

INSTITUTE OF AERONAUTICAL ENGINEERING

(Autonomous) Dundigal - 500 043, Hyderabad, Telangana

COURSE CONTENT

ANALOG ELECTRONICS LABORATORY								
IV Semester: ECE								
Course Code	Category	Hours / Week			Credits	Maximum Marks		
AECD14	Core	L	Т	Р	С	CIA	SEE	Total
		-	-	2	1	40	60	100
Contact Classes: Nil	Tutorial Classes: Nil	Practical Classes: 45				Total Classes: 45		
Prerequisite: Electronic Devices and Circuits								

I. COURSE OVERVIEW:

This course provides hands-on experience in designing electronic circuits and pulse circuits using transistors. The course aims at practical experience with the characteristics and theoretical principles of linear and nonlinear devices and pulse circuits. It provides the capability to design and simulate amplifiers and wave-shaping circuits. Design power amplifiers, feedback amplifiers, clippers, and clampers, and end determine the gain, bandwidth of amplifiers, and calculation of distortion in power amplifiers.

II. COURSES OBJECTIVES:

The students will try to learn

- I. The design and analysis of single-stage and multistage amplifiers such as a basic amplifier, and two-stageamplifiers and determine the gain and bandwidth.
- II. Design and analysis of the transistor clipper and clamper circuits, also measure the voltage limits of bothbiases and unbiased clipping circuits.
- III. The concept of feedback and design feedback amplifiers, such as current series feedback amplifiers and RC phase shift oscillators.

III. COURSE OUTCOMES:

At the end of the course students should be able to:

- CO 1 Illustrate Bipolar Junction Transistor (BJT) amplifier circuits and their frequency response at low, mid, and high frequencies for determining amplifier characteristics.
- CO 2 Summarize the concept of feedback amplifiers for the distinction between negative and positive feedback.
- CO 3 Design the RC and LC Oscillators for a given frequency and compare the theoretical and practical frequency values.
- CO 4 Identify the suitable large signal amplifiers or power amplifiers for practical applications with given specifications.
- CO 5 Analyze the response of linear and non-linear wave shaping circuits for impulse and pulse inputs with different time constants.
- CO 6 Build bi-stable, monostable, and Astable multivibrator circuits using transistors for real-time applications.

EXERCISES FOR ANALOG ELECTRONICS LABORATORY

Note: Students are encouraged to bring their own laptops for laboratory practice session

1. Getting Started Exercises

1.1 Introduction to MULTISIM

Step 1: Open Multisim.Step 2: Place components.Step 3: Wire components.Step 4: Place a simulation source.Step 5: Place measurement instruments.Step 6: Run a simulation.

1.2 Frequency Response of Basic Amplifier

The basic amplifier is an electronic circuit which amplifies the given weak input signal over certain band of frequency range. The input signal is applied to base terminal of the transistor and amplifier output is taken across collector terminal is known as CE amplifier. The input signal is applied to emitter terminal of the transistor and amplifier output is taken across collector terminal is known as CB amplifier. Frequency response of an amplifier is defined as the variation of gain with respective frequency. To explore and understand basic amplifier design you need to perform the following practical exercises.

1.2.1 Simulate the frequency response of common emitter amplifier



Figure 1.2 Common emitter amplifier circuit diagram



1.2.2 Simulate the frequency response of common base amplifier

Figure 1.3 Common base amplifier circuit diagram

Try

- 1. Plot the frequency response of CE amplifier with different values input and output capacitors?
- 2. Design a CB amplifier with different biasing values, Summarize your comparisons with CE Amplifier? Justify your conclusion?
- 3. Determine the lower cutoff frequency for the basic CE amplifier with following parameters. $C_s = 10\mu F$, $C_E = 20\mu F$, $C_C = 1\mu F$, $R_s = 1K\Omega$, $R_1 = 40K\Omega$, $R_2 = 10K\Omega$, $R_E = 2K\Omega$, $R_C = 4K\Omega$, $R_L = 2.2K\Omega$.

