SOCIETY. INTEGRATION. EDUCATION
Proceedings of the International Scientific Conference. Volume V, May 25th-26th, 2018. 409-419

APPLICATION OF MULTISIM AND LTSPICE SOFTWARE PACKAGES TO SIMULATE THE OPERATION OF ELECTRONIC COMPONENTS AS AN ALTERNATIVE TO MEASUREMENTS OF REAL ELEMENTS

Pawel Ptak
Czestochowa University of Technology, Poland

Abstract. *The paper presents the process of carrying out simulations of the operation of electronic elements by means of software packages Multisim and LTspice. Using the software, students can test a possibility of replacing real measurements by results obtained from a simulation. The can reach the relevant conclusions by comparing the results and analysing the data obtained. In this way students gain experience needed for carrying out a didactic project in a creative way.*
Keywords: *simulation, problem task, electronics lab didactics.*

Introduction

Programme LTSpice is a free app manufactured by Linear Technology Corporation®. It is used for constructing electronic systems and performing simulations on them. Equipped with an ample library of electronic components and subsystems, it offers a possibility of constructing fairly complex systems (Jędrzejczyk, 2017). The inventory of components can be expanded by installing new libraries provided by the app producer.

To insert an electronic component into the simulation screen, one has to select this component from a menu or using a keyboard shortcut (Prauzner, 2013). After the components have been placed in the simulation window, they have to be connected in accordance with the diagram of the measuring system (Prauzner, 2013).

Another program intended for constructing electronic systems and their simulation is NI Multisim manufactured by National Instruments Corporation®. It includes an extended base of subsystems consisting of about 20,000 components, both real and ideal. They can be used for constructing highly complex electronic systems (Ptak, 2016). The program is more extended and offers wider possibilities that the similar in application LTSpice. Multisim has an in-built drag and drop mechanism for inserting and measuring electronic

http://dx.doi.org/10.17770/sie2018vol1.3120

components into a simulation by choosing these components from scroll-down menus, in which the available components are displayed (Ptak, 2017). Simulations carried out in Multisim are interactive, and it is possible to modify the parameters of input and output signals during a simulation and to observe how such modifications affect the operation of the systems under scrutiny thanks to the in-built measuring and registering systems (Jakubiec, 2017). The signals which are being measured can be also observed on the screen of an in-built oscilloscope and recorded as graphic files (Prauzner, 2016).

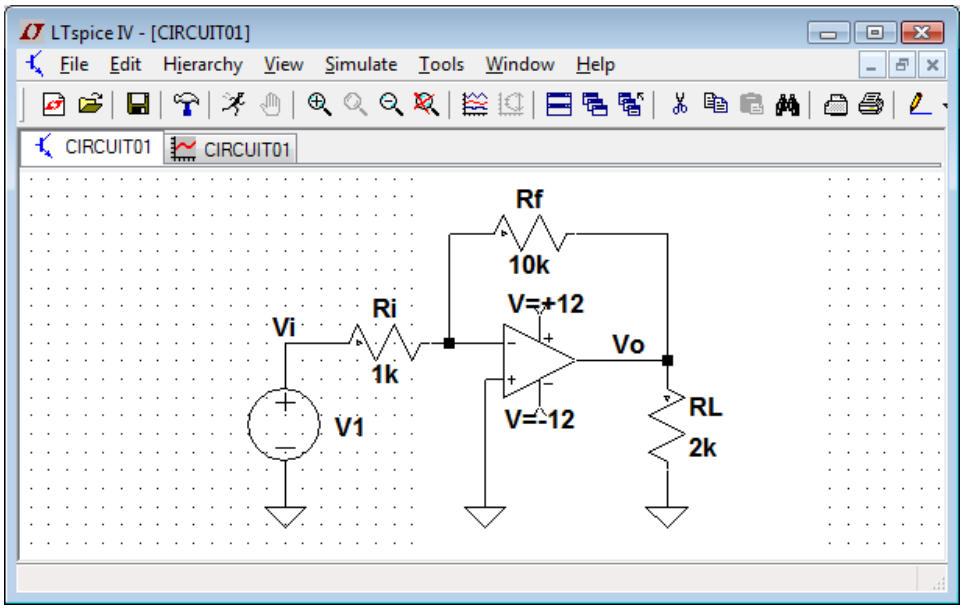

Figure 1. **Presents an example of a system constructed in the LTSpice environment** (University of Evansville, 2009)

Some of the measuring instruments are real devices, which are built into the simulation program together with a realistically looking control panel. Modifying parameters of the system during its operation and observing the results of such modifications resemble the work in a real electronic laboratory with real measuring equipment (Prauzner & Ptak, 2014). Fig. 2 presents an example of an electronic system constructed in the NI Multisim environment.

