

Circuit Analysis and Simulations through Internet

Jiří Hospodka

Department of Circuit Theory
Czech Technical University in Prague
Prague, Czech Republic
Email: hospodka@fel.cvut.cz

Jan Bičák

ASICentrum
a Company of the SWATCH GROUP
Prague, Czech Republic
Email: Jan.Bicak@asicentrum.cz

Abstract—This paper presents an application suitable for analysis of electric and electronic circuits through internet. The application is based on PHP scripts and uses SpiceOpus and Maple program with special library PraCAN as a computation engine. Continuous-time linear and nonlinear circuits as well as periodically switched linear circuits can be analyzed. Results can be obtained in symbolic or semisymbolic form for the case of linear circuits analyzed by Maple with PraCAN. Description of the circuit can be entered through a graphical schematic editor. It is a Java applet for scheme drawing and netlist creation. The possibilities of the application are demonstrated on a number of circuit analyses. The whole system was developed at the Department of Circuit Theory, for research and teaching support.

Keywords—web-based application; simulations; analysis.

I. INTRODUCTION

Nowadays, many systems for circuit analysis are available. Conventional programs like PSpice[®], Micro-Cap, WinSpice [3], [4], [5], etc. are single-purpose programs. Evaluation versions of these programs are frequently used for teaching support. Most of them solve the task only numerically, hardly any program makes symbolic or semisymbolic computing possible. Its use is connected with installation of mentioned software to the user computer. In contrast to this a web-based system offers the advantages of open and remote system for circuit analysis. These systems combine rich client technology, and circuit simulation, and provide convenient user interface for simulation capability. Web-based simulation environments, combining distance education, group training and real-time interaction, can serve as a good approach. The web-based virtual laboratory system for electronic circuit simulation (ECVlab) [7], trainer for electrical circuit analysis [8] or application [11] can serve as examples. Pages [9] based on C++ CGI Toolkit in Ch [10], provide interactive web-based calculation (computation of mathematical formulas in C expression, complex, matrix computation, and grade point average calculation), 2D/3D plotting, numerical analysis (analysis of linear systems, differential equation solving, integration, non-linear equations, Fourier analysis), OpenGL graphics and control system design and analysis (analysis of continuous-time or discrete-time linear time-invariant (LTI) control systems described by particular transfer function). Special application called "Remote Wiring and Measurement Laboratory" (RwmLAB) is described in [14]. RwmLAB is intended to address real-time remote wiring of electrical circuits and real

data acquisition over the Internet instead of using simulated data. Simulations through internet using Spice (Spice internet Packages – SIP) were introduced in [11], [12], [13], where internet serves as a graphical user interface for the program simulator – Spice. It is based on CGI (common gateway interface) scripts, PERL and PHP scripting language. Similar application is also the system for electric filter design. Web-based application [15] uses Maple[™] [6] as a computational engine with special package Syntfil [16] developed by authors. However synthesis of electric circuit is performed instead of circuit simulations.

The application presented in this article is based on client-server concept [17], which uses special simulation program on the server side. However the described system offers better user interface and greater scope of simulations including symbolic analysis and analysis of periodically switched linear (PSL) circuits, i.e., circuit with switched capacitors (SC) or switched currents (SI). Representation of circuit description is graphical using schematic editor like Java applet [1].

Our motivation was to create application for circuit analysis with a great potential. This goal can be realized by an easy-to-use web-based application, which is available for wide range of users. It can combine several simulation engines, can be accomplished using any computer with internet connection and can be easily administrated and updated, because it runs on a server.

The paper is divided into the three main parts. Programs used for analysis are described first. Introduction of application description including schematic editor follows. Examples of circuit analyses are shown to demonstrate usability of the application and its facilities at the end of the paper.

II. PROGRAMS FOR ANALYSIS

A forementioned application uses client-server conception where programs are installed and run on the server side. It is necessary that programs on the server side have to enable batch processing for easy operation through control scripts. Circuit analysis is provided by two main programs. The numeric analyses are performed by SpiceOpus, while semisymbolic and symbolic analyses including switched circuit are powered by Maple program with special package named PraCAN.

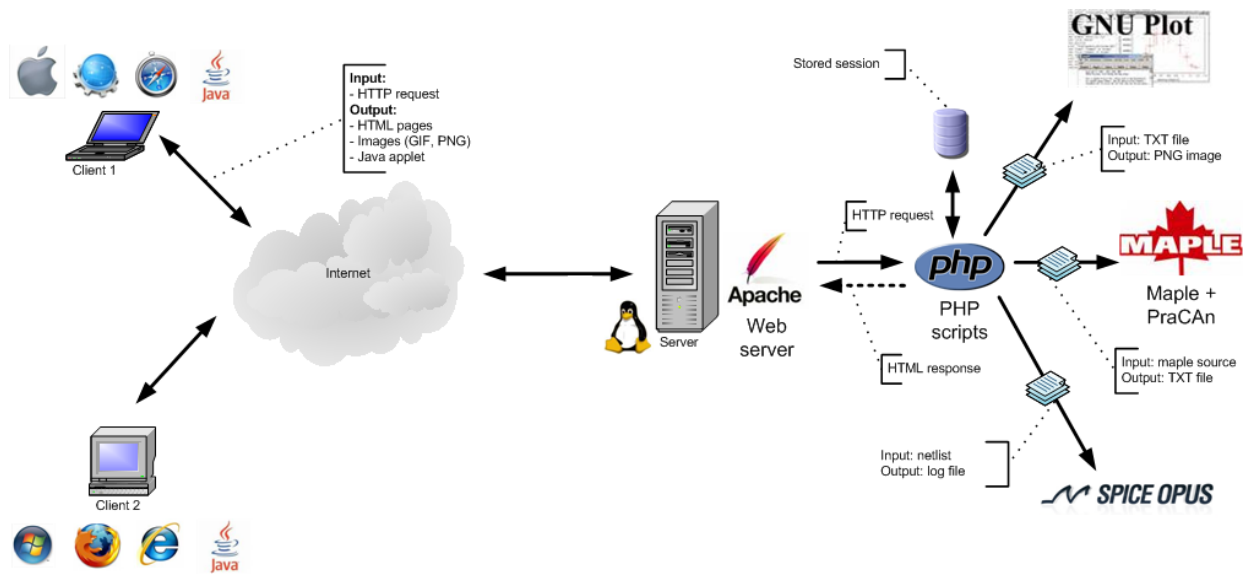


Fig. 1. Principle of the client-server concept used for the interface realization.