2.Exercises on Current Series Feedback Amplifier

This laboratory continues the study of Feedback which plays a very important role in electronic circuits and the basic parameters, such as input impedance, output impedance, current and voltage gain and bandwidth, may be altered considerably using feedback for a given amplifier. A portion of the output signal is taken from the output of the amplifier and is combined with the normal input signal and thereby the feedback is accomplished. To explore and understand feedback amplifier design you need to perform the following practical exercises. 2.1 Simulate the Current series feedback amplifier without capacitor



Figure 2.1: Current series feedback amplifier without capacitor

2.2 Simulate the Current series feedback amplifier with capacitor



Figure 2.2: Current series feedback amplifier with capacitor

2.3 Simulate Current shunt feedback amplifier

Design a current shunt feedback amplifier in Figure 2.1. and draw the frequency response of respective design.



Figure 2.3: Current shunt feedback amplifier



2.4 Simulate Voltage Series feedback amplifier

Try

1. Derive the expression for the gain of a current series amplifier with and without feedback and implement practically for the different values of resistances and capacitances.

- 2. The input resistance of a current amplifier is 10 k Ω without feedback. If the feedback factor is 10 and the input resistance with feedback is 10 Ω , then what is the current gain with and without feedback?
- 3. Design the voltage shunt feedback amplifier and plot the frequency response.
- 4. The gain of a voltage amplifier decreased to 1000 with negative feedback from its gain of 5,000 without feedback. Calculate the feedback factor and express the amount of negative feedback in dB.
- 5. Derive the expression for Input Resistance and Output Resistance for a transconductance amplifier with and without feedback and comment on the results.

3. Exercises on RC Phase Shift Oscillator

RC phase shift oscillator has a CE amplifier followed by three sections of RC phase shift feedback networks. The output of the last stage is return to the input of the amplifier. The values of R and C are chosen such that the phase shift of each RC section is 60°. Thus, the RC ladder network produces a total phase shift of 180° between its input and output voltage for the given frequencies. Since CE amplifier produces 180° phase shift the total phase shift from the base of the transistor around the circuit and back to the transistor will be exactly 360°. To explore and understand oscillator design you need to perform the following practical exercises.

3.1 Design an RC phase oscillator with Resistance R =10 K\Omega and Capacitance C =10 μF and determine the frequency of oscillation.



Figure 3.1: RC Phase shift oscillator

- 1. Design an RC phase shift oscillator to produce an oscillation frequency of 2 K Hz and compare the theoretical and practical frequencies.
- 2. Design an RC phase shift oscillator with a target frequency of 1.5 K Hz and an amplitude of 5 Volts peak-to-peak.
- 3. Determine the individual and total phase shift introduced by each stage for a given RC phase shift oscillator for a frequency of 3.5 K Hz.

4.Exercises on Linear and Non-Linear Wave Shaping Circuits

A Signal can also be called a wave. Every wave has a certain shape when it is represented in a graph. This shape can be of different types such as sinusoidal, square, triangular, etc. which vary concerning period or they may have some random shapes disregarding of the period. Linear elements such as resistors, capacitors, and inductors are employed to shape a signal in this linear wave shaping. A Sine wave input has a sine wave output and non-sinusoidal inputs are used to understand the linear wave shaping.

Along with resistors, the non-linear elements like diodes are used in nonlinear wave shaping circuits to get required altered outputs. Either the shape of the wave is attenuated or the dc level of the wave is altered in the Non-linear wave shaping. The process of producing non-sinusoidal output wave forms from sinusoidal input, using non-linear elements is called as nonlinear wave shaping.

To explore and understand the linear and non-linear wave shaping circuits you need to perform the following practical exercises.

