Software Environments Applicable in Teaching Analog and Mixed Signals Circuits Design

Daniela Antonova Shehova

Abstract – **Contemporary requirements for the transformation of the educational process in tertiary technical educational institutions in a modern educational environment for the preparation of future engineers demands that students master methods of learning some of which are of increasing importance, and involve the simulation examination of electronic circuits.**

The paper presents an interactive development of the concept of analog and mixed signal circuits through the use of suitable software environments.

Special emphasis has been put on the prospects of using these environments in the teaching of analog circuits design to the students from the corresponding specialties in the Technical college of Smolyan.

Keywords – **simulation research, software environments, electronic means**

I. INTRODUCTION

Innovational engineering education demands that its contents be updated in accordance with the knowledge of world's information resources. It aims the acquiring of a certain amount of knowledge, skills, and complex preparation of the specialists in the area of engineering and technology through the use of innovative educational methods and techniques. Such are the innovative educational technologies that consist of a complex of modern paradigms of the educational process; methods and modern educational techniques; educational information resources aimed at the development of innovative educational environment for the professional training of engineering specialists who acquire a complex of professional and social competences and innovative thinking [1].

The electronic tools for teaching are modern educational means that provide an opportunity not only to demonstrate processes that are difficult to replicate within the training environment, but also to clarify the influence of certain parameters on the processes under investigation in the electronic devices. This allows their use as simulators of laboratory equipment and for the development of skills to run modeled processes. They lower costs in the cases when experimentation with physical equipment will prove difficult due to financial or technical impediments [2].

The fact is that the improvement of the measuring devices leads to the increase in costs for the equipment necessary for the labs. This is the reason why this integrated approach of the combination of software and physical tools of training and education in the courses of

D. Shehova is with the Technical College by Plovdiv University "Paisiy Hilendarski", 4700 Smolyan, Bulgaria, e-mail: dani_shehova@abv.bg.

analog circuits design is widely used as an alternative of expensive physical lab equipment.

II. FACTORS DETERMINING THE CHOICE OF SOFTWARE ENVIRONMENTS AS ELECTRONIC TOOLS FOR TEACHING THE COURSES IN ANALOG AND MIXED SIGNALS CIRCUITS DESIGN

The use of software environments as electronic tools for practical training is the basis for the modernization of the educational system of any country as well as an important element of European educational policy. The idea for the application of electronic tools and models in the practical training of technical specialties directly corresponds with ISO 9001 – educational quality management system which is concordant with the Higher Education Act.

The use of software environments as modern electronic educational tools in the courses in analog and mixed signals circuits design has the following premises:

▪ It is a widely known fact that when technical specialties are concerned the specifics of the fast changes in technologies and equipment development presents financial burdens when buying modern equipment for highly specialized applications (electronics, communications technologies etc.). This leads to difficulties for the technical universities and colleges and limits their being up to date with the trends of these changes. One way out of this situation is the approbation of experimental simulation exploration of electronic circuits in the teaching process;

▪ Software environments for teaching analog and digital devices through circuits modeling and analysis are used with free licensing for students. Their interface is constantly being improved and their capabilities are being extended;

▪ Young generations possess much better computer literacy and are keenly interested in new technologies;

▪ The number of students who work full time but want to continue their education and to improve their qualifications is also increasing;

The laboratory in which students are taught can be used for laboratory experimentation in various courses in the curriculum and for conducting various short-term training courses. The software applications are identical for all the workstations in the laboratory which themselves are connected in a local area network. The students conduct practical and laboratory experiments integrating the theoretical material taught with the acquiring of professional experience.

III. Software Environments with Free Access Used in the Teaching of Analog and Mixed Signals Circuits Design

When using software environments as electronic tools of training the most important factor is the careful decision that is to be made as to which environment to use. The major criterion is the presence of a wide spectrum of applicability in several different courses included in the curricula. With regard to the improvement of the quality of the educational process and the optimization of the economic costs it is advisable to take into consideration the following factors when choosing a software environment:

▪ Wide spread of the software applications;
▪ Potential for maintenance and updating of the software applications;
▪ Availability of literature suitable for the teaching of the software applications;
▪ Requirements as to the parameters of the computer hardware to be used.

