Heaven's Light is Our Guide

Rajshahi University of Engineering & Technology

Department of Computer Science & Engineering

Lab Manual

Course Title: Electronic Devices and Circuits

Course No.: EEE 1252

Course Credits: 1.5

Course Teacher

Professor Dr. Abdul Khaleque

Department of Electrical & Electronics Engineering

Rajshahi University of Engineering & Technology

<u>Guidance for report writing:</u>

1. **Cover Page:** All lab reports should have a uniform cover page. It must contain Exp. No, name of the experiment, Student's Name, SID and SID of your group mates.
2. **Objective:** In the second page, you should briefly write what was the aim of the experiment. In other words, write what your intent to achieve by doing the experiment.
3. **Theory:** Next, provide the background information on the experiment or concept under investigation.
4. **List of required apparatus:** List all the equipment, components, and materials used in the experiment. Include model numbers if necessary. Write full specification.
5. **Circuit Diagram:** Draw appropriate circuit diagram with details. It should be clean and readable.
6. **Experimental Procedure:**
   - Describe the step-by-step procedure you followed during the experiment.
   - Include any diagrams or schematics to illustrate the circuit setup.
   - Be specific about measurements, settings, and any adjustments made.
7. **Data Collection:**
   - Present the data you collected during the experiment. This may include tables, graphs, or charts.
   - Ensure all measurements have appropriate units and are labelled clearly.
8. **Data Analysis and Results:**
   - Analyse the data to draw conclusions and support or reject your hypotheses.
   - Discuss any patterns or trends you observed.
   - Include relevant calculations, equations, and formulas.
   - Present the main results of your experiment in a clear and organized manner.
   - Discuss any errors or uncertainties in your measurements.
9. **Discussion:**
   - Interpret the results in the context of the experiment's objectives and the underlying theory.
   - Compare your findings with expected or theoretical values.
   - Discuss any sources of error and their potential impact on the results.
   - Address any unexpected outcomes and offer explanations.
10. **Conclusion:**
    - Summarize the key findings and their significance.
    - State whether your experiment supported or contradicted your hypotheses.
    - Mention any practical applications or real-world implications.

Pre-laboratory: Read this laboratory experiment carefully to become familiar with the background procedural steps in this experiment. Download the two user manuals for the function generator and oscilloscope and become familiar with their use. Using the simulation package of your choice in which you are the most familiar with: Multisim, Workbench or LTSpice IV simulate the following circuits/ experiments. Prepare a one-page report based on pre-laboratory which includes circuits and input-outputs obtained from the software. For example, if Exp. 1 is due on Saturday, student should complete prelab and print one page on the circuit and summary of results before starting Exp. 1 in the lab.

List of the experiments:

Exp. No.	Experiment Name
01	Oscilloscope Fundamentals.
02	Study of diode Half-wave rectifier circuit.
03	Study of diode Full-wave rectifier circuits.
04	Study of BJT voltage divider biasing circuit.
05	Study of the frequency response of FET amplifier
06	Experimental study of inverting, noninverting, summing, subtracting/differential, and unity gain buffer amplifier using Op-amp
07	Experimental study of op-amp based integrator and differentiator
08	Observation the effect of Slew rate of op-amp
09	Experimental studies on Precision half wave and full wave rectifier
10	Experimental study on Op-amp based square wave generator
11	Experimental study on Op-amp based triangular wave generator
12	Experimental study on rectangular and square wave generator using 555 timer
13	Design of first order active low pass Butterworth Filter

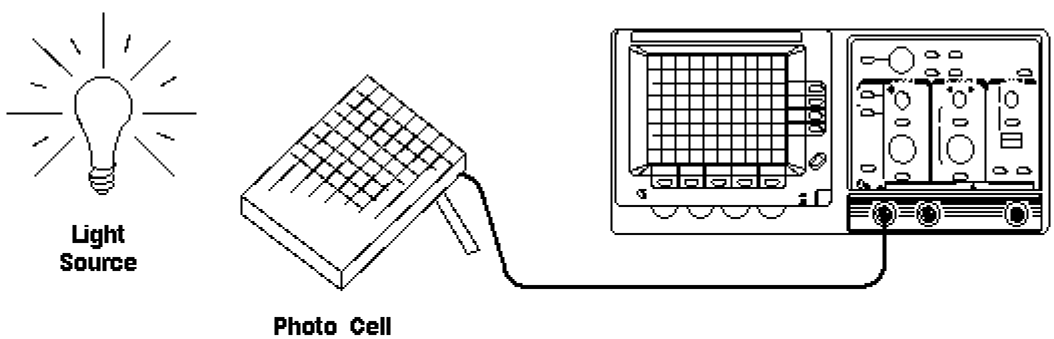
Exp No.: 01

Name of the Experiment: Study of Oscilloscope fundamentals.

Introduction: The word “Oscilloscope” can be separated into two parts “oscillo” and “scope”; the first is short for “oscillations” and the second means “to view or see”. The oscilloscope is basically a graph-displaying device - it draws a graph of an electrical signal. In most applications the graph shows how signals change over time: the vertical (Y) axis represents voltage and the horizontal (X) axis represents time. The intensity or brightness of the display is sometimes called the Z axis.

An oscilloscope is easily the most useful instrument available for testing circuits because it allows us to *see* the signals at different points in the circuit. The best way of investigating an electronic system is to monitor signals at the input and output of each system block, checking that each block is operating as expected and is correctly linked to the next.

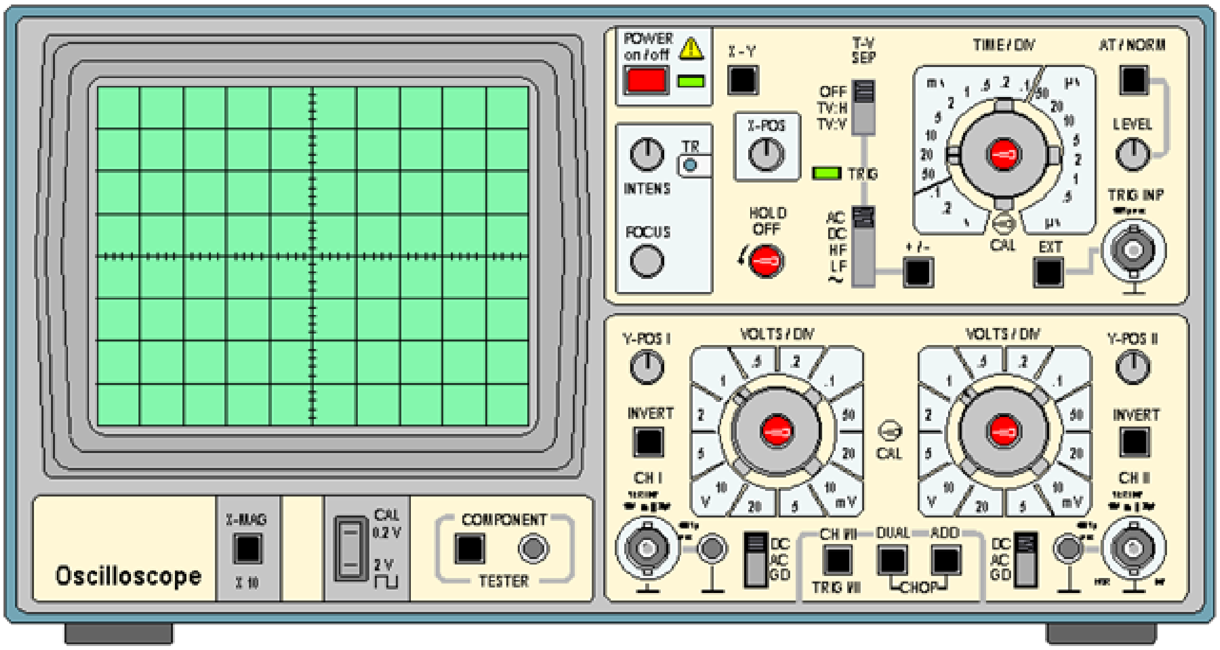
Oscilloscopes are used by everyone from television repair technicians to physicists. They are indispensable for anyone designing or repairing electronic equipment. The usefulness of an oscilloscope is not limited to the world of electronics. With the proper transducer, an oscilloscope can measure all kinds of phenomena. A transducer is a device that creates an electrical signal in response to physical stimuli, such as sound, mechanical stress, pressure, light, or heat. For example, a microphone is a transducer. An automotive engineer uses an oscilloscope to measure engine vibrations. A medical researcher uses an oscilloscope to measure brain waves. The possibilities are endless.



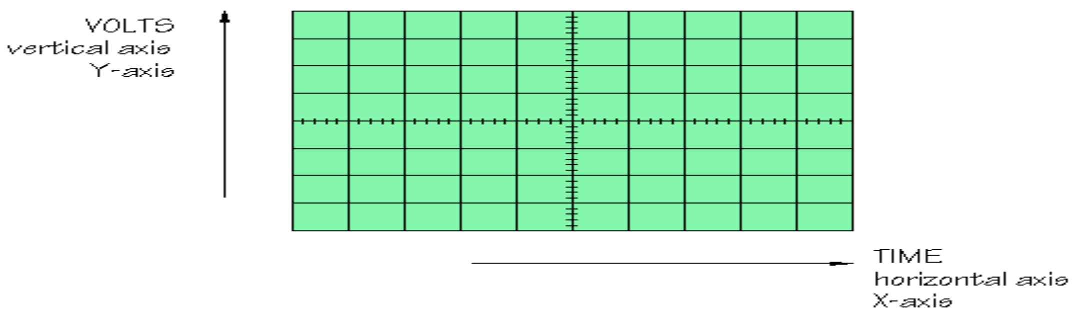
Scientific Data Gathered by an Oscilloscope (in Diagram)

Oscilloscope Front Panel:

The diagram shows a*n* oscilloscope, it may look different but will have similar controls. Faced with an instrument like this, we typically respond either by twiddling every knob and pressing every button in sight, or by adopting a glazed expression. Neither approach is especially helpful. Following the systematic description below will give us a clear idea of what an oscilloscope is and what it can do.

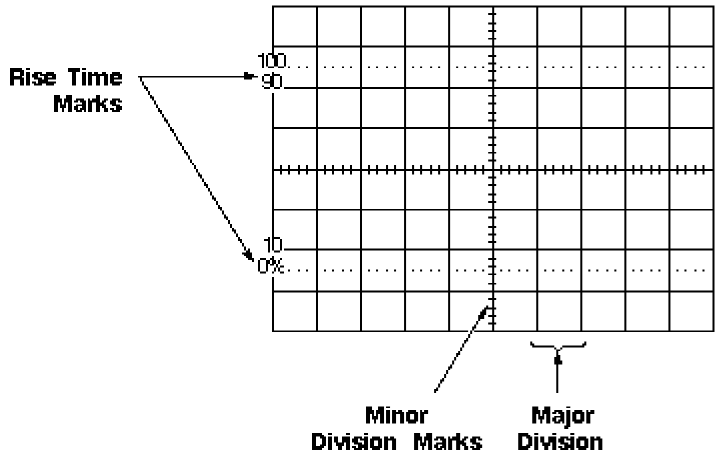


The function of an oscilloscope is extremely simple: it draws a *V/t* graph, a graph of voltage against time, voltage on the vertical or Y-axis, and time on the horizontal or X-axis.



Take a look at the oscilloscope display. Notice the grid markings on the screen - these markings create the graticule. Each vertical and horizontal line constitutes a major division. The graticule is usually laid out in an 8-by-10 division pattern. Labeling on the oscilloscope controls (such as volts/div and sec/div) always refers to major divisions. The tick marks on the center horizontal and vertical graticule lines are called minor divisions.

