Islamic University of Gaza

Faculty of Engineering

Electrical Engineering Department

Electronics 1 Laboratory Manual

Developed by:

Eng. Nael Sofyan Dokhan

Eng. Israa Hamdi Abu Rayya

Supervisor:

Dr. Jawdat Abu-Taha

2019

Experiment 1

Introduction to Electronics 1 Laboratory

❖ Objectives

- To be familiar with electronic components.
- To be familiar with lab instruments and its functions.
- To be familiar with the LTspice simulation.

❖ Theory

There are numerous basic electronic components that are used for building electronic circuits. Without these components, circuit designs are never complete or didn't function well. These components include resistors, diodes, capacitors, integrated circuits, and so on. Some of these components consists of two or more terminals which are soldered to circuit boards, here is a brief overview on each of these basic electronic components.

In designing of an electronic circuit following are taken into consideration:

- Basic electronic components: capacitors, resistors, diodes, transistors, etc.
- Power sources: Signal generators and DC power supplies.
- Measurement and analysis instruments: Oscilloscope, Digital Multimeter, etc.

➢ <u>Basic Electronics Components:</u>

An electronic circuit comprises of various types of components, which are classified into two types: active components like transistors, diodes, IC's; and passive components like capacitors, resistors, inductors, etc.

a. Passive Electronic Components

These components can store or maintains energy either in the form of current or voltage. Some of these components are discussed below.

1. Resistors

A resistor is a two-terminal passive electronic component, used to oppose or limit the current. Resistor works based on the principle of Ohm's law which states that voltage applied across the terminals of a resistor is directly proportional to the current flowing through it $V=IR$.

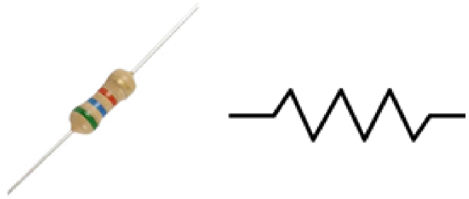


Figure 1 Resistor and circuit symbol

2. Capacitors

A capacitor made from two conductive plates with an insulator between them and it stores electrical energy in the form of an electric field. A capacitor blocks the DC signals and allows the AC signals. The stored charge is Q=CV

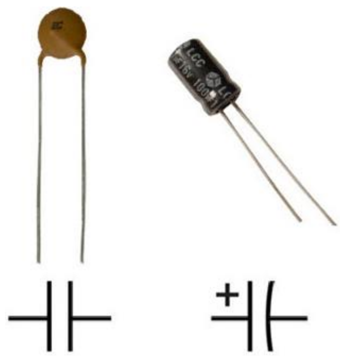


Figure 2 Capacitor and circuit symbol

3. Inductors

An inductor is also referred as AC resistor which stores electrical energy in the form of magnetic energy. It resists the changes in the current and the standard unit of inductance is Henry. Capability of producing magnetic lines is referred as inductance.

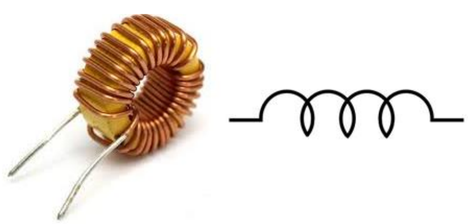


Figure 3 inductor and circuit symbol

b. Active Electronic Components:

These components rely on a source of energy and are able to control the electron flow through them. Some of these components are semiconductors like diodes, transistors, integrated circuits, various displays like LCD, LED, and CRTs.

1. Diodes

A diode is a device that allows current to flow in one direction and usually made with semiconductor material. It has two terminals, anode and cathode terminals. These are mostly used in converting circuits like AC to DC circuits. These are of different types like Rectifier diodes, Zener diodes, LEDs, photo diodes, etc.

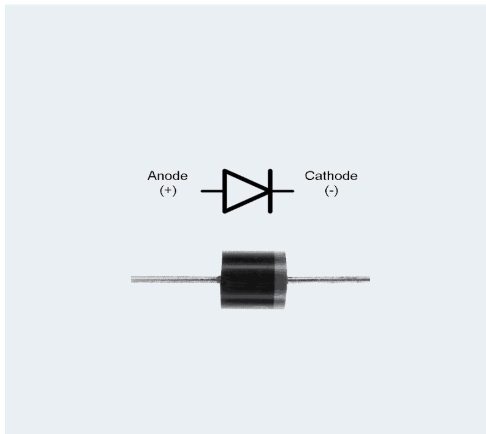


Figure 4 Rectifier diode and circuit symbol

2. Transistors

A transistor is a three terminal semiconductor device. Mostly it is used as switching device and also as an amplifier. This switching device can be a voltage or current controlled. By controlling the voltage applied to the one terminal controls the current flow through the other two terminals. Transistors are of two types, namely bipolar junction transistor (BJT) and field effect transistors (FET). And further these can be PNP and NPN transistors.

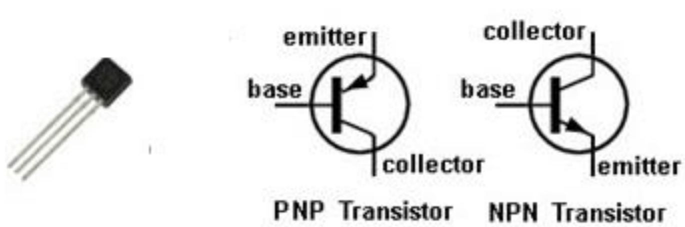


Figure 5 Transistor and circuit symbol

➢ Power sources:

1. The Power Supply

Sometimes in electronics lab components, you may need to supply voltages other than those available on the trainer boards. For this purpose, you will be using a power supply Figure 6. The power supplies present in the lab are able to provide 2 independent variable voltages ranging from 0V to up to 30V and a fixed voltage supply of 5V. Each voltage output block consists of positive terminal (red) and a negative terminal (black). In some cases, a third terminal labeled Gnd (yellow/green) may be present but this can be ignored. The voltage being supplied by the power supply is essentially the voltage of the positive terminal with respect to the negative terminal. To generate positive voltages, the negative terminal is shorted with the ground of the circuit and voltage is obtained from the positive terminal.

Figure 6 Dc power supply

2. The Function Generator

The function generator Figure 7 is used to produce wave forms of variable frequency, amplitude and offset. The generators in the lab are capable of producing sine waves, triangle/saw-tooth waves and square waves. The principle of voltage generation is again similar to the power supply in which the negative (black) clip is connected to the ground and the voltage at the positive (red) clip oscillates with respect to it. While you may be familiar with the terms frequency and amplitude, offset is a relatively new yet equally important quantity.

Figure 7 Function generator

- <u>Measurement and analysis instruments:</u>

1. The Digital Multi Meter (DMM)

The multimeter Figure 8 as suggested by its name, is used to measure multiple parameters in a circuit. It can be used to measure voltage, current, resistance, capacitance and it even has a mode for checking for short circuits. In logic circuits, you will typically be using only the voltage.

Figure 8 Digital multimeter

2. The Oscilloscope

The oscilloscope Figure 9 (a) is one of the most interesting pieces of equipment that you will be using in the lab. Although it can only be used to measure voltages, it allows for a large amount of functions to be performed on the acquired data. It is particularly helpful in measuring voltages that change with time as it is able to not only plot the wave form but also automatically measure many parameters associated with the wave. The probes of an oscilloscope are similar to the one shown in Figure 9(b) The small black clip is connected to the ground of the circuit whereas the larger grey part of the probe is connected to the point whose voltage needs to be measured.

(a)

(b)

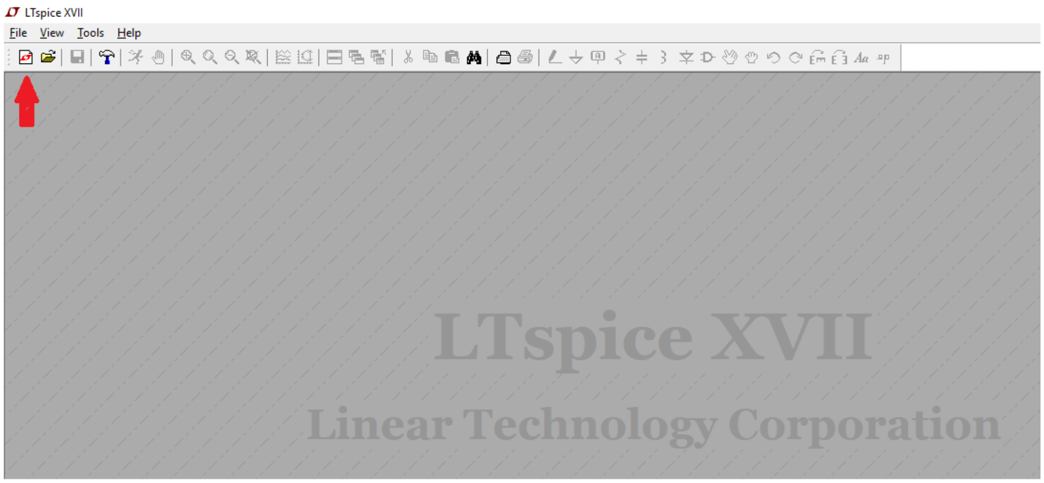
Figure 9 (a) Oscilloscope (b) Cable of oscilloscope

➢ <u>LTspice Simulation</u>

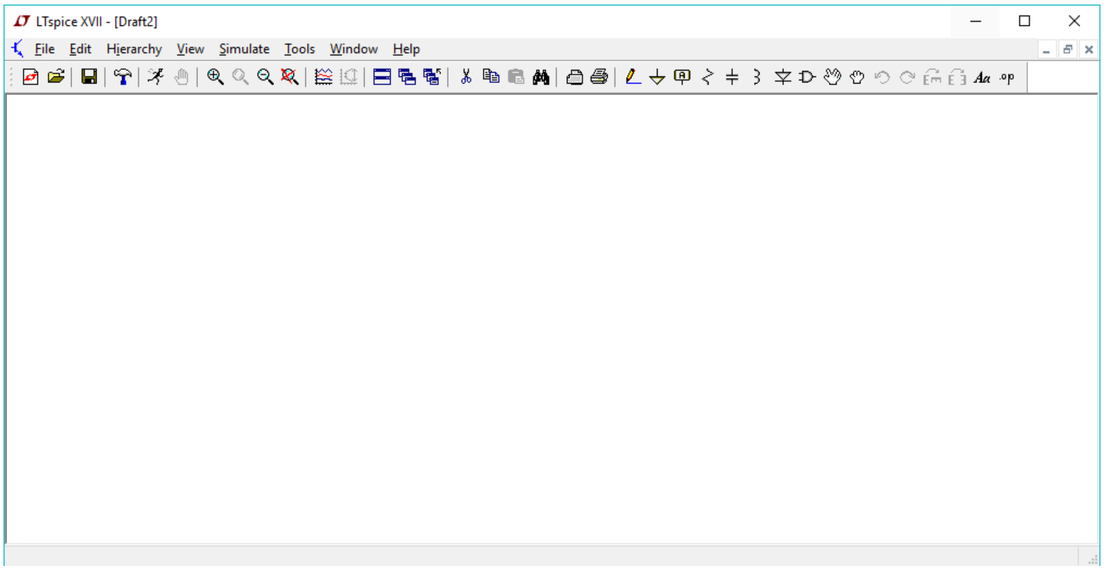
SPICE (Simulation Program for Integrated Circuit Engineering) is a powerful general-purpose analog and mixed-mode circuit simulator that is used to verify circuit designs and to predict the circuit behavior.

- Basic Setup & Getting Started

After opening LT Spice, you can start a new schematic or open an existing one. To start a new schematic, press the new schematic button in the upper left of the window:



(a)



(b)

Figure 10 Initial Window (a) before (b) after Opening new schematic

- Building a Circuit

These are the basic tools on the toolbar along the top of the Schematic Editor. They can all be accessed with hotkeys which are intuitive for common components: (R) Resistor, (L) Inductor, (C) Capacitor, (G) Ground and (D) Diode.

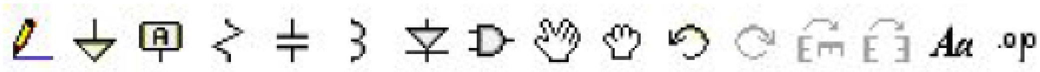


Figure 11 Circuit Building Toolbar

- ## Important Default Hotkeys

These are a few of the important default hotkeys that you will use constantly when working in LT spice.

Table 1 Hotkeys in LTspice

Shortcut	Symbol	Function	Description
F2		Component Catalog	Open component catalog for selecting
F3		Draw Net	Draw wires between components
F4		Create Port	Create a port for naming nodes in the circuit or create a ground
F5		Delete	Click on a component or wire to delete, or drag an area to delete objects in the area
F6		Copy	Click an element or wire to create a copy of it or drag to copy a group of parts
F7		Grab	Grab and move an element, disconnects from circuit. Drag to select multiple parts.
F8		Drag	Grab and move an element, keeps current connections. Drag an area to select multiple parts
F9		Undo	Undoes previous action
G		Ground	Create Ground node
R*		Resistor	Place a resistor
C*		Capacitor	Place a capacitor
L*		Inductor	Place an inductor
D*		Diode	Place a diode
Ctrl-R		Rotate	Rotate part 90 degrees clockwise
Ctrl-E		Mirror	Flip part horizontally
T		Text	Place text (for documentation/notes)
S		SPICE Directive	Place SPICE directive

In this lab we will learn two type of analysis:

- Dc operation points.
- Transient analysis.
- DC sweep analysis.

1. Dc operation points:

We want to analysis the voltage divider circuit shown in Figure 12.

