

APPLICATION OF THE SOFTWARE PACKAGE LTSPICE FOR DESIGNING AND ANALYSING THE OPERATION OF ELECTRONIC SYSTEMS

Pawel Ptak

Czestochowa University of Technology, Faculty of Electrical Engineering, Poland

Abstract. *The paper presents application of the software package LTspice in the teaching of designing and analysis of electronic systems operation. The program has been used for simulating the operation of typical electronic systems, such as filters. The systems created in the simulation can provide a basis for preparing students to participate in laboratory classes. The results obtained in a simulation can be subsequently verified by testing real systems during the electronics laboratory classes. The results of simulation and real experiments can also be compared and utilised for making an analysis and writing a report on the experiment.*

Keywords: *simulation, problem task, electronics laboratory didactics.*

Introduction

Manufactured by Linear Technology Corporation® and available free of charge, application LTSpice is a kind of software intended for designing electronic systems and simulating their operation in the virtual environment (Praužner, 2013). Constructing a measuring system involves selecting symbols denoting electronic components from the menu and inserting them into the virtual simulation screen. The connected components form a data-flow sheet, with data travelling between the components of the system (Praužner, 2016). The components are built into the application but their inventory can also be expanded by installing libraries available at the software manufacturer's website. The program offers fairly complex functionalities and can be used for building complicated electronic systems and for observing their simulated operation. It is likewise possible to observe the behaviour of electrical and electronic parameters during a simulation thanks to in-built measuring and registering instruments (Praužner, 2012). The value of current and voltage can be checked at a number of predetermined points of the system under test. An example of a system constructed by means of the program LTSpice is presented in Fig. 1 (Electronic Circuit, 2018).