Many oscilloscopes display on the screen how many volts each vertical division represents and how many seconds each horizontal division represents. Many oscilloscopes also have 0%, 10%, 90%, and 100% markings on the graticule to help make rise time measurements



An Oscilloscope Graticule

Setting the Controls:

After plugging in the oscilloscope, take a look at the front panel. It is divided into three main sections labeled Vertical, Horizontal, and Trigger. Oscilloscope may have other sections, depending on the model and type (analog or digital).

Notice the input connectors on oscilloscope. This is where probes are attached. Oscilloscopes have at least two input channels and each channel can display a waveform on the screen. Multiple channels are handy for comparing waveforms.

Some oscilloscopes have an AUTOSET or PRESET button that sets up the controls in one step to accommodate a signal. If oscilloscope does not have this feature, it is helpful to set the controls to standard positions before taking measurements.

Standard positions include the following:

- Set the oscilloscope to display channel 1
- Set the volts/division scale to a mid-range position
- Turn off the variable volts/division
- Turn off all magnification settings
- Set the channel 1 input coupling to DC
- Set the trigger mode to auto
- Set the trigger source to channel 1
- Turn trigger holdoff to minimum or off
- Set the intensity control to a nominal viewing level
- Adjust the focus control for a sharp display

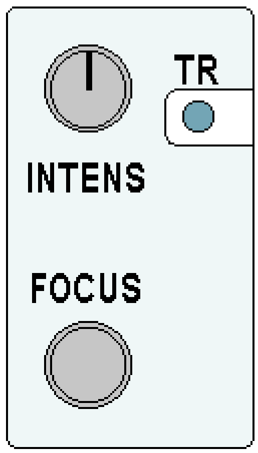
These are general instructions for setting up the oscilloscope.

Y-POS and X-POS Setting:

Changing the scales of the X-axis and Y-axis allows many different signals to be displayed. Sometimes, it is also useful to be able to change the *positions* of the axes. This is possible using the X-POS and **Y-POS** controls. For example, with no signal applied, the normal trace is a straight line across the centre of the screen. Adjusting Y-POS allows the zero level on the Y-axis to be changed, moving the whole trace up or down on the screen to give an effective display of signals like pulse waveforms which do not alternate between positive and negative values.

This is useful when we want to use the grid in front of the screen to make measurements, for example, to measure the period of a waveform.

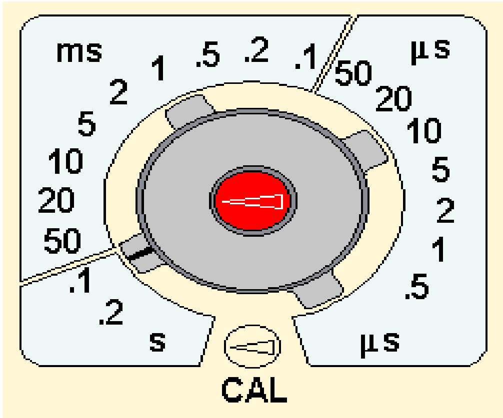
INTENSITY and FOCUS Setting:



When these are correctly set, the spot will be reasonably bright but not glaring, and as sharply focused as possible. (The TR control is screwdriver adjusted. It is only needed if the spot moves at an angle rather than horizontally across the screen with no signal connected.)

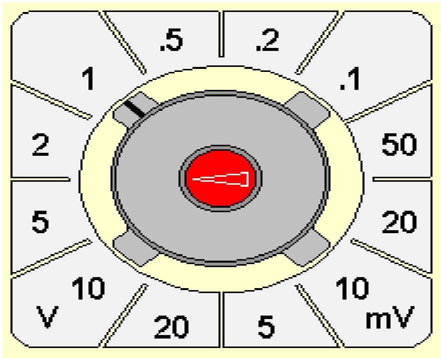
The TIME/DIV Setting:

The slope of the rising phase varies with the frequency of the saw tooth and can be adjusted, using the TIME/DIV control, to change the scale of the X-axis. Dividing the oscilloscope screen into squares allows the horizontal scale to be expressed in seconds, milliseconds or microseconds per division (s/DIV, ms/DIV, µs/DIV). Alternatively, if the squares are 1 cm apart, the scale may be given as s/cm, ms/cm or µs/cm.



The VOLTS/DIV Setting:

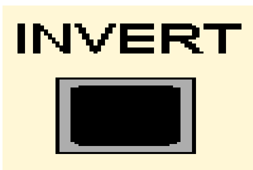
The Y-amplifier is linked in turn to a pair of **Y-plates** so that it provides the Y-axis of the *V/t* graph. The overall gain of the Y-amplifier can be adjusted, using the VOLTS/DIV control, so that the resulting display is neither too small nor too large, but fits the screen and can be seen clearly. The vertical scale is usually given in V/DIV or mV/DIV.



The *oscilloscope* has a built in source of signals which allow us to check that the oscilloscope is working properly.

Invert:

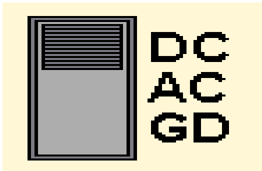
When the INVERT button is pressed IN, the corresponding signal is turned upside down, or inverted, on the oscilloscope screen.



This feature is sometimes useful when comparing signals.

DC/AC/GND slide switches:

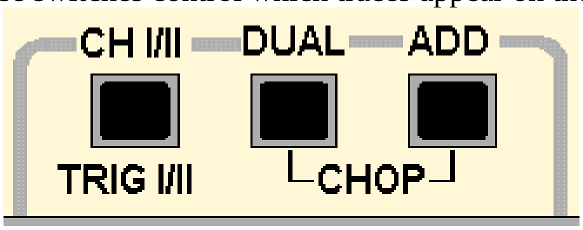
In the DC position, the signal input is connected directly to the Y-amplifier of the corresponding channel, CH I or CH II. In the AC position, a capacitor is connected into the signal pathway so that DC voltages are blocked and only changing AC signals are displayed.



In the GND position, the input of the Y-amplifier is connected to 0 V. This allows us to check the position of 0 V on the oscilloscope screen. The DC position of these switches is correct for most signals.

Trace selection switches:

The settings of these switches control which traces appear on the oscilloscope screen.



Settings highlighted in yellow are used frequently. Experience with the oscilloscope will help us to decide which setting is best for a particular application.

XY Mode:

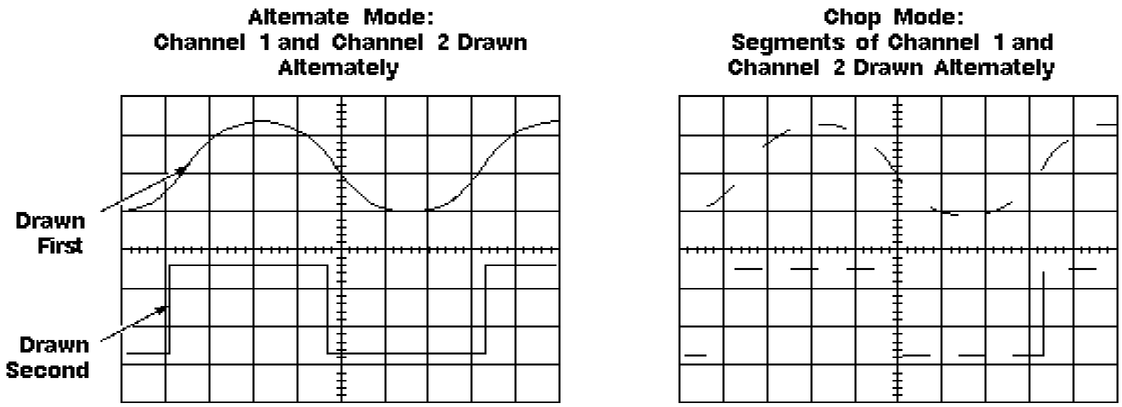
Most oscilloscopes have the capability of displaying a second channel signal along the X-axis (instead of time). This is called XY mode

Alternate and Chop Display:

On analog scopes, multiple channels are displayed using either an alternate or chop mode. (Digital oscilloscopes do not normally use chop or alternate mode.)

Alternate mode draws each channel alternately - the oscilloscope completes one sweep on channel 1, then one sweep on channel 2, a second sweep on channel 1, and so on. Use this mode with medium- to high-speed signals, when the sec/div scale is set to 0.5 ms or faster.

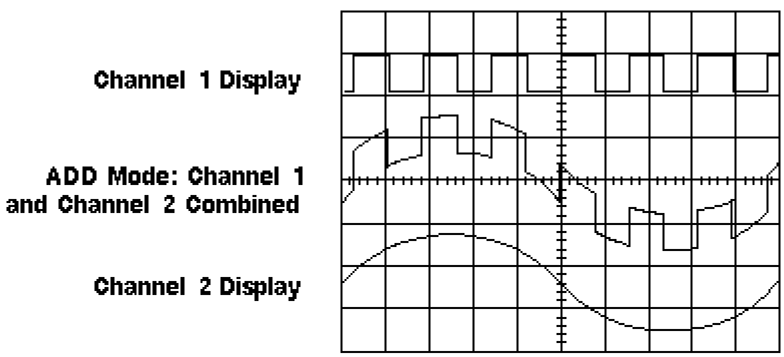
Chop mode causes the oscilloscope to draw small parts of each signal by switching back and forth between them.



Multi-Channel Display Modes

Add Operation:

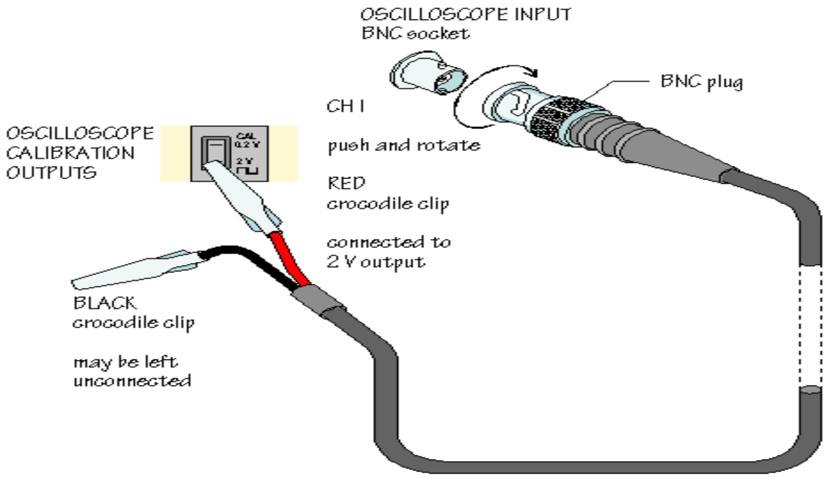
Oscilloscope may also have operations to add waveforms together, creating a new waveform display. Analog oscilloscopes combine the signals while digital oscilloscopes mathematically create new waveforms. Subtracting waveforms is another math operation. Subtraction with analog oscilloscopes is possible by using the channel invert function on one signal and then use the add operation. Digital oscilloscopes typically have a subtraction operation available. Following Figure illustrates a third waveform created by adding two different signals together.



