Multitool Online Assisted Design of Communication Circuits and Systems

Galia Marinova[1]

Abstract **– The paper deals with a Multitool environment for computer-aided design assisted by a portal for online calculators. The structure of the portal is presented as well as design examples (projects of loudspeakers, LED circuits and PCBs, etc.) illustrating join application of Computer-aided design (CAD) tools and online calculators.**

Keywords **– CAD tools, Online calculators, Communication circuits and systems.**

I. INTRODUCTION

Recently there is a boom of online calculators for different engineering tasks. The paper describes an extension of the Multitool environment for computer-aided design, described in [1], with a portal for online calculators, applicable in communication circuits and systems.

The Multitool environment from [1] integrates CAD tools as: MATLAB, ORCAD/PSpice/Layout, WARP 6.2, ISE Webpack, Quartus II, Microwind, FilterCAD, PAC Designer, MMICAD, etc., which cover different aspects of realistic projects in communications (system level, analog, digital, analog-digital, programmable, microwave, microelectronic design and PCB design). The design process can be assisted by specialized online calculators which add more features to the Multitool environment. The structure of the portal for online calculators is presented further. The approach of joint application of online tools and downloaded tools for solving communication circuit design tasks is illustrated with four examples in the area of crossover design, PCB design, SMPS design and LED array and driver design.

II. PORTAL FOR ONLINE CALCULATORS FOR COMMUNICATION CIRCUITS AND SYSTEM DESIGN

The portal is in experimental phase of development and it will be connected to E-content for PSpice and other CAD tools for communications from [2]. It organizes different groups of online calculators:

- Online design tool PowerEsim for switch-mode power supply design [3];
- Online calculators for resistors and capacitors [4, 5, etc.];
- Online calculators for PCBs and transmission lines [6, etc.];
- Online matching network tools – Smith Chart

[1]Galia Marinova is with the Faculty of Telecommunications at Technical University of Sofia, 8 Kl. Ohridski Blvd, Sofia 1000, Bulgaria, E-mail: gim@tu-sofia.bg.

diagram tools [7, etc.];
- Online calculators and wizards for LED array design [8, etc.];
- Online analog/digital/crystal filters design tools [9, etc.];
- Online 555 timer calculators [10, etc.];
- Online design tools for loudspeakers and microphones (mono and stereo) [11, etc.];
- Online antenna calculators and design tools [12, etc.];
- Online SCA based open source software defined radio – OSSIE [13, etc.].

The online calculator groups are connected to a set of online calculators. Each calculator is described shortly and a link is provided to its original web address (see Fig.1).Instructions are given for joint online and downloaded tools tasks solutions for communication circuits.

III. EXAMPLES OF ONLINE CALCULATOR ASSISTED DESIGN OF COMMUNICATION CIRCUITS IN THE MULTITOOL ENVIRONMENT.

Four examples illustrating the joint approach are presented in details.

A. Example 1. Crossover design through joint application of loudspeakers' online calculator and PSpice

Figure 2 illustrates Two way crossover design (with 6th order Linkwitz-Riley LC filters) using the online calculator [11], modeling the loudspeakers' drivers with Thile small equivalent circuit as indicated in [14] and simulating the whole design in PSpice for specification verification. The output voltages of the tweeter and the woofer are plotted separately and then the resulting sum is obtained. All three voltages are plotted in dB.

B. Example 2. PCB design through joint application of ORCAD/Layout, online calculator for transmission line impedance calculation and PSpice

Figure 3 illustrates the characteristic impedance calculation with the online tool [6] of a PCB trace, taking its geometric parameters from ORCAD/LAYOUT tool. Then the PCB trace is modeled as a transmission line and simulated in PSpice as proposed in [15]. The rise time of the input pulse is 5ns. The parasitic picks in the output voltage are attenuated with a serial resistor R4 and then a new PSpice simulation illustrates the elimination of the parasitic picks.

Fig.1. Portal for online calculators for communication circuit and system design

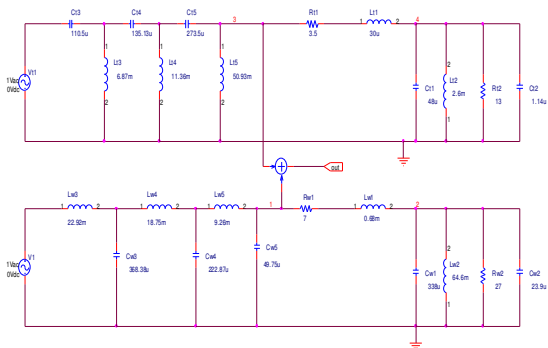Fig. 2a. Crossover circuit generated in Loadspeakers' online calculator

