

GUJARAT TECHNOLOGICAL UNIVERSITY

ELECTRONICS (10), ELECTRONICS & COMMUNICATION (11), ELECTRONICS & TELECOMMUNICATION ENGINEERING (12)

SIMULATION AND DESIGN TOOLS

SUBJECT CODE: 2141006

B.E. 4th SEMESTER

Type of Course: Introduction of various Simulation and Design tools for Electronics

Prerequisite: Basic knowledge of computer programming and Analog and Digital Electronics.

Rationale: The course provides introduction to Simulation and Designing softwares for electronics. Using simulation software the students can design and analyze various analog and digital circuits. The students can design PCB layout for the desired circuits using PCB design tools. Using MATLAB/SCILAB various waveforms can be generated and various electronics systems can be implemented.

Teaching and Examination Scheme:

Teaching Scheme			Credits	Examination Marks						Total Marks
L	T	P		Theory Marks			Practical Marks			
			ESE (E)	PA (M)		ESE (V)		PA (I)		
				PA	ALA	ESE	OEP			
0	0	2	2	0	0	0	50	30	20	100

Content:

Sr. No.	Topics	Teaching Hrs.	Module Weightage
1	Introduction to PSpice software General purpose circuit simulation using Schematic Editor, Introduction to netlist command based SPICE simulation, basic netlist commands. Basic circuit analyses: DC, AC Transient.	20	56%
2	Introduction to PCB Design software Schematic Entry, Netlist Creation, Working with component libraries, Design of Boards, Layout of Parts, Optimizing Parts Placements, Pads and Via, Manual and Auto Routing, Handling Multiple Layers	08	22%
3	Introduction to SCILAB Introduction to SCILAB, use SCILAB functions. Writing simple programs using SCILAB, handling arrays, files, plotting of functions etc. Writing SCI files for Creation of analog & discrete signals, plotting of signals etc. Simulation of electronic circuits using SCILAB	08	22%

Reference Books:

Modelling and Simulation in Scilab/Scicos By. Stephen L. Campbell et. Al. Springer
(This subject aims "Learning by doing" so best practice is use software and its help document and guidance from faculty, however above book is suggested for SCILAB)

Course Outcomes:

After successful completion of the course students should be able to:

1. Design the electronics circuits using software tools like NGspice/LTSpice/Multisim.
2. Simulate various analog and digital circuits using NGspice/LTSpice/Multisim
3. Able to design PCB for given circuit using PCB Software like EAGLE, ExpressPCB,OrCAD.
4. Use open source SCILAB tool and write simple programs
5. Plot various waveforms using SCILAB.
6. Simulate basic electronic system blocks using SCILAB

List of suggested Experiments:

1. Simulation of one rectifier circuit and one clipper/clamper circuit.
2. Simulation of any one transistor biasing circuit.
3. Simulation of CE single/double stage amplifier circuit.
4. Simulation of any one power amplifier circuit.
5. Simulation of any one JFET/MOSFET amplifier circuit.
6. Simulation of any one negative feedback circuit.
7. Simulation of encoder/multiplexer circuit.
8. Simulation of decoder/de multiplexer circuit.
9. Simulation of any one flip-flop circuit using gates.
10. Simulation of any one register/counter circuit.
11. Design of PCB for any one circuit from experiment 1 to 6.
12. Design of PCB for any one circuit from experiment 7 to 10.
13. Plot the sine, cosine, triangle and exponential waveform using SCILAB.
14. Plot sampled sine, cosine, triangle and exponential waveform using SCILAB.
15. Study of Simulink. (only source and sink available in Simulink library).

Design based Problems (DP)/Open Ended Problem:

The student has to carry out any one project based on the following topics :

- Design of any complex analog or digital or hybrid circuit, implement it using simulation software and design the PCB for the same circuit.
- Generate and observe basic signals using XCOS.
- Design of GUI for any analog or digital circuit using SCILAB

Major Equipments:

Personal Computers with necessary simulation and design tools

List of Software/learning website:

NGspice, LTSpice,MULTISIM, Orcad, Proteus or other open source PCB design tools, SCILAB

Website: <http://www.scilab.org/> (To download SCILAB open source software)

<http://www.linear.com/>,

<http://www.expresspcb.com/>

<http://ngspice.sourceforge.net/>