

E-content for Computer-Aided Design of Communication Circuits with PSpice Simulator

Galia I. Marinova¹

Abstract – The paper presents the e-content of a Knowledge base with algorithms for analyzing different classes of communication circuits in PSpice simulator. Algorithms are developed for 22 classes of circuits and there are more than 200 circuits in the Knowledge base.

Keywords – PSpice, Communication circuits, Algorithms for analysis, Knowledge base, E-content.

PSpice simulator is one of the tools used to teach Computer-aided design – a third year bachelor degree course in the Telecommunications Faculty of Technical University – Sofia. The goal of the course is to give students the abilities to design communication circuits and systems integrating knowledge from several previous and parallel courses:

- Circuit theory methods from the course of Electrotechnics;
- Analog and digital filter design from the course of Communication networks;
- Analog and digital circuits design from the courses of the same name;
- Radiocommunications and Radiocommunication systems courses.

I. INTRODUCTION

The PSpice simulator is a basic tool in university classes of Electrotechnics, Electronics and Communications. Different teaching environments integrating the PSpice software are presented in [1, 2, 6, 7, 8].

TABLE 1

Class of circuits	Number of circuits	Circuits types or examples
Filters with operational amplifiers	35	Low pass, High pass, Band pass and Stop band filters with different approximations
Passive LC filters	10	Example: Band pass filter for communication antenna system
Digital filters	10	FIR filters with one or more stages [9,10]
Equalizers	2	Examples for 4 and 10 frequencies
Filters with quartz crystals	10	Cohn crystal ladder filters with 3 or 4 stages [12]
SC filters	2	Low pass filters, Integrators [5]
Filters with transmission lines	15	Realized with different number of transmission line elements [9]
Generators	15	Square wave, triangle wave and sine wave realized with transistors, 2 or 3 operational amplifiers
Generators with quartz crystal stabilization	4	Sine wave generators
Arithmetic circuits with operational amplifiers	20	Integrator, Differentiator, Comparator, Summing and difference amplifier, Logarithmic and exponential amplifier, Square root circuit
Voltage regulator circuits	10	Linear voltage regulator circuit with BJT at the output, Voltage regulator circuit with output BJT in switch mode, Voltage regulator circuit based on direct converter principle, with output BJT in switch mode, Voltage regulator circuit with output MOS transistor in switch mode [2]
Resonance amplifier	3	Realized with transistors
Amplifiers	5	Realized with transistors and with operational amplifiers
Programmable gain amplifiers	3	Realized with switches
Circuits with timer LM555	25	PWM circuit, Schmitt trigger, Basic astable oscillator, Monostable timer [11]
Digital combinatorial circuits	10	Logical circuits
Arithmetic digital circuits	10	Adders and multipliers
Digital circuits with memory	5	Latches, Flip-flops
Radiocommunication circuits with transmission lines	10	Mixers, Modulators, Demodulators, Wilkinson power divider /adder, Directional coupler, Hybrid ring, Balun [1]
VCOs	2	Realized with OA and with timer LM555 [11]
Mixers, balance modulators	2	Realized with transistors
Frequency converters	2	Realized with transistors

¹ Galia I Marinova is with the Telecommunications faculty, Technical University – Sofia, Sofia-1612, 8, bul. Kliment Ohridski, E-mail: gim@tu-sofia.bg

The course of Computer-aided design is structured in lectures, laboratory exercises and individual or group projects. All these aspects are evaluated and then students are estimated.

In order to give students a global view, closer to real applications and the ability to solve a large number of design problems, a Knowledge base with algorithms to analyze different classes of communication circuits is developed.

II. E-CONTENT IN THE KNOWLEDGE BASE

The e-content serving the Knowledge base in the Learning environment is structured following the general methodology for circuit design with PSpice and gives the specific step-by-step algorithms for each circuit class.

The main steps in the algorithms are:

- Choice of circuits from a circuit class suitable for the desired application.
- Determination of circuit element parameters for meeting the design specification and constraints. For some classes of circuits on-line calculators are available and they can be used, for example for crystal filters [14] or timers with LM555 circuit [11].
- Creation of project, designs and schematics for the circuits in OrCAD Capture.
- Choice of the input sources – voltage or current stimulus in analog or digital form.

- Definition of the set of analyses to be performed in order to obtain the representative curves for the circuits.
- Definition of the set of curves to be visualized by the graphical postprocessor Probe.
- Definition of the set of parameters to be extracted from the curves and from the output file.
- Definition of the criteria for estimation of circuit properties and for comparison.
- Conclusions for the circuit performance, its area of application, the restrictions for its application, optimization advices and advices for multisolution synthesis.

Algorithms are developed for 22 classes of circuits and a set of circuits taken from books [3, 4] and internet sites [12, 13] is considered in each class. The Knowledge base contains more than 200 circuits. The e-content of the Knowledge base is presented in Table 1.

