STUDY MODULE DESCRIPTION FORM

Name of the module/subject **Computer Aided Design**		Code **1010811161010833605**
Field of study **Electronics and Telecommunications**	Profile of study (general academic, practical) **general academic**	Year /Semester **3 / 6**
Elective path/specialty **Radio Communications**	Subject offered in: **Polish**	Course (compulsory, elective) **elective**
Cycle of study: **First-cycle studies**	Form of study (full-time,part-time) **full-time**	
No. of hours Lecture: **1** Classes: **-** Laboratory: **2** Project/seminars: **-**		No. of credits **3**
Status of the course in the study program (Basic, major, other) **other**	(university-wide, from another field) **university-wide**	
Education areas and fields of science and art **technical sciences** **Technical sciences**		ECTS distribution (number and %) **3 100%** **3 100%**

Responsible for subject / lecturer:

dr inż. Sławomir Michalak
email: michalak@et.put.poznan.pl
tel. +48 616653824
Faculty of Electronics and Telecommunications
ul. Piotrowo 3A 60-965 Poznań

Prerequisites in terms of knowledge, skills and social competencies:

1	**Knowledge**	Has a basic knowledge of the fundamentals of circuit theory, together with necessary mathematical background; this knowledge allows him/her to understand, analyze and evaluate the operation of electrical circuits. Has a basic knowledge about basic electronic elements and theirs characteristics. Have very basic knowledge about measurements and metrology.
2	**Skills**	Is able to extract information from Polish or English language literature, databases and other sources. Is able to use known mathematical analysis, algebra and theory to solve basic problems in electronics.
3	**Social competencies**	Is aware of the limitations of his knowledge and skills; is committed to further self-study. Is active in solving technical electronics problems. Is able to consulting in group.

Assumptions and objectives of the course:

Computer Aided Design in electronics. SPICE - a general-purpose circuit simulation program for nonlinear DC, nonlinear transient, and linear AC analyses. Models of decices: resistors, capacitors, inductors, independent and dependent voltage and current

sources, switches, the most common semiconductor devices: diodes, BJTs, JFETs,

MESFETs, and MOSFETs.

Study outcomes and reference to the educational results for a field of study

Knowledge:

1. Has a wide, systematic knowledge of the properties and characteristics of electronic components, as well as of construction, analysis and design of electronic circuits. - [K1_W08]

2. Knows the theoretical foundations and principles of design of digital circuits, and of construction of digital electronic elements; knows the theoretical foundations of analysis and design of digital circuits and CAD. - [K1_W12]

Skills:

1. Is able to analyze, design and build digital circuits , using appropriate methods and engineering tools, and taking into consideration predefined criteria. Is able to use models, catalogue cards and application notes of semiconductor electronic elements. Is able to analyze and design circuits and systems using CAD. - [K1_U18]
2. Is able to extract information from Polish or English language literature, databases and other sources. Is able to synthesize gathered information, draw conclusions, and justify opinions. - [K1_U01]
3. Is able to communicate in English or in Polish in the professional environment and other environments. - [K1_U02]
4. Is capable of studying autonomously. - [K1_U05]

Social competencies:

1. Demonstrates responsibility for designed electronic and telecommunication systems. Is aware of the hazards they pose for individuals and communities if they are improperly designed or produced. - [K1_K03]
2. Is aware of the limitations of his/her current knowledge and skills; is committed to further self-study. - [K1_K01]

Assessment methods of study outcomes

1. Projects
2. Reports from laboratory execercises
3. Activity durning labs

Course description

- Basic analyes: DC, AC, Transient and FFT analysies.
- Parametric analysie.
- Temperature analysie.
- Worst Case and Monte Carlo analysies.
- Noise analysie.
- Models of basic electronic passive devices (resistor, capacitor, inductor) used in CAD programms.
- Models of active elements (dioda, Zener dioda, bipolar transitor, unipolar transistor).
- Models of voltage and current sources (DC, AC, SIN, PULSE, EXP)
- Models i macromodels OpAmp.
- Models of devices used in SPICE and APLAC.

Basic bibliography:

1. Baranowski K., Matuszczyk M., Welo A., Symulacja układów elektronicznych: PSpice pakiet DESIGN CENTER, MIKOM, Warszawa, 1996.
2. Dobrowolski A., Pod maską Spicea. Metody i algorytmy analizy układów elektronicznych, BTC, 2004.
3. Michalak S., Symulacja układów elektronicznych w środowisku APLAC, Wydawnictwo PP, Poznań, 2005.

Additional bibliography:

1. Porębski J. Korohoda P., SPICE program analizy nieliniowych układów elektronicznych, WNT, Warszawa, 1996.
2. Zachara Z., Wojtuszkiewicz K., PSpice: symulacje wzmacniaczy dyskretnych, MIKOM, Warszawa, 2001.
3. Sidor T., Komputerowa analiza elektronicznych układów pomiarowych, Kraków, Wydawnictwo AGH, 2006.
4. Walczak J., Pasko M., Komputerowa analiza obwodów elektrycznych z wykorzystaniem programu SPICE: zagadnienia podstawowe, Wydawnictwo Politechniki Śląskiej, Gliwice, 2002.

Result of average student's workload

Activity	Time (working hours)
1. Lectures	15
2. Labs	30
3. Reports	30
4. Project	20

Student's workload

Source of workload	hours	ECTS
Total workload	85	3
Contact hours	50	2
Practical activities	52	2