COURSE GUID8
Academic year 2019-2020

Dean,
Professor Daniela Tărniceriu, PhD

1. Program info

1.1 Higher education institution	"Gheorghe Asachi" Technical University of Iaşi
1.2 Faculty / Department	Electronics, Telecommunications and Information Technology
1.3 Department	Telecommunications and Information Technologies
1.4 Field	Electronics Engineering, Telecommunications and Information Technologies
1.5 Study level	Bachelor's Degree Studies
1.6 Study program / Qualification	Telecommunications Technologies and Systems

2. Course info

2.1 Course name	**Computer-Aided Analysis of Electronic Circuits**					Code	EDIS203
2.2 Course organizer (lecturer)	Dănuţ Burdia, Associate professor, PhD						
2.3 Applications	Felix Diaconu, Lecturer, PhD						
2.4 Year of study	**2**	2.5 Semester	**3**	2.6 Assessement	**Exam**	2.7 Category	**DS**

203

3. Estimated total time (hours per semester for teaching activities)

3.1 Number of hours per week	**4**	3.2 lecture	**2**	3.3 seminary/laboratory	**2**
3.4 Total number of hours in curricula	**56**	3.5 lecture	**28**	3.6 seminary/laboratory	**28**

Time distribution	hours
Textbook, course support, references and course notes study	24
Library, electronic platforms and on site documentation	8
Seminar/laboratory preparation, homework, reports, portfolios and essays	18
Tutoriing	7
Assessment	3
Other activities – advice, consultation	4

3.7 Total individual study hours	**64**
3.9 Total hours per semester	**120**
3. 10 Number of credit points	**5**

4. Prerequisites (where applicable)

4.1 curricula	
4.2 competences	Basic of linear algebra, working with matrixes, knowledge of electrical fundamentals, basic skills in using computers

5. Infrastructure (where applicable)

5.1. for lectures	Course hall or lecture theater with capacity of minimum 50 students. Equipements: blackboard or whiteboard with accesorises, desktop computer or laptop, video projector (beam), projection screen, crete or white board marker.
5.2. for laboratories	- Laboratory room with min. 9 workstations with internet access. Software tools: OrCad PSpice, internet browser, text editor, Adobe Reader. - Laboratory work must be carried out fully, the results being noted at each session. Students will be presented to the laboratory with a brief summary of the laboratory's essay. Homework must be presented at deadline. Delays in delivery have to be justified. Presentation at exam is conditioned by attending and graduating the applications.

6. Specific skills acquired

Professional competences	• To know the types of models and modeling techniques of electronic devices • To know the concepts of electrical circuits topology and the computer-based generation techniques of topological matrixes • To understand the methods of formulating equations for linear or nonlinear circuits • To knows the methods for matrix memorize and the sparse matrix techniques • To shows the capability of using appropriate algorithms for the analysis of linear or nonlinear resistive networks. • To demonstrate abilities to identify and resolve the convergence issues • To knows the SPICE syntax and using the model libraries. • To knows what type of analysis have to be done to evaluate the circuit performances • To choose the appropriate parameters for each type of simulations • To develops skills for using the SPICE simulation tools • To show the capability of understanding the simulation results
Transversal competences	- To use efficiently the information and communication resources in the field of computer-aided analysis - Demonstrate concern for professional development through training critical thinking skills and to improve their training and education throughout the lifespan of the activity. - To develop teamwork skills and familiarize easily in an environment dedicated to computer-aided analysis of electronic circuits

7. Course target (as resulting from **6. Specific skils acquired** table)

7.1 Course main target	- The development of theoretical and practical competence in the field of electronic modeling and computer-aided techniques for analyzing analog circuits.
7.2 Course specific targets	- To demonstrate knowledge of modeling techniques of electronic devices - To demonstrate knowledge of concepts and techniques of electrical circuits topology - To apply appropriate methods for the formulations of the linear or nonlinear circuit equations - To understand the principles of the algorithms used in the linear or nonlinear networks analysis. - To use the SPICE simulator for describing and simulating of electronic circuits - To understand the source of errors and to resolve the convergence issues. - To demonstrate knowledge in using the Probe graphic analyzer to represent the simulation results (waveforms, characteristics)