Adding Channels

Compensating the Probe:

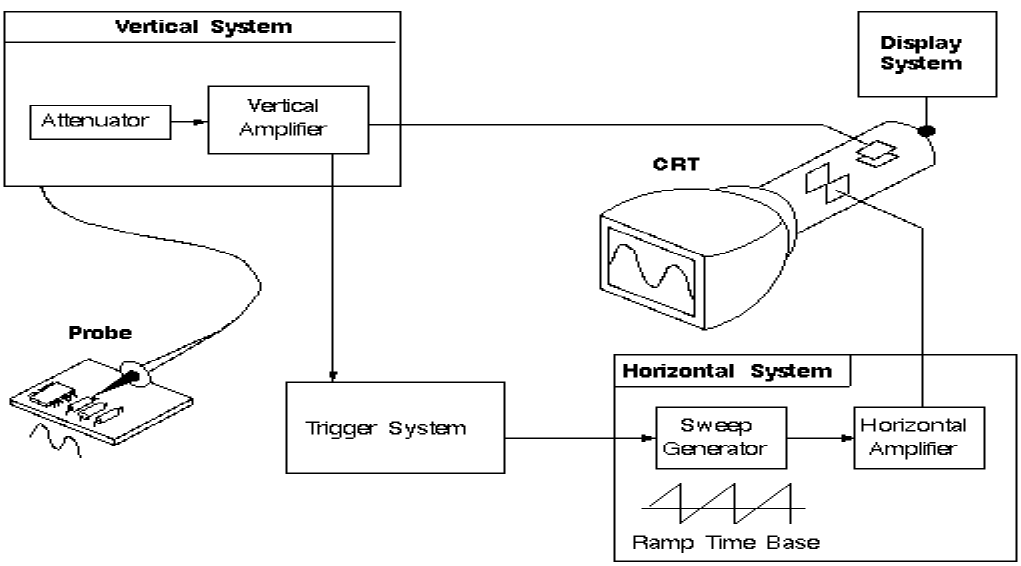
Before using a probe, it should be compensated - to balance its electrical properties to a particular oscilloscope. Most oscilloscopes have a square wave reference signal available at a terminal on the front panel used to compensate the probe. The probe can be compensated by:



- Attaching the probe to an input connector
- Connecting the probe tip to the probe compensation signal
- Attaching the ground clip of the probe to ground
- Viewing the square wave reference signal
- Making the proper adjustments on the probe so that the corners of the square wave are square

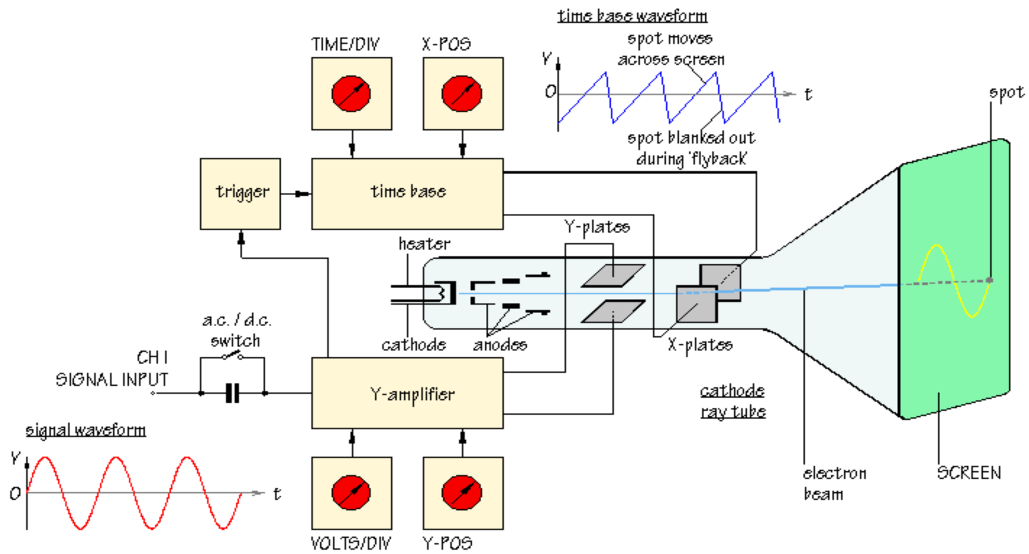
Basic Block Diagram of Oscilloscope:

When we connect an oscilloscope probe to a circuit, the voltage signal travels through the probe to the vertical system of the oscilloscope. Following Figure is a simple block diagram that shows how an oscilloscope displays a measured signal.



Depending on the vertical scale setting (volts/div control), an attenuator reduces the signal voltage or an amplifier increases the signal voltage.

Next, the signal travels directly to the vertical deflection plates of the cathode ray tube (CRT). Voltage applied to these deflection plates causes a glowing dot to move. (An electron beam hitting phosphor inside the CRT creates the glowing dot.) A positive voltage causes the dot to move up while a negative voltage causes the dot to move down.



The signal also travels to the trigger system to start or trigger a "horizontal sweep." Horizontal sweep is a term referring to the action of the horizontal system causing the glowing dot to move across the screen. Triggering the horizontal system causes the horizontal time base to move the glowing dot across the screen from left to right within a specific time interval. Many sweeps in rapid sequence cause the movement of the glowing dot to blend into a solid line. At higher speeds, the dot may sweep across the screen up to 500,000 times each second.

Together, the horizontal sweeping action and the vertical deflection action trace a graph of the signal on the screen. The trigger is necessary to stabilize a repeating signal.

In conclusion, to use an oscilloscope, we need to adjust three basic settings to accommodate an incoming signal:

- The attenuation or amplification of the signal. Use the volts/div control to adjust the amplitude of the signal before it is applied to the vertical deflection plates.
- The time base. Use the sec/div control to set the amount of time per division represented horizontally across the screen.
- The triggering of the oscilloscope. Use the trigger level to stabilize a repeating signal, as well as triggering on a single event.

Also, adjusting the focus and intensity controls enables you to create a sharp, visible display.

Measurements with Oscilloscope:

Voltage Measurements

Voltage is the amount of electric potential, expressed in volts, between two points in a circuit. Usually one of these points is ground (zero volts) but not always. Voltages can also be measured from peak-to-peak - from the maximum point of a signal to its minimum point.

The oscilloscope is primarily a voltage-measuring device. Once the voltage is measured, other quantities are just a calculation away. For example, Ohm's law states that voltage between two points in a circuit equals the current times the resistance. From any two of these quantities we can calculate the third. Another handy formula is the power law: the power of a DC signal equals the voltage times the current. Calculations are more complicated for AC signals, but the point here is that measuring the voltage is the first step towards calculating other quantities.

Ohm's Law:

$$\text{Voltage} = \text{CurrenttimesResistance}$$

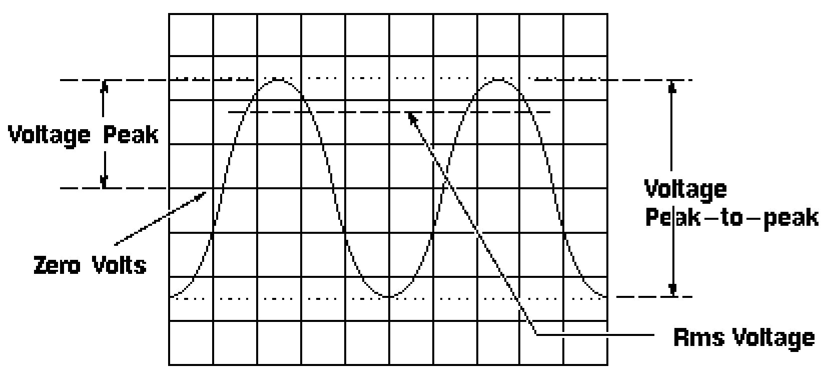
$$\text{Current} = \frac{\text{Voltage}}{\text{Resistance}}$$

$$\text{Resistance} = \frac{\text{Voltage}}{\text{Current}}$$

Power Law:

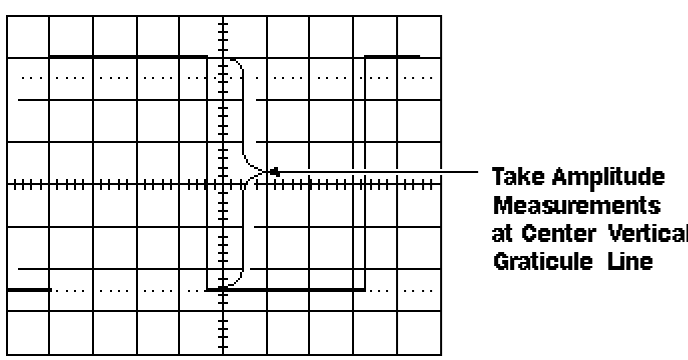
$$\text{Power} = \text{VoltagetimesCurrent}$$

Following figure shows the voltage of one peak - V[p] - and the peak-to-peak voltage - V[p-p] -, which is usually twice V[p]. Use the RMS (root-mean-square) voltage - V[RMS] - to calculate the power of an AC signal.



Voltage Peak and Peak-to-peak Voltage

The voltage is measured by counting the number of divisions a waveform spans on the oscilloscope's vertical scale. Adjusting the signal to cover most of the screen vertically, then taking the measurement along the center vertical graticule line having the smaller divisions makes for the best voltage measurements. The more screen area is used, the more accurately data can be read from the screen.



Measure Voltage on the Center Vertical Graticule Line

Many oscilloscopes have on-screen cursors that can be used take waveform measurements automatically on-screen, without having to count graticule marks. Basically, cursors are two horizontal lines for voltage measurements and two vertical lines for time measurements that we can move around the screen. A readout shows the voltage or time at their positions.

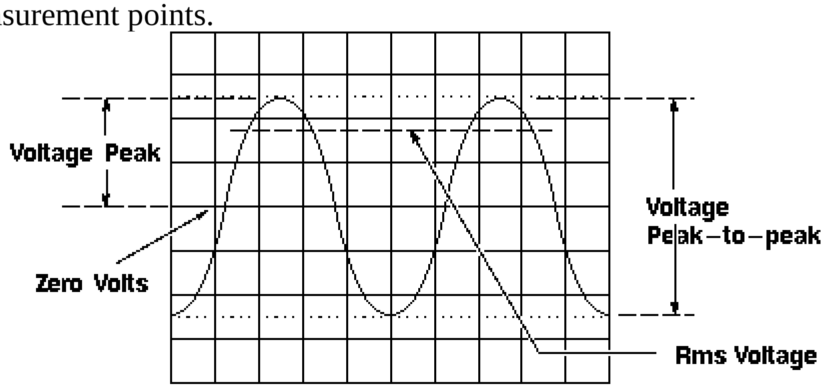
Time and Frequency Measurements

We can take time measurements using the horizontal scale of the oscilloscope. Time measurements include measuring the period, pulse width, and timing of pulses. Frequency is the reciprocal of the period, so once we know the period, the frequency is one divided by the period. Like voltage measurements, time measurements are more accurate when we adjust the portion of the signal to be measured to cover a large area of the screen. Taking time measurement along the center horizontal graticule line, having smaller divisions, makes for the best time measurements.

Pulse and Rise Time Measurements

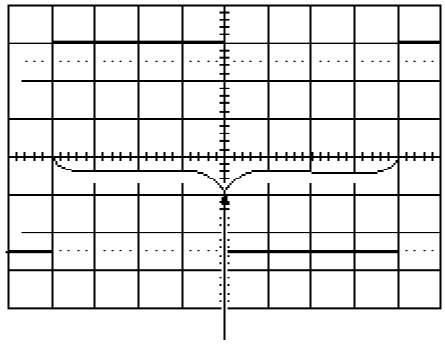
In many applications, the details of a pulse's shape are important. Pulses can become distorted and cause a digital circuit to malfunction, and the timing of pulses in a pulse train is often significant.

Standard pulse measurements are pulse width and pulse rise time. Rise time is the amount of time a pulse takes to go from the low to high voltage. By convention, the rise time is measured from 10% to 90% of the full voltage of the pulse. This eliminates any irregularities at the pulse's transition corners. This also explains why most oscilloscopes have 10% and 90% markings on their screen. Pulse width is the amount of time the pulse takes to go from low to high and back to low again. By convention, the pulse width is measured at 50% of full voltage. See Figure below for these measurement points.



Rise Time and Pulse Width Measurement Points

Pulse measurements often require fine-tuning the triggering. To become an expert at capturing pulses, we should learn how to use trigger holdoff and how to set the digital oscilloscope to capture pretrigger data, as described earlier in the Controls section. Horizontal magnification is another useful feature for measuring pulses.



