The Use of SPICE Lumped Circuits as Sub-grid Models for FDTD Analysis

Vincent A. Thomas, Michael E. Jones, Melinda Piket-May,
Allen Taflove, and Evans Harrigan, *Associate Member, IEEE*

***Abstract*— A general approach for including lumped circuit elements in a finite difference, time domain (FDTD) solution of Maxwell's equations is presented. The methodology allows the direct access to SPICE to model the lumped circuits, while the full 3-Dimensional solution to Maxwell's equations provides the crosstalk and dispersive properties of the microstrips and striplines in the circuit.**

I. Introduction

FDTD solution of Maxwell's equations has many applications [1]. Recently in [2], a method was proposed that treats a few circuit elements such as resistors, capacitors, inductors, diodes, and transistors as subgrid models on the FDTD grid. A three-dimensional implementation with improvements was presented in [3]. Another example of a more complicated active elment used in FDTD is presented in [4]. Here, we present an extension of these works that allows direct access to all SPICE models (via SPICE itself) for simulation of lumped circuits in FDTD calculations. This is a more general approach than the harmonic balance method [5], which has been used previously for treating the problem of nonlinear elements coupled to linear networks.

II. Finite Difference Equations

The coupling of SPICE with FDTD relies upon the use of a different way to obtain the difference form of Ampere's law

$$\epsilon\frac{\partial \mathbf{E}}{\partial t} + \mathbf{J}(\mathbf{E}) = \nabla \times \mathbf{H}. \tag{1}$$

The difference equation is obtained by treating (1) as an ordinary differential equation in time with the right hand side constant, but in general keeping the time dependence for $\mathbf{J}(\mathbf{E})$. The integration is performed for a timestep and connects the electric field values at two time steps. This is the same idea expressed for the time variable as the approach given in [6] for the spatial variables in device simulations. For many simple cases (the resistor, the capacitor, the inductor, and the diode), this equation can be integrated analytically. In some regimes, this treatment shows better stability and accuracy properties than methods found in [2] and [3], whereas no regimes were found where the reverse is true.

For general circuit elements $\mathbf{J}$ is a complicated nonlinear function of the electric field and its derivatives, and neighboring values, and analytic results would not be possible.

III. The SPICE Connection

The circuit simulator SPICE gives the current through a circuit element as a function of the voltage across the device. Thus, in effect, it can be used to give $\mathbf{J}$ as a function of the electric field $\mathbf{E}$. This value of $\mathbf{J}$ could be used in (1) and a separate numerical integration could be done to provide the difference equation for Ampere's law.

However, an even simpler and probably more robust prescription can be obtained by rewriting (1) as

$$C\frac{dV}{dt} + I(V) = I \tag{2}$$

where V is the voltage across the device, $C = \epsilon A/dx$ is a capacitance (A is the area of the finite difference cell, and dx is its height), $I(V)$ $(= AJ(E)$) represents the current flowing through the lumped circuit, and I represents the total current $A\nabla \times \mathbf{H}^{\mathbf{n+1/2}}$. Equation (2) can be represented as the equivalent circuit shown in Fig. 1(a). Thus, instead of using SPICE just to determine $\mathbf{J}(\mathbf{E})$, SPICE can be used to integrate (2) directly. The lumped element may be an arbitrarily large SPICE circuit whose description may be contained in a standard SPICE file. The extensive device models in SPICE can be used directly in the FDTD simulation without the need to duplicate the model development, and the efficient circuit integration methods used in SPICE are also directly available without user implemented integration schemes. The FDTD and SPICE computer programs may be coupled using various interprocess communication techniques.

IV. Circuit Examples and SPICE Comparisons

Comparisons have been made between coupled FDTD-SPICE and complete SPICE simulations for various types of lumped circuits placed at the end of a microstrip line in three dimensions. Calculations of microstrip lines terminated with individual resistors, capacitors, inductors, and diodes have shown good agreement with SPICE calculations under appropriate conditions. Calculations with microstrip lines terminated with more complicated, multi-element lumped circuits have also yielded good agreement with SPICE.

Manuscript received December 20, 1993. This work was performed under the auspices of the U.S. Dept. of Energy.

V. A. Thomas and M. E. Jones are with Los Alamos National Laboratory, Los Alamos, NM 87545 USA.

M. Piket-May is with the Department of Electrical Engineering, University of Colorado, Boulder, CO USA.

A. Taflove is with the Department of Electrical Engineering+, Northwestern University, Evanston, IL USA.

E. Harrigan is with Cray Research, Inc., Huntington Beach, CA USA.

IEEE Log Number 9401065.

