COURSE DESCRIPTION CARD - SYLLABUS

Course name
Printed circuit design [S1EiT1>POD]

Course

Field of study
Electronics and Telecommunications

Year/Semester
3/6

Area of study (specialization)
–

Profile of study
general academic

Level of study
first-cycle

Course offered in
polish

Form of study
full-time

Requirements
elective

Number of hours

Lecture
15

Laboratory classes
15

Other (e.g. online)
0

Tutorials
0

Projects/seminars
0

Number of credit points

3,00

Coordinators

dr hab. inż. Maciej Wawrzyniak
maciej.wawrzyniak@put.poznan.pl

Lecturers

Prerequisites

A student has basic knowledge in mathematics, physics, and fundamentals of circuit theory and electronics. Is able to extract information from literature, databases, and other sources. Is able to participate in collaborative projects.

Course objective

To acquaint students with the design and production of printed circuit boards. Presentation of standards introducing the rules for the design and production of PCBs. Digest of the design principles to reduce electromagnetic interference.

Course-related learning outcomes

Knowledge:
1. A student has knowledge of the construction, description, design, and manufacture of printed circuit boards.
2. Knows the standards introducing the rules for the design and production of PCBs.
3. Has knowledge of methods to reduce the level of electromagnetic interference in printed circuit boards.

Skills:
1. A student can obtain information from the literature and other sources and can integrate the information obtained, interpret it, draw conclusions, and justify opinions.
2. Can prepare well-documented assumptions for the project of a printed circuit board.
3. Can use catalogs to select appropriate electronic components taking into account the given criteria.
4. Can design a simple PCB using the appropriate engineering methods and tools.

Social competencies:
1. A student is aware of the need for a professional approach to solving technical problems and assumes responsibility for the proposed technical solutions.
2. Can work in a group in the laboratory and perform team tasks.
3. Can formulate opinions on the basic challenges faced by modern electronic devices technology.

Methods for verifying learning outcomes and assessment criteria

Learning outcomes presented above are verified as follows:

Passing the lecture on the basis of one written and/or oral test on the content of the lectures. The written test contains 12 open questions. The oral test contains 4-6 questions. Passing threshold 50% of the sum of points for the test. Questions will be prepared on the basis of slides published in the eKursy system. Grading scale: <50% - 2.0 (ndst); 50% to 59% - 3.0 (dst); 60% to 69% - 3.5 (dst +); 70% to 79% - 4.0 (db); 80% to 89% - 4.5 (db +); 90% to 100% - 5.0 (bdb). The passing threshold may change depending on the results of the tests.

Laboratory passing based on grades for reports, preparation for classes, behaviour and commitment during classes, and one written and/or oral test. The written test consists of solving 8 tasks, scored differently. The oral test consists of solving 4-6 tasks with different points, assigned by the lecturer. The final grade is based on the weighted average: Sw=0.45*SO+0.55*OzK where: SO is the average of the grades obtained for the preparation of reports, preparation for laboratory exercises, behavior and participation in the laboratory and OzK is the grade from the test. Grading scale: Sw>4,75 - 5,0 (bdb); 4,25=4,75 - 4,5 (db+); 3,75=4,25 - 4,0 (db); 3,25=3,75 - 3,5 (dst+); 2,75=3,25 - 3,0 (dst); Sw<=2,75 - 2,0 (ndst) where Sw – the weighted arithmetic mean of all partial grades.

Programme content

Lecture
Production stages of making a printed circuit board, rules for drawing a block diagram, schematic diagram, graphic symbols used in diagrams, international standard IEC 60617, Polish standard PN - EN 60617/2003, standard ANSI Y32 / IEEE 315, rules for the correct electronic component placement, coding system of package outlines for semiconductor device packages - standard IEC 60191-4, computer methods of PCBs design, metric and imperial basic raster, technologies of printed circuit boards , soldering technologies, testing of the printed circuit boards.

Standardization of the design and production of printed circuit boards: SMT technology, THT technology, IPC density levels, what is a "footprint"?, pads for surface mount - layers and their purpose, placement courtyard, silkscreen outlines, assembly drawing outlines, pads for through-hole assembly - layers and their purpose, the diameter of drilled holes diameter, plane thermal relief, plane anti-pad, solder mask, padstack naming convention, default values and modifiers.

Surface mount pad pattern standard, tolerances and solder joint analysis, solder joint tolerance, component lead space tolerance, pad pattern pad length tolerance, rules for rounding pad dimensions and their position coordinates, creating a pad array for a given integrated circuit package, standard SMD packages, naming convention for standard SMT pad patterns, soldering quality control, BGA, LGA, CGA, and PCGA packages.

