INSTITUTE OF AERONAUTICAL ENGINEERING
(Autonomous)
Dundigal - 500 043, Hyderabad, Telangana

COURSE CONTENT

ANALOG AND PULSE CIRCUITS LABORATORY								
III Semester: ECE								
Course Code	**Category**	**Hours / Week**			**Credits**	**Maximum Marks**		
		L	**T**	**P**	**C**	**CIA**	**SEE**	**Total**
AECC13	**Core**	0	0	3	1.5	30	70	100
Contact Classes: Nil	**Tutorial Classes: Nil**	**Practical Classes: 45**				**Total Classes: 45**		
Prerequisite: There are no prerequisites to take this course								

I. COURSE OVERVIEW:

This experimental laboratory course explores the design and simulate of analog electronic circuits. This course builds on the courses electronic circuit analysis and pulse and digital circuits which are mandatory for all students of electronics and communication engineering. The course aims at practical experience with the characteristics and theoretical principles of linear and non- linear devices and pulse circuits.

II. COURSES OBJECTIVES:
The students will try to learn

    I.   Single stage and multi stage amplifiers and oscillators.
    II. The principles of feedback amplifiers and oscillators through simulation.
    III. The operations of circuits for linear and nonlinear wave shaping.
    IV. The characteristics of different multivibrators

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: Interpret the expressions to find frequency of oscillations for RC and LC type Oscillator circuits.

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 multi vibrator circuits using transistors for real time applications.

EXERCISES FOR ANALOG AND PULSE CIRCUITS LABORATORY

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

1. Getting Started Exercises

Introduction to MULTISIM and Digilent analog discovery kit

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

The Digilent Analog Discovery 2, developed in conjunction with Analog Devices, is a multi-function instrument that allows users to measure, visualize, generate, record, and control mixed signal circuits of all kinds. The low-cost Analog Discovery 2 is small enough to fit in your pocket, but powerful enough to replace a stack of lab equipment, providing engineering students, hobbyists, and electronics enthusiasts the freedom to work with analog and digital circuits in virtually any environment, in or out of the lab.
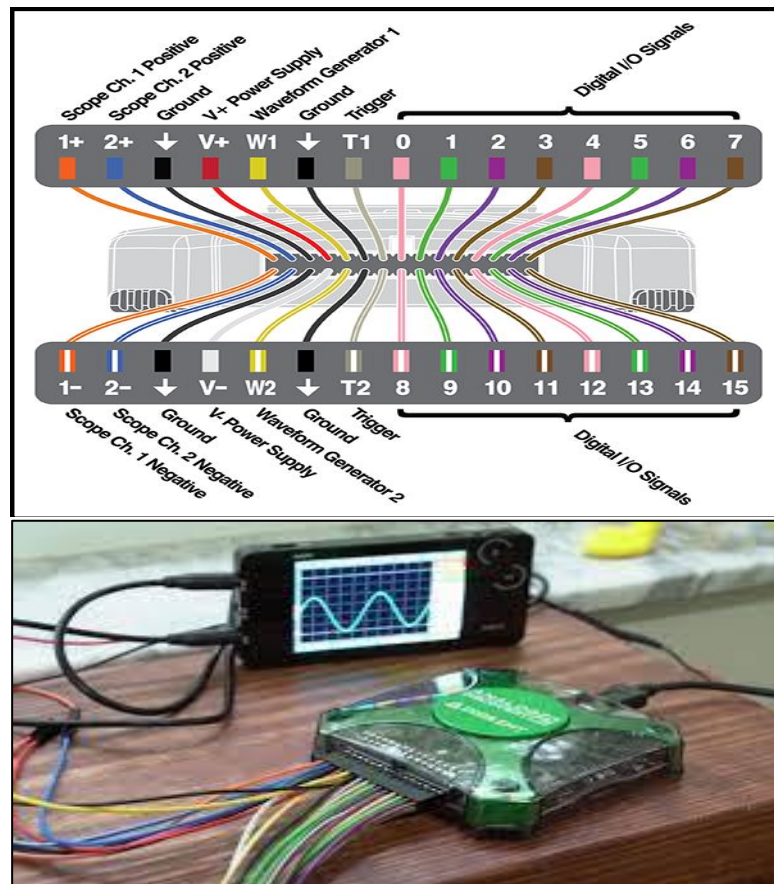