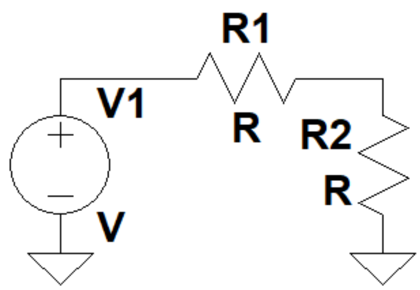


Figure 12 Voltage Divider Circuit

- <u>Procedures:</u>
  1. Drawing the Circuit in Schematic Window as shown in Figure 12.
  2. Put V=12v, R1=R2=1kΩ.
  3. Go to Simulate →Edit Simulation CMD. Choose the type of analysis you want to simulate→DC op pnt as shown in Figure 13 then ok.
  4. After finished Click on  button or go to Simulate>Run to run your simulation.
  5. A simulation showing only the DC operating points as shown in Figure 14.

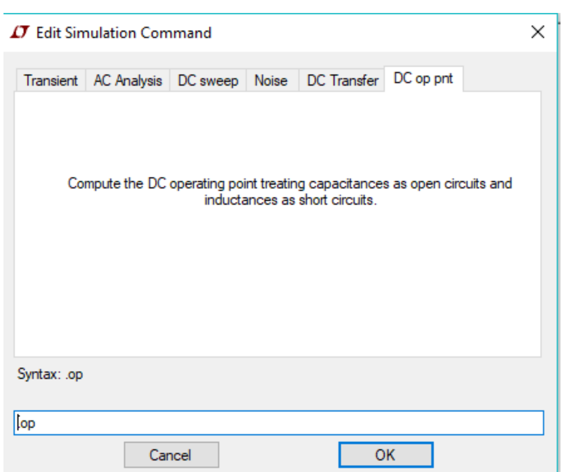


Figure 13 Dc operating point simulation

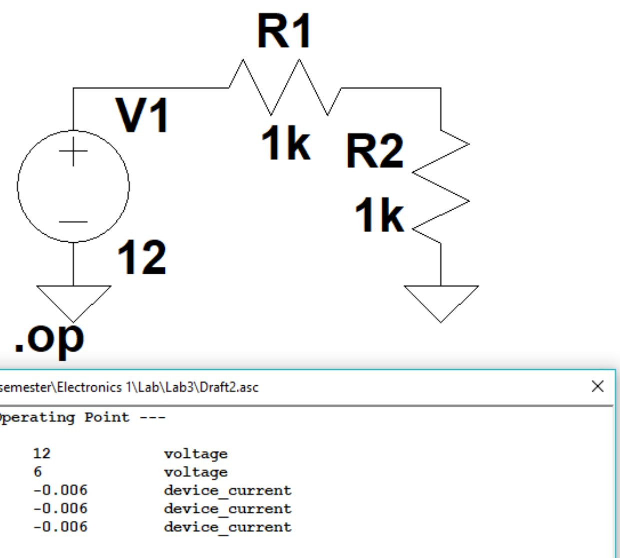


Figure 14 Dc values for electric circuit

2. Transient analysis

We want to analysis the voltage divider circuit shown in Figure 12.
Note: Here I want the voltage source to be AC voltage (sine wave)

- <u>Procedures:</u>
  1. Drawing the Circuit in Schematic Window as shown in Figure 12.
  2. To get the Ac voltage source right click on voltage then click on the advanced button as shown in Figure 15.

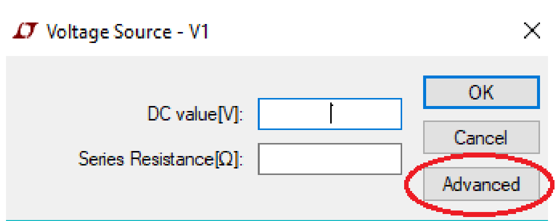
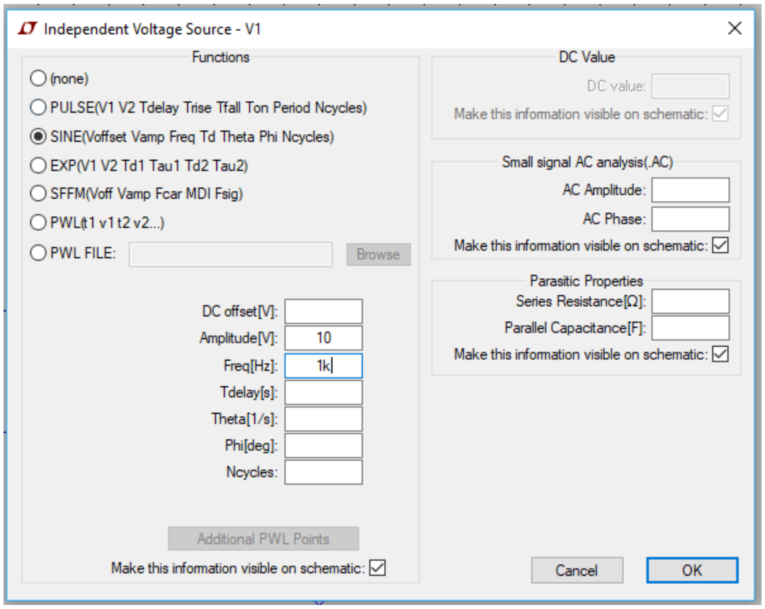
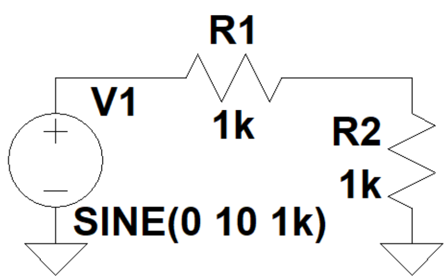


Figure 15 Select Ac voltage Source

  3. Select the source type (sinusoidal, pulse, piecewise linear etc.) then Edit the sources as the desired circuit as shown in Figure 16.



(a)



(b)

Figure 16 (a) Choose Sinusoidal (b) Sinusoidal Voltage Source

4. Run the simulation then choose transient analysis and put the stop time equal 10ms.

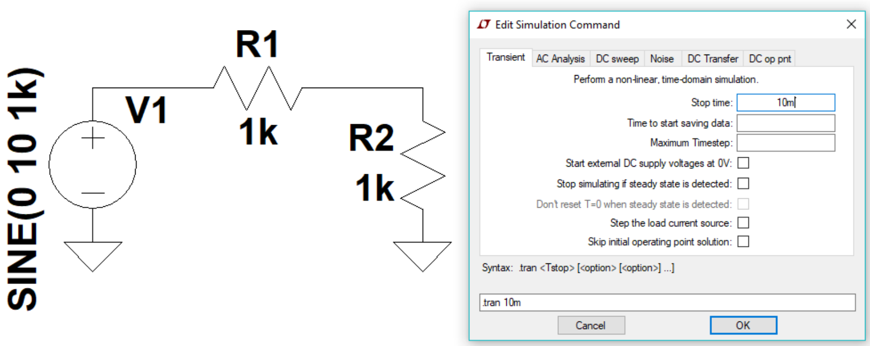


Figure 17 Transient Simulation

5. After running the simulation, the simulation showing the transient analysis will look like as in Figure 18.

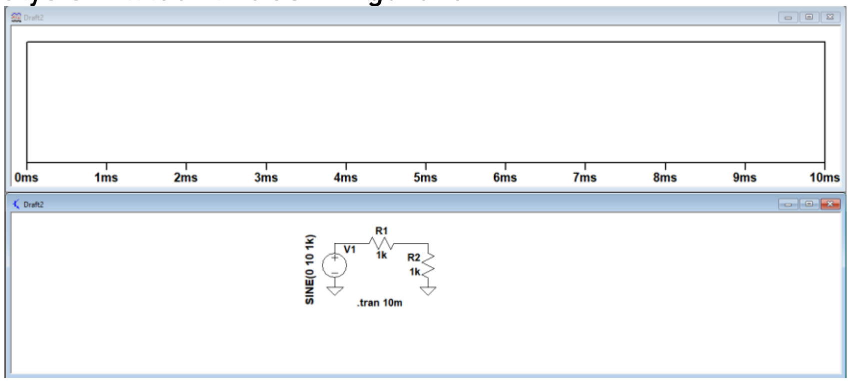


Figure 18 waveform and Schematic Window

6. To draw the voltage at any node by using voltage probe cursor, by left click on any wire to plot the voltage on the waveform viewer.

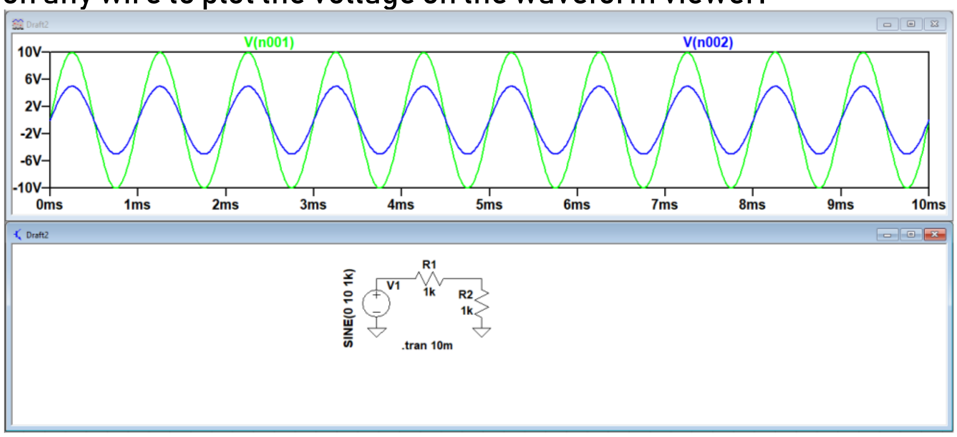


Figure 19 Voltage Waveform for voltage source and R2

7. To plot the current passes through any element. You can use the current probe cursor. By left click on the body of the component.

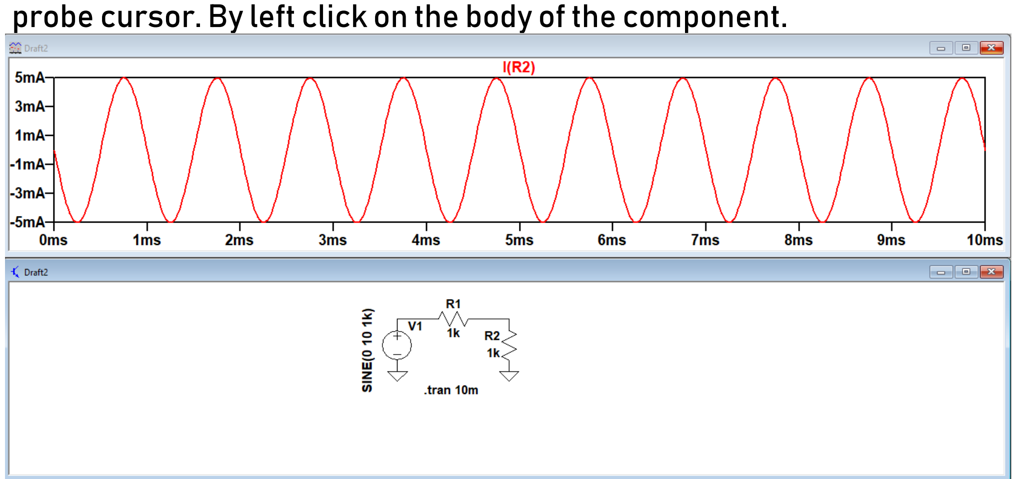


Figure 20 Current waveform passes on R2

8. To measure the RMS value for any signal, hold down Ctrl and left click on the I or V trace label in the waveform viewer.
   For example, the RMS value for input signal equal $\frac{Vm}{\sqrt{2}} = \frac{10}{\sqrt{2}} = 7.07$

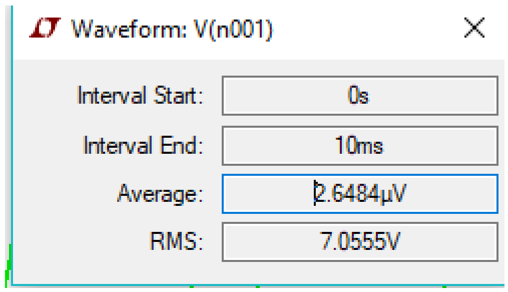


Figure 21 DC and RMS vales for voltage source

3. DC sweep analysis

DC Sweep Analysis is used to calculate a circuits' bias point over a range of values. This procedure allows you to simulate a circuit many times, sweeping the DC values within a predetermined range. You can control the source values by choosing the start and stop values and the increment for the DC range.

We want to make Dc sweep analysis for the circuit shown in Figure 12.

- Procedures:
  1. Drawing the Circuit in Schematic Window as shown in Figure 12.
  2. Put V=12v, R1=R2=1kΩ.
  3. Run the simulation then choose Dc sweep and fill the command window with appropriate values as shown in Figure.