A. SpiceOpus

SpiceOpus is a circuit simulator with optimization utilities [18]. It is a recompilation of the original Berkeley's source code [2] for Windows and Linux operating systems. Georgia Tech Research Institute's XSpice mixed-mode simulator was added to the Berkeley code. The simulator includes an interpreted programming language called Nutmeg, which allows interactive Spice sessions. The program compilation is powerful enough to support the mentioned application.

B. PraCAN Package

PraCAN package is a library of functions for Maple, which facilitates the symbolic and semisymbolic analysis of continuous and discrete-time linearized circuits. PraCAN is the acronym for Prague Circuits Analyzer [19]. The input syntax for circuit description is almost the same as in Spice program. The package contains functions for parsing circuit description (netlist), which enable easy identification of syntax errors. Therefore the package is used for parsing netlist in the application. Next functions are designed for the analysis of continuous-time linear circuits as well as the periodically switched linear (PSL) circuits.

The package originates from SCSyrup package [20], [21] for frequency analysis of idealized SC and SI circuits. The functions used in SCSyrup package are based on the analysis of SC circuits using nodal charge equations. The analysis of SI circuits is also based on nodal equations but the currents are used instead of charges [26]. The package was modified and algorithms for PSL circuits were added to make it possible to analyze SC or SI circuits with real qualities such as switch resistances r_{on} and r_{off} . The package was renamed to PraSCAN (Prague Switched Circuits Analyzer), which uses algorithms based on modeling of periodically switched networks using mixed s - z description [25]. SCSyrup

was a table-based package whereas PraSCAN is a module-based package.

PraCAN has been created from PraSCAN package [22], [23], which was completely rewritten and new functions were added for continuous-time circuit analysis. The package preserves all functions of SCSyrup and it can be used as its complete replacement with the same results. The circuit is processed by modified nodal voltage method [26]. Unlike the methods in [24], the presented method for multiphase PSL circuits provides symbolic analysis in frequency domain and closed form solution in time domain. The linear system with the time-varying parameters is modeled by nonstationary transfer functions $K(s, t)$. If the parameters vary periodically (e.g., in SC and SI circuits with externally controlled switches), then the system response contains both continuous and discrete time parts and it can be described by a generalized transfer function GTF [25]. The frequency response of the system is obtained by substitution of both s and z . PraCAN package also contains function for direct calculation of time response. The response can be calculated with respect to real input signal character, i.e., including also the so-called leakage effect if Sample&Hold circuit is not used in the input of the analyzed circuit. This way actual spectrum of the signals can be calculated including frequency response of the circuit with undersampling.

III. WEB-BASED APPLICATION

The application is based on www interface, which utilizes www client-server concept. The computation and the interface programs run on the server and a user uses standard www browser (Internet Explorer, Mozilla Firefox, Opera, etc.) as a graphical user interface. This principle is illustrated on the following flow-chart, see Figure 1.

The server runs under operating system Linux. The analysis of the required circuit is solved using SpiceOpus program or

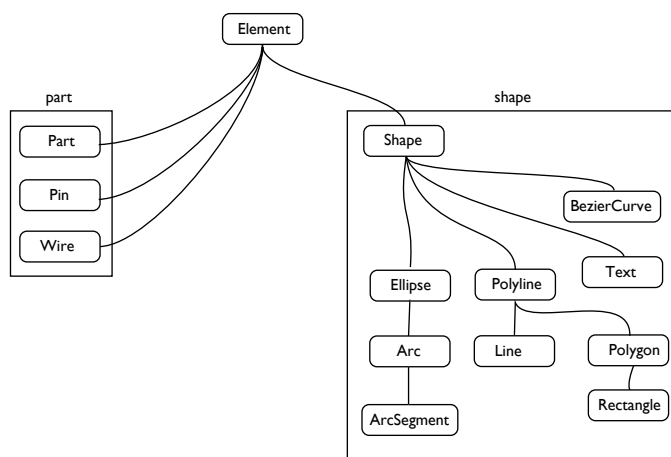


Fig. 2. Elements hierarchy of schematic editor.

PraCAan package in Maple. It runs using batch-processing, which is necessary for the interface where the programs are called by the PHP scripts [17]. According to client requests the results may be presented by the dynamically created www pages. These pages are provided to the client by means of HTTP server Apache. Described application of circuit analysis was realized according to this model.

Input requests are inserted into the forms of the application using www browser. The program in JavaScript tests the validity of these requests before sending them to the server, where they are tested too. Input files for Maple are generated from the input requests by scripts in PHP and results are saved in separate files. The PHP scripts process these files and create the structure of dynamic www pages, which are sent to the client. It is necessary to solve many other problems, for example, to distinguish simultaneously connected users, deleting temporary files and directories, etc. These tasks are solved using cookies and session variables (PHP). Other programs are used for additional functions – GnuPlot program is used for graph drawing and typographical system \LaTeX for graphical representation of terms. Procedure of calling GnuPlot and \LaTeX is analogous to a procedure of calling Maple or OpusSpice.

In contrast to conventional programs it is possible to create nonstandard analyses and different types of parametric analyses, using batch processing managed by PHP scripts using this application. These analyses can be easily defined by user without studying any program routines as is demonstrated in examples in Section III-B.

A. Schematic Editor – Application for Visual Scheme Editing

Schematic editor is being developed to help users with the creation of electronic circuit schemes by providing means of visual editing. It also allows an export of created schemes using variety of formats from SVG to netlist. The application supports plugins, which are loaded dynamically during the application start. Plugins can receive notification in case of scheme modification, so they can update properly.

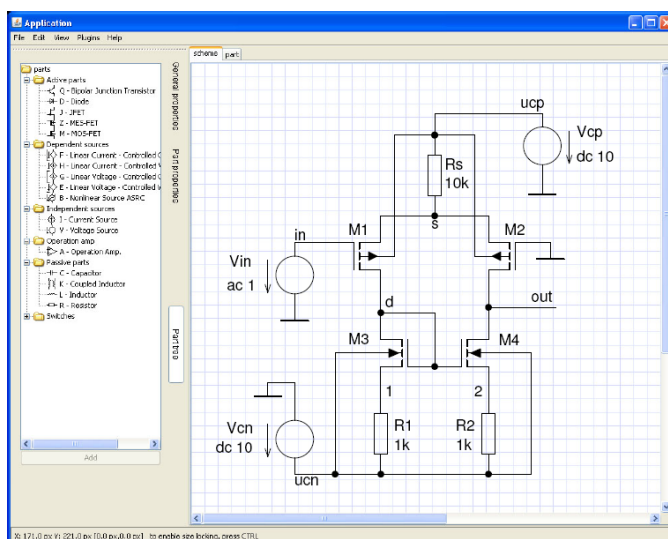


Fig. 3. Graphical interface of schematic editor.