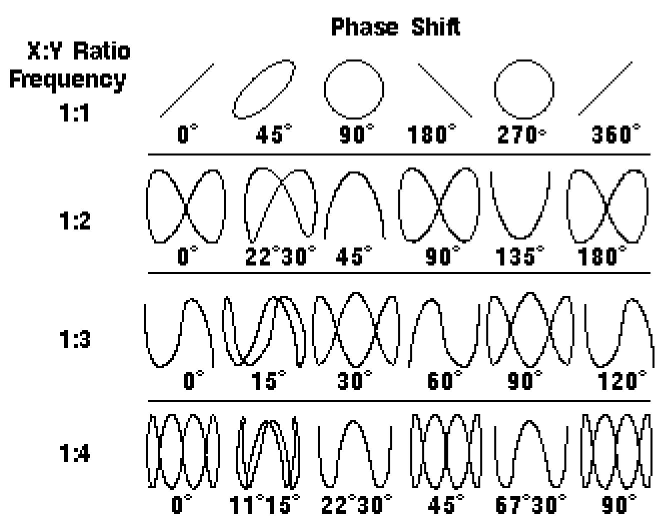
Measure Time on the Center Horizontal Graticule Line

Phase Shift Measurements

The horizontal control section may have an XY mode that display an input signal rather than the time base on the horizontal axis. (On some digital oscilloscopes this is a display mode setting.) This mode of operation opens up a whole new area of phase shift measurement techniques.

The phase of a wave is the amount of time that passes from the beginning of a cycle to the beginning of the next cycle, measured in degrees. Phase shift describes the difference in timing between two otherwise identical periodic signals.

One method for measuring phase shift is to use XY mode. This involves inputting one signal into the vertical system as usual and then another signal into the horizontal system. (This method only works if both signals are sine waves.) This set up is called an XY measurement because both the X and Y axis are tracing voltages. The waveform resulting from this arrangement is called a Lissajous pattern (named for French physicist Jules Antoine Lissajous and pronounced LEE-sa-zhoo). From the shape of the Lissajous pattern, we can tell the phase difference between the two signals. We can also tell their frequency ratio. Figure shows below Lissajous patterns for various frequency ratios and phase shifts.



Lissajous Patterns

Exp No.: 02

Name of the experiment: Study of diode Half-Wave rectifier circuit.

Introduction: A rectifier converts an AC signal into a DC signal. From the characteristic curve of a diode we observe that it allows the current to flow when it is in the forward bias only. In the reverse bias it remains open. So, when an alternating voltage is applied across a diode it allows only the half cycle (positive half depending on orientation of diode in the circuit) during its forward bias condition, other half cycle will be clipped off. In the output the load will get DC signal.

Diode rectifier can be categorized in two major types. They are-

a) Half-wave rectifier.
b) Full-wave rectifier.

Half-wave rectifier: Half-wave rectifier circuit can be built by using a single diode. The circuit diagram and the wave shape of the input and output voltage of half wave rectifier are shown bellow:

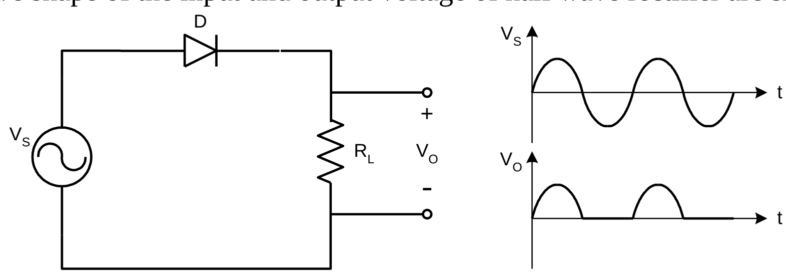


Figure 2.1: Half wave rectifier

In this circuit the load receives approximately half of the input power, Average dc voltage is low, Due to the presence of ripple output voltage is not smooth one.

Equipment:

1.	Diode	(1 piece)
2.	Signal generator	(1 piece)
3.	Resistor (10K)	(1 piece)
4.	Capacitor (0.22 μF, 10 μF)	(1 piece each)
5.	Oscilloscope chord	(2 pieces)
6.	Oscilloscope	(1 piece)

Procedure:

1. Connect the circuit in breadboard as shown in Fig: 2.1 without capacitor.
2. Observe the output and input voltages in oscilloscope and draw them.
3. Connect the 0.22μF capacitor and repeat step 2.
4. Connect the 10μF capacitor and repeat step 2. How does the output wave-shape differ from that in step 3?
5. Vary the frequency from 10 kHz to 100 Hz. What effect do you observe when frequency is changed?

PSPICE simulation:

```
* HALF WAVE RECTIFIER CIRCUIT

V_app 1 0 sin(0 12 1)          ; APPLIED SINUSOIDAL VOLTAGE
D_1 1 2 D1n4003             ; THE DIODE MODEL
R_1 2 0 2K
.lib nom.lib                        ;THE LIBRARY FILES USED TO OBTAIN
                                    THE REQUIRED INFORMATION FOR DIODE
                                    MODEL
.tran 1ns 2s                        ; THE TRANSIENT ANALYSIS
.probe                              ;THE PROBE ANALYSIS USED IN VIEWING
                                    GRAPHICAL RESULT
.end
```

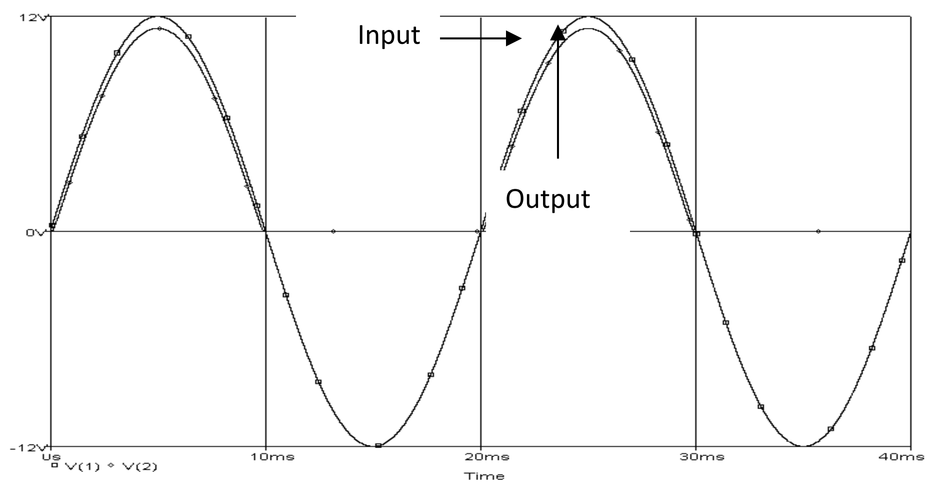


Fig: 2.2 Input, output wave-shapes of a Half wave Rectifier

Report:

1. Write the answers that were asked during the working procedure.
2. Draw the input wave, output wave (without and with capacitor) for both the circuits.
3. What is the frequency of the output of the circuit shown in Fig: 2.2.
4. What is the function of capacitor in the both circuits? Why a capacitor of higher value is preferable?

Exp No.: 03

Name of the experiment: Study of diode Full-Wave rectifier circuits.

Introduction: A rectifier converts an AC signal into a DC signal. From the characteristic curve of a diode we observe that it allows the current to flow when it is in the forward bias only. In the reverse bias it remains open. So, when an alternating voltage is applied across a diode it allows only the half cycle (positive half depending on orientation of diode in the circuit) during its forward bias condition, other half cycle will be clipped off. In the output the load will get DC signal.

Diode rectifier can be categorized in two major types. They are-

a) Half-wave rectifier.
b) Full-wave rectifier.

Full-wave rectifier: In the full-wave rectifier both the half cycle is present in the output. Two circuits are used as full-wave rectifiers are shown below:

a) Full-wave rectifier using center-tapped transformer and
b) Full-wave bridge rectifier.

Using a center-tapped transformer: Two diodes will be connected to the ends of the transformer and the load will be between the diode and the center tap. The circuit diagram and the wave shapes are shown in figure 3.1:

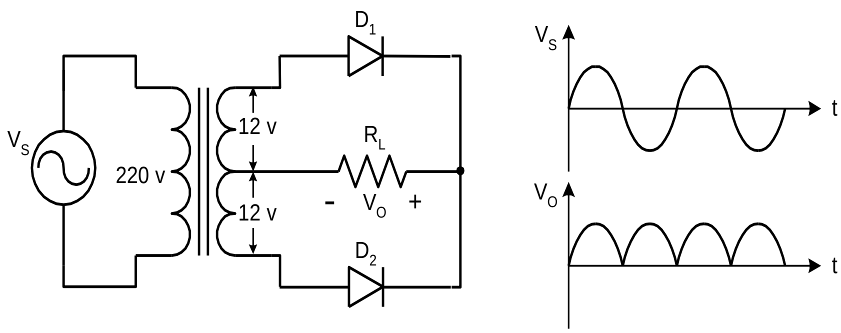


Figure 3.1: Full wave rectifier using center tapped transformer

This circuit has some advantages over the first one:

Wastage of power is less,
Average DC output increase significantly,
Wave shape becomes smoother.

Disadvantages are:

Require more space and becomes bulky because of the transformer.

Bridge rectifier: a bridge rectifier overcomes all disadvantages described above.

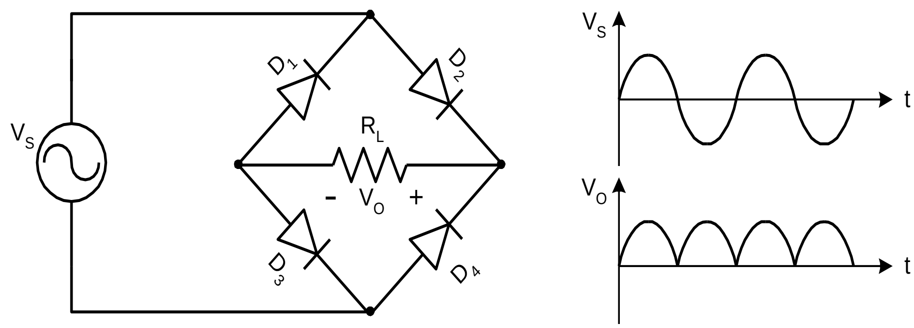


Figure 3.2: Bridge rectifier

A rectifier however cannot produce a smooth dc voltage. It produces some ripple in the output. This ripple can be reduced using filter capacitor across the load.

Equipment:

1. Diodes (4 pieces)
2. Signal generator (1 piece)
3. Resistor (10K) (1 piece)
4. Capacitor (0.22µF, 10µF) (1 piece each)
5. Oscilloscope chord (2 pieces)
6. Oscilloscope (1 piece)

Circuit diagram:

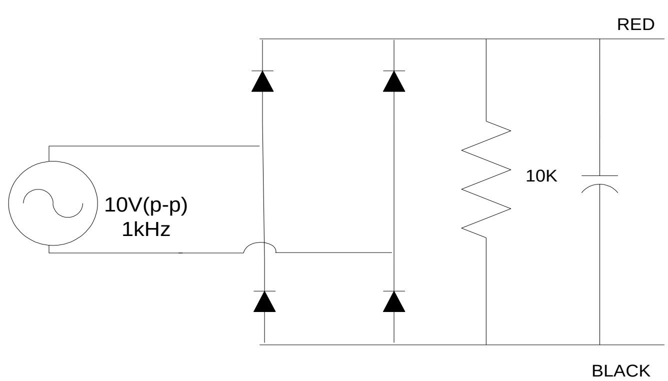


Fig: 2

Procedure:

1. Connect the circuit in breadboard as shown in Fig: 3.2 without capacitor.
2. Observe the output and input voltages in oscilloscope and draw them.
3. Connect the 0.22µF capacitor and repeat step 2. How does the output wave-shape differ from that in step 3?
4. Connect the 10µF capacitor and repeat step 2. Observe the wave-shapes.
5. Vary the frequency from 10 kHz to 100 Hz. Observe the wave-shapes. What effect do you observe when frequency is changed?

