COURSE DESCRIPTION CARD - SYLLABUS

Course name
Computer Aided Design [S1EiT1E>KWP]

Course

Field of study
Electronics and Telecommunications

Year/Semester
3/5

Area of study (specialization)
–

Profile of study
general academic

Level of study
first-cycle

Course offered in
english

Form of study
full-time

Requirements
elective

Number of hours

Lecture
15

Laboratory classes
30

Other (e.g. online)
0

Tutorials
0

Projects/seminars
0

Number of credit points

3,00

Coordinators

dr inż. Sławomir Michalak
slawomir.michalak@put.poznan.pl

Lecturers

Prerequisites

The student has basic knowledge of circuit theory and electrical metrology. Knows the basic symbols of electronic components. Has knowledge of the operation of basic electronic components and their characteristics. Demonstrates knowledge of basic electronic circuits. He knows the principles of operation and can design basic electronic circuits. Can use the catalog data of electronic components and systems. Uses the computer to perform the assumed tasks. Demonstrates the ability to obtain information (catalog data) on the Internet. Capable of independent learning (textbooks, computer programs). Behaves actively in class, puts questions, consciously uses contacts with the teacher (e.g. as part of consultations).

Course objective

To provide students with knowledge of the basics and tools of computer-aided design of electronic systems (CAD), the role of CAD in individual stages of designing electronic devices. Developing students' skills in creating schematic diagrams using CAD tools (e.g. LTSPICE, MULTISIM and APLAC), performing basic analyzes (DC, frequency, time) and extended analyzes (temperature, parametric, FFT, noise, Worst Case, Monte Carlo). Acquainting with models of elements, issues of simulation of analog, digital and analog-digital circuits, optimization of the parameters of the designed system.

Course-related learning outcomes

Knowledge:
Has ordered and wide knowledge of the properties and characteristics of electronic components, in the field of building models of electronic components, designing and analyzing electronic circuits, as well as designing printed circuits. Knows the theoretical foundations and principles of designing analog and digital circuits, building digital electronic components as well as analyzing and designing electronic circuits, computer aided design.

Skills:
Has the ability to analyze, design and simulate the operation of analog and digital circuits taking into account given criteria, using appropriate engineering methods and tools, is able to use models, catalog cards and application notes of semiconductor electronic components, is able to analyze and design systems and systems using CAD tools. Is able to obtain information from literature and databases as well as other sources in English; is able to integrate obtained information, interpret it, draw conclusions and justify opinions. Is able to communicate in English in a professional environment. He can continue to educate himself.

Social competences:
Has a sense of responsibility for the designed electronic and telecommunications systems and is aware of the potential dangers to other people or society in the event of improper use

Methods for verifying learning outcomes and assessment criteria

Learning outcomes presented above are verified as follows:

Knowledge acquired as part of the lecture, supplemented with practical skills acquired during laboratory classes, is verified by self-completion and passing a semester work (simulation of the operation of a given electronic system). Individual topics of final essays are given on 6/7 lectures. Students return their semester work (report in paper form and simulation files - sent by e-mail using the university e-mail system), receive note in accordance with the date of the final session.
The ability to use the knowledge acquired during lectures, correctness of simulation performance, correctness of selection and scope of analyzes according to the nature of the system, ability to modify the used element models are assessed. The scope of tasks of the semester work (degree of difficulty and labor intensity) is graded (satisfactory, good and very good). Students have the right to choose the range (minimum - satisfactory, maximum - very good). After issuing the grade, until it is approved in the e-proto system, students also have the option of individual consultation and verification of the grade (oral answer).
Laboratory classes are awarded on the basis of the student's report (in writing). The report is made after each laboratory unit (performing the assigned exercise).
The semester grade from the laboratory is determined on the basis of all reports (arithmetic mean value). The correctness and scope of the simulation are assessed (mandatory and additional tasks). Students have the option of individual consultation, verification of the grade (oral answer or additional tasks) and obtaining a higher grade.

Programme content

• Models of basic passive electronic components (resistor, capacitor, coil) used in simulation programs.
• Models of active elements (diode, Zener diode, bipolar, unipolar transistor).
• Models of voltage and current sources (DC, AC, SIN, PULSE, EXP), controlled sources.
• Models and macromodels of operational amplifiers.
• Comparison of element models used in SPICE (LTSpice, Multisim) and APLAC programs.
• DC analysis.
• AC frequency analysis.
• Time analysis and FFT.
• Parametric analysis.
• Temperature analysis.
• Statistical analysis (Worst Case, Monte Carlo).
• Noise analysis.
• Optimization.

Teaching methods

1. Lecture: traditional lecture; multimedia presentation, illustrated with examples of simulation problems.
2. Laboratory exercises: practical exercises at computer stations, performing simulation tasks given by the teacher, supported by examples of solutions (teahers' multimedia presentations).

Bibliography

Basic:
1. Sandler M.S., "SPICE Circuit Handbook", McGraw-Hill Education - Europe, 2006.
2. El Emam, K., Drouin, J., Melo, W., Dorling, A. "SPICE : the theory and practice of software process improvement and capability determination" , IEEE Computer Society, 1998.
2. Moscovici, A. "High speed A/D converters: understanding data converters through SPICE", Kluwer Academic Publischers, 2001.

Additional:
1. Nagel, L. W., and Pederson, D. O., "SPICE (Simulation Program with Integrated Circuit Emphasis)", Memorandum No. ERL-M382, University of California, Berkeley, Apr. 1973.
2. Shah, M.M., "Design of Electronic Circuits and Computer Aided Design", Halsted Press, 1993.
3. Tadej, T., Árpád B., "Circuit Simulation with SPICE OPUS THeory and Practice". Birkhäuser Press. 2009.

Breakdown of average student's workload

	Hours	ECTS
Total workload	90	3,00
Classes requiring direct contact with the teacher	55	2,00
Student's own work (literature studies, preparation for laboratory classes/ tutorials, preparation for tests/exam, project preparation)	35	1,00