INTERCONNECTION NETWORK SIMULATOR (INS)

Shahram Manighalam

Shahram Manighalam is a member of technical staff in the Interconnection Systems and Assembly Reliability Department at AT&T Bell Laboratories in Whippany, New Jersey. He is responsible for the electrical modeling of interconnection systems and for developing software tools to simulate the electrical performance of those systems. Mr. Manighalam joined the company in 1984. He has a B.S.E.E. and an M.S.E.E. in solid-state electronics and computer architecture from California State University in Los Angeles.

The increasing speed of today's digital systems has imposed new demands on interconnection network design. To help ensure quality designs, a way was needed to assess the performance of the interconnection network early in the design process. However, existing design systems did not support transmission line and circuit analyses. Now, the interconnection network simulator (INS)—an interactive, menu-driven simulation program—provides this capability. The INS allows a circuit designer to simulate the entire interconnection network under active conditions and provides graphs of the voltage, current, and frequency responses for evaluating the network's performance. Its on-line editing feature permits the designer to alter the electrical, geometrical, and physical parameters of the simulated network to increase its performance. The INS has been incorporated into several general computer-aided design systems, giving them the transmission line and circuit analysis capabilities that they lacked.

63

Introduction

Today's digital systems are becoming faster, which increases the demands on the design of the interconnection network. To ensure quality designs without redesign, circuit designers need a way to assess the performance of the interconnection network early in the design process. Several simulation systems (e.g., SPICE and ADVICE) already exist that can simulate electronic networks, but they cannot simulate the effects of interconnection networks on the whole system. (SPICE is a simulation program for integrated circuit engineering, while ADVICE is an aid for design verification for integrated circuit engineering.)

The interconnection network simulator (INS)—an interactive, menu-driven program that can be incorporated into a computer-aided

design (CAD) system—provides this capability. It can accurately model any linear tree-like network, including transmission lines, and produces a graphical response of the network to periodic waveforms of any frequency or shape.

The INS acts as an analytical oscilloscope, allowing a circuit designer to simulate the entire interconnection network under active conditions. It outputs graphs of the voltage, current, and frequency responses of the interconnection network. These results provide the information needed to evaluate the system's performance in terms of signal-carrying capability, reflections, time delays, and frequency limitations.

In addition, the INS permits on-line editing of the network to increase performance. This feature allows interactive modification of the electrical, geometrical, and physical parameters of the system being simulated.

The INS is available as a stand-alone package to circuit designers and others who may need it. It has already been incorporated into some of the more general CAD systems at AT&T to give them transmission line and circuit analysis capabilities.

A user's manual that describes the operational aspects of the program accompanies the INS software.

The Interconnection Network Simulator

Designed to be user friendly, the INS helps circuit designers analyze the electrical effects of interconnection networks in terms of frequency limitations, pulse distortion, wave reflections, propagation delay, and phase dispersion. Its visual displays are easy to understand, enabling a circuit designer to draw conclusions about the performance of an interconnection system to be used for a particular circuit.

The interconnection systems it can simulate include circuit packs, cables, backplanes, integrated-circuit (IC) packages, and vias. [A *via* is a conductive path provided between the circuits on different layers of a printed circuit board (PCB).] The INS program's level of simulation can be highly detailed. In most cases, an INS user need only define the physical and geometrical

aspects of the system. INS menus prompt the user for such details as:
- Transmission line configurations, e.g, microstrip, stripline, balanced
- Chip attachment methods
- Type and dimensions of conductor cross sections
- Dimensions of vias
- Loads and voltage sources.

With the INS menus, a user can define an arbitrary linear tree-like network and analyze its response to an arbitrary input signal. The user selects the point on the network where the signal is to be applied, as well as where the response is to be observed. The network's response can then be viewed as a plot of voltage or current versus time. The network's frequency response can also be observed as a plot of gain versus frequency over a frequency spectrum determined by the user.

The network may consist of purely lumped elements (R, L, C) or any combination of lumped parameters and transmission lines. (R, L, and C are resistance, inductance, and capacitance, respectively.)

To model lossy transmission lines, the INS uses accurate distributed-parameter techniques. It takes into account the effects of high frequencies on the transmission line parameters, such as the skin effect for R and L and for G (conductance per unit length of the dielectric). The INS user provides information about the dimensions of the conductors and their geometry. By using the INS's on-line editing feature, the user can also change the simulated system's electrical, geometrical, and physical parameters to improve performance.

