Short Papers

A Convolution-Based Approach to the Steady-State Analysis of Nonlinear Microwave Circuits Using SPICE

Patrick Halloran and Thomas J. Brazil

Abstract—**A convolution-based, black-box approach is proposed to incorporate linear circuit blocks into a nonlinear time-domain analysis program (SPICE) for the purposes of obtaining steady-state results. The technique used is straightforward and offers an effective method for incorporating linear circuits described in the frequency domain into a nonlinear simulation. The technique allows SPICE-users to couple the flexibility and accuracy of general-purpose linear microwave simulators together with SPICE nonlinear device models, and thereby obtain steady-state results which are comparable in accuracy to popular alternative methods such as harmonic balance.**

I. Introduction

A wide range of numerical techniques has been applied to the simulation of microwave circuits which contain both lumped nonlinear components and distributed linear components. A complete review is given in [1]. Techniques used include harmonic balance [2], [3] and power series analysis [4], in addition to several others, e.g., [5], [6]. In recent years, harmonic balance has become one of the most popular frequency-domain methods for the steady-state analysis of microwave circuits with periodic excitation. Prior to the advent of harmonic balance, traditional time-domain simulation techniques (for example, as applied in SPICE) formed the most important large-signal nonlinear analysis tool available to the microwave engineer. Thus, there has been considerable investment in time-domain simulation models and tools, and they continue to be widely used.

One of the major problems with time-domain analysis is the inability directly to utilize accurate frequency-domain descriptions of linear distributed 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 transient simulator is limited only to simple idealized types. The description of all components must either be in the form of an equivalent circuit or a dyadic impulse response. Much work has been carried out on building suitable time-domain, SPICE-compatible descriptions of frequency-domain transfer functions with the main goal of determining the response to an aperiodic excitation, [7], [8], [10]. That is, in developing the time-domain model, the aim is to synthesize an impulse response which approximates the desired frequency function over that region of the frequency-domain where the signals incident on the component have nonnegligible spectral energy. In general, where real distributed components are being simulated, it will not be possible to obtain an impulse response which exactly represents the desired function over the required band, thus the simulated transient will always offer an approximation to the actual transient response of the circuit. Equivalent circuit-type models, on the other hand, are inherently tied to a particular circuit, as, for example, in the case of modelling of multiple coupled transmission lines in [7], [8]. In this approach, an equivalent circuit is proposed for a structure, and standard parameter extraction techniques are used to calculate the equivalent circuit parameters.

Manuscript revised April 1, 1993; revised May 25, 1995.

The authors are with the Department of Electronic and Electrical Engineering, University College Dublin, Dublin 4, Ireland.

IEEE Log Number 9413411.

With either approach, a transient simulator solves the circuit equations progressively in time and so any impulse functions synthesized from a frequency function must be causal [11]. Since the frequency functions are generally represented numerically and only over a limited band, this almost always implies that the impulse response offers an approximation to the corresponding frequency function. Thus, if a transient simulation with periodic excitation is simulated into steady-state, and the results are compared to a frequency-domain method such as harmonic balance, then some discrepancy will be observed, due to the fact that the impulse response is only approximating the desired frequency function at the fundamental excitation frequency and its harmonics. Of course, there are many other factors which may contribute to the discrepancy but the simulations can be set up so that these influences are minimized.

In view of the fact that SPICE-type transient simulators are used in certain applications to perform steady-state analysis of microwave circuits, it is apparent that it would be useful to the microwave circuit designer to have a technique which would allow the direct incorporation of frequency-domain data within such simulators in a form which would guarantee the correct steady-state result. In this paper we present such a method. The discrete Fourier transform (DFT) is used to synthesize a time-domain model from frequency-domain data, this model being exact at a certain set of frequencies. These frequencies are chosen to coincide with the excitation frequency and its harmonics. A device has been added to the SPICE simulator to accept the resulting discrete impulse response and to perform the appropriate convolutions. A standard SPICE simulation is then performed, until steady-state is achieved. Results can be presented directly in either time- or frequency-domain by estimating the Fourier series of the response by means of the DFT.

