SWITCHED CIRCUITS SIMULATION SOFTWARE

Ing. Tomáš LUKL, Doctoral Degree Programme (1)
Dept. of Telecommunications, FEEC, BUT
E-mail: thomlu@post.cz

Supervised by: Dr. Vít Novotný

ABSTRACT

Switched circuits are often used in signal processing circuits (such as filters, modulators or A/D converters) or switched power sources. In this time there are modern active circuit elements, which allow us to increase switching frequency and therefore maximum frequency of processed signals. But these elements are not implemented in any known or available switched circuit simulation program. Text bellow briefly describes issue of numerical analysis of switched circuits and presents developed switched circuit simulation software for Matlab environment.

1   INTRODUCTION

Usually used programs for electric circuit simulation can not be used for switched circuit simulation. Not solved problem with inconsistent conditions at switching times is one of main reasons why not to use these programs for these purposes. Inconsistent conditions may be caused by e.g. parallel connection (using an ideal switch) of two capacitors with a different potential. We can bypass this by using of real switches (most often MOS transistors), but great extension of simulation time occurs. These programs also do not provide us results what we are interested in at switched circuit simulation. On the other hand programs for switched circuit simulation are usually not available and comfortable.

Due to these and other reasons it was necessary to develop software specialized for switched circuit simulation with graphical interface. It should allow us to create and edit electrical schema, models of new devices and so on. There was necessary to choose right numerical methods for switched circuit analysis, such as numerical integration and iteration methods, modified nodal analysis – MMA, during development. Simulation algorithm was developed for chosen numerical method and imbedded into environment of new program. This program names SCISIP – Switched CIrcuit SImulation Program. It is available in version for mathematical tool MathWorks[©] Matlab[®] r.13 at present, which is available at most universities.

2   SWITCHED CIRCUITS

Switched circuits are divided into three groups.

- Switched capacitor networks – SC are circuits with charge carrying. These circuits have high accuracy of transfer function. Base SC elements are capacitors (voltage memory cells), switches and active elements (such as OAs, CFAs, OTAs, BOTAs or current conveyors). These modern active elements can work in current mode. This mode brings us enhancements in higher reachable frequencies of an element and thus increasing of switching frequency [4].

- Switched current networks. Base switched current circuit elements are current memory cells, current mirrors and active elements.

- Switched power sources – high efficient power sources. They consist of diodes, transistors and thyristors which are, for simplicity and simulation acceleration, replaced by models with ideal switches. But we have to use nonlinear models for other circuit elements, because circuit variables achieve great ranges.

3   SCISIP

For program developing known mathematical tool *MathWorks© Matlab® 6.5* was used. There were a lot of reasons, such as C++ programming language similarity, large amount of pre-defined functions for work with vectors and matrices (it is useful for simulation algorithms applications – MNA) or easy GUI creation (GUI – **G**raphical **U**ser **I**nterface). Possibly only disadvantage is necessity of having and running Matlab environment. Abilities of Matlab OOP technique were fully used [3].

Modularity of program SCISIP allows us to use different (but very similar) tools to define a symbol (Symbol Designer) or model of a new device (Device Creator), to edit electrical scheme (Schematic Editor), to define simulation parameters and to display simulation results (Simulation Manager). All tools are integrated into main program - **Program Controller**. This program controller should create and manage projects (as known from other programs not only for circuit simulation).

3.1   SYMBOL DESIGNER

Symbol Designer serves as tool for a device symbol creating and editing. Drawing area is working desktop for placing graphical elements of a symbol. There is defined necessary amount of usual graphical elements such as circle, line and some special, e.g. a pin. Wires are connecting into the point defined by this pin object. There are pre-defined layers and two-layer grid for comfort and lucidity using. Created symbol is saved into a disc file with ***.sym** extension.

3.2   DEVICE CREATOR

Device creator is simple graphical tool for creating devices. Device is created by a schematic symbol and a model combining. Model can be created directly in the Device Creator tool or can be loaded from a disc file. *Template Model* can be used for creating a model. This template is pre-defined model of given type (e.g. diode – it has defined certain parameters). We can also directly create a template (in the case when any pre-defined

templates do not go with our device). This template is defined by e.g. a set of equations, which are determined by transfer, input or other characteristics or points of measured curve. These equations are parametric, where their parameters are the parameters of given type of device (e.g. for diode devices it can be cut-off current in forward direction I_S).

