COURSE DESCRIPTION

1. Program identification information

1.1 Higher education institution	Politehnica University of Bucharest
1.2 Faculty	Faculty of Electronics, Telecommunications and Information Technology
1.3 Department	Electronic Devices, Circuits and Architectures
1.4 Domain of studies	Electronic Engineering, Telecommunications and Information Techologies
1.5 Cycle of studies	Bachelor
1.6 Program of studies/Qualification	TSTeng

2. Course identification information

2.1 Name of the course				**SPICE models for the electronics components**			
2.2 Lecturer				Prof. dr. ing. Dragos Dobrescu, Prof. dr. ing. Lidia Dobrescu,			
2.3 Instructor for practical activities				Prof. dr. ing. Dragos Dobrescu, Prof. dr. ing. Lidia Dobrescu, Dr. Ing. Busu Iulian.			
2.4 Year of studies	2	2.5 Semester	4	2.6 Evaluation type	Verification	2.7 Course choice type	Optional

3. Total estimated time (hours per semester for academic activities)

3.1 Number of hours per week, out of which	2	3.2 course	1	3.3 practical activities	0/1
3.4 Total hours in the curricula, out of which	28	3.5 course	14	3.6 practical activities	14

Distribution of time	hours
Study according to the manual, course support, bibliography and hand notes	15
Supplemental documentation (library, electronic access resources, in the field, etc)	15
Preparation for practical activities, homeworks, essays, portfolios, etc.	15
Tutoring	0
Examinations	2
Other activities	3

3.7 Total hours of individual study	50
3.9 Total hours per semester	78
3. 10 Number of ECTS credit points	3

4. Prerequisites (if applicable)

4.1 curricular	Electronic devices
4.2 competence-based	Knowledge's of Electronic devices and simple electronic circuits

5. Requisites (if applicable)

5.1 for running the course	Accordingly with UPB chart and rules
5.2 for running of the applications	Accordingly with UPB chart and rules

6. Specific competences

Professional competences	**C1.** Using fundamentals to devices, circuits, systems, instrumentation and electronic technology. **C3**. Applying knowledge, concepts and methods to the underlying computing systems architecture, microprocessors, microcontrollers, programming languages and techniques
Transversal competences	**CT3** Adapting to new technologies, professional and personal development through training documentation using printed sources, specialized software and electronic resources in Romanian and at least one international language.

7. Course objectives (as implied by the grid of specific competences)

7.1 General objective of the course	Familiarize students with models of active components in SPICE simulators based on general theoretical and practical knowledge on key parameters of the model and methods for their extraction.
4.2 Specific objectives	- Highlighting the model parameters and methods for extracting them. - -Practical models selection in SPICE simulators for editing and analyzing electronic circuit diagrams. -- Using subprograms for parameters extraction in order to modify or creatie models for active components exist in libraries or for of new models validation. - Classification and ranking of SPICE models; - Efficient running simulation programs by ensuring optimum between complexity and number of parameters, robustness and accuracy of the model chosen.

8. Content

8.1 Lectures	Teaching techniques	Remarks
1. Spice presentation – electronic circuit simulator • Modeling problem in the nowaday state-of-the-art • Models – Parameters – Parameters Extractors • Extraction algorithms Emphasized extractor – PARTS	Teaching is based on using alternative projector and blackboard chalk detail (covering communication function and demonstration); oral communication methods are used and the method problematizării expository method used loader. The course material are lecture notes and presentations, books (theoretical and practical examples of solving scheme by Spice simulations). Course materials are available both in print publishers with ISBN UEFISCDI recognized, and in electronic format.	2
2. SPICE models of diodes • Diode statement • Static regime modeling of diodes in SPICE • Dynamic regime modeling of diodes in SPICE • Example of SPICE file including diodes		2
3. SPICE models of bipolar transistors. Static model. • Bipolar transistor statement • Ebers-Moll fundamental model, BRIEF: Series resistance, • Early effect • Generation-recombination effect • High level of injection		4
4. Bipolar transistor dynamic regime. Brief. • Transition capacitancies • Diffusion capacitancis • Specific parameters extraction • Complete circuit of the bipolar transistor		4
5. SPICE models of MOS transistors.		2

