Transactions on Engineering Sciences vol 11, © 1996 WIT Press, www.witpress.com, ISSN 1743-3533

A Windows package for symbolic and numerical simulation of analog circuits

A. Luchetta,[a] S. Manetti,[a] M.C. Piccirilli[b]
[a]*D.I.F.A., University of Basilicata, Via della Tecnica, I-85100, Potenza, Italy*
[b]*D.I.E., University of Florence, Via S. Marta 3, I-50139 Florence, Italy*

Abstract

A Windows program package for both symbolic and numerical simulation of analog circuits is presented in this paper. It is provided with functionalities which permit the graphical schematic entry of the circuit, the symbolic analysis, the approximation of the symbolic results, the use of an external numerical simulator and the graphical postprocessing of both the symbolic and numerical simulation results. These functionalities can result very useful in the design of analog circuits, because they allow us to immediately evaluate the influence of both topology and component value changes on the circuit behavior.

1 Introduction

Numerical analysis is a powerful verification tool that allows the designer to test its circuit without assembling it. Verification is part of the design itself, because the results of this operation are used to modify the circuit up to the final version. However the determination of the causes of malfunctioning is not an easy task and requires a not negligible effort. In fact numerical analysis requires a well defined circuit with all its parameters fixed and yields as output a series of numbers in tabulated or plotted form without any indication about the circuit component influence.

On the other hand, if the simulation results are provided as a function of circuit parameters, the determination of the causes of malfunctioning is highly simplified, because it is possible to understand the influence of each parameter on the circuit behavior. Symbolic analysis meets this need. In fact it is a technique that permits to obtain, as result of a computer program, circuit network functions in closed form, where some or all the circuit elements and complex frequency are represented by symbolic parameters. It allows not only to correct eventual malfunctioning, but also to improve the performance of the circuit or even to reduce its complexity.

Of course, after the circuit modification has been implemented, it is always necessary to have a final accurate verification through the use of a reliable numerical simulator as, for example, SPICE. This explains how symbolic and numerical analyses can be considered two complementary approaches.

In this paper a Windows program package which constitutes an environment for the analysis and design of analog circuits is presented. The program uses both symbolic and numerical simulation techniques exploiting all the potentialities allowed by the two different analysis procedures. In particular the program package is provided with functionalities which permit the graphical schematic entry of the circuit, the symbolic analysis, the approximation of the symbolic results, the use of an external numerical simulator and the graphical postprocessing of both the symbolic and numerical simulation results. These functionalities allow the user to interactively evaluate the influence of both topology and component value variations on the circuit performance, then they can result very useful not only for the above mentioned applications of analysis and design of analog circuits, but also for educational/training purposes.

The program is named SAPWIN (Symbolic Analysis Program for WINdows) and can use as external numerical simulator SPICE-like programs. It presents many attractive features, such as the Windows application potentialities, the graphical schematic entry, the possibility to have various small signal model for semiconductor devices and to work on approximated network functions.

2 Program package description

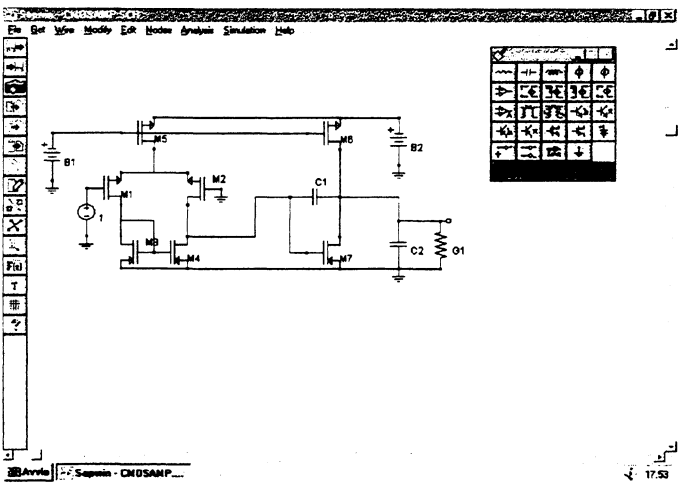

Figure 1: Sapwin main screen

SAPWIN is a Windows program package for symbolic and numerical simulation of analog circuits. It is constituted by the three modules SAPWIN, SAPEC and WINGRF, which can run in interactive or independent way. The first allows us to draw the circuit scheme and enter all commands, the second performs the circuit symbolic simulation, the third graphically processes the results of an already analysed circuit. In the following each of them will be described in detail, pointing out the main new features which they contain with respect to other similar program packages [1,2,3].

