European Design Automation Conference (EURO-DAC'90); Glasgow, UK, March 12-15, 1990, pp.2.6.1-4.

TEGESED, a tool for efficient technological and geometrical characterization of semiconductor devices

A. Konczykowska
Centre National d'Etudes des Telecommunications
Laboratoire de Bagneux
92220 Bagneux, France

W.M. Zuberek
Department of Computer Science
Memorial University
St. John's, NL, Canada A1C-5S7

A b s t r a c t

Characterization of semiconductor devices in terms of their technological and geometrical parameters is described in this paper. This characterization is generated by a software tool composed of the SPICE-PAC simulation package and the symbolic simulator SYBILIN. The paper discusses the general structure of the program, its principle of operation, some implementation details, and its computational efficiency. A characterization of Heterojunction Bipolar Transistor (HBT) for microwave applications is used as an illustration.

1. INTRODUCTION

A complete, detailed characterization of semiconductor devices is of significant importance not only to circuit designers, but also to all researchers interested in improved or new technologies and manufacturing processes. This paper presents a software tool, TEGESED, that provides characterization of semiconductor devices in terms of their technological and geometrical parameters. Its basic feature is flexibility which is due to an "open" structure of the simulation package that constitutes the "backbone" of this tool.

In order to illustrate TEGESED's properties, one particular example is presented in greater detail. It is a characterization of transistors in the frequency domain, as required in microwave applications. The program is composed of two computer-aided analysis tools, the SPICE-PAC simulation package [12] which is used for nonlinear and time-domain analyses, and the SYBILIN symbolic simulator [7,8] for (linear) frequency-domain analyses. SPICE-PAC has been chosen because of its modular form and very flexible interfacing capabilities, while SYBILIN because of its efficient implementation of repetitive AC analyses. A brief description of SPICE-PAC and a short discussion of symbolic simulation are presented in the next sections, and are followed by an outline of the TEGESED's structure as well as a practic study of Heterojunction Bipolar Transistor characterization.

2. SPICE-PAC

SPICE-PAC is a simulation package that is upward compatible with the popular SPICE simulator [3,9,11]. It means that SPICE-PAC accepts the same circuit description language as SPICE (with a few minor exceptions) and provides the same set of circuit analyses, but it also supports a number of new features, not available in the original SPICE simulators. Examples of these extensions include:

- a uniform hierarchic naming scheme for all levels of subcircuits; subcircuit elements can be used in parameter lists and output specifications,
- dynamic (i.e., at the "simulation-time") definitions of analyses, their parameters and outputs,
- parameterized subcircuit invocations; subcircuit definitions can be modified by parameters passed to the subcircuit expansion phase,
- static and dynamic circuit variables; circuit variables are those attributes of circuit elements that can be modified during a simulation session; static circuit variables must be defined within the circuit description, and these definitions are needed by the package to implement a very efficient access to static variables, as required in circuit optimization; dynamic circuit variables do not require any definition, so they are very flexible but relatively slow,
- enhanced circuit elements, i.e., circuit elements with characteristics defined by users in the form of formulas or tables of values, etc.,
- enhanced analyses, i.e., circuit analyses which are extended by user-defined operations.

However, the most important difference between the SPICE program and the SPICE-PAC package is in their internal organizations. SPICE is a program with one, fixed sequence of analyses, indicated by appropriate parameters within the circuit description. The order of these analyses as well as the form of results are always the same since they are "built-into" the simulation program. SPICE-PAC, on the other hand, is a rather "loose" collection of simulation "blocks" (implemented by different components of the package), which can be combined together in many different ways (similarly to building a variety of structures from a set of LEGO-type elements). Typical examples of such "simulation blocks" include reading a circuit description, performing an analysis, changing values of (some) circuit elements, or redefining analysis parameters. The operations of the package are thus performed "on demand", as required by a particular application. In the case of interactive simulation, it is the user who - during a simulation session - selects the order, type and all other parameters of analyses. This means that this type of circuit simulation provides the user with a “feedback” which is unavailable in traditional “batch-oriented” simulators; within one interactive simulation session, the results of one analysis can be used to determine the new values of (some) circuit elements as well as the next analysis and its parameters. In cases of integrated applications, when the simulation package is used as a “generator” of circuit responses (e.g., circuit optimization in which it evaluates the objective function and - possibly - the constraint functions), the sequence of package’s operations, their types and parameters are determined by other software tools.

This flexible structure of the package makes it possible to combine the same set of “standard” analyses with several input processors accepting different forms of circuit specification (e.g., the SPICE input language, a functional-type circuit description, an output of a circuit extractor, etc.), to represent the results in different ways (graphical for the user, binary for further processing by other tools, textual for storing in a file). Furthermore, it is possible to replace some of the “standard” modules by dedicated, user-defined methods specialized to particular applications. The integration with the symbolic simulator is an example of such an enhancement.