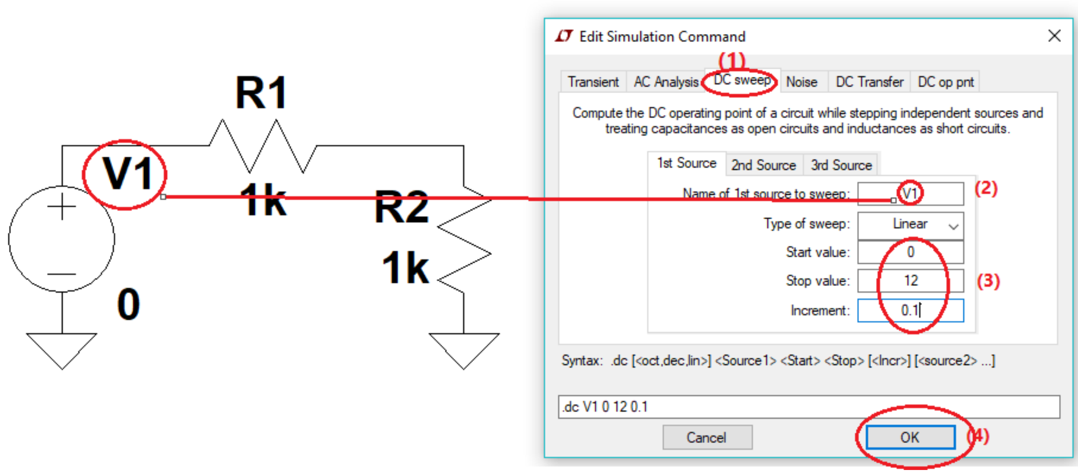


Figure 22 DC Sweep Simulation

4. After running the simulation, draw the Voltage source and Voltage on R2.

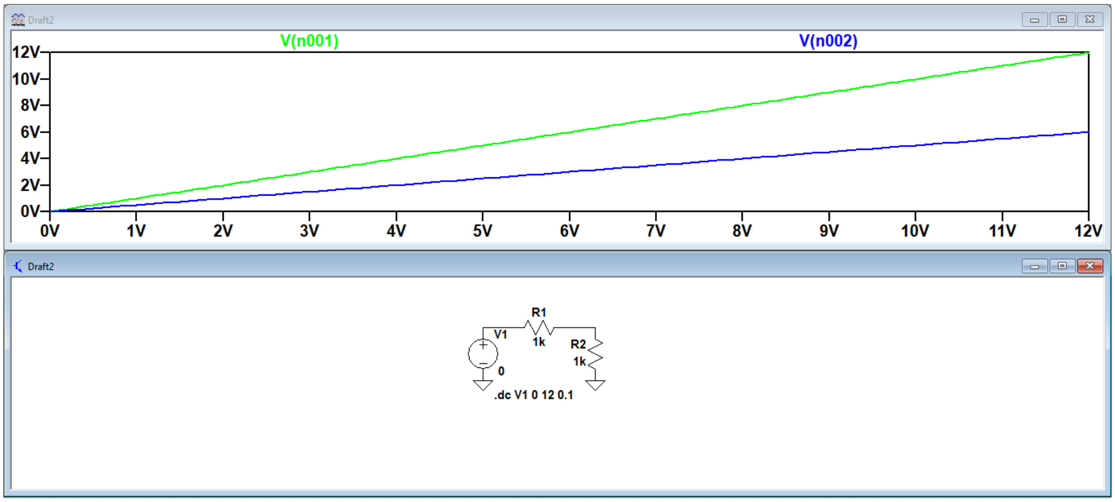


Figure 23 Voltage Waveform for voltage source and R2

Note: that at the dc sweep, the X axis represent the voltage that will increase from value to another; Here from 0V to 12V. But in transient simulation, the X axis represent the time.

Experiment 2
DC Characteristics of Diodes

❖ Objectives

- To plot V/I characteristics curve of semiconductor diode.
- To study How to check diode using multi-meter.
- To be familiar how can be simulate diode circuit with the LTspice simulation.

❖ Equipment

- Diode (1N4007).
- Resistors.
- Dc power supply.
- Digital Multi-meter.

❖ Theory

Donor impurities (pentavalent) are introduced into one-side and acceptor impurities into the other side of a single crystal of an intrinsic semiconductor to form a p-n diode with a junction called depletion region (this region is depleted off the charge carriers). This region gives rise to a potential barrier called Cut-in Voltage. This is the voltage across the diode at which it starts conducting. The P-N junction can conduct beyond this potential.

The P-N junction supports uni-directional current flow. If +ve terminal of the input supply is connected to anode (P-side) and –ve terminal of the input supply is connected the cathode. Then diode is said to be forward biased. In this condition the height of the potential barrier at the junction is lowered by an amount equal to given forward biasing voltage. Both the holes from p-side and electrons from n-side cross the junction simultaneously and constitute a forward current from n-side (injected minority current – due to holes crossing the junction and entering P- side of the diode). Assuming current flowing through the diode to be very large, the diode can be approximated as short- circuited switch.

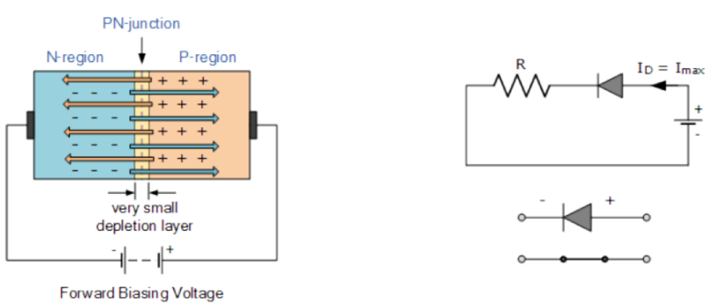


Figure 1 Forward Biasing P-N junction

If –ve terminal of the input supply is connected to anode (p-side) and +ve terminal of the input supply is connected to cathode (n-side) then the diode is said to be reverse biased. In this condition an amount equal to reverse biasing voltage increases the height of the potential barrier at the junction. Both the holes on P-side and electrons on N-side tend to move away from the junction there by increasing the depleted region. However the process cannot continue indefinitely, thus a small current called reverse saturation current continues to flow in the diode. This current is negligible hence the diode can be approximated as an open circuited switch.

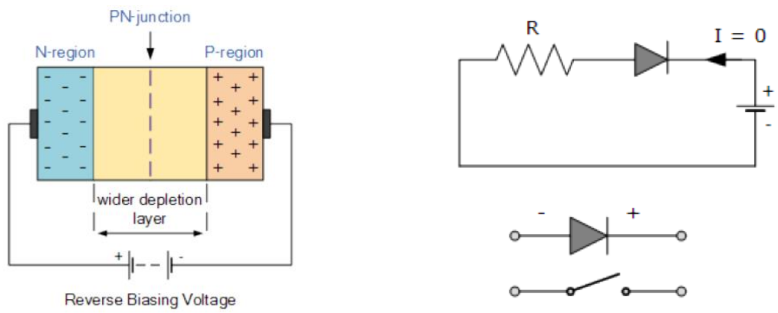


Figure 2 Reverse Biasing P-N Junction

The general relationship between the current and the voltage of a diode is as follows:

$$I_D = I_S(e^{\frac{V_D}{\eta V_T}} - 1)$$

Where

- I_D is the diode current.
- I_S is called the reverse saturation current and its value is constant for a specific diode. Usually this is a very small value of current since virtually no current flows in the reverse direction.
- V_Ds the applied voltage between the terminals of the diode.
- η is another constant whose value ranges from 1 to 2 depending on the material of the semiconductor diode.
- V_T is the thermal equivalent voltage. It is equal to $\frac{KT}{q}$ where
- K is the Boltzman's constant (1.38e-23 J/K),
- T is the temperature in Kelvins
- q is the basic electronic charge (1.6e-19C).
- The value of V_T at the room temperature is usually taken as 26mV.

- <u>V/I characteristics curve:</u>

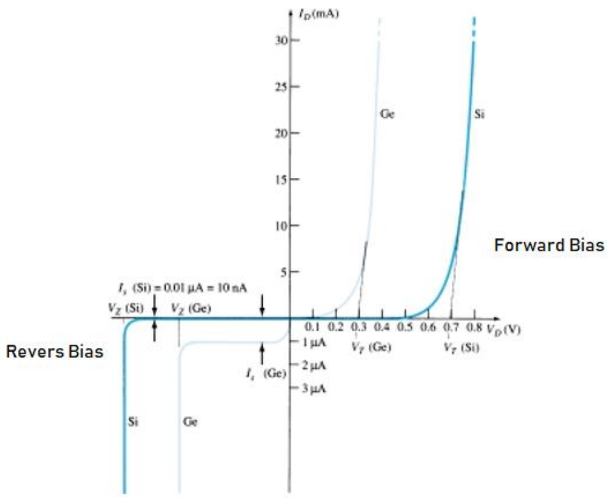


Figure 3 Silicon and germanium semiconductor diode characteristics

- <u>Types of Diode:</u>

There are many types of diodes we will learn about the most famous:

1. Rectifier Diode

   Diodes are widely used semiconductor device. A rectifier diode is a two-lead semiconductor that allows current to pass in only one direction. Generally, P-N junction Diode is formed by joining together n-type and p-type semiconductor materials. The P-type side is called the anode and the n-type side is called the cathode. Many types of diodes are used for a wide range of applications. Rectifier diodes are a vital component in power supplies where they are used to convert AC voltage to DC voltage.

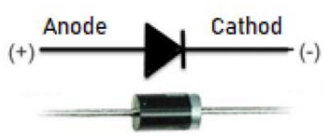


Figure 4 Rectifier Diode

2. Light Emitting Diode (LED)

   The light emitting diode is also a type of P-N junction diode that emits light in the forward bias configuration. LED is made up of a direct-band semiconductor. When the charge carriers (electrons) cross the barrier and recombine with electron holes on the other side, they emit photon particles (light). While the color of the light depends on the energy gap of the semiconductor.

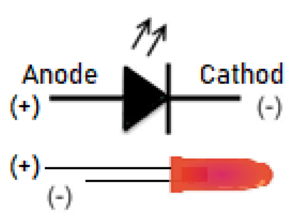


Figure 5 LED Diode

3. Zener Diode
   It is a passive element works under the principle of zener breakdown. It is similar to normal diode in forward direction, it also allows current in reverse direction when the applied voltage reaches the breakdown voltage. It is designed to prevent the other semiconductor devices from momentary voltage pulses. It acts as voltage regulator.

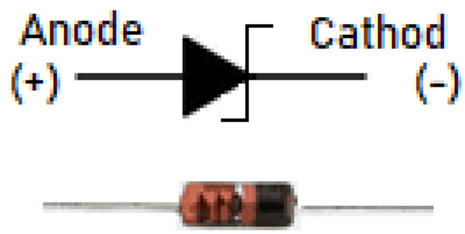


Figure 6 Zener Diode

➢ <u>How to check the diode using multi-meter?</u>

1. A multi-meter set to the Resistance mode (Ω).
2. A diode is forward-biased when the positive (red) test lead is on the anode and the negative (black) test lead is on the cathode.
   - ✓ The forward-biased resistance of a good diode should range from 250 Ω to 1 MΩ.
3. A diode is reverse-biased when the positive (red) test lead is on the cathode and the negative (black) test lead is on the anode.
   - ✓ The reverse-biased resistance of a good diode displays OL on a multi-meter. The diode is bad if readings are the same in both directions.

❖ Experiment Procedure

➢ Part1: Forward bias

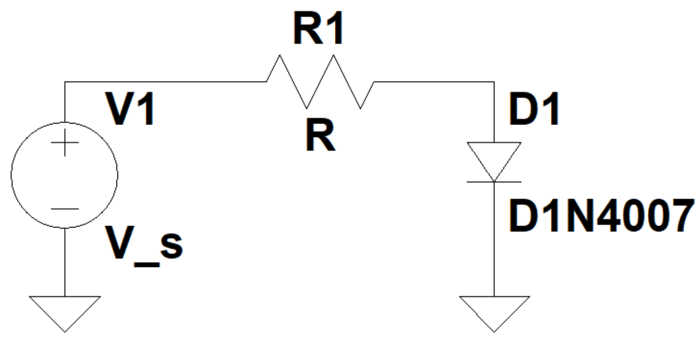


Figure 7 Forward Biased Diode Circuit

1. Connect the Circuit as shown in Figure 7.
2. Put R=330Ω.
3. Increase the voltage source from the power supply from 0V to 3as shown in table 1.
4. Measure voltage across diode and current through diode.
5. Draw VI characteristics for forward bias.
6. Simulate the circuit in LTspice Program.

Table 1 Forward Biased Diode Characteristics

V_S(v)	0.2	0.4	0.6	0.8	1	1.2	1.4	1.6	1.8	2	2.2	2.4	2.6	2.8	3
V_D(v)															
I_D(mA)															

➢ Part2: Reverse bias

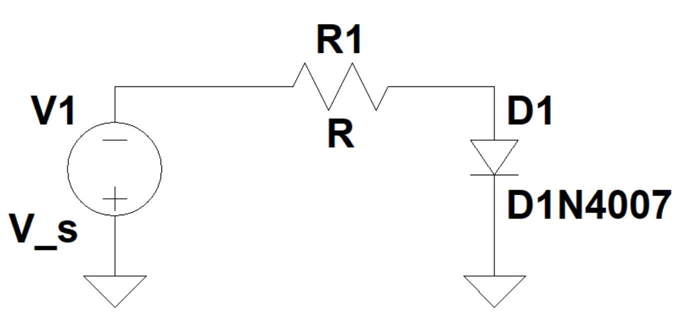


Figure 8 Reverse Biased Diode Circuit

1. Connect the Circuit as shown in Figure 8.
2. Put R=330Ω.
3. Increase the voltage source from the power supply from 0V to 3V as shown in table 2.
4. Measure voltage across diode and current through diode.
5. Draw VI characteristics for reverse bias.
6. Simulate the circuit in LTspice Program.

Table 2 Reverse Biased Diode Characteristics

V_S(v)	0.2	0.4	0.6	0.8	1	1.2	1.4	1.6	1.8	2	2.2	2.4	2.6	2.8	3
V_D(v)															
I_D(mA)															

Experiment 3
Half Wave Rectifier

❖ Objectives

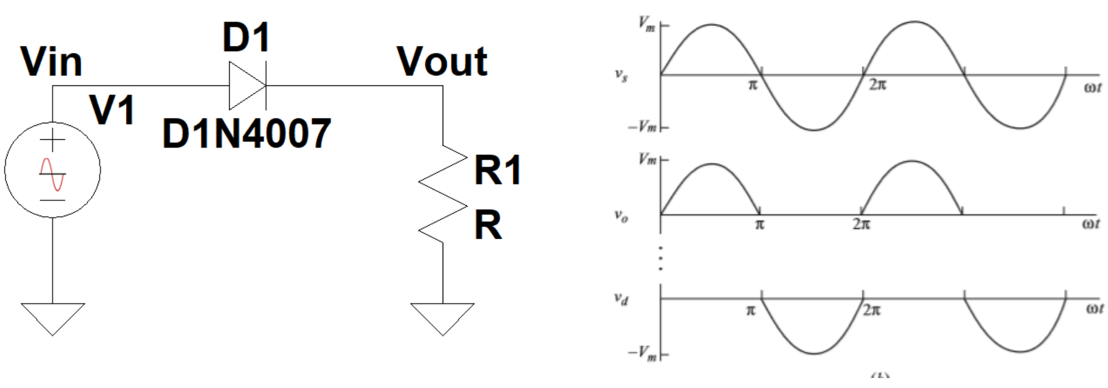
- To construct a half-wave rectifier circuit and analyze its output.
- To analyze the rectifier output using a capacitor in shunt as a filter.
- To be familiar how can be simulate half-wave-rectifier circuit using ltspice simulation.

