Virtual Instruments for Development of High Performance Circuit Technologies

Robert W. Dutton, Zhiping Yu, Francis Rotella,
Stephen Beebe*, Boris Troyanovsky, and Lydia So

Center for Integrated Systems
Stanford University, Stanford, CA 94305-4055
*Advanced Micro Devices, Sunnyvale, CA 94088-3453

Abstract

A set of virtual instruments based on computer-aided design tools for technology (TCAD) are described. These virtual instruments support the evaluation of new technologies for circuit applications, including both intrinsic and parasitic effects. Mixed-mode (circuit/device) simulation in both the frequency and time-domain is demonstrated including an example of a virtual network analyzer applied to evaluation of a GaAs FET. Virtual curve-tracing is demonstrated as a powerful means to obtain I-V curves and to zoom in on the regions of device characteristics where SPICE model parameters can effectively be extracted and parasitic effects such as failure mechanisms due to electrostatic discharge (ESD) can be analyzed. Finally large signal distortion behavior analyzed based on the device simulation using the harmonic balance (HB) method is demonstrated with application to extraction of intermodulation (IM) distortion in bipolar transistor circuits.

Introduction

The requirements for high performance circuits that support applications at radio frequencies (RF) place serious demands on present characterization and design tools. The difficulties of high frequency measurements and the dependence of the results on parasitics are becoming a limitation both in technology development and in support of circuit designers. Hence, the opportunity to use new computer tools at the device level to support such user needs in overcoming present limits is very promising. In addition, there are many parasitic effects where experimental verification is difficult and often not conclusive. For example, problems such as latchup (1), substrate noise (2), and ESD are major problems where circuit models either do not exist or are a major problem to develop and calibrate. Technology computer-aided design (TCAD) tools and the methodology discussed here offer powerful capabilities to support circuit designers where compact models are inadequate and experimental characterization is difficult because of parasitic effects. This paper will demonstrate how specific TCAD tools and a virtual instrument methodology can be applied to overcome many of these limitations.

Technology CAD and “Virtual Instruments”

The definition of Technology CAD has been broadly used to describe the union of three levels of simulation and design capabilities---circuits, device, and IC processing. For this paper, we focus only on the circuit and device levels, although the process technology data base supports these higher levels through parameters such as 2D doping profiles and topography. Figure 1 shows a simple schematic representation of the TCAD hierarchy used in this work and indications of parameters that can be extracted for other applications that support both characterization and design applications. Each of these application areas is discussed below based on specific examples.

The means to extract this data is based on a new CAD methodology called “virtual instrumentation” (3). The basic idea is to use the core device simulation capabilities of TCAD in concert with software “controllers” and associated algorithms that are targeted to achieve efficiency in simulation of complex technology problems. For example, the I-V characteristics of breakdown (see Figure 2) are complex both in shape and the underlying device physics. The “curve-tracing” algorithm, to be discussed and demonstrated below, provides the smooth tracing of these curves and support in their parameterization at the technology level. In the case of device behavior where parasitics are important, e.g. high frequency characterization, interaction with circuit parameters can easily be included using mixed-mode simulations of circuit-device models where arbitrarily complex circuitry boundary conditions are included in “virtual instrument” techniques for technology characterization.

Curve Tracer

This is the most straightforward of the virtual instrument algorithms, both conceptually and in TCAD implementation. The algorithm was recently published (4) and implemented in concert with both PISCES and MEDICI device simulators (5). Basically, the curve tracer guides a simulator through complex I-V curves by automatically adapting the load-line boundary conditions to maintain solution stability throughout the trace. Smooth I-V characteristics are obtained for complex device breakdown characteristics over a wide range of bias and device physics, including bipolar BVceo, MOSFET snapback due to ESD stress, and even GaAs MESFET breakdown. Figure 2 shows the ESD breakdown of a MOS protection device stressed with a positive pulse at the drain with the gate, source, and substrate grounded. The region of negative differential resistance is due to the turn-on of the parasitic bipolar transistor formed by the drain, channel, and source regions of the MOSFET. Such simulations have helped in the design of improved protection devices (6). An important phenomenon revealed by the simulations is the critical role of parasitic resistance and capacitance in the protection MOSFET which affect the onset of breakdown during an ESD event. These parasitics are actually useful in the optimization of a protection-circuit design.