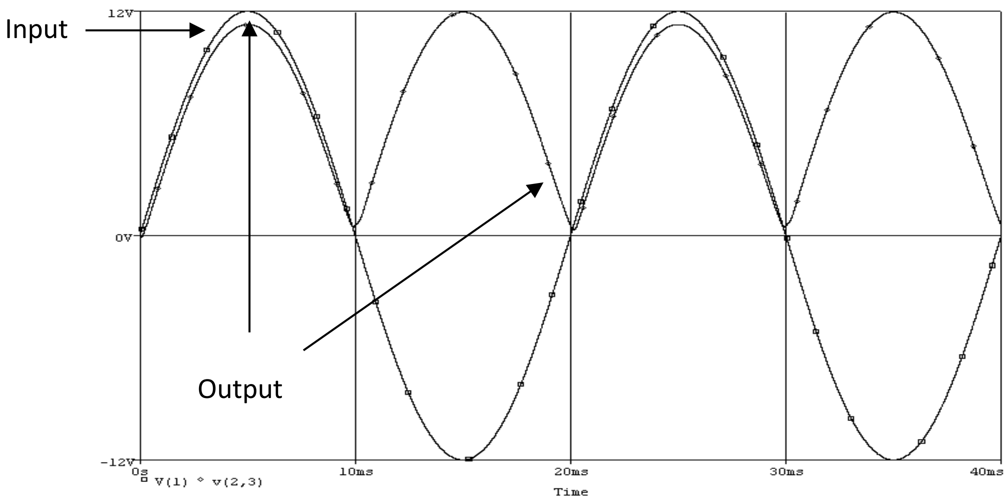


Fig. 3.3: Input, Output wave-shape of full wave rectifier

Report:

5. Write the answers that were asked during the working procedure.
6. Draw the input wave, output wave (without and with capacitor) for both the circuits.
7. What is the frequency of the output of the circuit shown in Fig: 3.3.
8. What is the function of capacitor in the both circuits? Why a capacitor of higher value is preferable?

Questions:

1. What is meant by rectifier?
2. Which type of transformer is used for the rectifier input?
3. Define ripple factor.
4. Write the efficiency of this rectifier.

Exp No.: 04

Name of the experiment: Study of BJT amplifier using voltage divider bias.

Introduction: This type of biasing is otherwise called Emitter Biasing. The necessary biasing is provided using 3 resistors: R_1, R_2 and Re. The resistors R1 and R2 act as a potential divider and give a fixed voltage to the base. If the collector current increases due to change in temperature or change in β, the emitter current I_e also increases and the voltage drop across R_e increases, reducing the voltage difference between the base and the emitter. Due to reduction in V_{be}, base current I_b and hence collector current I_c also reduces. This reduction in V_{be}, base current I_b and hence collector current I_c also reduces. This reduction in the collector current compensates for the original change in I_c.

The stability factor $S= (1+\beta) * ((1/ (1+\beta))$. To have better stability, we must keep R_b/R_e as small as possible. Hence the value of R1 R2 must be small. If the ratio R_b/R_e is kept fixed, S increases with β.

The stability factor S= (1+β) * ((1/ (1+β)). To have better stability, we must keep R_b/R_e as small as possible. Hence the value of R1 R2 must be small. If the ratio R_b/R_e is kept fixed, S increases with β.

$$R_{in} = \beta * R_e$$
$$Gain = \beta * R_e/R_{in}$$

Objective: To construct a voltage divider bias amplifier circuit and measure input resistance and gain and also to plot the dc collector current as a function of collector resistance.

List of Components:

S.No.	Name	Range	Quantity
1.	Transistor	BC 107	1
2.	Resistor	56kΩ,12kΩ,2.2kΩ,470Ω	1,1,1,1
3.	Capacitor	0.1µF, 47µF	2, 1
4.	Function Generator	(0-3)MHz	1
5.	CRO	30MHz	1
6.	Regulated power supply	(0-30)V	1
7.	Bread Board		1

Circuit Diagrams:

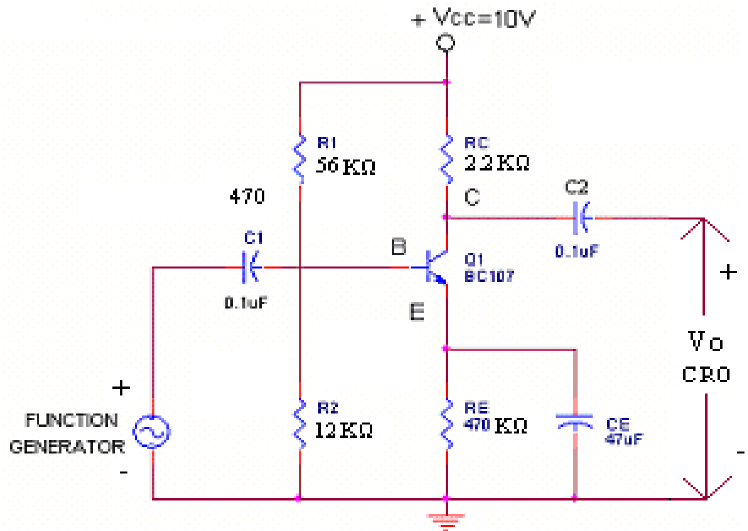


Fig. 4.1: Voltage divider bias circuit.

MODEL GRAPH

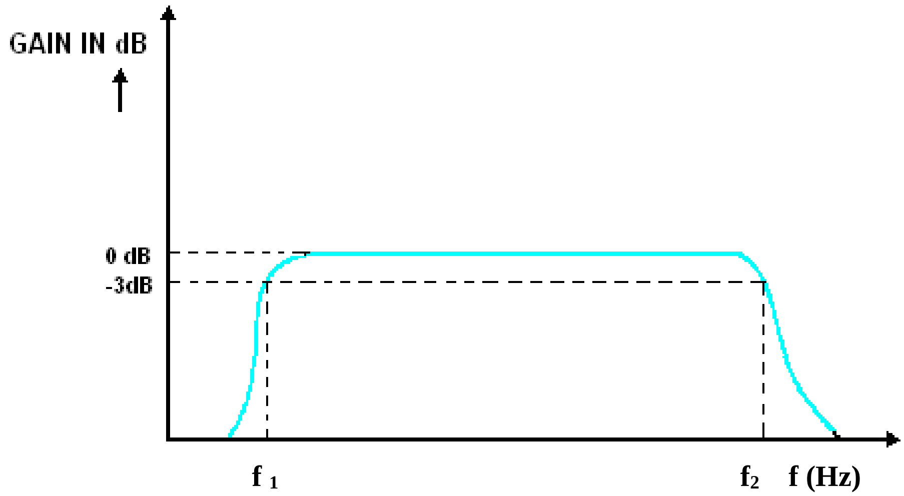


Fig. 4.2

Procedure:

1. Connections are given as per the circuit diagram.
2. Measure the input resistance as $R_{in}=V_{in}/I_{in}$ (with output open) and gain by plotting the frequency response.
3. Compare the theoretical values with the practical values.
4. Plot the dc collector current as a function of the collector resistance (ie) plot of V_{cc} and I_c for various values of Re.

Experimental Data: Keep the input voltage constant.

Vin =

Frequency (Hz)	Output Voltage (volts)	Gain= 20 log(Vo/Vin) (dB)

Result: Thus the voltage divider bias amplifier circuit was constructed and input resistance and gain were determined.

Questions:

1. Why the stability of voltage divider bias circuit is more than other bias circuits?

Exp No.: 05

Title: Study of the frequency response of FET amplifier.

Objective: To plot frequency response (gain in dB vs frequency) and measure of bandwidth of FET amplifier.

Theory: A JFET can be N-channel type or P-channel type. The structure of a P-channel JFET is similar to that of an N-channel JFET. Except that in its structure, N-type is replaced by P-type and P-type by N-type. The structure of an N-channel JFET is a bar of N-type silicon. This bar behaves like a resistor between its two terminals, called source and drain. We introduce heavily doped P-type region on either side of bar. These P-regions are called gates. Usually, two gates are connected together. This gate is used to control current flow from source to drain. This flow of electrons makes the drain current I_d. The electrons in the bar pass through the space between the two P-regions. As width of this space between the p-regions can be controlled by varying gate voltage that is called a channel. We apply a small reverse bias to the gate. Because of the reverse bias, the width of depletion increases. Since the N-type bar is lightly doped compared to the P-regions, the depletions region extends more into the N-type bar. This reduced the width of the channel. Reduction in the width of the channel (the conductive portion of the bar) increases its resistance. This reduces the drain current I_d. There is one important point about the channel shape. It is narrower at the drain end. This happens because the amount of reverse bias is not same throughout the length of the P-N junction. When current flows through the bar, a potential drop occurs across its length. As a result, the reverse bias between the gate and the drain end of the bar is more than that between the gate and the source end of the bar. The width of depletion region is more at the drain end than at the source end. As a result, the channel becomes narrower at the drain end. If the reverse gate bias is increased further, the channel becomes narrower at the drain end and drain current further reduces. If the reverse bias is made sufficiently large, the depletion region will extend into channel. This pinches off all current flow. The gate-source voltage at which pinch-off occurs is called PINCH-OFF voltage V_p.

Required Apparatus:

1. CRO
2. Signal generator
3. Power Supply
4. Bread-board
5. Resistors (1MΩ, 4.7kΩ, 1kΩ)
6. Capacitors(10µF,100MF)
7. FET-BFW10
8. Connecting wires

Circuit Diagram:

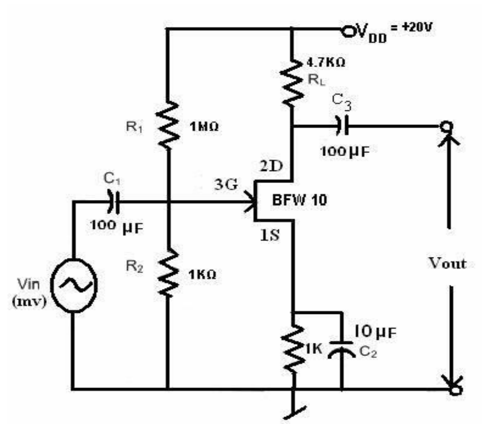


Figure 5.1: Single stage RC coupled FET amplifier

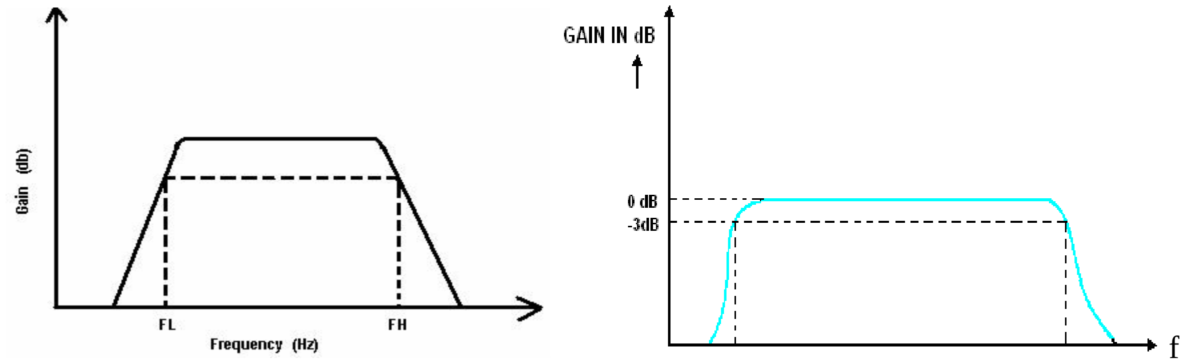


Figure 5.2: Frequency response

Procedure:

(1) Connect the circuit as per the circuit diagram.
(2) Apply V_i = 10 mv and V_{DD} = +20 v.
(3) Now vary the frequency of the input signal and measure the corresponding amplitude variation in output at different values frequency.
(4) Note down the readings and plot the graph between gain and frequency. This curve is known as frequency response curve.