4.1 Design RC low pass and high pass circuit for different constants.



Figure 4.2: RC Differentiator or High pass filter

4.2 Design transfer characteristics of clippers and clampers.

1. Positive Clipper



2. Positive Clipper with positive reference level



3. Positive Clipper with negative reference level



Figure 4.3: Non-Linear Wave Shaping Circuits - Clippers

1. Positive Clamping circuit



2. Positive Clamping circuit with negative reference voltage



3. Positive Clamping circuit with positive reference voltage



Figure 4.4: Non-Linear Wave Shaping Circuits - Clampers

Try

- 1. Design an RC low pass filter circuit to attenuate frequencies above 2 KHz with R = 1 KΩ, C= 100 nF
- 2. Determine the cutoff frequency of RC high pass filter with resistance R= 2.2 K Ω and capacitance C= 220 nF.
- 3. Design an RLC band-pass filter circuit with a center frequency of 1.5 K Hz and a bandwidth of 500 Hz, for a given R= $1.5 \text{ K}\Omega$, Inductor L = 10 mH, and capacitor C= 100 nF.
- 4. For a negative clamping circuit, what is the minimum peak value of the output waveform if the input waveform is 10V square wave with switching time of 1 second? Assume that the input switches between +10V and -10V DC levels.
- 5. For a positive clamping circuit with negative reference voltage of 10 V, find the peak value of the output. Assume that the input voltage is a sinusoidal waveform with 40 V_{p-p} .

5. Exercises on UJT as a Relaxation Oscillator

A Uni junction Transistor (UJT) relaxation oscillator is a circuit that uses a UJT to generate a periodic waveform, typically a sawtooth or pulse waveform. The UJT is a three-layer, two-terminal semiconductor device. In a relaxation oscillator circuit, the UJT switches between its high-resistance and low-resistance states, causing a charging and discharging cycle of a capacitor, which results in an oscillating output. To explore and understand power amplifier design you need to perform the following practical exercise.

5.1 Design UJT as a Relaxation Oscillator.



Figure 5: UJT as a Relaxation Oscillator

- 1. A UJT relaxation oscillator circuit has the following component values: R1=75 kohms C1=68 uf, R2=1.8k ohms , and R3=150 ohms The UJT's standoff ratio is given as n = 0.73 . Calculate for the oscillation frequency (in Hz).
- 2. Design a UJT relaxation oscillator to generate a sawtooth waveform at a frequency of 500Hz. Assume the supply voltage $V_{BB} = 20V$, $V_{\nu} = 2.9V$, $V_{\nu} = 1.118V$, $I_p = 1.6mA$ and $I_{\nu} = 3.5mA$

6. Exercises on Multivibrators

Multivibrator circuit is a switching circuit. It generates non-sinusoidal waves such as square waves, rectangular waves and saw tooth waves etc. Multivibrators are used as frequency generators, frequency dividers and generators of time delays and also as memory elements in computers etc. A Transistor basically functions as an amplifier in its linear region. If a transistor amplifier output stage is joined with the previous amplifier stage, such a connection is said to be coupled. If a resistor is used in coupling two stages of such an amplifier circuit, it is called as Resistance coupled amplifier. To explore and understand multivibrator design you need to perform the following practical exercises.

6.1 Simulate Astable multivibrator and plots its waveforms



Figure6.1: Astable Multivibrator





Figure 6.2: Monostable Multivibrator

6.3 Simulate Bistable multivibrator and plots its waveforms



Figure 6.3: Bistable Multivibrator

Try

- 1. An Astable Multivibrators circuit is required to produce a series of pulses at a frequency of 500Hz with a mark-to-space ratio of 1:5. If R2 = R3 = $100k\Omega$, calculate the values of the capacitors, C1 and C2 required.
- 2. An Astable Multivibrator circuit is constructed using two timing capacitors of equal value of 3.3 uF and two base resistors of value $10 \text{k}\Omega$. Calculate the minimum and maximum frequencies of oscillation if a $100 \text{k}\Omega$ dual-gang potentiometer is connected in series with the two resistors.
- Design an astable multivibrator to generate 5kHz square wave with a duty cycle of 40% and if amplitude 12V. Use NPN transistor having h_{fe} = 100, V _{BE}(sat) = 0.7V, VCE(Sat) = 0.2, I_C(max) = 100mA.