The following software products have been chosen for the teaching of analog and mixed signals circuit design in accordance with the aforementioned considerations.

A. NI Multisim

NI Multisim is an intuitive simulation educational environment that has libraries with over 26 thousand components (fig.1) of well-established and leading manufacturers of semiconductor components like Analog Devices, Infineon, Semiconductor and Texas Instruments [3]. The schematics behavior is evaluated through 20 modern SPICE analyses and 22 controlling and measuring devices (fig.2) that visualize the circuits behavior which students can investigate in the laboratories.

Fig. 1. Component libraries

Fig. 2. Controlling and measuring devices

NI Multisim is an appropriate training environment as it can be applied in: teaching the theoretical basics of analog circuits design; conducting laboratory experiments. It offers a complete approach in the innovative exploration of electronic circuits through the integration of the design of electronic circuits with the interactive simulation and analysis. Through the tuning and managing the virtual measuring devices the behavior of the scheme is visualized by using the "what if" scenario.

In the teaching of the students of analog circuits design in the Technical college of Smolyan the following simulation models developed within the NI Multisim software environment are being used:

▪ inverting and non-inverting amplifier designed with Operational Amplifiers (Op Amps);
▪ integrators and differentiators;

▪ sine wave oscillations generators;
▪ pulse generators,

which have been presented in previous publications. The simulation experiment precedes the laboratory experimentation with physical models which gives the students the opportunity to enhance their theoretical knowledge and to be aware of the implications of the laboratory experimentation results. Fig. 3 shows the schematics of a rectangular pulses generator designed with an integral timer in the graphics module schematic capture. The behavior of the schematics is explored with an oscilloscope and frequency meter. In this way knowledge is acquired at three levels: mastering of the theoretical fundamentals, simulation and experimentation. The verification of the simulation results increases the motivation of the students in order to acquire new knowledge.
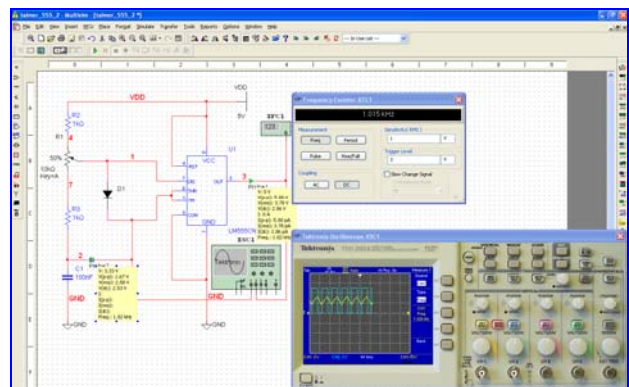

Fig. 3. Simulation exploration of a simulation exploration of a rectangular pulses generator

B. TINA-TI

TINA-TI is a free application – a product DesignSoft for Texas Instruments. It is fully functional but does not support some of the features present in the full version of TINA. The TINA-TI software package is intended for use in the teaching of electronic circuits through simulation and analysis. It works with linear and non-linear analog, digital and mixed circuits and offers a library containing more than 10,000 components. The schematic analysis is done with the help of about 20 analysis modes or 10 virtual instruments. The schemes explored can be edited while working and the results obtained can be analyzed.