Model Description

Figure 1a shows the general form of the network that the INS can analyze, while Figure 1b shows each segment of the network in detail. This tree-like network consists of:
- A typical source impedance to represent the voltage source's output impedance or the termination impedance of the transmission line.
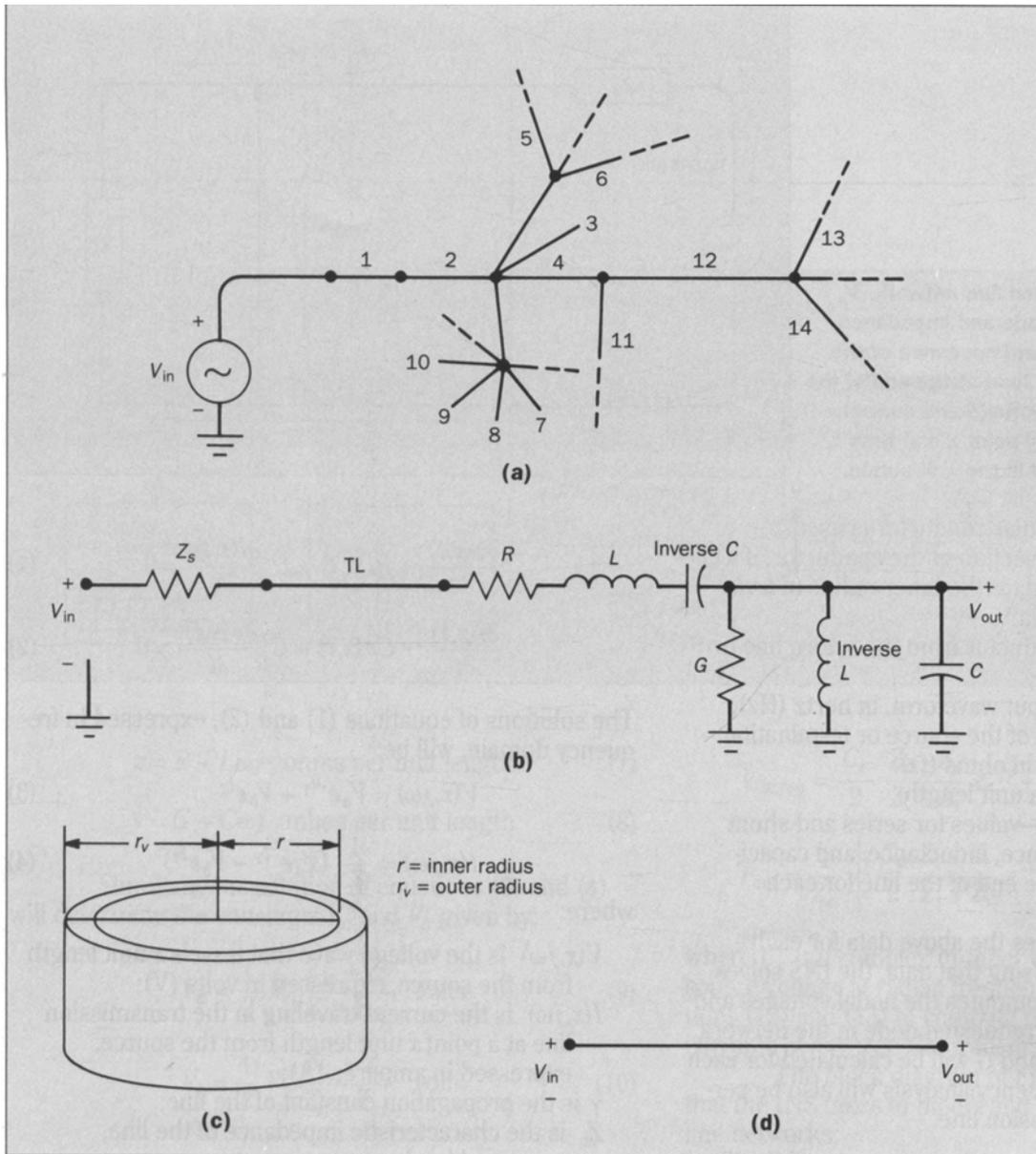- A transmission line.

64

Figure 1. General form of the network to be analyzed. (a) Basic tree-like network. (b) The network representing each segment of a tree-like network. (c) Via parameters and (d) a waveguide. V_{in} and V_{out} are input and output voltage, respectively; Z_s is the source impedance; TL is a transmission line; R, L, and C are the distributed resistance, inductance, and capacitance, respectively; and G is the conductance per unit length.

65

- A complex load placed at the end of the transmission line. This complex load consists of both a series RLC and a shunt RLC load, a configuration that permits achieving any combination of load.

To model the transmission line, INS uses the distributed inductance, capacitance, resistance, and conductance per unit length. These parameters can either be provided by the user or calculated by the INS. In addition, the INS can also simulate vias and waveguides (see Figures 1c and 1d).

