FRI-2G.302-1-CCT2-01

IMPROVEMENT OF THE LEARNING PROCESS IN THE SUBJECT OF SEMICONDUCTOR COMPONENTS THROUGH SIMULATIONS [14]

Assist. Prof. Ventsislav Keseev, PhD
Department of Telecommunications,
“Angel Kanchev” University of Ruse
Tel.: +359 82 888 831
E-mail: vkeseev@uni-ruse.bg

Abstract: *Possibilities of improving the learning process for Electronics subjects through simulations of analog electronic circuits are presented. Simulations could be used in every part of the education process. LTspice is suggested as a powerful free to use simulation platform, that could be used in the education process, and used by the students for their own investigations and homework. Our experiences with the application of simulations in the lectures and exercises for the semiconductor components subject are presented. This is possible due to the many widely available simulation models for many different components, that allow the best one to be chosen for more realistic results. The examples include the automatic drawing of component characteristics, measurement of different parameters and comparison with the real values, choosing an appropriate simulation model and design of a schematic based on the simulation characteristics and parameters. The possibilities for their beneficial application for the education process improvement are endless. The conclusion is that they considerably improve the understanding of the studied material and allow students to develop the necessary experience, skills and correct methods of work, and to do their own projects and investigations at home without the need for expensive equipment.*

Keywords: *Education, Simulation, Semiconductor components, characteristics, models, electronics.*

INTRODUCTION

The good education system is one of the main supporting pillars of every prosperous country. Its main objective is the students to acquire the necessary understanding in the studied field in order to be able to solve practical problems of a different nature. In order for this requirement to be fulfilled, the students must be able to visualize the undergoing processes and to be able to work independently at home. This is easier to achieve in certain education fields, but in others it is more difficult. In the Electronics education field, the students find it difficult to understand how different electronic and semiconductor components work and hence how electronic circuits consisted of such components function. The learning process could be improved through lengthy exercises with high quality contemporary measurement equipment and special setups, but it requires a lot of time and effort, and very often they don’t have the necessary prerequisites to work by themselves at home. At the same time, not everything could be easily and well visualized, even with modern equipment. Certain problems are even difficult to be solved without simulation, not ensuring any success and causing a long cycle time with a low success rate (Melis, T., Simeu, E., Auvray, E., & Armagnat, P., 2020). These problems could be solved with the help of simulations of analogue electronic circuits through specialized pspice software, but our previous experience was that such programs are very expensive, difficult to use, good for small circuits only and often lead to unrealistic simulation results. At the same time, software experiments require a prerequisite knowledge of appropriate simulation techniques (Abramovitz, A., 2011).

Recent search led to the conclusion that the mentioned limitations have considerably improved in the recent years and today there are powerful softwares for analogue circuit simulations that bring realistic results even for more complicated ones. Currently, some of these programs are even free to use and allow widely available online pspice simulation models of different electronic and semiconductor components to be incorporated in simulation circuits. Nowadays, many

[14] Докладът е представен на научната сесия на 27.10.2023 в секция „Комуникационна и компютърна техника“ с оригинално заглавие на български език: ПОДОБРЯВАНЕ НА УЧЕБНИЯ ПРОЦЕС ПО ДИСЦИПЛИНАТА „ПОЛУПРОВОДНИКОВИ ЕЛЕМЕНТИ“ ЧРЕЗ СИМУЛАЦИИ

semiconductor producing companies create pspice simulation models for their components and different companies also do so. Different free to use pspice models exist for certain components and usually they are with different build quality, which means that they mimic the real component characteristics with different levels of success in different simulation scenarios. The creation of a good simulation model is a lengthy time-consuming process that requires knowledge, experience and a lot of hard work, but they allow designers to optimize system performance (Stupar, A., McRae, T., Vukadinović, N., Prodić, A., & Taylor, J., 2019), analyze electromagnetic compatibility (Sakairi, H., Yanagi, T., Otake, H., Kuroda, N., & Tanigawa, H., 2018), and reduce the number of hardware iterations (Li, K., Evans, P., & Johnson, M., 2017). In this regard, the usefulness of time-domain system models in these roles is dependent on their accuracy, convergence reliability, and simulation speed (Nelson, B., & al., 2021). However, there are significant challenges in achieving all three criteria simultaneously (Ceccarelli, L., Kotecha, R., Iannuzzo, F., & Mantooth, A., 2018).

