, he joined the Dpto. de Señales, Sistemas y Radiocomunicaciones of the Universidad Politécnica de Madrid, where he is currently Associate-Professor. His research involves the simulation and computer aided of high speed digital circuits and of nonlinear mono-

lithic microwave circuits. He is currently engaged in development work on MMIC's for digital mobile communications.



José Borja, was born in Madrid, Spain, in 1966. He received the Ingeniero de Telecomunicación degree in 1990, from Universidad Politécnica de Madrid.

He research interests include simulation and development of CAD tools for the study of high-speed digital circuits and nonlinear microwave circuits.



Félix Pérez (M'91) was born in Castejón, Spain, in 1954. He received the Ingeniero de Telecomunicación and Ph.D. degree from the Universidad Politécnica de Madrid in 1977 and 1982, respectively.

From 1978 to 1987, he has been at the Microwave Department of the Polytechnic University of Madrid, first as Assistant Lecturer, and then as Associate Professor. He is presently Professor of the Signal, Systems and Radiocommunication Department. His research interests have been mainly in MESFET, front-ends for DBS-TV, and millimeter wave circuits. He is currently engaged in the developed of solid-state radar system and radar system simulation.