DEVICES AND CIRCUIT SIMULATIONS LAB

(Core Subject)

Course Code:	10B17EC471	Semester:	4th Semester, B. Tech (ECE)
Credits:	1	Contact Hours:	L-0, T-0,P-2

Course Objectives

The objectives are to make the student familiar with the basics of PSPICE and CAPTURE and to know the steps involved in the analysis and synthesis of analogue circuits.

Course Outcomes

After studying this course the students would gain enough knowledge After the successful completion of the course, student should be able to:

1. Draw various analogue circuits in PSPICE Capture and write the code for the same and verify them.
2. To synthesize different analogue circuits and observe different types of outputs depending on the analysis type.
3. To use the knowledge acquired in this lab in the project related works in future.
4. To use the knowledge acquired through this lab to design blocks that are used in communications systems and integrated circuits.

List of Experiments

1. Introduction to PSPICE and Capture. To verify the KCL and the KVL on a given circuit.
2. To perform the Nodal and Mesh analysis on a circuit containing independent and dependent sources.
3. To obtain the Thevenin's and Norton's equivalent circuits for a given circuit.
4. To perform transient analysis on RC, RL and RLC circuits. To design the series resonant and shunt resonator circuits.
5. To design a half-wave and a full wave rectifier circuits.
6. To design and obtain the voltage transfer characteristics of a positive, negative and double sided clippers and observe the time domain signals.
7. To design different transistor biasing circuits.
8. To obtain series and shunt feedback amplifiers-Frequency response, Input and output impedances.
9. To measure the common mode rejection ratio of a differential amplifier.

10. 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. To design active LPF and HPF using op-amp.
11. To design a Schmitt trigger circuit and to design Wien bridge, oscillators using op-amp.
12. To design astable and mono stable multi vibrator circuits using IC555 timer.
13. To design LC filters and compare their magnitude and phase responses.
14. To design bi-stable and mono-stable and astable multi-vibrators using transistors.

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, Muhamad H. "Introduction to PSpice Using OrCAD." *Prentice Hall of India* (2010).

2. Tobin, Paul. "PSpice for circuit theory and electronic devices." *Synthesis Lectures On Digital Circuits and Systems* 2.1 (2007): 1-159.