3.3 SCHEMATIC EDITOR

Schematic Editor is tool for graphical electrical circuit editing. It uses devices created by Device Creator Tool. Some functions for in-schematic model editing are added in this tool (this editing influences just the concrete device which is editing). Output of Schematic Editor Tool is a *Netlist*. *Netlist* is a text file describing electrical schema.
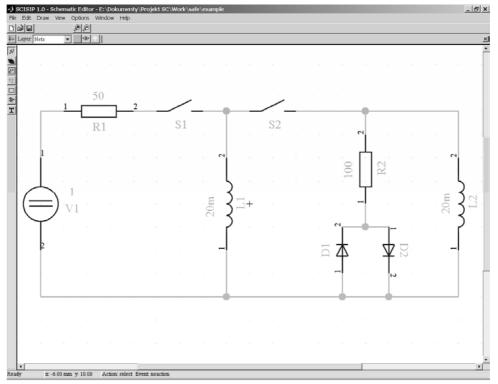

Fig. 1: *Schematic Editor Tool window with switched circuit schema.*

3.4 SIMULATION MANAGER

Simulation manager tool drives circuit simulation process. Its input is a file (or data structure) with *Netlist*. We can set some simulation parameters in this tool such as simulation time range. Tool automatically determines the circuit type (linear or non-linear, switched or non-switched). The output is naturally set of graphs and values (in the case of bias point computing).

3.4.1 SIMULATING CORE

Simulating core is implemented into Simulation Manager. This core is set of functions to obtain data from *Netlist* and to compute necessary curves of circuit variables.

Process of simulation consists of four steps [1]:

- simulation constants setting (e.g. $\varepsilon_r, \varepsilon$ for all variable)

- times vector creating, at which is necessary to compute solution (e.g. switching times)

- Bias point resolution through Newton-Raphson method

- MNA matrix compiling

Procedure of MNA matrix compilation for circuit with non-linear accumulating elements is as follows. We perform:

➢ Discretization – it is just time correspondence $t = t_n$ for linear resistor or conductance. For accumulating elements it means implementing of numeric integration formulae. Accumulating elements transform due to this to non-accumulating with constant either voltage or current source.

Used numeric integration methods:

○ Forward Euler method
$$x_{n+1} = x_n + hf\left(x_n, t_n\right),$$ (3.1)

is explicit first order method used as predictor.

○ Backward Euler method

$$x_{n+1} = x_n + hf\left(x_{n+1}, t_{n+1}\right)$$ (3.2)

is implicit one-step first order method.

○ BDF methods

$$\dot{x}_n = \sum_{k=0}^{l} \gamma_k^{(l)} x_{n-k},$$ (3.3)

which are backward Euler method generalization.

➢ Linearization – it is perform for non-linear elements by applying of iteration formulae for non-linear equations solution. Non-linear element transforms into linear element and either voltage or current source defined by state variables from previous iteration step. Newton-Raphson method was used as iteration method.

Example – base formulae for non-linear capacitor changes during solution as follows:

$i_C = \dot{q},$

$i_{C,n} = \eta\left(q\left(u_{C,n}\right) - l_n^{(p)}\right),$ diskretization, (3.4)

$i_{C,n}^{(k+1)} = \eta\left(q\left(u_{C,n}^{(k)}\right) - l_n^{(p)}\right) + \eta\dfrac{\partial q\left(u_{C,n}^{(k)}\right)}{\partial u_{C,n}}\left(u_{C,n}^{(k+1)} - u_{C,n}^{(k)}\right),$ linearization.
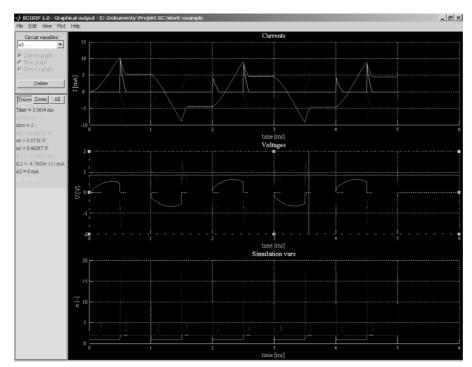
Fig. 2: *Results of simulation of circuit (see Fig. 1) with 500Hz/1V feeding.*

4 CONCLUSION

SCISIP is still being developed and new possibilities and abilities are added into single tools. Above all is working on increasing the robustness of NR algorithm (to improve ability counter to overflow, oscillations or divergence), extending supply of real device models and leveling the program comfort up.

Program can be used in theoretical courses like Electrical Circuit Theory. Students could improve their knowledge in the field of switched circuits and check up switched circuits properties with this program.

REFERENCES

[1] Ogrodzki, J.: Circuit Simulation Methods and Algorithms. CRC Press, Inc., ISBN 0-8493-7894-X, USA, 1994

[2] Kielkowski, R.: Spice – Practical Device Modelling. McGraw-Hill, Inc., ISBN 0-07-911524-1, USA, 1995

[3] The Mathworks, INC. Matlab 6.5 R13 manual, USA, 200

[4] Novotný, V.: MATSC - the program for SC circuit analysis in time domain. Workshop proceedings of international science symposium TELECOMMUNICATIONS AND SIGNAL PROCESSING - TSP'99, p. 56-59, Brno, 1999