Two measuring systems were constructed to perform a task involving the testing of a semiconductor diode and a transistor. The system designed for testing the diode was constructed in LTSpice and also in NI Multisim. The other system for testing a bipolar transistor was likewise constructed in LTSpice and then in NI Multisim. Constructing the same measuring system in both software packages was justified by the need to check whether and to what extent such applications can replace laboratory measurements of real electronic components, and if yes, whether the results so obtained are comparable with those from laboratory measuring instruments.

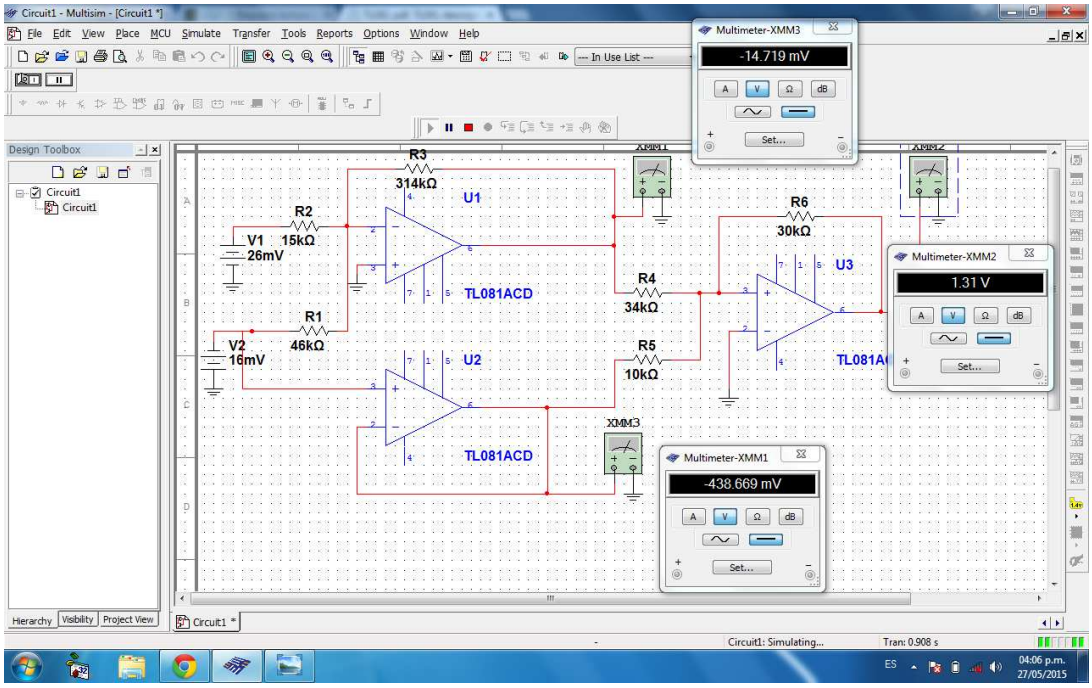

Figure 2. **Electronic system created in NI Multisim Measurements carried out in labs and simulated in programs** (Chavarin, 2015)