Figure 1.1: Digilent Analog discovery kit

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 the basic amplifier

1.1 Simulate the common emitter amplifier as shown in figure 1.2 using MULTISIM. Draw and analyze the gain versus frequency graph for CE amplifiers.
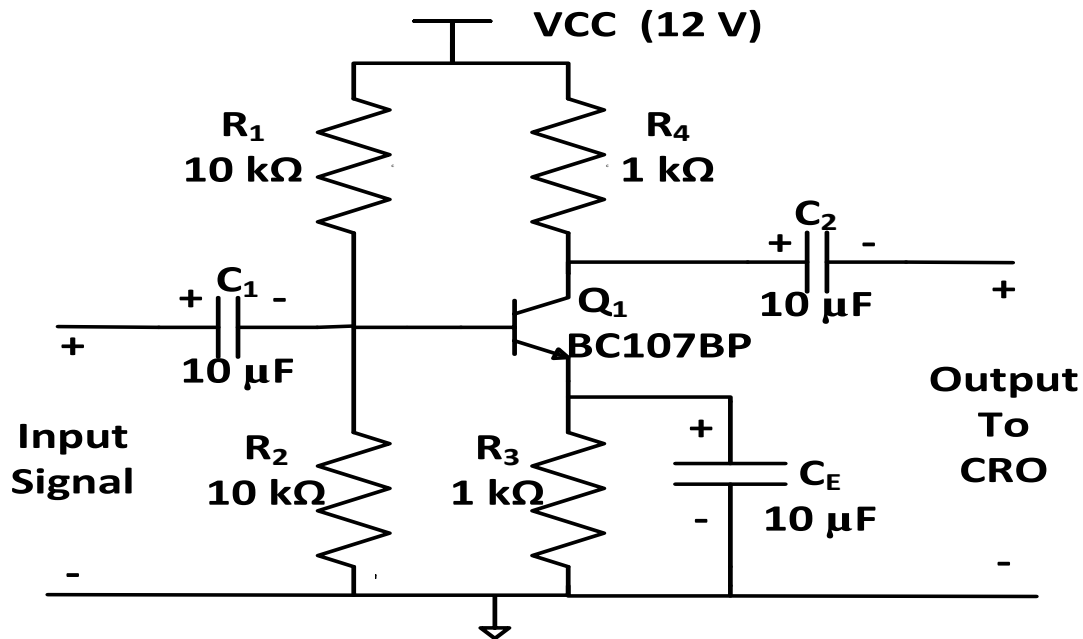
Figure 1.2 Common emitter amplifier circuit diagram

1.2 Simulate the common base amplifier as shown in figure 1.3 using MULTISIM. Draw and analyze the gain versus frequency graph for CB amplifiers.
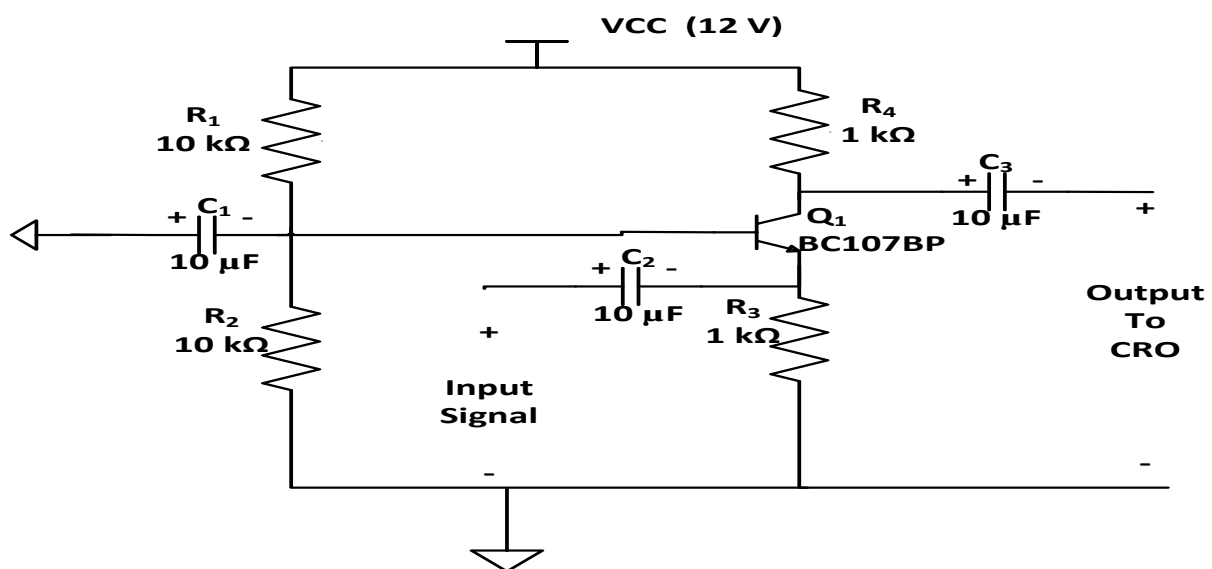

Figure 1.3 Common base amplifier circuit diagram

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 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. The process whereby the form of a non-sinusoidal signal is altered by transmission through a linear network is called linear wave shaping. To explore and understand the linear wave shaping circuits design and perform the following practical exercise using digilent analog discovery kit.

2.1 Design RC low pass and high pass circuit and draw the gain versus frequency graph as shown in figure 2.1 and 2.2.
2.2 Design RC low pass and high pass circuit for different constants for step and square input as shown in figure 2.1 and 2.2 and draw the waveform input and output waveforms.
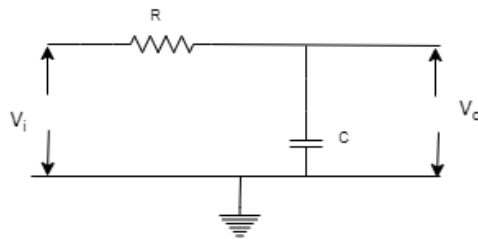

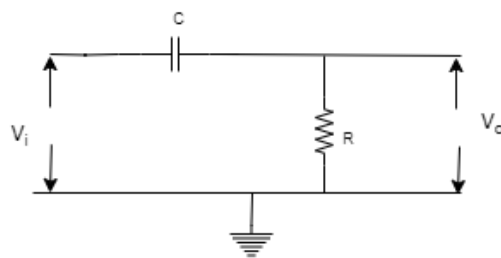Figure 2.1: RC Integrator or Low pass filter

Figure 2.2: RC Differentiator or High pass filter

Try
1. Design RC low pass and high pass circuit for different constants for step and square input as shown in figure 2.1 and 2.2 and draw the waveform input and output waveforms.
2. Determine the cutoff frequency of RC high pass filter with resistance R= 2.2 KΩ 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 KΩ, Inductor L = 10 mH, and capacitor C= 100 nF.