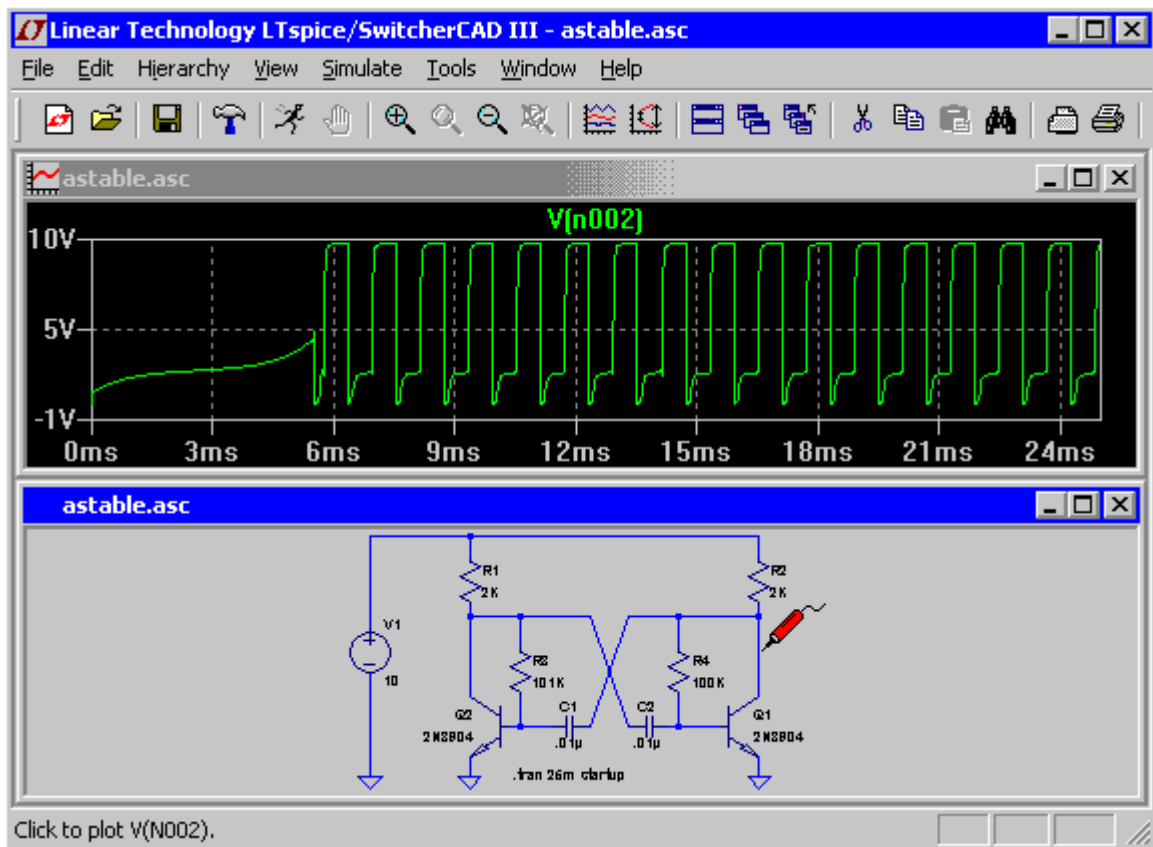


Figure 1. Electronic system created in the LTSpice environment

Simulation of a system under test in LTSpice

To perform a measuring task, an active band-pass filter was designed in LTSpice. The characteristics of the filter were determined by finding the value of output voltage for a constant value of the input voltage. The frequency of the filter was adjusted in the range of 20 to 20000 Hz. On the basis of the data obtained the amplification coefficient k_u was calculated during the simulated work of the system. Fig. 2 presents the design of the active band-pass filter system created in the software LTSpice, at the filter quality $Q = 5$, and Fig. 3 the same system is shown at the filter quality $Q = 707$ (Jędrzejczyk, 2017).

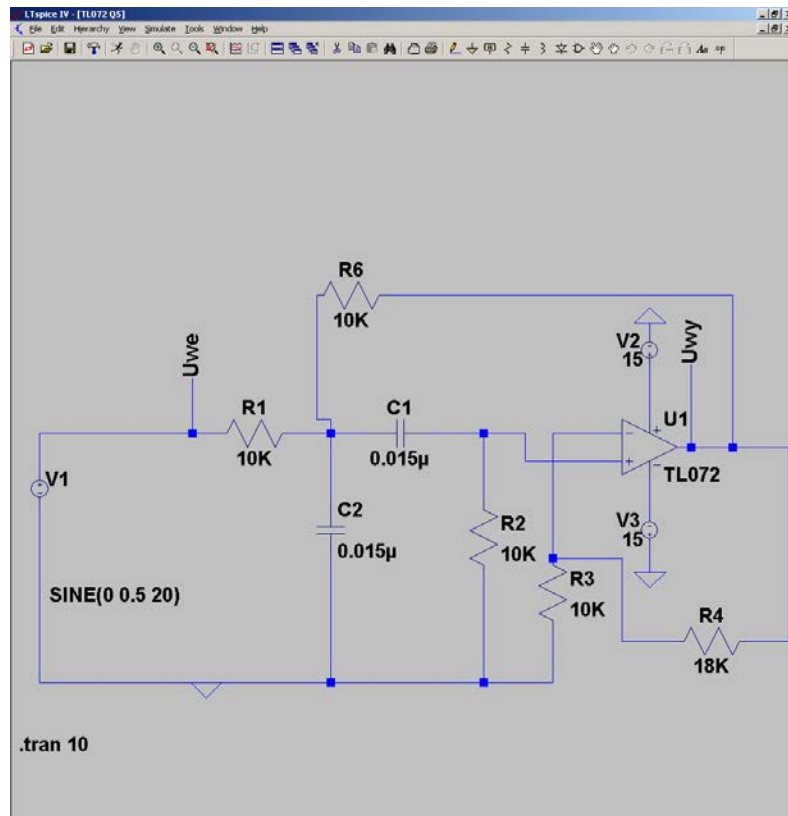


Figure 2. System for testing a band-pass filter created in the program LTSpice at $Q = 5$