The application of the editor is written with regards to object-oriented programming (OOP) techniques using Java 5. Architecture is modular, each module has specialized scope. The modules are divided into:

- *Configuration module* – implements classes, which are responsible for configuration,
- *Elements module* – implements all graphical elements used in application,
- *Gui* – contains classes responsible for graphical user interface (GUI) of application,
- *Manipulation* – contains classes, which implement all supported manipulations,
- *Units* – implements variety of units,
- *VectorEditorEngine* – core module, implements the most basic functionality of the vector editor.

Elements module for example, is represented by data structures, which can be divided into shapes and parts, see Figure 2.

Figure 3 shows the GUI of the editor. It consists of three parts, graphical editor, schematic parts and schematic editor. It is possible to draw basic vector objects (line, rectangle, polygon, Bezier curve, ellipse, etc.) in basic graphical editor. On this basis elements the editor was created where the schematic parts can be formed – to define shape, number of connecting pins and their location, part properties, ... The final part of the editor is the schematic editor, which consists of a menu, several toolbars and drawing pane used for drawing. All basic parts are defined and prepared in properties toolbar. The graphical elements can be used together with defined parts to create the circuit scheme in the main (schematic) editor. It is possible to choose one of the required parts and use it for circuit diagram creation in the schematic editor. After placing of the selected part a dialog appears for value or model definitions. The application is connected with an interactive catalogue, where the models are defined. After wiring the scheme the netlist is created and transmitted to the web server for analysis.

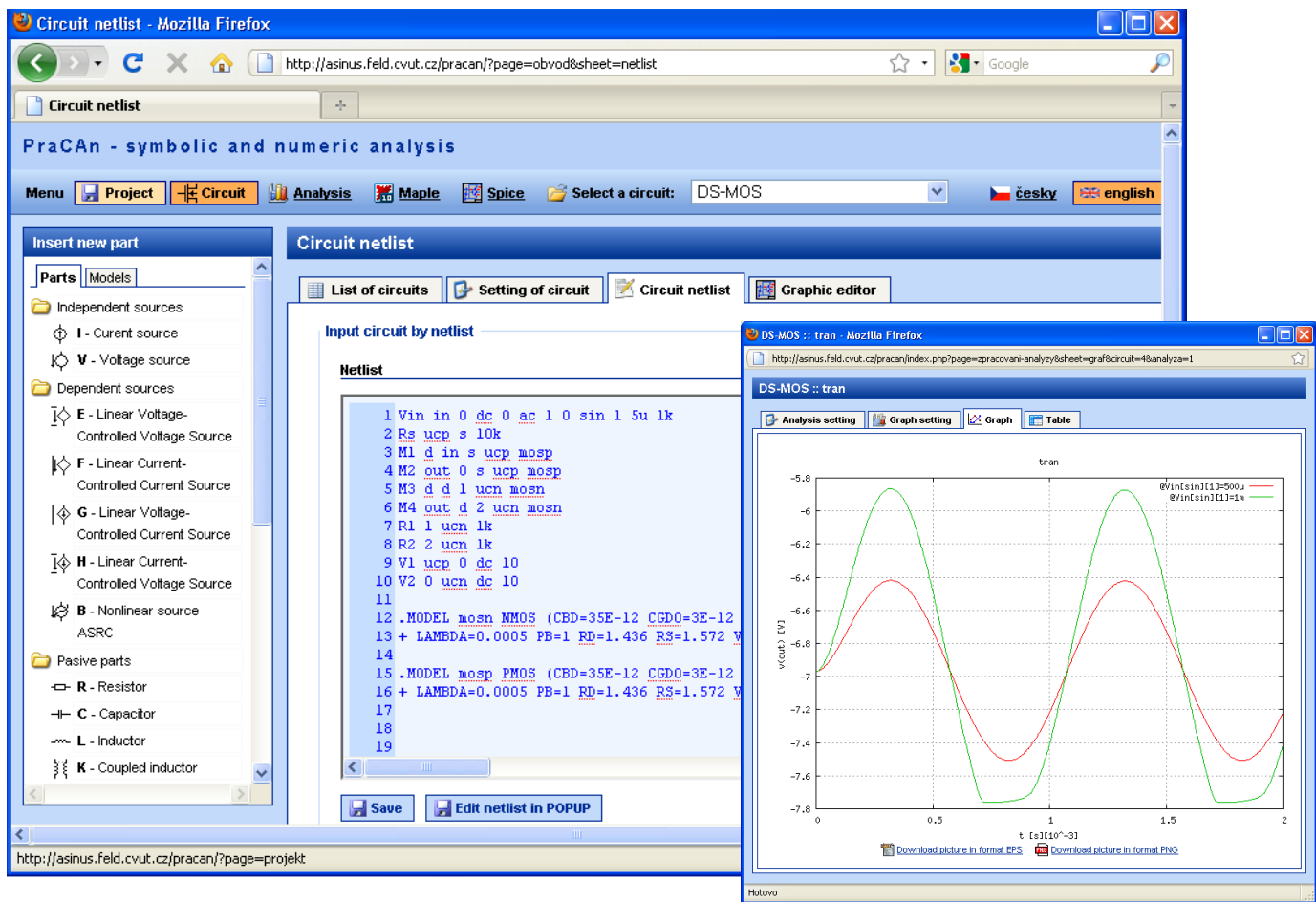


Fig. 4. Application page with inserted netlist of a circuit and window with the time response plot.

Besides the circuit netlist the scheme can be saved in a native format of the editor or exported into graphical format SVG or EPS.

User starts the editor as an applet included in the web application. It is necessary that the Java runtime environment has been installed on the client computer. The second way of starting the editor is by calling a special application installed on the user computer.

Connecting of schematic editor as an applet with www pages of application is made according [28], [29].

The applet is included at the level of class `cz.cvut.fel.schematicEditor.launcher.Applet` in Java. This class implements methods `void init()`, `void start()` and `void stop()` for applet initialization, start and stop. The methods come from `JApplet` class: `String getSession()` for getting of session from editor, `void setSession(String session)` for restoring of saved session and `String getNetlist()` for getting of netlist.

