



DISCIPLINE ACADEMIC SHEET
ACADEMIC YEAR 2025 – 2026

1. PROGRAMME DATA

1.1. Higher Education Institution	UNIVERSITY OF CRAIOVA
1.2. Faculty	Automation, Computers and Electronics
1.3. Department	Computers and Information Technology
1.4. Field of Study	Computers and Information Technology
1.5. Study level	L(License / Undergraduate)
1.6. Organizational form	IF
1.7. Study Program (name/code) /Qualification	Computers / L2060101010

2. DISCIPLINE DATA

2.1 Discipline Name	SPICE Models						
2.2 Course Activities Holder	Ș.l. dr. ing. Firincă Sanda-Diana						
2.3. Practical Activities Holder	Ș.l. dr. ing. Firincă Sanda-Diana						
2.4. Study Year	2	2.5. Semester	1	2.6. Evaluation type	V	2.7. Discipline regime	DFA

3. ESTIMATED TOTAL TIME (hours per semester of teaching activities)

3.1. Number of hours per week	2	in which: 3.2 course	1	3.3. sem./lab./pr.	1
3.4. Total hours of curriculum	28	in which: 3.5 course	14	3.6. sem./lab./pr.	14
3.7. Time distribution - hours/week					hours
Study after manual, course support, bibliography and notes					10
Additional documentation in library, on specialized electronic platforms and field					4
Training seminars / labs, homework, portfolios and essays					6
Tutorial					-
Examinations					2
Other activities.....					-
Total hours per individual activities					22
3.8. Total hours per semester					50
3.9. Number of credits					2

4. PRECONDITIONS (where appropriate)

4.1. of curriculum	The student must possess knowledge acquired in the following disciplines: Mathematical Analysis, Numerical Methods, Computer Programming, Physics I, Electrotechnics, Electronic Devices and Analog Electronics
4.2. of competence	General knowledge of simulating electronic circuits of small/medium complexity using CAD-CAM technologies and the standards in the field

5. CONDITIONS (where appropriate)

5.1. of the course	The course is taught using the video projector/online. The board is used for explanations and answers to questions. Course support in electronic format and access to up-to-date documentation are provided. The teaching process has the following structure:
5.2. of seminar/ laboratory/project	

6. COURSE OBJECTIVES – expected learning outcomes that are supported by completing and passing the course

Knowledge	The student/graduate describes, identifies and summarizes elementary concepts and methods regarding devices, circuits and the way they are applied to concrete problems.
Skills (Abilities)	The student/graduate uses specific methods for measuring electrical quantities and identifies digital and analog electronic devices. The student/graduate analyzes systems using the studied theories and designs, implements, diagnoses and debugs digital systems. The student/graduate uses specific theories and tools (applications, models, protocols etc.) for the analysis, simulation, design and implementation of computer networks.
Responsibility and autonomy	The student/graduate conducts processes in the management of computers and information technology projects, taking over different roles in the team and describing the results clearly and concisely, verbally and in writing.

7. CONTENT

7.1. Course	The method of implementation	Teaching methods	Allocated time (hours)
1. Introduction. Circuit simulation programs. 1.1 The structure of a simulation program. 1.2 Simulation environments and electronic circuit simulators - OrCAD		The course is taught in a mixed regime: online in the first 6 weeks according to the decisions of the UCv Senate and of the Faculty Council, and face to face in the following weeks. Course support in electronic format and access to up-to-date documentation are provided. The teaching process has the following structure: 75% presentation based	2
2. Types of analyses in PSpice. 2.1 DC analyses. 2.2 AC analyses, noise analysis. 2.3 Time-domain analyses, Fourier analysis.			2
3. Performance analysis. Optimization of the parameters of circuit elements and of circuits in Spice. Statistical analyses.			2