Some of the typical parameters that the user should enter are:

- μ—permeability of the dielectric, in henrys (H) per unit length.
- k—permittivity of the dielectric, in farads (F) per unit length.
- ρ—resistivity of the conductor of the transmission line, in ohms-unit length.
- φ—loss angle of the dielectric, in radians.
- a and b—dimensions of the cross section of the

Figure 2. Basic transmission line network. V_s and Z_s are the source voltage and impedance, respectively. Z_{in} is the input impedance of the line, and Z_l is the complex load at the end of the line. $v(x,t)$ and $i(x,t)$ are voltage and current, respectively, at an arbitrary point x and time t. dx is the length differential in the x direction.
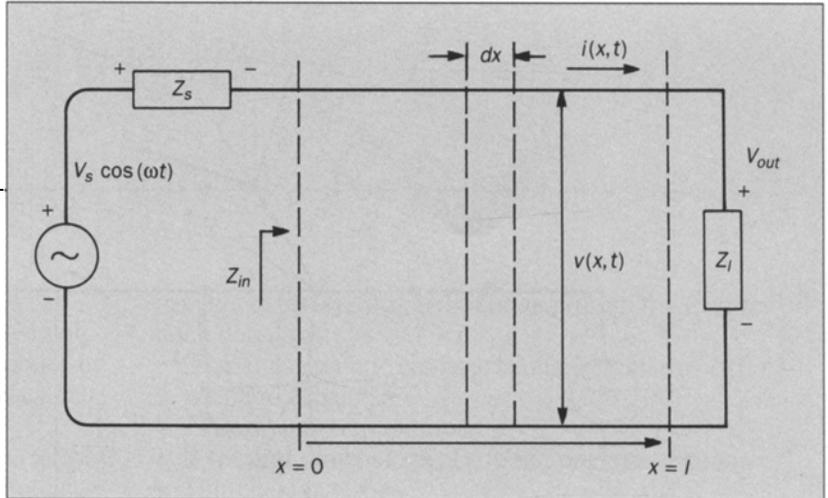
conductor, if a rectangular conductor is used.
- r—radius of the cross section of the conductor, if a circular conductor is used; or the inner radius of a via.
- r_v—outer radius of a via.
- d—distance of the conductor from the return line or ground plane.
- f—frequency of the input waveform, in hertz (Hz).
- Z_s—output impedance of the source or termination impedance of the line, in ohms (Ω).
- l—length of the line, in unit length.
- RLC shunt and series—values for series and shunt load elements (resistance, inductance, and capacitance) connected at the end of the line for each segment.

The user furnishes the above data for each branch of the network. Using that data, the INS solves the entire network and computes the nodal voltages and currents response at the requested node in the network. The parameters $R, L, C,$ and G will be calculated for each frequency; the high-frequency analysis will also be performed for each transmission line.

Evaluating a Network with Periodic Excitation

Now, let us look at the INS's approach to evaluating the nodal voltages and currents of a transmission line network with periodic excitation.

Consider the transmission line of Figure 2. Suppose $v(x,t)$ is the line to line voltage at an arbitrary point x (in Figure 2) and time t, and $i(x,t)$ is the current at an arbitrary point x and time t. Then, the following relations (which are in time domain) will hold for a transmission line with distributed $R, L, C,$ and G parameters:[1]

$$\frac{\partial v(x,t)}{\partial x} + Ri(x,t) + L\frac{\partial i(x,t)}{\partial t} = 0 \qquad (1)$$

$$\frac{\partial i(x,t)}{\partial x} + Gv(x,t) + C\frac{\partial v(x,t)}{\partial t} = 0 \qquad (2)$$

The solutions of equations (1) and (2), expressed in frequency domain, will be:[1]

$$V(x,j\omega) = V_a e^{-l\gamma} + V_b e^{l\gamma} \qquad (3)$$

$$I(x,j\omega) = \frac{1}{Z_c}(V_a e^{-l\gamma} - V_b e^{l\gamma}) \qquad (4)$$

where:

$V(x,j\omega)$ is the voltage wave that travels x unit length from the source, expressed in volts (V);

$I(x,j\omega)$ is the current traveling in the transmission line at a point x unit length from the source, expressed in amperes (A);

γ is the propagation constant of the line;

Z_c is the characteristic impedance of the line, expressed in ohms;

V_a and V_b are constants to be determined from the simultaneous solution of equations (3) and (4).

The equations for Z_c and γ are given below:

$$\gamma = \sqrt{z\,y} \qquad (5)$$

$$Z_c = \sqrt{(z/y)} \quad \text{ohms} \qquad (6)$$

Here, z is the loop series impedance of the line, and y is the shunt admittance of the line expressed as:
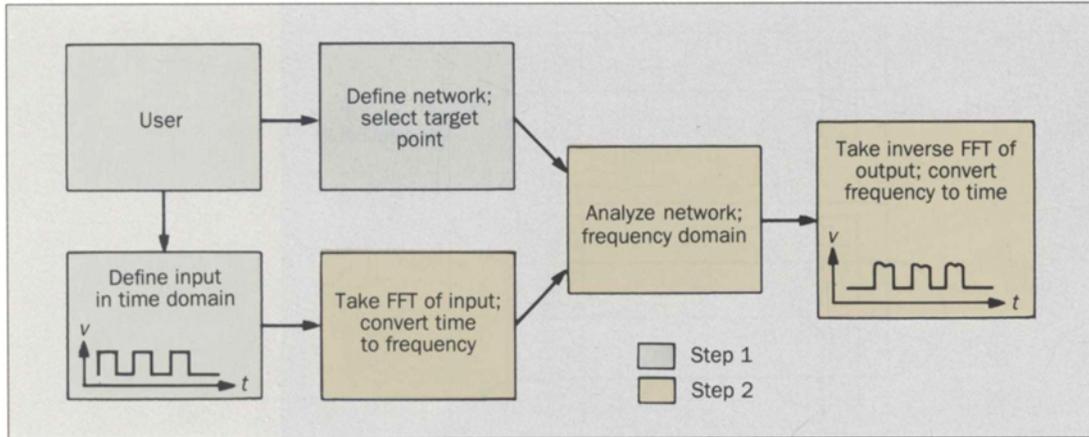
66

Figure 3. Block diagram showing the simulator's approach to solving complex networks. FFT is a Fast Fourier Transform and FFTI is an inverse FFT. (See the text for a description of the steps.)

$$z = R + L\omega j \quad \text{ohms per unit length} \qquad (7)$$

$$y = G + C\omega j \quad \text{mhos per unit length} \qquad (8)$$

Simultaneous solution of equations (3) and (4) will determine the constants V_a and V_b given by:

$$V_a = \frac{1}{2} V_s \frac{Z_{in} + Z_c}{Z_s + Z_{in}} \quad \text{volts} \qquad (9)$$

$$V_b = \frac{1}{2} V_s \frac{Z_{in} - Z_c}{Z_s + Z_{in}} \quad \text{volts} \qquad (10)$$

Here, V_s is the source voltage in volts; and Z_{in} is the input impedance of the line at the source in ohms, and is given by the following equation:[1]

$$Z_{in} = Z_c \frac{z_l + 1 + (z_l - 1) \ e^{[-2\gamma(l-x)]}}{z_l + 1 - (z_l - 1) \ e^{[-2\gamma(l-x)]}} \qquad (11)$$

where $z_l = Z_l/Z_c$, and Z_l is the complex load at the end of the line, in ohms. Therefore, equations (3) and (4) become:

$$V_{(x,j\omega)} = \frac{C_n}{2} \frac{Z_i + Z_c}{Z_s + Z_i} \ e^{-l\gamma} + \frac{C_n}{2} \frac{Z_i - Z_c}{Z_s + Z_i} \ e^{l\gamma} \qquad (12)$$
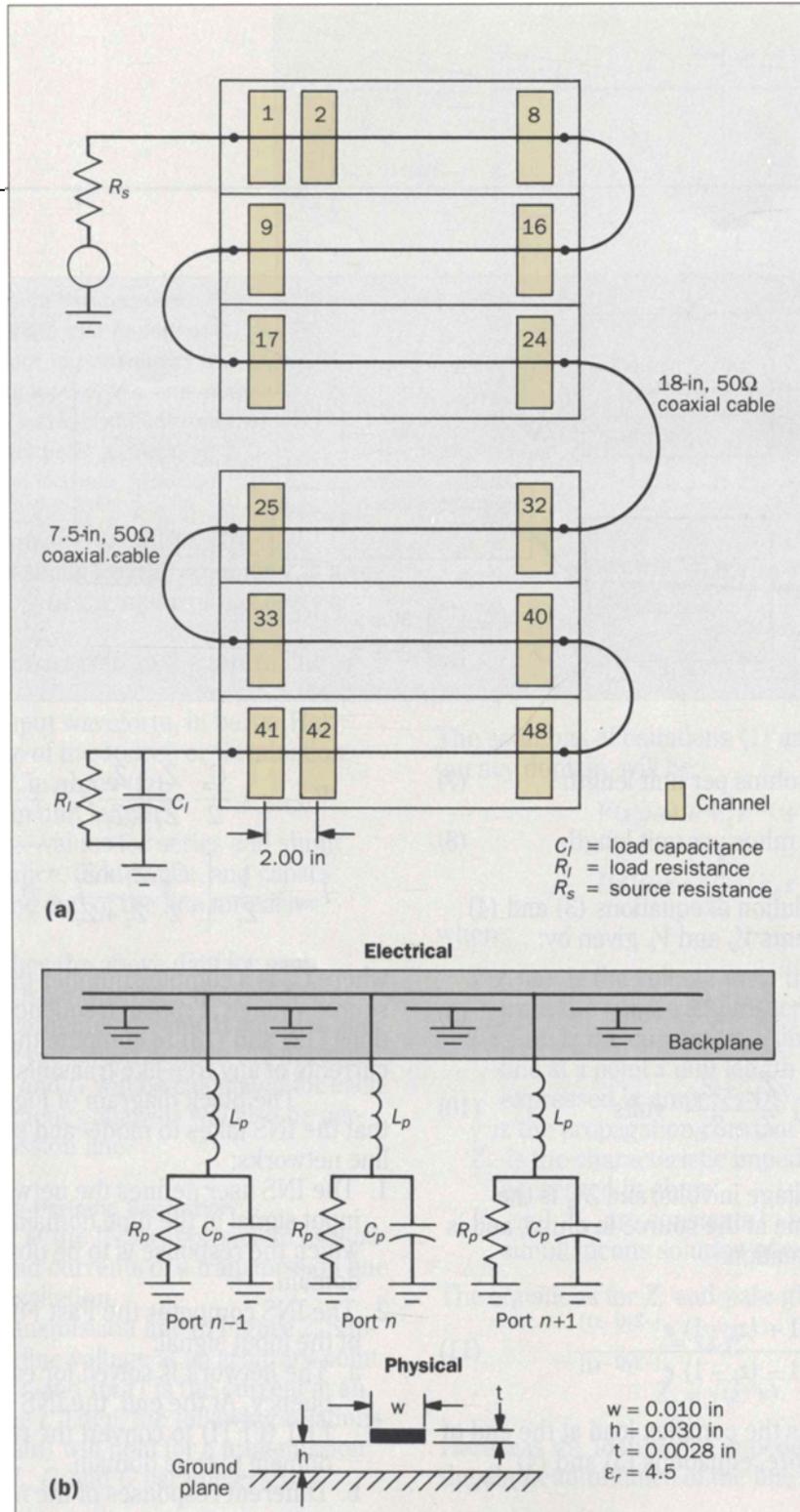
$$I_{out} = \frac{1}{Z_c} \left[\frac{C_n}{2} \frac{Z_i + Z_c}{Z_s + Z_i} \ e^{-l\gamma} - \frac{C_n}{2} \frac{Z_i + Z_c}{Z_s - Z_i} \ e^{l\gamma} \right] \qquad (13)$$

