LINEAR INTEGRATED CIRCUITS & APPLICATIONS LAB
PCC-ECE306-P

General course information

Course Credits : 1 Contact Hours : 2 per week per group (L T P : 0 0 2) Mode : Lab Work	**Course Assessment (Internal: 30; External: 70)**

Pre-requisites: Linear Integrated Circuits and Applications

Sr. No.	**Course Outcomes** **At the end of the semester, students will be able:**	**RBT Level**
CO 1	To describe linear integrated circuits and their application circuits.	L1
CO 2	To understand and explain operational amplifier circuits and their application circuits.	L2
CO 3	To operate various operational amplifier based circuits	L3
CO 4	To compare and analyze operational amplifier circuits and their application circuits.	H1
CO 5	To design operational amplifier based oscillators, filters, comparators and converters.	H3

List of Experiments

1. Design and simulate PSpice model of inverting amplifier and obtain plots of input signal voltage versus time and output signal voltage versus time.
2. Design and simulate PSpice model of noninverting amplifier and obtain plots of input signal voltage versus time and output signal voltage versus time.
3. Design and simulate PSpice model of differential amplifier and obtain plots of input signal voltages versus time and output signal voltage versus time.
4. Design and simulate PSpice model of inverting amplifier with feedback and obtain plots of input signal voltage versus time and output signal voltage versus time.
5. Create and simulate PSpice model of inverting averaging circuit and measure output voltage.
6. Create and simulate PSpice model of noninverting summing amplifier circuit and measure voltage at inverting, noninverting and output terminals.
7. Create and simulate PSpice model of voltage to current converter with grounded load and measure voltage at inverting, noninverting and output terminals. Also measure load current.
8. Create and simulate PSpice model of second order low pass Butterworth filter and obtain plot of output voltage versus frequency.
9. Create and simulate PSpice model of second order high pass Butterworth filter and obtain plot of output voltage versus frequency.

10. Create and simulate PSpice model of square wave generator and obtain plots of capacitor voltage versus time and output signal voltage versus time.
11. Design and simulate PSpice model of noninverting comparator and obtain plots of input signal voltage versus time and output signal voltage versus time.
12. Design and simulate PSpice model of inverting comparator and obtain plots of input signal voltage versus time and output signal voltage versus time.
13. Simple project (Any topic related to the scope of the course).

NOTE: At least eight experiments are to be performed in the semester, out of which at least six experiments should be performed from above list. Remaining experiments may either be performed from the above list or designed & set by the concerned institution as per the scope of the syllabus.