4. Behavioral modeling and hierarchical simulation		on the course support; 25% interactive activity (discussions with the students).	2
5. Modeling electronic devices. Modeling the semiconductor diode. Modeling the bipolar transistor. Modeling JFET transistors. Modeling MOS transistors			6
Bibliography:			
1. Gheorghe Pană, Marius Cătălin Carp, Tehnici de simulare, Aplicații în ingineria electrică și electronică, Editura Universității Transilvania din Brașov, 2011. 2. Nassir Sabah; Circuit Analysis with PSpice – A Simplified Approach; 2017, CRC Press. 3. Lucia Dumitriu, Mihai Iordache, Simularea numerică a circuitelor analogice cu programul Spice, București, 2006. 4. Allan Robbins, Wilhelm Miller; Circuit Analysis; Theory and Practice – Fifth Edition; 2013, Delmar, Cengage Learning; 5. Muhammad H. Rashid, SPICE for Power Electronics and Electric Power, Third Edition, CRC Press, 2012. 6. SPICE – User’s Guide			

7.2. Seminar/laboratory/project	The method of implementation	Teaching methods	Allocated time (hours)
1. Introduction to the PSpice simulation of electronic circuits.		The application classes will take place in a mixed regime: online in the first 3 weeks according to the decisions of the UCv Senate and of the Faculty Council, and face to face in the following weeks. 25% presentation of the theoretical notions and examples. 75% interactive activity (discussions with the students). The necessary materials will be made available to students in electronic format. For the laboratory activity a room equipped with computers and the ORCAD program is used	2
2. DC analyses.			2
3. AC analyses.			2
4. Time-domain analyses.			2
5. Modeling semiconductor diodes. Modeling the bipolar transistor. Modeling JFET and MOS transistors.			2
6. Functional and hierarchical simulation.			2
7. Lab recovery sessions, assessments, testing.			2
Bibliography:			
1. Gheorghe Pană, Marius Cătălin Carp, Tehnici de simulare, Aplicații în ingineria electrică și electronică, Editura Universității Transilvania din Brașov 2011. 2. Nassir Sabah; Circuit Analysis with PSpice – A Simplified Approach; 2017, CRC Press. 3. Lucia Dumitriu, Mihai Iordache, Simularea numerică a circuitelor analogice cu programul Spice,			

București, 2006.

4. Allan Robbins, Wilhelm Miller; Circuit Analysis; Theory and Practice – Fifth Edition; 2013, Delmar, Cengage Learning;

5. Muhammad H. Rashid, SPICE for Power Electronics and Electric Power, Third Edition, CRC Press, 2012. SPICE – User's Guide

8. Correlating the content of the discipline with the expectations of representatives from the epistemic community, professional associations and representative employers in the field related to the program

The course content was discussed with representatives of companies in the field.

9. EVALUATION

Activity Type	9.1. Evaluation criteria	9.2. Evaluation methods	9.3. Final mark weight
9.4. Course	C: Students must possess fundamental knowledge about analog circuit theory, the main analysis/synthesis methods and the main software tools used in computer-aided design.	Final written/oral exam	50%
9.5. Seminar/laboratory/project	L: Students must demonstrate knowledge about: editing the schematics of the circuits under analysis, mastering the way of setting the type of analysis, testing the schematic editing and analysis settings, simulating the circuits, interpreting the data.	Ongoing assessment and final examination: - Presentation and checking of homework (correlated with the laboratory works), verified during the semester. - Final examination: The laboratory test consists in simulating an electronic circuit (using the OrCAD16.6 program).	25% 25%
9.6. Minimum standard of performance			
Assimilation of the theoretical knowledge regarding the simulation of electronic circuits. Acquiring the skills for using electronic circuit simulation programs. Obtaining at least 50% of the score of the ongoing assessments, laboratory tests and final exam. The final			

grade is calculated by rounding the final score to a whole grade.

Date of Completion	Signature of the Course Activities Holder,	Signature of the Practical Activities Holder,
23.08.2025	Ș.l. dr. ing. Firincă Sanda-Diana	Ș.l. dr. ing. Firincă Sanda-Diana

Date of Approval in the Department	Signature of the Department Director,
25.08.2025	Ș.l.dr.ing. Nicolae ENESCU

Date of Approval in the Faculty	Signature of the Dean,
	Prof.dr.ing. Dorin ȘENDRESCU