SAPWIN is the graphical schematic entry program. The main screen is a white sheet where the user can draw a circuit, by employing all typical Windows tools to copy, cut, paste, move, edit a component or a part of the scheme. Referring to the Figure 1, by means of the Get and Wire options the circuit is drawn. The allowed components (Get options) are:

- resistor;
- capacitor;
- inductor;
- independent voltage and current sources;
- the four kinds of controlled sources;
- mutual inductance;
- ideal transformer;
- ideal (nullor) and finite gain operational amplifier;
- semiconductor devices (bipolar and MOS transistors n-p-n or p-n-p type).

The output may be a voltage between a node and ground, between two nodes, or a current on a bipolar component.

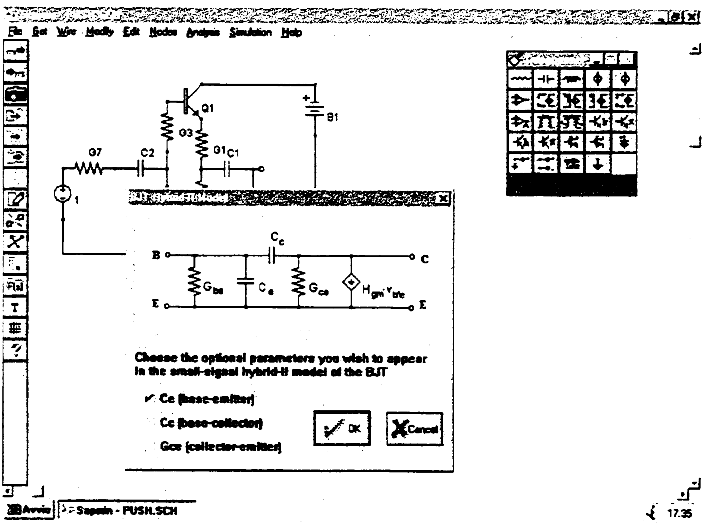

Figure 2: Get dialog box options for a MOS transistor

Transactions on Engineering Sciences vol 11, © 1996 WIT Press, www.witpress.com, ISSN 1743-3533

In particular for semiconductor devices linearized models of different complexity and with the chance of inserting in the analysis the chosen parasitic capacitances are available (Figure 2). This is an interesting innovation of the program which allows to employ user defined models as it can be done, for example, in SPICE.

There is the possibility to modify (copy, move, delete, etc..) the circuit scheme by means of the Modify option. The component values are assigned with the Edit option, while the Node option sets the circuit nodes. The File option allows to select already drawn circuit, while with the Analysis option the symbolic analysis starts.

An important characteristic of SAPWIN consists in its capability of generating the circuit netlist for an external numerical simulator (typically SPICE) and automatically launching it. With the Simulation option the external numerical simulator starts: the netlist is automatically generated and it is possible also to edit the desired commands for the used numerical simulator. Furthermore Windows environment allows us to compare numerical and symbolic results on the same screen, as it will be shown in the next section. The possibility of using both symbolic and numerical analysis can result very useful for the design of a circuit. In fact the influence of both topology and component value change on the circuit topology can be immediately evaluated by means of symbolic simulation, while the choice made with the approximate symbolic simulation can be accurately verified by means of a reliable numerical simulator.

Referring again to the Figure 1, on the left side of the screen all the allowed Windows tools are reported. A detailed on-line help indicated by a question mark (or Help option) is also available.

SAPEC (Symbolic Analysis Program for Electric Circuit) calculates the symbolic Laplace domain network functions of the circuit in expanded format with a variable "symbolism level": each component can appear in the network function with its name (a "symbol") or with its numerical value. It presents several innovative aspects with respect to previous versions [1, 2, 3]. In particular the latest version is provided with a completely renewed approximate analysis, during or after simulation. The approximation routine reduces the symbolic expression to the most significant terms and suppresses the others, with the maximum allowable error imposed by the user. The possibility of a symbolic expression approximation is necessary for medium and large size circuits, where otherwise the resulting functions would be lengthy and very difficult to interpret. In particular two different forms of approximation are possible: in the first form each coefficient of s is approximated by means of the elimination of the least significant terms, whose nominal value is smaller than a given magnitude, in the second form the least significant terms of each coefficient of s are replaced by their numerical values, while only the dominant terms remain in symbolic form in the final network function. In the approximation routine two different techniques are integrated into the same algorithm: a "local" technique, and a "global" one. The "local" approximation gradually removes all terms of the final expression smaller, in absolute value, than a given magnitude dependent on the user supplied error value and the original expression's mean value. A fine tune of the effective error is then obtained by applying a "global" approximation on the remaining part of the expression, until the exact user supplied error value is reached [4].

WINGRF is the graphical postprocessor. It is able to show the network function and moreover to plot gain and phase, time delay, pole and zero position, time waveform of step and pulse response, also in parametric form, plotting on the same graph different curves related to different component values. In particular impulse response and step response are obtained by numerical Laplace transform inversion, using Pade' approximation and stepping algorithm [7, 8]. If the function is not proper, the algorithm is preceded by long-divisions performed by the Vlach algorithm [9]. For pole and zero computation the Moore's method [10] is used.

