GDAŃSK UNIVERSITY OF TECHNOLOGY

Subject card

Subject name and code	Electric Circuits, PG_00050028		
Field of study	Electrical Engineering		
Date of commencement of studies	October 2024	Academic year of realisation of subject	2024/2025
Education level	second-cycle studies	Subject group	Obligatory subject group in the field of study Subject group related to scientific research in the field of study
Mode of study	Part-time studies	Mode of delivery	at the university
Year of study	1	Language of instruction	Polish
Semester of study	1	ECTS credits	2.0
Learning profile	general academic profile	Assessment form	exam
Conducting unit	Faculty of Electrical and Control Engineering		
Name and surname of lecturer (lecturers)	Subject supervisor	dr hab. inż. Jacek Horiszny	
	Teachers	dr hab. inż. Jacek Horiszny	

Lesson types and methods of instruction

Lesson type	Lecture	Tutorial	Laboratory	Project	Seminar	SUM
Number of study hours	10.0	0.0	10.0	0.0	0.0	20

E-learning hours included: 0.0

Learning activity and number of study hours

Learning activity	Participation in didactic classes included in study plan	Participation in consultation hours	Self-study	SUM
Number of study hours	20	4.0	26.0	50

Subject objectives

Mastering the skill of a comprehensive analysis of electric circuits using computer tools - the program PSpice. Acquiring knowledge and skills in the field of methods of analysis of transient states in electric circuits.

Learning outcomes

Course outcome	Subject outcome	Method of verification
[K7_W01] has an extended and deepened knowledge of mathematics, including selected issues of numerical methods and knowledge useful for solving tasks in the field of electrotechnology and electrodynamics, has a general knowledge of technical sciences covering their fundamentals and applications	Student can determine the nature of changes of basic electrical parameters in the transient state and determine the maximum values of these parameters. Can formulate equations to solve the transient state in the electrical circuit, eg. using a Laplace transform.	[SW1] Assessment of factual knowledge
[K7_U06] is able to analyse, model, simulate and design electrical systems	Student calculates transient state in electrical circuit eg using Laplace transform. Creates a mathematical model of the circuit in the PSpice program and conducts an analysis of the phenomena occurring in this circuit - frequency and time domain analysis.	[SU4] Assessment of ability to use methods and tools
[K7_K02] is aware of the impact of engineering activities on the environment, understands the the non-technical effects of those activities	Student is aware of the environmental impact of engineering activities, understands the non-technical effects of this activity.	[SK5] Assessment of ability to solve problems that arise in practice

Subject contents

Transients in electrical circuits. Commutation law. Formulation of equations. Initial conditions. Examples of analytical evaluation of current and voltage waveforms of circuit in transient state. Application of Laplace transform. Equivalent circuit in Laplace transformation. Basic information about PSpice software environment: basic modules of software components, creating a model of the circiut, basic modes of circuit analysis. Simulation of AC circuits in transient and quasi-steady states. Simulation analysis of transients in electrical systems, including in particular power electronic converters: selection of calculation parameters of the algorithm, modeling of the inverter gate signals, observation of typical transient phenomena, the selection of system components to limit overvoltage and overcurrent.

<table>
<tr><td>Prerequisites and co-requisites</td><td colspan="3">Knowledge of electrical circuits in the first degree level course in electrical engineering.</td></tr>
<tr><td>Assessment methods and criteria</td><td colspan="3"><table>
<tr><th>Subject passing criteria</th><th>Passing threshold</th><th>Percentage of the final grade</th></tr>
<tr><td>Written and practical exam</td><td>50.0%</td><td>70.0%</td></tr>
<tr><td>Homeworks during the semester</td><td>50.0%</td><td>24.0%</td></tr>
<tr><td>Results of own work during the tutorials</td><td>50.0%</td><td>6.0%</td></tr>
</table></td></tr>
<tr><td rowspan="3">Recommended reading</td><td>Basic literature</td><td colspan="2">1. Bolkowski S.: Teoria obwodów elektrycznych. WNT Warszawa 2012.
2. Osiowski J., Szabatin J.: Podstawy teorii obwodów elektrycznych. WNT Warszawa 1998.
3. Zimny P., Karwowski K.: Spice – klucz do elektrotechniki. Wydawnictwo Politechniki Gdańskiej. Gdańsk 2001.
4. Król A., Moczko J.: PSpice - Symulacja i optymalizacja układów elektronicznych. Wyd. Nakom. Poznań 2000.
5. Dobrowolski A.: Pod maską SPICE'a. Metody i algorytmy analizy układów elektronicznych. Wydawnictwo BTC. Warszawa 2004.
6. Wojtuszkiewicz K., Zachara Z.: PSpice. Przykłady praktyczne. Wyd. Mikom, Listopad 2000.</td></tr>
<tr><td>Supplementary literature</td><td colspan="2">1. Chua L.O., Pen-Min Lin:Komputerowa analiza układów elektronicznych. WNT Warszawa 1981.
2. Izydorczyk J.: PSpice. Komputerowa symulacja układów elektronicznych. Wydawnictwo Helion. Warszawa 1993.
3. Porębski J., Korohoda P.: Spice. Program analizy nieliniowej układów elektronicznych. WNT Warszawa 1994.</td></tr>
<tr><td>eResources addresses</td><td colspan="2">Adresy na platformie eNauczanie:
OBWODY ELEKTRYCZNE [ET][Niestacjonarne][2024/25] - Moodle ID: 39872
https://enauczanie.pg.edu.pl/moodle/course/view.php?id=39872</td></tr>
<tr><td>Example issues/ example questions/ tasks being completed</td><td colspan="3">Analytical calculation of current and voltage waveforms in a transient state in an exemplary linear circuit with DC excitations. Analytical calculation of current and voltage waveforms in a transient state in a selected simple linear circuit with sinusoidal excitations. Analytical calculation of the initial conditions in a transient state in a complex linear circuit with sinusoidal excitations, determination of the type of response. Analytical solution of a simple linear circuit with excitations other than DC or sinusoidal in a transient state. Simulation of transients using program PSpice in a complex linear circuit with excitation defined by function of a certain type. Simulation of transients using program PSpice in a complex circuit containing semiconductors devices, with the excitations defined by function of a certain type.</td></tr>
<tr><td>Work placement</td><td colspan="3">Not applicable</td></tr>
</table>

Document generated electronically. Does not require a seal or signature.