



**SAPTHAGIRI NPS**  
UNIVERSITY  
UNMATCHED EXCELLENCE, UNLIMITED POTENTIAL

**SCHOOL OF ELECTRONICS AND COMMUNICATION  
ENGINEERING**

#14/5, Chikkasandra, Hesarghatta Main Road, Bengaluru – 560057

**BASIC ELECTRICAL AND  
ELECTRONICS ENGINEERING  
LABORATORY MANUAL**

**Subject Code: 24BTELY104**

**SEMESTER - I**

**Academic Year**

**2024-25**

# **Sapthagiri NPS University, Bangalore - 57**

## **School of Electronics and Communication Engineering**



### **Laboratory Certificate**

This is to certify that Mr. / Ms \_\_\_\_\_  
\_\_\_\_\_  
has satisfactorily completed the course of  
Experiments in Practical \_\_\_\_\_  
prescribed by the Department during the year \_\_\_\_\_

Name of the Candidate: \_\_\_\_\_

USN No.: \_\_\_\_\_ Semester: \_\_\_\_\_

Signature of the staff in-charge:

Date:

Marks	
Max. Marks	Obtained
/	

## Basics of Electrical and Electronics Engineering Laboratory

Expt. No.	Date	TITLE	Duration in Hrs	Max. Marks	Marks Obtained
1.		i. Verification of Ohm's law ii. Study of effect of Open and Short Circuit in Simple Circuits.	2	10	
2.		Verification of KCL, KVL for DC circuits.	2	10	
3.		Measurement of Current , Power and Power factor of incandescent lamp, florescent lamp.& LED Lamp	2	10	
4.		Determination of Phase and Line quality in 3 star & delta.	2	10	
5.		Measurement of three phase power by two wattmeter method.	2	10	
6.		Determine the ripple factor and efficiency of the half wave using & Full wave rectifier using Centre tapped/ Bridge.	2	10	
7.		Determine the input and output characteristics of zener diode.	2	10	
8.		Determine input & output characteristic of a CE/CB/CC transistor configuration.	2	10	
9.		Demonstrate all Basic gates and realise Basic gates using Universal gates.	2	10	
10.		Implement and test the functionality of Half adder and Full adder.	2	10	
11.		Implement and test the functionality of SR, JK, D and T Flip-Flops.	2	10	
***Note: Also Verify the Expt. No 7 to 11 on software platforms like – PSpice 9.1 student version					
<b>TEST</b>					
<b>TOTAL MARKS</b>					

## *Laboratory Safety Information*

---

To work safely, it is important that you understand the prudent practices necessary to minimize the risks and what to do if there is an accident.

---

### **Electrical Shock**

---

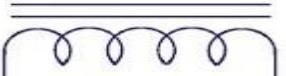
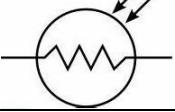
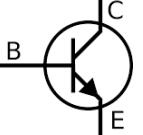
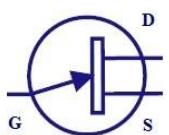
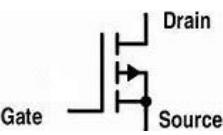
1. Avoid contact with conductors in energized electrical circuits.
2. Electrocution has been reported at dc voltages as low as 42 volts. Just 100 mA of current passing through the chest is usually fatal.
3. Muscle contractions can prevent the person from moving away while being electrocuted.
4. Do not touch someone who is being shocked while still in contact with the electrical conductor or you may also be electrocuted.
5. Make sure your hands are dry.
6. The resistance of dry, unbroken skin is relatively high and thus reduces the risk of shock. Skin that is broken, wet or damp with sweat has a low resistance.
7. When working with an energized circuit, work with only your right hand, keeping your left hand away from all conductive material. This reduces the likelihood of an accident that results in current passing through your heart.
8. Be cautious of rings, watches, and necklaces. Skin beneath a ring or watch is damp, lowering the skin resistance.
9. Shoes covering the feet are much safer than sandals.

## Circuit Trouble Shooting Hints

---

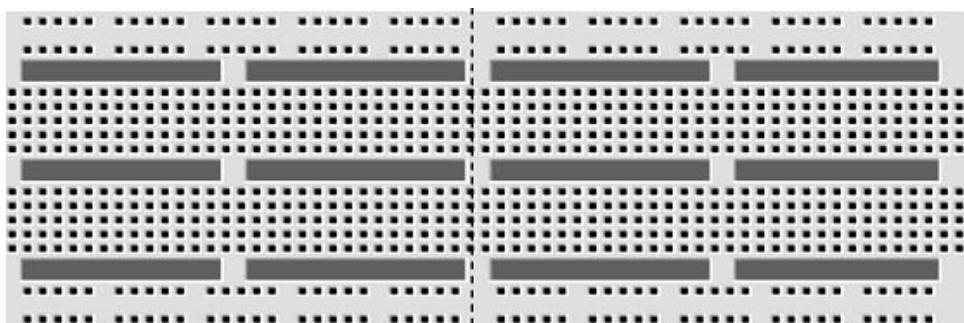
- ✓ Ensure that the power is turned on.
- ✓ Ensure the ground connections are common.
- ✓ Ensure the circuit you built is identical to that in the diagram. (Do a node-by-nodecheck)
- ✓ Ensure that the supply voltages are correct.
- ✓ Ensure you plug in cable to the right terminal in the multimeter to measure the voltage/resistance (upper terminal) or the current (lower terminal).
- ✓ Ensure that the equipment is set up correctly and you are measuring the correct parameter.
- ✓ Ensure the BJT's collector and emitter terminals are in correct orientation.
- ✓ If steps 1 through 5 are correct, then you probably have used a component with the wrong value or one that doesn't work.
- ✓ It is also possible that the equipment does not work (although this is not probable) or the bread-board you are using may have some unwanted paths between nodes.
- ✓ To find your problem you must trace through the voltages in your circuit *node by node* and compare the signal you have to the signal you expect to have.
- ✓ Finally, ask your lab assistant.

Component	Circuit Symbol	Function of Component
Wire		To pass current from one part of a circuit to another.
Wires joined		Wires connected at junctions should be staggered slightly to form two T-junctions, as shown.
Wires not joined		Wires crossing even though they are not connected.
Fuse		The Fuse reacts as safety element to protect circuit against large current and sudden urges of current.
Cell		Cell Supplies electrical energy. The larger terminal is positive (+). A single cell is often called a battery, but strictly a battery is two or more cells joined together.
Battery		Supplies electrical energy. A battery is more than one cell. The larger terminal is positive (+).
AC Supply		This represents AC supply in the circuit.
DC Supply		This represents the DC power supply. It applies DC supply to the circuit.
Ground		It is equivalent to theoretical 0 V and is used as zero potential reference. It is the potential of perfectly conducting earth.
Fixed Resistor		It is a device that opposes the flow of current in a circuit. These two symbols are used to represent fixed resistor.
Rheostat		It is a two terminal variable resistor. They are generally used to control the current in the circuit. Generally used in tuning circuits and power control applications like heaters, ovens etc
Capacitor		Capacitor stores the charge in the form of electrical energy. It can be used in both AC and DC circuits.
Electrolytic Capacitor		Almost all electrolytic capacitors are polarized and hence used in DC circuits. It can also be used as a filter, to block DC signals but pass AC signals.

Component	Circuit Symbol	Function of Component
Iron Core Inductor		A coil of wire which creates a magnetic field when current passes through it.
Transformer		Two coils of wire linked by an iron core. Transformers are used to step up (increase) and step down (decrease) AC voltages. Energy is transferred between the coils by the magnetic field in the core. There is no electrical connection between the coils.
Diode		A device which only allows current to flow in one direction.
LED		A transducer which converts electrical energy to light.
LDR		LDRs or photo-resistors are often used in circuits where it is necessary to detect the presence or the intensity level of light.
Zener Diode		A special diode which is used to maintain a fixed voltage across its terminals.
Transistor or(NPN)		A transistor amplifies current. It can be used with other components to make an amplifier or switching circuit.
Voltmeter		A voltmeter is used to measure voltage. The proper name for voltage is 'potential difference', but most people prefer to say voltage!
Ammeter		An ammeter is used to measure current.
JFET N-Channel		N-channel JFET is made by n-type silicon bars which form two PN junctions at the sides. Majority charge carriers here are electrons.
MOSFET P-Channel		The enhancement MOSFET structure has no channel formed during its construction. Voltage is applied to the gate, so as to develop a channel.

## Bread Board Connection Diagram

### Internal Wire Connection



- ← Connected Horizontally
- ↑ Connected vertically
- ↓ Connected vertically
- ← Connected Horizontally

### External Pin Connection

## Standard Resistor Values and Color Coding

The standard resistor color code table:

Color	Digit 1	Digit 2	Digit 3*	Multiplier	Tolerance	Temp. Coef.	Fail Rate
Black	0	0	0	$\times 10^0$			
B	Digit	Digit	Multiplier	Tolerance			
R							
O							
Y							
Green	5	5	5	$\times 10^3$	$\pm 0.5\% \text{ (D)}$		
Blue	6	6	6	$\times 10^6$	$\pm 0.25\% \text{ (C)}$		
Violet	7	7	7	$\times 10^7$	$\pm 0.1\% \text{ (B)}$		
Gray	8	8	8	$\times 10^8$	$\pm 0.05\% \text{ (A)}$		
White	9	9	9	$\times 10^9$			
Gold				$\times 0.1$	$\pm 5\% \text{ (J)}$		
Silver				$\times 0.01$	$\pm 10\% \text{ (K)}$		
None					$\pm 20\% \text{ (M)}$		

**4-band code**

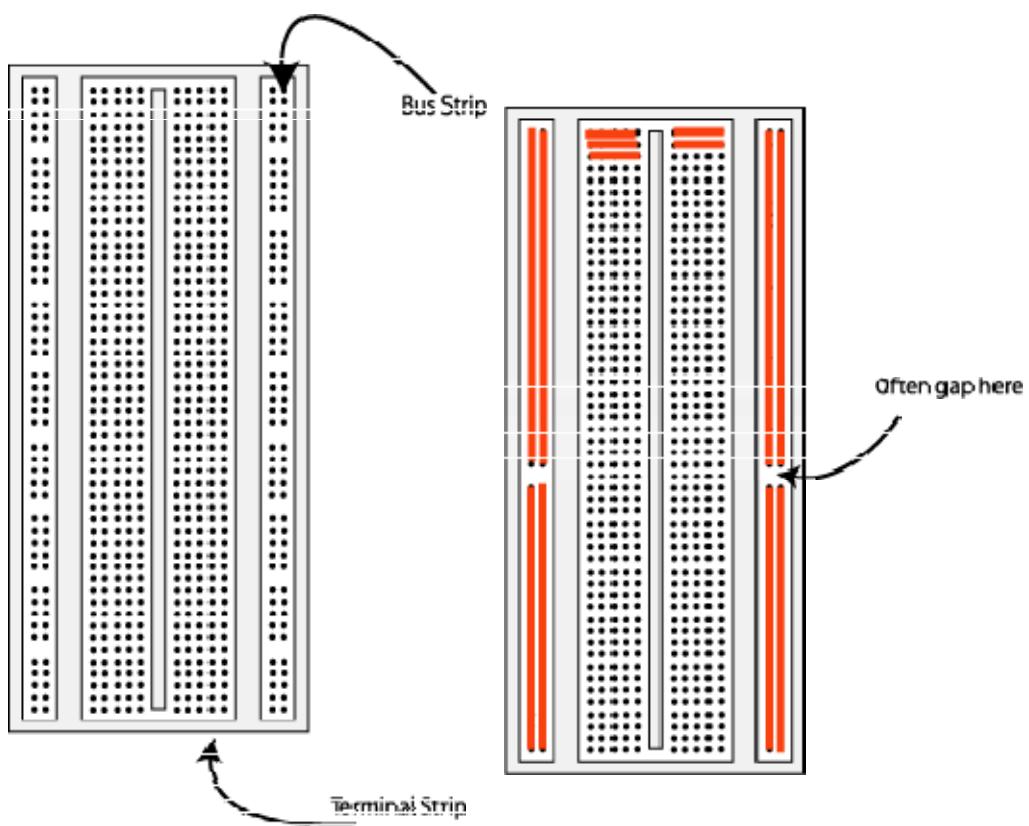
**5-band code**

\* 3rd digit - only for 5-band resistors

## Introduction to the Breadboard

The breadboard consists of two terminal strips and two bus strips ( often broken in the centre). Each bus strip has two rows of contacts. Each of the two rows of contacts is a node. That is, each contact along a row on a bus strip is connected together (inside the breadboard). Bus strips are used primarily for power supply connections, but are also used for any node requiring a large number of connections. Each terminal strip has 60 rows and 5 columns of contacts on each side of the centre gap. Each row of 5 contacts is a node.

You will build your circuits on the terminal strips by inserting the leads of circuit components into the contact receptacles and making connections with 22-26 gauge wire. There are wire cutter/strippers and a spool of wire in the lab. It is a good practice to wire +5V and 0V power supply connections to separate bus strips.



The 5V supply **MUST NOT BE EXCEEDED** since this will damage the ICs (Integrated circuits) used during the experiments. Incorrect connection of power to the ICs could result in them exploding or becoming very hot - with the **possible serious injury occurring to the**

**people working on the experiment! Ensure that the power supply polarity and all components and connections are correct before switching on power.**

## **Building the Circuit**

Throughout these experiments we will use TTL chips to build circuits. The steps for wiring a circuit should be completed in the order described below:

1. Turn the power (Trainer Kit) off before you build anything!
2. Make sure the power is off before you build anything!
3. Connect the +5V and ground (GND) leads of the power supply to the power and ground bus strips on your breadboard.
4. Plug the chips you will be using into the breadboard. Point all the chips in the same direction with pin 1 at the upper-left corner. (Pin 1 is often identified by a dot or a notch next to it on the chip package)
5. Connect +5V and GND pins of each chip to the power and ground bus strips on the breadboard.
6. Select a connection on your schematic and place a piece of hook-up wire between corresponding pins of the chips on your breadboard. It is better to make the short connections before the longer ones. Mark each connection on your schematic as you go, so as not to try to make the same connection again at a later stage.
7. Get one of your group members to check the connections, **before you turn the power on**.
8. If an error is made and is not spotted before you turn the power on. Turn the power off immediately before you begin to rewire the circuit.

### **Common Causes of Problems:**

1. Not connecting the ground and/or power pins for all chips.
2. Not turning on the power supply before checking the operation of the circuit.
3. Leaving out wires.
4. Plugging wires into the wrong holes.
5. Driving a single gate input with the outputs of two or more gates
6. Modifying the circuit with the power on.



## EXPERIMENT 01

- a) Verification of Ohm's law
- b) Study of effect of Open and Short Circuit in Simple Circuits.

### 1. OBJECTIVES

To conduct suitable experiment to verify, Ohm's law and study the effects of open circuit and short circuit on simple circuit.

### 2. THEORY

Ohm's law states that "The voltage across a conductor is directly proportional to the current flowing through it, provided all physical conditions and temperatures remain constant". Hence, according to Ohm's Law, the current flowing through the conductor is directly proportional to the voltage across the circuit, i.e.  $V \propto I$ . Thus, as Ohm's Law provides the basic relation between the voltage applied and current through the conductor.

$$V \propto I \quad \text{OR} \quad V = I \times R$$

$$I = V / R$$

Where,

- **R** - Constant of Proportionality known as Resistance,
- **V** - Voltage applied, and
- **I** - Current flowing through the electrical circuit.

#### I. Open circuit

An open circuit is defined as a condition in an electric circuit where the current does not flow. This occurs when there is no continuous path referred to as a "closed circuit." A break in any part of the circuit results in an open circuit, stopping the flow of current. In an open circuit, the terminals are disconnected, breaking the circuit's continuity. Although this prevents current from flowing, a voltage drop still exists between two points in the circuit.

