Serhiy RENDZINYAK, Natalia MUSYKHINA

Lviv Polytechnic National University

Simulation study of laboratory session of electrical engineering and electronics

Abstract. *The problems of the study of typical and specialized electrical and electronic circuits in laboratory works of electrical engineering and electronics with simulation experiment of schemes on a modern computer technology using available software in addition to natural experiment during the educational process are considered.*

Keywords: simulation study, educational process, electrical and electronic circuits.

In a wide application of computer technology in the student learning process of different specialties tendency to use Office software for editorial work with texts and databases was formed. However, for engineering training profile that is not enough. Since the professional specialized software relatively expensive, and as appropriate laboratory equipment, promising is the use of simplified versions of open-known computer-aided design of electronic circuits for the educational process.

Laboratory works on the theory of electronic circuits on a personal computer is to replace its natural experiment using computer modeling by student version of OrCAD PSpice 9.1 student version of the company OrCAD, Inc. [1], which is designed to simulate electronic circuits with analog devices (Fig. 1).

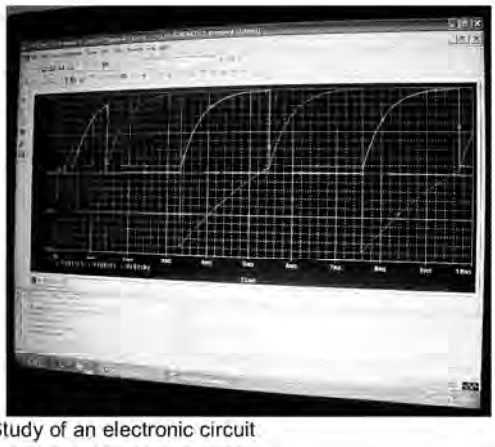

Fig.1. Study of an electronic circuit

System OrCAD PSpice to do any kind of analysis:
- Calculation of direct current with the change of parameters for a number of current-voltage characteristics and other calculations. They rank as the calculation of voltages and currents to changes in parameters of the sources and elements, the temperature within the specified limits, the sensitivity range of voltage as a function of the given value, small-signal transfer coefficients, input and output resistance as a function of the set parameters;
- Calculation of transients (time analysis) – time dependence of currents and voltages;
- Frequency analysis - dependence of voltage or current amplitudes and phases of linearized model electronic circuit on frequency of harmonic signal.

Note that the software system used OrCAD PSpice circuit builds image in compliance with standards accepted in the United States, which differs somewhat from the notation adopted in Ukraine.

To execute the specified laboratory work must be obtained from the teacher input data, enter upon the task values for circuit elements, power sources or generators that meet the parameters of the real laboratory model.

Typical tasks of laboratory practical work in electrical engineering and electronics include the following:
- Study of transients in linear circuits;
- Study of resonance in a simple oscillatory circuit;
- Experimental determination of parameters of passive two-port network;
- Study of frequency electrical filter;
- Research of simple differential and integrated RC-units;
- Research of semiconductor triode, etc.

Specialized tasks:
- Study of transistor amplifiers;
- Study of operational amplifiers;
- Study of the RC-generator on operational amplifiers;
- Research of multivibrator on bipolar transistors, etc.

This approach to laboratory work allows students to research not only real physical processes in the electrical circuits, and get to know about the features of professional specialized software.

REFERENCES

[1] www.electronics-lab.com/downloads/schematic/013/

[2] Бобало Ю.Я., Мандзій Б.А., Стахів П.Г., Писаренко Л.Д., Якименко Ю.І. Основи теорії електронних кіл. *Львів: «Магнолія плюс»*, 2006. – 296 с.

[3] OrCAD PSpice A/D. www.electronics-lab.com/downloads/schematic/013/

***Authors**: DSc, prof. Serhiy Rendzinyak, Lviv Polytechnic National University, Institute for Energy and Control Systems, Bandera str 12, 79013 Lviv, E-mail: emd@polynet.lviv.ua; senior teacher Natalia Musykhina, Lviv Polytechnic National University, Institute for Energy and Control Systems, Bandera str 12, 79013 Lviv, E-mail: nmusihina@polynet.lviv.ua.*