3. 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 is called RC coupled amplifier. To explore and understand the improvement in the gain design the following two stage amplifier.

3.1 Simulate the two stage RC coupled amplifier as shown in figure 3.1 using multisim. Draw the gain versus frequency graph for RC coupled amplifier as shown in figure 3.1.
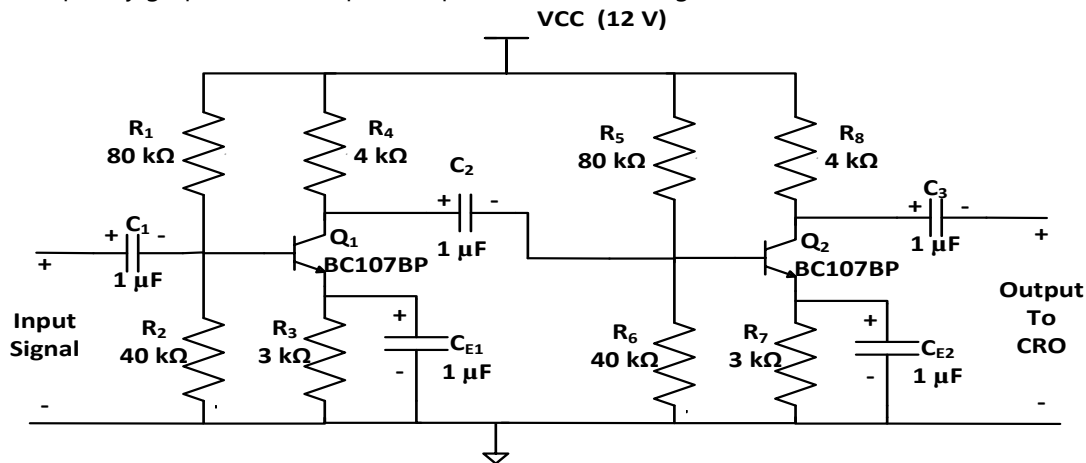

Figure 3.1: Two stage RC coupled amplifier

Try

1. Design and draw the gain versus frequency graph for two stage RC-coupled amplifier at middle and low frequency.

2. Draw the waveform for 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. And obtain the expressions for voltage gain, input resistance and output resistance.

3. A multistage amplifier has two stage gains of 10 and 100. Then what will be the overall voltage gain theoretically and practically from Figure 3 and what is the effect on the bandwidth of two stage amplifier?

4. Exercises on Non-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 no-linear wave shaping circuits perform the following circuits.

4.1 Design and check the transfer characteristics of clippers and clampers as shown in figure 4.1 and 4.2 using digilent analog discovery kit.

1. Positive Clipper
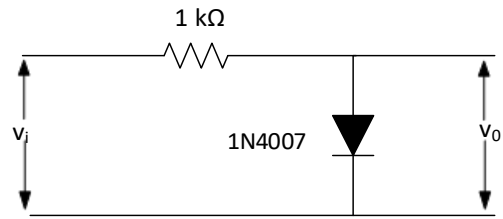
2. Positive Clipper with positive reference level
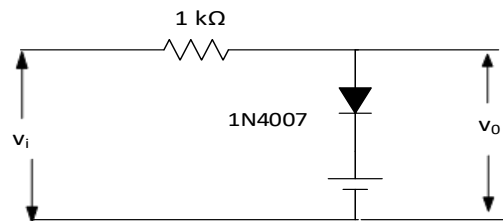
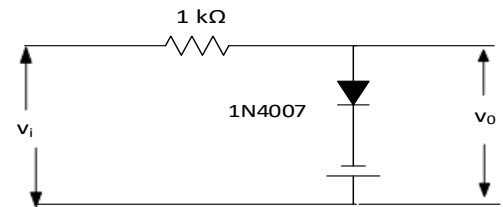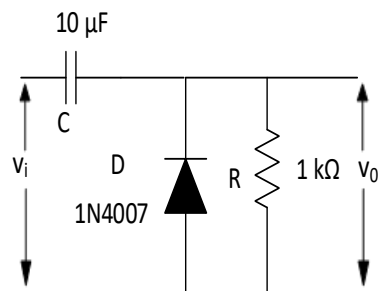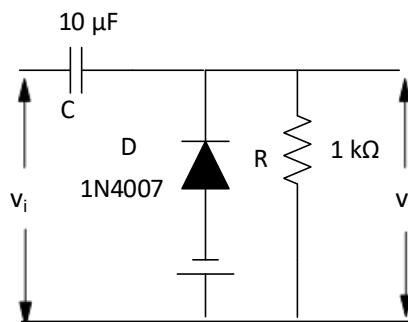
3. Positive Clipper with negative reference level

Figure 4.1: Non-Linear Wave Shaping Circuits - Clippers

1. Positive Clamping circuit

2. Positive Clamping circuit with negative reference voltage

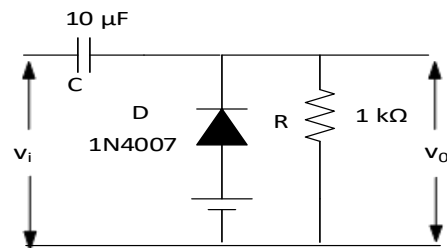3. Positive Clamping circuit with positive reference voltage

Figure 4.2: Non-Linear Wave Shaping Circuits - Clampers

Try

1. Design a series clipper at two independent level clipper as shown in figure with R=1KΩ as shown in figure 4.3.
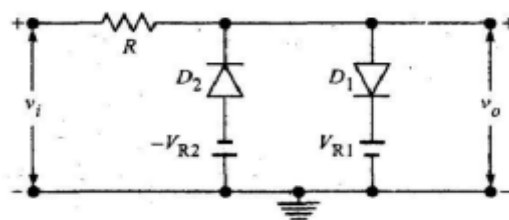

Figure 4.3 Series two level clipper

2. 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 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 perform the following practical exercises.