#### II. Short circuit

A short circuit is an abnormal connection between two nodes of an electric circuit, where a very high amount (infinite) of current passing through the circuit. And the resistance between two terminals of shorts circuits is ideally zero. But practically there is very low resistance. The voltage across short circuit terminals is zero.



### 3. HARDWARE REQUIRED

Sl. No.	NAME OF THE APPARATUS	RANGE / VALUE	QUANTITY
1	Resistors		
2	Regulated Power Supply		
3	Test Kit		
4	Digital Ammeter		
5	Digital Voltmeter		
6	Connecting wires		

#### A. VERIFICATION OF OHM'S LAW:

CIRCUIT DIAGRAM:

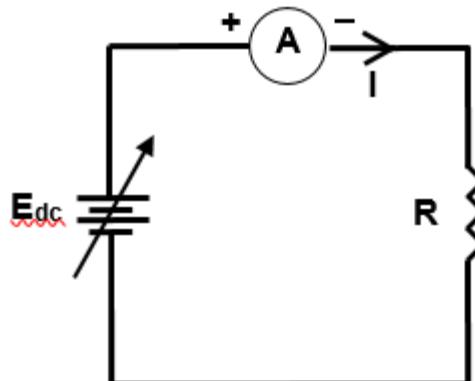


Figure: 1.1

#### 4. TABULAR COLUMN:

Sl. No.	E <sub>dc</sub> (Volts)	R(Ω)	I = E <sub>dc</sub> / R (A) (Theoretical)	Measured I (A)	Remarks
1	5	100			
2	7.5				
3	10				
4	12	100			



## 5. PROCEDURE

1. Rig up the circuit as per the above diagram.
2. For a given resistance, vary the DC input voltage in steps, and note down the respective values of current in the tabular column.
3. Compute the values of current (theoretical) for respective DC input voltages and tabulate them.
4. Observe the proportionality between current and voltage for the verification of Ohm's Law and record the same in the remarks column.
5. Now, for a given DC input voltage, replace the resistors of different values and note down the respective values of current in the tabular column.
6. Compute the values of current (theoretical) for respective resistances and tabulate them.
7. Observe the proportionality between current and resistance for the verification of Ohm's Law and record the same in the remarks column.

## B) STUDY OF EFFECT OF OPEN AND SHORT CIRCUIT IN SIMPLE CIRCUITS

### CIRCUIT DIAGRAM

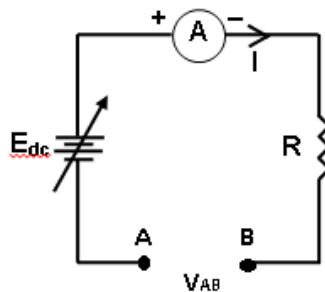


Figure: 2.1

## 6. PROCEDURE

1. Rig up the circuit as per the above diagram.
2. Apply the given DC voltage ( $E$ ) to the circuit.
3. Conduct the experiment with (a) terminals A & B OPEN. (b) terminals A & B SHORT.
4. Tabulate the readings as shown below.
5. Switch OFF DC supply.



## 7. TABULAR COLUMN

### a) OPEN CIRCUIT: (Terminals A & B OPEN)

DC Voltage (E): \_\_\_\_\_ Volts

Sl. No.	Voltage ( $V_{AB}$ ) Volts	Current ( $I_{AB}$ )Amps
1.		

### b) SHORT CIRCUIT (Terminals A & B CLOSE)

DC Voltage (E): \_\_\_\_\_ Volts

Sl. No.	Voltage ( $V_{AB}$ ) Volts	Current ( $I_{AB}$ )Amps
1.		

## 8. RESULT/ OBSERVATIONS



## 9. SAMPLE QUESTIONS

1. What is meant by current?
2. What is meant by voltage?
3. Distinguish between a Branch and a node of a circuit.
4. Define network and circuit?
5. Why ammeter connected in series only?
6. Why the internal resistance of voltmeter is very high, where as that of an ammeter is very low?
7. What is the function of resistance in the circuit?
8. What are the application of Ohm's law.
9. What are the limitations of Ohm's law?
10. State Ohm's law. What is S.I. Unit of resistance?
11. What are non-ohmic resistance? Give two examples.
12. What is difference between open circuit and short circuit?
13. What is the voltage across the terminals in open circuit condition?
14. Why the current is infinite in short circuit?



## EXPERIMENT NO: 02

Verification of KCL and KVL for DC circuits

### 1. OBJECTIVES

To conduct suitable experiments and verify:

- a. Kirchhoff's Current Law (KCL)
- b. Kirchhoff's Voltage Law (KVL).

### 2. THEORY:

Kirchhoffs Circuit Laws allow us to solve complex circuit problems by defining a set of basic network laws and theorems for the voltages and currents around a circuit.

**Kirchhoffs Voltage Law** or KVL, states that “In any closed loop network being driven by a voltage source, the total voltage around the loop is equal to the sum of all the voltage drops within the same loop which is equal to zero”.

In other words the algebraic sum of all voltage sources and voltage drops within a closed loop must be equal to zero.

**Kirchhoffs Current Law** or KCL, states that the “Total current or charge entering a junction or node is exactly equal to the charge leaving the node as it has no other place to go except to leave, as no charge is lost within the node”.

In other words the “Algebraic sum of all the currents entering and leaving a node must be equal to zero”.

### Applications of KCL and KVL:

KCL and KVL play essential roles in designing electronic circuits by ensuring proper current and voltage distribution. They help engineers analyze circuit behavior, solve complex circuit problems, and design efficient and functional electronic devices, such as integrated circuits and printed circuit boards.

### 3. HARDWARE REQUIRED

Sl. No.	Apparatus	Type	Range	Quantity
1.	Resistors			
2.	Regulated Power Supply			
3.	Test Kit			
4.	Digital Ammeter			
5.	Digital Voltmeter			
6.	Connecting wires			

### 4. CIRCUIT DIAGRAM

#### A. Verification of Kirchhoff's current law: -

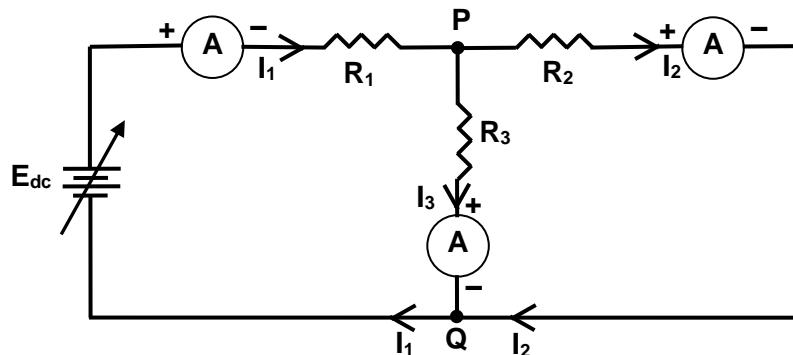


Figure: 2.1

#### Precautions:

1. Avoid loose connections.
2. Keep all the knobs in minimum position while switch on and off of the supply.

### 5. PROCEDURE:

1. Rig up the circuit as per the above diagram.
2. For a given set of resistances ( $R_1$ ,  $R_2$  and  $R_3$ ), by setting the DC input voltage to specific value, note down the respective values of current in the tabular column.
3. Observe the relation between various currents ( $I_1$ ,  $I_2$  and  $I_3$ ) for the verification of Kirchhoff's Current Law (KCL) and record it in the remarks column.
4. Repeat the above steps for different DC input voltages.

## 6. TABULAR COLUMN

Sl. No.	$E_{dc}$ (Volts)	$R_1$ ( $\Omega$ )	$R_2$ ( $\Omega$ )	$R_3$ ( $\Omega$ )	$I_1$ (mA)	$I_2$ (mA)	$I_3$ (mA)	Remarks
1								
2								

### B. Verification of Kirchhoff's voltage law:

Circuit diagram (single loop):

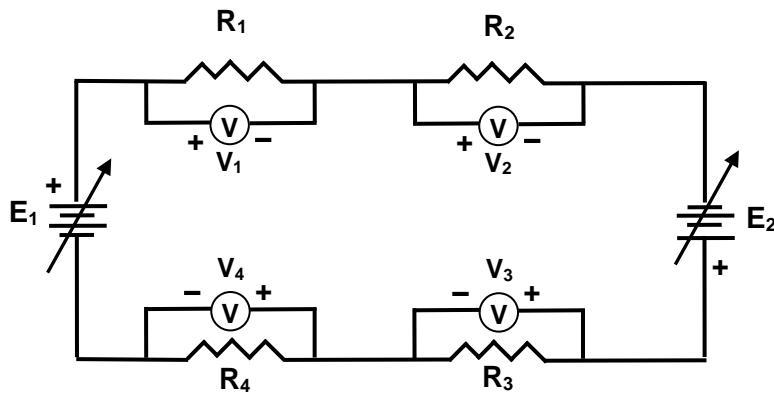


Figure: 2.2

## 7. PROCEDURE:

1. Rig up the circuit as per the above diagram.
2. For a given set of resistances ( $R_1, R_2, R_3, & R_4$ ), by setting the DC input voltages ( $E_1$  and  $E_2$ ) to specific values, note down the values of voltages across respective resistors in the tabular column.
3. Compute  $\Sigma E = E_1 + E_2$  and  $\Sigma V_R = V_1 + V_2 + V_3 + V_4$
4. Observe the relation between  $\Sigma E$  and  $\Sigma V_R$  for the verification of Kirchhoff's Voltage Law (KVL) and record it in the remarks column.
5. Repeat the above steps for another set of DC input voltages ( $E_1$  and  $E_2$ ).



## 8. TABULAR COLUMN:

Sl. No.	E <sub>1</sub> (V)	E <sub>2</sub> (V)	V <sub>1</sub> (V)	V <sub>2</sub> (V)	V <sub>3</sub> (V)	V <sub>4</sub> (V)	$\Sigma V_R$ (V)	$\Sigma E$ (V)	Remarks
1									
2									

## 9. RESULT/ OBSERVATIONS

Thus the Kirchhoff's Current Law and Kirchhoff's Voltage Law were verified for the given circuits and the followings are the remarks.

Kirchhoff's Current Law: \_\_\_\_\_

Kirchhoff's Voltage Law: \_\_\_\_\_



## **10. SAMPLE QUESTIONS**

1. What is meant by current?
2. What is meant by emf?
3. What is the meaning of closed circuit?
4. Distinguish between a Branch and a node of a circuit.
5. What is another name for KCL & KVL?
6. Define network and circuit?



## EXPERIMENT NO: 03

Measurement of Current, Power and Power Factor of Incandescent Lamp, Fluorescent Lamp and LED Lamp

### 1. OBJECTIVE

Measuring the parameters of current, power and power factor of different types of bulbs like incandescent lamp, fluorescent lamp, and LED lamp.

### 2. THEORY

The power factor is a crucial concept used in AC circuits. It's important to note that it has no relevance in DC circuits as these circuits don't exhibit any frequency or phase angle difference between current and voltage.

In an AC circuit, power consists of two components: real power (P) and reactive power (Q). Real power represents the actual energy transfer and performs useful work in the circuit. Reactive power represents the energy stored and released by the inductive or capacitive elements in the circuit, without performing useful work.

Power consumed by the load in AC circuit is given by

$$P=V \cdot I \cdot \cos\phi \text{ watts}$$

$$\text{Power factor } (\cos\phi) = P/VI$$

Where,

- P is active power in watt,
- V is supplied voltage in volts,
- I is current flowing through the circuit elements in Amp.

### 3. HARDWARE REQUIRED

SL. NO.	NAME OF THE APPARATUS	TYPE	RANGE	QUANTITY
1	Analog Ammeter (AC)			
2	Analog Voltmeter (AC)			
3	Wattmeter			
4	Power Factor Meter			

	Lamp load consisting of			
5	<ul style="list-style-type: none"> <li>• Incandescent Lamp</li> <li>• Fluorescent Lamp</li> <li>• LED Lamp (LED).</li> </ul>			
6	Connecting wires			

#### 4. CIRCUIT DIAGRAM

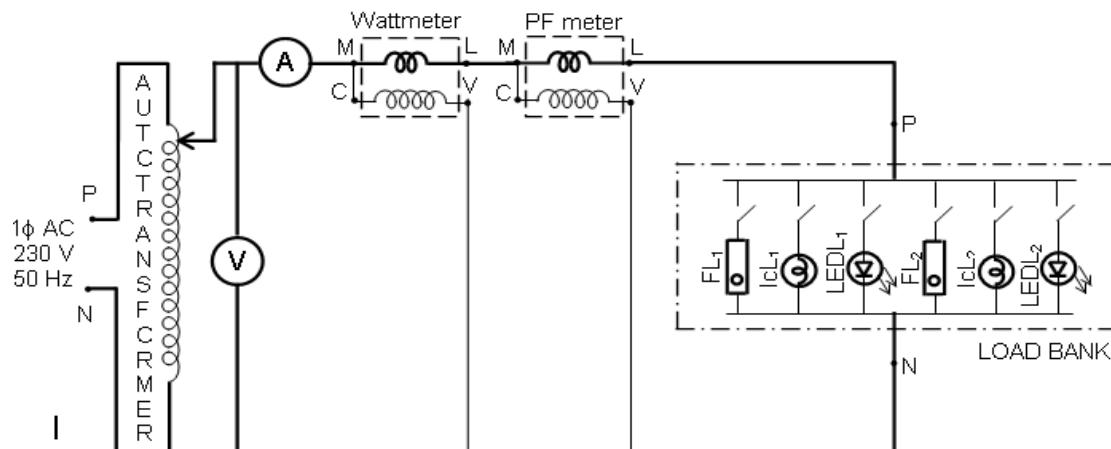


Figure: 3.1

#### Precautions:

1. All connection should be made tight and neat.
2. Care must be taken while using tools.
3. While wiring make sure to turn off the power.

#### 5. PROCEDURE

Rig up the circuit as per the above diagram.

1. Keep the autotransformer at zero voltage position.
2. Select the load as per the tabular column.
3. Switch ON the supply and vary the autotransformer from zero position to rated voltage of 230 V.
4. Note down the respective meter readings.
5. Switch ON different loads as per the tabular column and tabulate the respective readings.
6. Compare the power calculated ( $V I \cos \phi$ ) with the measured power by Wattmeter (P) for all the combinations of loads.
7. Compare the power factors for different loads under remarks.



## 6. TABULAR COLUMN

Sl. No.	Load type	Current <b>I</b> (A)	Voltage <b>V</b> (V)	pf meter reading $\cos \phi$	Wattmeter reading (W)	Power calculated ( <b>V I COS φ</b> ) (W)	Remarks
1	Incandescent (1 Lamp ON)						
2	Incandescent (2 Lamp ON)						
3	Fluorescent (1 Lamp ON)						
4	Fluorescent (2 Lamp ON)						
5	LED (1 Lamp ON)						
6	LED (2 Lamp ON)						
7	Combined All(1+1+1)						
8	Combined All(2+2+2)						

## 7. RESULT/ OBSERVATIONS



## **8. SAMPLE QUESTIONS:**

1. What is power?
2. What is Power Factor?
3. What is SI unit of power?
4. What is wattmeter?
5. What is an AC circuit?
6. What is reactive power?
7. What is active power?
8. What is power triangle of an AC circuit?
9. Mention the current behavior in a resistive, capacitive and inductive circuit.



## EXPERIMENT 04

Determination of Phase and Line Quantities in 3-Phase Star and Delta Connected Loads.

### 1. OBJECTIVE

To conduct suitable experiment(s) for determining phase and line quantities in 3-phase star and delta connected loads.