7. Exercises on Sampling Gates

A sampling gate is a transmission circuit that faithfully transmits an input signal to the output for a finite time duration which is decided by an external signal, called a gating signal (normally rectangular in shape). The input appears without a distortion at the output, but is available for a time duration T and afterwards the signal is zero. They can transmit a greater number of signals. The main applications of the sampling gates are multiplexers, choppers, D/A converter, sample and hold circuits, etc.

An ideal sampling gate is a transmission circuit that produces an output signal identical to the input signal during a selected time interval. The output of the sampling gate is zero outside this selected time interval. The sampling gate is open during the sampling interval and it is closed at all other times. The time interval for transmission is monitored by a control input signal, which is usually rectangular in shape. In practice the idealized transmission gate is not realized. As long as the output is produced at the correct time the performance of the practical sampling gates available is treated to be quite satisfactory.

7.1 Design a Unidirectional Diode Gate



Figure 7.1: Unidirectional Diode Gate

Try

1. In the circuit shown below, $RL = R1 = 100 \text{ k}\Omega$, $R2 = 50 \text{ k}\Omega$ and the signal has a peak value of 20 V. Find (a) A (b) V_c(min) (c) V_n(min) (d)Ri when the diodes are ON



2. For the four-diode gate shown in Fig. 11.20(a), $RL = R2 = 100 \text{ k}\Omega$ and $R1 = 1 \text{ k}\Omega$, $Rf = 25 \Omega$, Vs = 20 V. Calculate (a) A (b) V(min) (c) VC(min) (d) Vn(min) for V = V(min)



8. Exercises on Single Tuned Amplifier

Tuned amplifiers are used to for the amplification of a specific frequency signal or a narrowband frequency signal. Basically, high frequency or radio frequency signals are amplified using tuned amplifiers. Tuning (i.e., selecting) of frequency is done by using a tuned or resonant circuit at the load. To explore and understand tuned amplifier design you need to perform the following practical exercises.

8.1 Simulate a single tuned circuit as shown in Figure 8.1.



Figure 8.1: Single tuned amplifier

Try

- 1. Derive the expression for the Q-factor for the circuit shown in Figure 5.1
- 2. For the circuit shown in Figure 4.1 if the L1 = 1 mH and C2 = 500 pF the find (a) the resonant frequency and (d) d.c load resistance and a.c load resistance.
- 3. The Q of a tuned amplifier is 100. If the resonant frequency for the amplifier is 1000 kHz, find (i) bandwidth and (ii) cut-off frequencies.
- 4. A parallel resonant circuit has a capacitor of 250 pF in one branch and inductance of 1.25 mH plus a resistance of 10 Ω in the parallel branch. Find (i) resonant frequency (iii) Q-factor of the circuit.
- 5. A tuned amplifier has its maximum gain at a frequency of 1 MHz and has a bandwidth of 10 KHz. calculate the Q-factor.

9. Exercises on Two Stage RC Coupled Amplifier

As the gain provided by a single stage amplifier is usually not sufficient to drive the load, so to achieve extra gain multi-stage amplifier are used. In multi-stage amplifiers output of one- stage is coupled to the input of the next stage. The coupling of one stage to another is done with the help of some coupling devices. If the circuit is coupled by RC to next stage, it is called RC coupled amplifier as shown in figure 9.1.