Nowadays, the availability of more than one free pspice models often allows the best model to be selected for particular simulation study, which brings useful realistic results. This allows certain modern analog circuit simulation platforms to be incorporated successfully in the education process, because they have the potential to generate realistic and predictable simulation results, that are in accordance with the real parameters of the simulated components stated in their datasheets.

The aim of this work is to present our analogue circuit simulation findings and experiences with their current useful incorporation in the education process of the Semiconductor components subject. The simulations turned out to be very helpful for visual clearing of the difficult to understand studying matter. They are used in exercises as well as in certain lectures and our experience is that they improve considerable the students' understanding.

FIELDS OF APPLICABILITY OF SIMULATIONS AND APPROPRIATE SOFTWARE

Simulations could be used in every part of the education process for better visualization and clearance of the studying matter. In practice, the electronics experts use simulations for faster development of high-quality schematic solutions for different problems. The next step is a real model of the final design to be recreated, tested and improved. How successful and helpful simulations are depends on the quality of the simulation models and there are not perfect simulation models. Different ones are good for different types of analyses. In this regard, the use of simulations in the education process allows students to develop good working habits and knowledge of how to analyze and use the models correctly. They allow the project-based education approach to be implemented which provides a successful mechanism to help students achieve high-level learning goals and deal with real problem-solving activities (Chen, W., & Zhang, F., 2016).

Simulations can be used in all parts of the education process. They allow almost all types of analyses to be done, but how realistic the results are depend on the specific parameters of the copy models. Their incorporation in the education process allows the students to find or develop the right simulation models and to use them properly in the design process afterwards. These skills allow them to do their homework and to do more of their own investigations.

Our current search led to the conclusion that the LTspice software created by Linear Technology Corporation® is a very powerful free tool with many capabilities. It seems to be the best free option out there. It has been found that this software is used by many professionals working in the field as a high-quality free alternative. The LTspice simulation software offers a possibility of conducting a thorough study on the functioning of an electronic system (Ptak, P., 2018). A study compared the free LTspice and the paid NI Multisim simulation results to those obtained in laboratory measurements and concluded that they perform similarly and the results are close to the real ones (Ptak, P., 2018). The main drawback of LTspice is that compared to certain leading paid programs its user interface is not so user-friendly and there is lack of ready to use models of certain frequently used electronic components, as transformers, potentiometers, and others. The software allows such models to be created with the help of other available ones for other consisting components, but this requires knowledge of the process. Fortunately, there is a lot of information online considering how to use the software properly and how to overcome its drawbacks, which requires work in the field in order the know-how to be acquired.

APPLICATION OF SIMULATIONS IN THE LECTURE COURSE OF THE SEMICONDUCTOR COMPONENTS SUBJECT

The Semiconductor components subject studies the structure, characteristics, parameters, functioning, regimes and the correct application of these components in electronic circuits. The work of the components is not clearly visible and often students hardly understand their functioning, the practical meaning of their characteristics and parameters, and hence their proper application. During the lectures, the simulations of the work principles of semiconductor components are used for demonstrational purposes. The use of simulation and virtual systems helps students to comprehend the concepts and principles of the lecture materials (Noga, K., & Palczynska, B., 2018). They allow students to see how the different parameters of the components are interrelated. For example, the dependance of the output current of bipolar transistor from the input one could be clearly presented through simulations. Another subsequent example is their usage for visualization of the superimposition of a variable signal carrying information over the input constant current of a bipolar transistor and presentation of its relation to the output amplified variable signal superimposed over the output constant current. They are also used for simple calculations and demonstrations related to setting a given operating mode of the different semiconductor components.