Fig. 2c. PSpice simulation of the two-way crossover circuit: Tweeter output voltage in dB; Woofer output voltage in dB; Output voltage in dB for the two-way crossover circuit

Fig. 2b. PSpice model of the circuit from Fig.2a

Figure 2. Two way crossover design

Fig. 3a. Characteristic impedance calculation of a PCB trace modeled as transmission line
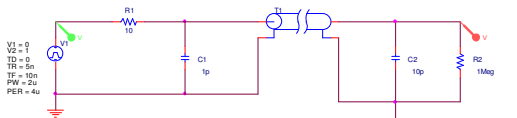
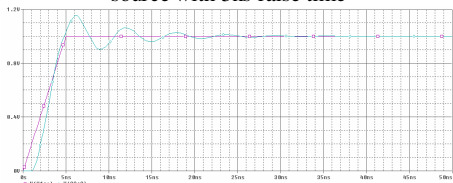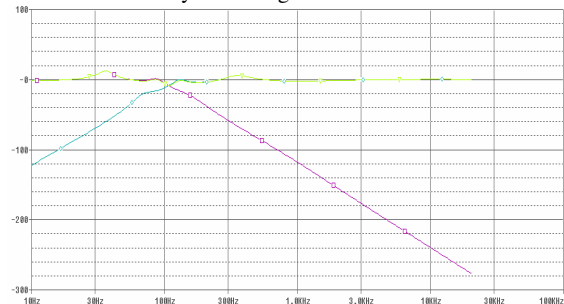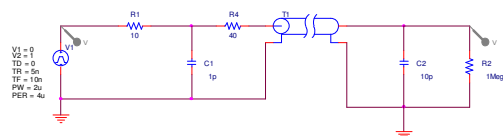Fig. 3b. PSpice model with transmission line and pulse voltage source with 5ns raise time

Fig. 3d. Circuit from Fig.3b with R4 for pick attenuation

Fig. 3c. PSpice simulation of the circuit from Fig.3b.: input pulse and output voltage with parasitic picks
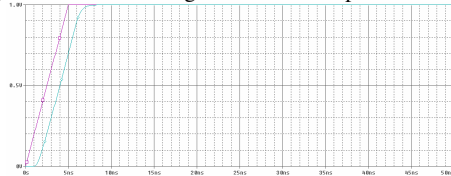

Fig. 3e. PSpice simulation of the circuit from Fig.3d.: Input pulse and Output voltage without parasitic picks

Fig. 3. Transmission line effect of a PCB trace for high speed

C. Example 3. Switch mode power supply design through joint application of PowerEsim and PSpice

Figure 4 illustrates the generation of LLC (Series Parallel) converter block in PowerEsim [3] and the possibility for simulation in PSpice of a circuit from this block for verification. A circuit from the LLC PWM (Pulse width modulation) block is simulated in frequency area.
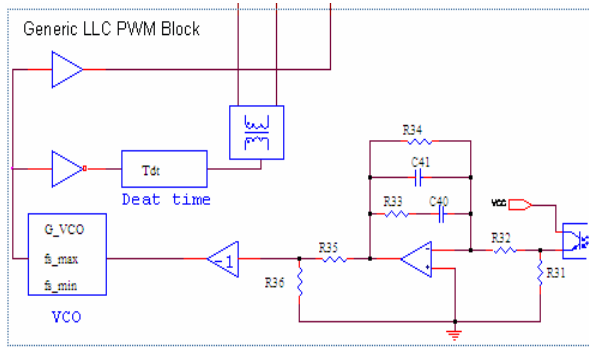

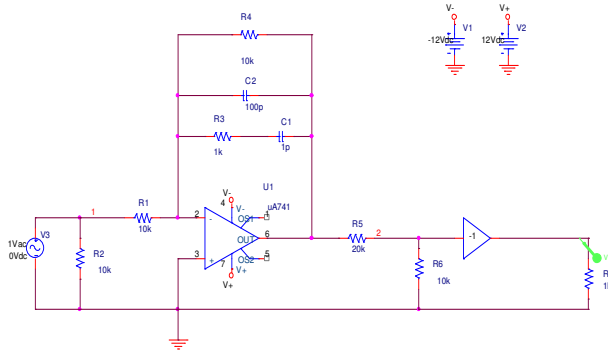Fig. 4a.LLC PWM block synthesized in PowerEsim

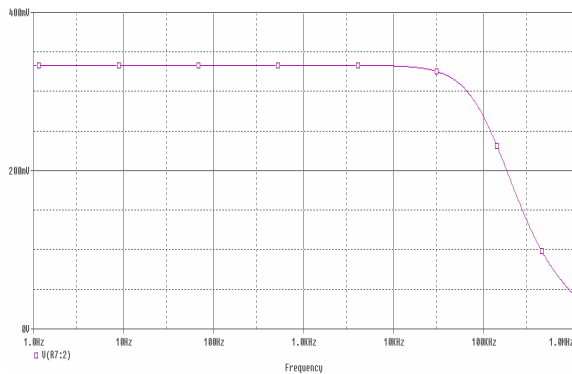Fig. 4b. Electrical circuit from the LLC PWM block in ORCAD/Capture