9.1 Simulate a Two stage RC coupled amplifier



Figure 9.1: Two stage RC coupled amplifier

Try

- 1. Obtain the expressions for voltage gain at middle and low frequency for two stage RC-coupled amplifier.
- 2. Obtain the expression for CE current gain with R_L and explain the variation of frequency response with R_L using hybrid- model.
- 3. Explain how Bandwidth and unity gain frequency (f_T) of a BJT can be determined to obtain the expression for the Gain Bandwidth product of a transistor.
- 4. A multistage amplifier has two stage gains of 10 and 100. Then what will be the overall voltage gain theoretically and practically? and what is the effect on the bandwidth of two stage amplifier?
- 5. Draw the circuit connection of multistage amplifier using 1st stage as CE amplifier and 2nd stage as CB amplifier to get the maximum benefit in terms of the voltage gain, input, and output impedance. Also obtain the expressions for voltage gain, input resistance and output resistance.

10. Exercises on LC Oscillators

The Hartley oscillator is an electronic oscillator circuit in which the oscillation frequency is determined by a tuned circuit consisting of capacitors and inductors. The Hartley oscillator consists of a single capacitor in parallel with two inductors in series. The feedback signal needed for oscillation is taken from the center connection of the two inductors. In the case of Colpitts oscillator the feedback for the active device is taken from a voltage divider made of two capacitors in series across the inductor. The frequency of oscillation is approximately the resonant frequency of the LC circuit, which is the series combination of the two capacitors in parallel with the inductor.

10.1 Simulate sine wave generated for a particular frequency by Hartley oscillator

Design an Hartley oscillator with inductance $L_1 = L_2 = 1$ mH and $C = 0.01 \mu$ F and determine the frequency of oscillation as shown in the Figure 10.1.



Figure 10.1: Hartley oscillator

10.2 Simulate sine wave generated for a particular frequency by Colpitts oscillator

Design Colpitts oscillator with $C_1 = C_2 = 100 \ \mu$ F and L = 10 mH and determine the frequency of oscillation as shown in Figure 11.2.



Figure 10.2: Colpitts oscillator

- 1. Determine the values of inductors and capacitors for each stage, aiming for an oscillation frequency of 6 M Hz and discuss the advantages of using multiple stages in the design.
- 2. Design a Hartley oscillator for a frequency of 2.8 M Hz and calculate the temperature coefficient for the designed oscillator.
- 3. Design a Colpitts oscillator with an initial frequency of 3.5 M Hz and determine the new values of the capacitor to adjust the frequency to 4 M Hz.
- 4. Modify an existing Colpitts oscillator with an initial frequency 1.8 M Hz, C1 = 100 pF, C2 = 68 pF, L = 120 μ H. Determine the new capacitor values to achieve a frequency range of 1.5 MHz to 2.2 MHz.

11. Exercises on Differential Amplifier

The typical BJT differential pair amplifier consists of a pair of transistors coupled at the emitters to a current source, having equal resistances in each collector and equal but opposite, signal sources in each base. The amplifier has several variations on this basic configuration. The basic configuration (Figure 11-1) will be studied in this experiment. The important characteristics for the differential pair amplifier to be studied in this experiment are: differential voltage gain (A_{Vd}), common mode gain (A_{Vcm}), common-mode rejection ration CMRR, and single-ended voltage gain (AVse). The common mode gain is found by applying the input signal to both the (+) and (-) inputs to the differential pair. The differential output voltage with a common mode input is zero (Figure 11-2).

11.1 Simulate Basic BJT Differential Pair Amplifier



Figure 11.1: Basic BJT Differential Pair Amplifier

11.2 Simulate Common mode circuit for differential pair



Fig 11.2: Common mode circuit for differential pair

- 1. Design the basic BJT differential amplifier to provide a differential input resistance of at least 10 K Ω and a differential voltage gain of 100 V/V. The transistor β is specified to be at least 100. The available positive power supply is 5V.
- 2. A BJT differential amplifier is biased from a 1mA constant--current source and includes a 200Ω resistor in each emitter. The collectors are connected to Vcc via $12k\Omega$ resistors. A differential input signal of 0.1V is applied between the two bases.