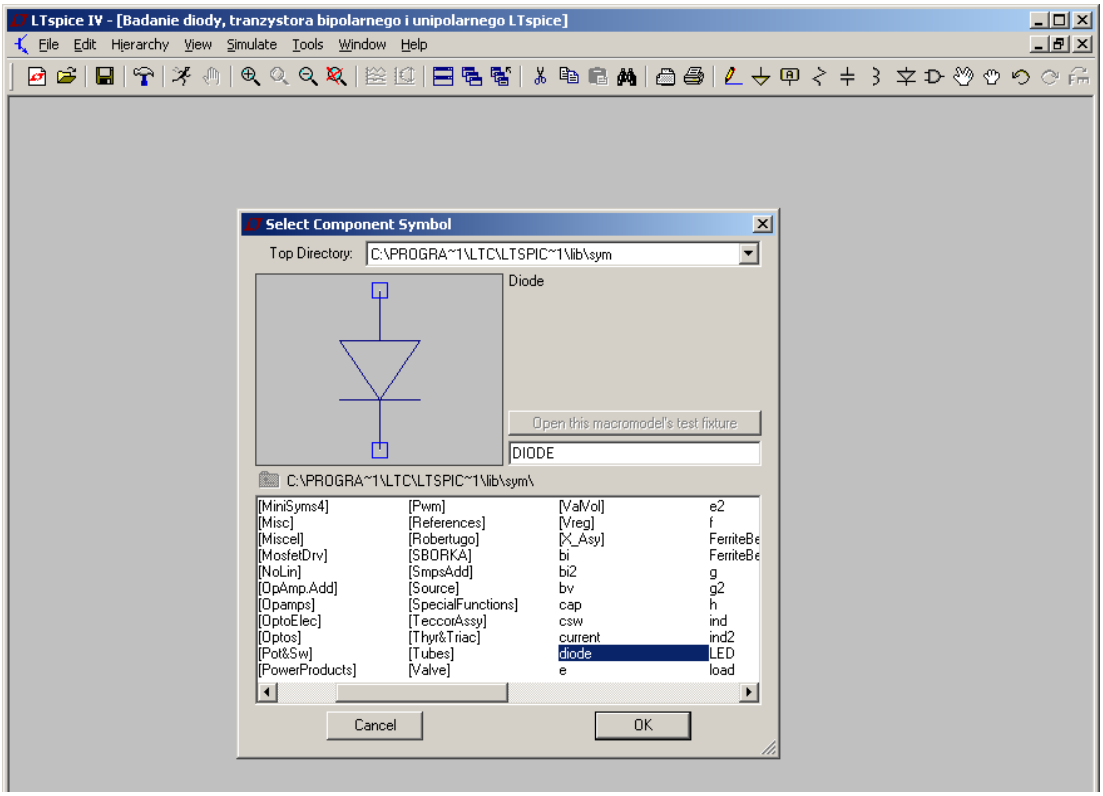

Figure 3. **Procedures for inserting electronic components to the simulation screen in LTSpice**

Fig. 3 presents the procedure of inserting electronic components to the simulation from the base available in LTSpice, and Fig. 4 shows a selection of electronic components used for a simulation in NI Multisim (Jędrzejczyk, 2017).

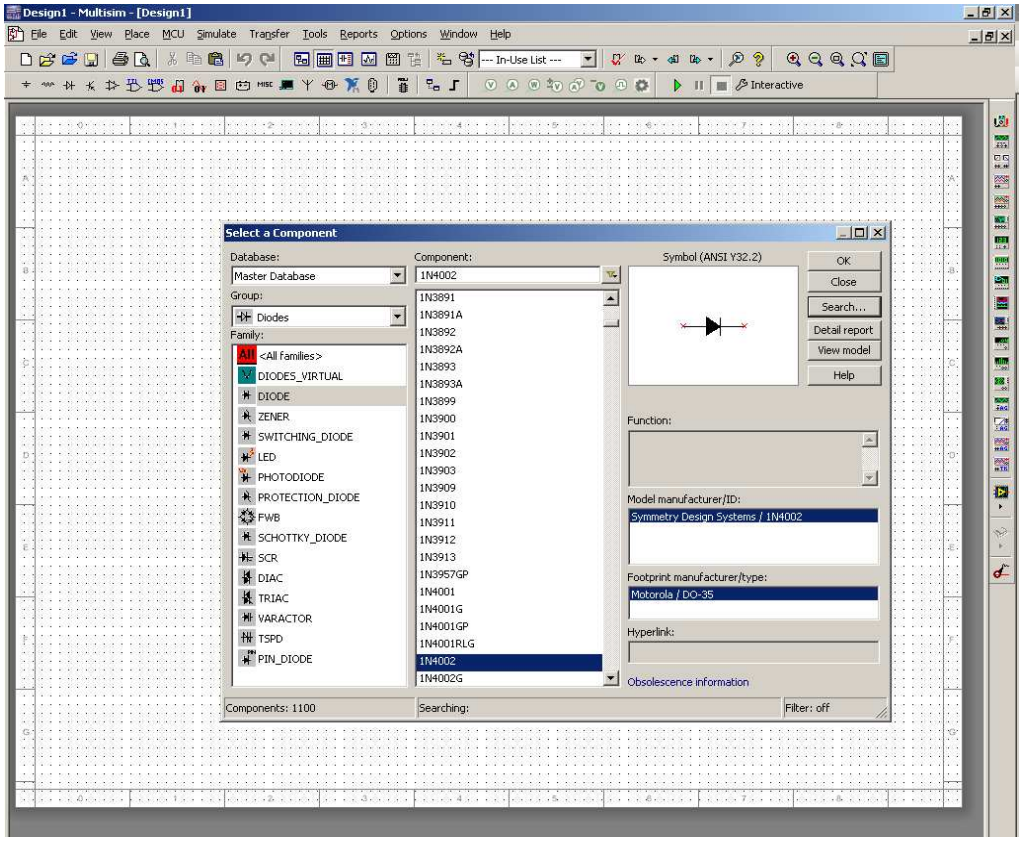

Figure 4. **Selection of electronic components for a NI Multisim simulation**

Fig. 5 presents the system designed for carrying out measurements on the semiconductor diode constructed in the LTSpice package and Fig. 6 presents the same system constructed in the program NI Multisim (Jędrzejczyk, 2017).