The hierarchy of IPC 'design specifications', standardization of routing, component placement, determination of trace impedance and capacitance, multilayer boards, reference layers, component and assembly issues, basic types of transmission line constructions: microstrip, embedded microstrip, symetric stripline and dual (asymetric) stripline, propagational speed of a signal within the transmission line, propagation delay of a signal, impedance control, impedance matching - elimination of reflections, 1/3 rise time rule, electrical conductor spacing, IPC recommended track widths.

Design of a printed circuit board for electromagnetic interference (EMI) suppression, electromagnetic compatibility (EMC), simple EMI model, methods of removing interferences, component characteristics at RF frequencies, return path for RF current, single-layer PCBs: arrangement of components, routing of ground and power paths, principles of designing double-layer boards, the effect of interference emission from the path, I and II Maxwell's equation in differential form, concept of magnetic flux cancellation, signal path and return path , right-hand rule, return path in multilayer boards, RLC circuits series and parallel resonance, capacitor equivalent diagram for high frequencies, types of capacitors and their parameters, frequency characteristics of SMD capacitors, decoupling and bypassing capacitors, resonances of parallel capacitors, bulk capacitors, rules for selecting bulk capacitors, high-speed circuit board signal integrity, signal reflections, reflection coefficient, multiple signal reflection (ringing), serial termination (source termination), parallel termination, RC termination, Thevenin termination, preventing cross-talk, current density distribution from trace to reference, plane 3-W rule, shielding paths.

Laboratory
Production stages of making a printed circuit board, rules for drawing a block diagram, schematic diagram, graphic symbols used in diagrams, international standard IEC 60617, Polish standard PN - EN 60617/2003, standard ANSI Y32 / IEEE 315, rules for the correct electronic component placement, coding system of package outlines for semiconductor device packages - standard IEC 60191-4, computer methods of PCBs design, metric and imperial basic raster, technologies of printed circuit boards , soldering technologies, testing of the printed circuit boards.

Standardization of the design and production of printed circuit boards: SMT technology, THT technology, IPC density levels, what is a "footprint"?, pads for surface mount - layers and their purpose, placement courtyard, silkscreen outlines, assembly drawing outlines, pads for through-hole assembly - layers and their purpose, the diameter of drilled holes diameter, plane thermal relief, plane anti-pad, solder mask, padstack naming convention. Impedance control, impedance matching - elimination of reflections, 1/3 rise time rule, electrical conductor spacing, IPC recommended track widths, design of a printed circuit board for electromagnetic interference (EMI) suppression, electromagnetic compatibility (EMC), simple EMI model, methods of removing interferences, decoupling and bypassing capacitors, resonances of parallel capacitors, bulk capacitors, rules for selecting bulk capacitors, high-speed circuit board signal integrity, signal reflections, reflection coefficient, multiple signal reflection (ringing), serial termination (source termination), parallel termination, RC termination, Thevenin termination, preventing cross-talk, shielding paths.

Teaching methods

Lecture: traditional multimedia presentation (examples also on the blackboard) and conversational lecture.
Lab: traditional multimedia presentation (examples also on the blackboard) and performance of tasks given by the teacher - practical exercises.

Bibliography

Basic
1. Horowitz P., Hill W., Sztuka elektroniki, cz. 1 i 2, WKiŁ, Warszawa 2009.
2. Kisiel R., Podstawy technologii dla elektroników, Poradnik praktyczny, Wydawnictwo BTC, 2005.
3. Pease R. A., Projektowanie układów analogowych : poradnik praktyczny, Wydawnictwo BTC, 2005.
4. Kisiel R., Podstawy technologii montażu dla elektroników, Wydawnictwo BTC, 2012.
Additional
1. Rymarski Z., Materiałoznawstwo i konstrukcja urządzeń elektronicznych, Wydawnictwo Politechniki Śląskiej, Gliwice 2000.
2. Thierauf S. C., High-speed circuit board signal integrity. Artech House, 2017.
3. Montrose M. I., EMC and the printed circuit board: design, theory, and layout made simple, John Wiley & Sons, 2004.

Breakdown of average student's workload

	Hours	ECTS
Total workload	75	3,00
Classes requiring direct contact with the teacher	31	2,00
Student's own work (literature studies, preparation for laboratory classes/ tutorials, preparation for tests/exam, project preparation)	44	1,00