5.1 Simulate a single tuned amplifier circuit as shown in Figure 5.1 using MULTISIM software. Draw and analyze the waveform for gain versus frequency of single tuned amplifier.
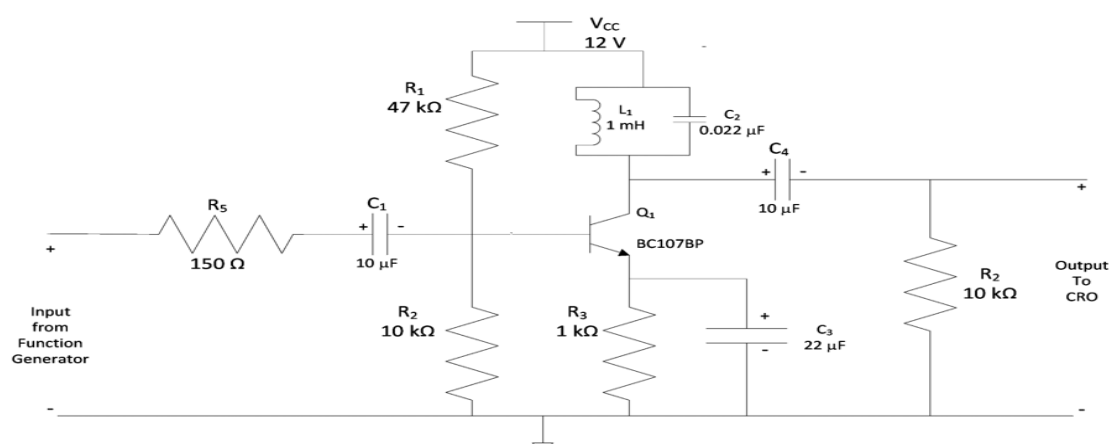

Figure5.1: Single tuned amplifier

Try

1. For the circuit shown in Figure5.1 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.

2. 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.
3. 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.
4. A tuned amplifier has its maximum gain at a frequency of 1 MHz and has a bandwidth of 10 KHz. calculate the Q-factor.

6. 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 need to perform the following practical exercise.

6.1 Design the transistor as a switch as shown in figure 6.1 using digilent discovery kit. Draw the input and output wave form for give sinusoidal input
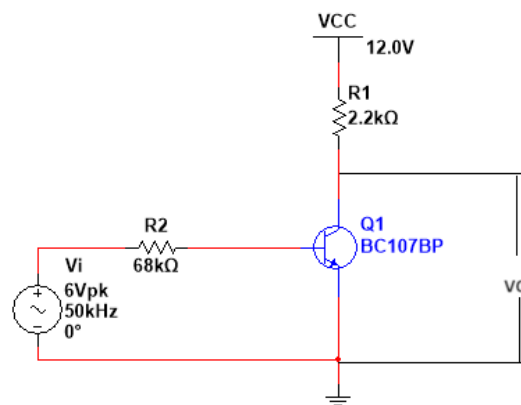

Figure 6.1: Transistor as a Switch

Try

1. Observe the function of this light-switching circuit, tracing the directions of all currents when the switch closes as shown in figure 6.2
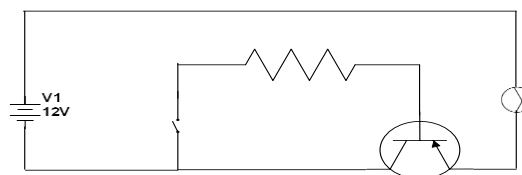

Figure 6.2 Light switching circuit

2. 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 as shown in figure 6.3.
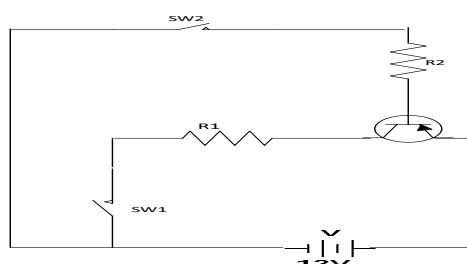

Figure 6.3 Switching circuit

7. Feedback Amplifiers

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 perform the following practical exercises.