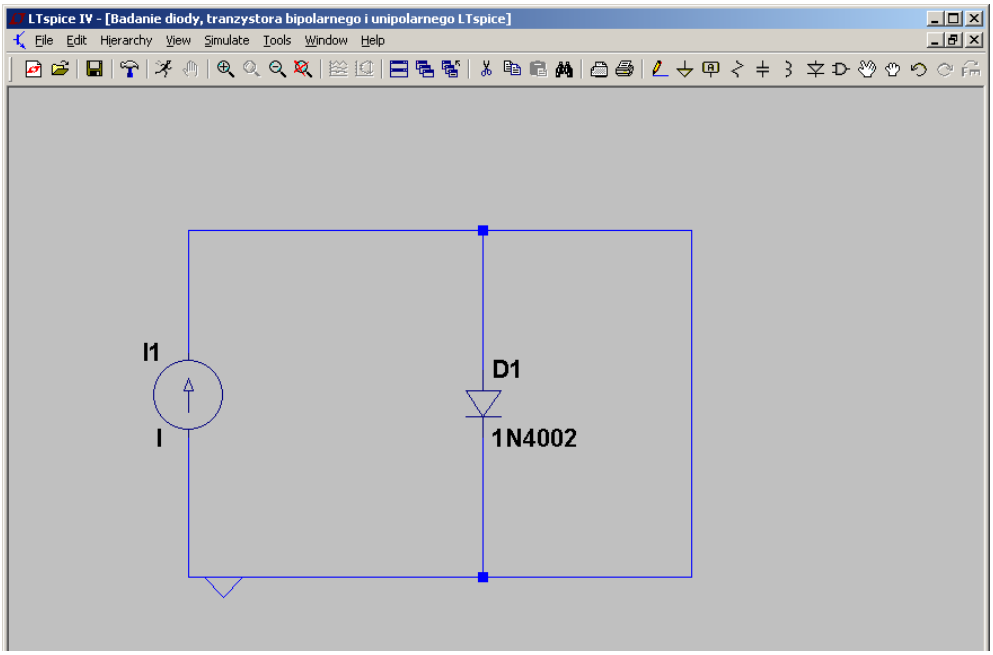

Figure 5. **System for measuring the parameters of a semiconductor diode created in LTSpice**

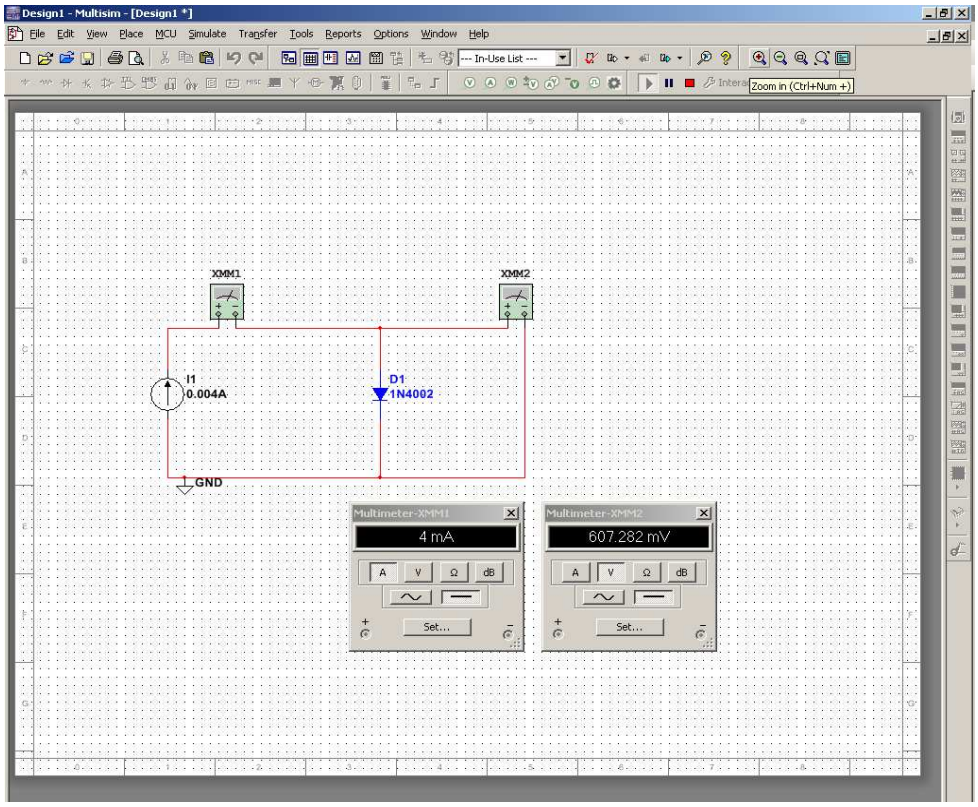

Figure 6. **System for measuring the parameters of a semiconductor diode created in NI Multisim**

After the measurements using the two software packages were performed, their results were written down in tables and represented as graphs comparing the lab results of real measurements with the simulation results. Fig. 7 presents a comparison of real vs. LTSpice results, whereas Fig. 8 real vs. NI Multisim results.

