Integration of Transient S-Parameter Simulation into HPSpice

Boris Troyanovsky[1], Norman Chang[1], and Dick Dowell[2]

[1]**Hewlett-Packard Labs, Palo Alto, CA**
[2]**ICBD/R&D Center, CA**

HPL–95–??
May, 1995

SPICE; measurement and simulation; packaging; high-speed interconnect; S-parameter; distributed element; HP 8510 network analyzer

Abstract —
Many modern RF/high-speed applications require the simulation of both active devices and high-frequency passive distributed elements together. Such applications include wireless communications, high-speed workstations, and advanced system packages. However, the required simulation capabilities for designing such systems do not exist in HPSPICE, and are not conveniently satisfied by other SPICE tools currently on the market.

We seek to bridge the gap between simulating active devices (which have internal HPSPICE models) and distributed passive elements characterized by measured S-parameters. We have developed an extension to the HPSPICE circuit simulator which allows transient analysis of circuits containing passive elements described by S-parameter data. Such data may be measured directly with network analyzers such as the HP8510, or extracted from simulators such HP's MDS (Microwave Design System) or HFSS (High Frequency Structure Simulator). In this report, we provide an overview of the algorithms used in the simulator, and present practical application examples.

1 Introduction

Many high-speed/RF applications require the simulation of circuits containing both high-speed active devices and passive distributed elements characterized in the frequency domain. Such applications include wireless communication systems, high-speed workstations, and advanced system packages. However, the capabilities required for designing such systems are not present in the standard version of the HPSPICE circuit simulator. This paper presents the algorithms necessary to integrate such capabilities into HPSPICE. A preliminary working version of the enhanced HPSPICE has been developed, with the resulting code being temporarily dubbed SSpice. The aforementioned software tool provides an effective solution for analysis of systems containing passive distributed frequency-domain elements in the same circuit with their lumped time-domain counterparts. Since it operates completely within the well known SPICE environment, it allows the user to stay within a familiar, comfortable framework. Furthermore, all the device models present in HPSPICE are still readily available.

2 An Overview of S-Parameters

Scattering parameters (or S-parameters) are particularly effective for high-frequency characterization of distributed passive structures [1]. Unlike some of their other counterparts, such as Y- and Z-parameters, S-parameters are measured through ratios of incident and reflected sinusoidal waves. For passive systems, the magnitude of an S-parameter will vary between zero and one. Hence, these parameters have superior dynamic range to most others, which vary between zero and infinity for the majority of applications [2].

Before proceeding to a description of the algorithms used in the program, we provide a brief overview of S-parameter fundamentals.

2.1 Traveling Waves in Microwave Networks

Consider a passive linear microwave N-port with waveguide terminals. We suppose the system is excited at precisely one frequency, and choose a reference plane along each waveguide terminal. Because we have a linear system in the sinusoidal steady-state, we may describe all voltages and currents at the chosen reference planes by phasors. Thus, at the reference plane of waveguide n, we will have an incident wave and a reflected wave with voltage phasors V_n^+ and V_n^-, respectively. The corresponding incident and reflected current phasors will be

$$I_n^+ = \frac{V_n^+}{Z_n}, \qquad I_n^- = -\frac{V_n^-}{Z_n}, \tag{1}$$

where Z_n is the characteristic impedance of the nth waveguide.

The incident and reflected waves are normalized [3] by the phasors

$$a_n = \frac{V_n^+}{\sqrt{Z_n}}, \qquad b_n = \frac{V_n^-}{\sqrt{Z_n}}. \tag{2}$$

Frequency-domain S-parameters relate the normalized reflected waves to the incident ones through a linear matrix transformation. In component form, the relationship is

$$b_m(\omega) = \sum_{n=1}^{N} S_{mn}(\omega)\, a_n(\omega) \tag{3}$$

where S_{mn} are the S-parameters. In matrix form, this is simply

$$\underline{b} = S\underline{a} \tag{4}$$

where $\underline{a}$ and $\underline{b}$ are both $N \times 1$ vectors, and S is an $N \times N$ matrix.

Thus, we see that the frequency-domain S-parameter S_{ij} is the (phasor) ratio of the "outgoing" wave at port i to the "incoming" wave at port j when the output ports are matched and port j is driven by a sinusoidal source.