### 2. THEORY

Voltage or potential difference between any two phases or live terminal is referred to as line voltage. Thus, assume that if the three phases are R, Y, and B, the line voltage can get across R-Y or Y-B or B-R. Line voltage is usually represented using the symbol,  $V_L$ . Voltage or potential difference between any one phase and the neutral point is referred to as Phase Voltage. The phase voltage is represented using the symbol,  $V_{pH}$ .

Star and delta connections are two types of electrical connections used in 3-phase power systems. In a star connection, three phases are connected at a central point, while in a delta connection; the three phases are connected in a loop. In the case of the Star Connection, the **Line Voltage** is  $\sqrt{3}$  times the **Phase Voltage**, but the **Line and Phase Currents** are the same. In the case of the **Delta** connection, the **Line Voltage** and **Phase Voltage** are the same, but the **Line Current** is  $\sqrt{3}$  times the **Phase Current**.

### 3. HARDWARE REQUIRED

Sl. No.	NAME OF THE APPARATUS	Type	RANGE	QUANTITY
1	Test Kit			
2	3-phase Lamp Load Bank			
3	3-phase AC Voltmeter			
4	3-phase AC Ammeter			
5	Connecting wires			

#### 4. CIRCUIT DIAGRAM

##### a. Star connected load

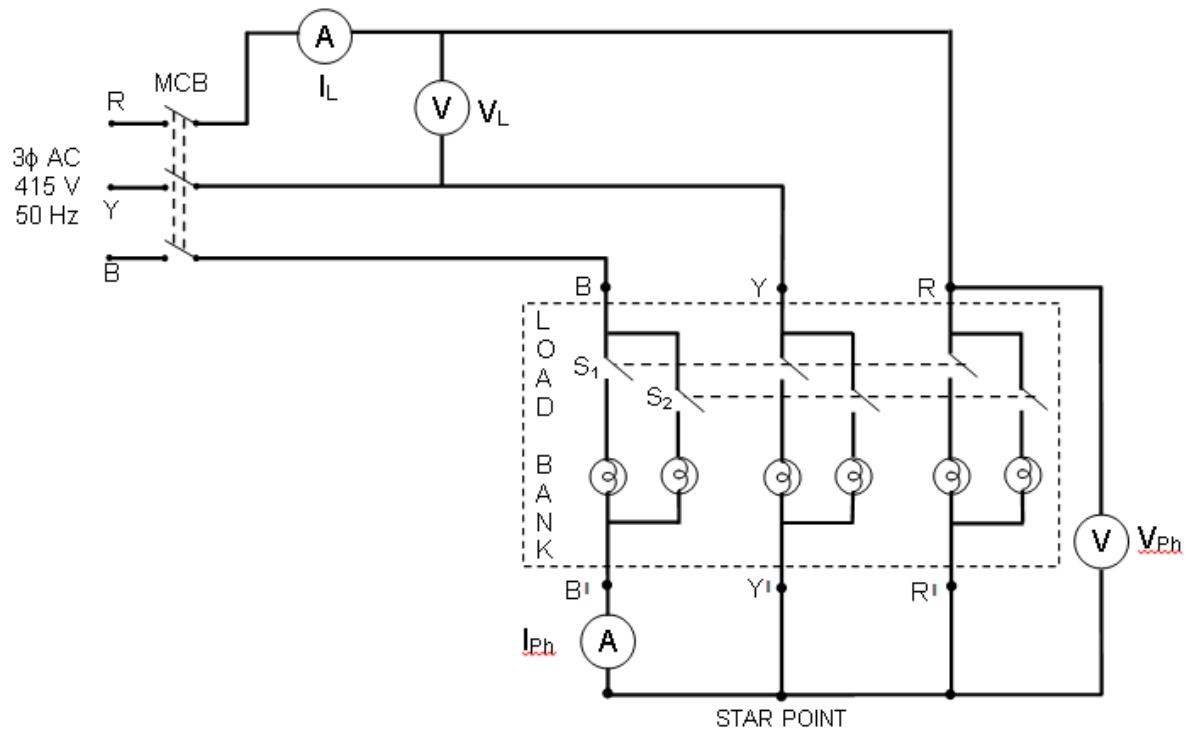


Figure: 4.1

##### b. Delta connected load

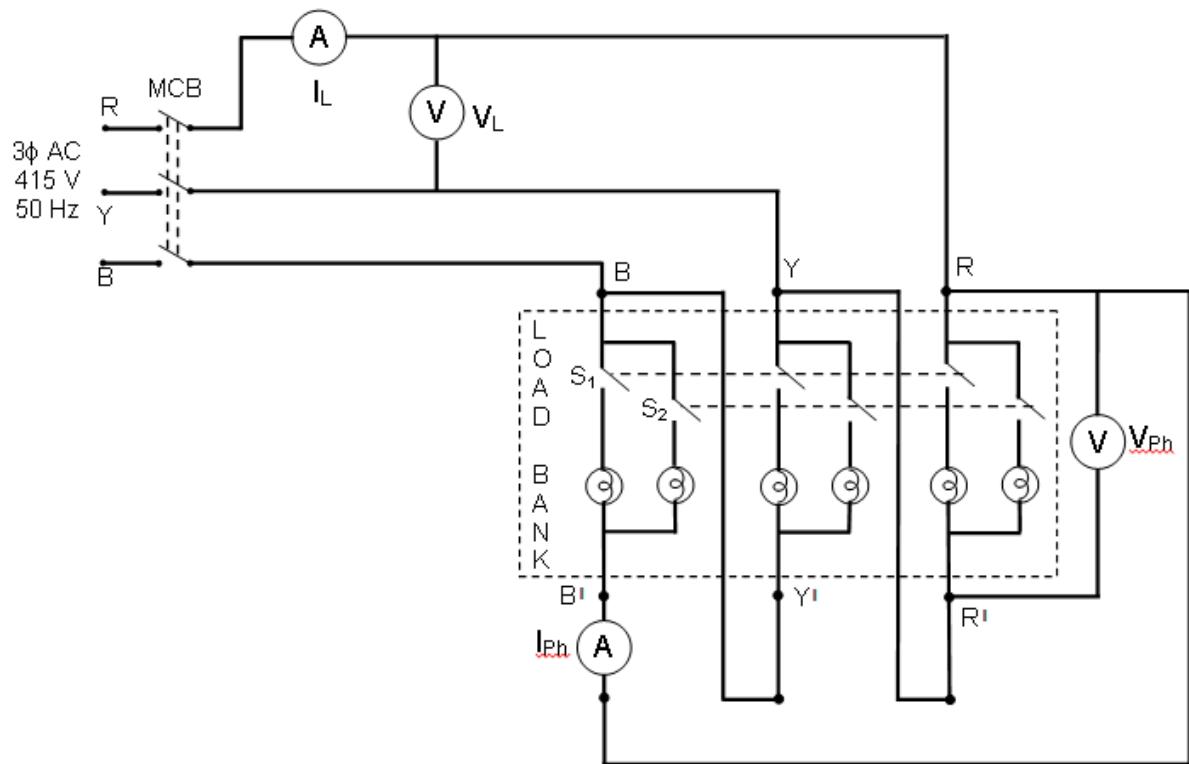


Figure: 4.2



## 5. PROCEDURE

1. Rig up the circuit as per the above diagram.
2. Switch ON the MCB.
3. Keeping switch  $S_1$  ON, note down the readings of AC Voltmeters & AC Ammeters.  
Similarly keeping both the switches  $S_1$  &  $S_2$  ON, note down the readings of AC Voltmeters & AC Ammeters.
4. Turn OFF the load switches  $S_1$  and  $S_2$ .
5. Switch OFF the MCB.

## 6. TABULAR COLUMN

Sl. No.	CONNECTION	Switch ON	Voltage $V_L$ (V)	Voltage $V_{Ph}$ (V)	Current $I_L$ (A)	Current $I_{Ph}$ (A)	REMARKS
1	STAR	$S_1$					
2	STAR	$S_1, S_2$					
3	DELTA	$S_1$					
4	DELTA	$S_1, S_2$					

## 7. RESULT/ OBSERVATIONS



## **8. SAMPLE QUESTIONS**

1. Define phase quantities of star and delta connection?
2. Define line quantities of star and delta connection?
3. Mention the power equation of 3-phase star or delta connection.
4. What is star connection?
5. What is delta connection?
6. What is 3-phase balanced circuit?
7. What is phase sequence?
8. What is the electrical angle displacement in phase sequence (RYB)?



## EXPERIMENT NO: 05

Measurement of 3-Phase Power using 2-Wattmeter method (Demonstration)

### 1. OBJECTIVES

To conduct suitable experiment(s) for measurement of 3-phase power using two wattmeter in a) Star connected load or b) Delta connected load.

### 2. THEORY

Two wattmeter methods are used to measure the power in three-phase circuits for both balanced and unbalanced load. The two-wattmeter method uses two voltage measurements referenced to the same phase (line) and the two currents flowing into that phase. The assumption is that the three-phase system is balanced, i.e., the summation of all voltages = 0 V and the summation of all currents = 0 A. This is true if there is no leakage current from neutral to ground.

### 3. HARDWARE REQUIRED

Sl. No.	NAME OF THE APPARATUS	RANGE / VALUE	QUANTITY
1	Test Kit		
2	3-phase Lamp Load Bank		
3	AC Voltmeter		
4	AC Ammeter		
5	Wattmeter		
6	Connecting wires		

#### 4. CIRCUIT DIAGRAM

##### a. Star connected load:

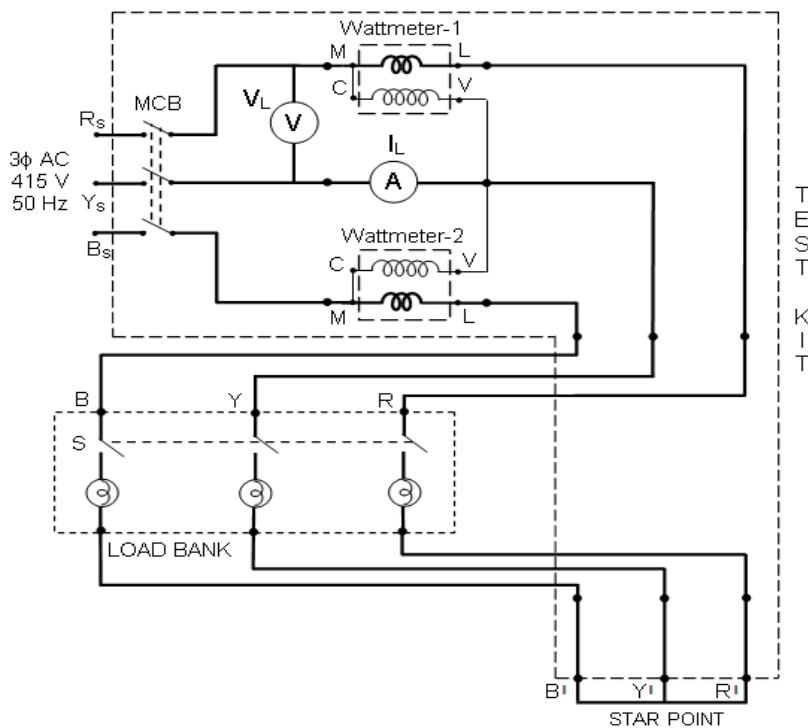


Figure: 5.1

##### b. Delta connected load

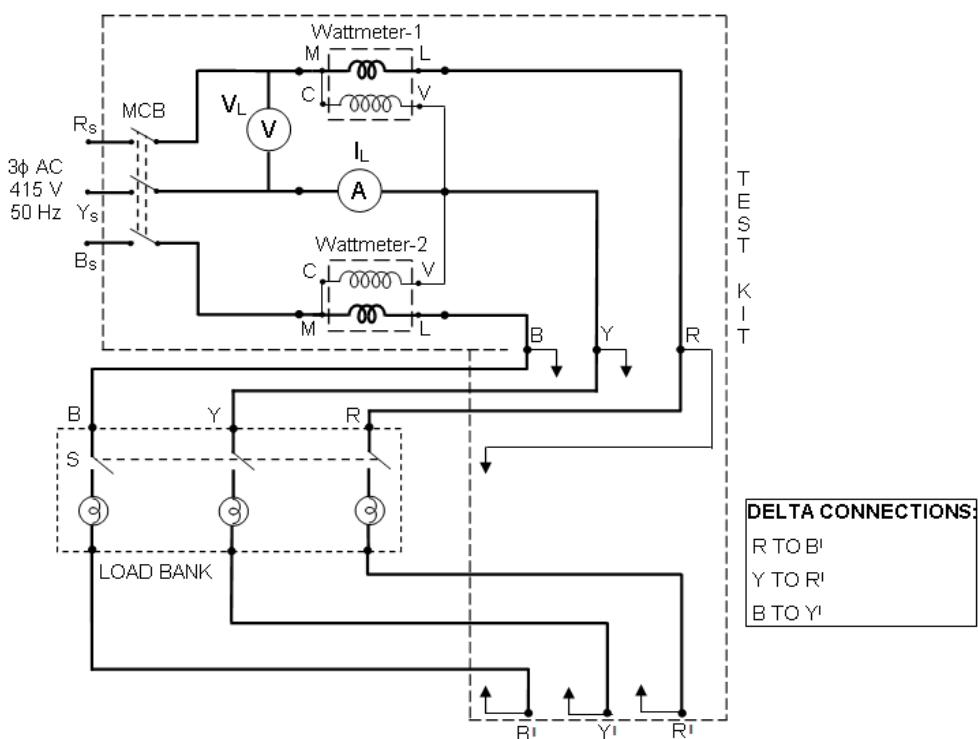


Figure: 5.2



## 5. PROCEDURE

1. Rig up the circuit as per the respective diagram.
2. Switch ON the MCB.
3. Keeping switch  $S_1$  ON, note down the readings of AC Voltmeter, AC Ammeter, Wattmeter-1 & Wattmeter-2.
4. Repeat the above step with
  - (a) Switch  $S_2$  ON
  - (b) Switches  $S_1$  &  $S_2$  ON
5. Tabulate the readings as shown below.
6. Turn OFF the load switches  $S_1$  and  $S_2$ .
7. Switch OFF the MCB.

## 6. TABULAR COLUMN

### 1. Star connected load:

Sl. No.	Switch Condition & Total Load	Voltage $V_L$ (V)	Current $I_L$ (A)	Calculated Power (W) $(\sqrt{3} V_L I_L)$	Wattmeter (W <sub>1</sub> ) (Watts)	Wattmeter (W <sub>2</sub> ) (Watts)	Total Power measured (W <sub>1</sub> +W <sub>2</sub> ) (Watts)
1	$S_1$ ON						
2	$S_2$ ON						
3	$S_1$ & $S_2$ ON						



## 2. Delta connected load:

Sl. No.	Switch Condition & Total Load	Voltage $V_L$ (V)	Current $I_L$ (A)	Calculated Power (W) $(\sqrt{3} V_L I_L)$	Wattmeter (W <sub>1</sub> ) (Watts)	Wattmeter (W <sub>2</sub> ) (Watts)	Total Power measured (W <sub>1</sub> +W <sub>2</sub> ) (Watts)
1	S <sub>1</sub> ON						
2	S <sub>2</sub> ON						
3	S <sub>1</sub> & S <sub>2</sub> ON						

## RESULT

3-Phase Power is measured and verified using 2-Wattmeter method.



## SAMPLE QUESTIONS

1. Which meter is used to measure the power?
2. Mention different methods to measure 3-phase power.
3. Why 2 wattmeters are used for measurement of 3-phase power?
4. What is current coil and voltage coil in wattmeter?
5. What is M, L, C and V in wattmeter?
6. What is a balanced load?
7. What are the advantages of using a three-phase AC circuit over a single-phase AC circuit?
8. What is difference between power factor meter and wattmeter?
9. What is MCB, Fuse and neutral connection?
10. What is earthing?