Spectrum Analyzer (S-parameters)

The extraction of high frequency device data using S-parameter characterization is well known and widely used for RF technology. The use of TCAD-based virtual spectrum analysis offers unique opportunities to understand both internal device effects and their interaction with circuit parasitics. Figure 3a shows schematically the connection of SPICE and a device-level simulator such as PISCES. The standard SPICE (version 3f) from UC Berkeley has been enhanced to include a numerical device model which allows SPICE to communicate with the device simulator through an interface routine. At each circuit iteration, SPICE determines the individual device bias conditions and executes a numerical device simulation. In return, the device simulator provides a current vector and an admittance matrix. SPICE continues to iterate until the requested tolerance have been found. As a result, a SPICE netlist may be used to write routines that electrically characterize a device by using physical models rather than an extracted model. Figure 3b shows the specific GaAs structure that was characterized using this method. The basic technology and layout information came from a commercial MESFET implementation (7). The LRC parameters are extracted directly using layout-based calculations based on the MIT multipole programs FASTCAP/FASTHENRY (8). Finally, Figure 3c shows typical S-parameter simulations using the mixed-mode virtual instrument. The results have been independently confirmed with experiments (8). In comparison to earlier simulations using only device simulations, the automated virtual instrument approach provides increased accuracy and efficiency by allowing SPICE to determine the appropriate device boundary conditions given a desired analysis.

Harmonic and Intermodulation (IM) Distortion

Improving the performance of wireless systems requires tighter specifications on bandwidth where distortion and intermodulation effects play a key and limiting role. The harmonic balance (HB) approach is widely used for intermodulation distortion analysis of nonlinear circuits. However, current frequency domain (HB) analysis packages are circuit simulation tools which require the active components to be characterized and modeled in advance. We have recently developed an HB device simulator that directly uses the PISCES physical model formulations in concert with new matrix solution algorithms. The result is the ability to directly determine the spectral content and IM distortion of semiconductor devices based on physical device simulation. The details of the numerical algorithms will be presented elsewhere (9). Figure 4b shows some very promising initial results. A two-tone large signal intermodulation distortion analysis is carried out on a 2D bipolar transistor (cross-section shown in Fig. 4a). The spectrum of the collector current clearly illustrates distortion components introduced by the nonlinearity of the device at high driving levels. Additional examples to be shown in the presentation will further extend the IM distortion analysis to modest sub-circuits (for example, power amplifiers for RF applications) where much more realistic circuit capabilities will be demonstrated.

Summary

This paper introduces a new concept of TCAD-based virtual instruments and demonstrates their application in a range of key technology evaluations that support high-performance circuit design. Curve-tracing is essential in obtaining smooth I-V data and can be most helpful in characterizing parasitic effects such as breakdown and ESD. In the area of high-frequency applications the analysis of S-parameters and IM distortion have also been considered. Here the emphasis is placed on the development of new analysis methods that support these virtual instruments and show interesting device-level effects that can be understood based on such enhanced TCAD capabilities.

Acknowledgment

The authors wish to thank Army Research Office, DAAL03-91-G-0152, and Semiconductor Research Corporations, 94-SJ-116, research support. Mentorship in the ESD work from Ben Tseng of AMD and fellowship support from AASERT/ARO (Rotella) and NSF (Troyanovsky) are gratefully acknowledged.

