



## 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.						

Prerequisites and co-requisites	Knowledge of electrical circuits in the first degree level course in electrical engineering.		
Assessment methods and criteria	Subject passing criteria	Passing threshold	Percentage of the final grade
	Written and practical exam	50.0%	70.0%
	Homeworks during the semester	50.0%	24.0%
	Results of own work during the tutorials	50.0%	6.0%
Recommended reading	Basic literature	<ol style="list-style-type: none"><li>1. Bolkowski S.: Teoria obwodów elektrycznych. WNT Warszawa 2012.</li><li>2. Osiowski J., Szabatin J.: Podstawy teorii obwodów elektrycznych. WNT Warszawa 1998.</li><li>3. Zimny P., Karwowski K.: Spice – klucz do elektrotechniki. Wydawnictwo Politechniki Gdańskiej. Gdańsk 2001.</li><li>4. Król A., Moczko J.: PSpice - Symulacja i optymalizacja układów elektronicznych. Wyd. Nakom. Poznań 2000.</li><li>5. Dobrowolski A.: Pod maską SPICE'a. Metody i algorytmy analizy układów elektronicznych. Wydawnictwo BTC. Warszawa 2004.</li><li>6. Wojtuszkiewicz K., Zachara Z.: PSpice. Przykłady praktyczne. Wyd. Mikom, Listopad 2000.</li></ol>	
	Supplementary literature	<ol style="list-style-type: none"><li>1. Chua L.O., Pen-Min Lin: Komputerowa analiza układów elektronicznych. WNT Warszawa 1981.</li><li>2. Izydorczyk J.: PSpice. Komputerowa symulacja układów elektronicznych. Wydawnictwo Helion. Warszawa 1993.</li><li>3. Porębski J., Korohoda P.: Spice. Program analizy nieliniowej układów elektronicznych. WNT Warszawa 1994.</li></ol>	
	eResources addresses	Adresy na platformie eNauczanie: OBWODY ELEKTRYCZNE [ET][Niestacjonarne][2024/25] - Moodle ID: 39872 <a href="https://enauczanie.pg.edu.pl/moodle/course/view.php?id=39872">https://enauczanie.pg.edu.pl/moodle/course/view.php?id=39872</a>	
Example issues/ example questions/ tasks being completed	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 semiconductor devices, with the excitations defined by function of a certain type.		
Work placement	Not applicable		

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