Extending SPICE with FDTD and Generalized Characteristic Model for Coupled Microstrip Lines Simulation

S. K. IP† and F. Y. Chang‡

†iTec Pro Ltd., P.O. Box No.68563, Kowloon East Post Office, Hong Kong
‡ Department of Communication Engineering, National Chiao Tung University, Taiwan, ROC

Abstract — **The implementation of an efficient full wave enabled SPICE simulator is presented. The generalized characteristic model has been integrated into FDTD-SPICE simulator so that full wave analysis with lumped terminal devices is much faster. Example with coupled microstrip lines terminated with diodes showed that it is 75% faster than FDTD-SPICE analysis.**

I. INTRODUCTION

With the ever-increasing density of integrated circuits, operating speed approaches to tens of gigabits per seconds. Interconnects consisting of parallel conductors embedded in dielectric media become the dominating factor in determining circuit performance. In order to guarantee circuit performances, interconnect effects such as reflection, dispersion, attenuation and crosstalk have to be modeled accurately.

The finite-difference time-domain (FDTD) method is a well-developed full-wave solution to model various electromagnetic phenomena and interactions [1]. It had been successfully extended to include lumped circuit elements modeled by a general circuit simulator such as SPICE [2]. Although this approach is versatile, FDTD is time consuming and requires high computational cost. If the source or loads change, the calculation must be carried out again, thus wasting time.

In 1997, time-domain characteristic model [3] was successfully applied to synthesize microstrip line. This generalized characteristic model (GCM) composed of transient characteristic impedance and propagation functions can be extracted from terminal response voltages simulated by FDTD. With this model, the response of any excitation of microstrip line with any loads can be rapidly simulated.

In this paper, we present an integration of SPICE, FDTD, and GCM such that terminal responses can be obtained by inputting a single circuit file. The coupling of GCM and SPICE is accomplished by resistors and voltage sources. Through add-on directives, user may request the simulator to generate GCM model via a FDTD analysis. When the GCM model has already been generated, it can be applied to simulate the terminal responses directly and the time consuming FDTD analysis can be skipped.

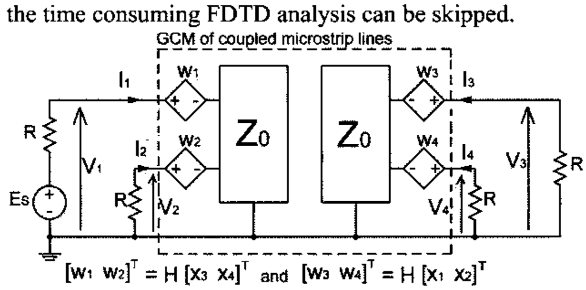

Fig. 1. Network configuration for extraction of characteristic functions, z_{11}, z_{12}, h_{11} and h_{12}. All labels are for Laplace domain.

II. GENERALIZED CHARACTERISTIC MODEL EXTRACTION

Similar to model extraction for a single microstrip line [4], coupled microstrip lines can be represented by a four-port network as shown in Fig. 1. The GCM of the network can be written for the Laplace domain as

$$\mathbf{V_a} = \mathbf{Z_0}\,\mathbf{I_a} + \mathbf{W_a} \tag{1}$$
$$\mathbf{V_b} = \mathbf{Z_0}\,\mathbf{I_b} + \mathbf{W_b} \tag{2}$$
$$\mathbf{W_a} = \mathbf{H}\,(\mathbf{V_b} + \mathbf{Z_0}\,\mathbf{I_b}) \tag{3}$$
$$\mathbf{W_b} = \mathbf{H}\,(\mathbf{V_a} + \mathbf{Z_0}\,\mathbf{I_a}) \tag{4}$$

where

$$\mathbf{V_a} = [V_1 \;\; V_2]^T; \quad \mathbf{V_b} = [V_3 \;\; V_4]^T;$$
$$\mathbf{I_a} = [I_1 \;\; I_2]^T; \quad \mathbf{I_b} = [I_3 \;\; I_4]^T;$$
$$\mathbf{W_a} = [W_1 \;\; W_2]^T; \quad \mathbf{W_b} = [W_3 \;\; W_4]^T;$$
$$\mathbf{Z_o} = \begin{bmatrix} Z_{11} & Z_{12} \\ Z_{12} & Z_{11} \end{bmatrix}; \quad \mathbf{H} = \begin{bmatrix} H_{11} & H_{12} \\ H_{12} & H_{11} \end{bmatrix}.$$

$\mathbf{Z_o}$ and $\mathbf{H}$ are the input impedance function matrix and the propagation function matrix of the coupled lines, respectively. Transformed into the time domain [1], (1) becomes

[1] We use uppercase and lowercase letters to represent Laplace domain and time domain functions, respectively.

