



Subject card

Subject name and code	Application of PSpice and FEMM programs in engineering simulations of electrical circuits and systems, PG_00053432						
Field of study	Electrical Engineering						
Date of commencement of studies	October 2020		Academic year of realisation of subject		2023/2024		
Education level	first-cycle studies		Subject group				
Mode of study	Part-time studies		Mode of delivery		at the university		
Year of study	4		Language of instruction		Polish		
Semester of study	7		ECTS credits		2.0		
Learning profile	general academic profile		Assessment form		assessment		
Conducting unit	Katedra Elektrotechniki i Inżynierii Wysokich Napięć -> Faculty of Electrical and Control Engineering						
Name and surname of lecturer (lecturers)	Subject supervisor		dr inż. Adam Młyński				
	Teachers		dr inż. Mikołaj Nowak				
Lesson types and methods of instruction	Lesson type	Lecture	Tutorial	Laboratory	Project	Seminar	SUM
	Number of study hours	15.0	0.0	15.0	0.0	0.0	30
	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	30		2.0		18.0	50
Subject objectives	The aim of the course is to acquire the skills to use software packages (PSpice, Femm) and to create circuits and systems needed for the analysis and determination of computer simulation parameters.						

Learning outcomes	Course outcome	Subject outcome	Method of verification
	K6_U09	The student is able to create simple simulation models of electrical devices	[SU4] Assessment of ability to use methods and tools
	K6_U10	The student is able to design simulation models of simple electrical and electronic devices and systems.	[SU4] Assessment of ability to use methods and tools [SU3] Assessment of ability to use knowledge gained from the subject
	K6_U05	The student is able to design simulation models of simple electrical and electronic devices and systems.	[SU4] Assessment of ability to use methods and tools
	K6_K05	The student is able to operate advanced simulation software and use computer equipment.	[SK5] Assessment of ability to solve problems that arise in practice [SK3] Assessment of ability to organize work
	K6_W11	The student is able to design simulation models of simple electrical and electronic devices and systems.	[SW1] Assessment of factual knowledge
	K6_W10	The student is able to design simulation models of electrical, electronic and power electronic systems as well as electrical machines, and is also able to use these models and draw conclusions regarding the construction and operation of real objects.	[SW3] Assessment of knowledge contained in written work and projects [SW1] Assessment of factual knowledge
	K6_W09	The student is able to design simulation models of electrical, electronic and power electronic systems as well as electrical machines.	[SW1] Assessment of factual knowledge [SW3] Assessment of knowledge contained in written work and projects
	K6_K01	The student is able to use various computer programs for engineering calculations, finds their updates. The student can search for information on the Internet and in literature.	[SK5] Assessment of ability to solve problems that arise in practice [SK3] Assessment of ability to organize work
Subject contents	<ol style="list-style-type: none"> 1. Introduction to the PSpice environment. 2. Simulations of DC circuits in PSpice (parametric, sensitivity, statistical analysis). 3. Simulation of AC circuits in PSpice. 4. Simulations of AC circuits in PSpice (complex circuits), resonance phenomena, filters, magnetic couplings. 5. Transient simulations in PSpice time domain analysis (transient analysis). 6. Transient simulations in PSpice time domain analysis (transient analysis). 7. Simulations of power electronics systems in PSpiceCircuits with distributed parameters. 8. Lines long analysis of phenomena in PSpice 9. FEMM program for the analysis of electric and magnetic fields 10. FEMM program for the analysis of electric and magnetic fields 		
Prerequisites and co-requisites	The ability to analyze electric circuits and describe electromagnetic fields		
Assessment methods and criteria	Subject passing criteria	Passing threshold	Percentage of the final grade
	Passing the laboratory	60.0%	60.0%
	Examination of theory	60.0%	40.0%
Recommended reading	Basic literature	<ol style="list-style-type: none"> 1. FEMM User Guide 2. PSpice User Guide 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. Izydorczyk J.: Pspice. Komputerowa symulacja układów elektronicznych. Wydawnictwo Helion. Gliwice 1993 6. Porębski J., Korohoda P.: Spice. Program analizy nieliniowej układów elektronicznych. WNT Warszawa 1994 	

	Supplementary literature	<ol style="list-style-type: none"> 1. Dobrowolski A.: Pod maską SPICE'a. Metody i algorytmy analizy układów elektronicznych. Wydawnictwo BTC. Warszawa 2004 2. Wojtuszkiewicz K., Zachara Z.: PSpice. Przykłady praktyczne. Wyd. Mikom, Warszawa, Listopad 2000 3. Griffiths D.J.: Podstawy elektrodynamiki. PWN Warszawa 2001 4. Krakowski M: <i>Elektrotechnika teoretyczna. , tom 1 i 2.</i> PWN, Warszawa 1995 5. Piątek Z., Jabłoński P.: Podstawy teorii pola elektromagnetycznego. WNT, Warszawa 2010 6. M. O. Sadiku Elements of Electromagnetics
	eResources addresses	Adresy na platformie eNauczanie:
Example issues/ example questions/ tasks being completed	<ul style="list-style-type: none"> • Analyze the electric circuit in a transient state. • Perform simulation of electromagnetic field distribution in FEMM software • Define the parameters of the computer simulation in PSpice • Determine the parameters of the computer simulation in FEMM 	
Work placement	Not applicable	