EXModeler software for constructing SPICE models and macro-models for complex VLSI structures

by

Akis Doganis, Kevin Walsh and Prof. Panos Linardes*

Electrical Engineering Software Inc.
4675 Stevens Creek Blvd. Suite 200
Santa Clara, California 95051
(408) 296-8151 Fax (408) 296-7563

* Computer Science Dept.
Aristotelion University
Thessaloniki, Greece

Abstract

Deriving transient parameter for bipolar transistors (TF, TR, VTF, XTF, ITF) and modeling complex VLSI structures like LDD and Darlingtons is time-consuming and difficult. This paper will present a new optimization technology that can be used to obtain critical parameters in DC, CV, and transient for completely characterizing the complex devices necessary for accurate circuit simulation.

Summary

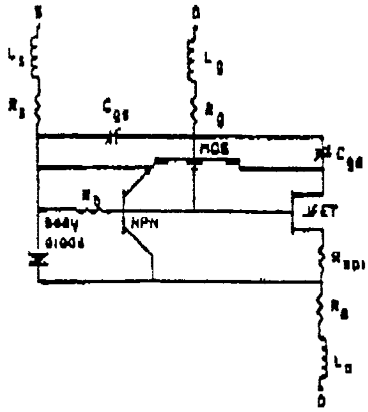
Characterization of macro-models representing complex devices as well as determination of the transient response device parameters is made possible using optimization and circuit simulation techniques. The application of simulation technology (Spice, Spice derivatives) and device optimization technology (Suxes, Suxes derivatives) allows engineers to easily and automatically determine all necessary model parameters affecting the device measured analog performance. This paper will describe an integration of both technologies (EXModeler) to determine the optimum device parameter values in the DC as well in AC and transient device performance.

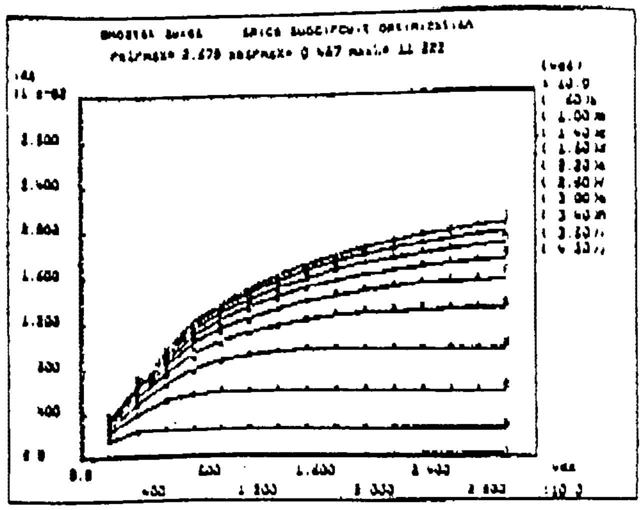
Obtaining bipolar device parameters for Spice based model transistors has developed from work done at Tektronix by Ian Getreu and work done on the SUXES program from Stanford. Still the problem of obtaining parameters that are consistent across all operating environments and which are closest to process data are difficult to obtain. Using the bipolar Gummel Poon Spice model, parameters TF, TR, VTF XTF, and ITF are critical transient parameters that required multiple guessing to obtain a close fit to the measured data. In the area of complex VLSI structures like LDD the limitations of the Spice based MOS models required special macro-models to obtain an accurate fit to measured data. These problems can now be addressed in a new software program called EXModeler.

The program uses a combination of gradient algorithms (steepest descent, Gauss-Newton and Levenberg-Marquardt) automatically selected according to the peculiarities of the error function. Minimizing the error function between the model performance and actual measured data generates the optimum set of device parameters The accurate algorithms available in Spice for direct simulation provide the basis for calculating the sensitivity of each element to the measured performance. The optimization algorithms quickly determine the changes necessary to element values, model parameters or device geometries to achieve better results. The program iterates using these algorithms until the measured performance and the simulated performance are within an error tolerance. The result is a set of device parameter values which give the best or closest match to the measured performance.