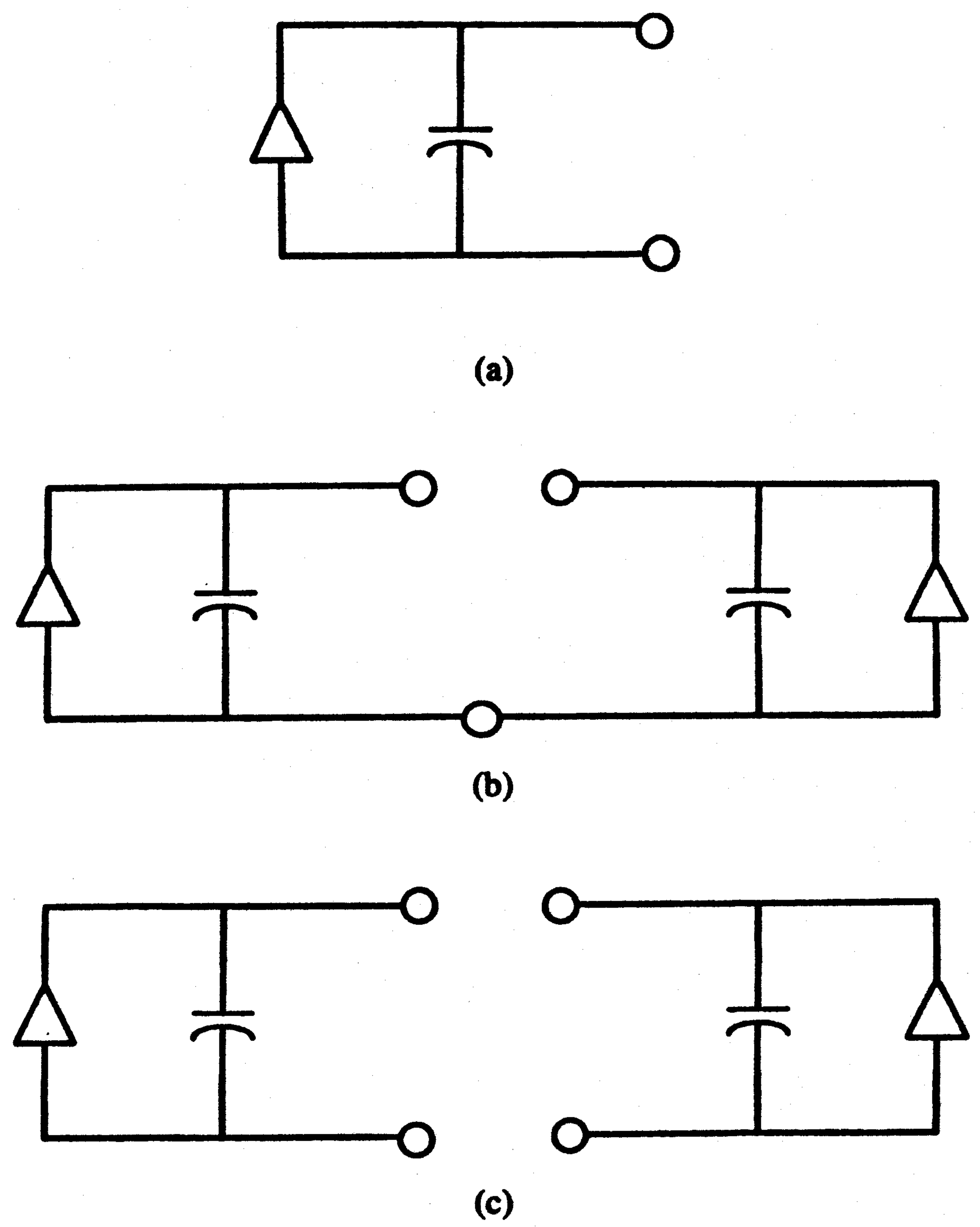


Fig. 1. Equivalent SPICE circuits for Ampere's law. The lumped circuit is included between the nodes. (a) Two-node circuit. (b) Three- node circuit. (c) Four-node circuit.