III. ILLUSTRATIVE EXAMPLES OF E-CONTENT IN THE KNOWLEDGE BASE

Three examples from the e-content of the Knowledge base are given on Fig. 1, 2, 3:

Fig. 1 presents a double crystal filter in radio receiver from [13] – the original circuit, the circuit with RLC models of the crystals, then AC curves and the parameter values for the resonance frequency, the low and high cut off frequencies and the frequency band.

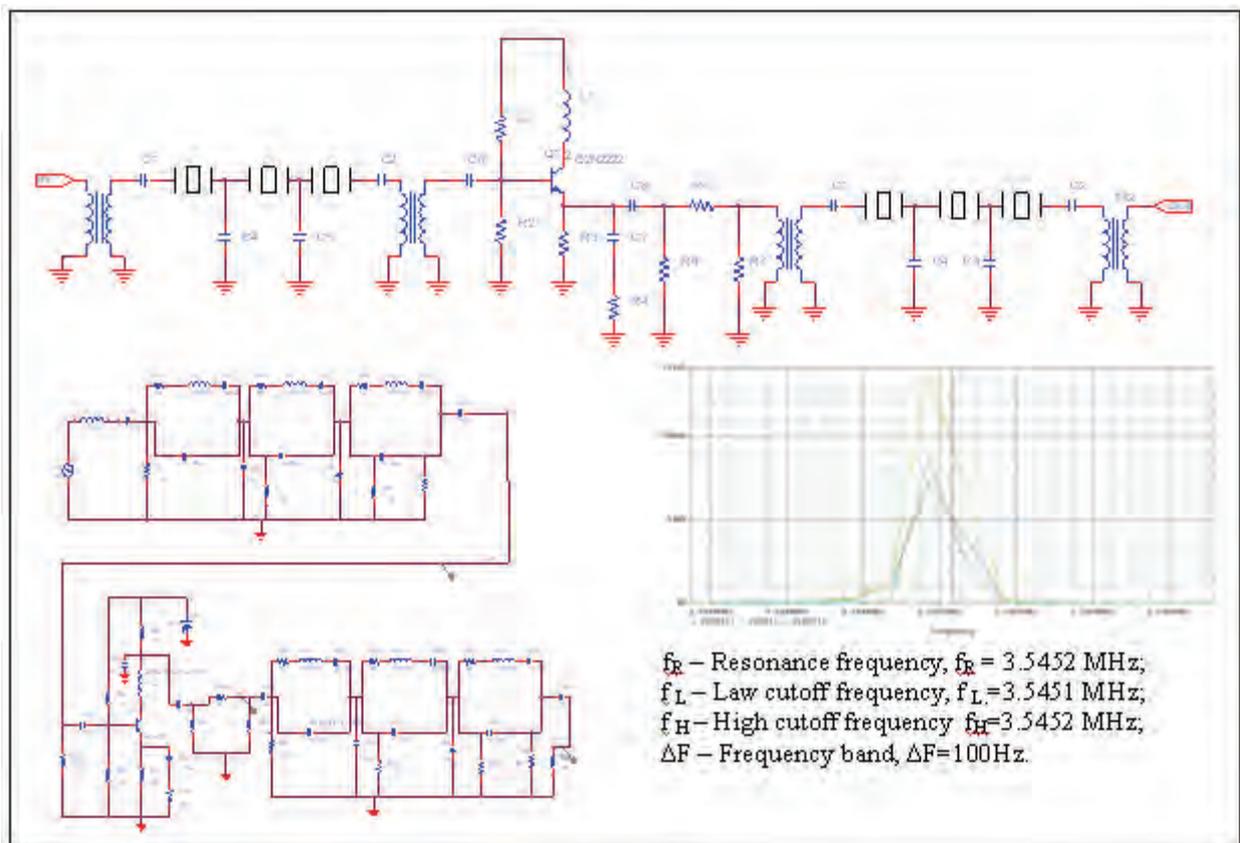


Fig. 1. Double crystal filter in radio receiver – original circuit, circuit with RLC model of the crystals, AC curves and parameters