7.1 Simulate a current shunt feedback amplifiers as shown in Figure 7.1 using MULTISIM. Draw the frequency response of current shunt feedback amplifiers.
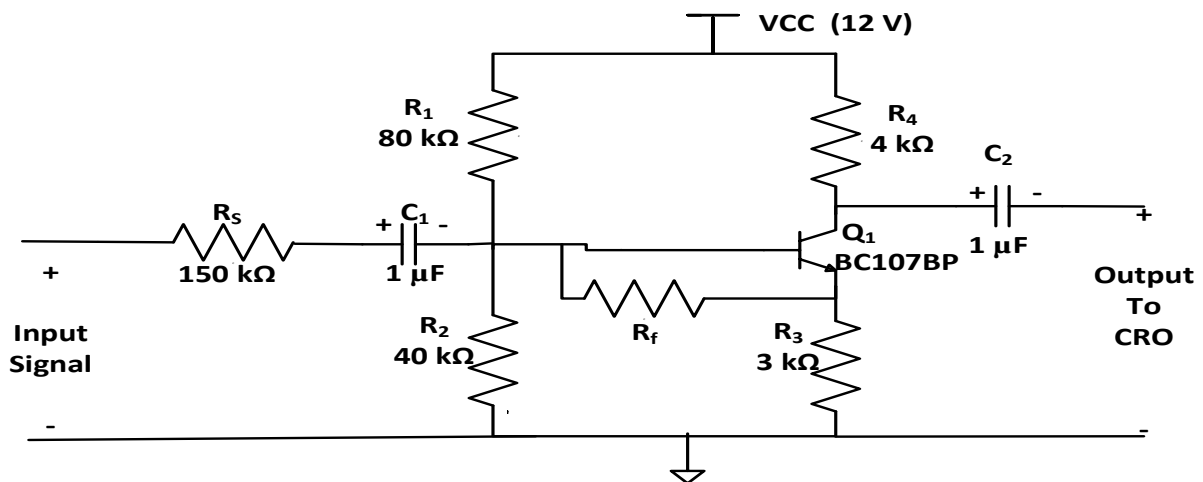

Figure7.1: Current shunt feedback amplifier

7.2 Simulate a voltage series feedback amplifiers as shown in Figure 7.2 using MULTISIM. Draw the frequency response of voltage series feedback amplifiers.
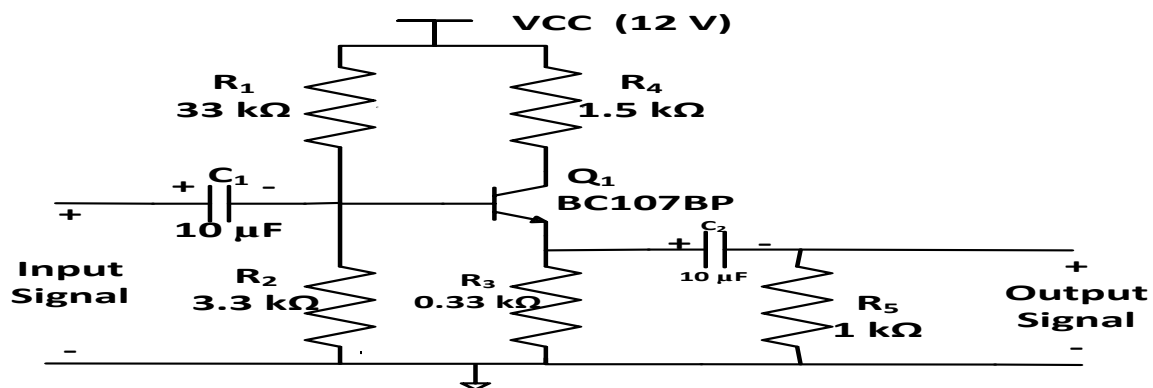

Figure7.2: Voltage series feedback amplifier

Try
1. Design the voltage shunt feedback amplifier and plot the frequency response.
2. 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.
3. Derive the expression for Input Resistance and Output Resistance for a transconductance amplifier with and without feedback and comment on the results.

8. 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 need to perform the following practical exercises as shown in figure8.

8.1 Design a comparator circuit as shown in figure 8.1 using digilent analog discovery kit and draw the output waveform.
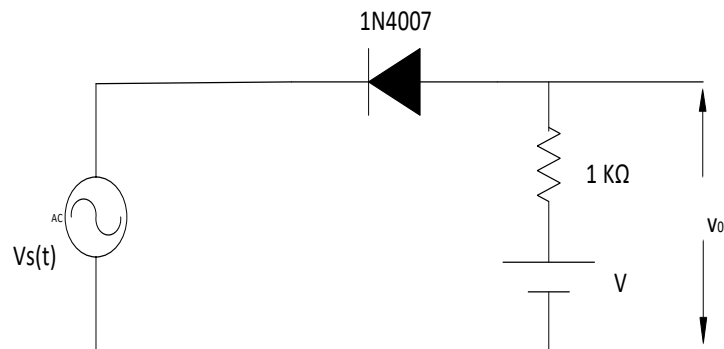

Figure 8.1 : Circuit diagram for Comparator

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.

9. 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 600. Thus, the RC ladder network produces a total phase shift of 1800 between its input and output voltage for the given frequencies. Since CE amplifier produces 1800 phase shift the total phase shift from the base of the transistor around the circuit and back to the transistor will be exactly 3600. To explore and understand oscillator design and need to perform the following practical exercises.

9.1 Design an RC phase oscillator with Resistance R1=Rc=10 KΩ, R_E=1 KΩ,R =10 KΩ, C_E=C_C=1 μF and C =10 μF for the circuit shown in figure.9 and determine the frequency of oscillation of the circuit.
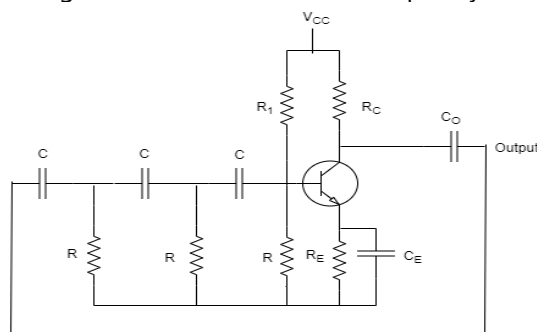

Figure 9: RC Phase shift oscillator

Try

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.