2.2 A Circuit Theory Viewpoint

Having discussed the physical motivation for the use of S-parameters in microwave networks, we now relate the traveling-wave formulation to more conventional circuit concepts. We can write for the total voltage and current at the reference plane of port n

$$V_n = V_n^+ + V_n^- = \sqrt{Z_n}(a_n + b_n) \tag{5}$$

$$I_n = I_n^+ + I_n^- = \frac{(a_n - b_n)}{\sqrt{Z_n}} \tag{6}$$

These two relations immediately allow us to express the normalized wave phasors as functions of voltage and current:

$$a_n = \frac{1}{2}\left(\frac{V_n}{\sqrt{Z_n}} + \sqrt{Z_n} I_n\right) \tag{7}$$

$$b_n = \frac{1}{2}\left(\frac{V_n}{\sqrt{Z_n}} - \sqrt{Z_n} I_n\right) \tag{8}$$

The preceding four equations readily give us a one-to-one mapping between voltage/current and normalized traveling waves. Thus, a scattering matrix relationship between the incident and reflected waves immediately implies a relationship between voltage and current.

3 Simulation Algorithms

As mentioned in the introduction, frequency-domain S-parameters are an effective way to characterize high-frequency passive structures. The characterization consists of scattering matrix "samples" at frequency points over a given range of interest. In other words, to characterize a passive element over a certain frequency range we require a set of S-parameter matrices, each corresponding to a certain frequency.

Once such data is available, it is straightforward to use convolution-based techniques for transient time-domain simulation. The frequency-domain data must first be inverse Fourier transformed into the time-domain. Once this is accomplished, convolution techniques may be used to represent the frequency-domain element as a linear, time-varying N-port. The N-port may then be readily included into the overall circuit matrix.

3.1 Basic Methods for Transient Simulation Based on Frequency-Domain S-Parameter Data

This section outlines the basic techniques used to incorporate circuit elements described by S-parameters into HPSPICE transient simulations. We assume for the moment that we have knowledge of the S-parameter functions $S_{ij}(\omega)$ for all frequencies. This allows us to define the time-domain S-parameter functions $\sigma_{ij}(t)$, which are the inverse Fourier transforms of $S_{ij}(\omega)$. (In practice, DFT/FFT methods are used to compute samples of σ_{ij}). We further define the quantities $\alpha_i(t)$ and $\beta_i(t)$ to be the inverse Fourier transforms of the incident and reflected waves $a_i(\omega)$ and $b_i(\omega)$, respectively.

With these definitions we have, by inverse transforming equation (3),

$$\beta_m(t) = \sum_{n=1}^{N} \sigma_{mn}(t) * \alpha_n(t) \tag{9}$$

where the operator * denotes convolution. Assuming that the S-parameters are all measured with a reference impedance Z_o, we also have the additional time-domain formulas

$$\alpha_n(t) = \frac{1}{2}\left(\frac{v_n(t)}{\sqrt{Z_o}} + \sqrt{Z_o}\, i_n(t)\right) \tag{10}$$

$$\beta_n(t) = \frac{1}{2}\left(\frac{v_n(t)}{\sqrt{Z_o}} - \sqrt{Z_o}\, i_n(t)\right) \tag{11}$$

which relate the time-domain reflected and incident waves to time-domain voltages and currents.

The HPSPICE circuit simulator's internal solution algorithms depend on an admittance (Y-) matrix formulation of the nodal equations. Consequently, while the convolution operation of eq. (9) may be performed in the S-parameter domain, the final submatrix of the linear N-port must be an admittance matrix for inclusion into the HPSPICE Jacobian. To be precise, the problem to be solved is this: given voltage samples $v_i(t_0), v_i(t_1), \ldots, v_i(t_n)$ at each port i, we must exhibit an admittance matrix and a current "source" vector which describe our N-port at time t_{n+1}. That is, we seek a relationship of the form

$$\underline{i}(t_{n+1}) = Y\underline{v}(t_{n+1}) + \underline{c} \tag{12}$$

where Y (an $N \times N$ matrix) and c (an $N \times 1$ vector) are both functions of $v_i(t_0), v_i(t_1), \ldots, v_i(t_n)$.