3. SYMBOLIC SIMULATOR SYBILIN

The principle of symbolic simulation (see for example the list of references in [1,10]) is to derive analytic (or symbolic) network functions from a representation of a network. This means that all (or some of) circuit parameters are represented by symbols in the derived functions, and then the circuit responses can be obtained very efficiently by evaluations of the derived analytic formulas.

Symbolic simulators may use different circuit representations and different algorithms to derive network functions. The algorithm implemented in SYBILIN is sketched in Fig.1.

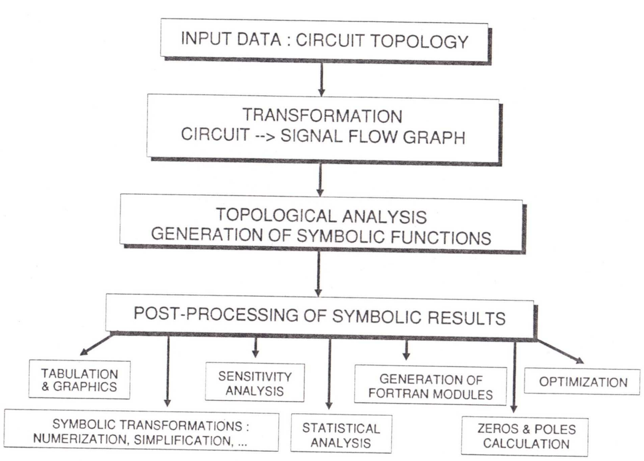

Fig.1. General organization of SYBILIN.

SYBILIN uses the Coates flowgraph representation [2]. Variables corresponding to graph nodes are the same as those used in the known Modified Nodal Admittance (MNA) method [6]. The characteristic functions are in the form of rational functions, in which the numerator and the denominator are the determinant and some subdeterminant of the Coates flowgraph. The formula for the determinant of the Coates flowgraph G is as follows :

$$det(G) = (-1)^n \sum_{t \in T} sgn(t) \prod_{1 \leq i \leq n} val(t_i)$$

where

n is a number of flowgraph nodes,

$t = (t_1, t_2, ..., t_n)$ is a 0-connection of a graph G,

T is a set of all 0-connections (0-connections are specific subgraphs),

$sgn(t)$ - is a sign of a 0-connection t,

$val(t_i)$ - is the weight of an element t_i

(the last three elements are described in greater detail in [1,2,10]).

Symbolic formulas generated by such a simulator may be used in several different ways. They can be analyzed (in the mathematical sense) to provide a qualitative evaluation of circuits under design. They can be subjected to “post-processing” in order to perform (approximate or exact) sensitivity analysis, or statistical analysis. They can be very useful in circuit optimization as they can replace many time-consuming circuit simulations with relatively simple evaluations of (symbolic) functions.

The network functions generated by symbolic simulators can also be represented in a form that is “computer executable”, i.e., a form that can be included into circuit simulations without any further processing. One of such possibilities is to generate a “machine code”, as was the case in some versions of SPICE (for CDC computers); obviously, such a solution is very “machine-dependent” because of the many differences between low-level machine instructions of different computers. A more general solution that is only slightly less efficient is to use interpretive techniques, i.e., to generate the symbolic functions in an intermediate representation (for example, the reverse Polish or the postfix notation) that is quite efficient to evaluate, but still preserves machine independence. Yet another solution (which is used in TEGESED) represents the symbolic functions by equivalent segments in a high-level programming language (in this case, Fortran); this generated code is then compiled, and linked with the simulation environment (using appropriate interfacing conventions to pass all the needed values of circuit parameters).

4. ORGANIZATION OF TEGESED

The TEGESED program has been obtained by combining the SPICE-PAC simulation package with the symbolic simulator SYBILIN. Only a brief outline of this program is given (a more detailed description will be presented in the final version of this paper).

TEGESED is composed of three basic parts:

A) input data processor and a generator of parameter values,

B) circuit analyzer,

C) generator of output results.

The circuit analyzer performs:

B1) calculation of electrical parameters for the current set of technological and geometrical data,

B2) analyses based on calculated electrical parameters,

B3) extraction of characteristic information.

This organization is presented in Fig.2.