## EXPERIMENT NO: 06(a)

To observe waveform at the output of half wave rectifier, full wave rectifier.

### 1. OBJECTIVES

Calculate the ripple factor, rectification efficiency and % regulation.

### 2. THEORY

Rectifier changes ac to dc and it is an essential part of power supply. The unique property of a diode, permitting the current to flow in one direction, is utilised in rectifiers.

#### Half Wave Rectifier

Mains power supply is applied at the primary of the step-down transformer. All the positive half cycles of the stepped down ac supply pass through the diode and all the negative half cycles get eliminated. Peak value of the output voltage is less than the peak value of the input voltage by 0.6V because of the voltage drop across the diode.

For a half wave rectifier,

$$V_{rms} = V_m/2 \text{ and } V_{dc} = V_m/\pi:$$

$V_{rms}$  = rms value of input,

$V_{dc}$  = Average value of input

$V_m$  = peak value of output.

$$\text{Ripple factor } r = V_{r, rms}/V_{dc}$$

Where  $V_{r, rms}$  is the rms value of the ac component

Since  $V_{rms}^2 = V_{r, rms}^2 + V_{dc}^2$ ,

$$\text{Ripple Factor} = r = \sqrt{\left(\frac{V_{rms}}{V_{dc}}\right)^2 - 1} = 1.21$$

#### Full Wave Rectifier

During the positive half cycle of the transformer secondary voltage, diode D1 is forward biased and D2 is reverse biased. So a current flows through the diode D1, load resistor  $R_L$  and upper half of the transformer winding. During the negative half cycle, diode D2 becomes forward biased and D1 becomes reverse biased. The current then flows through the diode D2, load resistor  $R_L$  and lower half of the transformer winding. Current flows through the load resistor in the same direction during both the half cycles. Peak value of the output voltage is less than the peak value of the input voltage by 0.6V because of the voltage drop across the diode.



For a full wave rectifier

$$V_{rms} = V_m \sqrt{2}, V_{dc} = 2V_m/\pi$$

$$\text{Ripple factor} = r = \sqrt{\left(\frac{V_{rms}}{V_{dc}}\right)^2 - 1} = 0.48$$

### Rectifier Efficiency

Rectifier efficiency is defined as the ratio of DC output power to the input power from the AC supply. Even with ideal rectifiers with no losses, the efficiency is less than 100% because some of the output power is AC power rather than DC which manifests as ripple superimposed on the DC waveform.

$$\eta = \frac{V_{dc}^2}{V_{ac}^2}$$

It is a measure of the variation of DC output voltage as a function of DC output current i.e. variation in load.

$$\% \text{ regulation} = \left( \frac{V_{NL} - V_{FL}}{V_{FL}} \right) * 100\%$$

$V_{NL}$ = Voltage across load resistance when minimum current flows through it.

$V_{FL}$ = Voltage across load resistance when maximum current flows through it.

### 3. HARDWARE REQUIRED

Sl no	Apparatus	Type	Range	Quantity
1	Transformer	Step down	Input 230V Ac, Output 6v AC, 500 Ma	1
2	Diode	D1N4002		
3	Resistor		1k	1
4	Capacitor		100uf	1
5	Voltmeter			1
6	Toggle Switch			1
7	Ammeter			1
8	Bread Board			1
9	Probes			2

#### 4. CIRCUIT DIAGRAM:

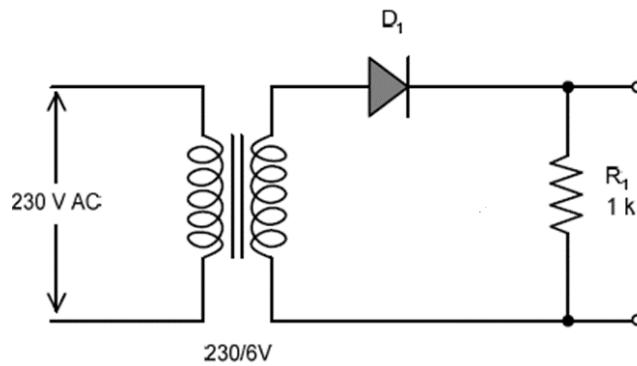


Figure: 6.1 Half wave rectifier

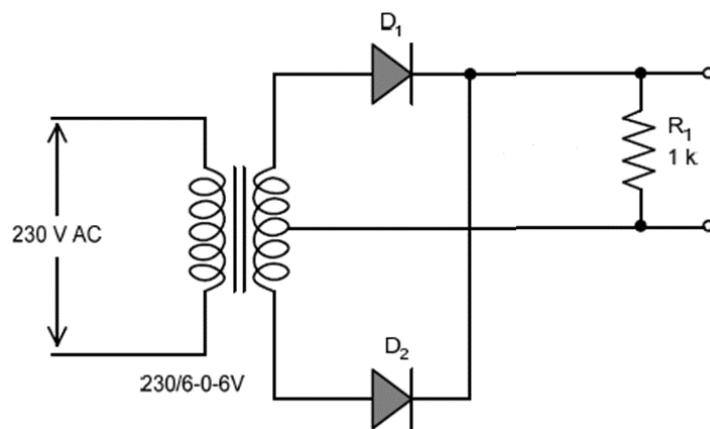


Figure: 6.2 Full wave rectifier

#### 5. WAVEFORM

Typical waveforms of half wave rectifier without filter and with filter are shown in the figure below:

##### a) Half wave Rectifier

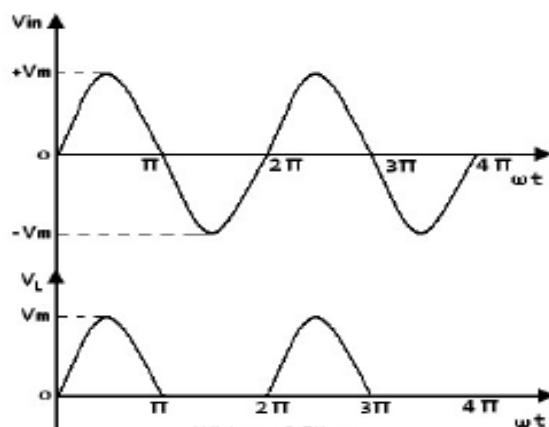


Figure: 6.3 Waveform of Half wave rectifier

b) Full wave Rectifier

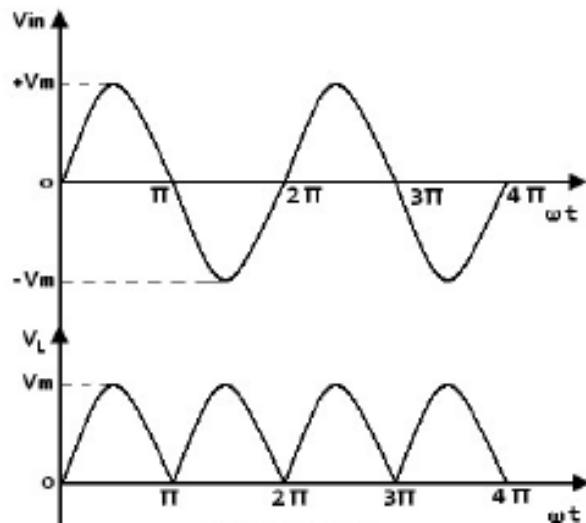


Figure: 6.4 Waveform Full wave rectifier

## 6. PROCEDURE:

1. Wire up the half wave rectifier circuit without capacitor after testing all the components.
2. Switch on the main supply. Observe the transformer secondary voltage waveform and output voltage waveform across the load resistor, simultaneously on the CRO screen. Note down  $V_m$  and calculate  $V_{rms}$  and  $V_{dc}$ .
3. Calculate the ripple factor, rectifier efficiency and % regulation using the expressions.
4. Repeat the above steps for full wave rectifiers.

## 7. PRECAUTIONS

1. The primary and secondary side of the transformer should be carefully identified
2. The polarities of all the diodes should be carefully identified.
3. While determining the % regulation, first full load should be applied and then it should be decremented in steps



## 8. OBSERVATION TABLE

<b>HWR</b>	$V_m$	$V_{rms} = V_m/2$	$V_{dc} = V_m/\pi$	$r = \sqrt{(V_{rms}/V_{dc})^2 - 1}$
<b>FWR</b>	$V_m$	$V_{rms} = V_m/\sqrt{2}$	$V_{dc} = 2V_m/\pi$	$r = \sqrt{(V_{rms}/V_{dc})^2 - 1}$

<b>HWR</b>	Rectifier Efficiency $\eta = \frac{I_{dc}^2}{I_{dc}^2 + I_{ac}^2} \times 100$	% Regulation $\frac{V_{dc(NL)} - V_{dc}}{V_{dc}} \times 100$
<b>FWR</b>		

## 9. RESULT/OBSERVATIONS

## Using Pspice Software:

### 1. OBJECTIVE:

Simulate and study Half-wave and Full-wave Rectifier using PSPICE windows.

### 2. CIRCUIT DIAGRAM:

#### Half wave rectifier

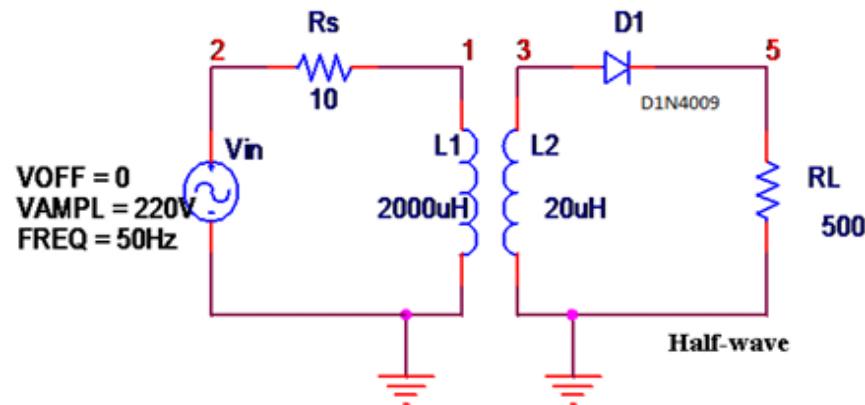


Figure: 6.5 Half wave rectifier

#### Full wave Rectifier

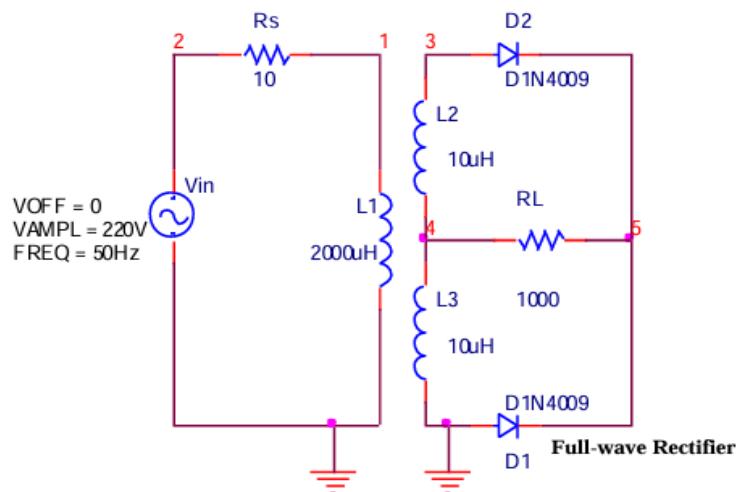


Figure: 6.6 Full wave rectifier

### 3. RESULT/OBSERVATIONS



#### **4. POST LAB QUESTION**

1. What is the mathematical relationship between rms input AC voltage and DC output voltage in half wave rectifier?
2. What is the frequency of AC component at the output of full wave rectifier? Give reason.

#### **5. SAMPLE QUESTIONS**

1. What is the efficiency of half wave rectifier?
2. What is the rectifier?
3. What is the difference between the half wave rectifier and Full Wave Rectifier?
4. Define peak inverse voltage (PIV)? And write its value for Full-wave rectifier?
5. If one of the diode is changed in its polarities what wave form would you get?
6. Does the process of rectification alter the frequency of the waveform?
7. What is ripple factor of the Full-wave rectifier?
8. What is the necessity of the transformer in the rectifier circuit?
9. What are the applications of a rectifier?
10. What is meant by ripple and define Ripple factor?
11. Explain how capacitor helps to improve the ripple factor?

## EXPERIMENT NO: 07

Determine the input and output characteristics of Zener diode.

### 1. OBJECTIVES

To analyse the Volt-Ampere characteristics of Zener diode and to measure the Zener break down voltage.

### 2. THEORY

The Zener diode is like a general-purpose signal diode. When biased in the forward direction it behaves just like a normal signal diode, but when a reverse voltage is applied to it, the voltage remains constant for a wide range of currents.

#### Avalanche Breakdown:

When the diode is in the reverse bias condition, the width of the depletion region is more. If both p-side and n-side of the diode are lightly doped, depletion region at the junction widens. In reverse bias, the minority charge carrier current flows through junction. As the applied reverse voltage increases the minority carriers acquire sufficient energy to collide with the carriers in the covalent bonds inside the depletion region. As a result, the bond breaks and electron hole pairs are generated. The process becomes cumulative and leads to the generation of a large number of charge carriers resulting in Avalanche Breakdown. At this stage maximum current will flow through the zener diode. This breakdown point is referred as “**Zener voltage**”.

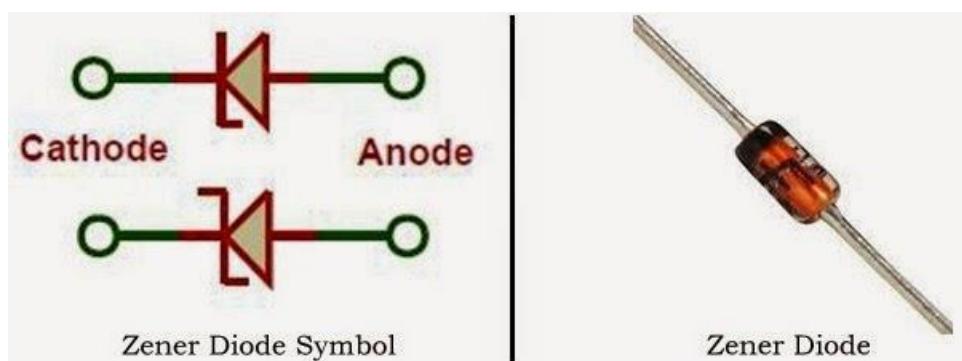


Figure: 7.1 Zener diode

#### Zener Break down:

If both p-side and n-side of the diode are heavily doped, depletion region at the junction reduces compared to the width in normal doping. Applying a reverse bias causes a strong



electric field get applied across the device. As the reverse bias is increased, the Electric field becomes strong enough to rupture covalent bonds and generate large number of charge carriers. Such sudden increase in the number of charge carriers due to rupture of covalent bonds under the influence of strong electric field is termed as Zener breakdown. The Zener Diode is used in its "reverse bias". From the V-I Characteristics curve we can study that the zener diode has a region in its reverse negative voltage regardless of the value of the current flowing through the diode and remains nearly constant even with large changes in current as long as the zener diodes current remains between the breakdown current  $I_Z$  (min) and the maximum current rating  $I_Z$  (max).

This ability to control itself can be used to great effect to regulate or stabilize a voltage source against supply or load variations. The fact that the voltage across the diode in the breakdown region is almost constant turns out to be an important application of the zener diode as a voltage regulator.