During lectures, the simulations can be used for improved understanding of the studied material through direct visual and practical representation of the relations. Any difficult to understand concept is easier to digest if it can be visualized through practical examples.

APPLICATION OF SIMULATIONS IN THE EXERCISE COURSE OF THE SEMICONDUCTOR COMPONENTS SUBJECT

Simulations are definitely useful in almost all exercises in many different aspects. During the introductory exercises for each separate component these steps are being followed in the presented sequence:

- First, the simulations are used for automatic capturing of the different component characteristics describing their way of work. Then they are explained and analyzed.
- Second, the datasheets of the example components are analyzed and their different parameter meanings are explained with examples. Some of the visible parameters are defined based on the captured by simulation characteristics.
- Different found spice models of the example components are compared based on their parameter adequacy compared to the basic real parameters visible from their datasheets.
- A simple project design for certain mode of operation is calculated and the results are demonstrated again through simulation of the example circuit based on the most adequate spice model. These exercises allow the relation between the different parameters, visible from the characteristics, to be demonstrated in real practical cases.
- Further more complicated simulation analyses could be done as AC analysis and others, but before each separate project the existent spice models for the example components are tested for adequacy compared to the original datasheet parameters. The most adequate model is used for the conduction of further design projects.

Widely available simulation models of an example bipolar transistor are presented in Table 1. Our experience is that usually the more complicated models, with more programing text, are more adequate for most of the possible simulation analyses, but this is not always the case. In this example for bipolar transistor 2N2222 the transition frequency f_T should be minimum 250 *MHz*, based on its original specifications, but for the first model it is about 33 *MHz*, for the 2nd model it is 330 *MHz*, for the 3rd one it is 62 *MHz* and for the 4th it is 270 *MHz*. In this case the 2nd and the 4th models are good for AC analysis and they could be further compared based on other parameters.

The other two models are going to return wrong unrealistic results. In this case, the more complicated 3rd model is not good for AC analysis.

Table 1. Example simulation models for bipolar transistor 2N2222

№	2N2222
1	.MODEL **Q2N2222A/ZTX** NPN IS =3.0611E-14 NF =1.00124 BF =220 IKF=0.52 + VAF=104 ISE=7.5E-15 NE =1.41 NR =1.005 BR =4 IKR=0.24 + VAR=28 ISC=1.06525E-11 NC =1.3728 RB =0.13 RE =0.22 + RC =0.12 CJC=9.12E-12 MJC=0.3508 VJC=0.4089 + CJE=27.01E-12 TF =0.325E-9 TR =100E-9
2	.MODEL **T2N2222** NPN (IS=2.20f NF=1.00 BF=240 VAF=114 + IKF=0.293 ISE=2.73p NE=2.00 BR=4.00 NR=1.00 + VAR=24.0 IKR=0.600 RE=0.194 RB=0.777 RC=77.7m + XTB=1.5 CJE=24.9p VJE=1.10 MJE=0.500 CJC=12.4p VJC=0.300 + MJC=0.300 TF=371p TR=64.0n EG=1.12)
3	.MODEL **TQ2n2222a** npn +IS=3.88184e-14 BF=929.846 NF=1.10496 VAF=16.5003 +IKF=0.019539 ISE=1.0168e-11 NE=1.94752 BR=48.4545 +NR=1.07004 VAR=40.538 IKR=0.19539 ISC=1.0168e-11 +NC=4 RB=0.1 IRB=0.1 RBM=0.1 +RE=0.0001 RC=0.426673 XTB=0.1 XTI=1 +EG=1.05 CJE=2.23677e-11 VJE=0.582701 MJE=0.63466 +TF=4.06711e-10 XTF=3.92912 VTF=17712.6 ITF=0.4334 +CJC=2.23943e-11 VJC=0.576146 MJC=0.632796 XCJC=1 +FC=0.170253 CJS=0 VJS=0.75 MJS=0.5 +TR=1e-07 PTF=0 KF=0 AF=1
4	.MODEL **TTQ2N2222A** NPN(IS=8.11E-14 BF=205 VAF=113 IKF=0.5 ISE=1.06E-11 + NE=2 BR=4 VAR=24 IKR=0.225 RB=1.37 RE=0.343 RC=0.137 CJE=2.95E-11 + TF=3.97E-10 CJC=1.52E-11 TR=8.5E-8 XTB=1.5)