The use of convolution to simulate a nonlinear circuit which contains components specified by their frequency-domain behavior has been reported by many authors, one of the first being Silverberg and Wing [9]. Djordevic *et al.* [10], have also utilized convolution to simulate distributed lossy dispersive interconnects with nonlinear terminations. In the context of the present contribution, what is important is how the impulse response is calculated and how the subsequent convolutions are performed. Existing convolution-based techniques seek to offer an approximation to the desired frequency response at all frequencies, their aim being to make possible a full transient analysis of the networks involved. In this paper, a technique is presented which allows the incorporation of frequency-domain data into a general-purpose time-domain simulator such as SPICE, for the purposes of obtaining only steady-state results. While the method is exact for the steady state, the transient part of the solution (which must be first integrated through) is considered meaningless. The overhead of performing the convolutions is not significant in the simulation, since the impulse responses are short and discrete convolution is used.

In Section II the time-domain model is presented, followed in Section III by details of the incorporation of this model into SPICE. Finally, as an example, the steady-state analysis of a 10 GHz microwave amplifier is presented in Section IV and the results are

0018-9480/95$04.00 © 1995 IEEE

compared with an harmonic balance simulator. Concluding remarks are given in Section V.

II. Linear Circuit Representation

In the steady-state regime, voltages and currents at any point in a network are periodic and so have spectral energy only at discrete frequencies, dc, $f_0, 2f_0, \ldots$ where f_0 is the excitation frequency. Also for most practical cases a frequency can be found beyond which the steady-state responses have negligible spectral energy. Thus, if we are only interested in the steady-state behavior of a network which has linear circuit blocks embedded within it, we only need an exact representation of those blocks at dc, the excitation frequency, and a finite number of harmonics of that frequency. Let $H(f)$ represent some frequency-domain system function associated with a linear network which is embedded into a nonlinear network. Suppose it is required to simulate the steady-state response of the network to a sinusoidal excitation at f_0. We propose to sample $H(f)$ at dc, $f_0, 2f_0, \ldots N/2f_0$ and form the sequence, $\tilde{H}(kf_0)$ $k = 0, 1, \ldots N/2$ (N assumed even for illustration). Applying $\tilde{H}(nf_0)$ and its hermitian part to the inverse DFT yields the real-valued sequence $h(nT)$ given by (1) with $T = 1/Nf_0$

$$h(nT) = \frac{1}{N} \sum_{k=0}^{k=N-1} \tilde{H}(kf_0) e^{j2\pi(nkTf_0)} \qquad n = 0, 1, 2, \ldots, N-1. \quad (1)$$

If $h(nT)$ is considered to be a (finite) train of impulse functions then its continuous Fourier integral becomes exactly a (finite) summation (i.e., a discrete-time Fourier transform) and this results in a continuous periodic frequency function $\hat{H}(f)$ given by

$$\hat{H}(f) = \sum_{n=0}^{n=N-1} h(nT) e^{-j2\pi f nT} \quad (2)$$

This frequency function is equivalent to the original function $H(f)$ at dc, $f_0, 2f_0, \ldots (N/2-1)f_0$, and it is continuous at all points since it is a finite summation of trigonometrical functions. The relationship between $H(f)$ and $\hat{H}(f)$ at points other than the original samples will depend upon the behavior of the original function and on the sampling process. If a transient SPICE-type analysis is performed using these impulse samples, the steady-state result will be correct and the simulated transient will be that of a network which contains an embedded linear block described by a frequency function given by $\hat{H}(f)$. However, the resulting steady-state response to a periodic excitation with fundamental frequency at f_0, is still valid for the original system function, because $H(f)$ and $\hat{H}(f)$ are equivalent at dc, $f_0, 2f_0, \ldots (N/2-1)f_0$. At the same time, the transient portion of the simulation is invalid, since over the continuous frequency band where the transient response has spectral energy, the original function and its approximation are, in general, not similar. Using an approximate time-domain model within any convolution based simulator can result in an unstable representation of the overall network. The technique presented here can suffer from this problem because the time-domain model only guarantees an exact representation of the original frequency function at the fundamental frequency and its harmonics.