8. Contents

8. 1 Lectures	**Teaching methods**	**Notes**
Introduction to computer aided simulation	-lecturing -beam using -explanations -debate - discussions	2 lectures
Circuit models of components and electronic devices usable in simulators		2 lectures
Network topology: the key of Kirchhoff laws formulation on computer		2 lectures
Linear network analysis by nodal method		5 lectures
Non-Linear resistive network analysis by nodal method		3 lectures

References

1. Chua L.O. and P.M. Lin, Computer Aided Analysis of Electronic Circuits, Prentice Hall, 1975.
2. D. Burdia, Computer-Aided Analysis of Electronic Circuits (Romanian), Tehnopres, Iaşi, 2009 (ch. 1-5)
3. D. Burdia, G.S. Popescu Computer-Aided Design of Electronic Circuits:. SPICE and VHDL (Romanian), Part I: SPICE, Matrixrom, 1999.
4. Vlach, J. and K. Singhal, Computer Methods for Circuit Analysis and Design, New York, van Nostrand Reinhold, 1983
5. Ruehli A.E., Circuit Analysis, Simulation and Design, Advances in CAD for VLSI, vol. 3, North-Holland, 1987
6. Jenkins D.G. and R.C. Welland, Software Engineering for Electronic Systems, IEE Computing Series 18, 1990.
7. Tuinenga, Paul W, SPICE – A Guide to Circuit Simulation & Analysis Using Pspice, Prentice Hall, 1992
8. Vladimirescu, A. – SPICE, Ed. Tehnică, Bucuresti, 1999.
9. Ioinovici, A. – Computer-Aided Analysis of Active Circuits, Ed. Marcel Dekker, NY, 1990.

8. 2 Laboratory	Teaching methods	Notes
1. Safety working rules, History of circuit simulators	Practical demonstration Exercise	
2. PSpice simulator – basic concepts		
3. SPICE syntax and modeling of passive devices		
4. SPICE syntax of independent voltage and current sources		
5. SPICE syntax of controlled sources and switches.		
6. DC analysis		
7. AC analysis		
8. Transient analysis		
9. Harmonic distorsions analysis		
10. Part 1: Parametric analysis. Part 2: SPICE syntax of device models and subcircuits		
11. SPICE models and syntax of diodes and bipolar transistors		
12. SPICE models and syntax of JFETs and MOSFETs		
13. Statistical analysis		
14. Test :. PSpice simulation of a given electronic circuit		

References

1. *** The Design Center, Circuit Analysis Reference Manual, MicroSim Corp., 1994
2. *** The Design Center, Circuit Analysis User's Guide, MicroSim Corp., 1994
3. http://www.orcad.com/buy/orcad-educational-program , Download the Capture and PSpice Lite / Student version
4. http://www.pspice.com/ PSpice User Forum
5. D. Burdia, G.S. Popescu Proiectarea asistata de calculator a circuitelor electronice. SPICE si VHDL, Partea I: SPICE, Matrixrom, 1999.
6. http:\\www.etti.tuiasi.ro/pac

9. Corroborating the contents discipline expectations epistemic community representatives, professional associations and employers representative for the field program

In determining the content of discipline and teaching methods / examination discipline holders have consulted with counterparts from both the Romanian academic community and from abroad. Also, it has taken into account the opinion and expectations of key industry players in Romania, with which the faculty has collaboration. Course objectives are consistent with the curriculum, passing on information and skills necessary for forming future specialists in electronics and telecommunications.

10. Assessment

Activity type	10.1 Assessment criteria	10.2 Assessment Methods	10.3 Percentage of the final grade
10.4 Course	• Theoretical knowledge acquired	Intermediate tests:	%

	(quantity, correctness, accuracy)	Homeworks:	%
		Final exam: writing exam	60 % (minim 5)
10.5b Laborator	Knowledge of equipment, how to use specific tools; assessment tools or achievements, processing and interpretation of results	- written questionnaire - oral answer - laboratory notebook (experimental works, essays) - practical demonstration	40 % (minim 5)
10.6 Minimum performance standard			
• Know the fundamentals of theory of circuit simulation, solving simple problems			

Completion date:
13.09.2019

Course organizer signature,
Dănuț Burdia, Associate Professor

Teaching assistant signature,
Felix Diaconu, Lecturer

Department approval date

16. SEP. 2019

Department director signature,

Luminița Scripcariu, Associate Professor, PhD