0-7803-7695-1/03/$17.00 © 2003 IEEE

$$v_1 = z_{11}*i_1 + z_{12}*i_2 + w_1 \quad (5)$$

$$v_2 = z_{12}*i_1 + z_{11}*i_2 + w_2 \quad (6)$$

where

$$w_1 = h_{11}*(v_3 + z_{11}*i_3 + z_{12}*i_4) + h_{12}*(v_4 + z_{12}*i_3 + z_{11}*i_4);$$

$$w_2 = h_{12}*(v_3 + z_{11}*i_3 + z_{12}*i_4) + h_{11}*(v_4 + z_{12}*i_3 + z_{11}*i_4).$$

$*$ is the convolution operator. As the transient terminal responses are sampled values, we have to use discrete operations. Suppose that

$$y(t) = x(t) * p(t) = \int_0^t x(t-\lambda)x(\lambda)d\lambda \quad (7)$$

The discrete-time convolution is

$$y[n] = \sum_{k=0}^{n} p[k]x[n-k] = \sum_{k=0}^{n} p[n-k]x[n] \quad (8)$$

where y[n], x[n] and p[n] are sampled data given by $y(n\Delta t)$, $x(n\Delta t)$ and $p(n\Delta t)$, respectively, for $n = 0, 1, 2, \dots$. (5) can be written in discrete-time form

$$v_1[n] = \sum_{k=0}^{n} z_{11}[k]i_1[n-k] + \sum_{k=0}^{n} z_{12}[k]i_2[n-k] + w_1[n] \quad (9)$$

To extract the characteristic functions, a unit-step excitation source E_S with resistance R is applied at port 1 while the other ports are terminated with the same resistance R. A single FDTD-SPICE simulation is adopted to obtain the terminal responses. Thus, V_1, V_2, V_3 and V_4 are known. The port currents can be obtained by

$$I_1 = \frac{E_S - V_1}{R}; I_2 = \frac{-V_2}{R}; I_3 = \frac{-V_3}{R}; I_4 = \frac{-V_4}{R}. \quad (10)$$

Due to the delay introduced by the finite propagation velocity along the distributed structure, we would observe terminal voltages emerge at port 2, port 3 and port 4 sequentially. Suppose that $k_2\Delta t$, $k_3\Delta t$, and $k_4\Delta t$ are the delay times for the voltage emerging at the corresponding port. We have

$$v_2[n] = 0;\ i_2[n] = 0; \qquad \text{for } n < k_2$$
$$v_3[n] = 0;\ i_3[n] = 0; \qquad \text{for } n < k_3$$
$$v_4[n] = 0;\ i_4[n] = 0; \qquad \text{for } n < k_4$$

and hence

$$w_1[n] = 0;\ w_2[n] = 0; \qquad \text{for } n < k_3$$

By removing those zero terms from (9), $z_{11}[n]$ can be obtained by

$$z_{11}[n] = (v_1[n] - s) / i_1[0] \quad (11)$$

where

$$s = \sum_{k=0}^{n-1} z_{11}[k]i_1[n-k] + \sum_{k=k_2}^{n} z_{12}[n-k]i_2[k] + w_1[n].$$

When the upper limit of summation is less than the lower limit, the term vanishes. Similarly, we can derive the equation for $z_{12}[n]$, $h_{11}[n]$ and $h_{12}[n]$, subsequently. The complete algorithm for extracting GCM of coupled lines is as follows:

Step 1: calculate currents using (10),

$$i_1[n] = (e_S[n] - v_1[n])/R, \quad i_2[n] = -v_2[n]/R,$$
$$i_3[n] = -v_3[n]/R, \quad i_4[n] = -v_4[n]/R$$