B. Examples of Circuits Analysis

The following figures are screen shots taken during the application running circuit analysis. As examples three circuit

were chosen – elementary twin-T-network with parametric part values (Figure 7), a third-order elliptic SC low pass ladder realized by the technique of switched capacitors filter [27], see Figure 12 and current mode circuit as an implementation of a simple serial resonant circuit (see Fig 15) by the technique of the switched transconductances, which is analyzed by PraCan package in Maple program and difference amplifier with active load in CMOS technology, which is analyzed by SpiceOpus.

Figure 4 shows one of the main pages of the application with imported netlist of circuit form Figure 3. The overlapping window shows the time response of the circuit for two levels of excitation as a result of numeric analysis powered by SpiceOpus. The analyses are defined on the main page. The user can define number of analyses, which are displayed into the separated windows. Figure 5 shows a window for basic setting of AC analysis. The result is displayed in “Graph” bookmark, see Figure 6. It is not classical example of frequency response, where arbitrary variable (x-axis) is frequency. It is a result of parametric analysis of AC analysis in one frequency (1 kHz, see Figure 5), where module of output voltage (gain) is calculated versus resistor R2 value. Huge possibilities of parametric analysis are demonstrated. Spectral analysis including total harmonic distortion can be calculated similarly as a special

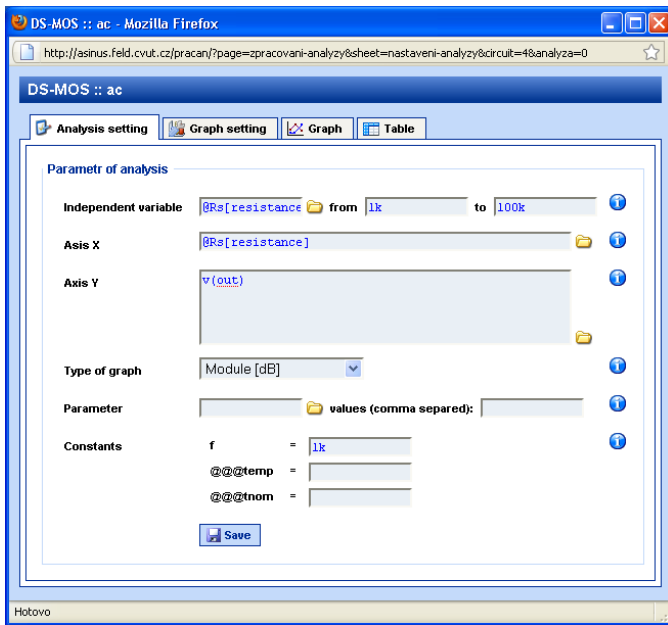


Fig. 5. Definition and settings of AC analysis – frequency response.

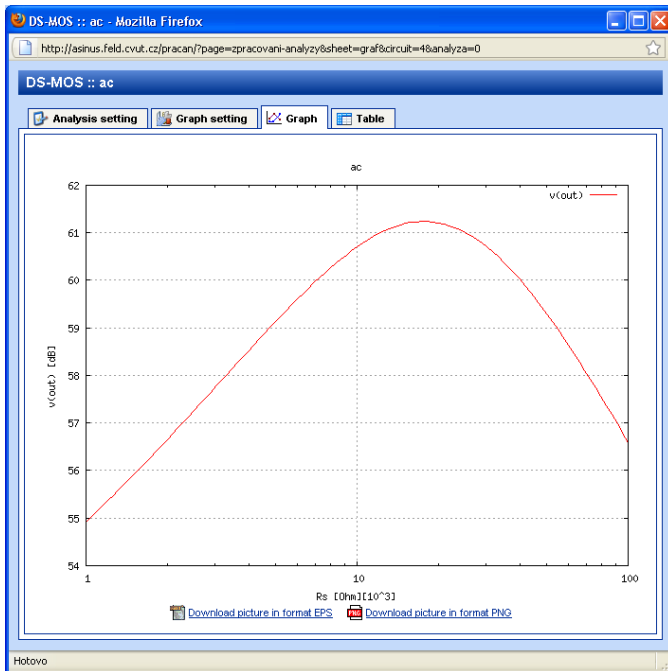


Fig. 6. Result of parametric AC analyses – modulus of output voltage vs. resistor R2 value.

case of transient analysis. A special page exist for "quick" summary analysis, where operational point can be calculated and displayed together with main circuit parameters (ac gain in one specified frequency or range of frequencies and time response also in one specified time or a time range).

Next examples demonstrate possibilities of semisymbolic and symbolic analyses based on PraCAN package powered by Maple program. Elementary circuit – twin-T-network with parametric part values form Figure 7 is analyzed first.

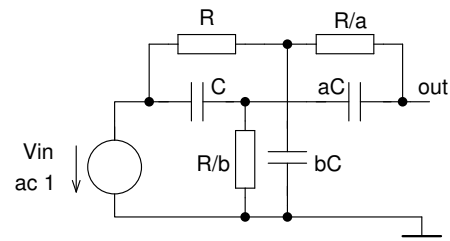


Fig. 7. Twin-T-network.

Figure 8 shows a symbolic frequency analysis of the circuit from figure 7, where transfer function is calculated. The result is displayed in basic text form (for clipboard copying) and also in graphic form (using \LaTeX). Next Figure 9 shows the benefit of Maple program implementation – all results can be mathematically treated in bookmark "Maple calculation". Resonant frequency (ω_0), transfer function in resonance (P_0), slope of phase characteristic in resonance and its evaluation for given parameter values are calculated in this case. Maple commands and all circuit variables (voltages and currents) can be used for direct calculation in Maple. Syntax of these commands is necessary to know of course. Circuit variables can be easily chosen in the right part of the window (see Figure 9).