As a first step towards deriving such a relation, we rewrite (9) as

$$\beta_i(t_{n+1}) = \sum_{j=1}^{N} \int_{-\infty}^{t_{n+1}} \alpha_j(\tau)\, \sigma_{ij}(t_{n+1} - \tau)\, d\tau \tag{13}$$

and note that we can split up the integration into two intervals, from $-\infty$ to t_n and from t_n to t_{n+1}. Integrating numerically over the second of these two intervals yields

$$\int_{t_n}^{t_{n+1}} \alpha_j(\tau)\, \sigma_{ij}(t_{n+1} - \tau)\, d\tau = A_{ij} + B_{ij}\alpha_j(t_{n+1}) \tag{14}$$

where A_{ij} and B_{ij} are functions of the (known) quantity $\alpha_j(t_n)$ and the samples of σ_{ij}. Using this relationship, (13) may be expressed in the matrix form

$$\underline{\beta}(t_{n+1}) = H\underline{\alpha}(t_{n+1}) + \underline{g}, \tag{15}$$

where the elements of the $N \times N$ matrix H are given by

$$H_{ij} = B_{ij} \tag{16}$$

and the components of the $N \times 1$ vector $\underline{g}$ are

$$g_i = \sum_{j=1}^{N} \left[A_{ij} + \int_{-\infty}^{t_n} \alpha_j(\tau)\, \sigma_{ij}(t_{n+1} - \tau)\, d\tau\right] \tag{17}$$

At Spice time-step t_{n+1}, the voltage samples $v_i(t_0), v_i(t_1), \ldots, v_i(t_n)$ are known. From the admittance relationship which was computed at time t_n, $\underline{i}(t_n)$ is readily found. This allows the normalized time-domain coefficients α_i to be calculated through (10), thus permitting the evaluation of g_i through numerical integration.

Substitution of (10) and (11) into (15) yields the expression necessary to obtain the form (12). Through some straightforward algebraic manipulation, we find that

$$Y = \frac{1}{Z_o}(I + H)^{-1}(I - H) \tag{18}$$

and

$$\underline{c} = -\frac{2}{Z_o}(I + H)^{-1}\underline{g} \tag{19}$$

where H and $\underline{g}$ have been given previously. These quantities may now be incorporated directly into HPSPICE data structures.

3.2 DC and AC Analyses

The DC and AC analysis algorithms are even more straightforward than their time-domain transient counterparts. This is, of course, due to the fact that they are both single-frequency analyses. In the event that an AC analysis is requested at a frequency point which is not one of the samples available, interpolation is used to obtain the approximate scattering matrix.

The DC analysis must precede all transient simulations. The appropriate admittance sub-matrix for the DC portion of the run is readily derived by substituting eqs. (7) and (8) into (4). The resulting Y-matrix is

$$Y = \frac{1}{Z_o} [I + S(0)]^{-1} [I - S(0)] \ . \tag{20}$$

The formula for AC analysis is identical, with the exceptions that the S-parameter matrix is taken at the appropriate frequency and that the admittance matrix may now be complex.

4 Application Example

SSpice is most useful when a design engineer wishes to run transient SPICE simulations on a circuit containing elements which are best described in the frequency-domain. Such frequency-domain data may come either from measurements (say, from an HP8510C) or from simulation tools such as the HP Microwave Design System (MDS) or the HP High-Frequency Structure Simulator (HFSS).

One real-world application using SSpice in interconnect/ bypass capacitor/power-ground plane analysis between chips on Multi-chip Modules (MCMs) [4] is illustrated below.

In this case, SSpice was used to model the power supply noise in a 19-chip MCM (Figure 1) with thin- and thick-film technology. Since the risetime of the CMOS drivers used in the module is less than 200ps (comparable to the time needed for the signals and noise to traverse the module), the whole signal path and power supply planes must be treated as distributed elements. Distributed frequency-domain models for the three-coupled lines, bypass capacitors, and planes were developed using measurement data from a Vector Network Analyzer (HP8510C) and simulation with HP's Microwave Design System (MDS). Figure 2 shows the six-port power supply plane model in MDS. 6x6 S-parameter matrices were calculated for a frequency range of 50 MHz - 10.05 GHz at 50 MHz intervals.

Using SSpice, each port of this 6-port S-parameter model was connected to a CMOS driver and a current-dependent current source representing the simultaneously switching drivers on the same chip. A transient simulation of the system was then performed. Twenty minutes were spent extracting frequency-domain models from MDS, and an additional twenty minutes were needed for the SSpice transient simulation. Figure 3 shows the calculated noise on the power supply and the output waveform of the driver on each chip when all drivers switch simultaneously on the MCM with the thin-film power planes. Port 6, where a large chip is connected, suffers the largest noise. The Vdd spike also causes distortion on the output waveform.