Figure 4c. Simulation in PSpice of the circuit from Fig.4b. in frequency area

Fig. 4. PWM block of LLC circuit designed in PowerEsim and PSpice simulation of a circuit in the block

D. Example 4. LED array and driver circuit design through joint application of online calculator and WARP 6.2

Figure 5 illustrates the LED array design in the online calculator [8] and the design of a Pseudo random number generator (PRNG) in VHDL and its simulation in the tool WARP 6.2 (CYPRESS). The PRNG is designed using shift register and XOR circuits.

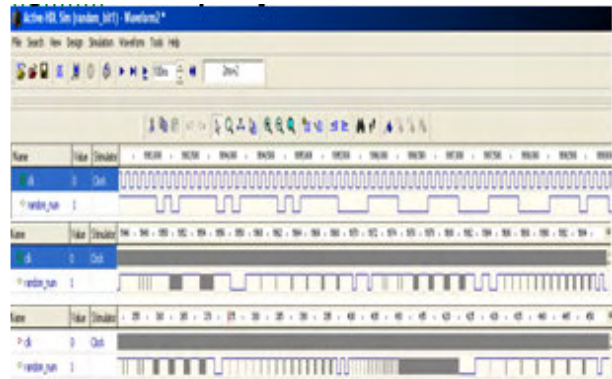Figure 5a. LED array design in the LED online calculator

Figure 5b. Simulation waveforms of a PRNG designed in VHDL with the tool WARP 6.2(CYPRESS).

Figure 5. LED array design in online calculator and design of a driving circuit in WARP 6.2.

IV. Conclusion

The paper is an attempt to make useful the large number of online calculators available nowadays to facilitate the design of communication circuits and systems adding them to a Multitool design environment through a new portal in the web.

Four different types of online calculators for loudspeakers' design, for PCB trace characteristic impedance calculators, SMPS tool design and LED array design tool are combined with ORCAD/Capture/PSpice, Layout and WARP 6.2. tools.

591

The PSpice simulator is applied mainly as a verification tool. ORCAD/Capture tool is applied for equivalent circuit's edition. ORCAD/Layout is applied as a source of data in the PCB trace Characteristic impedance calculator and WARP 6.2 is applied for VHDL description and simulation of a digital driver circuit.

The four examples, illustrating the joint approach, confirm the advantages of the approach and promise plenty of applications in students' education contributing to its accessibility and attractiveness. The portal and the applications proposed can be applied in the course of Computer-aided design for the Bachelor and/or Master degree students and they can serve for a diversity of individual projects.

REFERENCES

[1] G. Marinova, Multidisciplinary approach for teaching Computer-Aided Design incommunications using multiple software tools, *Advanced Aspects of TheoreticalElectrical Engineering Sozopol '2010,* 19.09.10 – 22.09.10, Sozopol, Bulgaria, Plenarylectures, pp. 34 -42

[2] E-content for PSpice and other CAD tools for communications http://www.pueron.org/pueron/E_CADSystems/E_CADSystems.htm

[3] Free Switch Mode Power Supply Circuit (SMPS) Design Software& Transformer Calculation/Simulation tool http://www.poweresim.com/

[4] Capacitor color code calculator http://www.csgnetwork.com/capcccalc.html

[5] Resistor color code calculator http://www.csgnetwork.com/resistcolcalc.html

[6] Microstrip transmission line Characteristic Impedance Calculator http://chemandy.com/calculators/microstrip_transmission_line_calculator_Hartley27.htm

[7] Smith Chart tutorial http://www.fourier-series.com/rf-concepts/smithchart.html

[8] Current Limiting Resistor Calculator for LEDs http://ledcalc.com/

[9] Homebrew Crystal Filter Design http://www.changpuak.ch/electronics/calc_17.php

[10]555 Calculator http://freespace.virgin.net/matt.waite/resource/handy/pinouts/555/index.htm

[11] 2 Way Crossover Designer/ Calculator http://www.diyaudioandvideo.com/Calculator/XOver/

[12] LPDA: Logarithmic Periodic Dipole Antenna Calculator http://www.changpuak.ch/electronics/lpda.php

[13] OSSIE - SCA based open source software defined radio http://ossie.wireless.vt.edu/

[14] How to simulate speaker/crossover using ORCAD http://www.shine7.com/audio/orcad.htm

[15] Johnson H., High Speed Digital Design, Prentice Hall, 1993