Experimental Data:

Sl. No.	Input voltage (V_i)	Input frequency (F_i)	Output voltage (V_o)	Gain (dB)

Result:

Thus plot a graph between gain and frequency to obtain a frequency response curve. The values of f_L and f_H from graph are given as

$$F_H =$$

$$F_L =$$

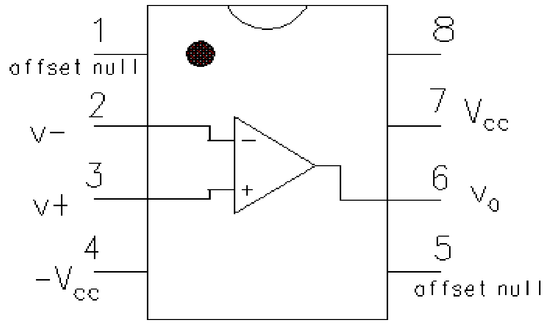
$$BW = F_H - F_L$$

Exp No.: 06

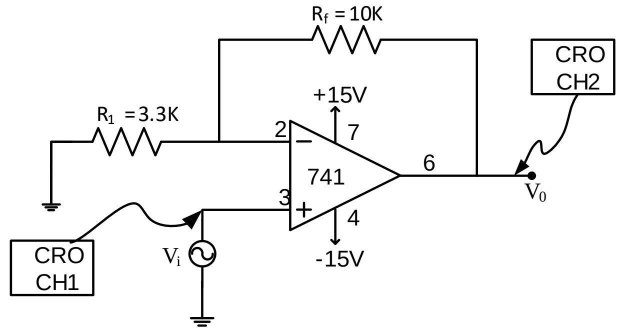
Title: Experimental study of inverting, noninverting, summing, subtracting/differential, and unity gain buffer amplifier

Objective: To understand the basic operations of an op-amp.

Theory: Identify the pins of the µA741C op-amp as follows:



Pin configuration of 741 Op-amp



(Sample Circuit diagram of an inverting amplifier)

Circuit Diagrams and operations:

1.Inverting operational amplifier:

1. Generate input signal from function generator of amplitude 10 V (P-P) with frequency 1 KHz.
2. Place $R_1 = 1$ KΩ and $R_2 = 10$ KΩ.
3. See the output V_0
4. Find the gain: $-R_2/R_1$.

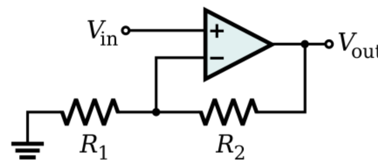


Figure 6.1

2.Noninverting operational amplifier:

1. Generate input signal from function generator of amplitude 10 V (P-P) with frequency 1 KHz.
2. Place $R_g = 1$ KΩ and $R_f = 10$ KΩ.
3. See the output V_0
4. Find the gain: $1+R_g/R_f$.

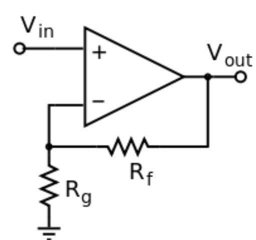


Figure 6.2

3.Summing amplifier:

1. Collect input signals from function generator each with frequency 1 KHz:
   $v_1 = 2\sin(\omega t)$
   $v_2 = 2\sin(\omega t)$
   $v_3 = 2\sin(\omega t)$
2. Place $R_1 = R_2 = R_3 = 1$ KΩ and $R_f = 10$ KΩ.
3. See the output V_0
5. Find the output: $v_0 = -\frac{R_f}{R_1}(v_1 + v_2 + v_3)$

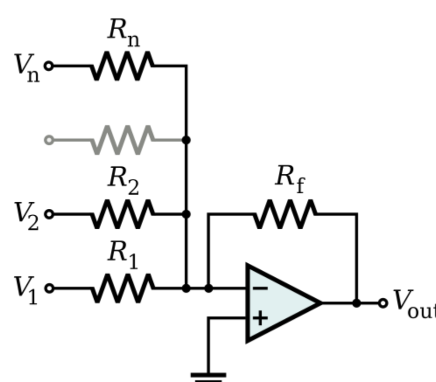


Figure 6.3

4.Summing amplifier:

1. Collect input signals from function generator each with frequency 1 KHz:
   $v_1 = 2\sin(\omega t)$
   $v_2 = 2\sin(\omega t)$
   $v_3 = 2\sin(\omega t)$
2. Place R_i = 1 KΩ, $R_1 = R_2 = R_3$ = 1 KΩ and R_f = 10 KΩ.
3. See the output V_0
4. Find the gain.

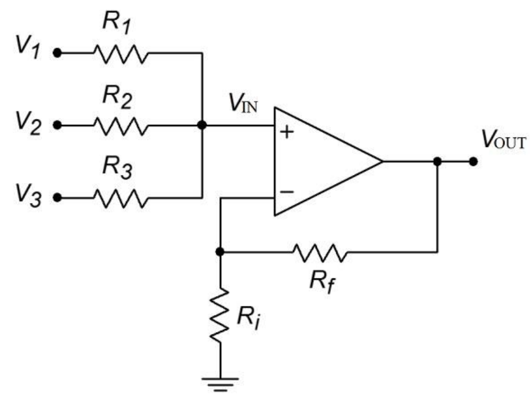


Figure 6.4

5. Differential amplifier:

1. Collect input signals from function generator each with frequency 1 KHz:
   $$v_1 = 2\sin(\omega t)$$
   $$v_2 = 2\sin(\omega t)$$
2. Place R_1 = 1 KΩ and R_f = 10 KΩ.
3. Place $R_2 = R_3$ =1 KΩ

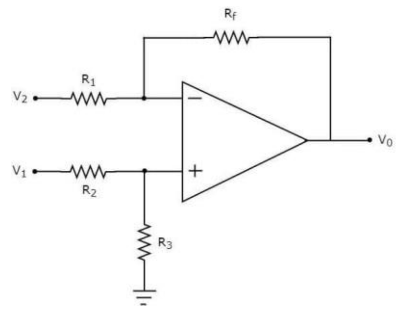


Figure 6.5

$$v_0 = 1 + \frac{R_f}{R_1}\frac{R_3}{R_2 + R_3}v_1 - \frac{R_f}{R_1}v_2$$

4. See the output V_0

6. Unity gain buffer amplifier:

1. Generate input signal from function generator of amplitude 10 V (P-P) with frequency 1 KHz.
2. See the output V_0

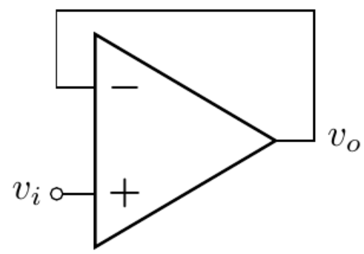


Figure 6.6

Experimental data Table where applicable:

SI No.	Clipping	V_i(p-p)	V_0(p-p)	$A_V = \frac{V_0}{V_i}$	$A_V = Equation\ in\ theory$
1	Before Clipping				
2	Just before clipped				
3	Too much clipped				

Conclusion:

Summarize the key findings of the experiments and discuss the practical applications of op-amp-based above circuits.

Exp No.: 07

Title: Experimental study of op-amped-based integrator and differentiator

Objective:

The objective of this lab is to understand and experimentally study the behavior of operational amplifier (op-amp) based integrators and differentiators. Students will gain hands-on experience in building and testing these circuits, and observe their input-output relationships.

Theory: Write a short theory based on op-amped-based integrator and differentiator.

Required Apparatus:

1. Operational Amplifiers (Op-Amps) - IC741
2. Resistors (write values)
3. Capacitors (write values)
4. Breadboard
5. Oscilloscope
6. Function Generator
7. Connecting wires
8. Power supply

Circuit Diagrams:

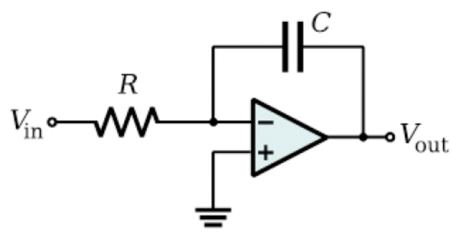


Figure 7.1

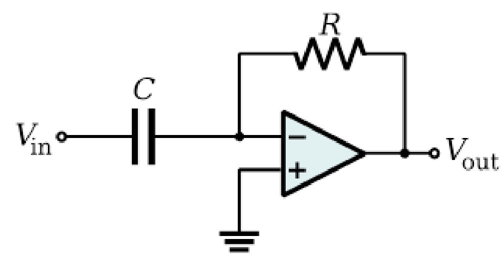


Figure 7.2

Operations:

1. Set up the above circuit on the breadboard.
2. Choose values for the resistor R = 1 kΩ and capacitor C = 0.05 µF for integrator.
3. Choose values for the resistor R = 10 kΩ and capacitor C = 0.1 µF for differentiator.
4. Connect the input signal (sine, square wave and triangular) from the function generator to the V_{in} terminal.
5. Connect the oscilloscope probe to the V_{out} terminal.
6. Power up the circuit and observe the input and output waveforms on the oscilloscope.
7. Change the frequency of the input signal in regular interval and observe how the output changes.
8. Record your observations and note any observations on the integration/ differentiation effect.
9. Plot Gain Vs frequency for both integrator and differentiator.

Data Table: Fill up the following data table for both integrator and differentiator.

Frequency on the input signal (KHz)	Vin(p-p)	Vout(p-p)	Gain = Vout(p-p)/ Vin(p-p)

Analysis and Questions:

1. Compare the observed waveforms in both experiments with the theoretical expectations for integration and differentiation.
2. Explain any discrepancies between theory and experiment.
3. What happens to the output waveform if the values of the resistor or capacitor are changed?
4. How does the input frequency affect the output waveform in each circuit?

Conclusion: Summarize the key findings of the experiments and discuss the practical applications of op-amp-based integrators and differentiators. Reflect on the challenges faced during the experiments and suggest improvements for future work.

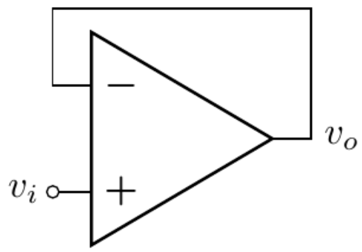
Exp No.:08

Title: Observation of the effect of Slew rate of op-amp

Objective:

In this exercise, the effects of slew rate on pulse and sinusoidal waveforms will be examined. Also, we will find the maximum input signal frequency that can be applied to op-amp. Slew rate places an upper “speed limit” on the rate of change of output voltage. This tends to slow the rising and falling edges of pulse signals, turning them into a trapezoidal shape. In the case of sinusoidal signals, slew rate limiting tends to turn waves into a more triangular shape. The maximum non-slewed sine wave frequency for a given output amplitude is termed the power bandwidth, or f_{max}. Any output signal that exceeds the power bandwidth at the stated output amplitude will exhibit slew rate induced distortion. Slew rate is determined by the internal characteristics of a given op amp. In most op amps, circuit gain or feedback resistor values do not affect the slew rate.

