Cracow University of Technology

Cracow University of Technology
Faculty of Electrical and Computer Engineering

COURSE TITLE:	**LT Spice - Power Electronics Control**
Institute/Division:	Department of Electrical Engineering Faculty of Electrical and Computer Engineering
Course code:	E-LTS
Erasmus subject code:	0713 Electricity and energy 0714 Electronics and automation
Number of contact hours	45
Course duration:	1 semester (Fall/Winter)
ECTS credits:	6
Course description:	The course comprises lectures, laboratory exercises and individual project. It covers basic aspects of power electronics control by means of the simulator LT Spice. LTspice is one of the most widely used free simulators. It is a powerful simulator with a simple interface to handle. All subjects are clarified and exercises and computer simulations are made. Subjects of the course are listed below: • Bridge rectifiers control • Voltage and current control of DC choppers • Voltage and current control of a three-phase voltage inverter • Control of current inverter • Pulse regulation of direct voltage • Three-Phase alternating current regulator
Course type:	Lectures (20h), Laboratory (20h), Project (5h)
Literature:	Brocard G., The LT Spice XVII Simulator: Command and Applications. Manual, Methods and Applications, Swiridoff Verlag, 2021 May C., Passive Circuit Analysis with LTspice®: An Interactive Approach, Springer, 2020 Pooja Mohindru P.,Pankaj Mohindru P., Electronic Circuit Analysis using LTSpice XVII Simulator. A Practical Guide for Beginners, CRC Press, 2021 Singh A.K, Singh R., Electronics Circuit SPICE Simulations with LTspice: A Schematic Based Approach , Kindle Edition, 2015
Assessment method:	Project and laboratory exercises
Contact Person:	Marcin Tomczyk, Ph.D Eng., marcin.tomczyk@pk.edu.pl