where C_n is a complex number that represents the source voltage at radian frequency ω. The INS uses equations (12) and (13) to compute the nodal voltages and currents of any tree-like transmission line network.

The block diagram of Figure 3 displays the steps that the INS takes to model and solve large transmission line networks:

1. The INS user defines the network and the periodic input signal in the time domain, and selects the point at which the response is to be observed in the time domain.
2. The INS computes the Fast Fourier Transform (FFT) of the input signal:
   a. The network is solved for each harmonic frequency. At the end, the INS will perform an inverse FFT (FFTI) to convert the results from frequency domain to time domain.
   b. Different responses of the network that the INS

67

Figure 4. Use of the INS to optimize the design of a cellular radio system.
(a) Physical layout of 48-channel model.
(b) System's TDM bus modeled for use by the simulator. R_p, L_p, and C_p are the resistance, inductance, and capacitance, respectively, for port p; ε_r is the dielectric constant of the bus material.

68

Figure 4 (a)

18-in, 50Ω coaxial cable

7.5-in, 50Ω coaxial cable

2.00 in

R_l C_l

☐ Channel

C_l = load capacitance
R_l = load resistance
R_s = source resistance

(a)

Electrical

Backplane

L_p L_p L_p

R_p C_p R_p C_p R_p C_p

Port $n-1$ Port n Port $n+1$

Physical

Ground plane

w = 0.010 in
h = 0.030 in
t = 0.0028 in
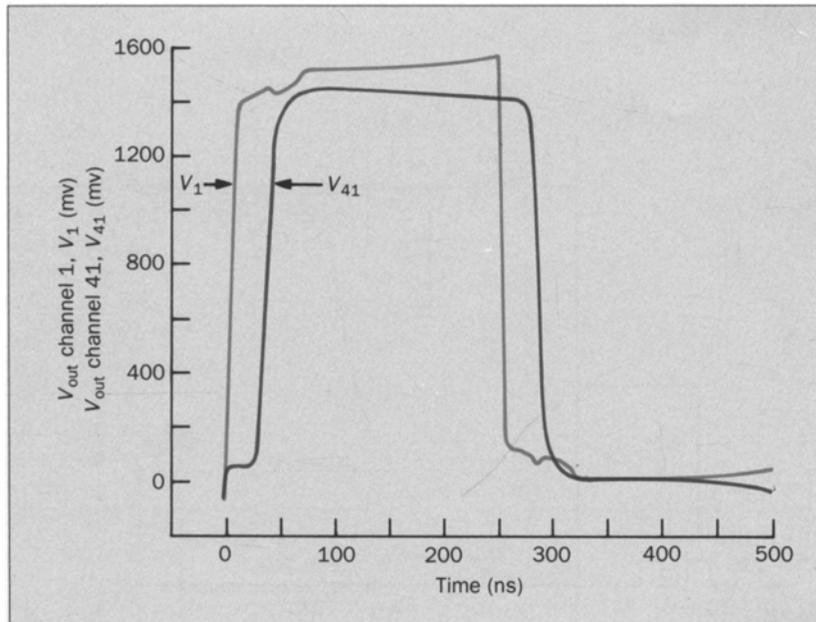ε_r = 4.5

(b)

Figure 5. System voltage response in the time domain at channels 1 (V_1) and 41 (V_{41}) for the simulated 48-channel model.

computes include: input and output voltage waveform versus time, output current waveform versus time, output frequency response curve, and input waveform frequency spectrum. Multiple output waveforms will provide timing information between different points on the network.