Modifications must be made for the more general lumped circuit, which is a two-port device. To investigate this more general situation, we have tried modeling the circuit depicted in Fig. 2. The lumped circuit consists of a dc voltage source, several capacitors, several inductors, and a transistor. The resistive voltage source and the load resistor could be modeled with SPICE, but they are modeled with the analytic integration of (1). The equivalent circuit for a time advance of the electric fields connecting the SPICE lumped circuit to the FDTD calculation is shown in Fig. 1(b). The calculation is carried out in a three-dimensional FDTD model with a ground plane, a lumped resistive voltage source at the end of one microstrip, the lumped circuit connecting the first and second microstrips, and then the second microstrip terminated by a matched load. In this implementation there may be several cells between the source microstrip line and the load microstrip line. This violates causality, since the field at one location is related to the instantaneous value of a field at another location. However, the time delay between the lines is assumed to be so small that the electric fields at the microstrip lines are treated as a lumped circuit. There are 30,000 time steps in the calculation, $c\Delta t = 0.5\Delta x$, uniform gridding, outgoing radiation boundary conditions are used, and the relative dielectric constant is unity everywhere. The wavelength is much larger than microstrip dimensions, and so good agreement between the coupled FDTD-SPICE calculation and the SPICE calculation should be obtained.

Fig. 3 shows the voltage at the input to the lumped transistor circuit and the voltage across the load resistor for SPICE and the coupled FDTD-SPICE algorithm. For this calculation, the