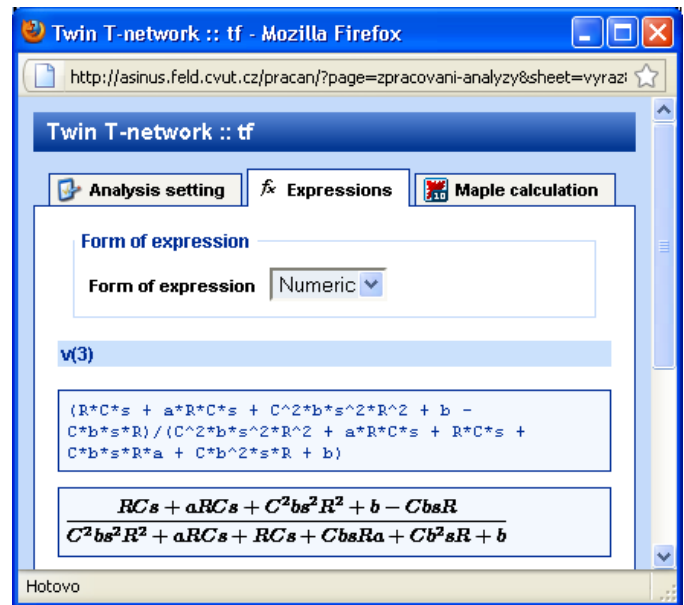


Fig. 8. Result of symbolic analyses of circuit from Figure 7.

The following two figures show frequency analysis of the circuit. The setting of the analysis (Figure 10) is nearly the same as in the previous case. Only the number of circuit constants is higher. In addition one constant (b) is chosen as a parameter with number of values (1, 1.9 and 2.1) for parametric analysis. The result of frequency response (modulus together with phase) is shown in Figure 11.

The last two examples demonstrate possibilities of discrete-time circuit analyses. The first circuit presents a third-order

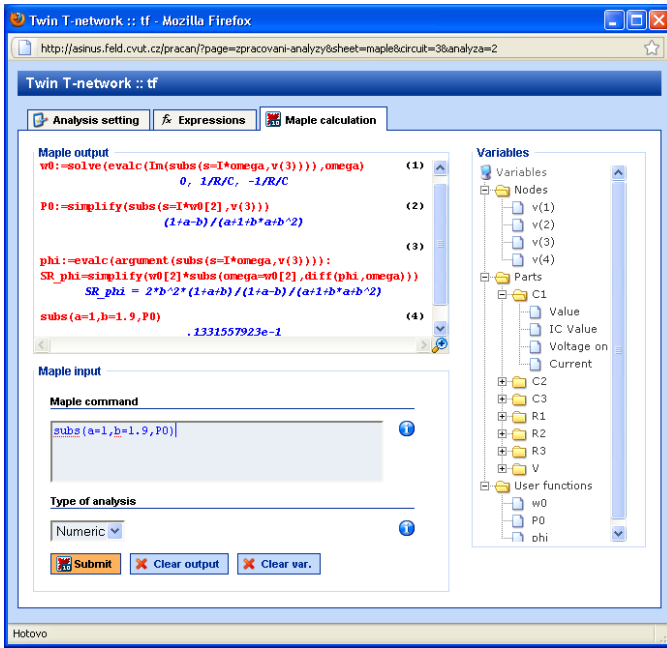


Fig. 9. Page for direct computation in Maple.

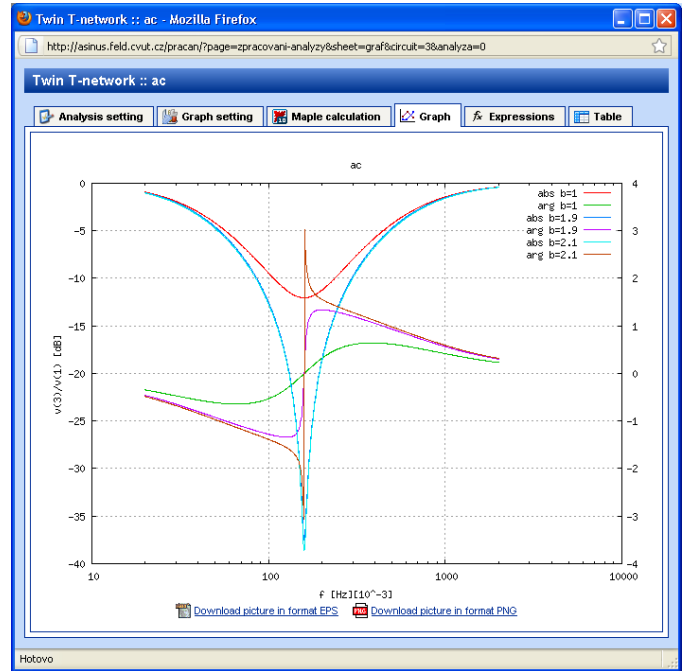


Fig. 11. Parametric frequency response (modulus and phase characteristic).

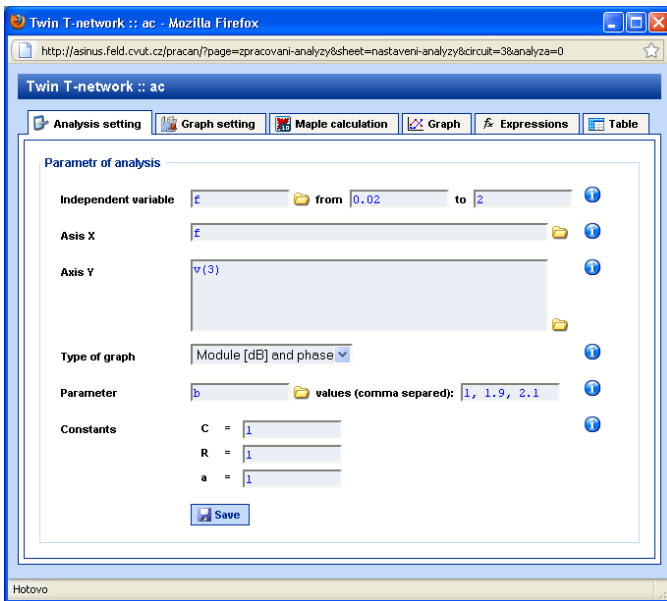


Fig. 10. Setting of frequency analysis of the circuit from Figure 7.