III. Incorporating Discrete Convolution into the SPICE Simulator

A direct time-domain simulator, such as SPICE, formulates and solves a set of coupled nonlinear integro-differential equations (associated with a nonlinear electrical circuit) directly in the time-domain. The circuit equations are solved progressively in time so that all device models must be causal. This is in contrast to simulation techniques which solve for the steady-state only, such as harmonic balance, where excitations and responses are periodic and exist for all time. The bandlimiting which is implicit in developing a time-domain model from a limited set of frequency-domain data, frequently causes the impulse response to become noncausal. The noncausal portion cannot be simulated, so it must be discarded and the positive-time portion of the impulse response adjusted in some way to compensate. Normally this is done in a manner which ensures correct dc performance of the approximation. However, it does cause an approximation error at all other frequencies of interest. In this paper, the sequence obtained from (1) is interpreted as a positive time sequence with no reference to its relationship to the actual continuous time-domain impulse response of the original frequency function. An advantage of (1) is the fact that it is discrete. This implies that the convolution operations are not as time-consuming as would be the case for a continuous-time model since only the convolving variable with a continuous character needs to be interpolated, and calculation of the convolution integral also reduces exactly to a simple sum-of-products operation. This interpolation is the only source of error in the simulation of the linear device.

We consider for illustration a linear two-port, represented in the time-domain by a set of impulse responses as described in Section II. If a scattering parameter representation of the two-port is used then the defining equations may be written in time as

$$\begin{aligned} b_1(t) &= h_{11}(nT) * a_1(t) + h_{12}(nT) * a_2(t) \\ b_2(t) &= h_{21}(nT) * a_1(t) + h_{22}(nT) * a_2(t). \end{aligned} \quad (3)$$

In (3) $*$ denotes convolution, a and b are the incident and reflected power waves, and the h_{mn} may be derived from (1) with $H(f)$ being the corresponding scattering parameter. The h_{mn} are discrete and finite and so the convolutions in (3) are exactly summations

$$h_{mn}(nT) * a_n(t) = \sum_{k=0}^{N-1} a_n(t-kT) h_{mn}(kT). \quad (4)$$

At each iteration at each timestep, (3) may be written in the following form for which a modified nodal analysis [12] stamp is readily calculated

$$\begin{aligned} i_1(t) &= G_1 v_1(t) + K_1 v_2(t) + H_1 i_2(t) + I_{01} \\ i_2(t) &= G_2 v_2(t) + K_2 v_1(t) + H_2 i_1(t) + I_{02}. \end{aligned} \quad (5)$$

Equation (5) may be represented by the equivalent circuit shown in Fig. 1. This is the resistive circuit presented to SPICE at each iteration, which may be readily incorporated within the normal time-domain numerical solution. It is a straightforward matter to extend this kind of representation to a P-port linear network, if required. In the following, a SPICE analysis incorporating this kind of description of linear frequency-domain blocks is described as a 'modified-SPICE' solution.

IV. Example of Application of Method

As an example of the technique described above, a 10 GHz MESFET amplifier on microstrip is presented in Fig. 2. The active device is represented by a standard nonlinear model, and the linear circuitry within which it is embedded is first analyzed in the frequency-domain using HP-MDS [13]. The microstrip structures which form the matching and bias networks are both lossy and dispersive, and eight harmonics are included in the analysis. The linear input and output matching and bias networks are represented in the frequency domain by two 3-port scattering matrices derived from HP-MDS, and

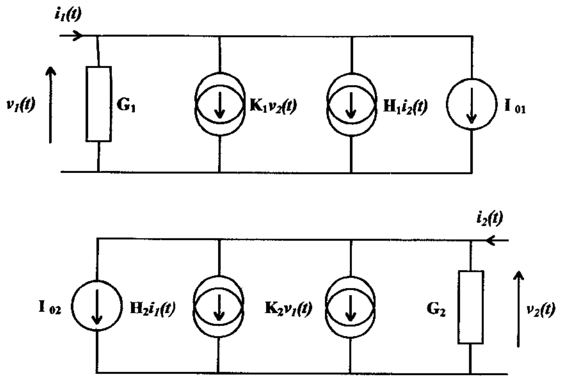

Fig. 1. Equivalent circuit presented to SPICE at each iteration.

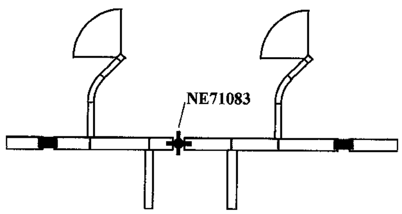

Fig. 2. 10 GHz MESFET amplifier.

