A NEW APPROACH FOR THE EXTRACTION OF SPICE COMPATIBLE MODELS FROM MEASURED PARAMETERS OF MICROWAVE CIRCUITS

J. M. Gómez [1] and J. I. Alonso [2]

[1] Dpto. de Teoría de la Señal y Comunicaciones. E. Politécnica.
Unversidad de Alcalá. Crta. Madrid-Barcelona. Km 33,6. 28871 Alcalá de Henares. Madrid.
[2] Dpto. Señales, Sistemas y Radiocomunicaciones. E.T.S.I. Telecomunicación.
Universidad Politécnica de Madrid. Ciudad Universitaria S/N. 28840 Madrid.

ABSTRACT

A new and simple technique for implementing microwave and MMIC circuits and components into a general purpose time-domain simulator, such as SPICE, will be presented. The developed technique is based in a generalization the Method of Characteristics and use the S-parameters as only requirement to obtain the model, which is composed by input impedances and voltage controlled voltage sources, both frequency dependent. These controlled generators can be easily implemented into SPICE using the Analog Behavioral Modelling. The technique will be validated by comparisons between measured S-parameters and those obtained with developed model.

INTRODUCTION

One of the major problems with time-domain analysis is the inability to utilize accurate frequency-domain descriptions of linear distributed and active elements, which form such an important part of any high-frequency circuit design. These descriptions need to include important effects such as loss and dispersion. Direct modeling of distributed linear components within a time domain simulator is limited only to simple idealized components.

Much work has been carried out on building suitable time domain, SPICE compatible descriptions. In general, these works have been oriented to describe components either by the development of SPICE compatible equivalent circuit [1] [2] [3] or by convolution-based techniques [4] [5]. First can involve complex equivalent circuits and are impractical in cases such as interconnect networks surrounding a pin grid array package or microwave components and circuits. The other use frequency domain data but are also limited to simple structures like a single transmission lines.

A efficient technique for implementing microwave and MMIC circuits and components into SPICE-type transient simulator will be presented. The developed technique is based in a generalization of the Method of Characteristics, and it is an extension of previous works in the field of modeling of non-uniform multiconductor transmission lines [6], and as will be shown in this communication, it has been successfully applied to model MMICs components and microwave circuits. The starting point of the developed technique is the knowledge of scattering parameters. This frequency domain data, gathered from simulation or measurements are used to implemented a model SPICE-compatible with only frequency-dependent input impedance and voltage controlled voltages sources, whose gains and phases are dependent on the frequency. These controlled generators can be easily implemented into SPICE using the Analog Behavioral Modelling option. The examples presented will validate the proposed technique.

BASIC PRINCIPLES

The starting point of the developed model, consist of finding the functional relations between the S parameters of the linear circuit and the input impedances, $Z_{ii}(s)$, and the frequency-dependent voltages .sources, $F_{ij}(s)$, which form the model, as is show in Fig.l. In order to find these relations, we assume that the S-parameters of the circuit whit a total of *n* ports are know from measurements or simulations.

According to the generalized characteristics method the circuit can be described by the equation:

$$\mathbf{V = ZI + E = ZI + F\,[V + Z_N]} \qquad (1)$$

where **Z** is a diagonal matrix of impedances, whose elements, $Z_{ii}(s)$, are the input impedances in each

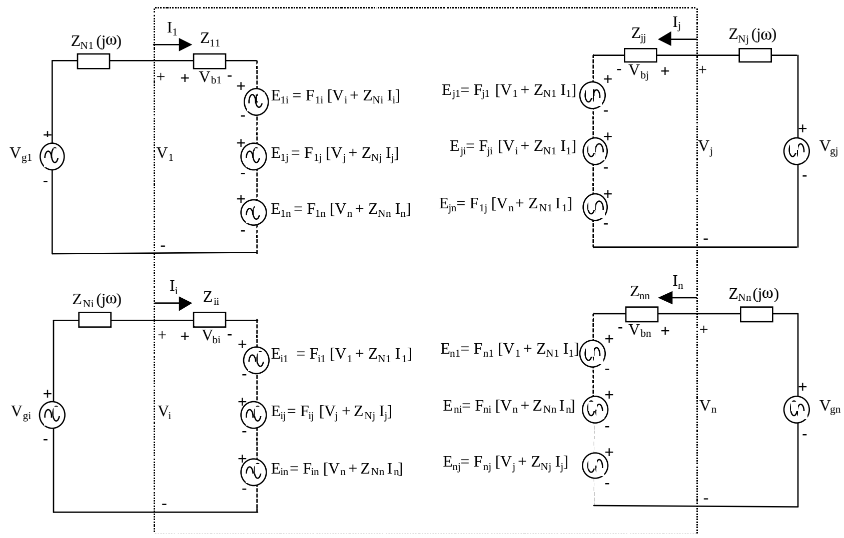

Fig.1. The generalized characteristics model of an n-ports circuit.