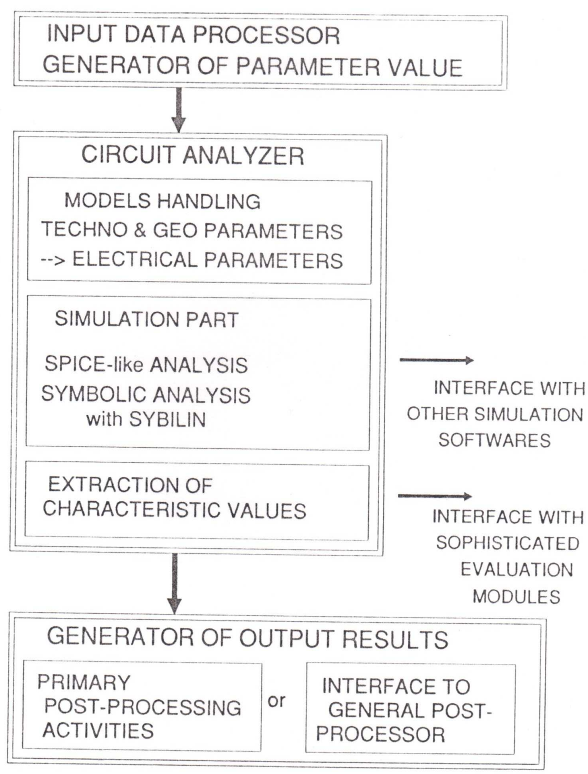

Fig.2, General organization of TEGESED.

Part A reads the input data and creates internal structures used by other modules. It also determines the changes of technological and geometrical parameters and organizes systematic generation of parameter combinations, as required by subsequent analyses.

Part B is the central part of TEGESED. It uses many modules from the SPICE-PAC package for changes of device parameters as well as for performing circuit analyses. It also contains an interface to the evaluation of symbolic functions generated by SYBILIN, that replace the original SPICE analyses in case of repetitive frequency-domain analyses of circuits with the same topologies.

Section B1 handles device models and evaluation of electrical model parameters on the basis of technological and geometrical data determined in the part A. SPICE-PAC provides simple mechanisms for manipulating parameters of the original SPICE models; it is thus straightforward to use any SPICE model with its electrical parameters related to technological and geometrical ones. Furthermore, it is possible to build models from SPICE primitive elements (resistors, capacitors, sources), and due to SPICE-PAC enhanced elements, the functions of such elements can be specified rather arbitrarily as table-driven elements or analytic functions implemented as user-defined routines linked with the package.

Section B2 is the proper simulation "engine". All types of analyses provided by SPICE, i.e., DC, AC, time-domain, noise, distortion, and Fourier analysis, can easily be performed using equivalent SPICE-PAC modules. Due to well-defined and standardized SPICE-PAC interfaces, it is possible to add new implementations of analyses to speed up the computations as well as to introduce new analyses. The symbolic simulation in the frequency domain is used at this level.

Section B3 is application-dependent since it extracts some characteristic values from the results of analyses performed in the step B2. Examples of such characteristic values include the delay time for time-domain analysis or the cut-off frequency and maximum oscillation frequency for frequency domain analysis. The set of implemented characteristic values is "open", so it is possible to add new definitions of many other parameters that can be extracted from the results of simulations. An interface with evaluation module based on expert system techniques are currently under investigation (see sections 5. and 6.).

Finally, part C is responsible for numerical and graphical representation of results. This part is not really sophisticated because the intention is to use a general postprocessor for final processing and presentation of results; in the future part C will be actually reduced to a postprocessor's interface.

5. CHARACTERIZATION OF HBT FOR MICROWAVE APPLICATIONS

The TEGESED software has been used for an evaluation of performances (in the microwave range) of the Heterojunction Bipolar Transistor (HBT) in GaAlAs/GaAs technology developed at CNET - Bagneux [5]. Such parameterized characterization usually means an exhaustive simulation in the frequency domain for different combinations of technological and geometrical parameters.

The HBT was represented by the modified Ebers-Moll model [4]. Some 30 technological and geometrical parameters were used to describe the device; examples of these parameters include doping density in the base, emitter and collector as technological ones and number of emitter stripes, length and width of the emitter, base and collector stripe, or emitter-base distance as geometrical parameters.

Each analysis step (B2 in section 4) executes the operating point analysis and then determines the frequency characteristic by repeatedly evaluating the symbolic functions generated earlier by SYBILIN. The operating point information is passed to the symbolic simulation where it substitutes some parameters in the symbolic formulas. The values of the cut-off frequency and the maximum oscillation frequency are extracted from the frequency characteristics. Fig.3 shows an example of maximum oscillation frequency as a function of the emitter stripe length and the collector doping density.