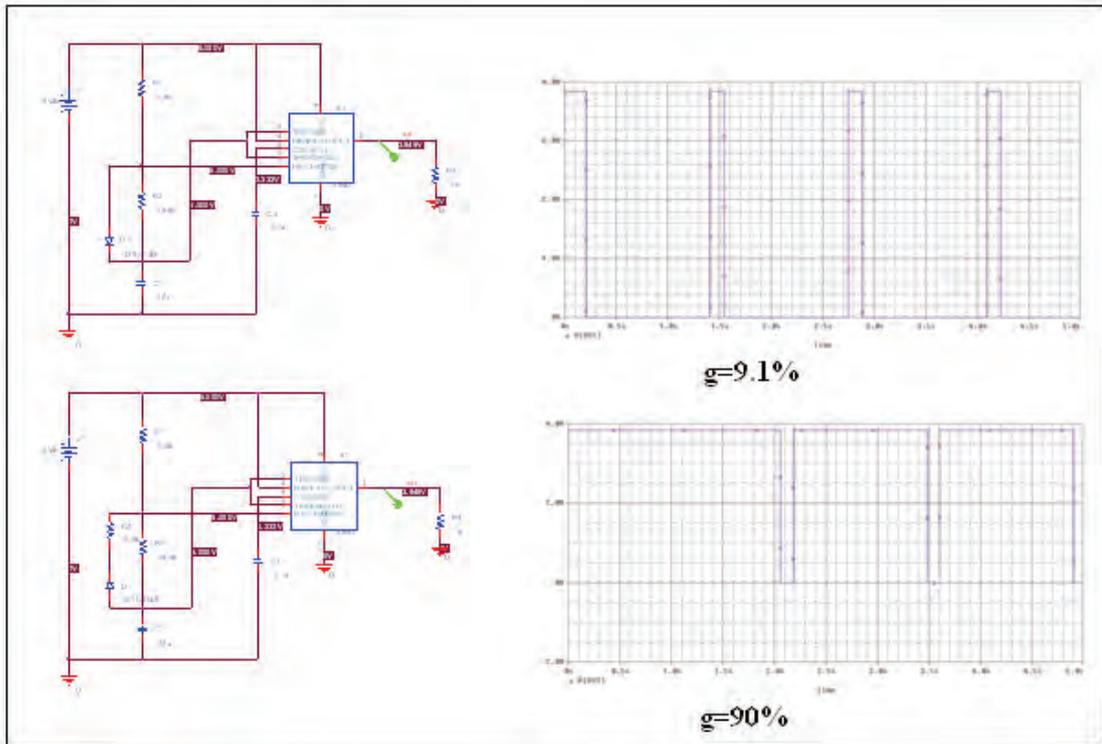


Fig. 2. Pulse generator circuits with short and large ON time. Circuits, output voltages and output duty cycle $g = T_{on}/T$, T - pulse period

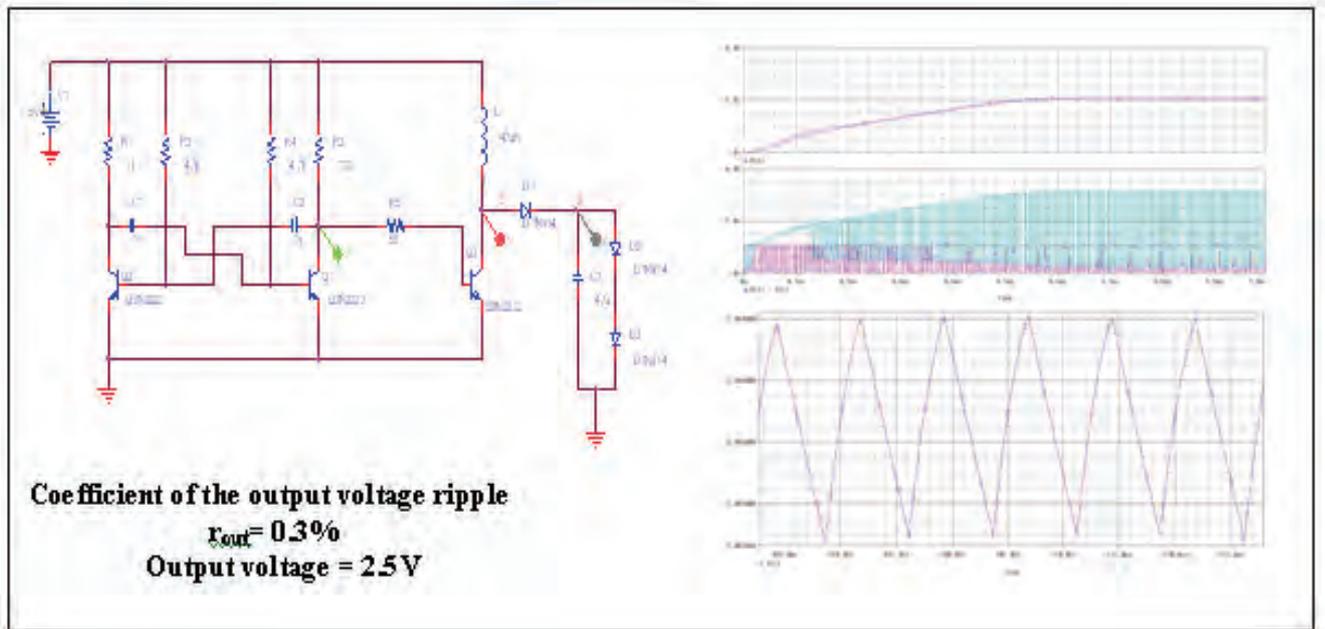


Fig. 3. Boost voltage regulator – circuit, voltages in nodes 1, 2 and 3. Circuit parameters: output voltage and coefficient of the output voltage ripple.