Fig. 1 presents the simulation results for the 2nd simulation model with name T2N2222. The small signal current gain is over 60, which also corresponds to the original data sheet.

Fig. 2 presents the output characteristics for Common Emitter for the 2nd simulation model of 2N2222. They are captured automatically by the example circuit presented in Fig. 3. In order these characteristics to be drawn automatically the independent parameter value, *Uce* in this case, must linearly rise, the parameter for each different linear curve of the whole family, *Ib* in this case, must be stepped, and the relation of the dependent and independent parameters is visualized. This methodology could be used for automatic capturing of all 4 types of static characteristics of bipolar transistor. They are used for comparison of different parameters of the models to the actual ones, presented in the original datasheet, as well as for example calculations of circuits and simulation tests. For example, in the original datasheet, the Collector-Emitter saturation voltage should be no more than 0.4 *V* for output current of *Ic* = 150 *mA* and the simulation model returns a realistic value of 0.1 *V*, Fig. 2.

After the example component simulation characteristics are investigated in detail, the real ones are also measured and compared. The next logical step is a real circuit to be created and investigated with the help of a breadboard and other necessary equipment. These steps create a complete education cycle consisting of theory, simulation investigations and verifications of parameters, practical study of the component and final application project, which in this case is design of a circuit of a transistor operating in the different possible static modes for given operating points.