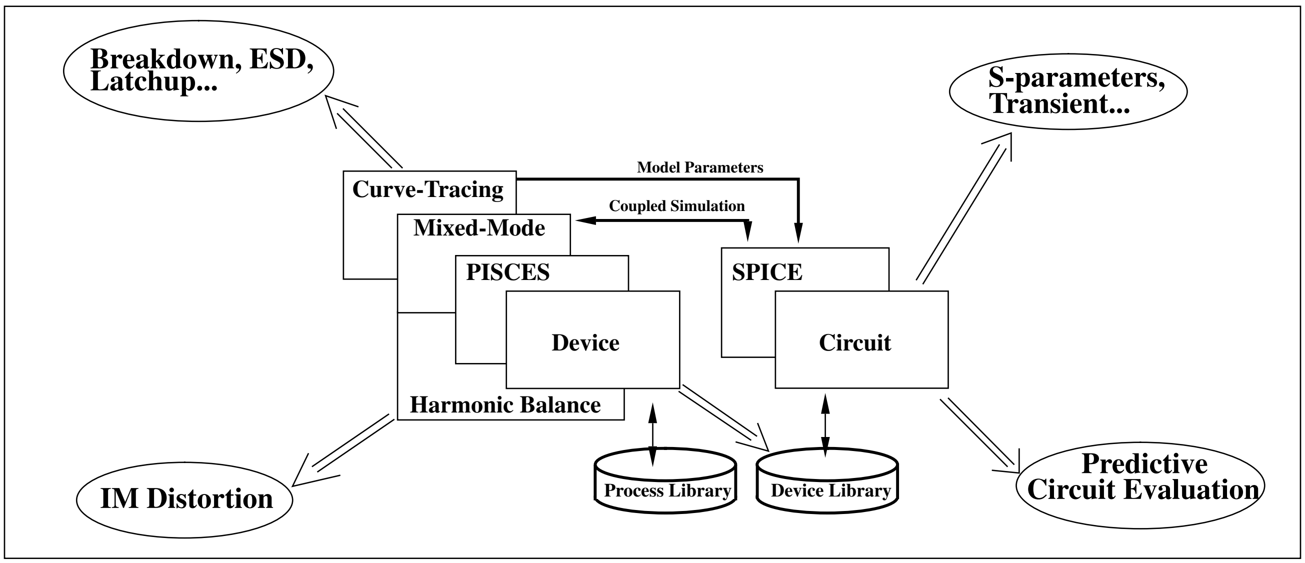
References

(1) D.B. Estreich, A. Ochoa, Jr., and R.W. Dutton, “An analysis of latch-up prevention in CMOS IC’s using an epitaxial-buried layer process,” IEDM Tech. Digest, pp. 201-204, 1978.

(2) D. Su, M. Loinaz, S. Masui, B.A. Wooley, “Experimental results and modeling techniques for substrate noise in mixed-signal integrated cir-

cuits," IEEE J. Solid State Circuits, p. 420, Apr. 1993.
(3) Z. Yu, R.W. Dutton, and R.J. Goossens, "Virtual instrument -- concept and implementation," Proc. SASIMI '93, pp. 289-294, Nara, Japan, Oct. 1993.
(4) R. J. G. Goossens, S. Beebe, Z. Yu, and R. W. Dutton, "An automatic biasing scheme for tracing arbitrarily shaped I-V curves," IEEE Trans. CAD, Vol. 13, No. 3, pp. 310-317, March 1994.
(5) Z. Yu, D. Chen, L. So, and R.W. Dutton, "PISCES-2ET---Two Dimensional Device Simulation for Silicon and Heterostructures," Stanford University.
(6) K. Taylor, Advanced Micro Devices, private communication.
(7) J. Sato-Iwanaga, Y. Ota, O. Ishikawa, K. Inoue, Z. Yu, and R.W. Dutton, "High-frequency small-signal analysis of GaAs MESFETs using AC simulation," Workshop on Advanced Nonlinear Device Modeling for Circuit Simulation, Asia-Pacific Microwave Conference (APMC), Tokyo, Dec. 1994.
(8) K. Nabors and J. White, "FastCap: A multipole accelerated 3-D capacitance extraction program," IEEE Tran. CAD, pp. 1447-1459, Nov. 1991.
(9) B. Troyanovsky, et al., "Large signal frequency-domain device simulation via the harmonic balance technique," to be submitted to SISDEP '95, Germany.

Figure 1: Schematic view of TCAD tool hierarchy which consists of three levels--circuit, device, and processing simulation. Note that the processing level is incorporated in process library in this representation.



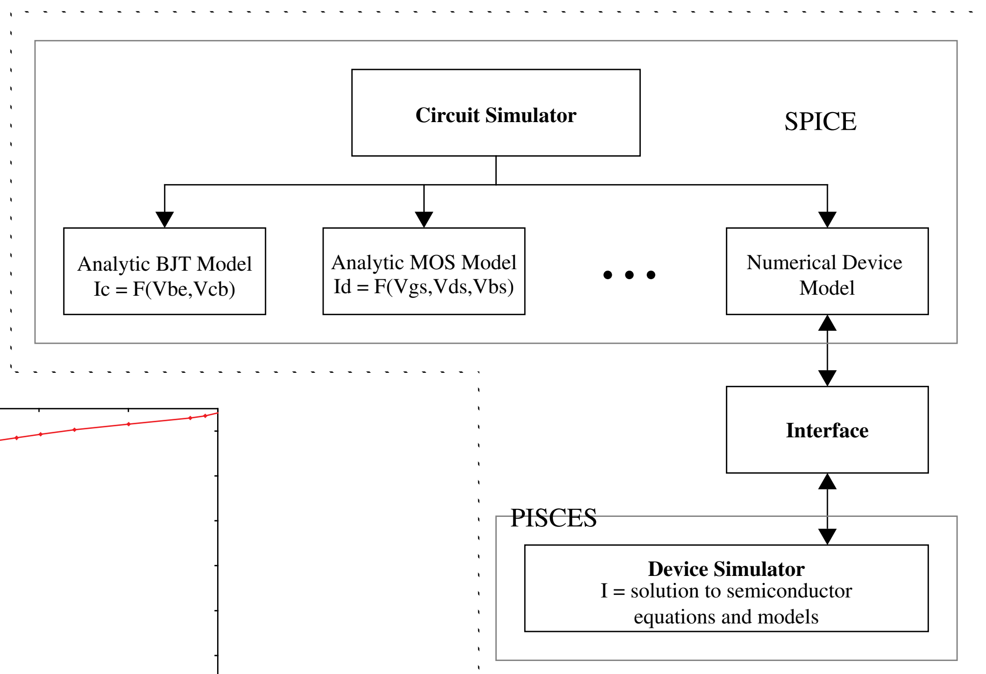


Figure 2: Circuit setting for investigating NMOS ESD behavior using pulse. The I_{ds} vs. V_{ds} is simulated by the curve tracer.

