COURSE DESCRIPTION CARD - SYLLABUS

Course name
Printed circuit board design [S1MiKC1>POD]

Course

Field of study
Microelectronics and digital communications

Year/Semester
4/7

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
30

Laboratory classes
15

Other
0

Tutorials
0

Projects/seminars
0

Number of credit points

3,00

Coordinators

mgr inż. Paweł Kubczak
pawel.kubczak@put.poznan.pl

Lecturers

Prerequisites

A student starting this subject should have basic knowledge of mathematics and physics as well as the basics of the theory of electronic circuits and systems. They should also have the ability to obtain information from indicated sources and be ready to cooperate as part of a team.

Course objective

The aim of the course is to familiarize yourself with the process of designing printed circuit boards. Presentation of standards standardizing the process of designing and manufacturing printed circuit boards. Demonstrate design principles to reduce various types of interference.

Course-related learning outcomes

Knowledge:
1. Has basic knowledge in the construction, description, design and fabrication of printed circuit boards.
2. Has knowledge of standardization of the design and manufacturing process of electronic devices.
3. Knows the internal and external causes of conflicts in printed circuits and the basic methods of their reduction.

Skills:
1. Can obtain information from literature and other sources, can integrate the obtained information, interpret it, draw conclusions and justify opinions.
2. Can use catalogs, select appropriate electronic components and systems. Can identify the problem and formulate the design specification of a simple electronic circuit.
3. Has the ability to analyze and design printed circuit boards taking into account the given criteria, using appropriate engineering methods and tools.

Social competences:
1. Is aware of the need for a professional approach to solved technical problems and taking responsibility for the technical solutions proposed by him/her.
2. Has the ability to work in a group and is able to carry out team projects.
3. Can formulate opinions on the basic challenges faced by modern technology of electronic devices.

Methods for verifying learning outcomes and assessment criteria

Learning outcomes presented above are verified as follows:

Lecture:
The knowledge acquired during the lecture is verified by a written test. To obtain a grade of 3.0, it is necessary to score more than half of the possible points. Rating scale: <0%-50%) - 2.0 ; <50% -60%) - 3.0; <60%-70%) - 3.5; <70% -80%) - 4.0; <80% -90%) - 4.5; <90% -100%> - 5.0.
Laboratory:
Assessment of learning outcomes is carried out by preparing reports and assessing preparation, behaviour and commitment during classes. Assessment is carried out on a continuous basis through each verification of knowledge by oral answers to questions asked during laboratory exercises.

Programme content

The lecture and laboratory program provides knowledge and skills related to the design of printed circuits, focusing on various technologies used in the production of these circuits. The labs provide hands-on design experience, enabling students to fully understand the challenges of the issue.

Course topics

Lecture
Stages of making a printed circuit board, block diagram and rules for its drawing, schematic diagram, graphic symbols used in diagrams, international standard IEC 60617, Polish Standard PN - EN 60617/2003, ANSI Y32/IEEE 315 standard, principles of correct arrangement of components, standardization of designations of electronic component enclosures - IEC 60191-4 standard, type of enclosure, type of lead, computer methods of print design, rasterbasic, metric and imperial measure, technologies for the production of printed circuit boards, technologies for soldering, testing the correctness of printed circuit boards.
Standardization of the design and production process of printed circuit boards: SMT technology, THT technology, component density levels, what is a "footprint"?, definition of surface mount pads - layers and their purpose, definition of through-hole pads - layers and their purpose, rules for selecting the diameter of drilled holes, vias and "anti-pads", rules for selecting the dimensions of pads that reduce heat transport, standardization of pad markings and vias default values and modifiers.
Principles of calculating pad dimensions for surface mounting, taking into account the tolerances of components and production inventory, rules of rounding pad dimensions and coordinates of its position, creating a pad grid for a given housing, conversion of pad grid markings, standard SMD chip housings, soldering quality control, BGA, LGA, CGA and PCGA housings.
IPC hierarchy of "design specifications", standardization of trace routing, component placement, determination of trace impedance and capacitance, component placement, determination of trace impedance and capacitance, multilayer boards, reference layers, basic models: "microstrip:, "embdeded microstrip", "symetric stripline" and "dual (asymetric) stripline", dielectric snag propagation rate, signal propagation time in trace, trace impedance control, reflection elimination signals, 1/3 rise time rule, design of boards with traces of known impedance, recommended trace widths, minimum spacing between traces for individual device classes.
Design of printed circuit boards for EMI reduction, electromagnetic compatibility, channels for emitting electromagnetic interference, simple EMI model - ways of removing interference, equivalent