❖ Equipment:

- Diode (1N4007).
- Resistors.
- Function generator.
- Oscilloscope.

❖ Theory

The process of converting an alternating current into direct current is known as rectification. The unidirectional conduction property of semiconductor diodes (junction diodes) is used for rectification. Rectifiers are of two types: (a) Half wave rectifier and (b) Full wave rectifier. In this lab we will discuss a half wave rectifier circuit as shown in Figure 1 (a), during the positive half-cycle of the input, the diode is forward biased and conducts. Current flows through the load and a voltage is developed across it. During the negative half-cycle, it is reverse bias and does not conduct. Therefore, in the negative half cycle of the supply, no current flows in the load resistor as no voltage appears across it. Thus, the dc voltage across the load is sinusoidal for the first half cycle only and a pure AC. input signal is converted into a unidirectional pulsating output signal.



(a) (b)

Figure 1 (a) Half Wave Rectifier Circuit with Resistive load (b) waveform voltages HWR when diode is ideal

- The basic parameters of the half wave rectifier circuit are:
  - <u>For the input:</u>
    - Average Value:

$$V_{dc_{avr}} = \frac{1}{T}\int_0^T v(t)dt = \frac{1}{2\pi}\int_0^{2\pi} V_m \sin\theta \, d\theta$$

$$= \frac{V_m}{2\pi}[-\cos\theta] = \frac{-V_m}{2\pi}[\cos 2\pi - \cos 0]$$

$$= \frac{-V_m}{2\pi}[1-1] = \mathbf{0}$$

    - RMS Value (effective value):

$$\boldsymbol{V_{rms}} = \sqrt{\frac{\mathbf{1}}{\boldsymbol{T}}\int_0^t \boldsymbol{v(t)^2\, dt}}$$

$$= \sqrt{\frac{1}{2\pi}\int_0^{2\pi} (V_m \sin\theta)^2 \, d\,\theta} = \sqrt{\frac{{V_m}^2}{2\pi}\int_0^{2\pi} \sin^2\theta \, d\theta}$$

$$= \sqrt{\frac{{V_m}^2}{2\pi}\int_0^{2\pi} \frac{1}{2}(1-\cos 2\theta)\, d\theta} = \sqrt{\frac{{V_m}^2}{2\pi}\left[\frac{1}{2}\left(\theta - \frac{\sin 2\theta}{2}\right)\right]}$$

$$= \sqrt{\frac{{V_m}^2}{2\pi} \times \frac{1}{2}\left[\left(2\pi - \frac{\sin 4\pi}{2}\right) - \left(0 - \frac{\sin 0}{2}\right)\right]} = \sqrt{\frac{{V_m}^2}{2\pi}[\pi]}$$

$$\boldsymbol{V_{rms}} = \frac{\boldsymbol{V_m}}{\sqrt{\mathbf{2}}}$$

  - <u>For the output:</u>
    - Average value:

$$\boldsymbol{V_{dc_{avr}}} = \frac{1}{T}\int_0^T v(t)dt = \frac{1}{2\pi}\int_0^{\pi} V_m \sin\theta \, d\theta + \int_{\pi}^{2\pi} 0 \, d\theta$$

$$= \frac{V_m}{2\pi}[-\cos\theta] + 0 = \frac{-V_m}{2\pi}[\cos\pi - \cos 0]$$

$$= \frac{-V_m}{2\pi}[-1-1] = \frac{\boldsymbol{V_m}}{\boldsymbol{\pi}}$$

- RMS Value (effective value):

$$V_{rms} = \sqrt{\frac{1}{T}\int_0^t v(t)^2\, dt}$$

$$= \sqrt{\frac{1}{2\pi}\int_0^\pi (V_m \sin\theta)^2\, d\theta} = \sqrt{\frac{V_m{}^2}{2\pi}\int_0^\pi \sin^2\theta\, d\theta}$$

$$= \sqrt{\frac{V_m{}^2}{2\pi}\int_0^\pi \frac{1}{2}(1-\cos 2\theta)\, d\theta} = \sqrt{\frac{V_m{}^2}{2\pi}\times\frac{1}{2}\left[\theta - \frac{\sin 2\theta}{2}\right]}$$

$$= \sqrt{\frac{V_m{}^2}{2\pi}\times\frac{1}{2}\left[\left(\pi - \frac{\sin\pi}{2}\right) - \left(0 - \frac{\sin 0}{2}\right)\right]} = \sqrt{\frac{V_m{}^2}{2\pi}\left[\frac{\pi}{2}\right]}$$

$$V_{rms} = \frac{V_m}{2}$$

Table1 Basic parameter of the HWR

HWR	V(avg)	V(rms)
Input	0	$\frac{Vm}{\sqrt{2}}$
Output	$\frac{Vm}{\Pi}$	$\frac{Vm}{2}$

➢ Rectifier output using a capacitor in shunt as a filter:

The output of a rectifier gives a pulsating DC signal as shown in Figure 2, because of presence of some AC. components whose frequency is equal to that of the AC supply frequency. Very often when rectifying an alternating voltage, we wish to produce a "steady" direct voltage free from any voltage variations or ripple. Filter circuits are used to smoothen the output. Various filter circuits are available such as shunt capacitor, series inductor. Here we will use a simple shunt capacitor filter circuit as shown in Figure3. Since a capacitor is open to DC and offers low impedance path to AC current, putting a capacitor across the output will make the DC component to pass through the load resulting in small ripple voltage.

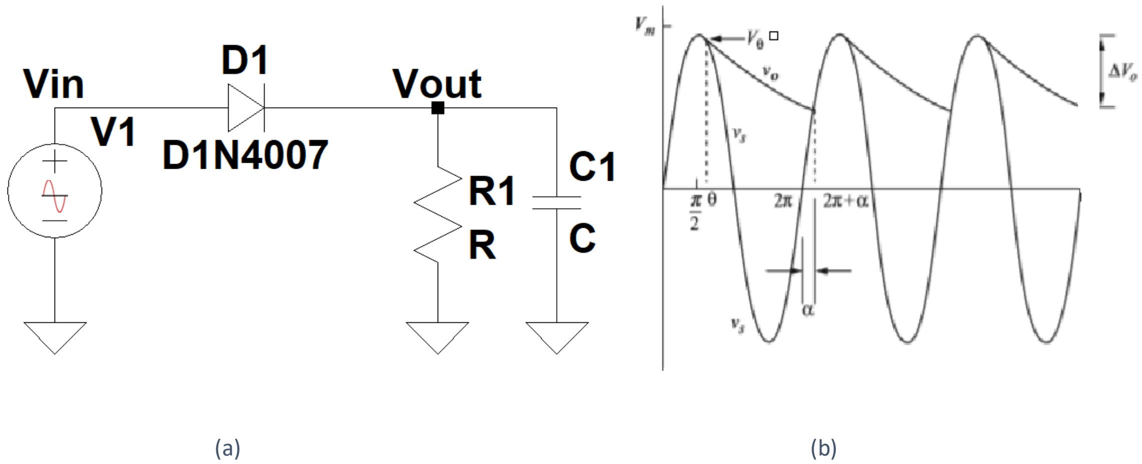


Figure 2 (a) Half Wave Rectifier Circuit with capacitive load(b) waveform voltages HWR

The working of the capacitor can be understood in the following manner. When the rectifier input voltage is increasing, the capacitor charges to the peak voltage Vm. Just past the positive peak the rectifier output voltage tries to fall. As the source voltage decreases below Vm, the capacitor will try to send the current back to diode making it reverse biased. Thus the diode separates/disconnects the source from the load and hence the capacitor will discharge through the load until the source voltage becomes more than the capacitor voltage. The diode again starts conducting and the capacitor is again charged to the peak value Vm and the process continues.

❖ Experiment Procedure:

➢ Part 1: HWR with resistive load

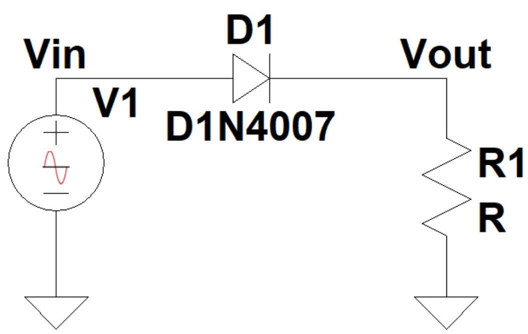


Figure 3 Half Wave Rectifier Circuit with Resistive load

1. Connect the circuit as shown in Figure 3.
2. Set function generator to a sine wave with 20Vp-p and 1 kHz.
3. Put resistor equal 1kΩ.
4. Measure the input and output voltage using oscilloscope.
5. Measure the V_{DC} ,and V_{rms} for output voltage in Table 2.
6. Simulate the circuit using LTspice program.
7. Repeat the previous step when
   a. F=1kHz
      i. R=4.7KΩ
      ii. R=10kΩ
   b. R=1kΩ
      i. F=10kHz
      ii. F=100kHz

Table 2 average, and effective output voltage

R(KΩ)	V_{DC}(V)	$V_{rms}(V)$
1		
4.7		
10		

Table 3 average, and effective output voltage

f(KHz)	V_{DC}(V)	$V_{rms}(V)$
1		
10		
100		

➢ Part 2: HWR with resistive load

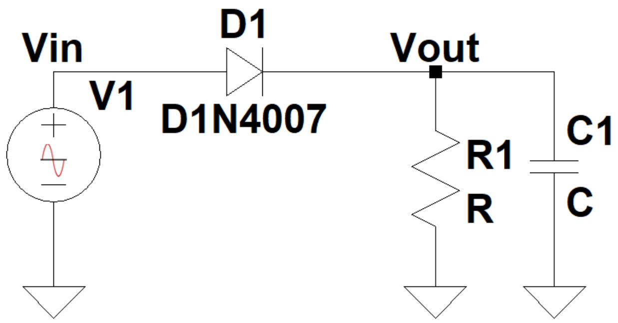


Figure 4 Half Wave Rectifier Circuit with capacitive load

1. Connect the circuit as shown in Figure 4.
2. Set function generator to a sine wave with 20Vp-p and 1 kHz.
3. Put R=1kΩ and C= 1µF.
4. Measure the input and output voltage using oscilloscope.
5. Simulate the same circuit using LTspice program.

Experiment 4
Full Wave Rectifier

❖ Objectives

- To construct a full-wave rectifier circuit and analyze its output.
- To analyze the rectifier output using a capacitor in shunt as a filter.
- To be familiar how can be simulate full-wave-rectifier circuit using ltspice simulation.

❖ Equipment:

- Diode (1N4007).
- Resistors.
- Function generator.
- Oscilloscope.

❖ Theory

The half-wave rectifier studied above blocks the negative half cycles of the input, allowing the filter capacitor to be discharged by the load for almost the entire period. The circuit therefore suffers from a large ripple in the presence of a heavy load (a high current).

It is possible to reduce the ripple voltage by a factor of two through a simple modification. Illustrated in Figure 1, the idea is to pass both positive and negative half cycles to the output, but with the negative half cycles inverted (i.e., multiplied by −1). We first implement a circuit that performs this function [called a "full-wave rectifier" (FWR)]

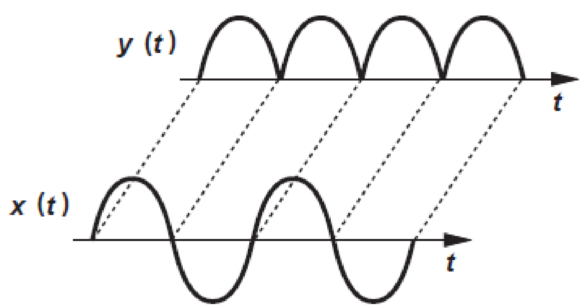


Figure 1 Input and output waveforms

- <u>Bridge full wave circuit:</u>

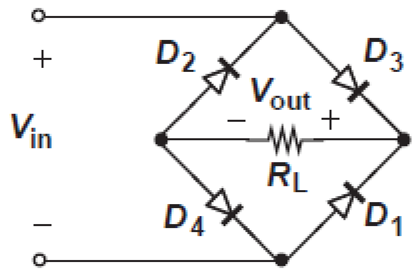


Figure 2 simple diagram full wave circuit

- <u>How the circuit work:</u>

The circuit has four diodes connected to form a bridge. The ac input voltage is applied to the diagonally opposite ends of the bridge. The load resistance is connected between the other two ends of the bridge.

If Vin<0, D2 and D1 are on and D3 and D4 are off, reducing the circuit to that shown in Figure3 and yielding Vout = −Vin.

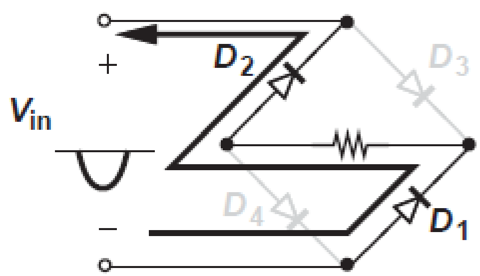


Figure3: Full wave negative side

On the other hand, if Vin > 0, D3 and D4 are on and D1 and D2 are off the bridge is simplified as shown in Figure4, and Vout = Vin.