Circuit Diagram:



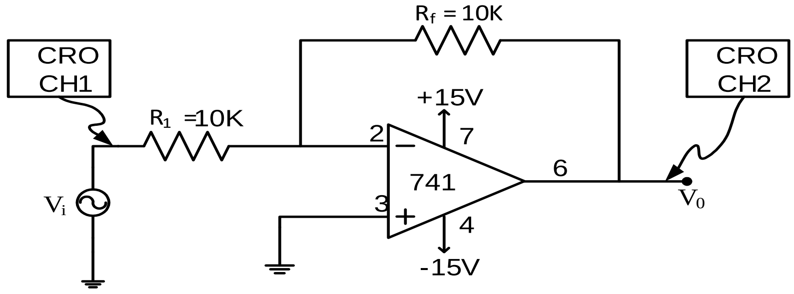


Figure 8.1 (Another sample Circuit diagram for determining Slew rate)

Required Apparatus:

1. Operational Amplifiers (Op-Amps) - IC741
2. Breadboard
3. Oscilloscope
4. Function Generator
5. Connecting wires
6. Power supply

Operations:

1. Set up the above circuit on the breadboard.
   Step-1: Determination of slew rate of op-amp experimentally
2. Apply 2V (peak-to-peak) square wave of 1 KHz frequency to the noninverting terminal and see the output.
3. Increase the time scale of the oscilloscope. You will see the following input-output waveshapes. Using oscilloscope measure ΔV and Δt. Calculate SR = ΔV/ Δt in V/μs.

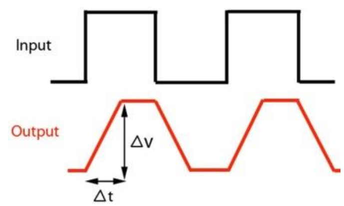


Figure 8.2

Step-2: Determination of maximum frequency

4. Apply 2V (peak-to-peak) square wave of 1 KHz frequency to the noninverting terminal and see the output.
5. Increase the frequency until you get triangular wave output for the square wave input. Record the maximum frequency (From oscilloscope) for which you get the triangular wave at the output. Using the triangular wave of the oscilloscope calculate the maximum frequency. Compare your results with theoretical maximum frequency.
6. Now increase the frequency beyond the maximum frequency and see the distortion. Compare your results with theoretical frequency for which output is more distorted.

Step-3: Determination of maximum amplitude

7. Apply 2V (peak-to-peak) square wave of 1 KHz frequency to the noninverting terminal and see the output.
8. Keep the frequency fixed and increase the magnitude of the input until you get the triangular wave at the output. (NB. If you do not found triangular wave at the output you can select different higher frequency at the input)
9. Record the maximum input signal magnitude and frequency for which the output become triangular.

Step-4: Determination the effect of gain on the slew rate

10. Construct any noninverting amplifier with a voltage gain of 2. Measure the SR as like as **Step-1**. Now increase the voltage gain and find the effect of voltage gain on the slew rate. Prepare a table.

Analysis and Questions:

Prepare necessary data table/ input-output figures to calculate maximum input signal frequency/ input signal magnitude.

Conclusion:

Summarize the key findings of the experiments.

Exp. No: 09

Title: Experimental studies on Precision half wave and full wave rectifier

Objective:

The objective of this lab is to understand the operation and characteristics of precision half-wave and full-wave rectifiers. By performing experiments with these rectifiers, you will gain practical insights into their functionality, efficiency, and applications.

Experiment 1: Precision Half-Wave Rectifier

Circuit Diagram:

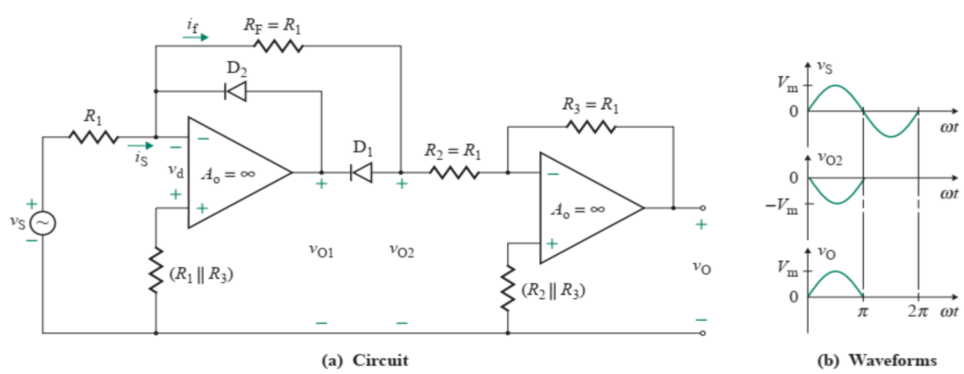


Figure 9.1

Required Apparatus:

1. Operational Amplifiers (Op-Amps) - IC741
2. Breadboard
3. Oscilloscope
4. Function Generator
5. Connecting wires
6. Power supply
7. Resistors (Write the values)
8. Diode (Write number)

Procedure:

1. Connect the circuit as shown in Fig. 9.1. Connect non-inverting terminal of the op-amp to ground.
2. Apply $v_i = 2sin\omega t$ V at the input of the circuit from the signal generator, where f= 1 KHz.
3. Use $R_1 = R_2 = R_3 = R_F = 1K\Omega$.
4. Adjust the CRO to measure the input and output voltage.
5. Observe input-output at the oscilloscope.

Experiment 2: Precision Full-Wave Rectifier

Circuit Diagram:

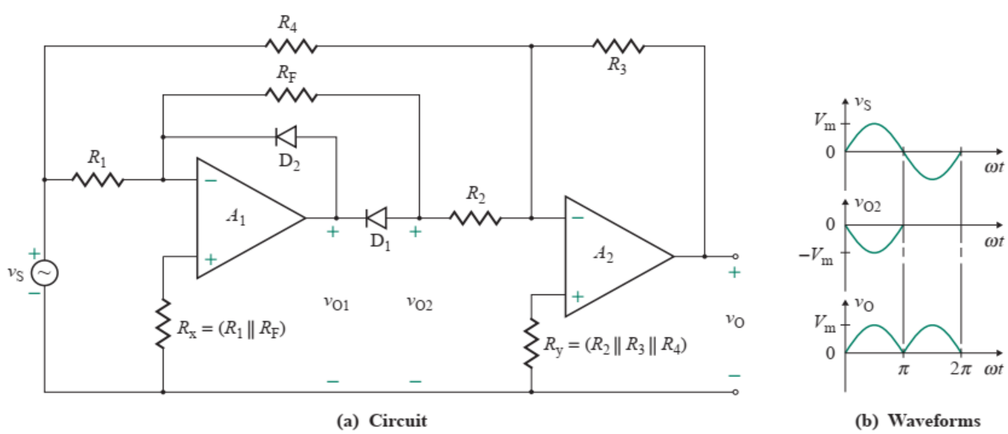


Figure 9.2

Procedure:

1. Connect the circuit as shown in Fig. 9.2.
2. Use $R_1 = R_2 = R_F = 1K\Omega$, $R_x = R_y = 0\Omega$ and $R_4 = R_3 = 2K\Omega$
3. Apply $v_i = 2sin\omega t$ V at the input of the circuit from the signal generator, where f= 1 KHz.
4. Adjust the CRO to measure the input and output voltage.
5. Observe input-output at the oscilloscope.

Discussion and Conclusion:

Discuss practical applications and advantages of precision rectifiers.

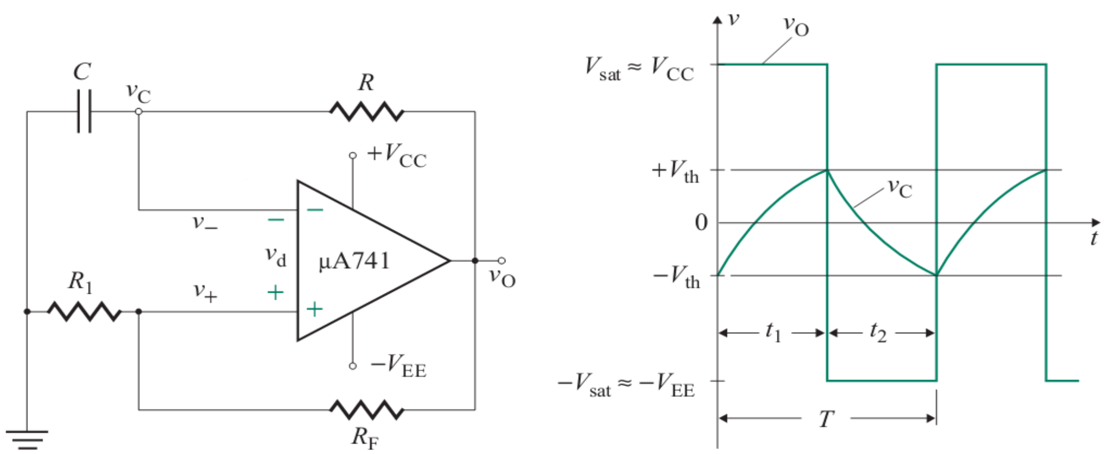
Summarize the key findings and any challenges encountered during the lab.

Exp. No: 10

Title: Experimental study on Op-amp based square wave generator

Objective: The objective of this lab is to design, construct, and analyze an operational amplifier-based square wave generator. The square wave generator utilizes an operational amplifier configured as an astable multivibrator to produce a stable square wave output.

Circuit Diagram:



(a) Circuit (b) Waveforms

Figure 10

Design Procedure:

The steps used to design the square-wave generator are as follows:

Step 1. Choose the value of R_1: Let $R_1 = 10\ \text{k}\Omega$.

Step 2. To simplify the design, choose $R_F = 1.164R_1$ and find R_F:

$$R_F = 1.164R_1 = 1.164 \times 10\ \text{k}\Omega = 11.64\ \text{k}\Omega \quad \text{(use a 20-k}\Omega\text{ potentiometer)}$$

Step 3. Choose a value of C: Let $C = 0.01\ \mu\text{F}$.

Step 4. Find the value of R

$$R = \frac{1}{2Cf_o} = \frac{1}{2 \times 0.01\ \mu\text{F} \times 5\ \text{kHz}} = 10\ \text{k}\Omega$$

Required Apparatus:

1. Operational Amplifiers (Op-Amps) - IC741
2. Breadboard
3. Oscilloscope
4. Function Generator
5. Connecting wires
6. Power supply
7. Resistors (Write the values)
8. Capacitors (Write the values)

Experimental Procedure:

1. Construct the circuit of Fig. 10 (a) using op-amp.
2. Choose appropriate values for R_1, R_F, and R, and C to determine the frequency of the square wave.
3. Observe the output waveform on the oscilloscope.
4. Measure the frequency of the generated square wave.

Discussion and Conclusion:

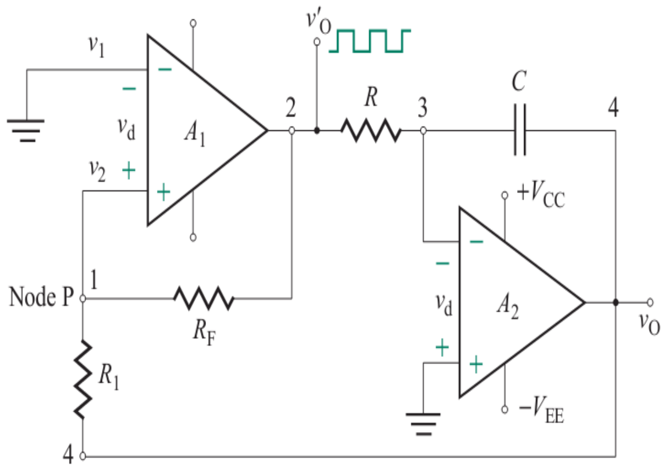
Compare the frequency obtained from the theoretical calculations with the measured frequency. Explore potential applications of operational amplifier-based square wave generators.