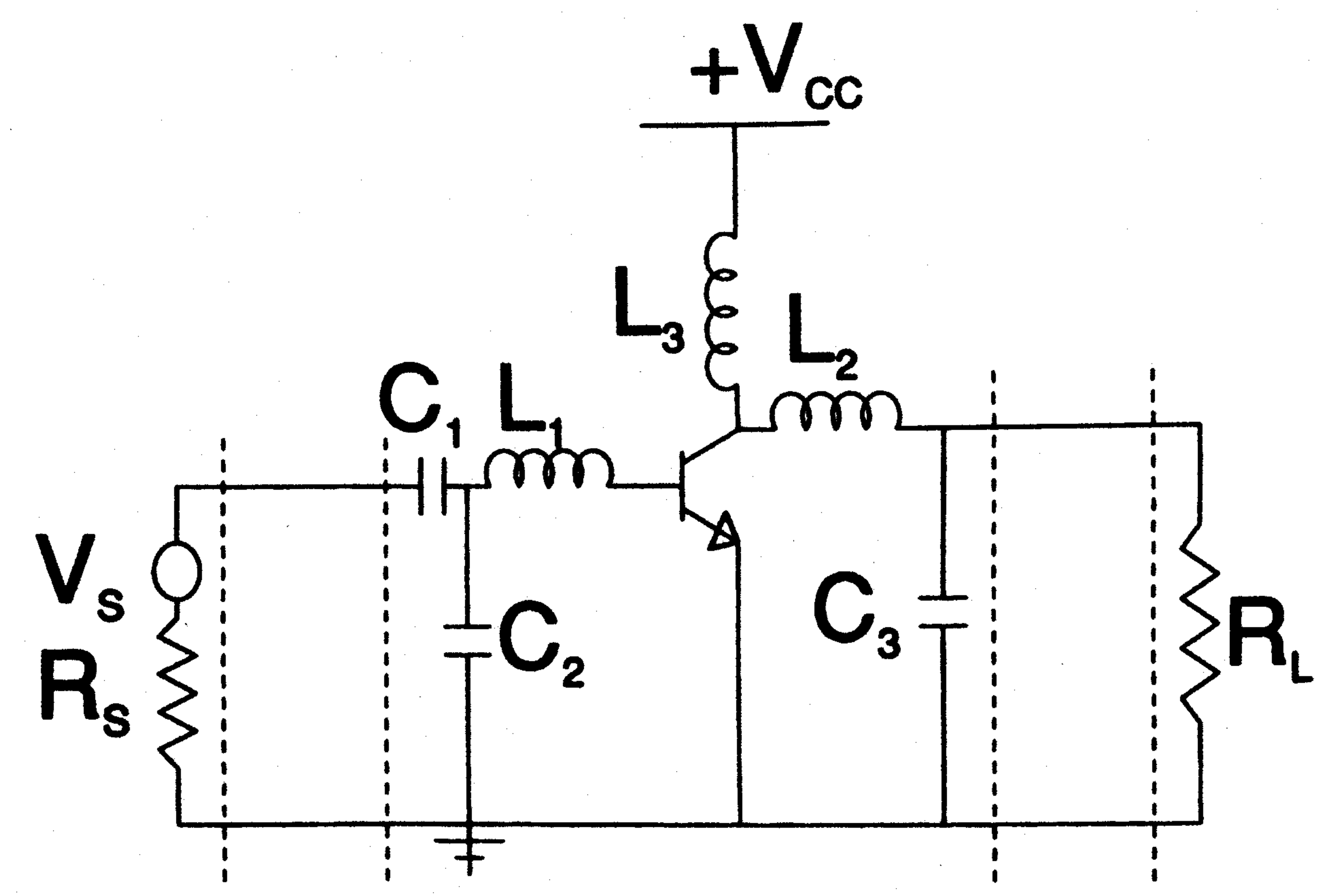


Fig 2. A simplified VHF amplifier circuit. Here $C_1 = 77pF$, $C_2 = 10pF$,$L_1 = 82nH$, $L_3 = 23nH$, $L_2 = 50nH$, $C_3 = 46pF$, $V_{CC} = 9$ volts, and default values are used for the bjt with a collector emitter capacitance of $60pF$, the impedances of the microstrip lines are 47.9 Ohms, the load resistor and the voltage source resistor each have a value to match the transmission lines, the voltage source is a sine wave with amplitude 1.3 V with frequency 137 MHz, and the delay times of the two transmission line segments is 0.16667 ns and 0.09667 ns, respectively.

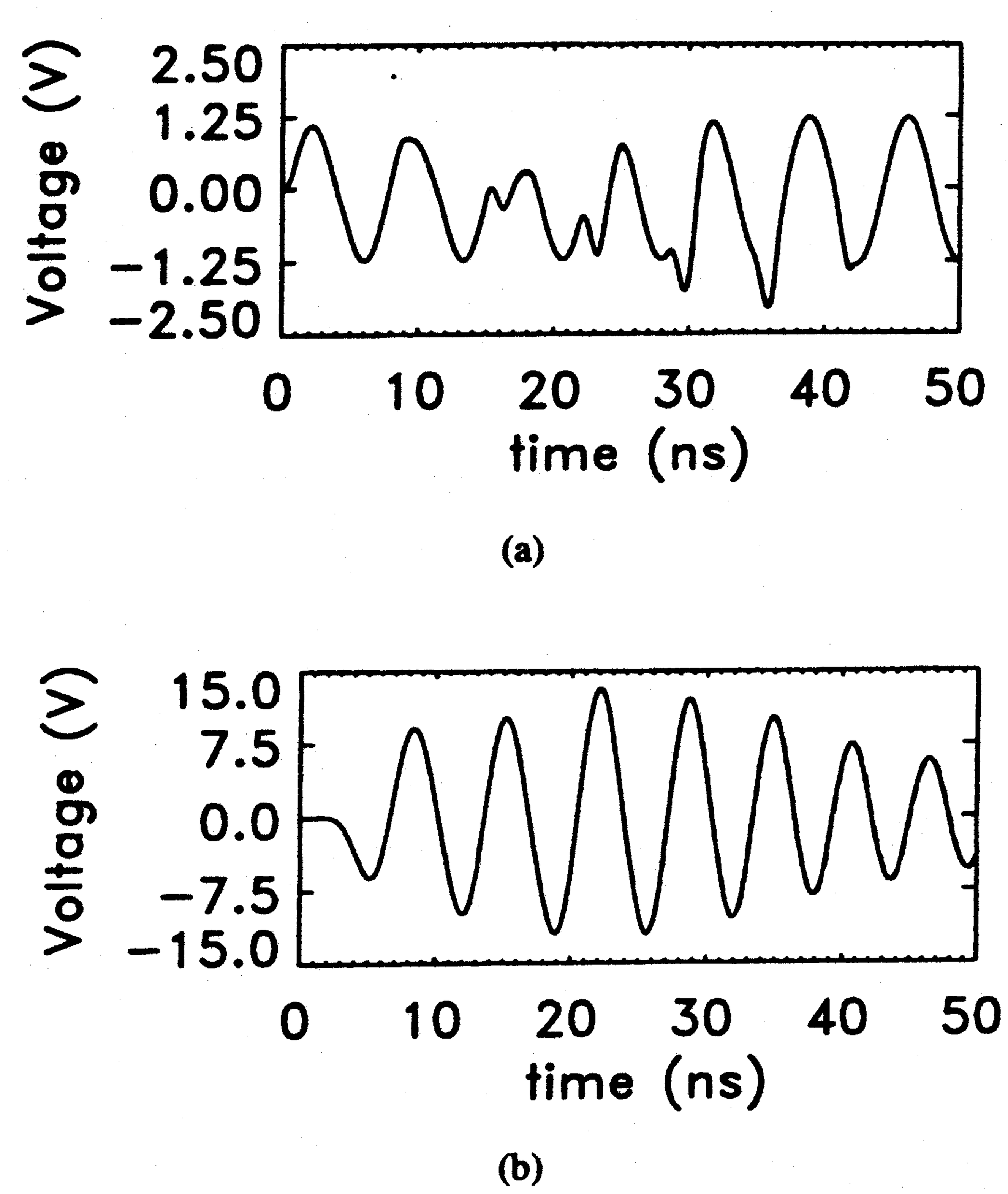
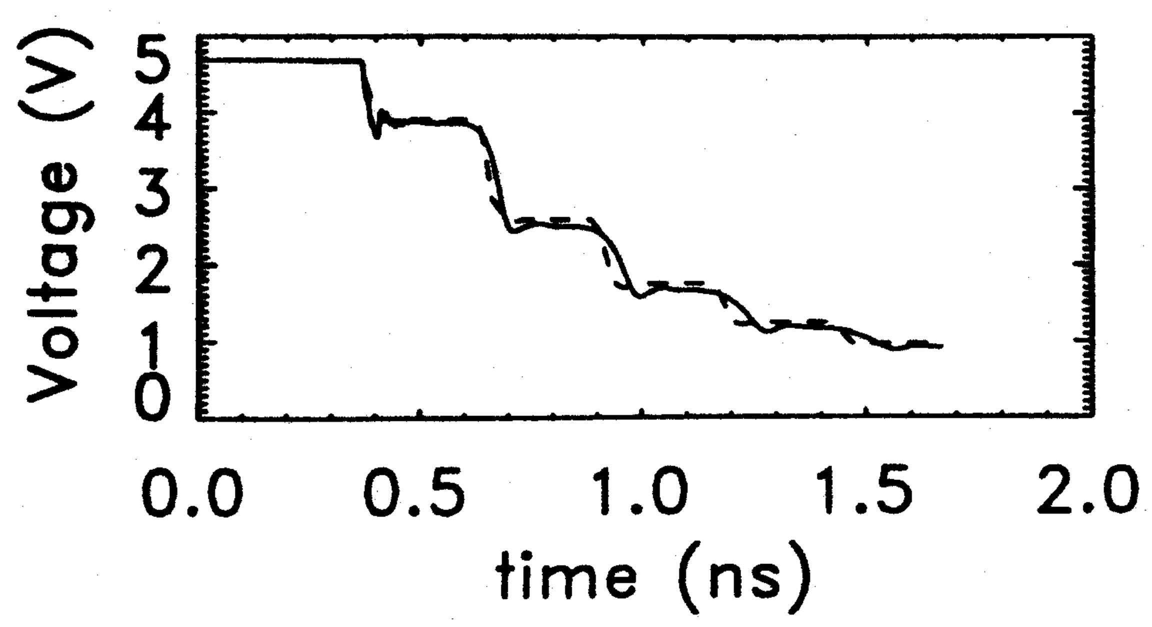