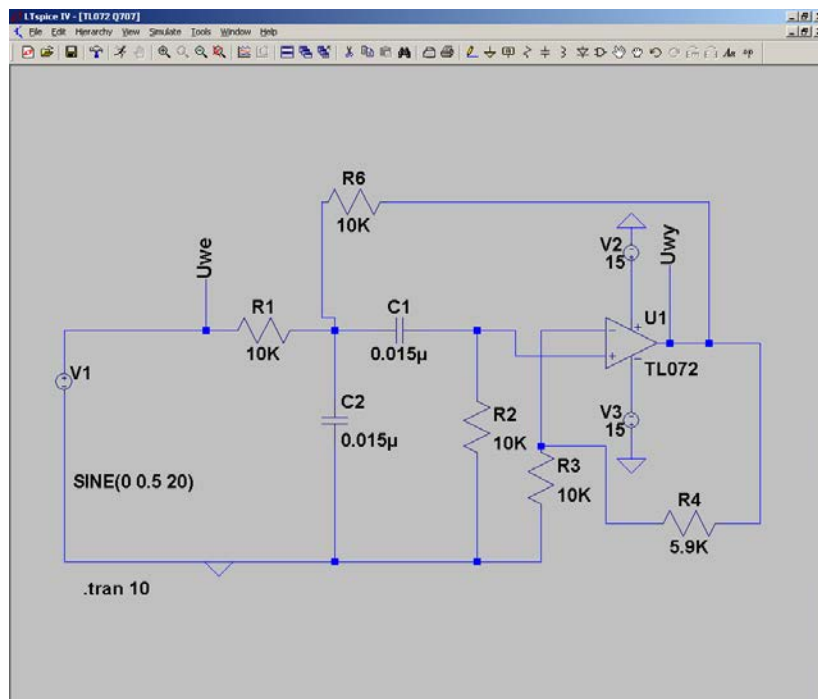


Figure 3. System for testing a band-pass filter created in the program LTSpice at $Q = 707$

Due to the fact that LTSpice represents the results of measurements as sinusoid curves, the values of input and output voltages of the filter under test had to be expressed as RMS values. Fig. 4 presents a screen of an LTSpice simulation of a low-pass filter system with sinusoid waveforms of input and output voltages (Jędrzejczyk, 2017).

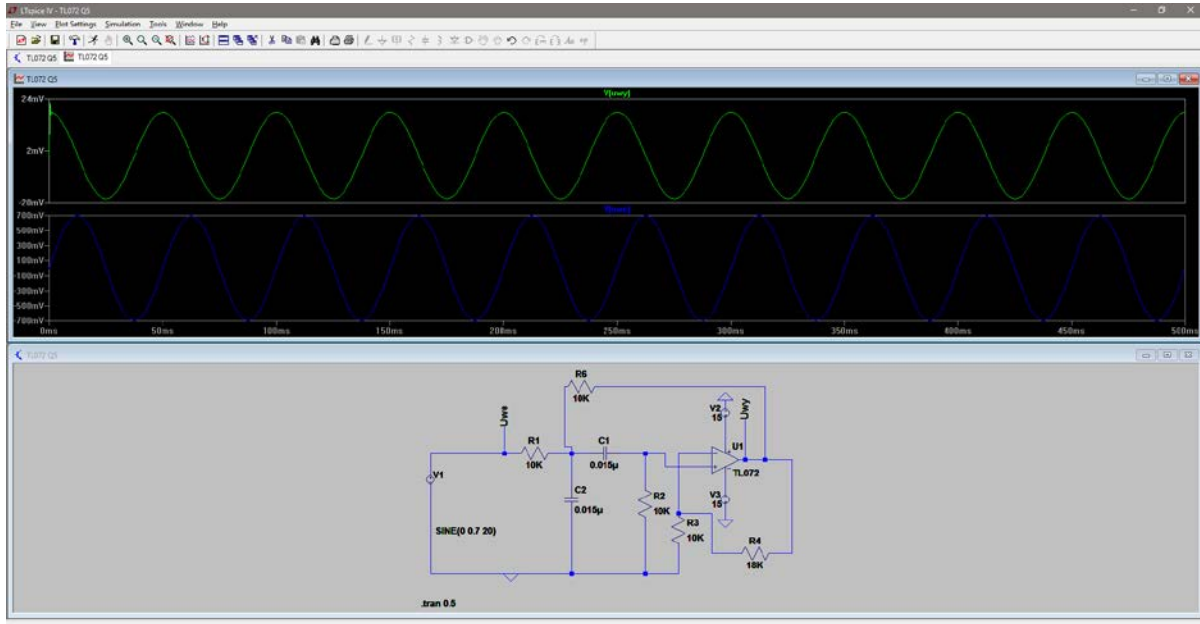


Figure 4. System for testing a low-pass filter during simulation with sinusoid voltage curves

The results obtained from the LTSpice simulation were further compared with results obtained from laboratory measuring instruments in real tests. The measuring data were included in tables and the results of the comparison were presented as graphs. Fig. 5 shows a juxtaposition of results obtained empirically by testing real filter systems in laboratory classes with results coming from a LTSpice simulation (Jędrzejczyk, 2017).