Step 2: calculate $z_{11}[n]$ and $z_{12}[n]$ as follows:

$$x_3[n] = v_3[n] + \sum_{k=k_3}^{n} z_{11}[n-k]i_3[k] + \sum_{k=k_4}^{n} z_{12}[n-k]i_4[k]$$

$$x_4[n] = v_4[n] + \sum_{k=k_3}^{n} z_{12}[n-k]i_3[k] + \sum_{k=k_4}^{n} z_{11}[n-k]i_4[k]$$

$$w_1[n] = \sum_{k=k_3}^{n} h_{11}[n-k]x_3[k] + \sum_{k=k_3}^{n} h_{12}[n-k]x_4[k]$$

$$w_2[n] = \sum_{k=k_3}^{n} h_{12}[n-k]x_3[k] + \sum_{k=k_3}^{n} h_{11}[n-k]x_4[k]$$

$$s = \sum_{k=0}^{n-1} z_{11}[k]i_1[n-k] + \sum_{k=k_2}^{n} z_{12}[n-k]i_2[k] + w_1[n]$$

$$z_{11}[n] = (v_1[n] - s) / i_1[0]$$

$$s = \sum_{k=0}^{n-1} z_{12}[k]i_1[n-k] + \sum_{k=k_2}^{n} z_{11}[n-k]i_2[k] + w_2[n]$$

$$z_{12}[n] = (v_2[n] - s) / i_1[0] \qquad (\text{i.e. } z_{12}[n] = 0 \text{ for } n < k_2)$$

Step 3: calculate $h_{11}[n]$ and $h_{12}[n]$ as follows:

$$x_1[n] = v_1[n] + \sum_{k=0}^{n} z_{11}[n-k]i_1[k] + \sum_{k=k_2}^{n} z_{12}[n-k]i_2[k]$$

$$x_2[n] = v_2[n] + \sum_{k=k_2}^{n} z_{12}[n-k]i_1[k] + \sum_{k=k_2}^{n} z_{11}[n-k]i_2[k]$$

$$u_3[n] = \sum_{k=1}^{n} h_{11}[n-k]x_1[k] + \sum_{k=k_2}^{n} h_{12}[n-k]x_2[k]$$

$$u_4[n] = \sum_{k=1}^{n} h_{12}[n-k]x_1[k] + \sum_{k=k_2}^{n} h_{11}[n-k]x_2[k]$$

$$s = \sum_{k=k_3}^{n} z_{11}[n-k]i_3[k] + \sum_{k=k_4}^{n} z_{12}[n-k]i_4[k] + u_3[n]$$

$$h_{11}[n] = (v_3[n] - s) / x_1[0]$$

$$s = \sum_{k=k3}^{n} z_{12}[n-k]i_3[k] + \sum_{k=k_4}^{n} z_{11}[n-k]i_4[k] + u_4[n]$$

$$h_{12}[n] = (v_4[n] - s) / x_2[0]$$

Step 4: $n = n + 1$. Go to step 1 if $n < N$.

III. The Coupling of GCM and SPICE

After the characterization of the terminal behavior of coupled lines, a coupling circuit that integrates GCM model into SPICE is derived. From the characterization algorithm mentioned in the last section, $w_1[n] = 0$, for $n < k_3$ and $z_{12} = 0$, for $n < k_2$. (9) can be written into

$$v_1[n] = z_{11}[0]i_1[n] + u_1 \tag{12}$$

where

$$u_1 = \sum_{k=1}^{n} z_{11}[k]i_1[n-k] + \sum_{k=k_2}^{n} z_{12}[k]i_2[n-k] + w_1[n].$$

(12) shows that the equivalent circuit looking into port 1 can be modeled by a Thévenin equivalent circuit. Similarly, the terminal equations for port 2, port 3 and port 4 can be derived as

$$v_m[n] = z_{11}[0]i_m[n] + u_m \tag{13}$$

where the subscript m is also the port number and

$$u_2 = \sum_{k=k_2}^{n} z_{12}[k]i_1[n-k] + \sum_{k=1}^{n} z_{11}[k]i_2[n-k] + w_2[n]$$

$$u_3 = \sum_{k=1}^{n} z_{11}[k]i_3[n-k] + \sum_{k=k_2}^{n} z_{12}[k]i_4[n-k] + w_3[n]$$

$$u_4 = \sum_{k=k_2}^{n} z_{12}[k]i_3[n-k] + \sum_{k=1}^{n} z_{11}[k]i_4[n-k] + w_4[n].$$