3. The user now does on-line editing of the network for immediate resimulation (step 2 is repeated), seeking to increase the network's performance.

INS for Design Optimization

Circuit designers have already used the INS to optimize designs. Here, we discuss applications of the INS to the design of a cellular radio system and a PCB.

Design of a Cellular Radio System. The objective here was to determine the optimum performance capability and design for the time-division-multiplexed (TDM) bus of an AT&T cellular radio system.

The INS was used to simulate a six-carrier radio frame (Figure 4a) to determine the overall bus delay and

signal deterioration. As Figure 4b shows, each channel was modeled using a microstrip configuration. Lumped elements represent the circuitry attached to each channel. A coaxial cable connects the end channel in each row to the first channel in the next row. The voltage generator was modeled to produce a pulse of 2 MHz (megahertz) at a rise and fall time of 10 ns (nanoseconds) and to have an output impedance of 50 ohms.

Figure 5 shows the voltage waveforms that the INS computed at channels 1 and 41, where we see the delay is 32 ns. The voltage deterioration at channel 41 can also be observed.

Printed-Circuit-Board Simulation. PCBs can be simulated with the INS for analysis of such electrical performance as delay, attenuation, dispersion, current requirements, and frequency limitation.

Figure 6a shows two patterns on a PCB. (The ICn are integrated circuits, and the TLn are transmission lines of the interconnection system.) These interconnection paths, along with their lumped parameter source