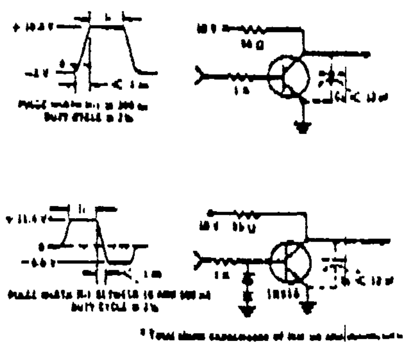
Measured device data in DC, CV and transient domains are simultaneously fitted to both internal Spice-like models and complex combinations of elements and transistors that form the macro-models. This operation involves coordination between device extraction and optimization and circuit simulation. Several examples of characterization will be described including the modeling of the bipolar transistors in all domains simultaneously, modeling LDD structures using a complex MOS model, modeling powerFETS with a complex element and transistor based macro-model. The results are very encouraging for offering methods of modeling other complex VLSI structures from detailed measured data and using the derived models in circuit simulation.

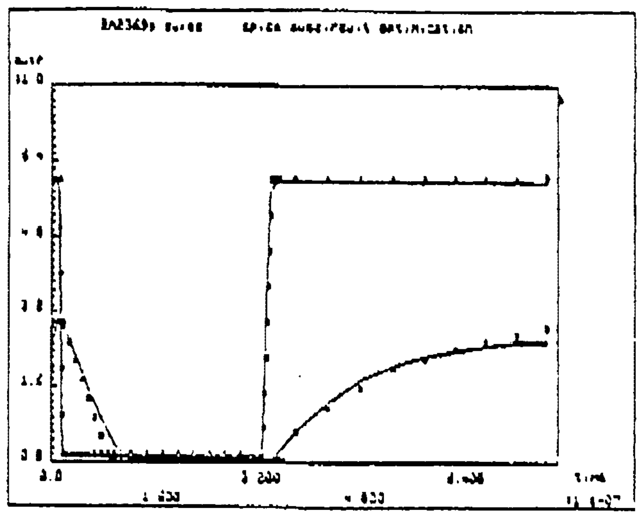
OPTIMIZATION OF POWER MOSFET SUBCIRCUIT



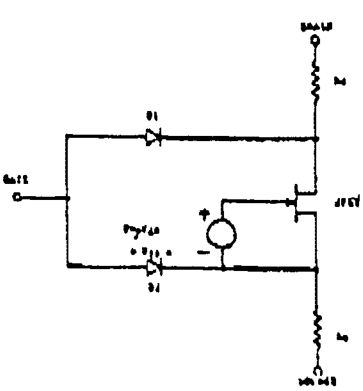


DELAY, RISE, STORAGE, AND FALL TIMES OPTIMIZATION





WHITE'S GaAs MACROMODEL OPTIMIZATION



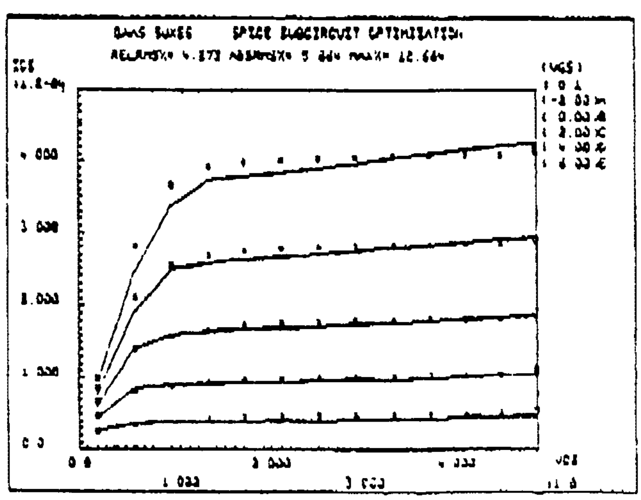


Figure 3