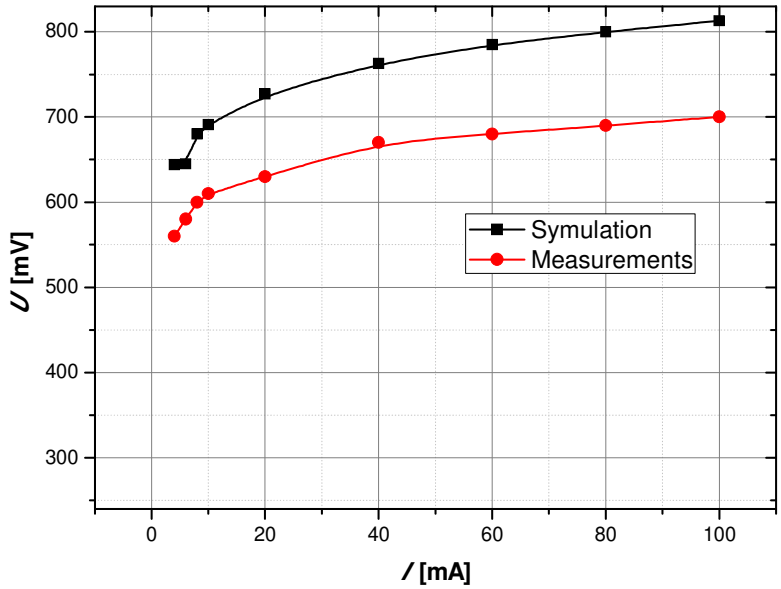

Figure 7. **Comparison of the results obtained in the program LTspice with those obtained in real measurements on a semiconductor diode type 1N4002**

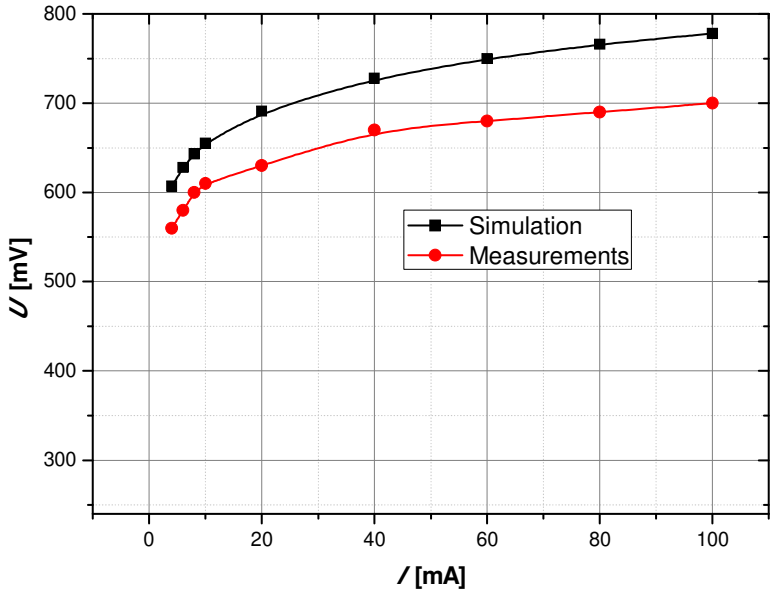

Figure 8. **Comparison of the results obtained in the program NI Multisim with those obtained in real measurements on a semiconductor diode type 1N4002**

As indicated by analyses, the characteristics representing the measurement results and those representing the simulation results are similar, with the latter being shifted vertically by a constant value. This is caused by the fact that the electronic components used in the simulations have ideal characteristics and inaccuracies and errors of real measurements are disregarded.

The other element put under test was a bipolar transistor in a measuring system. The task involved measuring the values of current at the transistor's base and collector for various values of resistance at the base and various values of voltage supplying the collector. Two measuring systems were created for this purpose, one in LTSpice and the other in NI Multisim. Figs 9 and 10 present the two bipolar transistor measuring systems constructed in LTSpice and in NI Multisim, respectively (Jędrzejczyk, 2017).