Zener diodes are available from about 2.4 to 200 volts as 2.4, 2.7, 3.0 3.3, 3.6, 3.9, 4.3, 4.7, 5.1, 5.6, 6.2, 6.8, 7.5, 8.2, 9.1, 10, 11, 12, 13, 15, 16, 18, 20, 22, 24, etc. All Zener diodes have a power rating,  $P_Z$ . From Watt's law the maximum current is  $I_{Z(MAX)}=P_Z / V_Z$ . Zener diodes are typically available with power ratings of 0.25, 0.4, 0.5, 1, 2, 3, and 5 watts although other values are available. We also carry Zener diodes with nominal working voltage up to 1 kV. Forward (drive) current can have a range from 200 uA to 200 A, with the most common forward (drive) current being 10 mA or 200 mA.

### 3. HARDWARE REQUIRED

Sl no	Particulars	Type	Range	Quantity
1	Zener diode	$I_Z$ 6.2		1
2	Resistance		1k ohm, 10% tolerance, 1/2 watt rating	1
3	Regulated power supply		(0 – 30V), 2A rating	1
4	Ammeter		(0-30)mA	1
5	Voltmeter		(0 – 1)V, (0 – 10)V	1
6	Breadboard, connecting wires			

#### 4. CIRCUIT DIAGRAM

##### Forward Bias

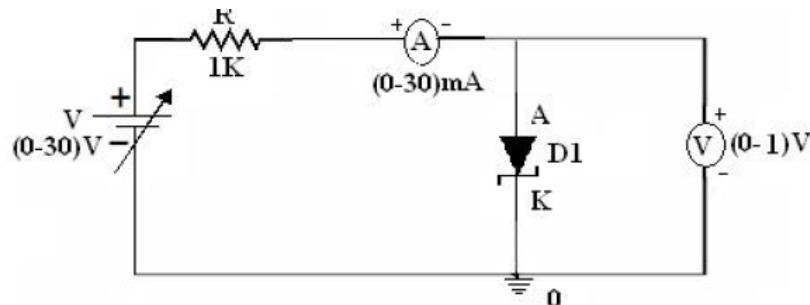


Figure: 7.2 Zener diode forward bias

##### Circuit diagram: Reverse Bias

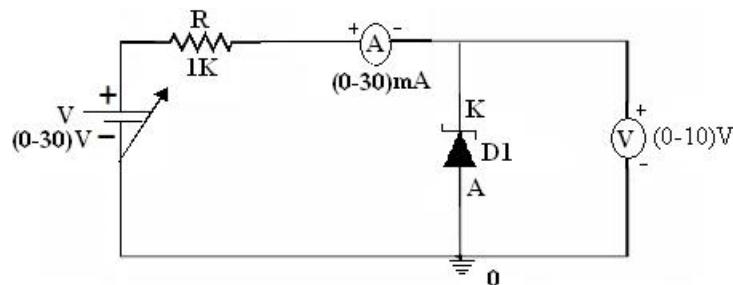


Figure: 7.3 Zener diode forward bias

##### Precautions:

1. While doing the experiment do not exceed the ratings of the diode. This may lead to damage of the diode.
2. Connect voltmeter and Ammeter in correct polarities as shown in the circuit diagram.
3. Do not switch **ON** the power supply unless you have checked the circuit connections as per the circuit diagram.

#### 5. MODEL GRAPH

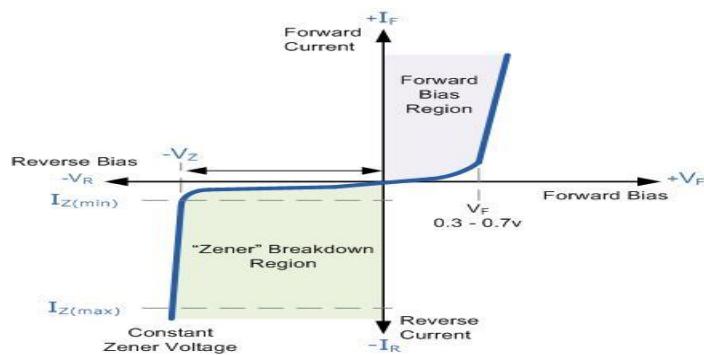


Figure: 7.4 Zener diode V-I Characteristics

### Forward Biased Condition:

1. Connect the Zener diode in forward bias i.e; anode is connected to positive of the power supply and cathode is connected to negative of the power supply as in circuit.
2. Use a Regulated power supply of range (0-30) V and a series resistance of  $1\text{k}\Omega$ .
3. For various values of forward voltage ( $V_f$ ) note down the corresponding values of forward Current ( $I_f$ ).

### Reverse biased condition:

1. Connect the Zener diode in Reverse bias i.e.; anode is connected to negative of the power supply and cathode is connected to positive of the power supply as in circuit.
2. For various values of reverse voltage ( $V_r$ ) note down the corresponding values of reverse current ( $I_r$ ).

## 6. TABULAR COLUMN

Table: 7.1 Forward Bias

Sl No	$V_f$ (in Volts)	$I_f$ (in mA)



Table: 7.2 Reverse Bias

Sl No	V <sub>r</sub> (in Volts)	I <sub>r</sub> (in $\mu$ A)

## 7. RESULTS/OBSERVATION

## Using Pspice Software:

### 1. OBJECTIVES

To analyse the Volt-Ampere characteristics of Zener diode and to measure the Zener break down voltage (DC Sweep analysis) using PSPICE software.

### 2. CIRCUIT DIAGRAM

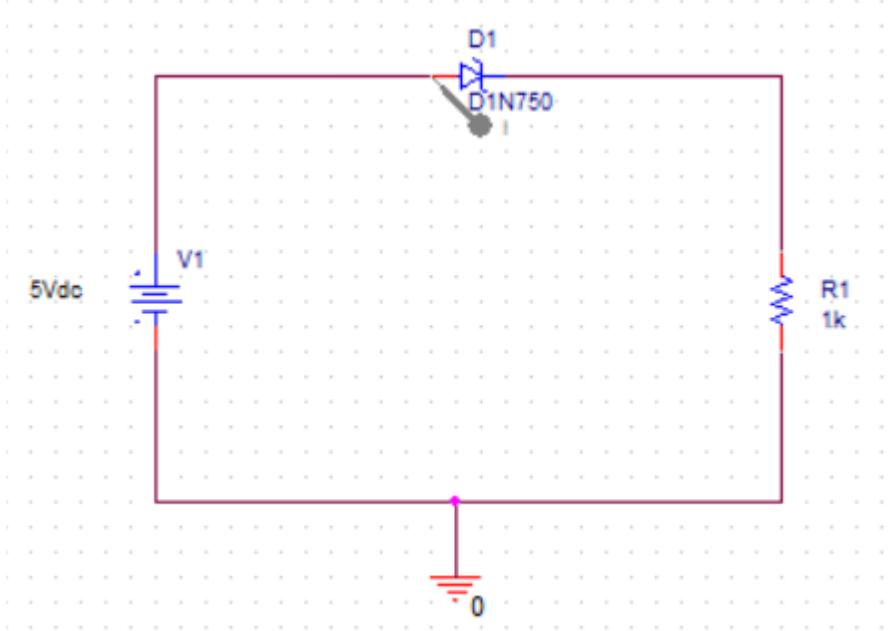


Figure: 7.5 Circuit diagram for Zener diode Characteristics

### 3. RESULTS/OBSERVATIONS



#### **4. POST LAB QUESTION**

1. Generate input and output characteristics in PN Junction diode using PSPICE and compare with the obtained output of Zener Diode.
2. Simulate Zener diode V-I Characteristics (using Time Domain Analysis).

#### **5. SAMPLE QUESTIONS**

1. Justify the use of zener diode in a stabilization circuit?
2. Explain the concept of Zener breakdown?
3. How does a zener diode protect meters from excess voltage that is applied accidentally?
4. State the reason why an ordinary diode suffers avalanche breakdown rather than Zener breakdown?
5. Give the reasons why Zener diode acts as a reference element in the voltage regulator circuits.

## EXPERIMENT NO: 08(a)

To observe input and output characteristics of Common Emitter (CE) NPN transistor.

### 1. OBJECTIVES

- a) Plot the input and output characteristics for CE NPN transistor.

### 2. THEORY

Transistor is three terminal active device having terminals collector, base and emitter. To understand operation of the transistor, we use three configurations common emitter, common base and common collector. In this practical, we will understand common emitter configuration.

#### Common Emitter (CE) Configuration.

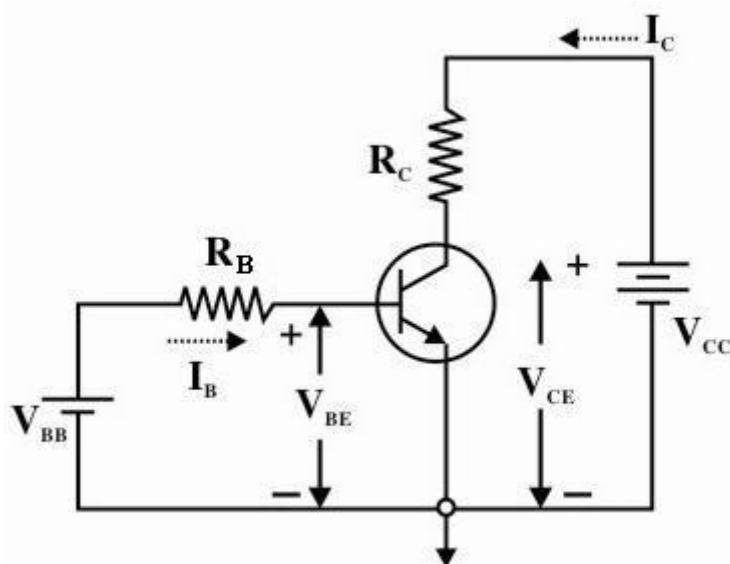


Figure: 8a.1 Common Emitter Configuration

As the name suggest, emitter is common between input and output. Input is applied to base and output is taken from collector. We will obtain input characteristics and output characteristics of common emitter (CE) configuration. We will connect variable DC power supply at  $V_{BB}$  and  $V_{CC}$  to obtain characteristics. Input voltage in CE configuration is base-emitter voltage  $V_{BE}$  and input current is base current  $I_B$ . Output voltage in CE configuration is collector to emitter voltage  $V_{CE}$  and output current is collector current  $I_C$ . We will use multimeter to measure these voltages and currents for different characteristics. Collector to emitter junction is reverse biased and base to emitter junction is forward biased. The CE configuration is widely used in amplifier circuits because it provides voltage gain as well as current gain. In CB configuration current gain is less than unity. In CC configuration voltage gain is less than

unity. Input resistance of CE configuration is less than CC configuration and more than CB configuration. Output resistance of CE configuration is more than CC configuration and less than CB configuration.

### 3. HARDWARE REQUIRED

Table: 8a.1 Hardware required

Sl. no	Apparatus	Type	Range	Quantity
1	NPN transistor			1
2	Resistor		1k	3
3	Voltmeter			1
4	Ammeter			1
5	Bread Board			1
6	Probes			2
7	Variable voltage source			2

### 4. CIRCUIT DIAGRAM:

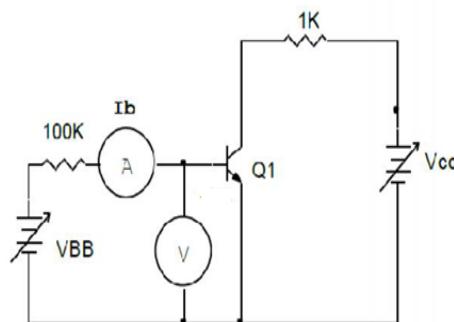


Figure: 8a.2 Circuit setup for input characteristics of CE NPN transistor

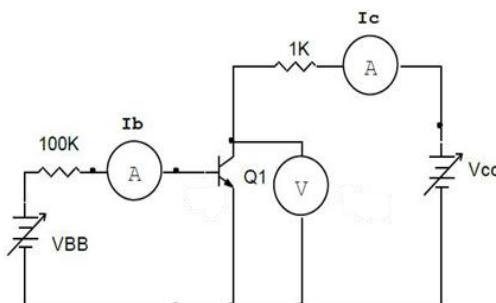


Figure 8a.3: Circuit setup for output characteristics CE NPN transistor

## 5. WAVEFORM

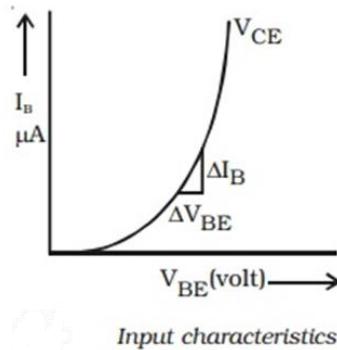


Figure: 8a.4 Input Characteristics of CE NPN transistor

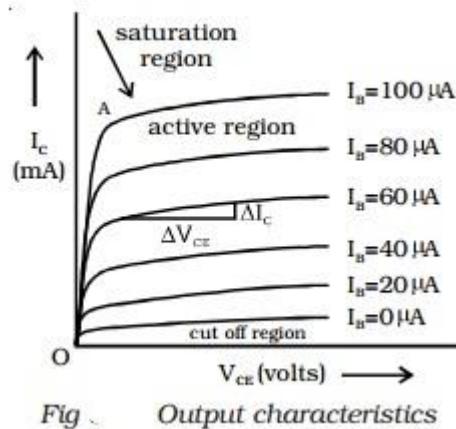


Figure: 8a.5 Output Characteristics of CE NPN transistor

## 6. PROCEDURE:

### Experiment Procedure for input characteristics:

1. Connect circuit as shown in the circuit diagram for input characteristics
2. Connect variable power supply 0-30V at base circuit and collector circuit.
3. Keep  $V_{CC}$  fix at 0V (Or do not connect  $V_{CC}$ )
4. Increase  $V_{BB}$  from 0V to 20V, note down readings of base current  $I_B$  and base to emitter voltage  $V_{BE}$  in the observation table.
5. Repeat above procedure for  $V_{CC} = +5V$  and  $V_{CC} = +10V$
6. Draw input characteristics curve. Plot  $V_{BE}$  on X axis and  $I_B$  on Y axis.



### **Experiment Procedure for output characteristics:**

1. Connect circuit as shown in the circuit diagram for output characteristics
2. Connect variable power supply 0-30V at base circuit and collector circuit.
3. Keep base current fix (Initially 0)
4. Increase  $V_{CC}$  from 0V to 30V, note down readings of collector current  $I_C$  and collector to emitter voltage  $V_{CE}$  in the observation table.'Repeat above procedure for base currents  $I_B = 5\mu A, 50 \mu A, 100 \mu A$ . Increase base current by increasing  $V_{BB}$ .
5. Draw output characteristics curve. Plot  $V_{CE}$  on X axis and  $I_{CE}$  on Y axis.

### **7. OBSERVATION TABLE:**

Table: 8a.2 Input plot observation table

Sr. No.	<b><math>V_{CC} = 0V</math></b>		<b><math>V_{CC} = +5V</math></b>		<b><math>V_{CC}=+10V</math></b>	
	<b><math>V_{BE}</math></b>	<b><math>I_B</math></b>	<b><math>V_{BE}</math></b>	<b><math>I_B</math></b>	<b><math>V_{BE}</math></b>	<b><math>I_B</math></b>
1						
2						
3						
4						
5						
6						

Table: 8a.3 Output Plot observation table

Sr. No.	<b><math>I_B = 0</math></b>		<b><math>I_B = 5\mu A</math></b>		<b><math>I_B = 50\mu A</math></b>		<b><math>I_B = 100\mu A</math></b>	
	<b><math>V_{CE}</math></b>	<b><math>I_C</math></b>	<b><math>V_{CE}</math></b>	<b><math>I_C</math></b>	<b><math>V_{CE}</math></b>	<b><math>I_C</math></b>	<b><math>V_{CE}</math></b>	<b><math>I_C</math></b>
1								
2								
3								
4								
5								
6								
7								
8								



## **PRECAUTIONS**

1. Power source should be off while connecting all components.
2. The polarities of all the transistor should be carefully identified.