Using this time-domain simulation methodology, simultaneous switching noise in large MCMs with different technologies can be analyzed, along with the effects of on-chip and off-chip bypass capacitors. Note that the whole signal path can also be treated as a black box. Time-domain simulations may then be readily performed by simply running SSpice with the S-parameter measurement data for the 6-port. On the other hand, an accurate time-domain model for the signal path may take weeks to develop, and may require know-how outside of the designer's immediate area of expertise.

5 Conclusion

This paper presented a description of SSpice, an extension to the HPSPICE circuit simulator which adds support for passive distributed circuit elements characterized in the frequency domain. The algorithms on which the extension is based were discussed, and a real-life example was presented. The tool should prove useful to designers who wish to directly simulate distributed structures in their HPSPICE circuits.

Acknowledgments

We wish to thank X. Xuan, J. Meyer, and B. Donecker of HP EEsof for the technical discussions and suggestions which they provided. K. Lee and A. Barber of HP Labs-SWG furnished us with much of the initial motivation for the project, helped immensely in testing early versions of the code, and provided a real-life application example. Additional thanks go to J. Kang, S.Y. Oh, and K.J. Chang for supplying additional test examples and suggestions. Lastly, we wish to express our appreciation to J. Coffron, R. Coverstone, and B. Shreve for their advice and support of this project.

References

[1] L.N. Dworsky, *Modern Transmission Line Theory and Applications,* New York: Wiley, 1979, pp. 46-51

[2] W.R. Eisenstadt and Y. Eo, "S-Parameter-Based IC Interconnect Transmission Line Characterization," *IEEE Trans. on Components, Hybrids, and Manufacturing Technology,* pp. 483-490, August 1983.

[3] S. Ramo, J.R. Whinnery, and T. Van Duzer, *Fields and Waves in Communication Electronics,* 2nd Edition, New York: Wiley, 1984, pp. 535-538

[4] K. Lee and A. Barber, "Modeling and Analysis of Multichip Module Power Supply Planes," HP Labs Technical Report, HPL-94-32.

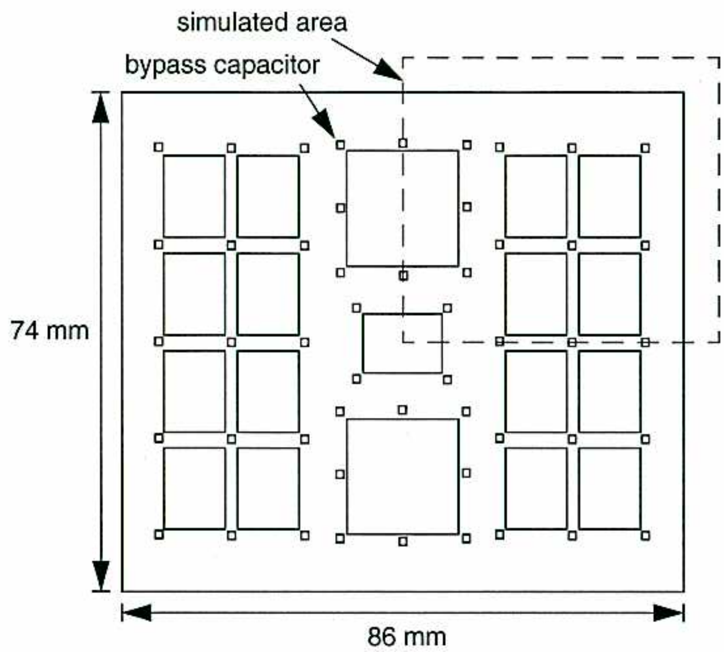

Figure 1. 19-chip MCM. Two large chips have 380 drivers each and 17 small chips have 45 drivers each. Substrate has one vdd/gnd pair of thin film planes and one vdd/gnd pair of thick film planes.