port with the others ports matched, and **E** is the vector which contains the frequency-dependent voltage waveform generators, as is show in the model of Fig.l. The matrix **F** is a propagation matrix, whose elements, $F_{ij}(s)$, simulate the delay and attenuation of signal propagation between *i*th and *j*th ports.

To obtain these relationships between the S-parameters of the circuit and the $Z_{ii}(s)$ and $F_{ij}(s)$ functions, we substitute in the definition of the generalized scattering matrix S of the n-port network the vectors **a** and **b** in terms of **V** and **I**, according to the definitions of complex power waves [7]. So, the next relationship is obtained:

$$\mathbf{(R - SR)V = (RZ_N^* + SRZ_N)I} \qquad (2)$$

If we split the matrix S in a diagonal matrix $\mathbf{S_d}$ and another matrix $\mathbf{S_r}$ which contains the rest of elements, that is to say $\mathbf{S = S_d + S_r}$, the equation (2) after some algebraic manipulations can be reformulated in this way:

$$\mathbf{V = \{(1_n - S_d)^{-1}\,[Z_N^* + S_dZ_N]\}I\ +} \qquad (3)$$
$$\mathbf{+\ \{R^{-1}(l_n - S_d)^{-1}(S - S_d)R\}\{V + Z_NI\}}$$

In this way, if we compare the equation (1) whit (3), we can obtain the values of matrices **Z** and **F** of the characteristic formulation in terms of scattering matrix **S** and the normalization impedances matrix $\mathbf{Z_N}$. The following result are obtained:

$$\mathbf{Z = (1_n - S_d)^{-1}\,[Z_N^* + S_dZ_N]} \qquad (4)$$

$$\mathbf{F = R^{-1}(1_n - S_d)^{-1}(S - S_d)R} \qquad (5)$$

the *i*th elements of the previous matrices are given by:

$$Z_{ii} = \frac{Z_{N\,i}^* + S_{ii}Z_{N\,i}}{1 - S_{ii}} \qquad (6)$$

$$F_{ij} = \sqrt{\frac{R_e(Z_{N\,i})}{R_e(Z_{N\,j})}}\,\frac{S_{ij}}{1 - S_{ii}} \qquad (7)$$

Therefore, frequency-dependent input impedances, $Z_{ii}(s)$, and the voltage controlled voltage sources, $E_{ij}(s) = F_{ij}(s)\,[V_j + Z_{Nj}\,I_j]$, whose gains and phases are dependent on the frequency, must be implemented in accordance whit [8]. In the proposed technique, the Analog Behavioral Modelling option is used to implement these controlled sources. In general, for most microwave circuits the normalizing impedances are 50 Ω, therefore the equations (6) and (7) are notably simplified.

EXAMPLES

To illustrate the advantages and the effectiveness of this method we give two representative examples: the modeling of a MMIC MESFET transistor and a microwave low pass filter.

In the first example, S-parameters of MESFET

0-7803-4471-5/98/$10.00 (c) 1998 IEEE

model for device whit 4 gate fingers for F20 process of GEC-MARCONY foundry are compared whit those obtained by the developed model. The figure 2 show the comparison between the S-parameters (magnitude and phase) of the foundry model ('-') and those obtained whit the proposed technique ('+').

The second example involves the obtaining of the model by the proposed technique for a microwave eight poles Chebyshev low pass filter. The filter has been fabricated in a microstrip substrate whit ε_r = 2,2 and h = 0,635 mm. This filter has been measured in an HP8610 Network Analyzer. Figure 3 compare the measured S-parameters, S_{11} and S_{21}, ('-') whit those obtained whit SPICE ('+'). In both cases, simulations and measurements are in excellent agreement.

CONCLUSIONS

A new and simple technique, which allows incorporate microwave and MMICs components and circuits into SPICE program, has been developed. The use of scattering parameters permits greater generality of the technique yielding a flexible transient time-domain simulator capable to simulate frequency dependent phenomena so important in MMICs and high-speed digital circuits. Simulated examples validate the technique by comparison between measurements and SPICE model.