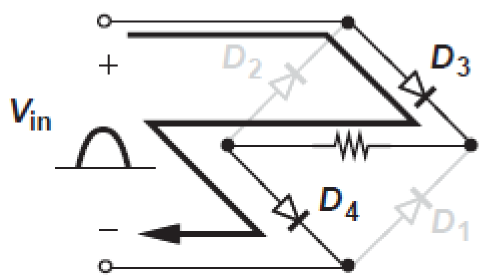


Figure 4: Full wave positive side

- The basic parameters of the full wave rectifier circuit are:
  - <u>For the input:</u>
    - Average Value:

$$V_{dc_{avr}} = \frac{1}{T}\int_0^T v(t)dt = \frac{1}{2\pi}\int_0^{2\pi} V_m \sin\theta \, d\theta$$

$$= \frac{V_m}{2\pi}[-\cos\theta] = \frac{-V_m}{2\pi}[\cos 2\pi - \cos 0]$$

$$= \frac{-V_m}{2\pi}[1-1] = \mathbf{0}$$

    - RMS Value (effective value):

$$V_{rms} = \sqrt{\frac{1}{T}\int_0^t v(t)^2\, dt}$$

$$= \sqrt{\frac{1}{2\pi}\int_0^{2\pi} (V_m \sin\theta)^2 \, d\theta} = \sqrt{\frac{V_m{}^2}{2\pi}\int_0^{2\pi} \sin^2\theta \, d\theta}$$

$$= \sqrt{\frac{V_m{}^2}{2\pi}\int_0^{2\pi} \frac{1}{2}(1-\cos 2\theta)\, d\theta} = \sqrt{\frac{V_m{}^2}{2\pi}\left[\frac{1}{2}\left(\theta - \frac{\sin 2\theta}{2}\right)\right]}$$

$$= \sqrt{\frac{V_m{}^2}{2\pi}\times\frac{1}{2}\left[\left(2\pi - \frac{\sin 4\pi}{2}\right) - \left(0 - \frac{\sin 0}{2}\right)\right]} = \sqrt{\frac{V_m{}^2}{2\pi}[\pi]}$$

$$V_{rms} = \frac{V_m}{\sqrt{2}}$$

  - <u>For the output:</u>
    - Average value:

$$V_{dc_{avr}} = \frac{1}{T}\int_0^T v(t)dt = \frac{1}{\pi}\int_0^{\pi} V_m \sin\theta \, d\theta$$

$$= \frac{V_m}{\pi}[-\cos\theta] = \frac{-V_m}{\pi}[\cos\pi - \cos 0]$$

$$= \frac{-V_m}{\pi}[-1-1] = \frac{\mathbf{2V_m}}{\boldsymbol{\pi}}$$

- RMS Value (effective value):

$$V_{rms} = \sqrt{\frac{1}{T}\int_0^t v(t)^2\,dt}$$

$$= \sqrt{\frac{1}{\pi}\int_0^{\pi} (V_m \sin\theta)^2\, d\,\theta} = \sqrt{\frac{V_m{}^2}{\pi}\int_0^{\pi} \sin^2\theta\, d\theta}$$

$$= \sqrt{\frac{V_m{}^2}{\pi}\int_0^{\pi} \frac{1}{2}(1-\cos 2\theta)\, d\theta} = \sqrt{\frac{V_m{}^2}{\pi}\times\frac{1}{2}\left[\theta - \frac{\sin 2\theta}{2}\right]}$$

$$= \sqrt{\frac{V_m{}^2}{\pi}\times\frac{1}{2}\left[\left(\pi - \frac{\sin 2\pi}{2}\right) - \left(0 - \frac{\sin 0}{2}\right)\right]} = \sqrt{\frac{V_m{}^2}{\pi}\left[\frac{\pi}{2}\right]}$$

$$V_{rms} = \frac{V_m}{\sqrt{2}}$$

Table1 Basic parameter of the FWR

FWR	V(avg)	V(rms)
Input	0	$\frac{\mathrm{Vm}}{\sqrt{2}}$
Output	$\frac{2Vm}{\Pi}$	$\frac{\mathrm{Vm}}{\sqrt{2}}$

➢ Full wave rectifier using a capacitor in shunt as a filter:

We saw in the previous section that the single-phase half-wave rectifier produces an output wave every half cycle and that it was not practical to use this type of circuit to produce a steady DC supply. The full-wave bridge rectifier however, gives us a greater mean DC value with less superimposed ripple while the output waveform is twice that of the frequency of the input supply frequency. We can improve the average DC output of the rectifier while at the same time reducing the AC variation of the rectified output by using smoothing capacitors to filter the output waveform. Smoothing or reservoir capacitors connected in parallel with the load across the output of the full wave bridge rectifier circuit increases the average DC output level even higher as the capacitor acts like a storage device as shown in Figure 5.

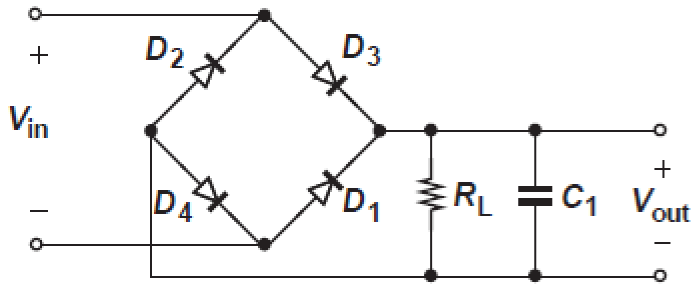


Figure 5 Full wave with capacitor

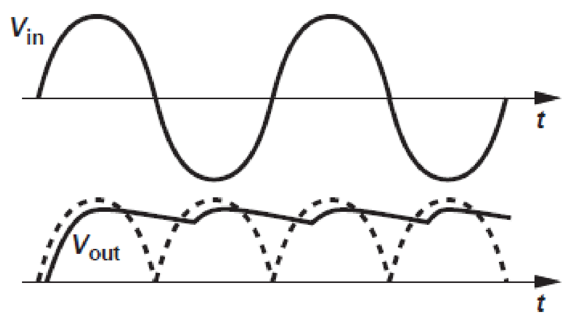


Figure 6 Input and output

❖ Experiment Procedure:

➢ Part 1: FWR with resistive load

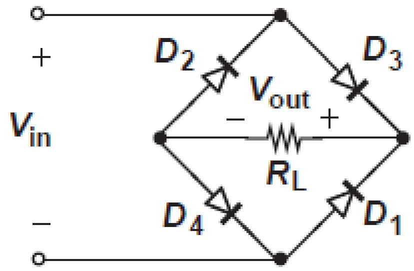


Figure 7 practical circuit full wave

1. Connect the circuit as shown in Figure 7.
2. Set function generator to a sine wave with 20Vp-p and 1 kHz.
3. Put resistor equal 1kΩ.
4. Measure the input and output voltage using oscilloscope.
5. Measure the V_{DC} ,and V_{rms} for output voltage in Table 2.
6. Simulate the circuit using LTspice program.
7. Repeat the previous step when
   a. F=1kHz
      i. R=4.7KΩ
      ii. R=10kΩ
   b. R=1kΩ
      i. F=10kHz
      ii. F=100kHz

Table 2 average, and effective output voltage

R(KΩ)	V_{DC}(V)	$V_{rms}(V)$
1		
4.7		
10		

Table 3 average, and effective output voltage

f(KHz)	V_{DC}(V)	$V_{rms}(V)$
1		
10		
100		

➢ Part 2: FWR with capacitor filter

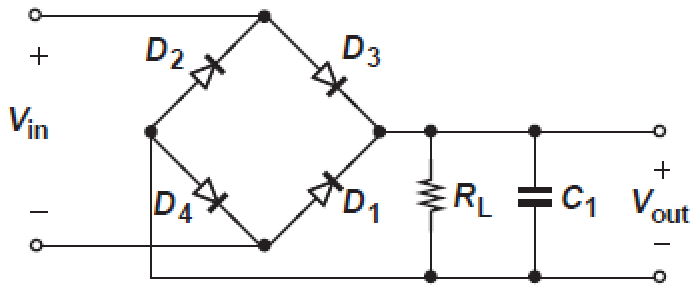


Figure 8 Full wave with capacitor load

1. Connect the circuit as shown in Figure8.
2. Set function generator to a sine wave with 20Vp-p and 1 kHz.
3. Put R_L=1kΩ and C_1= 1µF.
4. Measure the input and output voltage using oscilloscope.
5. Simulate the same circuit using LTspice program.

Experiment 5
Clipper and Clamper Circuits

❖ Objectives

- Design a clamping circuit for the given output.
- Design a clipping circuit for the given values.

❖ Equipment:

- Diode (1N4007).
- Resistors.
- Capacitors.
- Function generator.
- Oscilloscope.

❖ Theory

Clamper Circuits:

A clamper is one, which provides a DC shift to the input signal. The DC shift can be positive or negative. The clamper with positive DC shift is called positive clamper and clamper with negative shift is called negative clamper. Consider a clamper circuit shown below.

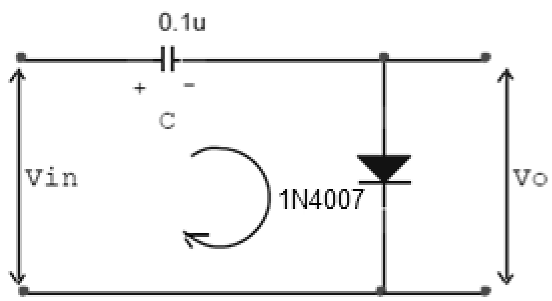


Figure 1 Clamper Circuit

In the positive half cycle as the diode is forward biased the capacitor charges to the value (Vin –VD) with the polarity as shown in the Figure 1. In the negative half cycle the diode is reverse biased. Hence the output is Vo = Vin -VC.

Initially let us assume that the capacitor has charged to (Vin –VD), Note: VD = 0.5V. i.e. (5 – 0.5) = 4.5V. Then in the positive half cycle diode is forward biased and applying KVL to the loop,

$$V_{in} - V_c - V_o = 0 \rightarrow V_o = V_{in} - V_c$$

When,

Vin = 0 Vo= 0 - 4.5 = - 4.5V

Vin = 5V Vo= 5 – 4.5 = 0.5V

In the negative half cycle,

When,

Vin = -5V Vo = -5 – 4.5 = -9.5V

The output shifts between 0.5V and – 9.5V. Here the output has shifted down by 4.5V.

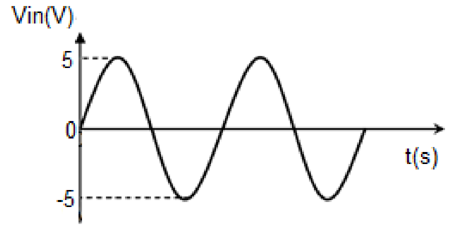


Figure 2 Input Signal

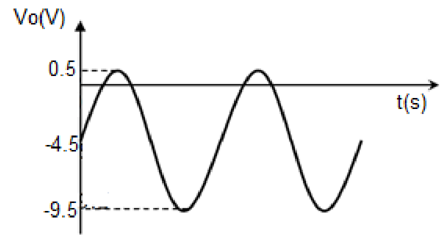


Figure 3 Output Signal

Clipper Circuits:

The process by which the shape of a signal is changed by passing the signal through a network consisting of linear elements is called linear wave shaping. Most commonly used wave shaping circuit is clipper. Clipping circuits are those, which cut off the unwanted portion of the waveform or signal without distorting the remaining part of the signal. There are two types of clippers namely parallel and series. A series clipper is one in which the diode is connected in series with the load and a parallel clipper is one in which the diode is connected in parallel with the load.

- **CIRCUIT DIAGRAM AND DESIGN:**
  Consider the circuit in Figure 4, Given VB = 3V, and diode is ideal VD=0

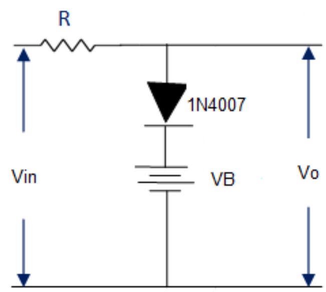


Figure 4 Clipper Circuit

In the positive half cycle,

(i) When |Vin| > |VD + VB|, D is forward biased

Applying KVL, we get

Vin = VR+Vo

Vo = VD+VB

Vo= 0+3=3v

(ii) When |Vin| < |VD + VB|, D is reverse biased

Vo= Vin

In the negative half cycle, D is reverse biased

Vo = Vin

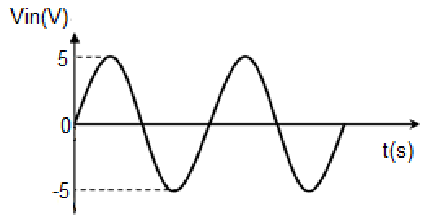


Figure 5 Input Signal

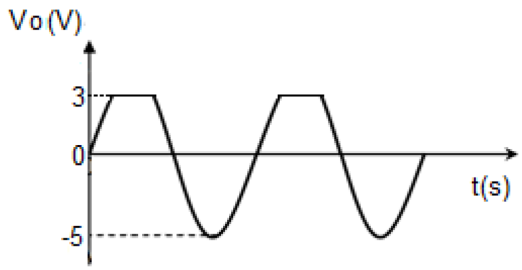


Figure 6 Output Signal

❖ Experiment Procedure:

➢ Part 1: Clamper Circuit

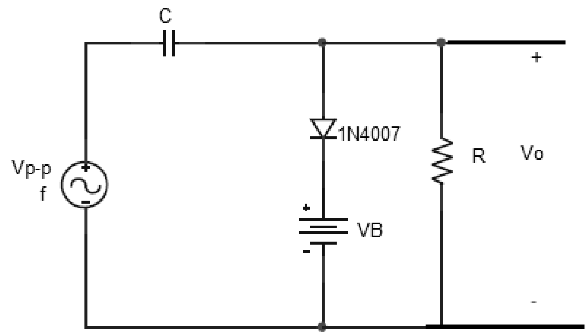


Figure 7 Practical Clamper Circuit

1. Connect the circuit as shown in Figure7.
2. Set function generator to a sine wave with 20Vp-p and 1 kHz.
3. Put R=100kΩ, VB=1Vand C= 1µF.
4. Measure the input and output voltage using oscilloscope.
5. Measure the maximum and minimum values for output voltage in Table 1.
6. Simulate the circuit using LTspice program.
7. Repeat the previous step when
   a- VB = 3V
      i. R=1KΩ
      ii. R=1MΩ
   b- R = 100k
      i. VB = 3V
      ii. VB = 5V
      iii. VB = 7V