Figure 6. The INS analysis of a PWB. IC*n* are integrated circuits, and TL*n* are transmission lines of the interconnection system. (a) Two networks on a PWB. IC1 and IC4 are clock generators; and IC2, IC3, and IC5 act as loads. (b) Schematic modeled for use by the simulator. Dashed lines separate the different branches, and the circled numbers represent the user's name for each branch.
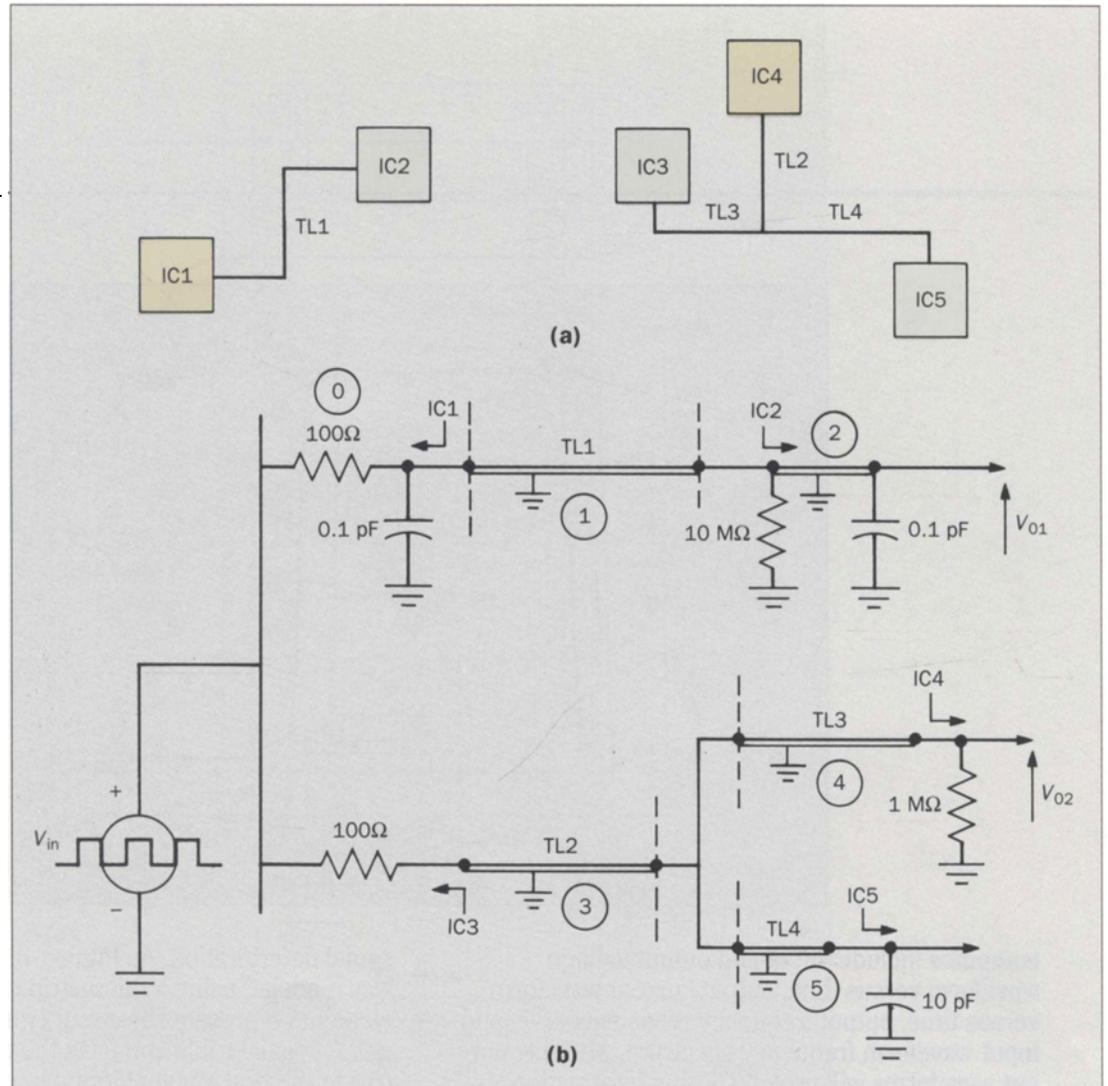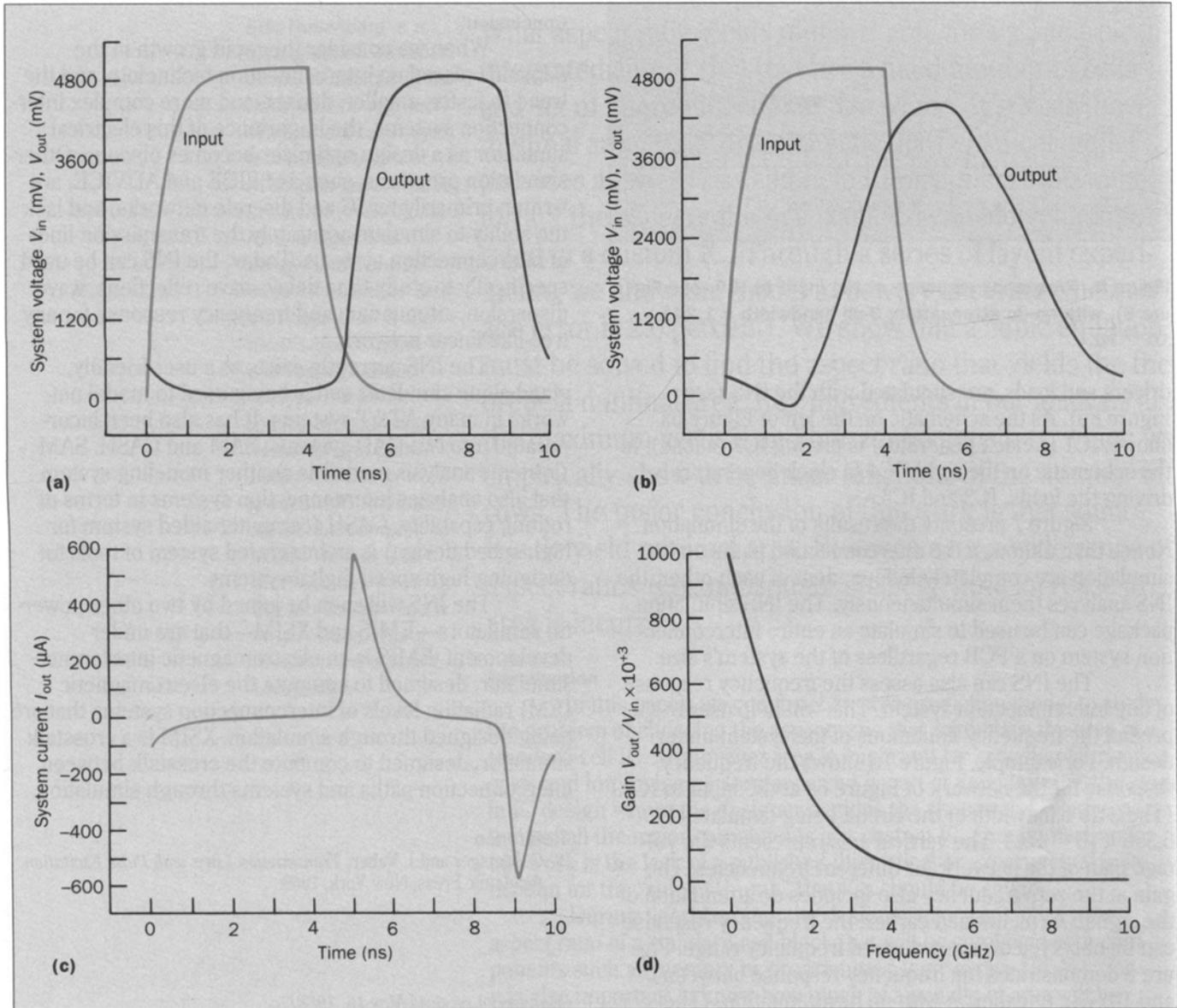
70