▪ The software application offers analog macromodels: Amplifiers and Linear Circuits;
▪ Switched Mode Power Supplies;
▪ Noise Sourses;
▪ Sensor Simulators,

and each of them has been tested and is available to the user. They provide fast and effective way of simulating the efficiency of the circuits. Fig. 4 shows the simulation model of a generator of rectangular and triangular pulses created in the graphics editor Schematic Editor of the software environment and the oscillograms resulting from the oscilloscope readings.
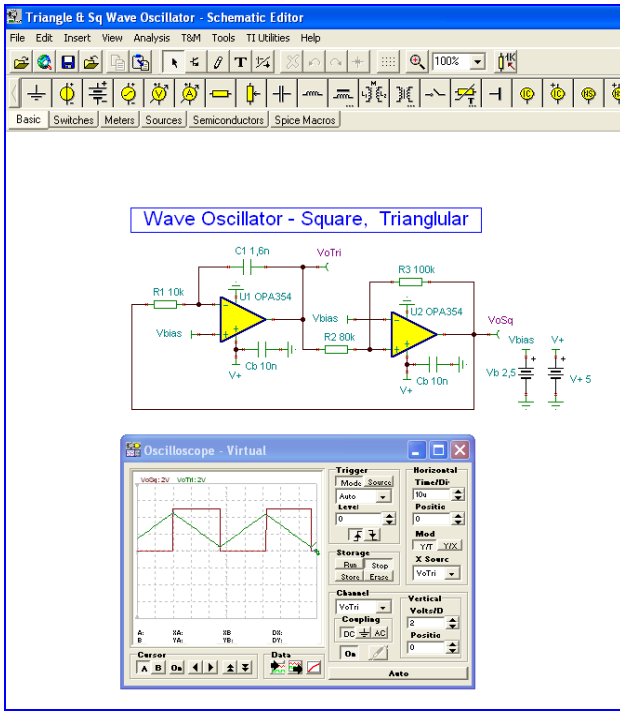
Fig. 4. Simulation model of a generator of rectangular and triangular pulses

TINA-TI possesses improved instruments (transfer functions and time responses) for the computation of calculations [4]. They are helpful in the self-study process of the students which includes writing papers and developing projects as well as exam preparation. They are conductive for the deeper understanding of the processes realized in the circuits and save valuable time by eliminating the need to check results of the computations based on formulae.

C. OrCAD

OrCAD is design software which prepares students from the corresponding specialties for the future engineering challenges through high precision design and visual interactive simulations. The different stages of the design process use various programming modules: graphical editor for the design, editing and storing of electronic circuits OrCAD Capture which can utilize extensive component libraries, including some available over Internet (registered users can access over 200 000 components); analog and analog-to-digital simulator PSpice A/D; graphical editor for the design of printed circuit boards OrCAD PCB Editor; an application for analysis of parasitic effects in printed circuit boards OrCAD Signal Explorer; an application for automated routing of printed circuit boards SPECCTRA for OrCAD; interface module SLPS for the integration of PSpice end MATLAB/Simulink [5]. OrCAD is suitable for the design and exploration of electronic circuits in the courses studied in accordance to the accepted curriculum. Fig. 5 shows the overall scheme of inverting Schmitt trigger, while fig.6

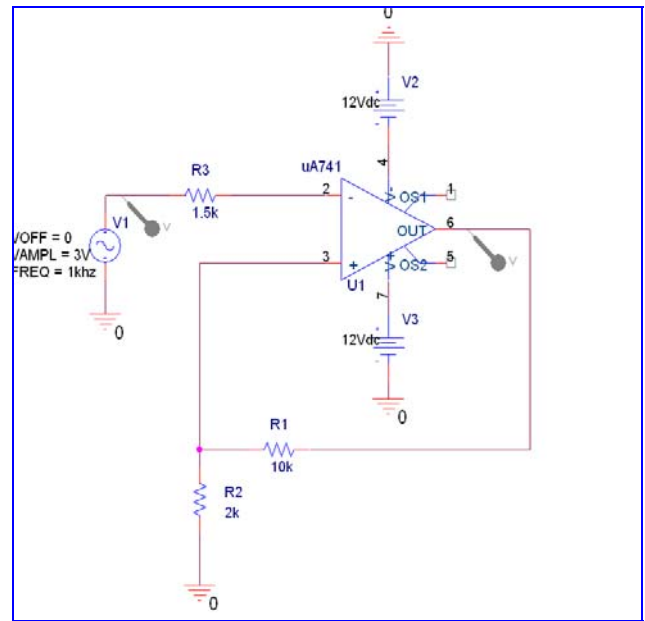shows the analysis of the processes in time domain visualized in Probe.