Fig 3. The voltage at (a) the input to the lumped transistor circuit and (b) across the output resistor with SPICE alone (dashed lines) and FDTD-SPICE (solid lines).

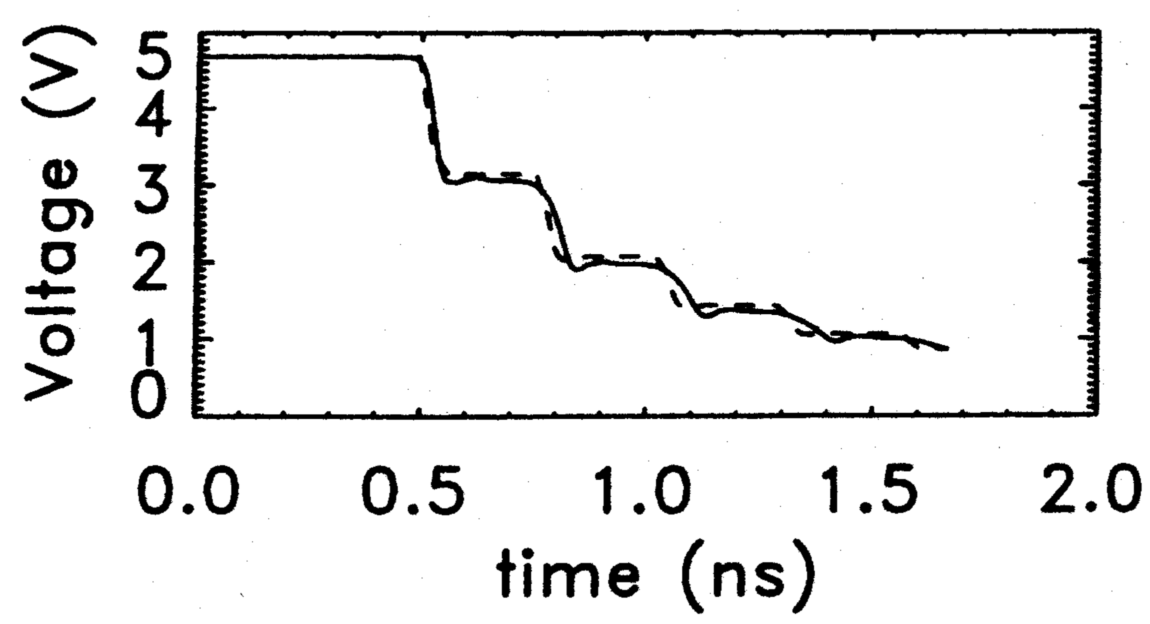
lines were separated by 2.5 times the width of the microstrip. The agreement is excellent. Poorer agreement is obtained as the separation of the microstrip lines is made comparable to their width, which is presumably due to parasitic coupling.