## **8. RESULT/OBSERVATIONS**

## USING P-SPICE SOFTWARE:

### 1. CIRCUIT DIAGRAM

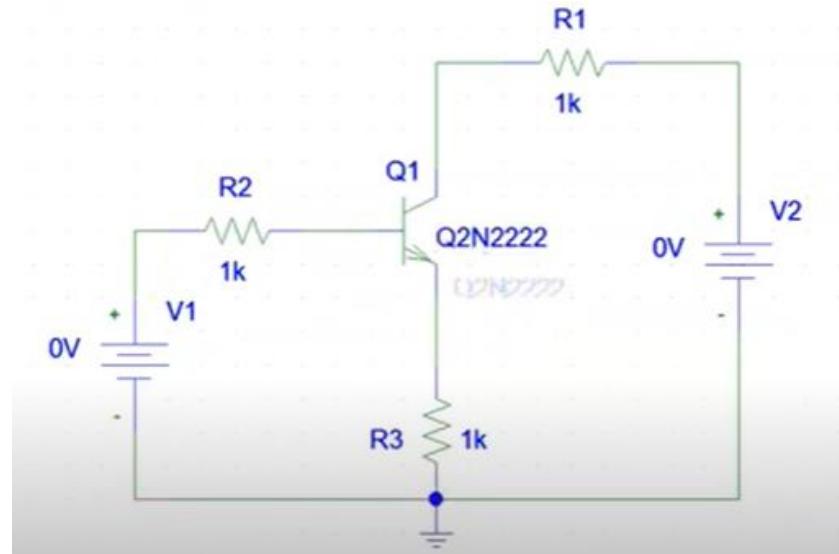


Figure 8a.6 Common Emitter configuration Circuit Diagram using PSPICE

### 2. RESULT/OBSERVATIONS

## EXPERIMENT NO: 08(b)

To observe input and output characteristics of Common Base (CB) NPN transistor.

### 1. OBJECTIVES

Plot the input and output characteristics for CB NPN transistor.

### 2. THEORY

#### Common Base (CB) Configuration

In a common base configuration, base terminal is common between input and output. The output is taken from collector and the input voltage is applied between emitter and base. The base is grounded because it is common. To obtain output characteristics, we will measure collector current for different value of collector to base voltage ( $V_{CB}$ ). Input current is emitter current  $I_E$  and input voltage is  $V_{EB}$ . To plot input characteristics we will plot  $V_{EB}$  versus  $I_E$ . Current gain for CB configuration is less than unity. CB configuration is used in common base amplifier to obtain voltage gain. Output impedance of common base configuration is very high. CB amplifier is used in multi-stage amplifier where impedance matching is required between different stages. In common base configuration circuit is shown in figure. Here base is grounded, and it is used as the common terminal for both input and output. It is also called as grounded base configuration. Emitter is used as a input terminal whereas collector is the output terminal.

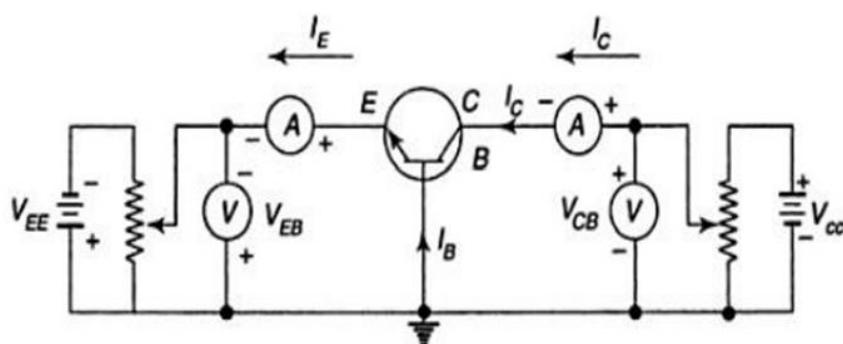


Figure: 8b.1 Common base NPN transistor

### 3. HARDWARE REQUIRED

Table: 8b.1 Hardware Required

Sl no	Apparatus	Type	Range	Quantity
1	NPN transistor			1
2	Resistor		1k	3
3	Voltmeter			1
4	Ammeter			1
5	Bread Board			1
6	Probes			2
7	Variable voltage source			2

### 4. CIRCUIT DIAGRAM:

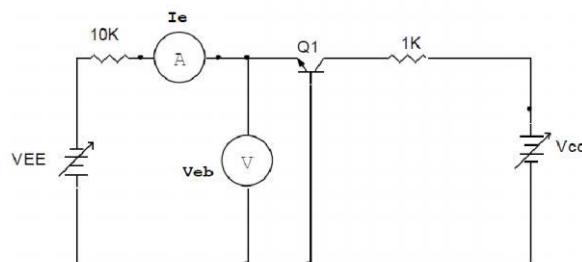


Figure: 8b.2 Circuit diagram to obtain input characteristics

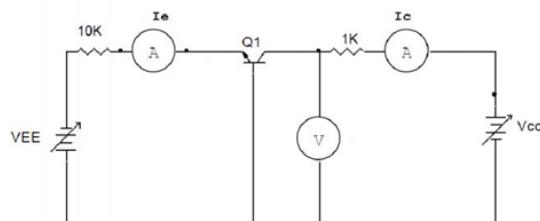


Figure: 8b.3 Circuit diagram to obtain output characteristics

### 5. WAVEFORM

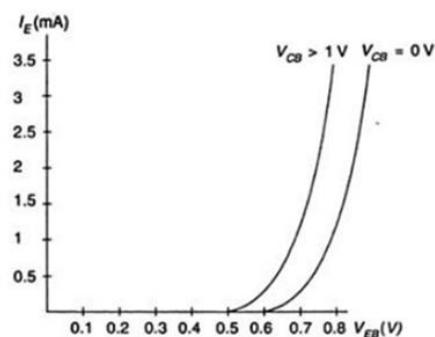


Figure 8b.4: Input Characteristics

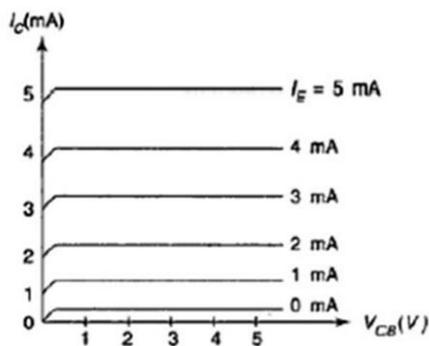


Figure: 8b.5 Output Characteristics

## 6. PROCEDURE:

### Experiment procedure to obtain input characteristics:

1. Connect circuit as shown in the circuit diagram for input characteristics 55
2. Connect variable power supply 0-30V ( $V_{EE}$ ) at emitter base circuit and another power supply 0-30V at collector base circuit ( $V_{CC}$ ).
3. Keep  $V_{CC}$  fix at 0V (Or do not connect  $V_{CC}$ )
4. Increase  $V_{EE}$  from 0V to 20V, note down readings of emitter current  $I_E$  and emitter to base voltage  $V_{EB}$  in the observation table.
5. Repeat above procedure for  $V_{CC} = +5V$  and  $V_{CC} = +10V$
6. Draw input characteristics curve. Plot  $V_{EB}$  on X axis and  $I_E$  on Y axis.

### Experiment procedure to obtain output characteristics:

1. Connect circuit as shown in the circuit diagram for output characteristics
2. Connect variable power supply 0-30V at emitter circuit and collector circuit.
3. Keep emitter current fix (Initially 0)
4. Increase  $V_{CC}$  from 0V to 30V, note down readings of collector current  $I_C$  and collector to base voltage  $V_{CB}$  in the observation table.
5. Repeat above procedure for base currents  $I_E = 1\text{mA}$ ,  $5\text{ mA}$  and  $10\text{mA}$ . Increase emitter current by increasing  $V_{EE}$ .
6. Draw output characteristics curve. Plot  $V_{CB}$  on X axis and  $I_C$  on Y axis.



## 7. OBSERVATION TABLE:

Table: 8b.2 Input Plot observation Table

Sr. No.	V <sub>cc</sub> = 0V		V <sub>cc</sub> = +5V		V <sub>cc</sub> =+10V	
	V <sub>cb</sub>	I <sub>c</sub>	V <sub>cb</sub>	I <sub>c</sub>	V <sub>cb</sub>	I <sub>c</sub>
1						
2						
3						
4						
5						
6						

Table: 8b.3 Output Plot observation table:

Sr. No.	I <sub>e</sub> = 0		I <sub>e</sub> = 1 mA		I <sub>e</sub> = 5 mA		I <sub>e</sub> = 10 mA	
	V <sub>cb</sub>	I <sub>c</sub>	V <sub>cb</sub>	I <sub>c</sub>	V <sub>cb</sub>	I <sub>c</sub>	V <sub>cb</sub>	I <sub>c</sub>
1								
2								
3								
4								
5								
6								
7								
8								

### PRECAUTIONS

1. Power source should be off while connecting all components.
2. The polarities of all the transistor should be carefully identified.



## **8. RESULT/OBSERVATIONS**



## USING P-SPICE SOFTWARE:

### 1. CIRCUIT DIAGRAM

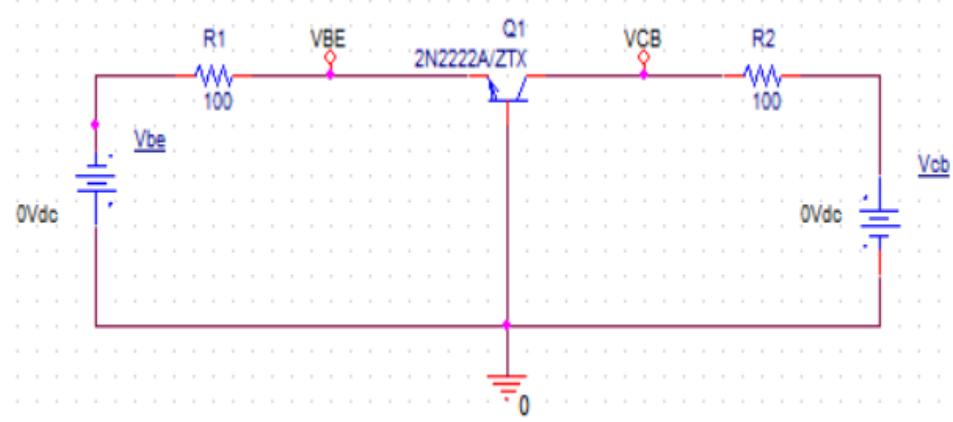


Figure: 8b.6 Common Base Circuit Diagram using PSPICE

### 2. RESULT/OBSERVATIONS



## EXPERIMENT NO: 08(c)

To observe input and output characteristics of Common Collector (CC) NPN transistor.

### 1. OBJECTIVES

Plot the input and output characteristics for CC NPN transistor.

### 2. THEORY

#### Common Collector (CC) Configuration

In common collector configuration, collector terminal is common between input and output. Input is applied between base and collector. Output is taken from emitter and collector. Voltage gain of CC configuration is less than unity. CC amplifier is used to provide current gain. It has very high input impedance and very low output resistance hence it is used to connect low impedance load to source which is having high output impedance. Thus, it can be used as impedance matching. Emitter current is approximately equal to collector current, hence output characteristics of CC configuration is very much same as CE configuration. However, input characteristics of CE and CC are quite different. In common collector configuration circuit is shown in figure. Here collector is grounded, and it is used as the common terminal for both input and output. It is also called as grounded collector configuration. Base is used as a input terminal whereas emitter is the output terminal.

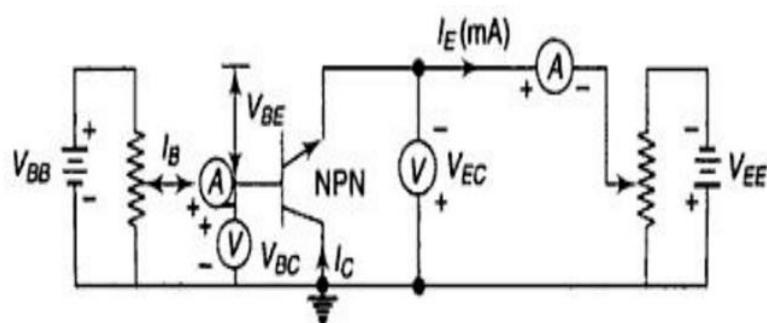


Figure: 8c.1 CC NPN transistor

### 3. HARDWARE REQUIRED

Table: 8c.1 Hardware Required

Sl no	Apparatus	Type	Range	Quantity
1	NPN transistor			1
2	Resistor		1k	3
3	Voltmeter			1
4	Ammeter			1
5	Bread Board			1
6	Probes			2
7	Variable voltage source			2

### 4. CIRCUIT DIAGRAM:

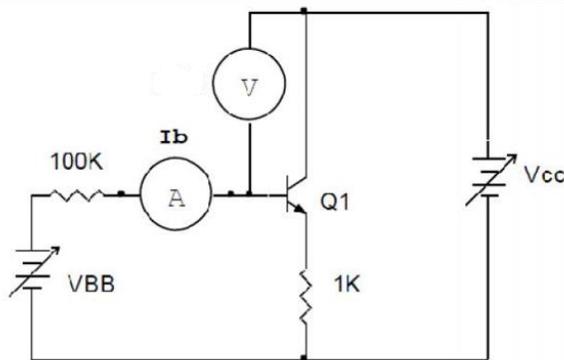


Figure: 8c.2 Circuit diagram to obtain input characteristics

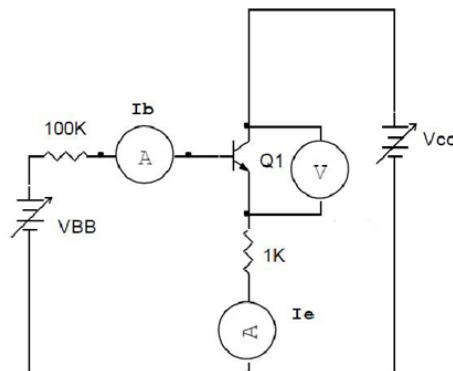


Figure: 8c.3 Circuit diagram to obtain output characteristics

## 5. WAVEFORM

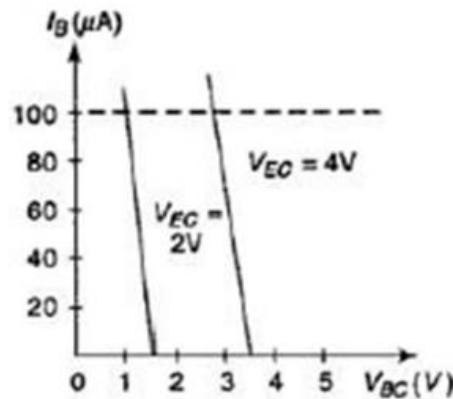


Figure: 8c.4 Input Characteristics

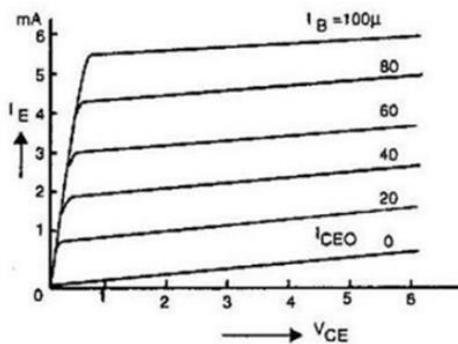


Figure: 8c.5 Output Characteristics

## 6. PROCEDURE:

### Experiment procedure to obtain input characteristics:

1. Connect circuit as shown in the circuit diagram for input characteristics
2. Connect variable power supply 0-30V ( $V_{BB}$ ) at base-emitter circuit and another power supply 0-30V at collector emitter circuit ( $V_{CC}$ ).
3. Keep  $V_{CE}$  fix at 0V (Or do not connect  $V_{CC}$ )
4. Increase  $V_{BB}$  from 0V to 20V, note down readings of base current  $I_B$  and collector to base voltage ( $V_{CB}$ ) in the observation table.
5. Repeat above procedure for  $V_{CE} = +1V$  and  $V_{CE} = +2V$
6. Draw input characteristics curve. Plot  $V_{CB}$  on X axis and  $I_B$  on Y axis.