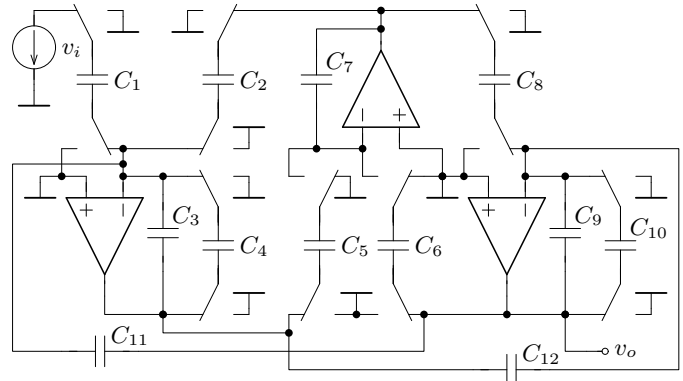


Fig. 12. A third-order elliptic SC lowpass filter.

elliptic lowpass filter realized by switched capacitor technique (Figure 12) and the second one is a resonant circuit realized by the technique of switched currents, in this example the so-called technique of switched transconductances (Figure 15). The results of analysis of circuit from Figure 12 is displayed in the following figures. Figure 13 represents the result of parametric AC analysis for two values of switching frequency f_c . This way the filter can be tuned as is seen from the figure. The overlapping window shows the poles and zeros of computed filter transfer function in Z -domain. Figure 14 shows results of transient analysis of this SC filter excited by

sinusoidal signal. One can see that input signal leaks directly to output signal because Sample&and Hold circuit is not used in the filter input and direct signal way exists from input to output.

A current mode switched circuit is analyzed in the last example. It is an implementation of a simple serial resonant circuit (see Fig 15) by switched transconductances, as was noted above. The symbolic analysis is provided first, see Figure 16. Using Maple direct computation interface, the input "impedance" in node 2 in the first phase is calculated as is shown in Figure 17. Then transformation of the impedance from z to s domain is done by means of inverse backward Euler transformation (BD). The input impedance corresponds to impedance of the RLC series resonant circuit, which is evident from expanded result.

The application enables analysis of number of circuits and their management. Different analysis can be created for each

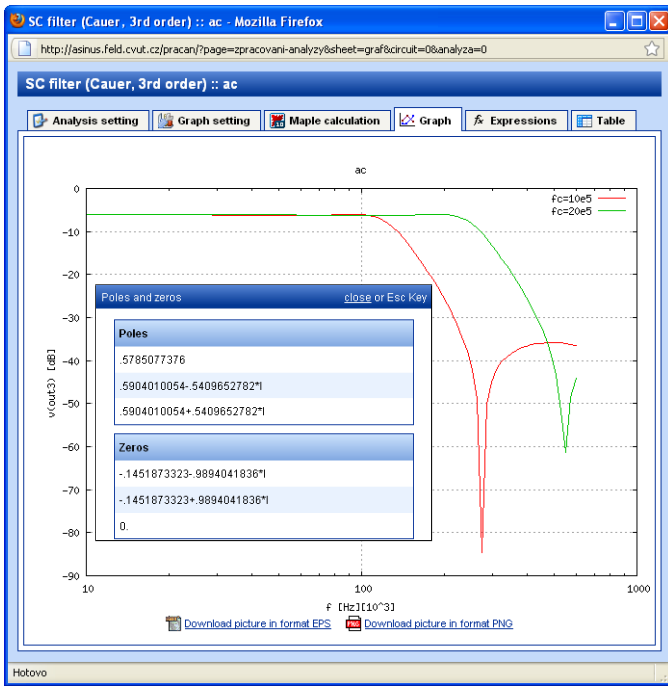


Fig. 13. Frequency response of SC filter and window with pole and zero calculation of transfer function in Z -domain.

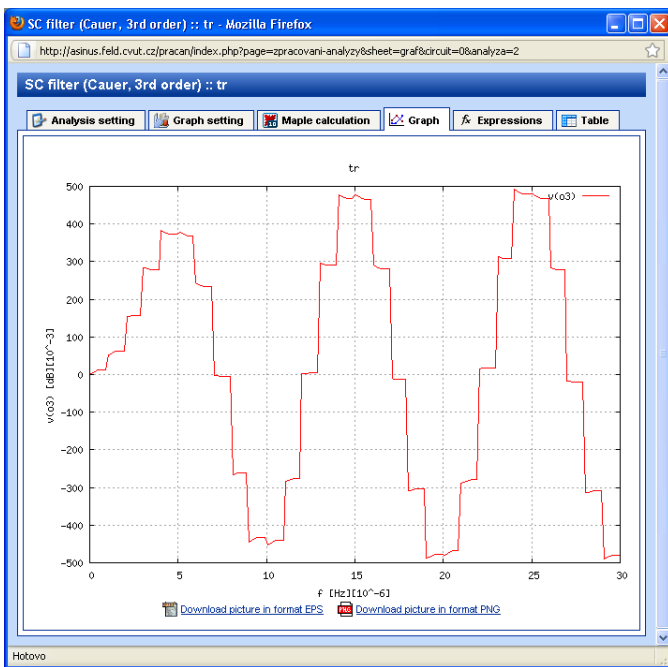


Fig. 14. Result of transient analyses of SC filter.

circuit. The circuit (scheme, netlist, setting, ...), analysis definitions and results can be saved to the project, see Figure 18. User can save complete work and present it this way.

IV. CONCLUSION

The web-based application has been created to enable analysis of electric and electronic circuits for a wide range of

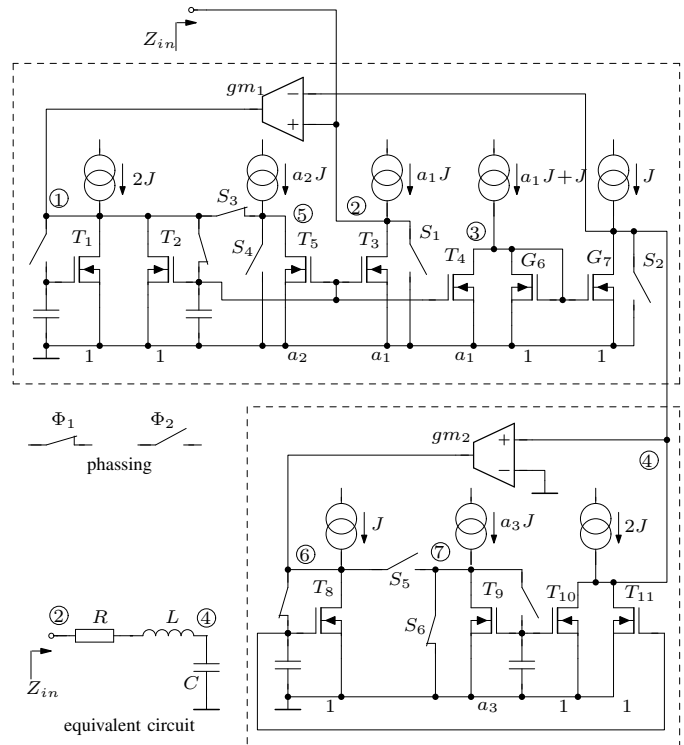


Fig. 15. Realization of RLC circuit by switched transconductances.