The coupling circuit is illustrated in-side the doted-line box in Fig. 2. At each time step, the time dependent voltage sources, u_m, are calculated from the previous terminal voltages and currents. SPICE is then used to analyze this equivalent circuit together with the attached lumped element to obtain the terminal responses.

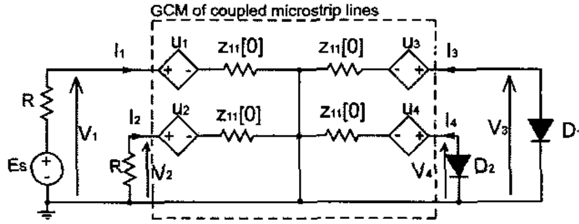

Fig. 2. The coupling of GCM and SPICE.

IV. The Implementation of Extending SPICE with FDTD and GCM

The description statements of microstrip lines, which start with *!, have been added to the syntax of SPICE. As SPICE treats a statement beginning with * as a comment line, the new circuit files are compatible to the original circuit files. In addition, devices for FDTD current source described in [2] and for GCM voltage source described by (13) are added.

User can specify the method of analysis for microstrip lines by a control statement.

**! Analysis = FDTD*

specifies that FDTD would be used to obtain the terminal behavior of the microstrip lines.

**! Analysis = GCM : z.dat, h.dat*

specifies that GCM would be used to obtain the terminal behavior of the microstrip lines. If the files z.dat and h.dat do not exist, simulator will run a characterization analysis first, which will replace the terminal lumped elements by the characterization terminal setting shown in Fig. 1 with $R = 50\Omega$ driven by a unit-step voltage source. And then a GCM analysis with the original terminal lumped elements and GCM–SPICE coupling model as shown in Fig. 2 will be executed. If the files exist, however, the characterization analysis will be skipped and the GCM simulation will be run directly.

V. Example

To illustrate the efficiency of this simulator we consider two coupled microstrip lines as an example. The two microstrip lines have a width of $6\Delta x$ and length of $50\Delta y$. The separation between the two lines is $2\Delta x$. The lines are printed on a dielectric substrate having thickness of $3\Delta z$ with a relative permittivity ε_r of 2.2. The configuration of nonlinear load terminated coupled lines is shown in Fig. 2, in which diodes having 10^{-16}A of reverse saturation current at room temperature (300K) are terminated at port

3 and port 4. The resistors at port 1 and port 2 are both 50Ω. The excitation source is a 2.2676GHz sinusoidal waveform with 20V peak-to-peak value. The description statements of the coupled microstrip lines are as follows:

```
Coupled microstrip lines terminated by diodes
*! Domain Dimensions:      ;specify the substrate's size
*!   X = 38                ;  38 delta-x
*!   Y = 74                ;  74 delta-y
*!   Z = 13                ;  13 delta-z
*!   e_r = 2.2             ;  relative permittivity = 2.2
*! Layer Height = 3        ;thickness of the substrate
*! Strip No. = 1           ;the first microstrip line
*! Strip Edges to Dielectric Substrate Edges:
*!   edge2sub = 12         ;the edge to substrate edge
*!   head2sub = 12         ;the near end to substrate edge
*! Strip Dimensions:
*!   w = 6                 ;width of the microstrip line
*!   l = 50                ;length of the microstrip line
*! Strip No. = 2           ;the second microstrip line
*! Strip Edges to Dielectric Substrate Edges:
*!   edge2sub = 20         ;the edge to substrate edge
*!   head2sub = 12         ;the near end to substrate edge
*! Strip Dimensions:
*!   w = 6                 ;width of the microstrip line
*!   l = 50                ;length of the microstrip line
*! Strip End               ;end of line description
*! Computational Domain:
*!   deltax = 0.4233e-3
*!   deltay = 0.4046e-3
*!   deltaz = 0.2650e-3
*!   deltat = 0.441e-12
*! Analysis = GCM : z.dat, h.dat
```

SPICE version 3f5 was modified to include both FDTD and GCM analyses in a Pentium IV computer at 1.6GHz and 512Mbyte of memory under Microsoft Windows XP environment. With a time step Δt of 0.441ps, $4000\Delta t$, i.e. 1.764ns, was simulated. The comparisons of the terminal voltages to the direct FDTD results are shown in Fig. 3.