Figure 7. Responses computed by the INS for the simulated PCB in Figure 6. Voltage response for (a) the first network at the input to IC2, and (b) the second network at the input to IC5. (c) Current input to IC2. (d) Frequency response at IC2; includes dc attenuation of the signal; 3-db bandwidth = $6.556 \times 10^{+08}$ Hz.
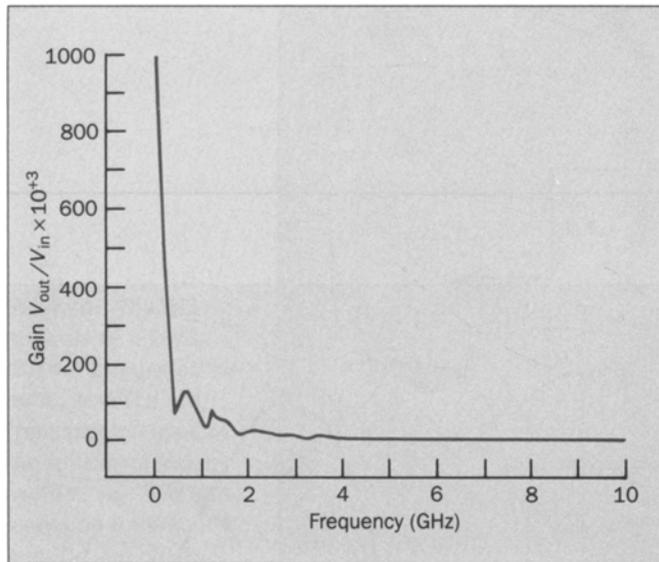
(a)

(b)

(c)

(d)

71

Figure 8. Frequency response at the input to IC5 (see Figure 6), with no dc attenuation; 3-db bandwidth = 1.241 × 10^{+08} Hz.

drivers and loads, are simulated with the INS (see Figure 6b). As the schematic on the left of Figure 6a shows, IC1 (a clock generator) is driving IC2 (a load); in the schematic on the right, IC4 (a clock generator) is driving the loads, IC3 and IC5.

Figure 7 presents the results of the simulation. Notice that, although the interconnection paths under simulation are completely independent of each other, the INS analyzes them simultaneously. The INS simulation package can be used to simulate an entire interconnection system on a PCB regardless of the system's size.

The INS can also assess the frequency response of any interconnection system. This will help users predict the frequency limitations of the system under design. For example, Figure 7d shows the frequency response for the network of Figure 6a at the input to IC2. (The 3-db bandwidth of the circuit being simulated is $6.556 \times 10^{+08}$ Hz.) The vertical axis represents the voltage gain of the network for different frequencies. The gain at the zero frequency also includes dc attenuation of the signal. As mentioned earlier, the frequency response can be observed over any desired frequency range. Figure 8 demonstrates the frequency response between 0 and 10 GHz (gigahertz) with no dc attenuation. (The 3-db bandwidth of the circuit being simulated here is $1.241 \times 10^{+08}$ Hz.)

Conclusion

When we consider the rapid growth in the demands placed on interconnection technology and the trend to faster, smaller, denser, and more complex interconnection systems, the importance of this electrical simulator as a design optimizer becomes obvious. Other simulation programs, such as SPICE and ADVICE, are written primarily for IC and discrete networks, and lack the ability to simulate accurately the transmission lines of interconnection systems. Today, the INS can be used specifically to study time delay, wave reflections, wave dispersion, attenuation, and frequency response for any tree-like linear network.

The INS currently exists as a user-friendly, stand-alone simulator and is being used to model networks in many AT&T systems. It has also been incorporated into two CAD systems: SAM and CASH. SAM (systems analysis model) is another modeling system that also analyzes interconnection systems in terms of routing capability. CASH (computer-aided system for high-speed design) is an integrated system of tools for designing high-speed digital systems.

The INS will soon be joined by two other powerful simulators—EMIS and XSIM—that are under development. EMIS is an electromagnetic interference simulator, designed to compute the electromagnetic (EM) radiation levels of interconnection systems that are being designed through simulation. XSIM is a crosstalk simulator, designed to compute the crosstalk between interconnection paths and systems through simulation.

Reference
1. G. Metsger and J. Vaber, *Transmission Lines with Pulse Excitation*, Academic Press, New York, 1969.

72