Fig. 2 presents pulse generator circuits with short and large ON time - circuit, output voltages in time domain and output duty cycle coefficients

$$g = T_{on}/T,$$

where T is the pulse period.

Students are expected to give proposals for the possible applications of the two circuits.

Fig. 3 presents a boost voltage regulator – circuit, voltages in nodes 1, 2 and 3 in time domain and circuit parameters - output voltage and coefficient of the output voltage ripple.

Students are expected to conclude whether the circuit meets the restriction for voltage regulator circuits:

$$r_{out} < 1\%$$

They should also check whether the boost regulator circuit can aliment a LED with 1.5V battery.

During the regular course students simulate about 10 different circuits to get familiar with the PSpice design methodology and then they work on individual or group projects chosen on a random principle from the Knowledge base.

IV. CONCLUSION

The e-content of the Knowledge base presented in the paper helps solving a great variety of design problems using PSpice simulator which is useful for students and engineers. It serves as well to illustrate effectively the lectures in the course of Computer-aided design.

Besides the PSpice simulator, students use also OrCad Capture and OrCad Layout tools [15] for schematic capture and PCB design of the different circuits in the Knowledge base.

In forth year they have an additional optional course for Computer-aided design of digital communication circuits with VHDL where they study design on programmable logic with tools as WARP 6.2 [16], ISE WEBPACK [17] and QUARTUS II [18]. Additional e-content is developed for these software tools applications in a larger Learning environment for Computer-aided design in communications.

REFERENCES

[1] A. Rusek, Barbara Oakley, "PSpice Applications in the Teaching of Wireless and High Frequency Electronics",

- Proceedings of the 2001 American Society for Engineering Education Annual Conference & Exposition*, American Society for Engineering Education, 2001
- [2] G. Marinova, D. Dimitrov, "Learning optimal synthesis of voltage regulator circuits through comparative study in PSpice", *XV International Symposium on Theoretical Electrical Engineering*, 22-24 June 2009, Lubeck, Germany, pp. 211-215
- [3] G. Marinova, *Guide for computer-aided design systems*, Filvest, Sofia, 1999
- [4] E. Gadjeva, T.Kumdjiev, S. Farhy, *Computer modeling and simulation of electronic and electric circuits with OrCAD PSpice*, Meridian 22, Sofia, 2001
- [5] E. Shoikova, I.Pandiev, "Behavioral PSpice macromodel of universal SC-filter LTC1060(MF10)", *Electronics*, Sozopol, 2001
- [6] K.A. Connor and J.Braunstein, *Fields and waves I*, 2.19.2002/31.01.2003, Rensselaer Polytechnic Institute, Troy, New York, USA, ESCE-2100/Spring 2002
- [7] Kh. S. Al-Olimat, "Shifting Teaching and Training to Online Learning: PSpice through CAMTASIA", *Proceedings of the 2008 ASEE North Central Section Conference*, American Society for Engineering Education, 2008
- [8] P. Tobin, "The role of PSpice in the engineering teaching environment", *International Conference on Engineering Education – ICEE 2007*, Coimbra, Portugal September 3 – 7, 2007
- [9] P. Tobin, *PSpice for Filters and Transmission Lines*, Book 2, Morgan Claypool publishers, Mar 2007.
- [10] P. Tobin, *PSpice for Digital Signal Processing*, Book 5, Morgan Claypool publishers, Mar 2007.
- [11] <http://home.cogeco.ca/~rpaisley4/LM555.html>
- [12] http://www.cliftonlaboratories.com/cohn_crystal_filter.htm,
- [13] <http://www.pan-tex.net/usr/r/receivers/sfilter.htm#dual>
- [14] <http://www.giangrandi.ch/electronics/crystalfilters/xtalfilters.shtml>
- [15] www.cadence.com
- [16] www.cypress.com
- [17] www.xilinx.com
- [18] www.altera.com

A list of circuit simulator softwares to download for free. Electronic circuit design, analysis, and simulation software based on linux, windows, Mac OS. PSpice is a mixed signal, industry standard circuit simulator. This software is used by many professionals for electronic system design and development. Orcad, the printed circuit board (PCB) design software is developed by the same company, Cadence. MultiSim is a great simulation software from National Instruments for electronic design engineers, academic professionals and educators, colleges, students, universities and professionals. Highlight features of this software are written above (under free resources).