UNIVERSITY OF MITROVICA "ISA BOLETINI"

Course Curriculum Model (Syllabus)

Fakulty:	Faculty of Mechanical and Computer Engineering	
Department:	Computer Science and Engineering	
Level:	VI	
Code of the course:	312-CSE	
Course:	COMPUTER MODELING AND SIMMULATION	
Course Status:	Elective	(mandatory)
Semester:	VI	(spring)
Number of hours per week:	2+2	
ECTS:	5	
Time / location:		
Year of studies	III	
Lecturer:	Prof. Ass. Dr. Fitim Zeqiri	
Assistent:	Msc. Granit Hajra	
Contact details:	Lecturer	Assistant
Email:	fitim.zeqiri@umib.net	granit.hajra@umib.net
Telefon:	+38344355401	

Content	Opening and creation of project. Analysis of DC circuits. Analysis of DC circuits with dependent voltage and current sources. Application of Thevenin's and Norton's theorems. Analysis circuits in time domain. Analysis of bistable circuits and the invertors working. Analysis of operations of the operational amplifiers and their applications. Analysis circuits in the frequency domain. Work analyses of circuits by Fourier coefficients.
Purpose	By using simulation program (Pspice) to be analyzed: DC circuits; DC circuits with dependent sources; Sweep DC circuits, AC circuits in time domain; AC circuits using phasors; AC circuits in frequency domain; logic gates, etc.
Accessibility	Upon completion of this course students will be able to: - realize DC circuits analyses and discuss the presented results, - apply the main theorems on electrical circuits and verify obtained results, - present the main characteristics of electrical circuits, - realize AC circuits analyses and discuss the presented results, - realize analysis by phasors, - realize in the frequency domain analyses, - realize the analysis by logic gates, etc.

Program	weeks	Lecture
	First week:	Introduction, opening and creating the project, following elements.
	Second week:	Analysis of DC circuits
	***Third week*:**	Analysis of DC circuits with dependent voltage and current sources
	Fourth week:	Application of Thevenin's and Norton's theorems
	Fifth week:	Analysis of DC circuits with electronics componets
	***Sixth week*:**	Analysis of the circuits with global parameters
	Seventh week:	Analysis circuits in time domain
	Eighth week:	Analysis of bistable circuits and the invertors working
	***Ninth week**:*	Analysis of operations of the operational amplifiers and their applications
	***Tenth week**:*	Analysis of the circuits in the frequency domain
	***Eleventh week**:*	Analysis of AC circuits using phasors
	Twelfth week:	Analysis of the transformers in time domen and using phasors
	***Thirteenth week*:**	Analysis of the circuits by Fourier coefficients
	Fourteenth week:	Analysis of the counters and shift registers
	Fifteenth week :	Presentation the seminary works (projects) and discussions
Literature	**Principal literature:** [1] James A. Svoboda, "PSpice for Linear Circuits" John Wiley & Sons, 2007. [2] Dennis Fitzpatrick, "Analog design and simulation using OrCAD capiture and PSipce", Elsevier Ltd. 2012 **Recommended Literature:** [3] Muhammed H. Rashid, Hasan M. Rashid, "SPICE for Power Electronics and Electric Power", Taylor & Francis 2006	
Teaching methodology	Lectures, exercises, seminar, discussions, design electrical circuts.	

Contribution to student workload (which should correspond to student learning outcomes - 1 ECTS credit = 25 hours)

Activity	Hours	Days/weeks	Total
Lectures	2	15	30
Exercise sessions (with TA)	2	15	30
Practical work	0	0	0
Office hours	1	15	15
Fieldwork	0	0	0
Midterms, seminars	2	2	4
Homework	2	2	4
Self-study	1	20	20
Final exam preparation	2	9	18
Time spent in exams	2	1	2
Projects, presentations, etc	2	1	2
Total			**125**

Evaluation

Teaching methodology:
(according to the Statute and Regulation for studies of UMIB)

Tests / Colloquia	20%+20%
Practical test during exercises	
Seminary work	20
Interpretation and presentation of artistic creativity and other works	
Final exam + Attendance	35%+5%

Mitrovica **Course provider:**

(Name Surname)

(Signature)