The program package is completely developed in a C++ object-oriented methodology. This fact gives us a good flexibility; the main simulation module SAPEC, for example, is subdivided into several classes as shown in the block diagram of Figure 3. Each component class is derived by the higher level abstract class Device and contains the routines for the single component manipulation. The class Matrix, together with its "friend class" Input, interprets the input ASCII file created by SAPWIN leaving from the drawn circuit scheme and contains an extremely efficient Laplace expansion algorithm for the determinant calculation [4] of the circuital symbolic matrix written in the Modified Nodal form [5]. The determinant calculation is improved performing a dynamic selection, at each recursion step, of row i and column j along which to develop the minor [6].

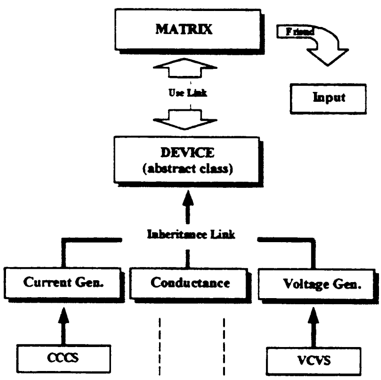

Figure 3: SAPEC class block diagram

3 Applicative example

As example to show the program potentialities let us consider again the circuit shown in Figure 1, a two stage CMOS op amplifier. Part of the complete symbolic network function is shown in Figure 4. It is produced by the Analysis option: in general, the approximate symbolic network function has a much simpler form with respect to the complete expression and few percent of user supplied error may cause the cancellation of hundreds of terms in a medium size circuit.

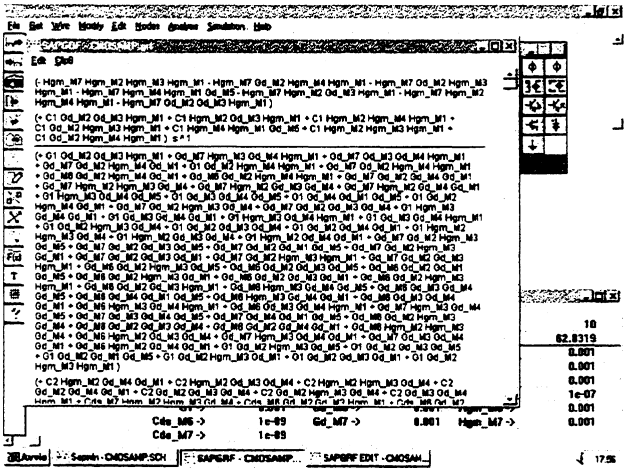

Figure 4: Complete symbolic network function and graphic postprocessor entry

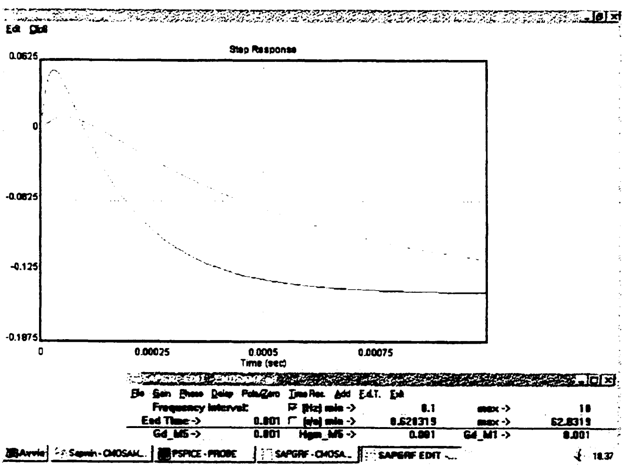

Figure 5: Graphic postprocessor output relevant to symbolic function (gain with two different values for C_2)

Transactions on Engineering Sciences vol 11, © 1996 WIT Press, www.witpress.com, ISSN 1743-3533

Once the symbolic network function is obtained, it is possible to use the graphic postprocessor by choosing the desired function and selecting the component fields and filling them with numerical values (if the component values have already been chosen with the Edit option, this operation is not required). The gain relevant to two different values for the capacitor C_2 of the MOS amplifier is shown in Figure 5. In Figure 6, instead, the gain obtained with the external numerical simulator (SPICE) is shown together with the gain relevant to the approximate symbolic analysis. Finally an example of time-domain step response is shown in the Figure 7.

