DISCIPLINE SPECIFIC ELECTIVE COURSE : Computer Aided Design (INDSE5C)

CREDIT DISTRIBUTION, ELIGIBILITY AND PRE-REQUISITES OF THE COURSE

Course title & Code	Credits	Credit distribution of the course			Eligibility criteria	Pre-requisite of the course (if any)
		Lecture	Tutorial	Practical/ Practice		
Computer Aided Design (INDSE5C)	04	02	-	02	Class XII passed with Physics + Mathematics/Applied Mathematics+ Chemistry / Computer Science/Informatics	Analog and Digital Electronics

Learning Objectives

The Learning Objectives of this course are as follows:

- To familiarize with MultiSim and PSPICE circuit simulation tools
- To verify response of various analog and digital circuits
- To provide knowledge of Industry standard TCAD simulation tools like Silvaco-ATLAS and and Synopsis-SENTAURUS

Learning outcomes

The Learning Outcomes of this course are as follows:

- Simulate and verify the functionality of diodes and transistor circuits using MultiSim and PSpice software
- Design and verify devices/ circuits using TCAD tools

SYLLABUS OF DSE-3

UNIT – 1 **(6 hours)**

Introduction to Multisim software: MultiSim Environment: Design Process, setting environment preferences, Multisim GUI, Schematic capture of circuits: Placing components, wiring components, Measuring instruments in MultiSim, simulation and result display in MultiSim

UNIT – 2 **(6 hours)**

Electronics circuit design using Multisim: Resistive circuits, Design of Bridge rectifier, Half-Wave rectifier, clippers and clampers using a diode, DC transfer curve analysis, Transient analysis, simulation of digital circuits.

UNIT – 3 **(8 hours)**

Introduction to PSpice software Understanding the SPICE Environment, Schematic Designing Brief Introduction of p spice simulator, Using Model Editor, Understanding the PSPICE Environment, Using Magnetic Parts Editor, Using Stimulus Editor, Drawing a Circuit Preparation for Simulation: Preparing schematic for simulation, Understand the sources for simulation, Understand different markers and errors

UNIT – 4 **(10 hours)**

Introduction to Industry standard TCAD tools, Silvaco- ATLAS device simulation software, Synosis-SENTAURUS. Online Simulation resources-NANOHUB. Simulation of n-channel MOSFET; Silicon on Insulator.

Practical component: **(60 hours)**

1. Designing RC Low pass filter using MULTISIM
2. Designing active RC Low pass filter (OpAmp based) using MULTISIM
3. Half wave rectifier using MULTISIM
4. Wein bridge Oscillator using MULTISIM
5. Simulating high pass filter Circuit using PSPICE
6. Designing active RC High pass filter (OpAmp based) using PSPICE
7. Half wave rectifier using PSPICE
8. Designing and Simulating Full wave rectifier using PSPICE
9. Output characteristics of MOSFET using SILVACO-ATLAS/ Synopsis TCAD
10. Transfer characteristics of MOSFET using SILVACO-ATLAS/ Synopsis TCAD

Essential/recommended readings

1. Introduction To PSpice Using OrCADfor Circuits and Electronics, Muhammad H. Rashid,Paperback – Import,3rd Edition, 2003.
2. Electronic Devices and circuit theory, Robert Boylstead and Louis Nashelsky, PHI, 10th Edition, 2009.
3. https:i/nanohub.org/resources/tools
4. https ://www.silvaco. com/contenVkbase/device.pdf

Note: Examination scheme and mode shall be as prescribed by the Examination Branch, University of Delhi, from time to time.