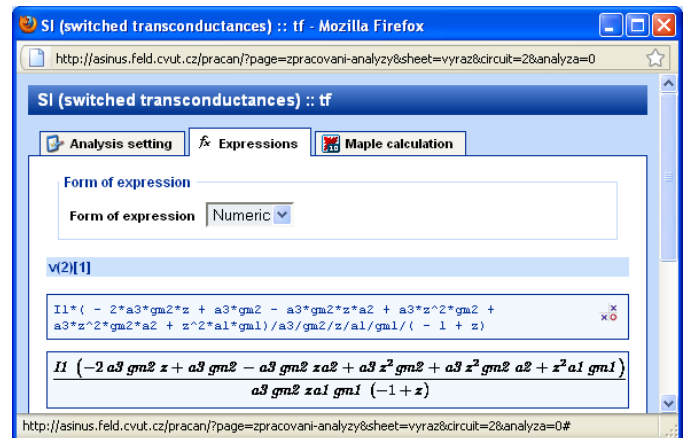


Fig. 16. Result of symbolic AC analysis of the circuit from Figure 15 – voltage in node 2 in the first phase.

users. Designed system combines technologies as rich as client technology, PHP, Java, and circuit simulation, and provides the user with vivid interface, convenient operation and powerful simulation capability. The application uses facilities of SpiceOpus program for numeric analysis and PraCan package in Maple program for symbolic and semisymbolic analysis of continuous-time as well as periodically switched linear circuits. No known web-based system offers such range of capabilities.

Operating of the interface is very easy. Circuit description can be entered using graphical interface of schematic editor. All pages of application are supplemented by interactive help.

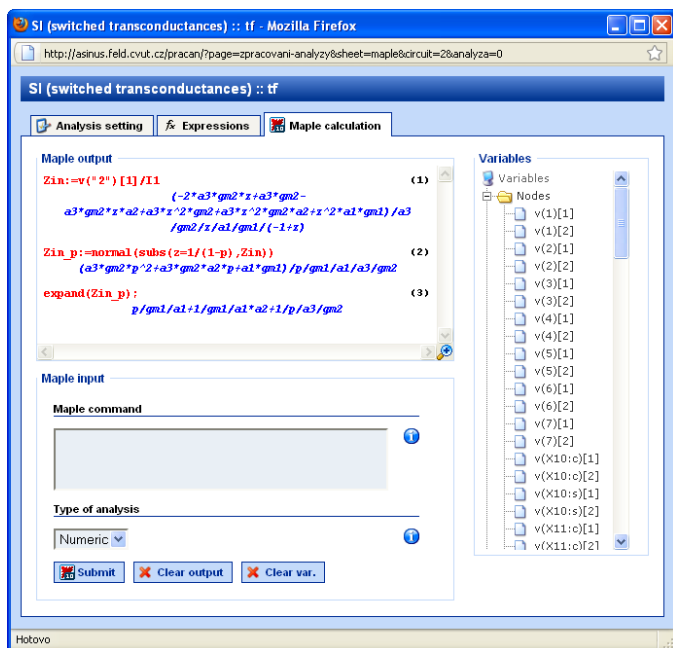


Fig. 17. Page with direct Maple interface for computation and result treatment.

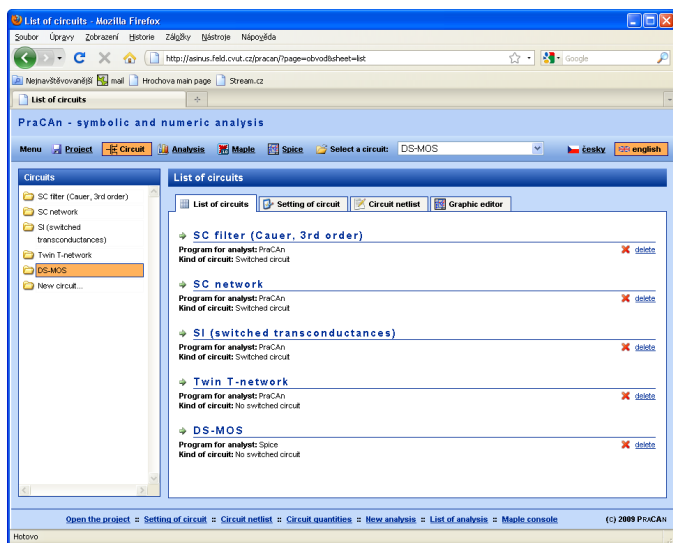


Fig. 18. Page for circuit management.

User can use the interface without any manual or study of syntax. The analysis can be very simply created and modified. It is possible to create number of different analysis of one or more circuits in one project. Results of the analyses can be displayed together and they are simultaneously recompiled (redisplayed) if the netlist of the corresponding circuit is changed. It represents with the symbolic analysis very good tool for engineers and students dealing with electronics.

The application was created especially for teaching support on the Faculty of Electrical Engineering, Czech Technical University (CTU) in Prague. The easy-to-use web based interface will aid analysis of electric and electronic circuits

without any program installation and without learning of any command syntax. Students make significant learning gain as a result of using this system. Their interest for electric circuits rises thanks to operation capability and potential of the application. Learning process can be well facilitated if tools are widely available, not just in the dedicated laboratories. Web-based simulation environment, combining distance education, group training and real-time interaction, can serve as a good approach.

The analyses are powered by Maple program whose utilization is restricted by license. This is the reason why the application is not free for all users. Nowadays the system is open from CTU domain; nevertheless it can be open for others who meet the license requirements.

ACKNOWLEDGMENT

The work has been supported by the grant of Ministry of Education, Youth and Sports No. 2388F1d, grant GA102/07/1186 of the Grant Agency of the Czech Republic and by the research program No. MSM6840770014 of the CTU in Prague.

Maple is trademark of Waterloo Maple Inc.; Maplesoft is a division of Waterloo Maple Inc. All other trademarks are property of their respective owners.

PSpice is registered trademark of Cadence Design Systems, Inc.