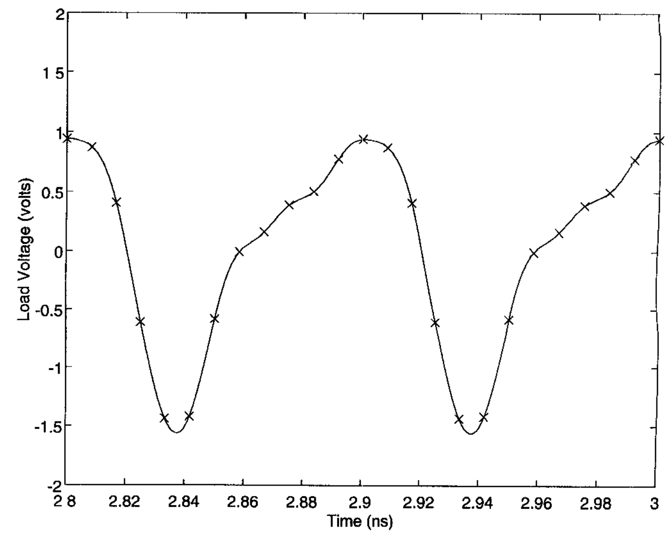

Fig. 3. Load voltage waveform, × harmonic balance, – modified-SPICE.

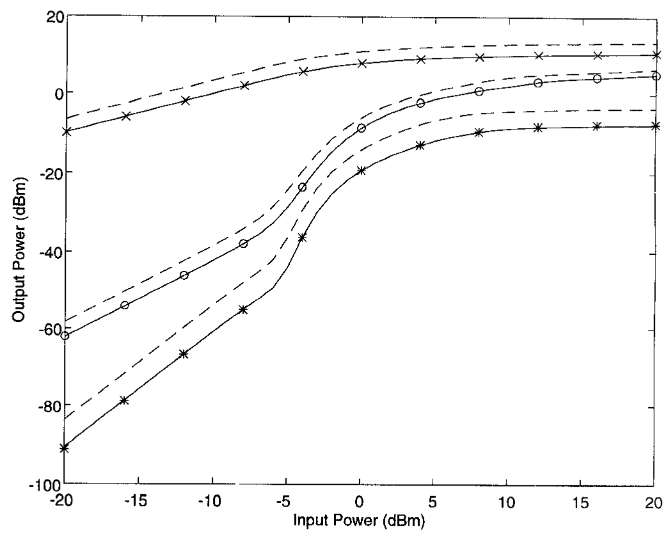

Fig. 4. Power sweep showing output power at 1st (×), 2nd (o), and 3rd (*) harmonics. ×/o/* modified SPICE simulator, — harmonic balance, — — idealized amplifier.

the corresponding time-domain models are calculated as outlined in Section II. A transient simulation using the modified-SPICE simulator is performed until a steady-state regime may be identified. An harmonic balance analysis of the same network is performed, also using HP-MDS. Identical device and circuit models are used both in the harmonic balance and modified-SPICE analysis. Since both simulation methods offer similar capabilities this test provides a good verification of the technique proposed.

Fig. 3 shows the output voltage in the time domain at an input drive level of 12 dBm with the MESFET biased in the linear region. The harmonic balance result has been transformed into the time-domain using the inverse Fourier series. The waveform obtained using harmonic balance and that obtained from the modified-SPICE simulator proposed in this work, are seen to agree very closely. Defining v_{hb} as the harmonic balance result and v_{ms} as the modified-SPICE result, we define the relative difference as

$$\varepsilon(t) = \frac{|v_{hb}(t) - v_{ms}(t)|}{|v_{hb}(t)|}. \tag{6}$$

The average relative difference over one period is less than 0.5%. Simulation run time on a HP700-series workstation for harmonic balance is five seconds while the modified-SPICE simulator requires 22 seconds. The duration of the time-domain transient, which is dependent on the circuit and the way in which the linear components are modeled, dictates the run-time for the modified-SPICE simulator. Note that the impulse response records need contain only 15 samples each in this case.

