ECE DESIGN AND SIMULATIONS LAB-II

(Core Subject)

Course Code:	10M17EC271	Semester:	2nd Semester, M. Tech (ECE)
Credits:	2	Contact Hours:	L-0, T-0,P-2

Course Objectives

At the completion of this course, the student should be able to design and test the analog and digital circuits.

Course Outcomes

At the completion of this course, the student should be able to:

1. Analyze and synthesis analog and digital circuits by writing the PSPICE code and using the CAPTURE.

2. Use the knowledge acquired through this lab to design blocks that are used in communications systems and integrated circuits.

3. To use the knowledge acquired in this lab in the project related works in future.

List of Experiments

1   To perform different types of analysis on a given circuit which contains resistors and reactive elements.

2   To obtain the Thevenin's and Norton's equivalent circuits for a given circuit which contains dependent and independent sources.

3   To design simple RC/RL integrator and differentiation circuits for a given input signal. Obtain the frequency characteristics of integrator and differentiation as well.

4   To obtain the series resonant and shunt resonator circuits. To design different types of damping circuits using RLC series and shunt resonance.

5   To design an half-wave and full wave rectifier.

6   To design and obtain the voltage transfer characteristics of a positive, negative and double sided clippers.

7   To obtain the transistor input and output characteristics for common base, common emitter and common collector configurations.(Both npn and pnp transistors).

8   To design different transistor biasing circuits.

9   To obtain series and shunt feedback amplifiers-Frequency response, Input and output impedances.

10   Study transistor phase shift oscillator and observe the effect of variation in R andC on oscillator frequency and compare with theoretical value.

11   To design bi-stable and mono-stable and astable multi-vibrators using transistors.

12   To design inverting, non-inverting amplifiers using an op-amp. To obtain the transfer characteristic of an op-amp in open loop and design a voltage reference circuit.

13   To design active LPF and HPF using op-amp and to design a Schmitt trigger circuit.

14   To design Wien bridge, Colpits and Hartelys oscillators using op-amp.

15   To design astable and mono stable multi vibrator circuits using IC555 timer.

16   To design LC filters and compare their magnitude and phase responses.

17   To design a basic NAND and NOR TTL family circuits.

18   To design the basic CMOS NAND and NOR circuits and observe the wave forms.

19   To obtain the timing diagrams for basic gates and universal gates. To design a 4 to 1 multiplexer.

20   To design a digital to analogue converter.

Evaluation Scheme

1. Mid Sem Evaluation   20 Marks
2. End Sem Evaluation   20 Marks
3. Attendance   15 Marks
4. Class response   30 Marks
5. File   15 Marks

Total Marks   **100 Marks**

Text Books

1   Rashid, Muhammad H. Introduction to PSpice using OrCAD for circuits and electronics. Prentice-Hall, Inc., 2003.

2   Tobin, Paul. PSpice for Circuit Theory and Electronic Devices. Morgan & Claypool Publishers, 2007.