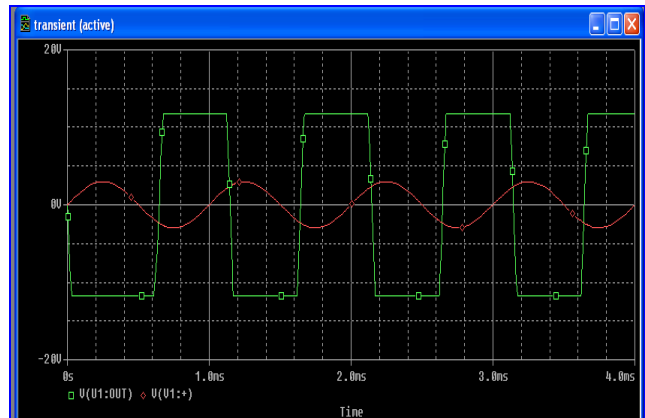Fig. 5. Overall scheme of inverting Schmitt trigger

Fig. 6. Analysis of the processes in inverting Schmitt trigger in time domain

Table 1 compares the principal features of the software environments NI Multisim, TINA–TI и OrCAD, used in the courses in analog circuits design for students in the Technical college of Smolyan, studying in Communications and computer technologies and pursuing an associate degree. The software environment NI Multisim is used in the laboratory experiments, TINA-TI is used by the students in their self-study, while OrCAD is used for working on papers and projects.

In future is yet to come creation of analog circuits models in environments of LT Spice and WEBENCH® Designer. Both of them can regard as virtual laboratory for modeling of analog circuits.

TABLE .1 COMPARISON BETWEEN THE MAJOR FEATURES OF THE SOFTWARE ENVIRONMENTS NI MULTISIM, TINA TI AND ORCAD, USED IN THE EDUCATIONAL PROCESS IN THE TECHNICAL COLLEGE OF SMOLYAN

Software environment / Features	NI Multisim *National Instruments*	TINA TI *Texas Instruments*	OrCAD *Cadence*
Components (number)	26 000	10 000	200 000
3D virtual components	present	present	-
Virtual measuring devices (number)	22	10	-
LabVIEW instruments	present	-	-
Graphical editor	Schematic Capture	Schematic Editor	OrCAD Capture
Simulation Feature	SPICE	SPICE	PSpice
Analyses (number)	20	10	10
Educational applications	Conducting laboratory experiments	For: ■ Self-study preceding lab experimentation; ■ Preparation of projects and papers; ■ Exam preparation.	For computer modelling and design of electronic circuits

IV. CONCLUSION

The paper discusses software environments used in the courses in analog circuits design taught to the students from the corresponding specialties in the Technical college of Smolyan. By using their features the lecturer and the trainees can work with existing models, and also design models through the combination of existing elements. Computer technologies allow the modelling of investigations into the properties of non-existing objects.

The electronic laboratory experimentation sessions promote the re-creation of the processes encountered in the physical technical devices being studied and the opportunities for their experimental exploration. The combination of the electronic and physical tools of education has an innovative character and creates a synergic effect on the quality of the educational and training process.

ACKNOWLEDGEMENT

REFERENCES

[1] Н. Шумакова, *Инновационные технологии в системе профессиональной подготовки студентов*, Молодой ученый, 5, pp787-789, 2013.
[2] Р. Николов, *Глобалният кампус*, Първо издание, Авангард Прима, София ,2009.
[3] http://www.ni.com/multisim/benefits/education/
[4] http://www.tina.com/student_version
[5] http://cio.bg/659