ACKNOWLEDGMENTS

This work was supported by the projects TIC/96-0724-C06-01 and ESP/95-0612 of the National Board of Scientific and Technological Research (CICYT).

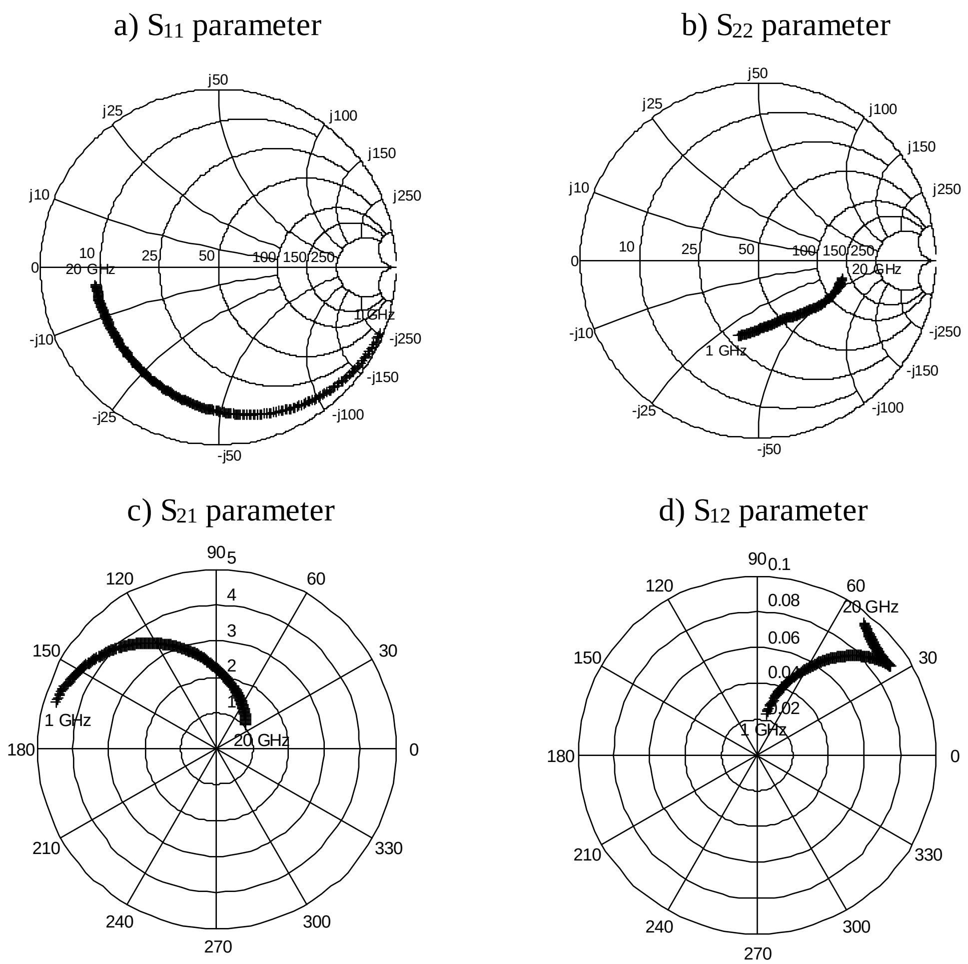

Figure 2. GEC-MARCONI foundry transistor ('-' equivalent circuit, '+' model obtained) a) S_{11} b) S_{22} ,c) S_{21} and d) S_{12} parameter.

