COURSE CODE	COURSE NAME	L-T-P-C	YEAR OF INTRODUCTION
EC233	ELECTRONICS DESIGN AUTOMATION LAB	0-0-3-1	2016

Prerequisite: Nil

Course Objectives :

The primary objective of this course is to familiarize the students, how to simulate the electronics/digital circuits, signals and systems using the soft-wares which are available for the modern design methodologies for the rapid design and verification of complex electronic systems.

List of Exercises / Experiments

1. **Introduction to SPICE**

[Institution can use any one circuit simulation package with schematic entry like EDWinXP, PSpice, Multisim, Proteus or CircuitLab.]

Introduction to SPICE software. Recognize various schematic symbols /model parameters of resistor, capacitor, inductor, energy sources (VCVS, CCVS, Sinusoidal source, pulse, etc), transformer, DIODE, BJT, FET, MOSFET, etc., units & values. Use SPICE Schematic Editor to draw and analyse (DC, AC, Transient) simple analog and digital electronic circuits.

List of Experiments using SPICE [Six experiments mandatory]

Simulation of following circuits using SPICE [Schematic entry of circuits using standard package, Analysis –Transient, AC, DC]

1. Potential divider network
2. RC integrating and differentiating circuits
3. Diode, BJT and MOSFET characteristics
4. Diode Circuits (Clipping, Clamping, Rectifiers)
5. RC coupled amplifier (Single & two stages)
6. RC oscillator (RC phase shift / Wien Bridge)
7. Astable multivibrator
8. Truth table verification of basic and universal gates
9. Half adder /full adder circuits using gates
10. 4 bit adder/BCD adder
11. Encoder/Multiplexers
12. Flipflops/Counters

2. **Introduction to MATLAB**

[Institution can use any one numerical computational package like SciLab, Octave, Spyder, Python (scipy) or Freemat instead of MATLAB]

Fundamentals, basic operations on array, matrix, complex numbers etc., Script and function files, plotting commands, control statements.

Writing simple programs for handling arrays and plotting of mathematical functions, plotting of analog, discrete and noise signals, analysing the simple electronic circuits/network using node and mesh equations.

List of Experiments [Four experiments mandatory]

Write program and obtain the solutions

1. Solve /plot the mathematical equations containing complex numbers, array, matrix multiplication and quadratic equations etc

	2. Obtain different types of plots (2D/3D, surface plot, polar plot) 3. Generate and plot various signals like sine square, pulse in same window. 4. Plot the diode/transistor characteristics. 5. Solve node, mesh and loop equations of simple electrical/network circuits. 6. Find the poles and zeros hence plot the transfer functions/polynomials 7. Sort numbers in ascending order and save to another text file using text read and sort function after reading n floating point numbers from a formatted text file stored in the system. 8. Plot a full wave rectified waveform using Fourier series
3	**Introduction to HDL** [Institution can choose VHDL or Verilog as language to describe the problem and any one simulation/synthesis tool like Xilinix ISE, Modelsim, QSim, verilog, VHDL, EDwinXP or ORCAD etc. for the simulation.] **List of Experiments using HDL** Write the HDL code to realise and simulate the following circuits: (at least 4 of the following) 1. Basic gates/universal gates 2. Combinational Circuits (Half adder/Half subtractor) 3. Full adder in 3 modelling styles (Dataflow/structural/Behavioural) 4. Multiplexer/De-multiplexer 5. Decoder/Encoder 6. 4 bit adder/BCD adder 7. Flipflops (SR,JK,T,D) 8. Binary Counters 9. Finite state machines
Expected outcomes: 1. An ability to apply knowledge of computer, science, and engineering to the analysis of electrical and electronic engineering problems. 2. An ability to design systems which include hardware and software components. 3. An ability to identify, formulate and solve engineering problems. 4. An ability to use modern engineering techniques	