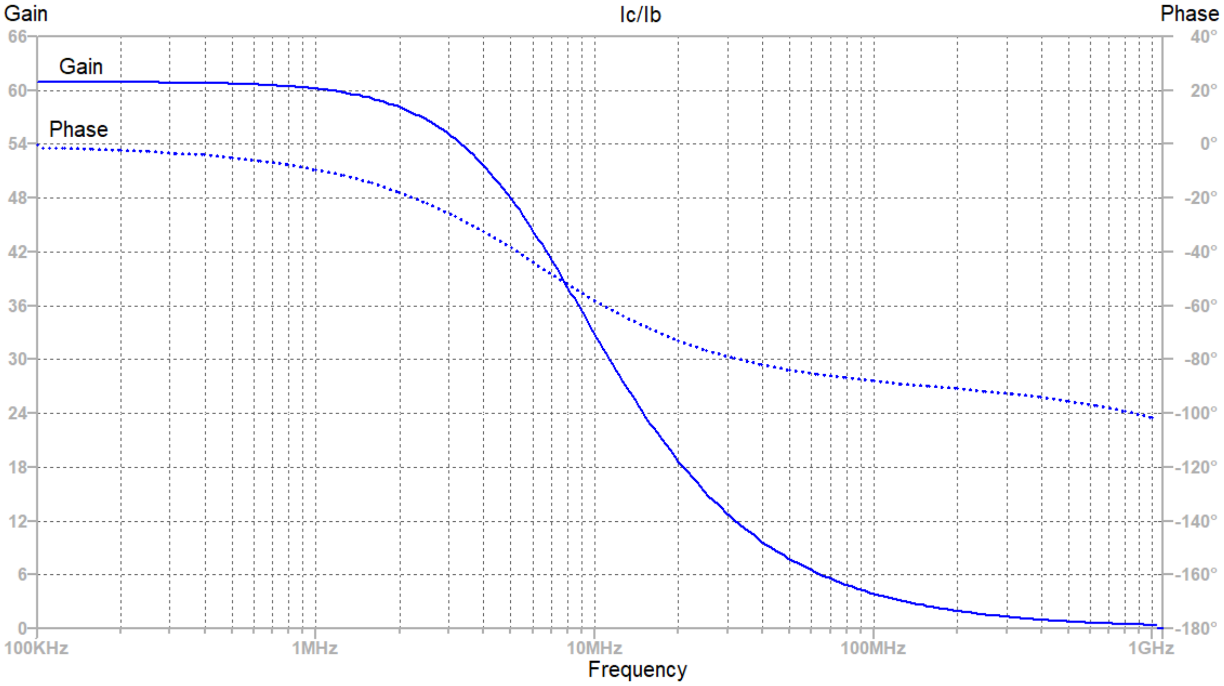

Fig. 1. AC analysis for the 2nd simulation model T2N2222

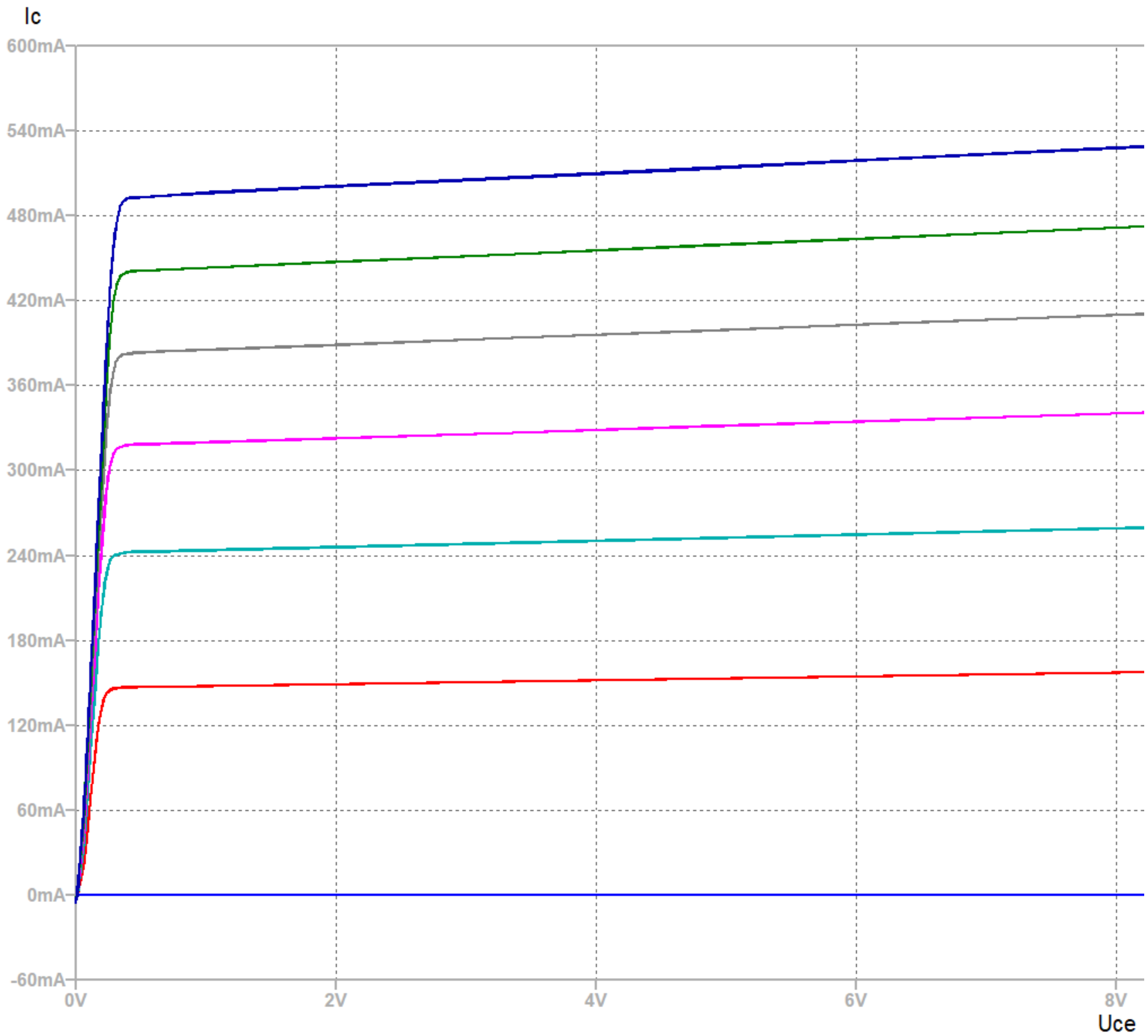

Fig. 2. Output characteristics for Common Emitter for the 2nd simulation model of 2N2222

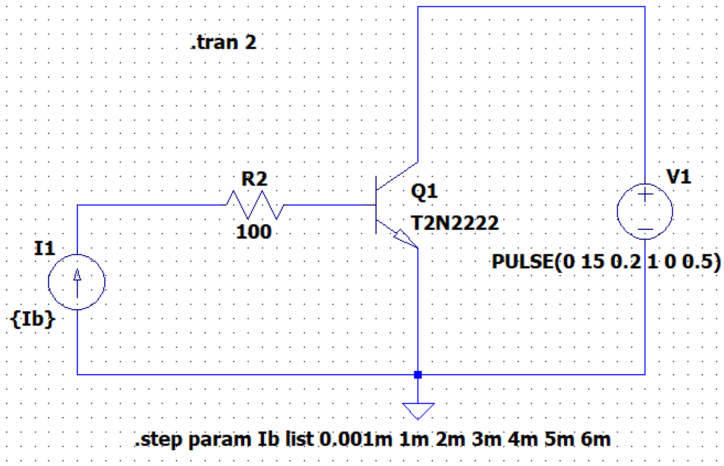

Fig. 3. Circuit for the capturing of the output static characteristics of the 2N2222 bipolar transistor

CONCLUSION

Nowadays, there are affordable softwares for analogue circuit simulations that bring realistic results, when used with realistic simulation models. There are also many widely available simulation models for many components and this creates opportunities for many education process improvements in the Electronics field. Simulation-based learning allows students to put theory into practice in a virtual laboratory environment (Barbarosou, M., Paraskevas, I., Kliros, G., & Andreatos, A., 2017). They give students the ability to see concepts, that they could otherwise not visualize (Dickerson, S., & Clark, R., 2018).

The applications of simulations in the electronics education lead to higher quality more understandable active-learning processes. A study results support active-learning as the preferred, empirically validated teaching practice in regular classrooms (Freeman, S., Eddy, S., McDonough, M., & al., 2014). They have been applied in the described manner for first year, and the students find it easier to understand the functioning, the parameters and the characteristics of the components in such a way, because they can see the results with their own eyes, and many different types of investigations could be easily done. The possibilities of their applications are almost endless. They allow students to do their own tests and designs in order to improve their understanding and skills. The regular usage of simulations creates also proper working habits in students because when done properly they could considerably accelerate the design process of a circuit and improve the overall quality. The simulations made the classes more practical and more interesting for students. The example investigations are always related to close to students' lives examples which further grasps their attention.

Because of the many benefits of the simulations, further work is done for development of additional interesting and fruitful methods for their application in all electronics related education subjects.

ACKNOWLEDGMENTS

The work presented in this paper is completed with the support of Project 2023 – FEEA – 03 "Simulation and experimental study on the methods and mechanisms for data confidentiality and data integrity in the modern local and wireless communication networks", financed under the Scientific and Research Fund of the University of Ruse "Angel Kanchev".