In another test, the coupled FDTD-SPICE code was used to simulate avoltage source connected to an inverter by a microstrip, which is in turn connected by a microstrip to another inverter. For these simulations, a simple TTL inverter is used that consists of two bjt transistors, two resistors, and a power supply. The FDTD model is similar to that described for the previous simulation.

Qualitative agreement with the SPICE simulations is achieved as shown in Fig. 4, but it is not as good as the



(a)



(b)

Fig 4. The voltage at (a) the output of the first inverter and at (b) the input of the second inverter with SPICE alone (dashed lines) and FDTD-SPICE (solid lines).

previous case. This is possibly related to the higher frequencies present in these simulations. The maximum frequencies are determined by the rise time of the input voltage (which is a ramped square wave with a rise time of about 100ps) and the rise time of the output of the first inverter (which is about 50ps for the parameters used in this simulation). The highest frequency components of the pulse are not highly resolved resolved by the finite difference grid (there are 1000 time steps in the total simulation).

The still more general four-node lumped circuit is depicted in Fig. 1(c). For this case, the lumped circuit still related electric fields at two different locations. However, each node may possess a unique voltage. This method may be extended to an arbitrarily large number of nodes.

V. Conclusions

A simple and effective approach has been developed for coupling SPICE lumped elements into FDTD calculations. Test calculations confirm the approach. This technique has also been extended to couple PISCES device simulations directly with the electromagnetic solver. In this way the work in [7] can be extended to include not only SPICE and PISCES devices, but also the electromagnetic field calculations.

References

[1] A. Taflove, "Basis and Application of Finite-Difference Time-Domain (FD-TD) Techniques for Modeling Electromagnetic Wave Interactions," (short course notes) *1992 IEEE Antennas and Propagation Society Int. Symp. and URSI 21 Radio Science Meeting*, Chicago, IL, July, 1992.

[2] W. Sui, D. A. Christensen, and C. H. Durney, "Extending the two-dimensional FD-TD method to hybrid electromagnetic systems with active and passive lumped elements," *IEEE Trans. Microwave Theory Tech.*, vol. 40, pp 724–730, Apr. 1992.

[3] M. Piket-May, A. Taflove, and J. Baron, "FD-TD modeling of digital signal propagation in 3-D circuits with passive and active loads," submitted to *IEEE Trans. Microwave Theory Tech.*

[4] B. Toland, J. Lin, B. Houshmand, and T. Itoh, "FDTD analysis of an active antenna," *IEEE Microwave and Guided Wave Lett.*, vol. 3, pp. 423–425, Nov. 1993.

[5] I. Wolff, "Finite difference time domain simulation of electromagnetic fields and microwave circuits," *Int. J. Numerical Modeling*, vol. 5, pp 163–182, 1992.

[6] D. L. Scharfetter and H. K. Gumel, "Large-signal analysis of a silicon Read diode oscillator," *IEEE Trans. Electron Dev.*, vol. ED-16, pp. 64–77, Jan. 1969.

[7] J. Geogory Rollins and John Choma, Jr., "Mixed-mode PISCES-SPICE coupled circuit and device solver," *IEEE Trans. Computer-Aided Design*, vol. 7, pp 862–867, Aug. 1988.