### **Experiment procedure to obtain output characteristics:**

1. Connect circuit as shown in the circuit diagram for output characteristics
2. Connect variable power supply 0-30V at base circuit and collector circuit.
3. Keep base current fix (Initially 0)
4. Increase  $V_{CC}$  from 0V to 30V, note down readings of emitter current  $I_C$  and collector to emitter voltage  $V_{CE}$  in the observation table.
5. Repeat above procedure for base currents  $I_B = 10\mu A$ ,  $50 \mu A$  and  $100\mu A$ . Increase base current by increasing  $V_{BB}$ .
6. Draw output characteristics curve. Plot  $V_{CE}$  on X axis and  $I_E$  on Y axis for different values of base currents.

### **7. OBSERVATION TABLE:**

Table: 8c.2 Input plot observation table

<b>Sr. No.</b>	<b><math>V_{CE} = 0V</math></b>		<b><math>V_{CE} = +1V</math></b>		<b><math>V_{CE}=+2V</math></b>	
	<b><math>V_{CB}</math></b>	<b><math>I_B</math></b>	<b><math>V_{CB}</math></b>	<b><math>I_B</math></b>	<b><math>V_{CB}</math></b>	<b><math>I_B</math></b>
1						
2						
3						
4						
5						
6						

Table: 8c.3 Output plot observation table

<b>Sr. No.</b>	<b><math>I_B = 0 \mu A</math></b>		<b><math>I_B = 10 \mu A</math></b>		<b><math>I_B = 50 \mu A</math></b>		<b><math>I_B = 100 \mu A</math></b>	
	<b><math>V_{CE}</math></b>	<b><math>I_E</math></b>	<b><math>V_{CE}</math></b>	<b><math>I_E</math></b>	<b><math>V_{CE}</math></b>	<b><math>I_E</math></b>	<b><math>V_{CE}</math></b>	<b><math>I_E</math></b>
1								
2								
3								
4								
5								
6								
7								
8								



## **PRECAUTIONS**

1. Power source should be off while connecting all components.
2. The polarities of all the transistor should be carefully identified.

## **8. RESULT/OBSERVATIONS**

## USING P-SPICE SOFTWARE:

### 1. CIRCUIT DIAGRAM

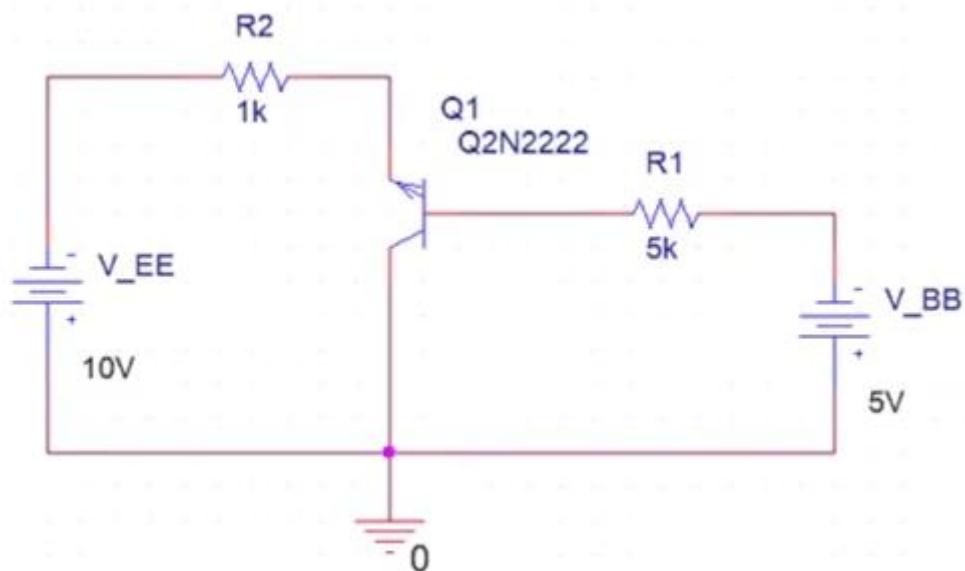


Figure: 8c.6 Common Collector Configuration Circuit Diagram using PSPICE

### 2. RESULT/OBSERVATIONS



### **3. POST LAB QUESTION**

1. Try and observe the input and output plot for CE PNP transistor configuration.
2. Try and observe the input and output plot for CE PNP transistor configuration.
3. Try and observe the input and output plot for CE PNP transistor configuration.

### **4. SAMPLE QUESTIONS**

1. What is transistor and why it is called like that?
2. What is the difference between CC, CB, and CE configuration?
3. Which configuration is best used for voltage amplifier and why?
4. Which configuration is best used for current amplifier and why?
5. Why CC configuration is not used commonly?



## EXPERIMENT NO: 09

Demonstrate all basic gates and realize basic gates using universal gates.

### 1. OBJECTIVES

- i. To realize and verify the truth table of all logic gates.
- ii. To realize and verify the truth table of all basic gates using universal gates.
  - i. To realize and verify the truth table of all logic gates

### 2. THEORY

Basically logic gates are electronic circuits because they are made up of number of electronic devices and components. Logic gates are electronic circuits which perform logical functions on one or more inputs to produce one output. Inputs and outputs of logic gates can occur only in two levels. These two levels are term HIGH and LOW, or TRUE and FALSE, or ON AND off, OR SIMPLY 1 AND 0 There are seven logic gates (3 basic, 2 universal and XOR, XNOR gates). When all the input combinations of a logic gate are written in a series and their corresponding outputs written along them, then this input/ output combination is called “Truth Table”.

#### AND GATE

An AND gate has two or more inputs but only one output. The output assumes the logic 1 state only when each one of its inputs is at logic 1 state. The output assumes logic 0 state even if one of its input is at logic 0 state. AND gate is also called an „all or nothing“ gate.

The logic symbol & truth table of two input AND gate are shown in figure 1.a & 1.b respectively. The symbol for AND operation is “.”

With input variables A & B the Boolean expression for output can be written as;

$$X = A \cdot B$$

Logic symbol



AND Gate

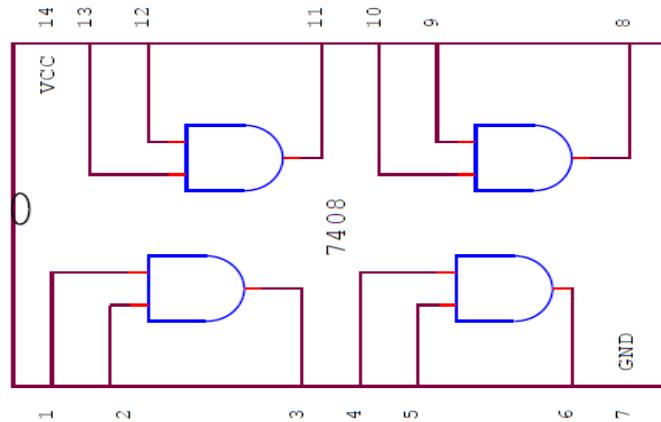
Figure 9.a

Truth Table

A	B	X
0	0	0
0	1	0
1	0	0
1	1	1

Figure 9.b

Pin diagram of IC74LS08:



## OR GATE

Like an AND gate, an OR gate may have two or more inputs but only one output. The output assumes the logic 1 state, even if one of its inputs is in logic 1 state. Its output assumes logic 0 state, only when each one of its inputs is in logic 0 state. OR gate is also called an „any or all“ gate. It can also be called an inclusive OR gate because it includes the condition „both the input can be present“.

The logic symbol & truth table of two input OR gate are shown in figure 1.c & 1.d respectively.

The symbol for OR operation is “+”

With input variables A & B the Boolean expression for output can be written as;

$$X = A + B$$

**Logic symbol**

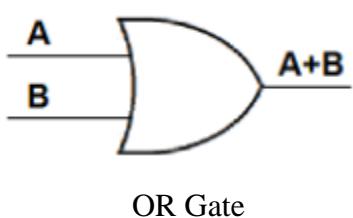


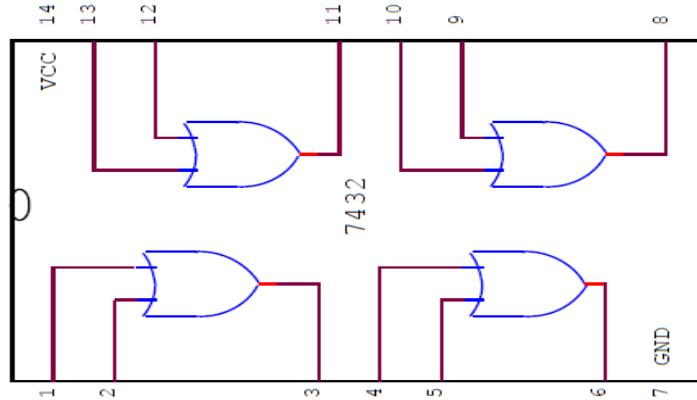
Figure: 9.c

**Truth table**

A	B	X
0	0	0
0	1	1
1	0	1
1	1	1

Figure: 9.d

Pin diagram of IC74LS32:



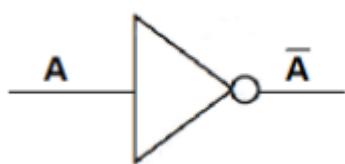
### NOT GATE

A NOT gate is also known an inverter, has only one input and only one output. It is a device whose output is always the complement of its input. That is the output of a not gate assumes the logic 1 state when its input is in logic 0 state and assumes the logic 0 state when its input is in logic 1 state.

The logic symbol & truth table of NOT gate are shown in figure 1.e & 1.f respectively. The symbol for NOT operation is (bar).

With input variable A the Boolean expression for output can be written as;  $A = \bar{A}$   
This is read as “X is equal to a bar”.

Logic symbol



NOT Gate

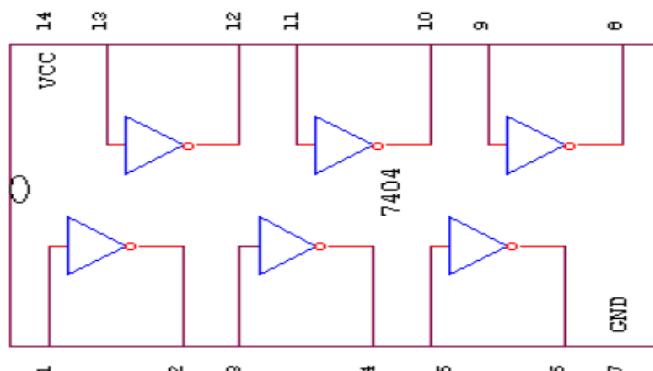
Figure: 9.e

Truth table

A	X
0	1
1	0

Figure: 9.f

Pin diagram for IC74LS04:



## UNIVERSAL GATES

A universal gate is a gate which can implement any Boolean function without need to use any other gate type. The NAND and NOR gates are universal gates. In practice, this is advantageous since NAND and NOR gates are economical and easier to fabricate and are the basic gates used in all IC digital logic families. In fact, an AND gate is typically implemented as a NAND gate followed by an inverter not the other way around!! Likewise, an OR gate is typically implemented as a NOR gate followed by an inverter not the other way around!!

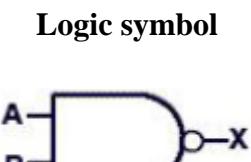
### NAND GATE

NAND gate is universal gate. It can perform all the basic logic function. NAND means NOT AND that is, AND output is NOTed so NAND gate is combination of an AND gate and a NOT gate. The output is logic 0 level, only when each of its inputs assumes a logic 1 level. For any other combination of inputs, the output is logic 1 level. NAND gate is equivalent to a bubbled OR gate.

The logic symbol & truth table of two input NAND gate are shown in figure 1.g & 1.h respectively.

With input variables A & B the Boolean expression for output can be written as:

$$X = \overline{A} \cdot \overline{B}$$



NAND Gate

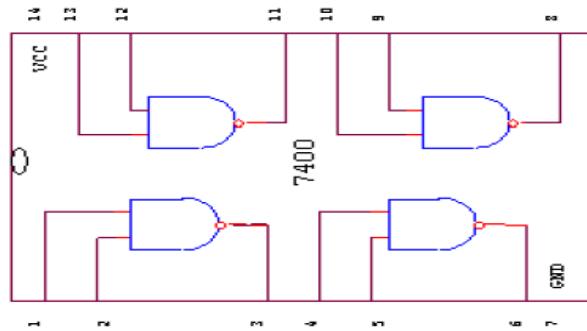
Figure 9.g

**Truth table**

A	B	X
0	0	1
0	1	1
1	0	1
1	1	0

Figure 9.h

Pin diagram for IC74LS00:

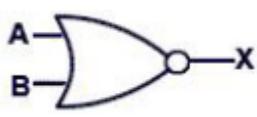


## NOR GATE

NOR gate is universal gate. It can perform all the basic logic function. NOR means NOT OR that is, OR output is NOTed so NOR gate is combination of an OR gate and a NOT gate. The output is logic 1 level, only when each of its inputs assumes a logic 0 level. For any other combination of inputs, the output is logic 0 level. NOR gate is equivalent to a bubbled AND gate. The logic symbol & truth table of two inputs NOR gate are shown in figure 1.i & 1.j respectively. With input variables A & B the Boolean expression for output can be written as;

$$X = \overline{A + B}$$

**Logic symbol**



NOR Gate

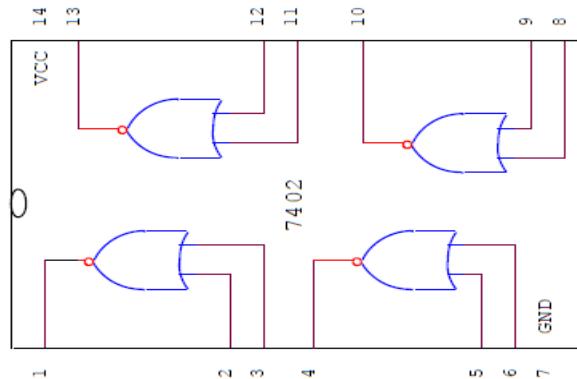
Figure 9.i

**Truth table**

A	B	X
0	0	1
0	1	0
1	0	0
1	1	0

Figure 9.j

Pin diagram for IC74LS02:



### 3. HARDWARE REQUIRED

Sl no	Particulars	Type	Range	Quantity
1	Logic gates trainer kit			
2	logic gates / ICs (AND,OR,NOT,NAND and NOR)			
3	Patch cords			

#### Precautions:

1. Connections must be tight on the bread board.
2. Identify the pins of the IC properly.
3. Take care while removing and inserting the IC on bread board.

### 4. PROCEDURE

1. Connect the trainer kit to ac power supply.
2. Connect the inputs of any one logic gate to the logic sources and its output to the logic indicator.
3. Apply various input combinations and observe output for each one.
4. Verify the truth table for each input/ output combination.
5. Repeat the process for all other logic gates.
6. Switch off the ac power supply.