Exp. No: 11

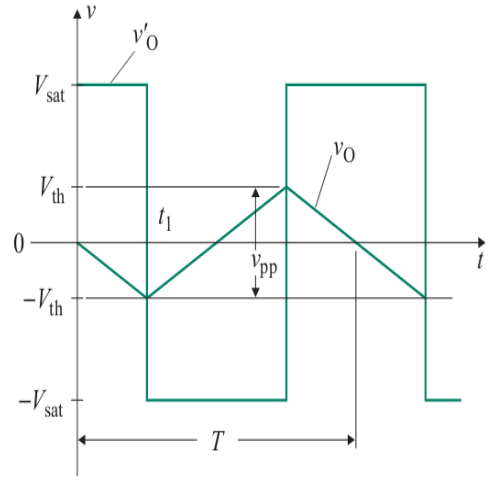
Title: Experimental study on Op-amp based triangular wave generator

Objective:

The objective of this lab is to design, construct, and analyze an operational amplifier-based triangular wave generator. The triangular wave generator utilizes an operational amplifier configured as an astable multivibrator to produce a stable triangular wave output.



(a) Circuit



(b) Waveforms

Figure 11

Design Procedure:

Designing a triangular-wave generator

Design the triangular-wave generator so that $f_o = 4$ kHz and $V_{th} = |-V_{th}| = 5$ V. Assume $+V_{sat} = |-V_{sat}| = 14$ V.

The steps used to design the triangular-wave generator are as follows:

Step 1. Find the values of R_1 and R_F.

$$\frac{R_1}{R_F} = \frac{V_{th}}{V_{sat}} = \frac{5}{14} = 0.36$$

Let $R_1 = 10\ \text{k}\Omega$; then

$$R_F = \frac{R_1}{0.36} = \frac{10\ \text{k}\Omega}{0.36} = 28\ \text{k}\Omega \quad \text{(use a 30-k}\Omega\text{ potentiometer)}$$

Step 2. Choose a suitable value of C: Let $C = 0.01\ \mu$F.

Step 3. Find the value of R.

$$R = \frac{R_F}{4 f_o C R_1}$$

$$= \frac{28\ \text{k}\Omega}{4 \times 4\ \text{kHz} \times 0.01\ \mu\text{F} \times 10\ \text{k}\Omega} = 17.5\ \text{k}\Omega \quad \text{(use a 20-k}\Omega\text{ potentiometer)}$$

Required Apparatus:

1. Operational Amplifiers (Op-Amps) - IC741
2. Breadboard
3. Oscilloscope
4. Function Generator
5. Connecting wires

6. Power supply
7. Resistors (Write the values)
8. Capacitors (Write the values)

Experimental Procedure:

1. Construct the circuit of Fig. 11 (a) using op-amp.
2. Choose appropriate values for R_1, R_F, and R, and C to determine the frequency of the triangular wave.
3. Observe the output waveform on the oscilloscope.
4. Measure the frequency of the generated triangular wave.

Discussion and Conclusion:

Compare the frequency obtained from the theoretical calculations with the measured frequency. Explore potential applications of operational amplifier-based triangular wave generators.

Exp. No: 12

Title: Experimental study on rectangular and square wave generator using 555 timer

Objectives: The objective of this lab is to design, construct, and analyze 555 timer-based rectangular and square wave generators.

Circuit Diagram:

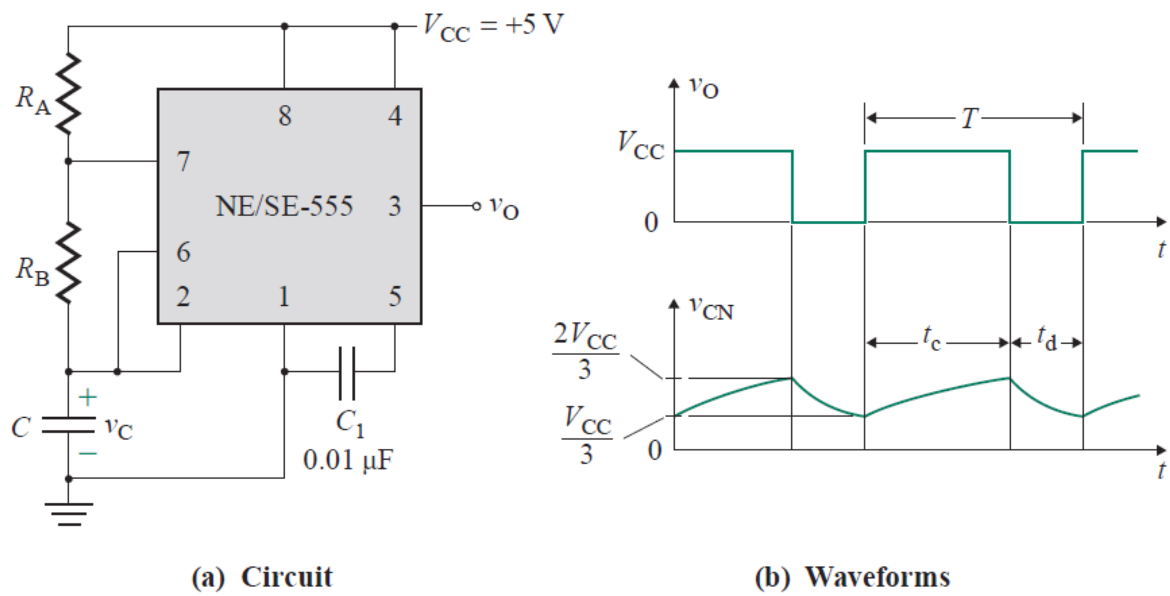


Figure 12.1

Design Procedure:

Design an astable multivibrator so that $k = 75\%$ and $fo = 2.5\ kHz$.

$k = 75\% = 0.75$, and $T = 1/f_o = 1/2.5$ kHz $= 400$ μs. The steps used to design an astable multivibrator are as follows:

Step 1. Find the charging time t_c and the discharging time t_d:

$$t_c = kT$$
$$= 0.75 \times 400\ \mu s = 300\ \mu s$$
$$t_d = (1 - k)T$$
$$= (1 - 0.75) \times 400\ \mu s = 100\ \mu s$$

Step 2. Chose a suitable value of C; Let $C = 0.1\mu F$

Step 3. Find the value of R_B.

$$R_B = \frac{t_d}{0.69C} = \frac{100\mu s}{0.69 \times 0.1\mu F} = 1449\Omega$$

Step 4. Find the value of R_A.

$$t_c = 0.69(R_A + R_B)$$

$$R_A = \frac{t_c}{0.69C} - R_B = \frac{300\mu s}{0.69 \times 0.1\mu F} - 1449 = 2889\Omega$$

Now design an astable multivibrator so that $k = 50\%$ and $fo = 2.5\ kHz$.

$$t_c = t_d = KT = 0.5 \times 400 = 200\ \mu s$$

Chose a suitable value of C; Let $C = 0.1\mu F$

$$R_B = \frac{t_d}{0.69C} = \frac{200\mu s}{0.69 \times 0.1\mu F} = 2899\Omega$$

$$t_c = 0.69(R_A + R_B)$$

$$R_A = \frac{t_c}{0.69C} - R_B = \frac{200\mu s}{0.69 \times 0.1\mu F} - 2899 = 2901\Omega$$

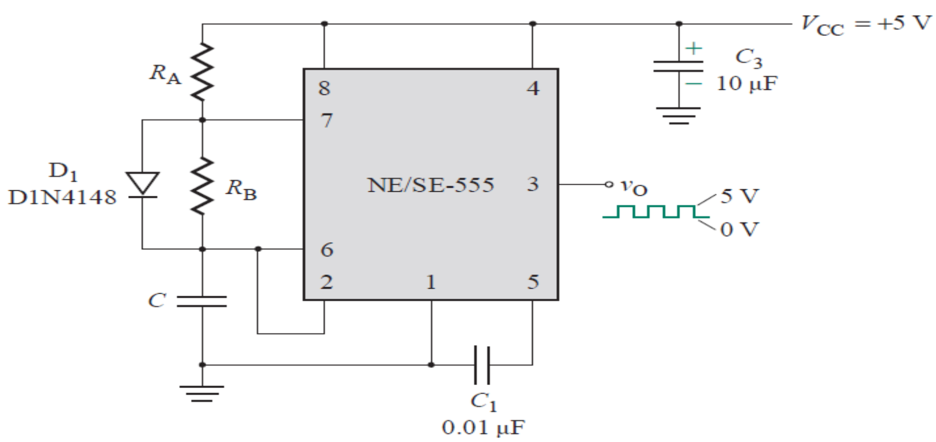


Figure 12.2

Required Apparatus:

1. 555 timer
2. Breadboard
3. Oscilloscope
4. Function Generator
5. Connecting wires
6. Power supply
7. Resistors (Write the values)
8. Capacitors (Write the values)
9. Diode (Write diode number)

Experimental Procedure:

1. Construct the circuit of Fig. 12.1 using 555 timer.
2. Choose appropriate values for R_A, R_B, C, and C_1 to determine the frequency of the output wave.
3. Observe the output waveform on the oscilloscope. Measure output signal frequency.
4. Measure the frequency of the generated square wave.
5. Repeat the above procedure and construct the circuit of Fig. 12.2 using 555 timer to generate the square wave. Measure output signal frequency.

Conclusion:

Summarize the key findings of the experiments.

Exp. No: 13

Title: Design of first order active low pass Butterworth Filter

Objectives: The main objective of this experiment is to plot the frequency response curve of first order active low pass Butterworth Filter.

Circuit Diagram:

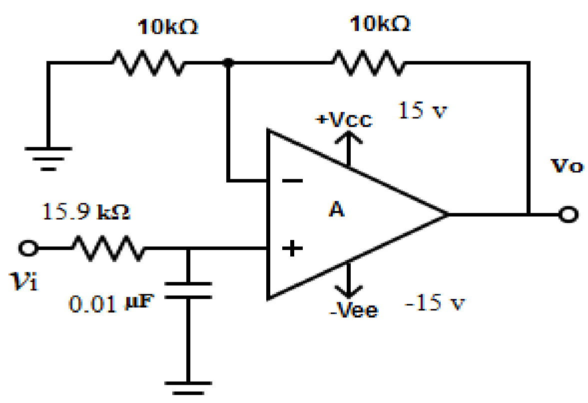


Figure 13.1

Required Apparatus:

1. Operational Amplifiers (Op-Amps) - IC741
2. Breadboard
3. Oscilloscope
4. Function Generator
5. Connecting wires
6. Power supply
7. Resistors (Write the values)
8. Capacitors (Write the values)

Procedure:

1. Connect the circuit as shown in Fig. 13.1.
2. Supply 1(p-p) Volt sine wave from the signal generator to the input.
3. Adjust the CRO to measure the input and output voltage.
4. Observe input-output voltage wave-shapes by gradually increasing the input frequency from low to high.
5. Fill up the table.
6. Take a semi-log graph paper and plot a curve Gain(dB) versus frequency as per scale.

Data Analysis: Fill up the following Table:

f	|Vi| in Volt	$\|v0/vi\|$	$20\|vo/vi\|$
	1		

Discussion and Conclusion: Summarize the key findings of the experiments.

Report writing marks distribution:

Category	**Marks**
Cover page that must include exp. no., exp. Title, student name, SID, SID of group members	1
Theory/ Circuit Diagram + Objectives	1
Experimental Procedure + Related mathematical formulas	1
Input signals	1
Output Signals	4
Discussion + Conclusion	2
Total	10

Lab Test:

Following the completion of all laboratory experiments, students are required to undergo an individual lab test. This examination aims to assess each student's competency in applying the knowledge gained during the experiments.

Lab test marks distribution:

Category	**Marks**
Write Title on answer script	1
Draw Circuit Diagram on answer script	2
Experimental set up Connection	2
Working of Circuit: show input and output waves on oscilloscope	1+2
Show Data collection/ any output graphs on answer script	2
Total	10

Quiz:

Upon concluding all laboratory experiments, students must participate in a quiz test. The quiz will encompass fundamental concepts from the conducted experiments, featuring various question formats such as short questions, True/False inquiries, and fill-in-the-gaps exercises.

Marks Distribution of this sessional course:

Attendance	**10**
Report writing	**30**
Lab Test	**10**
Lab viva	**05**
Quiz	**20**
Total	**75**