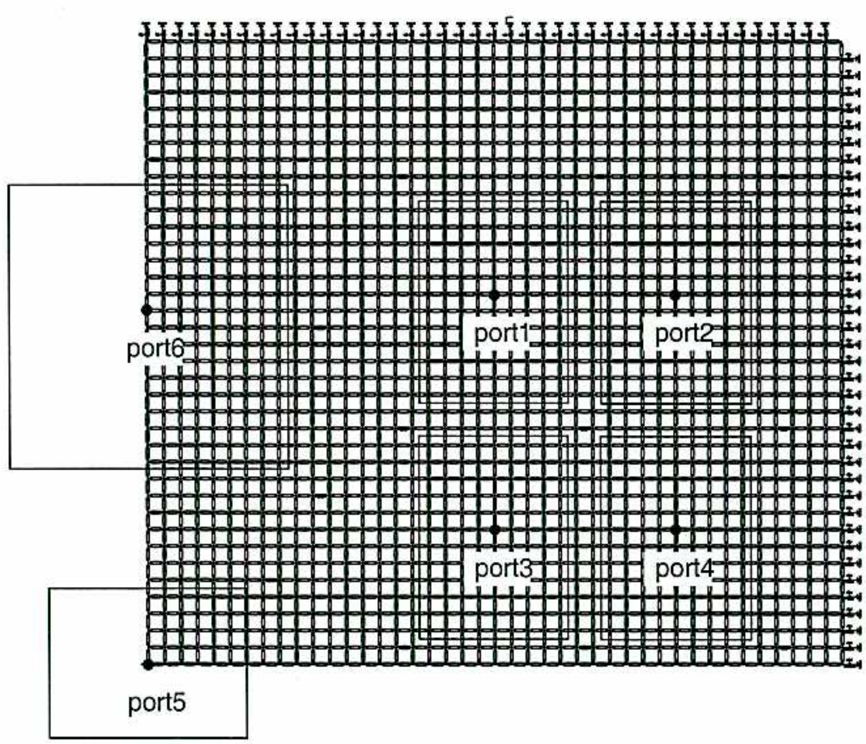

Figure 2. Plane model of 1/4 of 19-chip MCM.

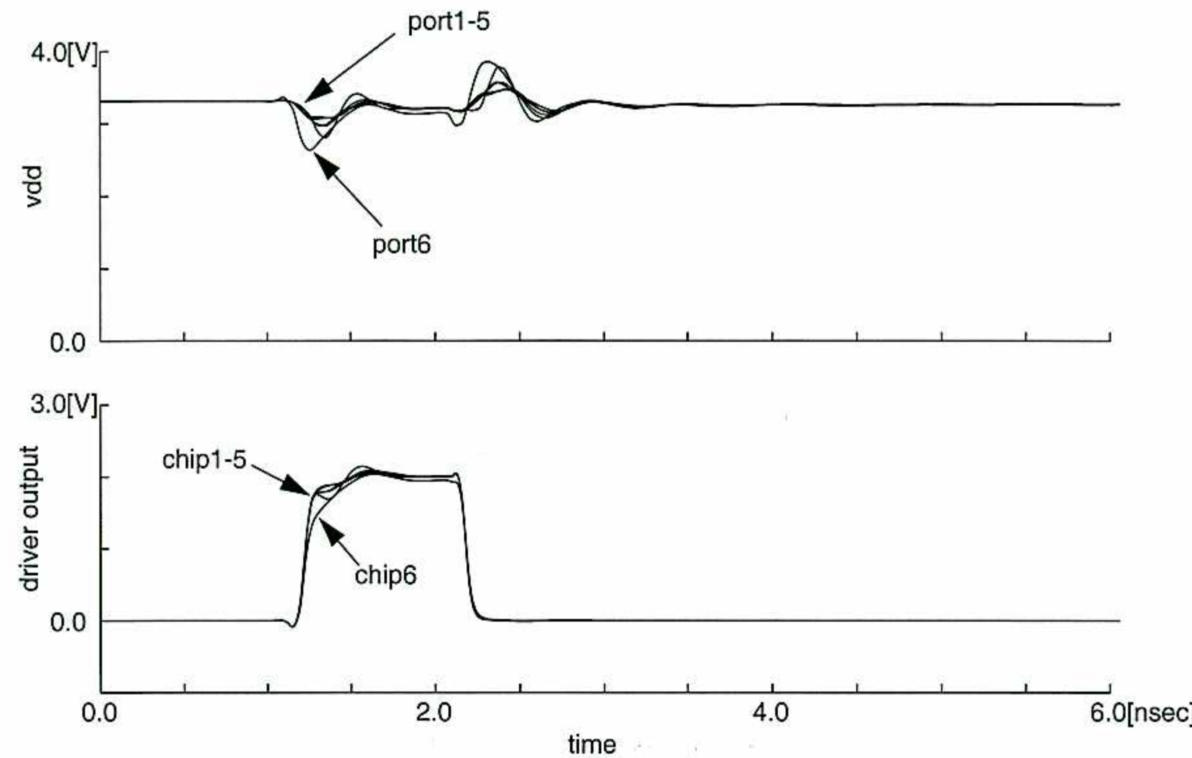

Figure 3. VDD noise and driver outputs of 19-chip MCM with thin film power planes.