Table 1 maximum and minmium values for output voltage

VB(V)	Max Value of Vo(V)	Min Value of Vo(V)
1		
3		
5		
7		

➢ Part 2: Clipper Circuit

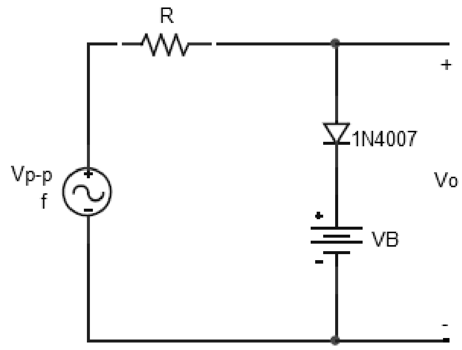


Figure 8 Practical Clipper Circuit

1. Connect the circuit as shown in Figure 8.
2. Set function generator to a sine wave with 20Vp-p, 1 kHz.
3. Put R=1kΩ, VB=1V
4. Measure the input and output voltage using oscilloscope.
5. Measure the maximum and minimum values for output voltage in Table 2.
6. Simulate the same circuit using LTspice program.
7. Repeat the previous step when
   i. VB = 3V
   ii. VB = 5V
   iii. VB = 7V

Table 2 maximum and minmium values for output voltage

VB(V)	Max Value of Vo(V)	Min Value of Vo(V)
1		
3		
5		
7		

Experiment 6
Characteristics of BJT Transistor

❖ Objectives

- To study the input and output characteristics of an NPN transistor in Common Emitter mode and determine transistor parameters.
- To plot the Characteristics of a BJT in Common Emitter Configuration.

❖ Equipment:

- NPN Transistor (2N2222A).
- Resistors.
- DC Power Supply.
- Multi-Meter.

❖ Theory

A BJT is called as Bipolar Junction Transistor and it is a three terminal active device which has emitter, base and collector as its terminals. It is called as a bipolar device because the flow of current through it is due to two types of carriers i.e., majority and minority carriers.

A Bipolar Junction Transistor (BJT) has three terminals connected to three doped semiconductor regions. In an NPN transistor, a thin and lightly doped P-type base is sandwiched between a heavily doped N-type emitter and another N-type collector; while in a PNP transistor, a thin and lightly doped N-type base is sandwiched between a heavily doped P-type emitter and another P-type collector.

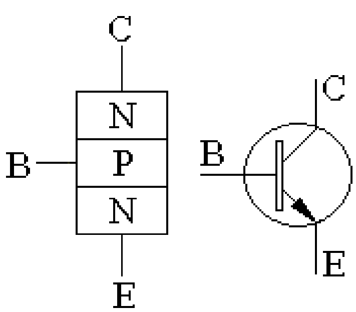


Figure 1 NPN Transistor

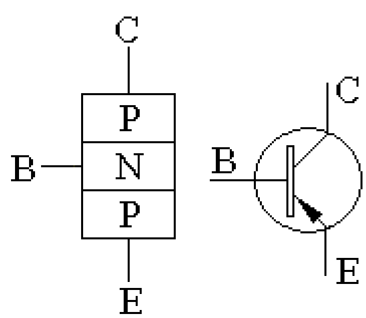


Figure 2 PNP Transistor

A transistor can be in any of the three configurations Common base (CB), common emitter (CE) and Common Collector (CC).

The relation between, β, α of CB, CE, CC are

$$\alpha = \frac{\beta}{1+\beta} \quad \beta = \frac{\alpha}{1-\alpha}$$

In CE configuration base will be input node and collector will be the output node .Here emitter of the transistor is common to both input and output and hence the name common emitter configuration.

The collector current is given as

$$I_C = \beta I_B$$

A transistor in CE configuration is used widely as an amplifier. While plotting the characteristics of a transistor the input voltage and output current are expressed as a function of input current and output voltage.

Transistor characteristics are of two types:

1. Input characteristics are obtained between the input current and input voltage at constant output voltage. It is plotted between VBE and IB at constant VCE in CE configuration.

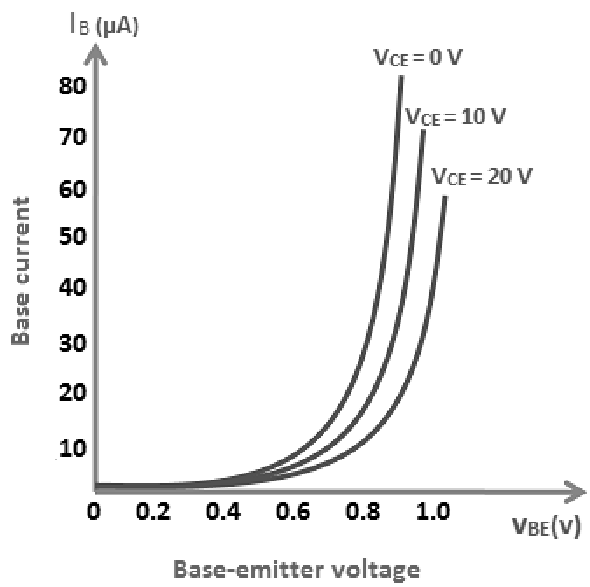


Figure 3 Input Characteristic

2. Output characteristics are obtained between the output voltage and output current at constant input current. It is plotted between VCE and IC at constant IB in CE configuration.

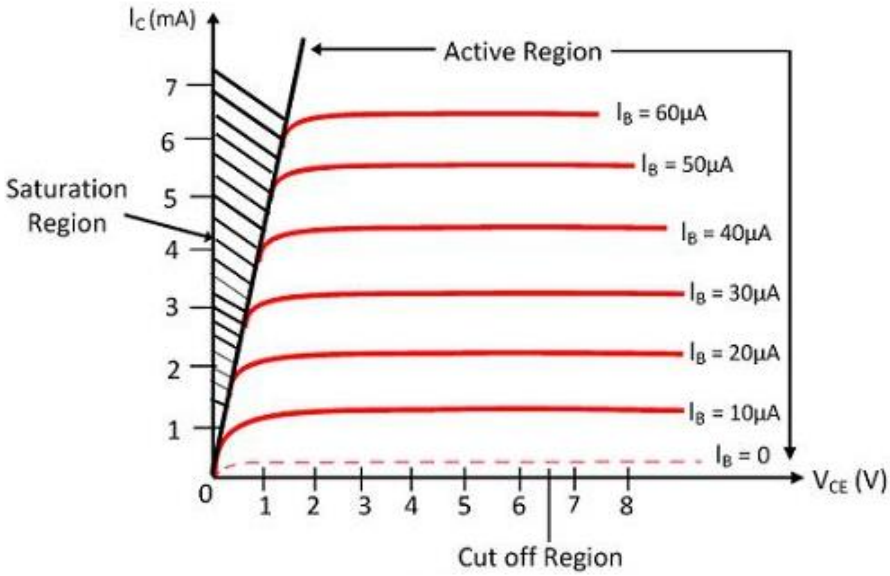


Figure 4 Output Characteristic

The different regions of operation of the BJT are:

- RB: Reverse Bias.
- FB: Forward Bias.

Table 1 Regions of operation

Emitter Junction	Collector Junction	Region	Application
RB	RB	CUTT OFF	OFF SWITCH
FB	FB	SATURATION	ON SWITCH
FB	RB	ACTIVE	AMPLIFIER
RB	FB	REVERSE ACTIVE	ATTENUATOR

❖ Experiment Procedure:

➢ Part 1: Input Characteristic

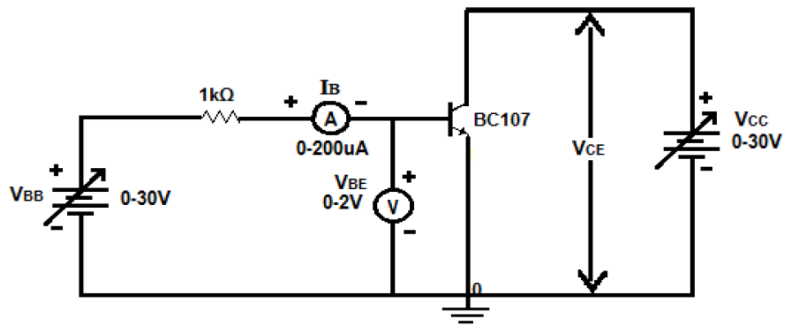


Figure5 Practical input characteristic

1. Connect the circuit as shown in Figure5. Adjust all the knobs of the power supply to their minimum positions before switching the supply on.
2. Adjust the V_{CE} to 0 V by adjusting the supply V_{CC}.
3. Vary the supply voltage V_{BB} so that V_{BE} varies in steps of 0.1 V. In each step note the value of base current I_B in Table 2.

Table 2: Input Characteristic

V_{CE} = 0		
VBB(V)	VBE(V)	IB (µA)
0		
0.1		
0.2		
0.3		
0.4		
0.5		
0.6		
0.7		
0.8		
0.9		
1		
1.1		
1.2		
1.3		
1.4		
1.5		

4. Adjust VCE to 1, 2V and repeat step-3 for each value of VCE in Table 3.

Table 3

	V_{CE} = 1V		V_{CE} = 2V	
V_{BB}(V)	V_{BE}(V)	I_B (μA)	V_{BE}(V)	I_B (μA)
0				
0.1				
0.2				
0.3				
0.4				
0.5				
0.6				
0.7				
0.8				
0.9				
1				
1.1				
1.2				
1.3				
1.4				
1.5				

5. Plot a graph between V_{BE} and I_B for different values of V_{CE}. These curves are called input characteristic.

➢ Part 2: Output Characteristic

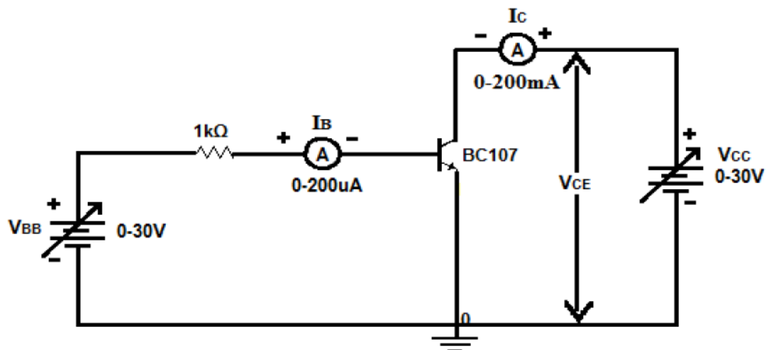


Figure 6 Practical Output Characteristic

1. Connect the circuit as shown in Figure 6. Adjust all the knobs of the power supply to their minimum positions before switching the supply on.
2. Adjust the base current I_B to 20 μA by adjusting the supply V_{BB}.
3. Vary the supply voltage V_{CC} so that the voltage V_{CE} varies in steps of 0.2 V from 0 to 2 V and then in steps of 1 V from 2 to 10 V. In each step the base current should be adjusted to the present value and the collector current I_C should be recorded in Table 4.

Table 4

IB = 20 µA		
VCC(V)	V_{CE}(V)	I_C (mA)
0		
0.2		
0.4		
0.6		
0.8		
1.0		
1.2		
1.4		
1.6		
1.8		
2.0		
3		
4		
5		
6		
7		
8		
9		
10		

4. Adjust the base current at 40, 60 µA and repeat step-3 for each value of IB in Table 5.

Table 5

	IB = 40 µA		IB = 60 µA	
VCC (v)	V_{CE}(V)	I_C (mA)	V_{CE}(V)	I_C (mA)
0				
0.2				
0.4				
0.6				
0.8				
1.0				
1.2				
1.4				
1.6				
1.8				
2.0				
3				
4				
5				
6				
7				
8				
9				
10				

5. Plot a graph between the output voltage VCE and output current IC for different values of the input current IB. These curves are called the output characteristics.

Experiment 7
Common Emitter Amplifier

❖ Objectives

- To construct a common emitter amplifier circuit and make DC and AC analyze of the system.
- To find all parameters of equivalent model.
- To be familiar how can be simulate common emitter amplifier circuit using LTspice simulation.

❖ Equipment:

- Transistor (2N2222A).
- Resistors.
- Capacitors.
- Power Supply.
- Multi-meter.
- Function generator.
- Oscilloscope.

❖ Theory

An amplifier is an electronic circuit that can increase the strength of a weak input signal without distorting its shape. The common emitter configuration is widely used as a basic amplifier as it has both voltage and current amplification with 180 phase shift.

The factor by which the input signal gets multiplied after passing through the amplifier circuit is called the gain of the amplifier. It is given by the ratio of the output and input signals.

$$Gain = \frac{Output\ Signal}{Input\ Signal}$$

The aim of any small signal amplifier is to amplify all of the input signal with the minimum amount of distortion possible to the output signal, in other words, the output signal must be an exact reproduction of the input signal but only bigger (amplified).

To obtain low distortion when used as an amplifier the operating quiescent point needs to be correctly selected. This is in fact the DC operating point of the amplifier and its position may be established at any point along the load line by a suitable biasing arrangement.

The best possible position for this Q-point is as close to the center position of the load line as reasonably possible, thereby producing a Class A type amplifier operation, i.e. Vce = 1/2Vcc. Consider the Common Emitter Amplifier circuit shown below.