10. 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 need to perform the following practical exercises.

10.1 Simulate an astable multivibrator as shown in figure 10.1 using MULTISIM and plots output waveforms.
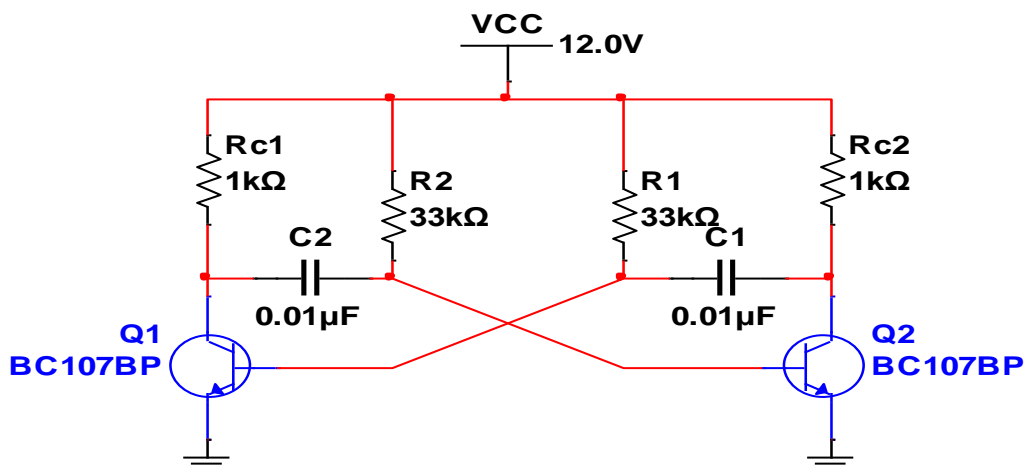

Figure10.1: Astable Multivibrator

10.2 Simulate monostable multivibrator as shown in figure 10.2 using MULTISIM and plots output waveforms.
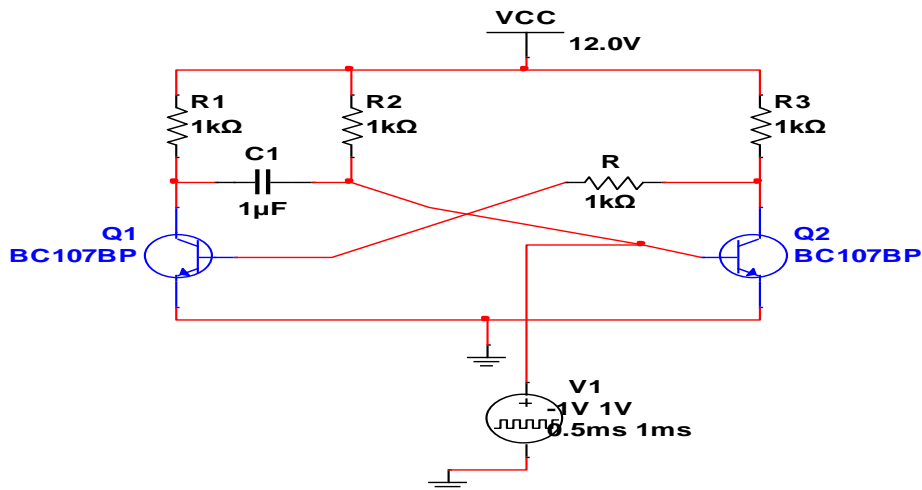
Figure10.2: Monostable Multivibrator

10.3 Simulate bistable multivibrator as shown in figure 10.3 using MULTISIM and plots output waveforms.
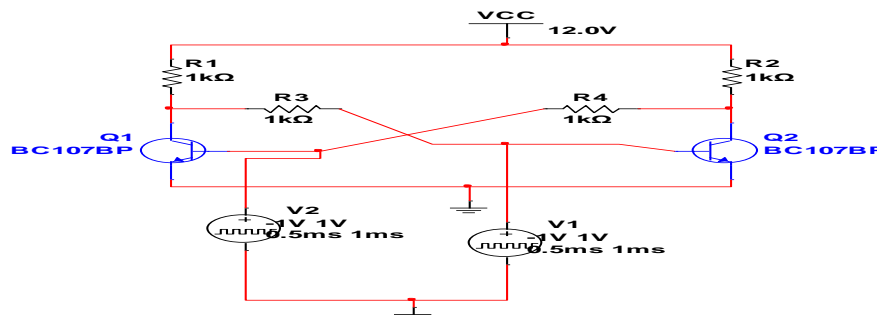

Figure10.3: Bistable Multivibrator

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 = 100kΩ, 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 3.3uF and two base resistors of value 10kΩ. Calculate the minimum and maximum frequencies of oscillation if a 100kΩ dual-gang potentiometer is connected in series with the two resistors.
3. 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.

11. Exercises on Hartley Oscillator and Colpitts Oscillator

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.