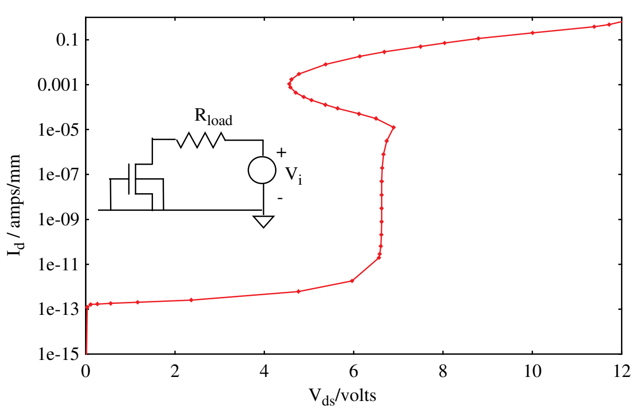


Figure 3a: Configuration including a device simulator as a numerical device model in SPICE simulation.All other features of the circuit simulator are preserved.

Figure 3b: GaAs MESFET with parasitics. The MESFET is represented by PISCES simulation and parasitics are extracted from the layout.

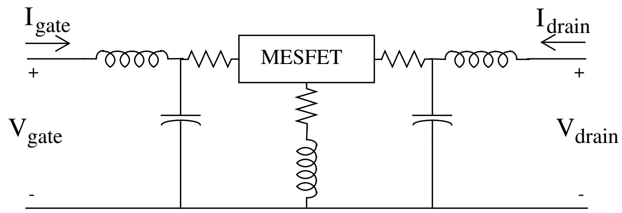


Figure 3c: Simulated S parameters of the circuit in Fig. 3b. Only S_{11} and S_{22} are shown.

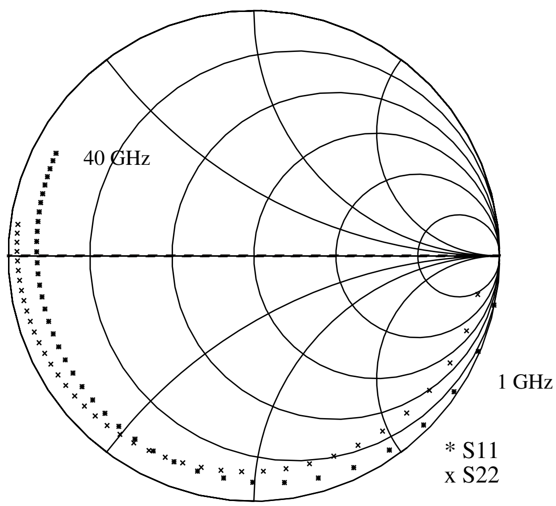


Figure 4a: Mesh for 2D section of a bipolar transistor used in HB intermodulation analysis.

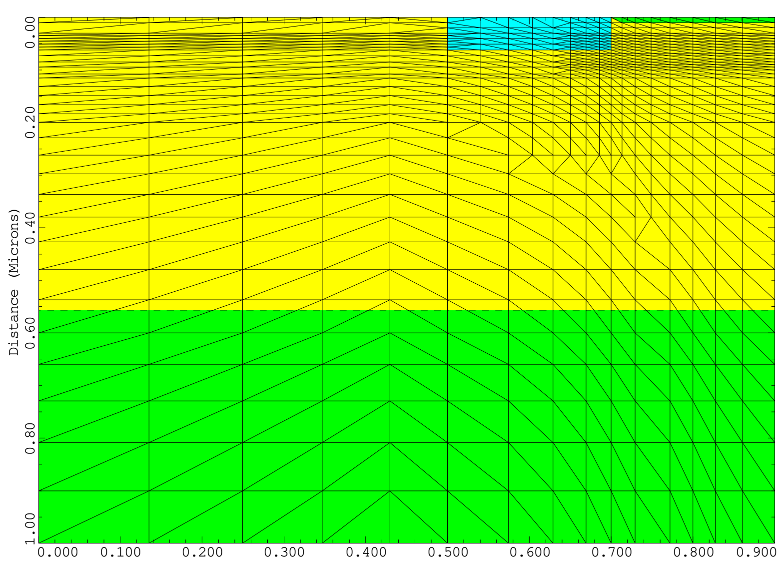


Fig 4b: Frequency spectrum of the collector current response to a two-tone large input signal. The extra frequencies due to the distortion is clearly shown.