- **The Common Emitter Amplifier Circuit:**

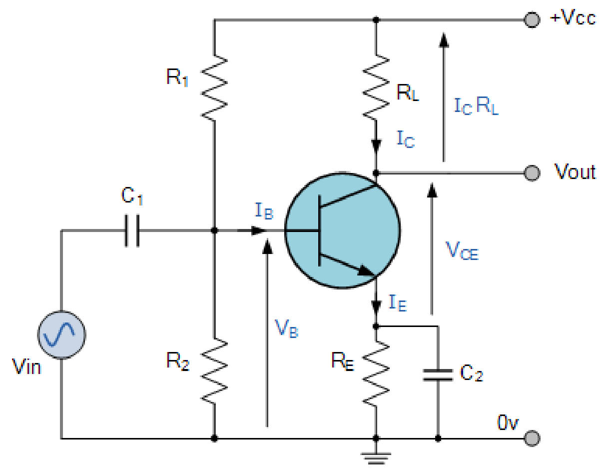


Figure 1 Common Emitter Amplifier Circuit

The single stage common emitter amplifier circuit shown above uses what is commonly called "Voltage Divider Biasing". This type of biasing arrangement uses two resistors as a potential divider network across the supply with their center point supplying the required Base bias voltage to the transistor. Voltage divider biasing is commonly used in the design of bipolar transistor amplifier circuits.

- Amplifier Coupling Capacitors:

In Common Emitter Amplifier circuits, capacitors C1 and C2 are used as Coupling Capacitors to separate the AC signals from the DC biasing voltage. This ensures that the bias condition set up for the circuit to operate correctly is not affected by any additional amplifier stages, as the capacitors will only pass AC signals and block any DC component. The output AC signal is then superimposed on the biasing of the following stages. Also a bypass capacitor, CE is included in the Emitter leg circuit.

This capacitor is effectively an open circuit component for DC biasing conditions, which means that the biasing currents and voltages are not affected by the addition of the capacitor maintaining a good Q-point stability.

However, this parallel connected bypass capacitor effectively becomes a short circuit to the Emitter resistor at high frequency signals due to its reactance. Thus, only RL plus a very small internal resistance acts as the transistors load increasing voltage gain to its maximum.

❖ Experiment Procedure:

➢ Part 1: DC Analysis

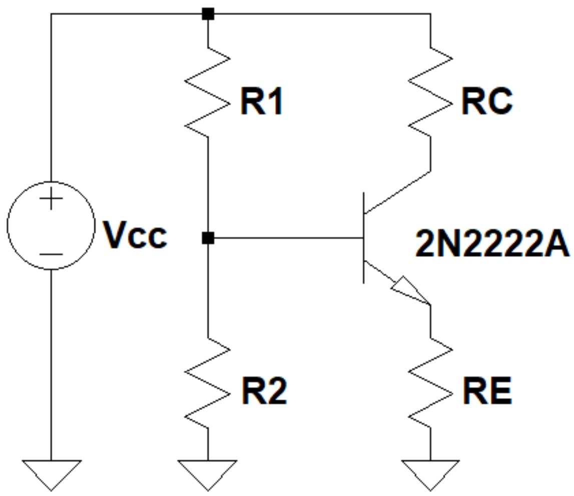


Figure 2 DC circuit for common emitter amplifier

1. Connect the circuit as shown in Figure 2.
2. Set Vcc=12v, R1=10 kΩ, R2=2.2 kΩ, Rc=1 kΩ, and RE=220 Ω.
3. Measure the voltages and currents in the circuits in Table 1.
4. Simulate the circuit using LTspice program.
5. Repeat the previous step when

   i. R_C=4.7KΩ
   ii. R_C=100kΩ

Table 1 DC Operating Values

R(KΩ)	V_B(V)	$V_C(V)$	$V_E(V)$	$V_{CE}(V)$	$I_B(\mu A)$	$I_C(mA)$	$I_E(mA)$
1							
4.7							
100							

➢ Part 2: AC Analysis

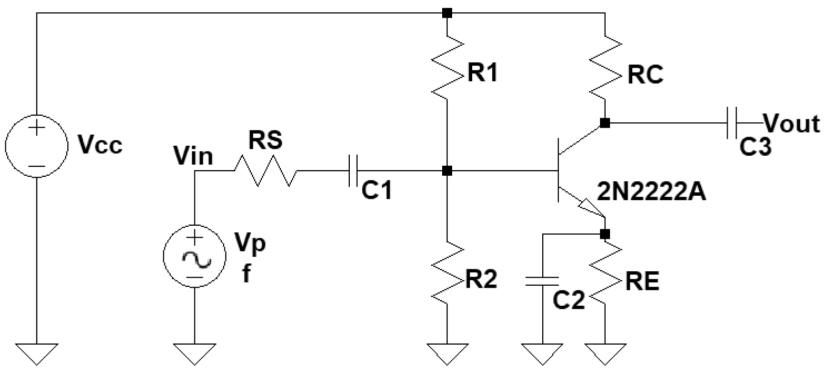


Figure 3 AC circuit for common emitter amplifier

1. Connect the circuit as shown in Figure3.
2. Set function generator to a sine wave with 500mVp and 1 kHz.
3. Set Vcc=12v, RS=1 kΩ, R1=10 kΩ, R2=2.2 kΩ, Rc=1 kΩ, RE=220 Ω, C_1=100µF, C_2= 100µF , and C_3= 22µF.
4. Measure the input and output voltage using oscilloscope.
5. Simulate the same circuit using LTspice program.
6. Repeat the previous step when

   i. R_C=4.7KΩ
   ii. R_C=100kΩ

Experiment 8
Common Base Amplifier

❖ Objectives

- To construct a common base amplifier circuit and make DC and AC analyze of the system.
- To find all parameters of equivalent model.
- To be familiar how can be simulate common base amplifier circuit using LTspice simulation.

❖ Equipment:

- Transistor (2N2222A).
- Resistors.
- Capacitors.
- Power Supply.
- Multi-meter.
- Function generator.
- Oscilloscope.

❖ Theory

The Common Base Amplifier is another type of bipolar junction transistor, (BJT) configuration where the base terminal of the transistor is a common terminal to both the input and output signals, hence its name common base (CB). The common base configuration is less common as an amplifier than compared to the more popular common emitter, (CE) or common collector, (CC) configurations but is still used due to its unique input/output characteristics.

For the common base configuration to operate as an amplifier, the input signal is applied to the emitter terminal and the output is taken from the collector terminal. Thus, the emitter current is also the input current, and the collector current is also the output current, but as the transistor is a three layer, two pn-junction devices, it must be correctly biased for it to work as a common base amplifier. That is the base-emitter junction is forward-biased.

- ### Characteristics of Common Base Amplifier Circuit:

The following are the characteristics of the Common Base amplifier circuit.

- High voltage gain.
- Low current gain.
- Low power gain.
- Input and output phase relation is 0.
- It has low input impedance.
- It has high output impedance.

- **Common base transistor amplifier circuits:**

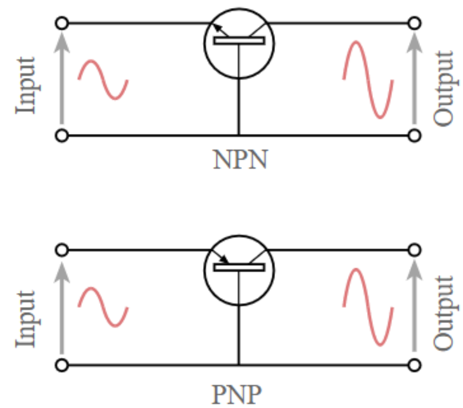


Figure 1 Common Base Circuit Configuration

For both NPN and PNP circuits, it can be seen that for the common base amplifier circuit, the input is applied to the emitter, and the output is taken from the collector. The common terminal for both circuits is the base. The base is grounded for the signal and for this reason the circuit may sometimes be called a grounded base circuit.

The below diagram shows how common base amplifier circuit is implemented.

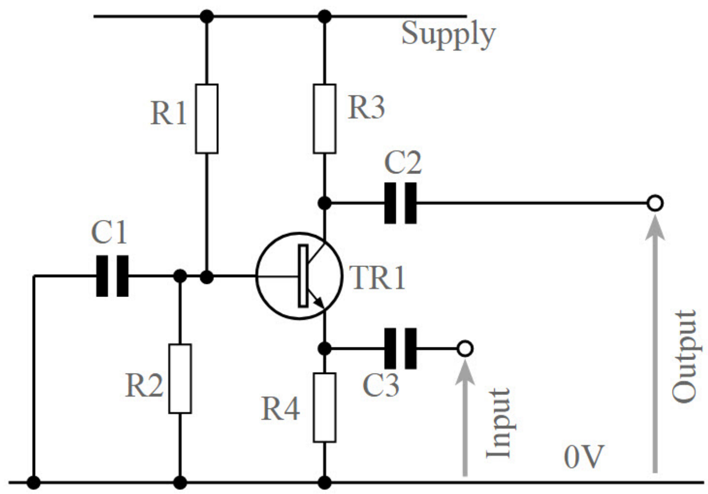


Figure 2 Common Base Circuit

The biasing constraints are same, but the applications of the signals are different. In this circuit, care has to be taken such that correct impedance match is provided to the input signal.

❖ Experiment Procedure:

➢ Part 1: DC Analysis

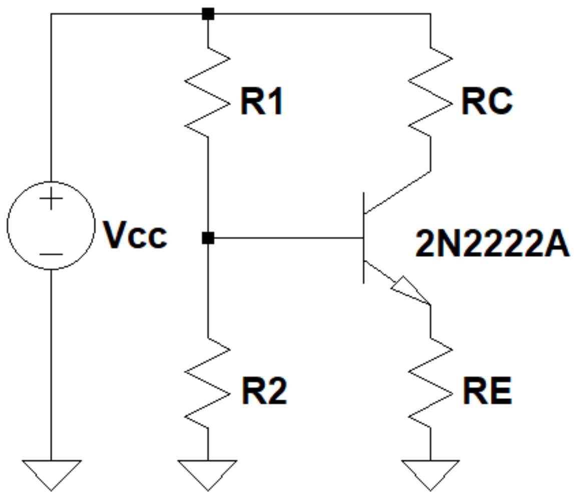


Figure 3 DC circuit for common base amplifier

1. Connect the circuit as shown in Figure 3.
2. Set Vcc=10v, R1=10 kΩ, R2=2.2 kΩ, Rc=1 kΩ, and RE=220 Ω.
3. Measure the voltages and currents in the circuits in Table 1.
4. Simulate the circuit using LTspice program.
5. Repeat the previous step when

   i. R_C=4.7KΩ
   ii. R_C=100kΩ

Table 1 DC Operating values

R(KΩ)	V_B(V)	$V_C(V)$	$V_E(V)$	$V_{CE}(V)$	$I_B(\mu A)$	$I_C(mA)$	$I_E(mA)$
1							
4.7							
100							

➢ Part 2: AC Analysis

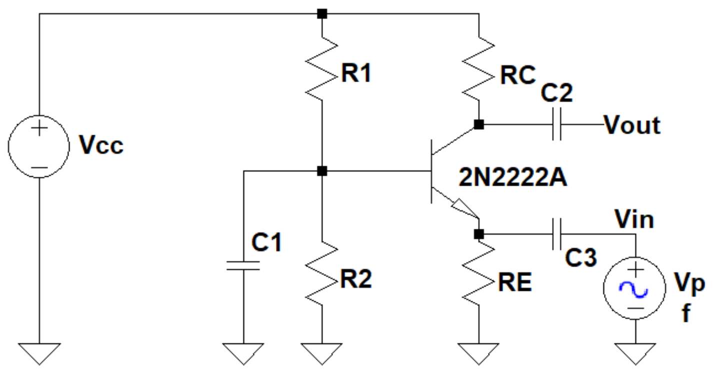


Figure 4 AC circuit for common base amplifier

1. Connect the circuit as shown in Figure4.
2. Set function generator to a sine wave with 500mVp and 1 kHz.
3. Set Vcc=10v, R1=10 kΩ, R2=2.2 kΩ, Rc=1 kΩ, RE=220 Ω, C_1=100µF, C_2= 100µF, and C_3= 22µF.
4. Measure the input and output voltage using oscilloscope.
5. Simulate the same circuit using LTspice program.
6. Repeat the previous step when

   i. R_C=4.7KΩ
   ii. R_C=100kΩ

Experiment 9
Common Collector Amplifier

❖ Objectives

- To construct a common collector amplifier circuit and make DC and AC analyze of the system.
- To find all parameters of equivalent model.
- To be familiar how can be simulate common collector amplifier circuit using LTspice simulation.

❖ Equipment:

- Transistor (2N2222A).
- Resistors.
- Capacitors.
- Power Supply.
- Multi-meter.
- Function generator.
- Oscilloscope.

❖ Theory

The Common Collector Amplifier is another type of bipolar junction transistor, (BJT) configuration where the input signal is applied to the base terminal and the output signal taken from the emitter terminal. Thus, the collector terminal is common to both the input and output circuits. This type of configuration is called Common Collector, (CC) because the collector terminal is effectively “grounded” or “earthed” through the power supply.
The emitter follower or common collector circuit configuration provides a high input impedance and a low output impedance.
This means that the emitter follower circuit provides an ideal buffer stage, and as a result it is used in many circuits where there is a need not to load a circuit like an oscillator or other circuit, but provide a lower impedance to the following stages.
The emitter follower is easy to design and implement, requiring just a few components.

- ### Characteristics of Common Collector Amplifier Circuit:

The following are the characteristics of the Common Base amplifier circuit.

- No voltage gain.
- High current gain.
- Medium power gain.
- Input and output phase relation is 0.
- It has high input impedance.
- It has low output impedance.

- **Emitter follower / common collector transistor amplifier basics:**

The emitter follower transistor amplifier has a very straightforward circuit. The base is connected to the previous stage, and often this may be directly connected as this can save on additional bias resistors which lower the input impedance and hence increase the loading to the previous stage.