REFERENCES

- [1] J. Hospodka and J. Bičák, *Web-based Application for Electric Circuit Analysis*, Proceedings of The Fourth International Multi-Conference on Computing in the Global Information Technology [CD-ROM]. Los Alamitos: IEEE Computer Society, 2009, p. 157-160, ISBN 978-0-7695-3751-1.
- [2] *SPICE – general-purpose circuit simulation*, URL: <http://bwrc.eecs.berkeley.edu/Classes/IcBook/SPICE/>, 2009.
- [3] Cadence Design Systems, Inc., *Spice – analog and mixed-signal circuit simulator*, URL: <http://www.cadence.com/>, 2009.
- [4] Spectrum Software, *Micro-Cap – schematic editor and mixed analog/digital SPICE circuit simulator*, URL: <http://www.spectrum-soft.com/>, 2009.
- [5] M. Smith, *WinSpice User's Manual*, <http://www.winspice.com>, 2007.
- [6] Maplesoft, a division of Waterloo Maple Inc., *Maple – essential technical computing software*, URL: <http://www.maplesoft.com/>, 2009.
- [7] Y. Ouyang, Y. Dong, M. Zhu, Y. Huang, S. Mao, and Y. Mao, *ECVlab: A web-based virtual laboratory system for electronic circuit simulation*, ICCS – International Conference on Computational Science 2005, Atlanta, Ga, USA, pp.1027-34, ISBN-10: 3540260323.
- [8] L. Weyten, P. Rombouts, and J. De Maeyer, *Web-Based Trainer for Electrical Circuit Analysis*, IEEE Transactions on Education, 2009 Volume: 52, Issue: 1, pp: 185-189, ISSN: 0018-9359.
- [9] SoftIntegration, Inc., *Web-Based Control System Design and Analysis*, URL: <http://www.softintegration.com/webservices/control/>, 2009.
- [10] SoftIntegration, Inc., *Ch CGI toolkit for CGI programming*, URL: <http://www.softintegration.com/products/toolkit/cgi/>, 2009.
- [11] B. M. Wilamowski, A. Malinowski, and J. Regnier, *Internet as a New Graphical User Interface for the SPICE Circuit Simulator*, IEEE transaction on industrial electronics, Vol. 48, No. 6, 0278-0046/01\$ 10.00, December 2001.
- [12] B. M. Wilamowski, A. Malinowski, and J. Regnier, *SPICE based Circuit Analysis using Web Pages*, ASEE 2000 Annual Conference, St. Louis, MO, June 18 to 2, 2000, CD-ROM session 2520.
- [13] J. Regnier and B. M. Wilamowski, *SPICE simulation and analysis through Internet and Intranet networks*, IEEE Circuits and Devices Magazine, Vol. 14, Issue 3, pp 9-12, ISSN 8755-3996, May 1998.

- [14] J.A. Asumadu, R. Tanner, J. Fitzmaurice, M. Kelly, H. Ogunleye, J. Belter, and Song Chin Koh, *A Web-based hands-on real-time electrical and electronics remote wiring and measurement laboratory (RwmLAB) instrument*, Proceedings of the 20th IEEE IMTC – Instrumentation and Measurement Technology Conference, Volume 2, 2003, pp: 1032-1035.
- [15] J. Bičák, J. Hospodka, J. Vrbata, and P. Martinek, *Design of Electric Filters in Maple and through WWW Interface*, Proceedings of ICECS – The 8th IEEE International Conference on Circuits and Systems, Vol. 3, pp. 1619–1622, ISBN: 0-7803-7058-9, Malta 2001.
- [16] J. Hospodka and J. Bičák: *Syntfil - Synthesis of Electric Filters in Maple*, MSW 2004 [CD-ROM]. Waterloo, ON: Maplesoft, a division of Waterloo Maple Inc., 2004.
- [17] J. Hospodka and O. Kobliha, *Internet Pages as an Interface between a User and Computing Program*, Digital Communications'03, EDIS Žilina University Publisher, Žilina, 2003, pp. 45-48.
- [18] SpiceOpus – SPICE with integrated OPTimization Utilities, URL: <http://www.fe.uni-lj.si/spice/> 28.3.2009.
- [19] J. Bicak and J. Hospodka, *PraCAN - Maple Package for Symbolic Circuit Analysis*, Digital Technologies 2008, EDIS Žilina University Publisher, Žilina, 2008, ISBN 978-80-8070-953-2.
- [20] J. Bicak, J. Hospodka, and P. Martinek, *Symbolic Analysis of SC Circuits in Maple*, ECCTD '99, pp. 1079-1082, Torino: Politecnico di Torino.
- [21] J. Bicak, J. Hospodka, and P. Martinek, *Analysis of SI Circuits in MAPLE Program*, Proceedings of the 15th European Conference on Circuit Theory and Design ECCTD'01, Helsinki: Helsinki University of Technology, 2001, vol. 3, pp. 121-124, ISBN 951-22-5572-3.
- [22] J. Bicak and J. Hospodka, *PraSCAN – Maple Package for Analysis of Real Periodically Switched Circuits*, Maple Conference 2005 Proceedings. Waterloo, ON: Maplesoft, a division of Waterloo Maple Inc., 2005, vol. 1, s. 8-18. ISBN 1-894511-85-9.
- [23] J. Bicak and J. Hospodka, *Symbolic Analysis of Periodically Switched Linear Circuits*, SMACD'06 – Proceedings of the IX. International Workshop on Symbolic Methods and Applications to Circuit Design, Firenze, Universita degli Studi, 2006, ISBN 88-8453-509-3.
- [24] F. Yuan and A. Opal, *Computer Methods for Switched Circuits*, IEEE Transactions on CAS I, Vol. 50, pp. 1013-1024, Aug. 2003.
- [25] D. Biolek, *Modeling of Periodically Switched Networks by Mixed s-z Description*, IEEE Transactions on CAS I, Vol. 44, pp. 750-758, 1997.
- [26] J. Vlach and K. Singhal, *Computer Methods for Circuit Analysis and Design*, Van Nostrand Reinhold Company Inc., New York 1994, 2nd Edition, ISBN 0-13-879818-4.
- [27] P. V. Ananda Mohan, V. Ramachandran, and M. N. S. Swamy, *SWITCHED CAPACITOR FILTERS Theory, Analysis and Design*, Prentice Hall 1995, ISBN 0-13-879818-4.
- [28] E. A. Musayev, *How to connect Java applet, Javascript and HTML form*, URL: <http://www.galiel.net/el/howto/jvjs.html>, 2009
- [29] *Call Javascript from a Java applet*, URL: <http://www.rgagnon.com/javadetails/java-0172.html>, 2009