components and frequency characteristics of paths and basic electronic components for low and high frequencies, sources of interference in digital circuits, signal return path for low and high frequencies, single-layer boards: arrangement of components, routing ground and power supply paths, principles of designing double-layer boards, effect of interference emission from the trace, Maxwell's I and II equation, routing traces to cancel (limiting) the flux of magnetic induction, 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 capacitors, parallel connection of decoupling capacitors - series and parallel resonance area, power supply support capacitors, principles of selection of decoupling and power supply backup capacitors, signal reflections due to lack of impedance matching, reflection coefficient, multiple signal reflection, path design with the use of matching thermistors, series terminator (source terminator), parallel terminator, RC terminator, Thevenin terminator, crosstalk, methods of eliminating signal crosstalk, current density distribution in the dielectric, elimination of crosstalk: 3-d principle, shielding path routing.
Laboratory
Stages of making a printed circuit board, block diagram, and the principles of its drawing, schematic diagram, graphic symbols used in diagrams, international standard IEC 60617, Polish Standard PN - EN 60617/2003, ANSI Y32/IEEE 315 standard, principles of correct arrangement of components, standardization of designations of electronic component enclosures - IEC 60191-4 standard, type of housing, type of lead, computer methods of print design, raster basic, metric and inch measurement. Standardization of the design and production process of printed circuit boards: SMT technology, THT technology, levels of packing density of components, what is a "footprint"?, definition of surface mount pads - layers and their purpose, definition of through-hole mount pads - layers and their purpose, rules for selecting the diameter of drilled holes, vias and "anti-pads", rules for selecting the dimensions of pads reducing heat transport, standardization of pad and via markingsdefault values and modifiers. Trace impedance control, elimination of signal reflections, 1/3 rise time rule, design of boards with traces of known impedance, recommended trace widths, minimum spacing between traces for individual device classes. Design of printed circuit boards to reduce EMI, electromagnetic compatibility, channels for emitting electromagnetic interference, simple EMI model - ways to remove interference, equivalent diagrams and frequency characteristics of traces and basic electronic components for low and high frequencies, sources of interference in digital circuits, signal return path for low and high frequencies, single-layer boards: arrangement of components, routing ground and power supply paths, principles of designing double-layer boards, return path in multilayer boards, principles of selecting decoupling and power supply backup capacitors, reflections due to lack of impedance matching, reflection coefficient, designing paths with the use of matching terminators, serial terminator (source terminator), parallel terminator, RC terminator, Thevenin terminator, crosstalk, method of elimination crosstalk, shielding paths.

Teaching methods

Lectures
1. Multimedia presentation: the lecturer presents materials using slides, supplemented with photos, videos and other visual elements, of real devices/layouts.
2. Interactive lecture: The lecturer engages students in discussion, asks questions, and encourages them to share their own thoughts, aiding a better understanding of the material and the development of critical thinking skills.
3. Case study: the lecturer discusses a specific example, analyzing the problem and proposing solutions. This allows you to apply theoretical knowledge in practice.
Laboratory:
1. Simulations: Students work with computer programs that imitate real-life situations.
2. Practical exercises: students perform tasks under the supervision of the teacher, learning how to use knowledge in practice.
3. Work in groups: students work together to solve a problem, sharing knowledge and developing communication and teamwork skills.

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:
[5] Rymarski Z., Materiałoznastwo i konstrukcja urządzeń elektronicznych, Wydawnictwo Politechniki Śląskiej, Gliwice 2000.
[6] Thierauf S. C., Hight-speed circuit board signal integrity, Artech House, 2017.
[7] Montrose M. I., EMC and the printed circuit board: design, theory, and layout made sipmle, John Wiley & Sons, 2004.

Breakdown of average student's workload

	Hours	ECTS
Total workload	85	3,00
Classes requiring direct contact with the teacher	45	1,50
Student's own work (literature studies, preparation for laboratory classes/ tutorials, preparation for tests/exam, project preparation)	40	1,50