11.1 Design a Hartley oscillator as shown in the Figure 11.1 and determine the frequency of oscillation using MULTISIM
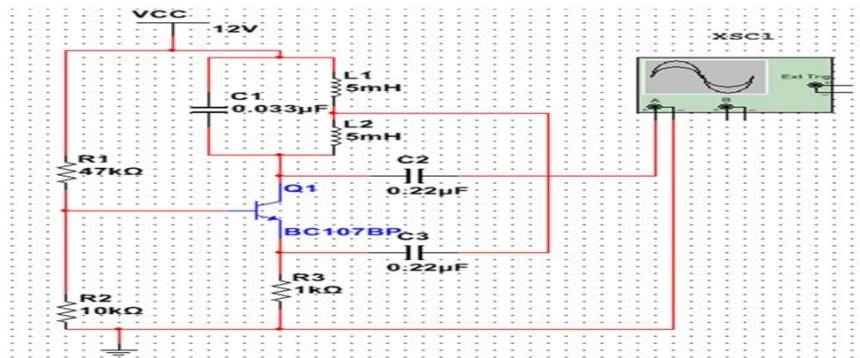
Figure 11.1: Hartley oscillator

11.2 Design Colpitts oscillator as shown in Figure 11.2 and determine the frequency of oscillation in MULTISIM
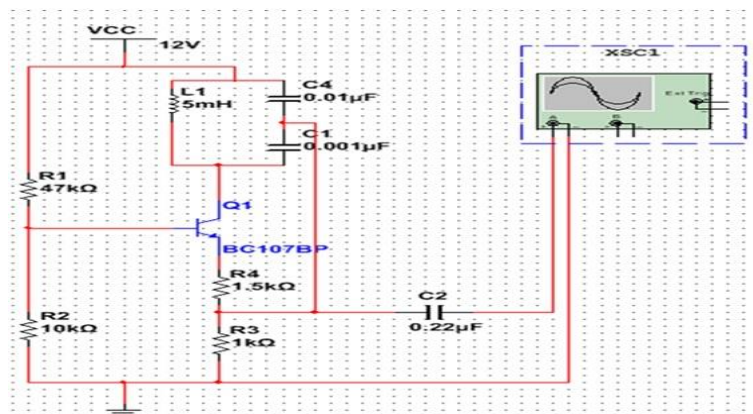

Figure 11.2: Colpitts oscillator

Try

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.

12. 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 the following practical exercise as shown in figure12.

12.1 Design schmitt trigger with the following parameters Rc1=6.8KΩ, RC2=3.9kΩ, R1=R2=2.2kΩ, Re=3.3kΩ, Rs=100Ω using digilent analog discovery kit
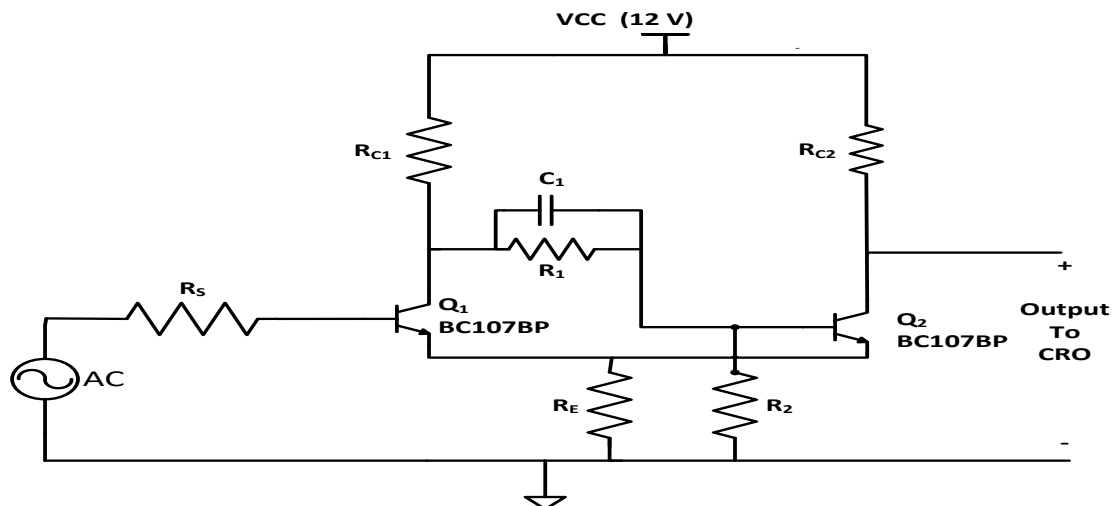
Figure 12.1 : Circuit diagram for Schmitt Trigger

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 $h_{fe} = 60$.

13. Exercises on Power Amplifier

Power amplifiers are mainly used to deliver more power to the load. To deliver more power it requires large input signals, so generally power amplifiers are preceded by a series of voltage amplifiers. In class-A power amplifiers, the Q-point is located in the middle of the DC-load line. In the class B amplifier, the Q-point is below the active region. To explore and understand power amplifier design the following exercise using figure13.1 and 13.2.