To further demonstrate the capabilities of the modified-SPICE simulator a power sweep is performed to an input drive level of 20 dBm. First-, second- and third-harmonic powers are plotted against available input power in Fig. 4. Again, both harmonic balance and the modified-SPICE simulator offer good agreement. A simulation using idealized transmission line models for the microstrip lines is also shown to demonstrate the effect of failing to account for the lossy substrate used in this example.

V. Conclusion

A technique has been presented which allows the use of frequency-domain data within a SPICE analysis for the purposes of obtaining steady state results. The modifications to the SPICE simulator are at a device level. The technique allows SPICE users to couple the flexibility of a linear microwave simulator with SPICE nonlinear device models, and obtain steady-state results. If circuit transients are long then the technique will be inefficient since in order to obtain the steady-state results an artificial transient must be integrated through. However it is certainly useful in situations where these transients do not persist over a long time interval. A realistic example of a 10 GHz microwave amplifier is presented, and results compared to an harmonic balance analysis showing excellent agreement.

References

[1] V. Rizzoli and A. Neri, "State of the art and present trends in nonlinear microwave cad techniques," *IEEE Trans. Microwave Theory Tech.*, vol. 36, pp. 343–365. Feb. 1988.

[2] M. S. Nakha and J. Vlach, "A piecewise harmonic balance technique for determination of periodic response of nonlinear systems," *IEEE Trans. Circuits Syst.*, vol. CAS-23, pp. 85–91, Feb. 1976.
[3] K. S. Kundert and A. Sangiovanni-Vincentelli, "Simulation of nonlinear circuits in the frequency domain," *IEEE Trans. Computer-Aided Design*, vol. CAD-5, pp. 521–535, Oct. 1986.
[4] G. W. Rhyne and Michael B. Steer, "Generalized power series analysis of intermodulation distortion in a MESFET amplifier: Simulation and experiment," *IEEE Trans. Microwave Theory Tech.*, vol. MTT-35, pp. 1248–1255, Dec. 1987.
[5] D. R. Frey and Orhan Norman, "An integral equation approach to the periodic steady-state problem in nonlinear circuits," *IEEE Trans. Circuits Syst.-I & II*, vol. 39, pp. 744–755, Sept. 1992.
[6] A. Buonomo, "Time domain analysis of nonlinear circuits with periodic excitation," *Electron. Lett.*, vol. 27, pp. 65–66, 1991.
[7] 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, Feb. 1988.
[8] 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.
[9] M. Silverberg and O. Wing, "Time domain computer solutions for networks containing lumped nonlinear elements," *IEEE Trans. Circuit Theory*, vol. CT-15, pp. 292–294, Sept. 1968.
[10] A. R. Djordjevic and T. K. Sarkar, "Analysis of lossy transmission lines with arbitrary nonlinear terminal networks," *IEEE Trans. Microwave Theory Tech.*, vol. MTT-34, pp. 660–666, June 1986.
[11] Thomas J. Brazil, "A new method for the transient simulation of causal linear systems described in the frequency domain," in *IEEE Microwave Theory Tech.-S Dig. 1992*, pp. 1485–1488.
[12] C. W. Ho, A. E. Ruehli, and P. A. Brennan, "The modified nodal approach to network analysis," *IEEE Trans. Circuits Syst.*, vol. CAS-22, pp. 504–509, June 1975.
[13] HP85150B Microwave Design System (HP-MDS).

Are Nonreciprocal Bi-Isotropic Media Forbidden Indeed?

Ari H. Sihvola

***Abstract*— Doubt is cast in this article on the universality of the conclusion that linear bi-isotropic media have to be reciprocal, which claim has recently been set forth by Lakhtakia and Weiglhofer.**

I. Introduction

Bianisotropic materials have a more complicated response to electricity than ordinary isotropic materials. The polarization behavior of bianisotropic media is contained in the constitutive relations between the electric ($\bar{D}$) and magnetic ($\bar{B}$) displacements, and the electric ($\bar{E}$) and magnetic ($\bar{H}$) field vectors

$$\bar{D} = \overline{\overline{\epsilon}} \cdot \bar{E} + \overline{\overline{\xi}} \cdot \bar{H} \tag{1}$$
$$\bar{B} = \overline{\overline{\zeta}} \cdot \bar{E} + \overline{\overline{\mu}} \cdot \bar{H}. \tag{2}$$

Here the material parameter dyadics are permittivity $\overline{\overline{\epsilon}}$, permeability $\overline{\overline{\mu}}$, and the magnetoelectric crosspolarisations $\overline{\overline{\xi}}$ and $\overline{\overline{\zeta}}$. Nonisotropy makes the four material parameter relations dyadic with nine components in general, and the total number of independent material parameters of bianisotropic media is hence 36.