On the basis of the results obtained it can be observed that for the factor $Q = 5$ the shapes of the curves are similar but one is shifted with respect to the other, which can be an effect of using ideal electronic elements built into the simulation program. For $Q = 707$, the curve representing real measurements is somewhat deformed, which can result from measuring errors. The vertical shift of the measurement characteristics with respect to the simulation characteristics can be explained, like in the case of the quality factor $Q = 5$, by the fact the in the simulation ideal elements were used.

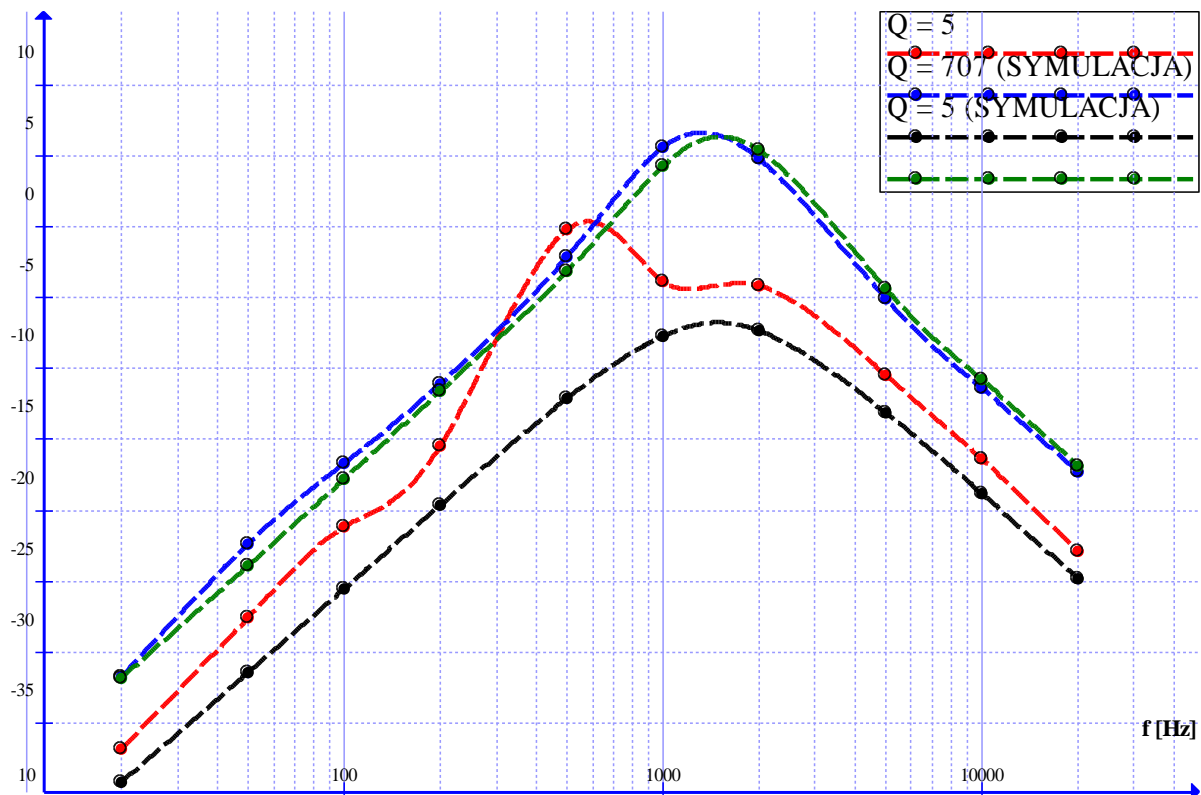


Figure 5. Comparison of two sets of results of testing the amplification coefficient of a low-pass filter: obtained empirically and obtained in a simulation

Concluding remarks

The research described in the paper indicates that the simulation software LTSpice offers a possibility of conducting a thorough study on the functioning of an electronic system. Thanks to that, students can test the operation of such a system at the stage of designing. They can also compare results obtained in simulations with those obtained in laboratory classes (Ptak & Prauzner, 2010; Ptak, 2016; Ptak, 2017). In this way, they have greater chances of gaining knowledge and skills in the field of electronics (Ptak, 2014; Ptak, 2015; Prauzner, 2017).