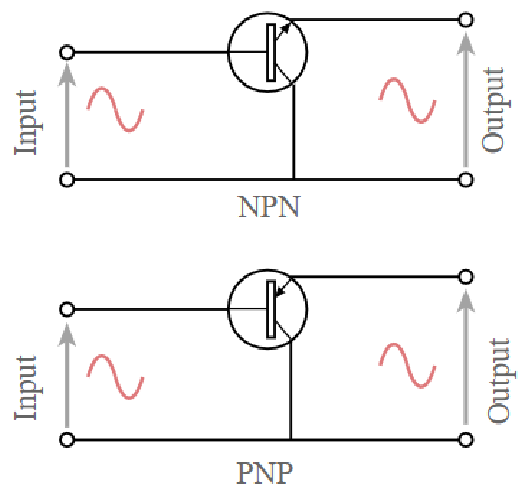


Figure 1 Common Collector Circuit Configuration

The emitter follower gains its name from the fact that the emitter follows the voltage on the base. It is actually slightly less than the voltage on the base by the amount of the base emitter diode voltage drop. This also means that the input and output are exactly in phase and not shifted by 180° as in the case of the common emitter amplifier.

The below diagram shows how common Collector amplifier circuit is implemented.

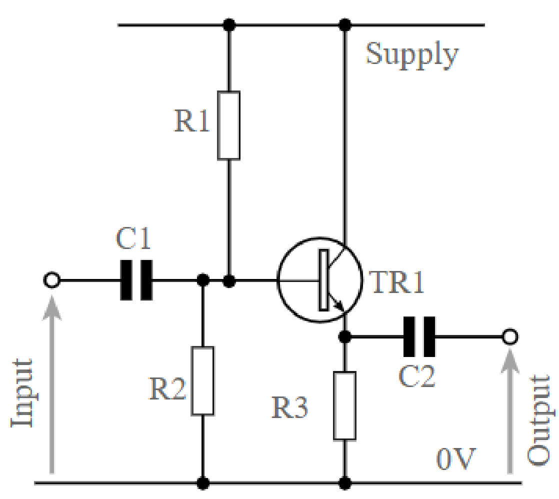


Figure 2 Emitter Follower Circuit

❖ Experiment Procedure:

➢ Part 1: DC Analysis

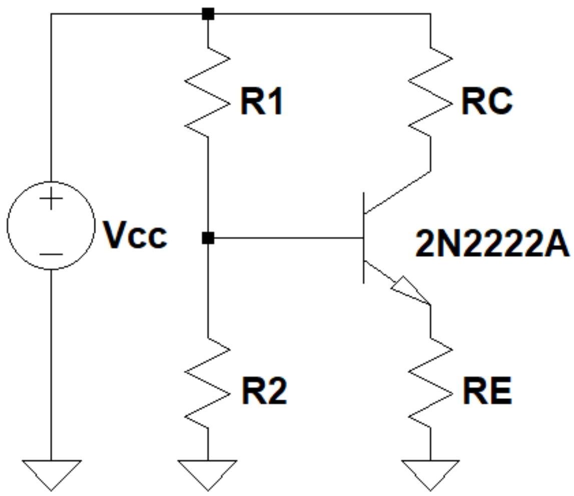


Figure 3 DC circuit for common collector amplifier

1. Connect the circuit as shown in Figure 3.
2. Set Vcc=10v, R1=10 kΩ, R2=2.2 kΩ, Rc=1 kΩ, and RE=220 Ω.
3. Measure the voltages and currents in the circuits in Table 1.
4. Simulate the circuit using LTspice program.
5. Repeat the previous step when

   i. R_C=4.7KΩ
   ii. R_C=100kΩ

Table 1 DC Operating Values

R(KΩ)	V_B(V)	$V_C(V)$	$V_E(V)$	$V_{CE}(V)$	$I_B(\mu A)$	$I_C(mA)$	$I_E(mA)$
1							
4.7							
100							

➢ Part 2: AC Analysis

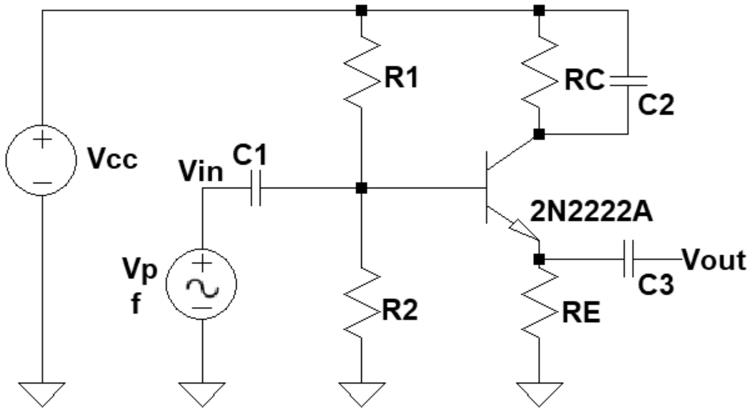


Figure 4 AC circuit for common collector amplifier

1. Connect the circuit as shown in Figure4.
2. Set function generator to a sine wave with 500mVp and 1 kHz.
3. Set Vcc=10v, R1=10 kΩ, R2=2.2 kΩ, Rc=1 kΩ, RE=220 Ω, C_1=100μF, C_2= 22μF, and C_3= 100μF.
4. Measure the input and output voltage using oscilloscope.
5. Simulate the same circuit using LTspice program.
6. Repeat the previous step when

   i. R_C=4.7KΩ
   ii. R_C=100kΩ

Experiment 10
JFET Characteristics

❖ Objectives

- To study JFET transistor and the difference between BJT transistor.
- Plot the transfer and drain characteristics of a JFET
- To be familiar how can be simulate JFET circuit using LTspice simulation.

❖ Equipment:

- JFET Transistor.
- Resistors.
- Power Supply.
- Multi-meter.
- Bread Board.

❖ Theory

A JFET is called as Junction Field effect Transistor. It is a unipolar device because the flow of current through it is due to one type of carriers i.e., majority carriers where as a BJT is a Bi - Polar device, It has 3 terminals Gate, Source and Drain. A JFET can be used in any of the three configurations viz, Common Source, Common Gate and Common Drain.

A FET is a three-terminal device, having the characteristics of high input impedance and less noise, the gate to source junction of the FET always reverse biased. In response to small applied voltage from drain to source, the n-type bar acts as sample resistor, and the drain current increases linearly with V_{DS}. with increase in I_D the ohmic voltage drop between the and the channel region reverse biases the junction and the conducting position of the channel begins to remain constant.

The V_{DS} at this instant is called “pinch of voltage”. If the gate to source voltage (V_{GS}) is applied in the direction to provide additional reverse bias, the pinch off voltage is decreased.

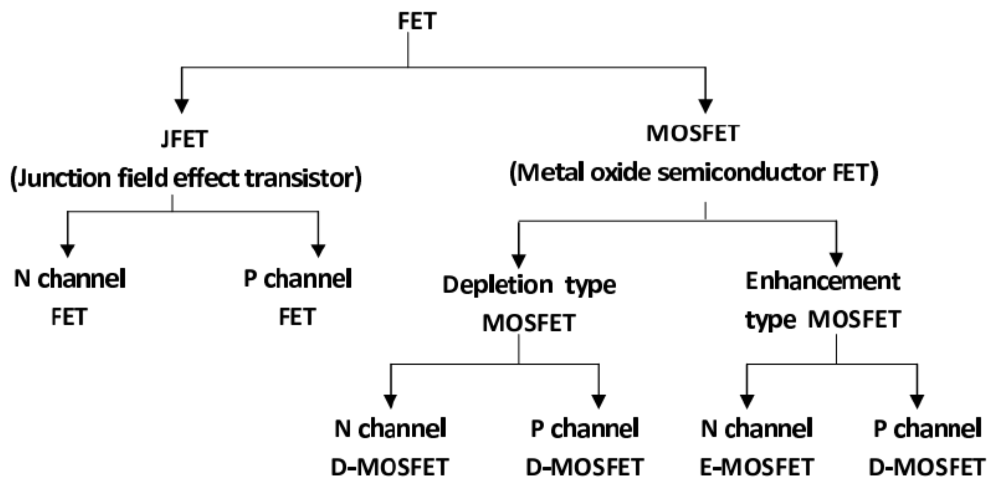
In CS configuration Gate is used as input node and Drain as the output node. A JFET in CS configuration is used widely as an amplifier. A JFET amplifier is preferred over a BJT amplifier when the demand is for smaller gain, high input resistance and low output resistance

The drain current equation and trans-conductance is given as

$$I_D = I_{DSS}\left(1 - \frac{V_{GS}}{V_P}\right)^2 \qquad g_m = \frac{\partial I_D}{\partial V_{GS}} = \frac{2}{|v_p|}\sqrt{I_D\, I_{DSS}}$$

Where I_{DSS} is called as Drain to Source Saturation current and V_p is called as the Pinch off voltage.

The Field Effect Transistor (FET) can be broadly classified into following categories:



Bipolar Junction Transistor (BJT)	Junction Field Effect Transistor (JFET)
Current controlled device (Ic function of IB)	Voltage controlled device (ID function of VGS)
Two types (PNP, NPN)	Two types (N-channel, P-channel)
High sensitivity to change of applied voltage	High Input Impedance
Less temperature stable	More temperature stable
Unipolar, depends on either electron or holes	Bipolar depends on both electron and holes
Need more area	Small ,useful in integrated-circuit (IC) chips
More power consumption	Low power consumption

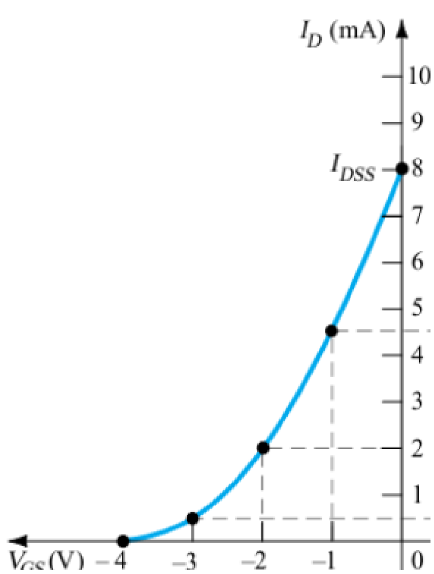


Figure 1 Transfer Characteristics

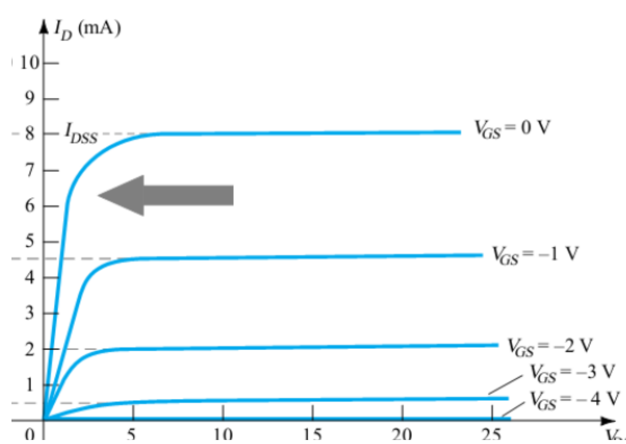


Figure 2 Drain Characteristics

❖ Experiment Procedure:

➢ Part 1: Transfer Characteristics

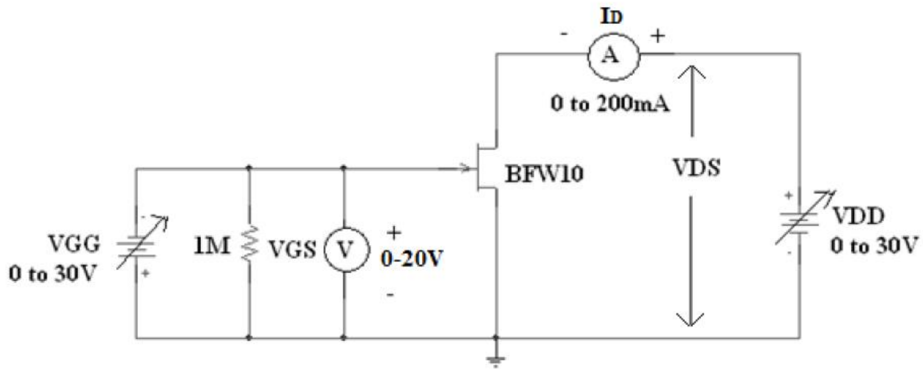


Figure 3 Characteristics of FET

1. Connect the circuit as shown in Figure3. All the knobs of the power supply must be at the minimum position before the supply is switched on.
2. Adjust the output voltage V_{DS} to 4V by adjusting the supply V_{DD}.
3. Vary the supply voltage VGG so that the voltage VGS varies, in each step note the drain current I_D. This should be continued till I_D becomes zero.
4. Repeat above step for V_{DS} = 6V.
5. Plot a graph between the input voltage V_{GS} and output current I_D. This curve is called the transfer characteristics.

Table 1: Transfer Charactaristics table

Transfer Characteristics			
$V_{DS} = 4V$		$V_{DS} = 6V$	
V_{GS}(V)	I_D(mA)	V_{GS}(V)	I_D(mA)

➢ Part 2: Drain Characteristics

1. Connect the circuit as shown in Figure3. All the knobs of the power supply must be at the minimum position before the supply is switched on.
2. Adjust the input voltage V_{GS} to 0 V by adjusting the supply V_{GG}.
3. Vary the supply voltage V_{DD} so that V_{DS} varies in steps of 1 V from 0 to 10 V. In each step note the value of drain current I_D.
4. Adjust V_{GS} to -1 and -2 V and repeat step-3 for each value of V_{GS}.
5. Plot a graph between V_{DS} and I_D for different values of V_{GS}. These curves are called drain characteristics.

Table 2: Drain Characteristics table

Drain Characteristics					
V_{GS} = 0V		V_{GS} = -1V		V_{GS} = -2V	
V_{DS}(V)	I_D(mA)	V_{DS}(V)	I_D(mA)	V_{DS}(V)	I_D(mA)