(a) Find the signal current in the emitters (io) and the signal voltage vbe for each BJT

- (b) What is the total emitter current in each BJT?
- 3. Design a BJT differential amplifier that provides two single-ended outputs (at the collectors). The amplifier is to have a differential gain (to each of the two outputs) of at least 100 V/V, a differential input resistance ≥10kΩ and a common mode gain (to each of the two outputs) no greater than 0.1 V/V. Use a 2mA current source for biasing.

12. Exercises on Transistor as A Switch

One of the most common uses of transistors in an electronic circuit is as simple switches. In short, a transistor conducts current across the collector-emitter path only when a voltage is applied to the base. When no base voltage is present, the switch is OFF. When base voltage is present, the switch is ON. To explore and understand switching activity of a transistor you need to perform the following practical exercise.

12.1 Design transistor as a switch



Figure 12.1: Transistor as a Switch

Try

- 1. For the BJT, $\beta = \infty$, V_{BE} on = 0.7V V_{CE} sat = 0.7V. The switch is initially closed. At t=0, it is opened. At which time the BJT leaves the active region?
- 2. Explain the function of this light-switching circuit, tracing the directions of all currents when the switch closes:



3. Trace the directions of all currents in this circuit, and determine which current is larger: the current through resistor R1 or the current through resistor R2, assuming equal resistor values.



13. Exercises on Comparator

A comparator circuit compares two voltages and outputs either a 1 (the voltage at the plus side, VDD in the illustration) or a 0 (the voltage at the negative side) to indicate which is larger. Comparators are often used, for example, to check whether an input has reached some predetermined value. To explore and understand comparator design you need to perform the following practical exercises.

13.1 Design the following comparator circuits



Figure 13: Comparator

Try

- 1. Generate the output waveform for reference voltage 2v and input 4v p-p.
- 2. Modify the design of your comparator with a positive feedback network to add a total hysteresis of approximately 0.2 V.
- 3. Design a comparator that will switch when the input signal crosses +2.5 V with no hysteresis.

14. Exercises on Schmitt Trigger

A Schmitt trigger is a type of electronic circuit that converts a noisy input signal into a clean digital output signal. It's commonly used in digital electronics for applications such as signal conditioning, noise filtering, and waveform shaping. The Schmitt trigger has hysteresis, meaning it has two different voltage thresholds for the rising and falling edges of the input signal. To explore and understand Schmitt trigger design you need to perform the following practical exercise.

14.1 Design a Schmitt trigger circuit.



Figure 14: Schmitt Trigger

Try

- 1. Given a Schmitt trigger circuit with positive and negative threshold voltages of 3 V and 2 V, respectively, and a supply voltage of 10 V, calculate the hysteresis window.
- 2. For a Schmitt trigger circuit with positive and negative threshold voltages of 4 V and 2 V, respectively, draw the voltage transfer characteristic curve. Assume an ideal op-amp and provide the key points on the curve.
- 3. Design a Schmitt trigger circuit to have V_{cc} = 12V, UTP=6V, LTP=3V, using two silicon npn transistors with $h_{fe} = 60$.

V. TEXT BOOKS:

- 1. Jacob Millman, Electronic Devices and Circuits, Tata McGraw Hill Education, 3rd edition, 2014.
- 2. Jacob Millman, Herbert Taub, Mothiki S. PrakashRao, "Pulse Digital and Switching Waveforms", TataMcGraw-Hill, 3rd edition, 2008.

VI. REFERENCE BOOKS:

- 1. Robert L. Boylestead, Louis Nashelsky, Electronic Devices and Circuits Theory, Pearson, 11th edition,2009.
- 2. David A. Bell, "Solid State Pulse Circuits", PHI, 4th edition, 2002

VII. ELECTRONICS RESOURCES:

- https://archive.org/details/ElectronicDevicesCircuits
 https://www.electronics-tutorials.ws/

VIII. MATERIALS ONLINE 1. Course template

- 2. Lab Manual