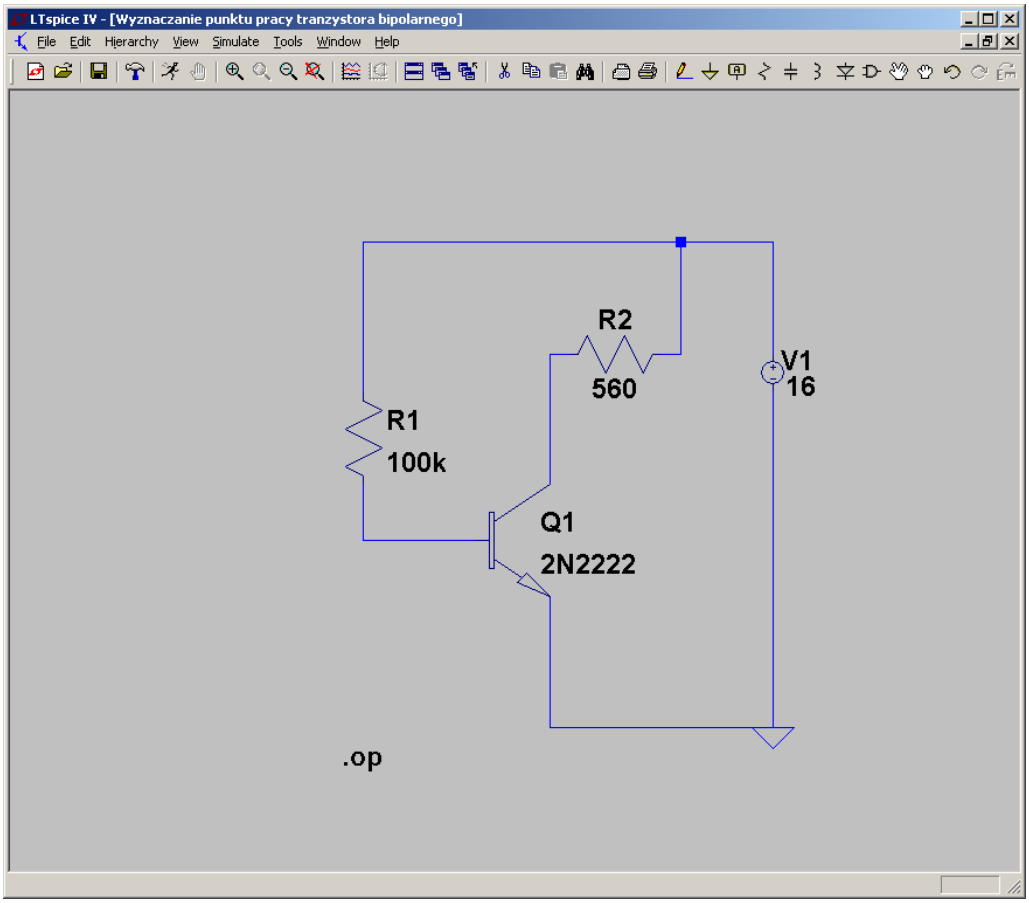

Figure 9. **System for measuring the parameters of a bipolar transistor created in LTSpice**