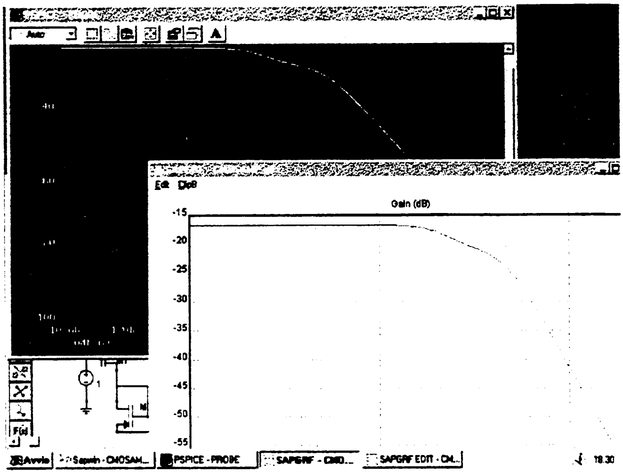

Figure 6: Gain relevant to the symbolic and numerical simulations shown on the same screen

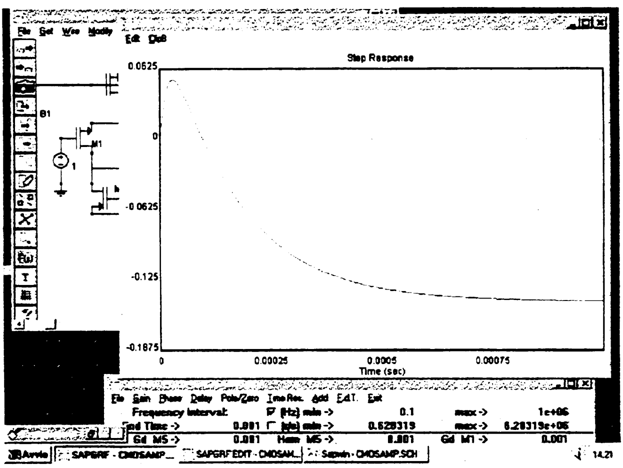

Figure 7: Step response relevant to the symbolic network function

Transactions on Engineering Sciences vol 11, © 1996 WIT Press, www.witpress.com, ISSN 1743-3533

4 Conclusions

A Windows program package for both symbolic and numerical simulation of analog circuits has been presented. The program, named SAPWIN, can result very useful for the design of analog circuits and for educational/training purposes. It is entirely written in C^{++} language and runs under Windows 3.1 on a PC 80386 CPU or more (the coprocessor is not required, but it is clearly useful). Finally the program has many attractive features for the user:

- it works in Windows environment allowing all the Windows applications potentialities;
- it can use, as external numerical simulator, a SPICE-like programs simply using a graphical schematic entry, without the need of specific input files;
- it allows voltage and current output, it is so very easy to obtain also input, output or transfer impedances and admittances;
- it can provide approximated network functions where not significant terms are eliminated, so allowing an easier understanding of the circuit itself.

References

1. Liberatore A., Manetti S. "SAPEC - A personal computer program for the symbolic analysis of electric circuits", in IEEE ISCAS/88, pp. 897-900, *Proceedings of International Symposium on Circuits and Systems*, Helsinki, Finland, June 1988.
2. Manetti S. "New approach to automatic symbolic analysis of electric circuits", *IEEE Proceedings Part G*, Vol. 138, No. 1, pp. 22-28, February 1991.
3. Liberatore A., Luchetta A., Manetti S., Piccirilli M. C. "A new symbolic program package for the interactive design of analog circuits", in IEEE ISCAS /95, pp. 2209-2212, *Proceedings of International Symposium on Circuits and Systems*, Seattle, Washington, USA, May 1995.
4. Gielen G., Sansen W. *Symbolic analysis for automated design of analog integrated circuits*, Kluwer Academic Publishers, 1991.
5. Vlach J. and Singhal K. *Computer methods for circuit analysis and design*, Van Nostrand Reinhold Co., New York, 1983.
6. Liberatore A., Luchetta A., Manetti S., Piccirilli M. C. "Symbolic generation of devoted simulators for non-linear circuits", in IEEE ISCAS/94, pp. 456, *Proceedings of International Symposium on Circuits and Systems*, London, United Kingdom, May 1994.
7. Singhal K., Vlach J. "Computation of time domain response by numerical inversion of the Laplace transform", *Journal of the Franklin Institute*, Vol. 299, No. 2, pp. 109-126, February 1975.

Transactions on Engineering Sciences vol 11, © 1996 WIT Press, www.witpress.com, ISSN 1743-3533

8. Singhal K., Vlach J. "Method for computing time response of systems described by transfer functions", *Journal of the Franklin Institute*, Vol. 311, pp. 123-130, 1981.
9. Vlach J. *Computerized approximation and synthesis of linear network*, Wiley, New York, 1969.
10. Moore J. B. "A convergent algorithm for solving polynomial equations", *J. ACM*, pp. 311-315, April 1967.