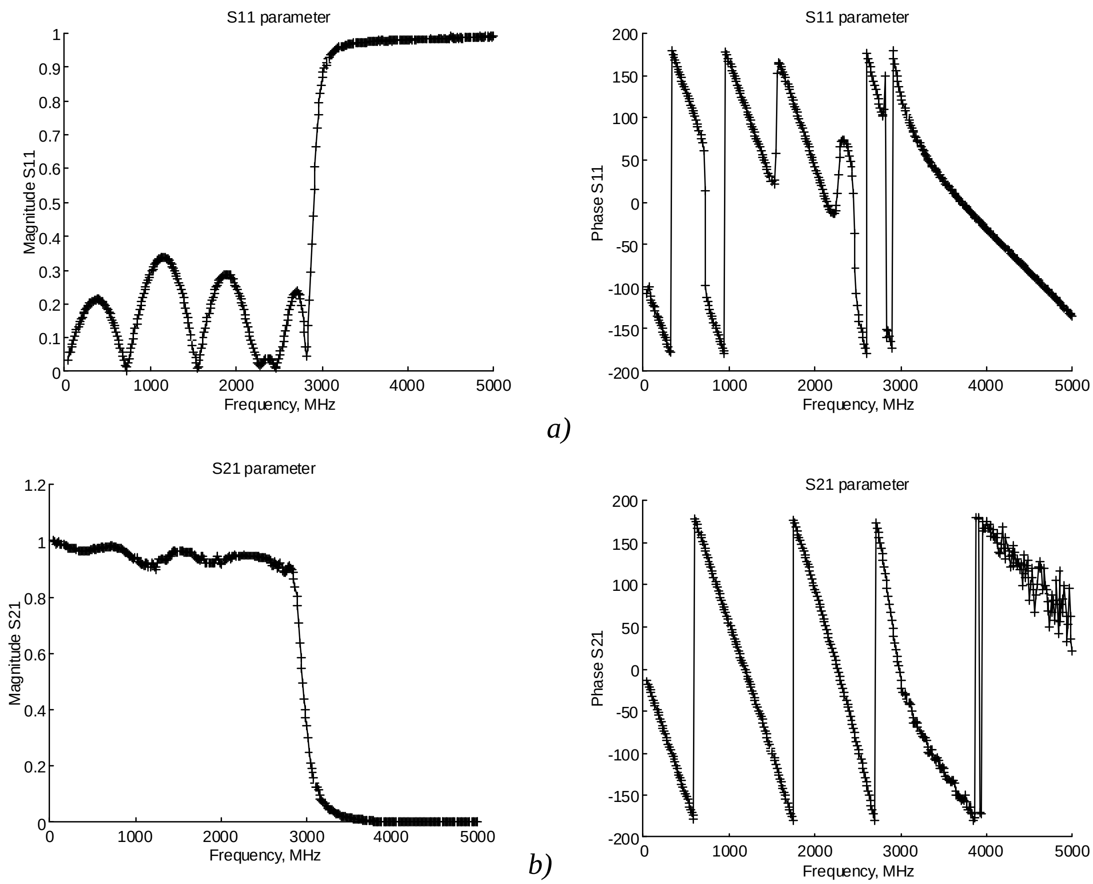

Figure. 2. Eight pule Chebyshev low pass filter ('-' measured data, '+' simulated data) a), b) Magnitude and phase of S11 and S21 parameters.

REFERENCES

[1] J.I.Alonso, J.Borja and F.Pérez. "*A Universal Model for Lossy and Dispersive Transmission Lines for Time Domain CAD of Circuits*", IEEE Trans. Microwave Theory Tech., Vol.40, pp.938-946, May 1992.

[2] V.K.Tripathi and A.Hill, "*Equivalent Circuit Modeling of Losses and Dispersion in Single and Coupled Lines for Microwave and Millimeter-Wave Integrated Circuits*", IEEE Trans. Microwave Theory Tech., Vol.36, pp.256-262, February 1988.

[3] 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, July 1992.

[4] A.R.Djordjevic, T.K.Sarkar and R.F.Harrington, "*Analysis of Lossy Transmission Lines With Arbitrary Nonlinear Terminal Networks*", IEEE Trans. Microwave Theory Tech. Vol.34, pp.660-666, June 1986.

[5] P.Halloran and T.J.Brazil, "*A Convolution-Based Approach to the Steady-State Analysis of Nonlinear Microwave Circuits Using Spice*", IEEE Trans. Microwave Theory Tech., Vol.43, pp.2157-2160, September 1995.

[6] J.I.Alonso and J.M.Gómez, "*Analysis of Transients in Coupled Tapered Lossy and Frequency-Dependent Transmission Lines using Spice*", Proc. European Microwave Conference, EuMC'97, Israel, pp.385-390, September 1997.

[7] K.Kurokawa, "*Power Waves and the Scattering Parameters*". 1965. IEEE Trans. Microwave Theory Tech., Vol.13, pp.194-202, March 1965.

[8] J.I.Alonso et al., "*Simulation of Arbitrary Nonuniform Frecuency- Dependent Interconnections Using Spice*". Electronics Letters, Vol.13, n°4, pp.253-254. February 1995.