Using ideal elements in the simulation program brings to notice an important observation, namely, that the functioning of electronic systems depends not only on their construction but also on the parameters of their particular electronic components (Prauzner, 2012; Ptak & Prauzner, 2013; Ptak & Prauzner, 2014). This sheds light on how electronic systems work and how their operation can be influenced, which should appear interesting to students (Zloto et al., 2012). By active participation in designing electronic

systems, student are offered an opportunity to experiment with a number of possible designs in a creative way (Prazner, 2015; Krzywanski et al., 2017).

References

- Electronic Circuit, (2018). *LTspice IV Circuit Simulation Schematic Capture Tool*. web site: <http://www.electroniccircuits.com/electronic-software/ltspice-iv-circuit-simulation-schematic-capture-tool>
- Jędrzejczyk, M. (2017). *Analiza możliwości i zastosowania programów symulacyjnych wspomagających nauczanie elektrotechniki i elektroniki*. Wydział Elektryczny, Politechnika Częstochowska
- Krzywanski, J., Grabowska, K., Herman, F., Pyrka, P., Sosnowski, M., Prazner, T., & Nowak, W. (2017). Optimization of a three-bed adsorption chiller by genetic algorithms and neural networks, *Energy Conversion and Management*, 153 (2017), 313–322.
- Prazner, T. (2013). Information Technology in Contemporary Education – Individuals' Research. *American Journal of Educational Research*, 2013, Vol. 1, No. 10.
- Prazner, T. (2015). Analysis of the results of the pedagogical research and EEG in the aspect of effective modern teaching aids in the technical education. *Society. Integration. Education, Proceedings of the International Scientific Conference. Volume IV, May 22th-23th 2015*, 4 80-489.
- Prazner, T. (2017). The effectiveness of school education - featured implications considerations. *Society. Integration. Education, Proceedings of the International Scientific Conference. Volume III, May 26th-27th 2017*, 558-564.
- Prazner, T. (2012). Zakłócenia elektromagnetyczne w elektronicznych systemach alarmowych. (The electromagnetic interferences in the electronic alarm systems). *Przegląd Elektrotechniczny*, 88 (2012), nr.12b, 205-208.
- Prazner, T. (2016). Interactive computer simulation as a response to contemporary problems of technical education. *Society. Integration. Education, Proceedings of the International Scientific Conference. Volume II*, 579-588.
- Ptak, P., & Prazner, T. (2010). Wykorzystanie pakietu DasyLab w nauczaniu podstaw elektroniki. *Edukacja. Studia, Badania, Innowacje*, NR 02/2010, 159-164.
- Ptak, P., & Prazner, T. (2013). Badanie czujników detekcji zagrożeń w systemach alarmowych. *Przegląd Elektrotechniczny*, NR 10/2013, 274-276.
- Ptak, P., & Prazner, T. (2014). Analiza parametrów pracy wybranych czujników pola magnetycznego. *Przegląd Elektrotechniczny*, R. 90 NR 12/2014, 273-276.
- Ptak, P. (2014). Software package dasyLab supporting research and teaching. E-Technologies in Engineering Education eTEE'2014, *Zeszyty Naukowe Wydziału Elektrotechniki i Automatyki PG, Nr 37/2014, Gdańsk*, 71-74.
- Ptak, P. (2015). Application of DasyLab in Teaching Electrical Engineering. *Society, Integration, Education. Proceedings of the International Scientific Conference. Volume IV, May 22nd-23rd, 2015*, 490-499.
- Ptak, P. (2016). Application of software packages in research and didactics. *Society, Integration, Education. Proceedings of the International Scientific Conference. Volume II, May 27th-28th, 2016*, 589-597.
- Ptak, P. (2017). *Application of computer programmes in research projects and teaching*. *Society, Integration, Education. Proceedings of the International Scientific Conference. Volume III, May 26th-27th, 2017*. 565-574.

Zloto, T., Ptak, P., & Prauzner, T. (2012). Analysis of signals from inductive sensors by means of the DasyLab software. *Annales Universitatis Mariae Curie-Skłodowska Sectio AI Informatica XII, NR 1/2012*, 31-37.