The FDTD characterization requires 1347s while GCM analysis requires 337s. Total simulation time required is 1684s. For a direct FDTD full-wave simulation, the simulation time is 1352s. It is clear that for a single run direct FDTD simulation is faster than FDTD characterization plus GCM simulation. For a second run with terminal devices changed, however, the time consuming FDTD characterization can be skipped and the computational time taken by GCM simulation is about 75% faster than direct FDTD simulation.

V. Conclusion

We have successfully extended SPICE version 3f5 to include both FDTD and GCM analyses. The efficiency of the simulator has been demonstrated by a coupled microstrip lines terminated with diodes. When the circuit elements are modified and re-simulation is performed, the computation is speeded up by approximately 75%. Good agreement between the GCM simulation and direct FDTD simulation has confirmed the accuracy of this approach.

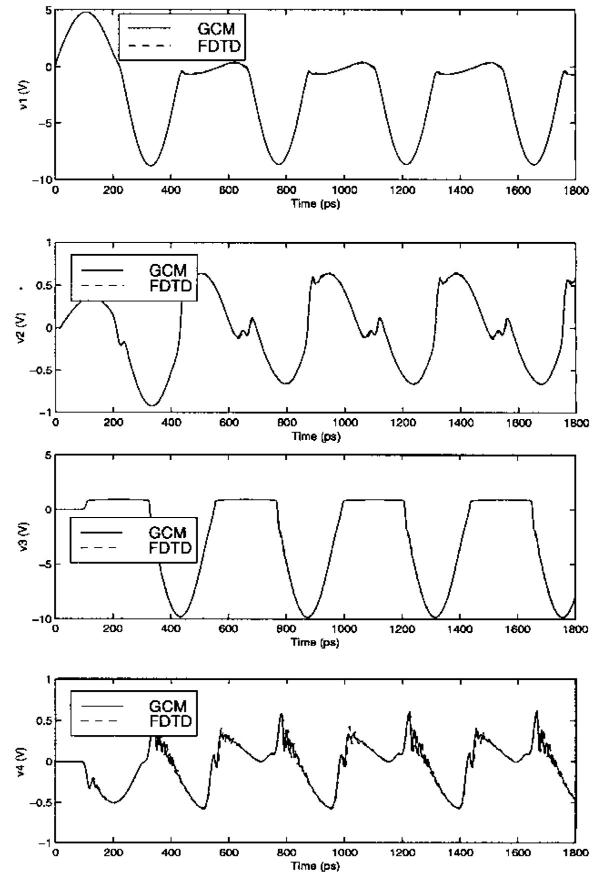

Fig. 3. Comparisons of the terminal response voltages.

References

[1] A. Taflove, "Review of the formulation and application of the finite-difference time-domain method for numerical modeling of electromagnetic wave interactions with arbitrary structures," *Wave Motion*, vol. 10, pp. 547-582, December 1988.

[2] V. A. Thomas, M. E. Jones, M. Piket-May, A. Taflove, and E. Harrigan, "The use of SPICE lumped circuits as sub-grid models for FDTD analysis," *IEEE Microwave Guided Wave Lett.*, vol. 4, no. 5, pp. 141-143, May 1994.

[3] Q. X. Chu, F. Y. Chang, Y. P. Lau, and O. Wing, "Time-domain model synthesis of microstrip," *IEEE Microwave Guided Wave Lett.*, vol. 7, no. 1, pp. 9-11, January 1997.

[4] S. K. Ip and F. Y. Chang, "Direct generalized characteristic model extraction for high-speed interconnects," *Electron. Lett.*, vol. 35, no. 11, pp. 893-894, May 1999.