It should be noted that Fig.3 requires some 150 to 200 plotted points, i.e., 150 to 200 frequency characteristics (not frequency domain analyses) which correspond to different values of the selected parameters. To calculate each point, a complete set of analyses has to be performed, i.e., the operating point analysis and about 100 to 200 frequency domain analyses. As values of characteristic frequencies for different parameters can vary from tens of MHz to hundreds of GHz, the range of exploration is not known a priori. Therefore, first a global evaluation is carried out, and then the region for detailed exploration is determined. Since all these analyses are performed for the same circuit topology, the symbolic functions for all these analyses are invariant.

The extraction of characteristic information is sometimes quite difficult as unexpected situations may happen which are not covered by the definitions. These situations are easily identified and classified by qualified users, but may be quite troublesome for automatic processing. Therefore, some criteria on the quality of extracted information have been introduced. Three types of situations are identified: regular extraction, extraction with anomalies, and no-extraction (or impossible extraction). Each device characterization is accompanied by an anomaly report, which indicates all those cases which need user assistance in extraction of characteristic information. The set of built-in criteria may be tuned to fit better the characterized device.

6. CONCLUDING REMARKS

TEGESED performs characterization of semiconductor devices especially efficiently in the frequency domain, which is due to the symbolic approach; it allows to undertake device characterizations in cases when the classical simulators are rather useless because of prohibitive computation times. For example, the HBT characterization presented in the section 5 was carried out on a DEC VAX-11/750 computer running VMS, and it required about 120 hours of the CPU time (distributed over the period of about 1 month). An estimation of the speed-up factor due to symbolic simulation (with respect to the all-SPICE simulations) is between 15 and 20 (depending on the number of points on the frequency characteristic). This means that the same characterization without the TEGESED tool would require 1800 to 2400 hours of the VAX-11/750 CPU time.

As was mentioned earlier, automated extraction of characteristic information requires a flexible set of results evaluation criteria. Generally, these criteria are modified during the study as new information is acquired. Recent developments in expert systems provide techniques which could handle such unexpected situations much easier that the traditional algorithmic methods. A possibility of coupling TEGESED with an "expert" evaluation module is investigated. Some details about this approach might be included in the final version of the paper.

Another interesting area of further modifications of TEGESED is in full integration of SPICE-PAC and SYBILIN. It means that the same internal structures would be used by both tools, and this would simplify interchange of (internal) information. Also, the use of symbolic simulation in speeding-up nonlinear analyses is under investigation.

References

[1] W.K. Chen, "Applied Graph Theory - Graphs and Electrical Networks"; North Holland, 1976.

[2] C.L. Coates, "Flow graph solutions of linear algebraic equations"; IRE Trans. on Circuit Theory, vol.CT-6, pp.170-187, 1959.

[3] E. Cohen, "Program reference for SPICE 2"; Memorandum UCB/ERL M592, University of California, Berkeley, CA 94720, 1976.

[4] J. Dangla et al., "A CAD Model for Heterojunction Bipolar Transistor'; Proc. ESSDERC, Bologna, Italy, 1987.

[5] C. Gerard, "Analysis and optimization of frequency performances of the Heterojunction Bipolar Transistor GaAlAs/GaAs" (in French); PhD Thesis, Universite Paris VI, 1987.

[6] C.W. Ho, A.E. Ruehli, P.A. Brennan, "The modified nodal approach to network analysis"; IEEE Trans. on Circuits and Systems, vol.22, pp.504-509, 1975.

[7] A. Konczykowska, V. Morin, J. Godin, M. Bon, "Symbolic analysis for CAD of microwave circuits", Coll. Computer Aided Design of Microwave Circuits, London, 1985.

[8] V. Morin, "SYBILIN - a program for symbolic design of microwave integrated circuits" (in French); PhD Thesis, Universite de Bretagne Occidentale, Brest, 1988.

[9] D.O. Pederson, "A historical review of circuit simulation"; IEEE Trans. Circuits and Systems, vol.31, no.1, pp.103-111, 1984.

[10] J.A. Starzyk, A. Konczykowska, "Flowgraph analysis of large electronic networks"; IEEE Trans. on Circuits and Systems, vol. CAS-33, pp.302-315, 1986.

[11] A. Vladimirescu, K. Zhang, A.R. Newton, D.O. Pederson, A.L. Sangiovanni-Vincentelli, "SPICE Version 2G - User's Guide (10 Aug. 1981)"; Department of Electrical Engineering and Computer Sciences, University of California, Berkeley, CA 94720, 1981.

[12] W.M. Zuberek, "SPICE-PAC, a package of subroutines for interactive simulation and circuit optimization"; Proc. IEEE Int. Conf. on Computer Design (ICCD-84), Port Chester, NY, pp.492-496, 1984.