The material parameters are limited by principles dictated by physical laws. For example, the well-known causality requirement leads to the Kronig–Kramers equations. And since the results of bianisotropic electromagnetics are relevant to the growing number of microwave applications of complex materials, it is evermore important to find out and articulate physical restrictions for the magnetoelectric parameters.

A recent publication [1] provides us with an argument that gives one restriction to the 36 parameters for linear bianisotropic media. Lakhtakia and Weiglhofer use tensor analysis and the general covariance requirements given by Post [2] for uniform media and structural fields, and the claim resulting from their analysis is strong: the sum of the traces of the magnetoelectric dyadics must vanish. Applied to the special case of bi-isotropic media[1], the conclusion of the analysis in [1] is that isotropic materials have to be reciprocal. In other words, the claim stands that NRBI (nonreciprocal bi-isotropic materials) cannot exist.

It is the purpose of this article to discuss possible counterexamples to this NRBI-nonexistence result. For that end, let us reformulate the constitutive relations (1)–(2) into a form where reciprocity becomes visible

$$\bar{D} = \overline{\overline{\epsilon}} \cdot \bar{E} + (\overline{\overline{\chi}}^T - j\overline{\overline{\kappa}}^T)\sqrt{\mu_0\epsilon_0} \cdot \bar{H} \tag{3}$$
$$\bar{B} = (\overline{\overline{\chi}} + j\overline{\overline{\kappa}})\sqrt{\mu_0\epsilon_0} \cdot \bar{E} + \overline{\overline{\mu}} \cdot \bar{H} \tag{4}$$

where now the magnetoelectric parameters are contained in the chirality dyadic $\overline{\overline{\kappa}}$ and the nonreciprocity dyadic $\overline{\overline{\chi}}$. The superscript T denotes the transpose operation.[2] The nonreciprocity decomposition (3)–(4) is in accord with the reciprocity definition for bianisotropic media [4]

$$\overline{\overline{\epsilon}} = \overline{\overline{\epsilon}}^T, \quad \overline{\overline{\mu}} = \overline{\overline{\mu}}^T, \quad \overline{\overline{\xi}} = -\overline{\overline{\zeta}}^T, \quad \text{(for reciprocal media).} \tag{5}$$

Bi-isotropic media have material dyadics that are multiples of a unit dyadic $\overline{\overline{I}}$. The well-known isotropic chiral medium has dyadics $\overline{\overline{\kappa}} = \kappa\overline{\overline{I}}$ and $\overline{\overline{\chi}} = 0$ [3]. And in particular, the nonreciprocity dyadic of a sample of NRBI material is of the form $\overline{\overline{\chi}} = \chi\overline{\overline{I}}$, where $\chi \neq 0$.

II. Consequences of the Magnetoelectric Tracelessness

The crucial result of [1] is a condition for the trace of the magnetoelectric dyadics. In particular, it restricts the nonreciprocal part of these dyadics with the condition

$$\mathrm{tr}\{\overline{\overline{\chi}}\} = 0 \tag{6}$$

where trace means the sum of the diagonal elements of the dyadic. Note that the constitutive relations used here relate the pair $(\bar{D}, \bar{B})$ to the pair $(\bar{E}, \bar{H})$ whereas [1] follows the "Boys–Post" relations where $\bar{D}$ and $\bar{H}$ are given as material functions of the primary fields $\bar{E}$ and $\bar{B}$. However, the magnetoelectric parameters have the same

[1] For bi-isotropic media, the four dyadics reduce to scalars since there is no direction dependence in the medium. The magnetoelectric coupling remains through two parameters.

[2] In (3)–(4), the additional coefficients (the imaginary unit j and the free-space parameters μ_0, ϵ_0) have been included for conformity with earlier notation [3].

Manuscript received July 7, 1994; revised May 25, 1995.
The author is with the Helsinki University of Technology, Electromagnetics Laboratory, FIN-02150 Espoo, Finland.
IEEE Log Number 9413413.