



MARRI LAXMAN REDDY INSTITUTE OF TECHNOLOGY AND MANAGEMENT

(AN AUTONOMOUS INSTITUTION)

(Approved by AICTE, New Delhi & Affiliated to JNTUH, Hyderabad)

Accredited by NAAC with 'A' Grade & Recognized Under Section 2(f) & 12(B) of the UGC act, 1956

COURSE CONTENT

ELECTRONIC CIRCUIT ANALYSIS LABORATORY								
IV Semester: ECE								
Course Code	Category	Hours/ Week			Credits	Maximum Marks		
2540475	Core	L	T	P	C	CIA	SEE	Total
		0	0	2	1	40	60	100
Contact Classes: Nil	Tutorial Classes: Nil	Practical Classes: 32			Total Classes:32			
Prerequisites: Basic knowledge of electronic devices and circuits								

Couse Overview

The Electronic Circuit Analysis Laboratory is designed to provide hands-on experience in designing, building, and analyzing analog electronic circuits. It focuses on the practical implementation of amplifiers, oscillators, power amplifiers, multivibrators, and waveform generators using discrete components and simulation tools. The lab strengthens understanding of frequency response, gain, feedback, waveform shaping, and time base generation.

Course Objectives

The students will try to learn

- Design and test basic analog circuits like amplifiers, oscillators, and multivibrators using hardware components
- Analyze waveforms using instruments like CRO/DSO to understand real-time circuit behavior
- Simulation of analog circuits using tools like LTspice or Multisim and compare results with practical experiments
- Calculation of key performance parameters such as gain, efficiency, frequency response, and distortion
- Working of time base generators and multivibrators through both hardware and software experiments
-

Course Outcomes

After successful completion of the course, students shall be able to

- Design and analyze multistage and power amplifiers and evaluate their frequency response and efficiency
- Implement and examine feedback and oscillator circuits and validate theoretical conditions for sustained oscillations
- Develop and interpret waveform generation circuits such as multivibrators and time base generators

- Perform simulations to validate analog circuit performance using industry-standard software tools
- Correlate practical results with theoretical predictions and identify deviations due to real-world constraints

List of Experiments

A. Hardware Experiments (7)

Perform practical design, implementation, and waveform analysis of amplifiers, oscillators, power stages, and multivibrators to validate theoretical concepts and observe real-world circuit behavior.

1. Design and analyze a two-stage RC coupled amplifier to demonstrate gain enhancement and study coupling capacitance effects.
2. Design Hartley and Colpitts oscillators for a specified frequency and observe their output waveforms.
3. Design an RC phase shift oscillator and derive the practical gain condition for oscillations at a given frequency.
4. Design a transformer-coupled class A power amplifier, observe input/output waveforms, and calculate efficiency.
5. Design a class B power amplifier, analyze input/output waveforms, and evaluate harmonic distortion.
6. Design a bistable multivibrator, analyze commutating capacitor effects, and record transistor waveforms.
7. Design an astable multivibrator and observe transistor base and collector waveforms.

B. Software Simulations (7)

Use circuit simulation software to design, analyze, and verify the performance of feedback amplifiers, waveform generators, and power amplifier circuits through virtual experimentation and frequency response evaluation.

1. Simulate four feedback amplifier topologies and compare their frequency responses with and without feedback.
2. Simulate a monostable multivibrator and analyze its input/output waveforms.
3. Simulate a Schmitt trigger for gain values greater than and less than one and analyze response behavior.
4. Simulate a bootstrap time base generator using BJT and observe the output sweep waveform.
5. Simulate a Miller sweep circuit using BJT and observe the time base output waveform.
6. Simulate a complementary symmetry push-pull amplifier and verify elimination of crossover distortion.
7. Simulate a single tuned amplifier and determine the quality factor (Q) of its tuned circuit.

Open ended Experiments

1. Two Stage RC Coupled Amplifier
2. Cascade Amplifier circuit
3. RC Phase Shift Oscillator using Transistor

Software Requirements

Simulation Tools: LTspice / Multisim / PSpice / Proteus / NI Multisim Live or equivalent

Operating System: Windows or Linux (Ubuntu preferred)

Hardware Requirements:

1. Dual Power Supply ($\pm 15\text{V}$, 0–30V)
2. Function Generator (up to 1 MHz)
3. CRO / DSO (Dual Channel, 20 MHz or more)
4. Digital Multimeter
5. Breadboards and Connecting Wires
6. BJTs: BC107, BC547, BC557, 2N2222, etc.
7. Resistors, Capacitors (Wide range of values)
8. Transformers (for power amplifiers)
9. Inductors, Crystals (1 MHz, 4 MHz, etc.)
10. Heat sinks, transistors for power stages (e.g., TIP41, TIP42 etc.)

Reference Link

1. https://onlinecourses.nptel.ac.in/noc20_ee27/preview
2. https://onlinecourses.nptel.ac.in/noc24_ee81/preview
3. https://onlinecourses.nptel.ac.in/noc22_ee27/preview
4. https://onlinecourses-archive.nptel.ac.in/noc18_ee45/course

Materials Online

1. Lab Manual
2. Open-ended experiments