REFERENCES

Abramovitz, A. (2011). *Teaching Behavioral Modeling and Simulation Techniques for Power Electronics Courses*. IEEE Transactions on Education, vol. 54, no. 4, pp. 523-530.

Barbarosou, M., Paraskevas, I., Kliros, G., & Andreatos, A. (2017). *Implementing Transistor Roles for Facilitating Analysis and Synthesis of Analog Integrated Circuits.* 2017 IEEE Global Engineering Education Conference (EDUCON), Athens, Greece, pp. 423-430.

Ceccarelli, L., Kotecha, R., Iannuzzo, F., & Mantooth, A. (2018). *Fast Electro-Thermal Simulation Strategy for SiC MOSFETs Based on Power Loss Mapping.* 2018 IEEE International Power Electronics and Application Conference and Exposition (PEAC), Shenzhen, China, pp. 1-6, doi: 10.1109/PEAC.2018.8590288.

Chen, W., & Zhang, F. (2016). *A Project Based Approach to Teaching Microelectronics Circuit Analysis and Design.* Int. Journal Inform. and Education Technology (IJIET), vol. 6, No. 9, pp. 737-740.

Dickerson, S., & Clark, R. (2018). *A Classroom-Based Simulation-Centric Approach to Microelectronics Education.* Comput. Appl. Eng. Educ., pp. 1-14.

Freeman, S., Eddy, S., McDonough, M., & al. (2014). *Active Learning Increases Student Performance in Science, Engineering, and Mathematics.* Proc. Natl. Acad. Sci. 111, pp. 8410-8415.

Li, K., Evans, P., & Johnson, M. (2017). *Using Multi Time-Scale Electro-Thermal Simulation Approach to Evaluate SiC-MOSFET Power Converter in Virtual Prototyping Design Tool.* 2017 IEEE 18th Workshop on Control and Modeling for Power Electronics (COMPEL), Stanford, CA, USA, pp. 1-8, doi: 10.1109/COMPEL.2017.8013278.

Melis, T., Simeu, E., Auvray, E., & Armagnat, P. (2020). *Analog and Mixed-Signal Circuits Simulation for Product Level EMMI Analysis*. Microelectronics Reliability, vol. 114, 113881.

Nelson, B., & al. (2021). *Computational Efficiency Analysis of SiC MOSFET Models in SPICE: Dynamic Behavior.* IEEE Open Journal of Power Electronics, vol. 2, pp. 106-123, doi: 10.1109/OJPEL.2021.3056075.

Noga, K., & Palczynska, B. (2018). *The Simulation Laboratory Platform Based on Multisim for Electronic Engineering Education.* 2018 International Conference on Signals and Electronic Systems (ICSES), Kraków, Poland, pp. 269-274.

Sakairi, H., Yanagi, T., Otake, H., Kuroda, N., & Tanigawa, H. (2018). *Measurement Methodology for Accurate Modeling of SiC MOSFET Switching Behavior Over Wide Voltage and Current Ranges.* IEEE Transactions on Power Electronics, vol. 33, no. 9, pp. 7314-7325, doi: 10.1109/TPEL.2017.2764632.

Stupar, A., McRae, T., Vukadinović, N., Prodić, A., & Taylor, J. (2019). *Multi-Objective Optimization of Multi-Level DC–DC Converters Using Geometric Programming.* IEEE Transactions on Power Electronics, vol. 34, no. 12, pp. 11912-11939, doi: 10.1109/TPEL.2019.2908826.

Ptak, P. (2018). *Application of the Software Package LTspice for Designing and Analysing the Operation of Electronic Systems*. Society. Integration. Education Proceedings of the International Scientific Conference, vol. V, 25-26 May 2018, pp. 402-408.

Ptak, P. (2018). *Application of Multisim and LTspice Software Packages to Simulate the Operation of Electronic Components as an Alternative to Measurements of Real Elements*. Society. Integration. Education Proceedings of the International Scientific Conference, vol. V, 25-26 May 2018, pp. 409-419.