13.1 Simulate class A power amplifier (transformer less) as shown in figure 13.1 using MULTISIM.

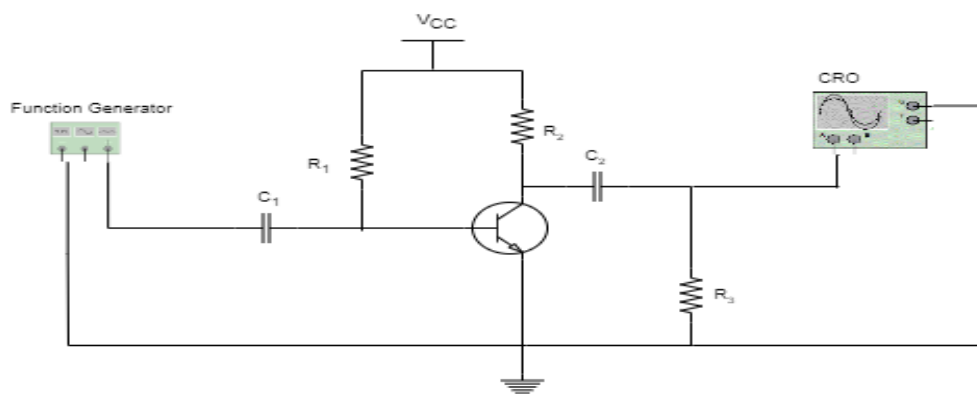
Figure13.1: Class A power amplifier

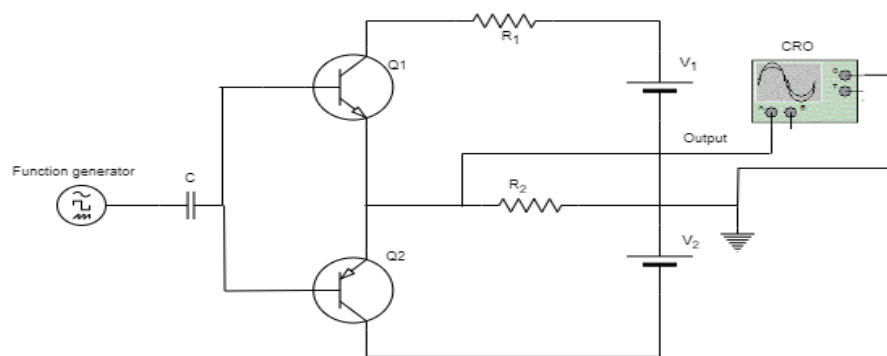13.2 Simulate class B power amplifier as shown in figure 13.2 using MULTISIM

Figure 13.2 Class B power amplifier

Try

1. Design a Class A power amplifier with a collector current of 200 mA, assuming a silicon NPN transistor with β = 100, V_{CC} = 15 V, V_{CEQ} = 7.5 V.
2. Given a Class A power amplifier with VCC = 20 V, RC = 500 Ω and IC = 100 mA, draw the load line on the output characteristics curve and determine the Q-point.
3. Design a Class B push pull amplifier with VCC = 22 V, RC = 250 Ω, and IC = 120 mA per transistor and calculate the power dissipation, and power delivered to the load.
4. Determine the efficiency of a Class B power amplifier with VCC = 24 V, RC = 200 Ω and IC = 250 mA per transistor.

14. 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 relaxation oscillator design the following practical exercise as shown in figure14.

14.1 Design the ujt relaxation oscillator as shown in figure 14 using digilent analog discovery kit.
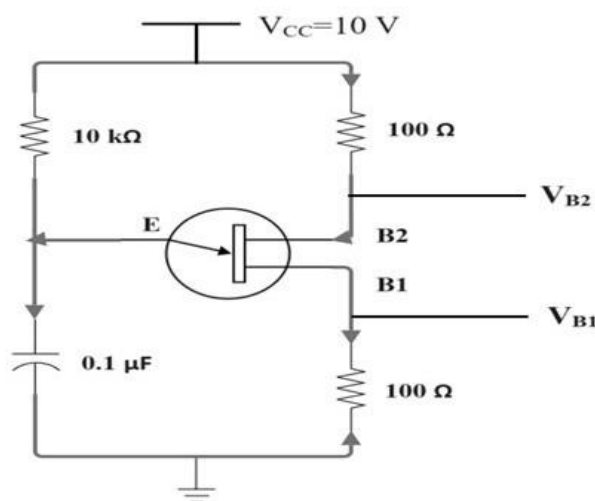

Figure 14.1: UJT as a Relaxation Oscillator

Try

1. For the UJT relaxation oscillator in the following figure, it is known that; $R_{BB} = 2K\Omega$, n =0.7, $V_v = 2V$ $I_v = 20mA$, $I_P = 15\mu A$, and $R_B = 150\Omega$.
2. 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).
3. Design a UJT relaxation oscillator to generate a sawtooth waveform at a frequency of 500Hz. Assume the supply voltage $V_{BB} = 20V$, $V_v = 2.9V$, $V_v = 1.118V$, $I_p = 1.6mA$ and $I_v = 3.5mA$

V. TEXT BOOKS:

1. Jacob Millman, Electronic Devices and Circuits, Tata McGraw Hill Education, 3rd edition, 2014.
2. Robert L. Boylestead, Louis Nashelsky, Electronic Devices and Circuits Theory, Pearson, 11th edition, 2009.

VI. REFERENCE BOOKS:

1. Jacob Millman, Herbert Taub , Mothiki S. PrakashRao, "Pulse Digital and Switching Waveforms", Tata McGraw-Hill, 3rd edition, 2008.
2. David A. Bell, "Solid State Pulse Circuits", PHI, 4th edition, 2002.
3. J. Millman, C. C. Halkias, "Integrated Electronics", Tata McGraw-Hill. 1st edition, 2008.
4. B. P. Singh, Rekha Singh, "Electronic Devices and Circuits", Pearson, 1st edition, 2006.
5. Behzad Razavi, "Design of Analog CMOS Integrated Circuits", Tata McGraw-Hill, 1st edition, 2002

VII. WEB REFERENCES:

1. http://www.tedpavlic.com/teaching/osu/ece327/
2. http://www.ee.iitkgp.ac.in