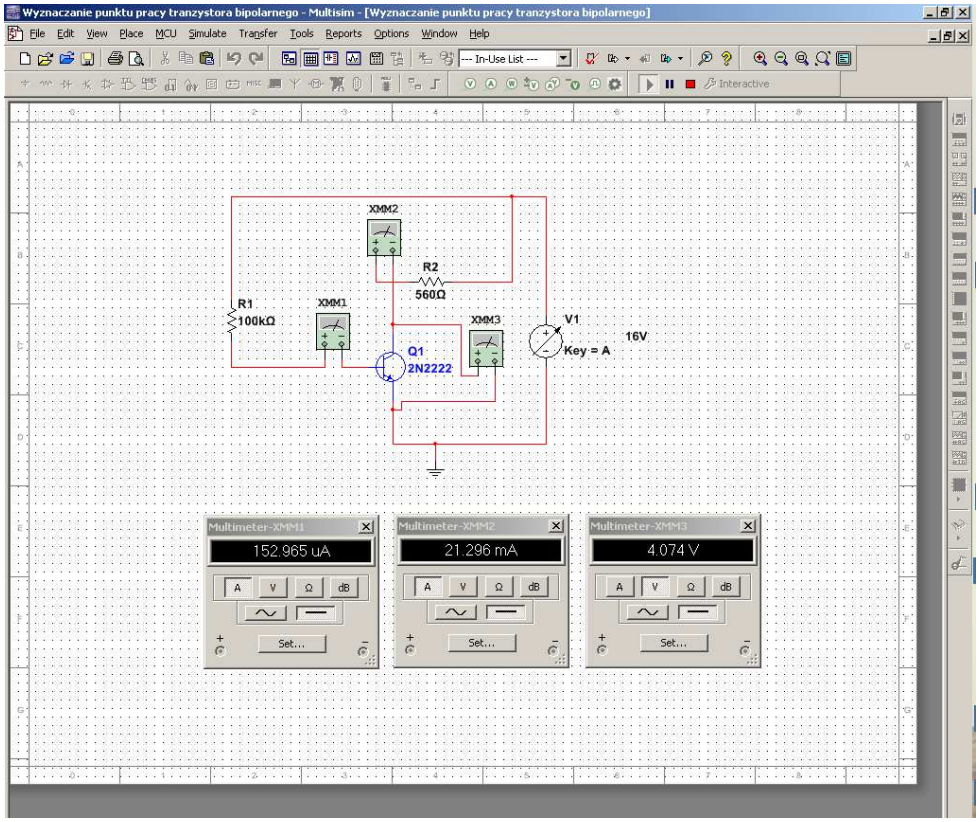

Figure 10. **System for measuring the parameters of a bipolar transistor created in NI Multisim**

The results of laboratory measurements and results obtained from simulations carried out in the software packages were written down in tables and represented as graphs, so that they could be subjected to analysis. Fig. 11 shows a comparison of results obtained from a LTSpice simulation to laboratory measurement results, whereas Fig. 12 juxtaposes NI Multisim simulation results with laboratory measurement results.

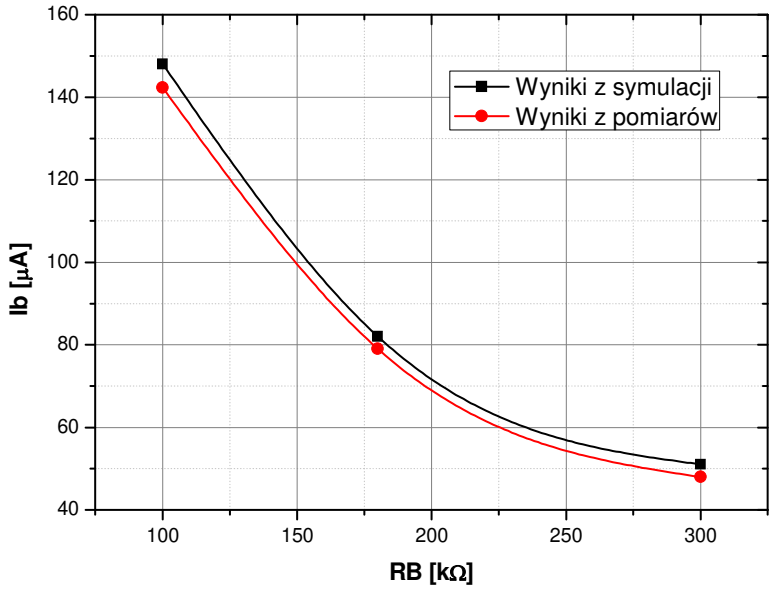

Figure 11. **Comparison between results of a LTspice simulation and results of real measurements on a bipolar transistor type 2N2222**

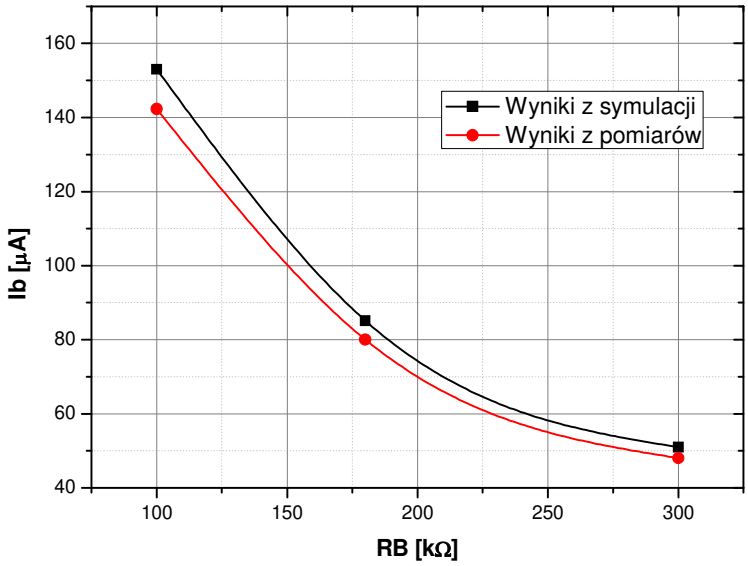

Figure 12. **Comparison between results of a NI Multisim simulation and results of real measurements on a bipolar transistor type 2N2222**

Concluding results

The research presented in this paper indicates that the results obtained from simulations carried out in the two software packages, i.e. LTSpice and NI Multisim are close to those obtained in laboratory measurements. It can be noted that the shapes of the characteristics are similar, but the values are shifted with respect to each other. The shift can be attributed to interfering factors and

variable parameters of electronic elements used in real measurements (Prauzner, 2012).