<table>
<tr><td colspan="3">Bibliography
• “ SPICE “, Andrei Vladimirescu, Tehnical Publishing House, 1999.
• “Active Components Modelling “, A. Rusu, Romanian Academy Publishing House, 1990.
• “ Electronic Circuits Analysing from function towards device “, Dragos Dobrescu, Printech Publishing House, , 2004.
• “Modele SPICE ale componentelor microelectronice “– C. Ravariu, A. Rusu, Ed. Matrix, 2006.
• “Basics of the semiconductor devices physics “, Lidia Dobrescu, Dragos Dobrescu, Printech Publishing House, 2005.
• “ SPICE   for Electronic Circuis Simulation, C. Ravariu, S. Ionel. Bucharest 2011, http://roma.mcma.pub.ro/~lidiad</td></tr>
<tr><td>8.2 Practical applications</td><td>Teaching techniques</td><td>Remarks</td></tr>
<tr><td>1. SPICE circuit simulation in multiple diodes with a focus on editing the list of model parameters, environment CADENCE ORCAD PSPICE from Cis-LITE 16.6</td><td rowspan="3">Students extract model parameters for Spice creates a model of active components and simulates an electronic circuit modeled with the active component; facilities use the computers and software environment. The teaching materials are available in electronic format platforms.</td><td>3 hours</td></tr>
<tr><td>2. Extraction of parameters for diode sub Parts, comparative and advanced simulations (diode as a temperature sensor on various materials etc.) using CADENCE ORCAD PSPICE from Cis-LITE 16.6</td><td>3 hours</td></tr>
<tr><td>3.Using LTSPICEIV for bipolar circuit simulation; .DC analysis, .AC, .TRAN</td><td>3 hours</td></tr>
<tr><td>4.Using LTSPICEIV circuit simulation for MOS transistors; NAND applications, highlighting amplifier sub-threshold currents.</td><td></td><td>3 hours</td></tr>
<tr><td>Laboratory tests</td><td></td><td>2 hours</td></tr>
<tr><td>Total</td><td></td><td>14 hours</td></tr>
<tr><td colspan="3">Bibliography
Spice-Analog circuits simulation, Emil Sofron, . Military Publishing House, 1994.
• SPICE laboratory workbook “ – C. Ravariu, 2001.
http://roma.mcma.pub.ro/~lidiad/, www.arh.pub.ro\cravariu,</td></tr>
</table>

9. Bridging the course content with the expectations of the epistemic community representatives, professional associations and employers representatives for the domain of the program

The increasing complexity of electronic circuits and systems and the need to reduce costs and cycle research and design-manufacturing techniques required the development of simulation, design and optimization of computer assisted form of various software tools. Most companies design-fabrication of semiconductor industry calls today to schedule Spice simulation at the transistor of a circuit or portion of the circuit. In SPICE programs used by design firms there is the possibility to select models of active components (bipolar, MOS etc.). Selection can be done individually, choosing a model program available from libraries or by attaching an external model supplied by independent producers. Selection can also be made by specifying default parameter LEVEL program. This parameter defaults to 1 incorporates a specific analytical model with a degree of complexity to describe transistor behavior. A circuit design engineer must choose between robustness, simplicity and efficiency and to make the best fit its application. If LEVEL1 supports simplified model for the transistor current regime, sometimes enough to simulate a digital circuit, the Level3 incorporates an empirical model developed, suitable for analog applications. Discipline provides graduates with skills appropriate to the needs of current skill and training scientific and technical modern, quality and competitive, enabling them hiring rapidly after graduation, being perfectly framed policy Politehnica University of Bucharest, both in terms of content and structure and in terms of skills and international openness offered to students.

10. Evaluation

Type of activity	10.1 Evaluation criteria	10.2 Evaluation methods	10.3 Weight in the final mark
10.4 Lectures			
Fundamental of theory	test	Written test	80%
10.5 Practical applications			
Spice simulations	test	Oral test	20% - lab test
10.6 Minimal performance standard: minimum 50 point; lab is obligatory.			
¬ Identify the main parameters of model ¬ Extraction model parameters from experimental curves, or catalog;			

Date Lecturer

25.09.2017

Prof. dr. ing. Dragos Dobrescu,

Prof. dr. ing. Lidia Dobrescu

Instructor for practical activities

Prof. dr. ing. Dragos Dobrescu,

Prof. dr. ing. Lidia Dobrescu

Dr. Ing. Busu Iulian.

Date of department approval

26.09.2017

Director of Department,

Prof. dr. ing. Claudius Dan