Having analysed the results obtained, it can be assumed that software packages can be successfully applied for simulating the operation of electronic systems (Zloto et al., 2012; Olesiak, 2017; Migo & Noga, 2015). Even though the results obtained in simulations do not accurately correspond to the real laboratory measurements, they sufficiently well represent the operation and parameters of the electronic components under test (Ptak, 2015; Ptak, 2014; Prauzner & Ptak, 2010).

Simulation software provides an excellent tool to complement the traditional measuring techniques employed in laboratory classes. They can be used by students to get prepared for the classes and to analyse the operation of electronic systems when they process results obtained in a lab (Krzywanski et al., 2017; Depesova et al., 2008; Noga, 2009). When it is not possible to perform measurements on real elements, e.g. due to a breakdown of a measuring stand, the work can be done using the software packages, both LTSpice and NI Multisim. Additionally, they can be used for carrying out complicated tasks involving creative experiments as it minimises possible damage to costly equipment and electronic components (Noga et al., 2014; Prauzner, 2015; Prauzner, 2017).

References

Chavarin, E. (2015). *Practica 12, Amplificador Operacional.* Electronica 1, Ultimas Entrados, web site: https://eduardochavarin.files.wordpress.com/2015/06/simulacion-multisim1.png

Depesova, J., Vargova, M., & Noga, H. (2008). *Edukacja techniczno informatyczna w opinii nauczycieli.* Uniwersytet Pedagogiczny, Kraków, 149-157.

Jakubiec, B. (2017). The utilisation of industrial process models in acquiring the practical skills of plc programming. *Society, Integration, Education. Proceedings of the International Scientific Conference. Volume III, May 26th-27th, 2017*, 483-491.

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., Prauzner, T., & Nowak, W. (2017). Optimization of a three-bed adsorption chiller by genetic algorithms and neural networks. *Energy Conversion and Management 153*, 313–322.

Noga, H. (2009). Sociometric methods in technological and information technology education. In: *Trends in Education. Information technologies and technical education. Olomouc, Vols.*1, 165-169.

Noga, H, Piaskowska-Silarska, M., Depešová, J., Pytel, K., & Migo, P. (2014). Neuro - didactic erspective of creative attitude towards education in the third millennium - examination of individual cases. In: *Emerging eLearning Technologies and Applications (ICETA), 2014 IEEE 12th International Conference on*, 355 – 360.

Migo, P., & Noga, H. (2015). Start-up of SSTC semiconductor tesla coil - an example of an educational project. *Przeglad Elektrotechniczny, Volume 1,* Issue 12, 167-169.

Olesiak, K. (2017). Application of fuzzy logic toolbox for modelling fuzzy logic controllers. *Society, Integration, Education. Proceedings of the International Scientific Conference. Volume III, May 26th-27th, 2017*, 539-546.

Prauzner, T. (2013). Information Technology in Contemporary Education – Individuals' Researche. *American Journal of Educational Research*, *Vol. 1*, No. 10.

Prauzner, 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*, 480-489.

Prauzner, 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.

Prauzner, T. (2012). Zakłócenia elektromagnetyczne w elektronicznych systemach alarmowych. (The electromagnetic interferences in the electronic alarm systems). *Przegląd Elektrotechniczny*, *88* (2012), nr.12b, 205-208.

Prauzner, 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*, *Rēzekne, Latvia, May 27th – 28th 2016*, 579-588.

Ptak, P., & Prauzner, T. (2010). Wykorzystanie pakietu DasyLab w nauczaniu podstaw elektroniki. *Edukacja. Studia, Badania, Innowacje, NR 02/2010*, 159-164.

Ptak, P., & Prauzner, T. (2013). Badanie czujników detekcji zagrożeń w systemach alarmowych. *Przegląd Elektrotechniczny, NR 10/2013*, 274-276.

Ptak, P., & Prauzner, 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*, 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.

University of Evansville LTSpice IV Library and Tutorials, (2009). *Transient Analysis, With Time Varying Sources*, University of Evansville, web site: https://csserver.evansville.edu/~richardson/courses/Tutorials/LTspiceIV/04_TimeVaryingSources/html/04_TimeVaryingSources.html

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.