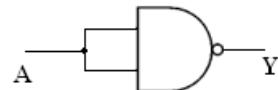
## **5. RESULT/OBSERVATION**

**ii. To realize and verify the truth table of all basic gates using universal gates.**

Circuit Diagram:

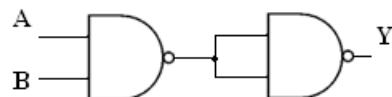
I. Implementation using NAND gate:

(a) NOT Gate:  $Y = A'$



(b) AND Gate:

$$Y = A \cdot B$$



(c) OR Gate:

$$Y = A + B$$

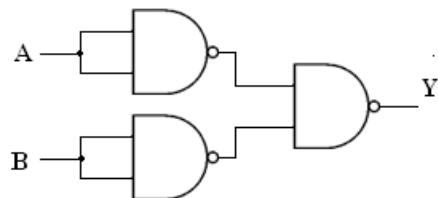


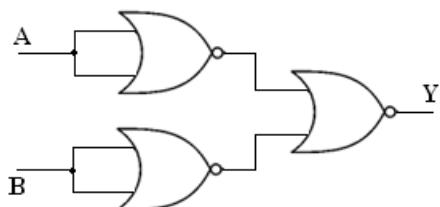
Figure: 9.1 Basic gates using NAND gate

II. Implementation using NOR gate:

(a) NOT gate:  $Y = A'$



(b) AND Gate:  $Y = A \cdot B$



(c) OR Gate:  $Y = A + B$

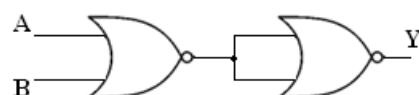


Figure: 9.2 Basic gates using NOR gate



## 1. HARDWARE REQUIRED

Sl no	Particulars	Type	Range	Quantity
1	Logic gates trainer kit			
2	logic gates / ICs (AND,OR,NOT,NAND and NOR)			
3	Patch cords			

## 2. PROCEDURE

1. Connect the trainer kit to ac power supply.
2. Connect the NAND gates/NOR gates for any of the logic functions to be realized.
3. Connect the inputs of first stage to logic sources and output of the last gate to logic indicator.
4. Apply various input combinations and observe output for each one.
5. Verify the truth table for each input/ output combination.
6. Repeat the process for all logic functions.
7. Switch off the ac power supply.

## 3. RESULT/OBSERVATION

## Using Pspice Software:

### 1. OBJECTIVES

To realize and verify the truth table of all logic gates using Pspice software.

### 2. CIRCUIT DIAGRAM

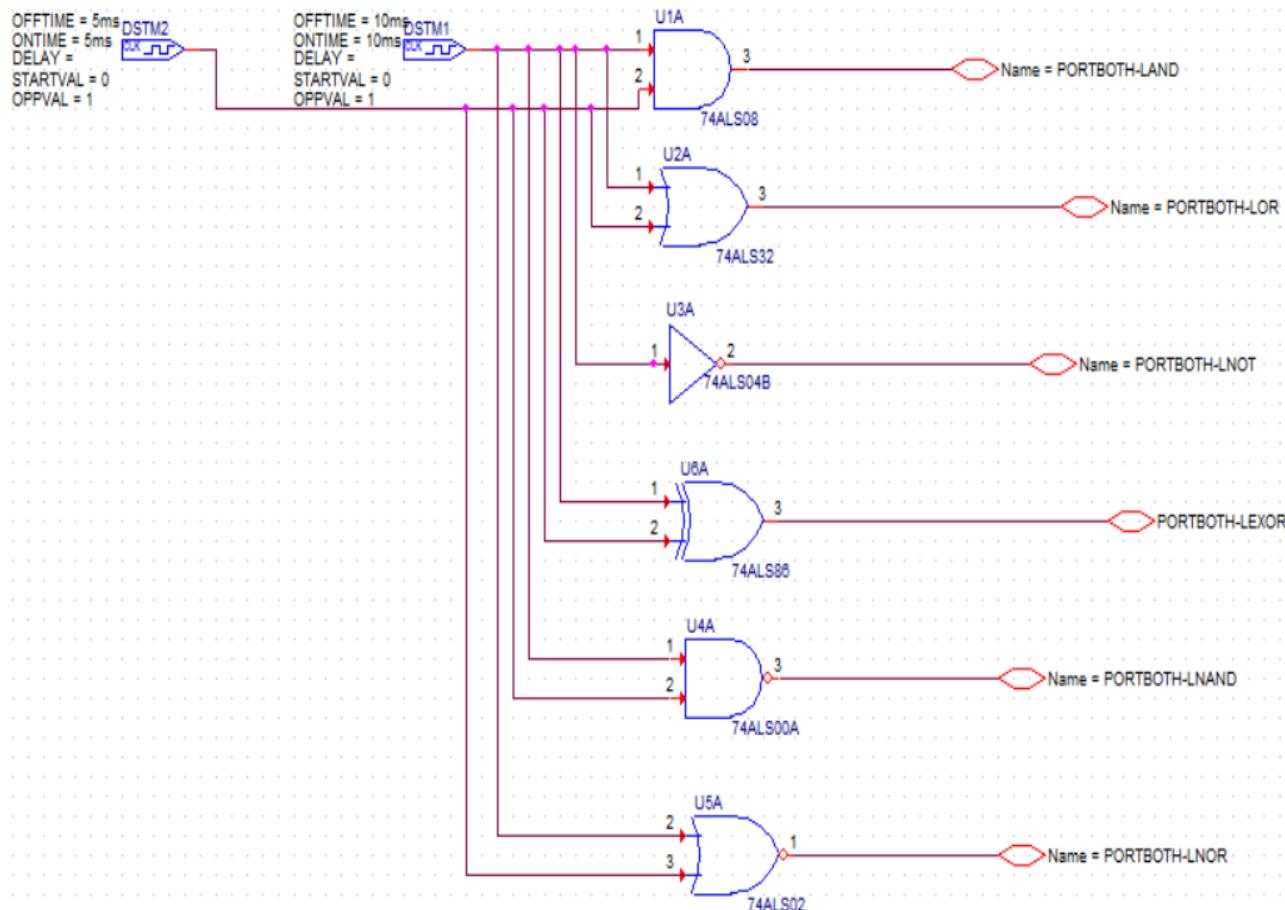


Figure: 9.3 Realization of basic gates using Pspice

### 3. RESULTS/OBSERVATIONS



#### **4. POST LAB QUESTION**

1. Verify the truth table of X-OR and XNOR gates using Pspice software.
2. Verify the truth table of all basic gates (using NAND/NOR only) using Pspice software.

#### **5. SAMPLE QUESTIONS**

1. What is an X-OR gate?
2. What is an X-NOR gate?
3. What type of gate is equivalent to a NAND gate followed by an inverter?
4. Define universal gates? What does the small bubble on the output of the NAND gate logic symbol mean?
5. Explain Boolean expression for digital networks



## EXPERIMENT: 10

Implement and test the functionality of half adder and full adder

**1. OBJECTIVES:** To realise half adder and full adder and test the functionality.

### 2. THEORY:

An Adder is a circuit which performs addition of binary numbers. Producing sum and carry. A half adder is a digital circuit which performs addition of two binary numbers which are one bit each and produces a sum and a carry (one bit each). A full adder is a digital circuit which performs addition of three binary numbers (one bit each), to produce a sum and a carry (one bit each). Full adders are basic block of any adder circuit as they add two numbers along with the carry from the previous addition.

**Half-Adder:** A combinational logic circuit that performs the addition of two data bits, A and B, is called a half-adder. Addition will result in two output bits; one of which is the sum bit, S, and the other is the carry bit, C. The Boolean functions describing the half-adder are:

$$S = A \oplus B \quad C = A \cdot B$$

**Full-Adder:** The half-adder does not take the carry bit from its previous stage into account. This carry bit from its previous stage is called carry-in bit. A combinational logic circuit that adds two data bits, A and B, and a carry-in bit, Cin, is called a full-adder. The Boolean functions describing the full-adder are:

$$S = (x \oplus y) \oplus Cin \quad C = xy + Cin(x \oplus y)$$

### 3. HARDWARE REQUIRED

Sl no	Particulars	Quantity
1	IC 7400, IC 7408, IC 7486, and IC 7432,	

#### 4. CIRCUIT DIAGRAM

a.) Half Adder using basic gates:

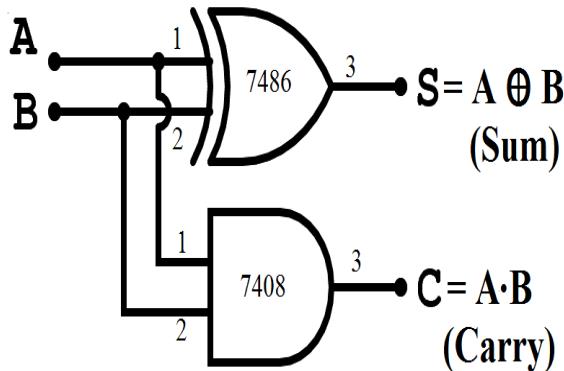


Table: 10.1 Truth table of half adder

Inputs		Outputs	
A	B	Sum(S)	Carry(C)
0	0	0	0
0	1	1	0
1	0	1	0
1	1	0	1

Figure: 10.1 Half Adder using basic gates circuit diagram

b.) Half Adder using only NAND gates:

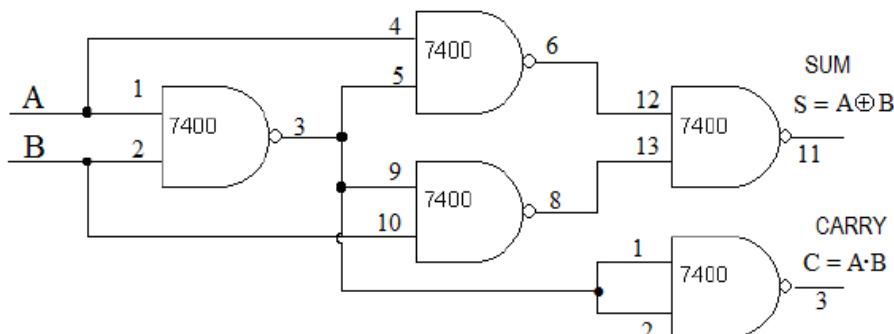


Figure: 10.2 Half Adder using only NAND gates circuit diagram

c.) Full Adder using basic gates:

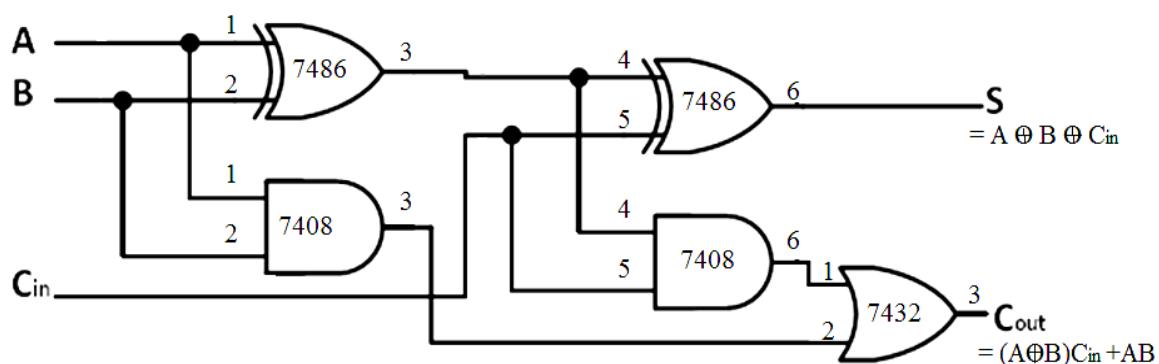


Figure: 10.3 Full Adder using basic gates

### Full Adder using NAND gates only:-

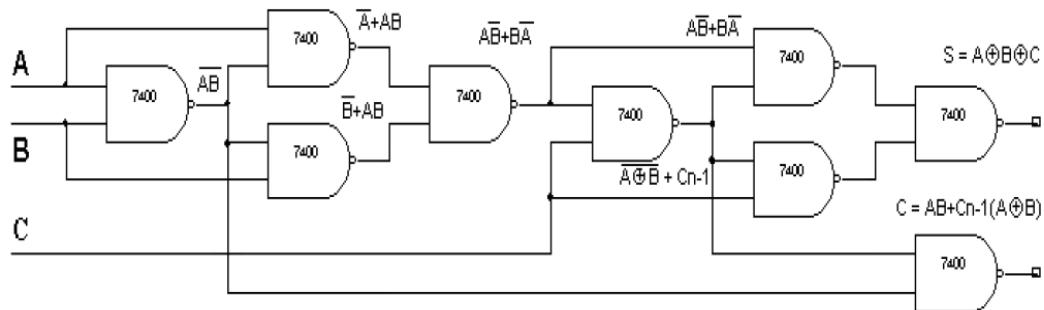


Figure: 10.4 Full Adder using NAND gates only

Table: 10.2 Truth Table of Full Adder

Inputs			Outputs	
A	B	C <sub>in</sub>	Sum (S)	Carry (C <sub>out</sub> )
0	0	0	0	0
0	0	1	1	0
0	1	0	1	0
0	1	1	0	1
1	0	0	1	0
1	0	1	0	1
1	1	0	0	1
1	1	1	1	1

## 5. PROCEDURE

1. Connect the trainer kit to ac power supply.
2. Make the connections as shown in the circuit diagram.
3. Apply various input combinations and observe output for each one.
4. Verify the truth table for each input/ output combination.
5. Switch off the ac power supply.

## 6. RESULTS/OBSERVATION

## Using Pspice Software:

## **1. OBJECTIVES**

To realise half adder and full adder and test the functionality using Pspice Software.

## 2. CIRCUIT DIAGRAM

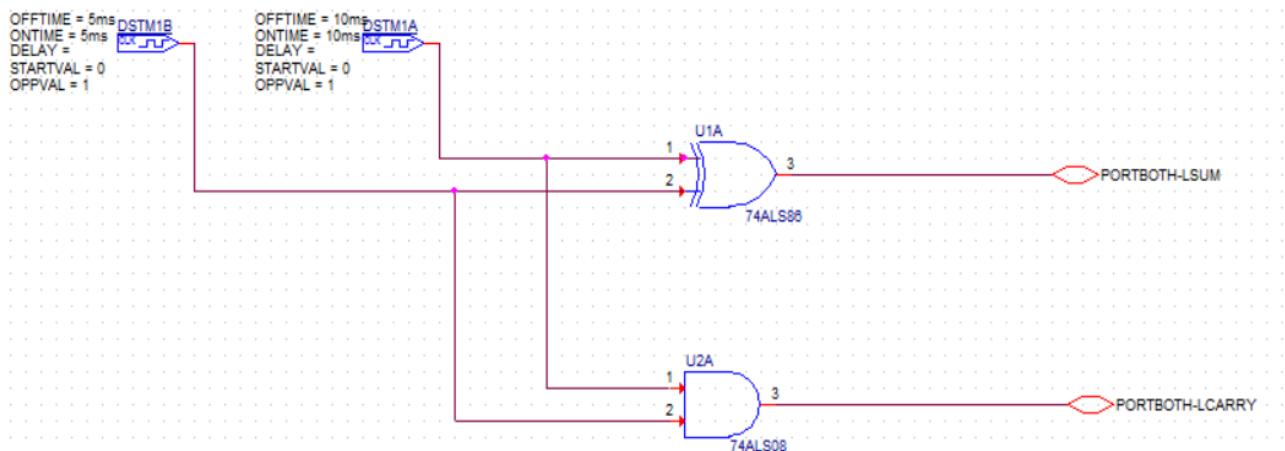


Figure: 10.5 Half adder circuit

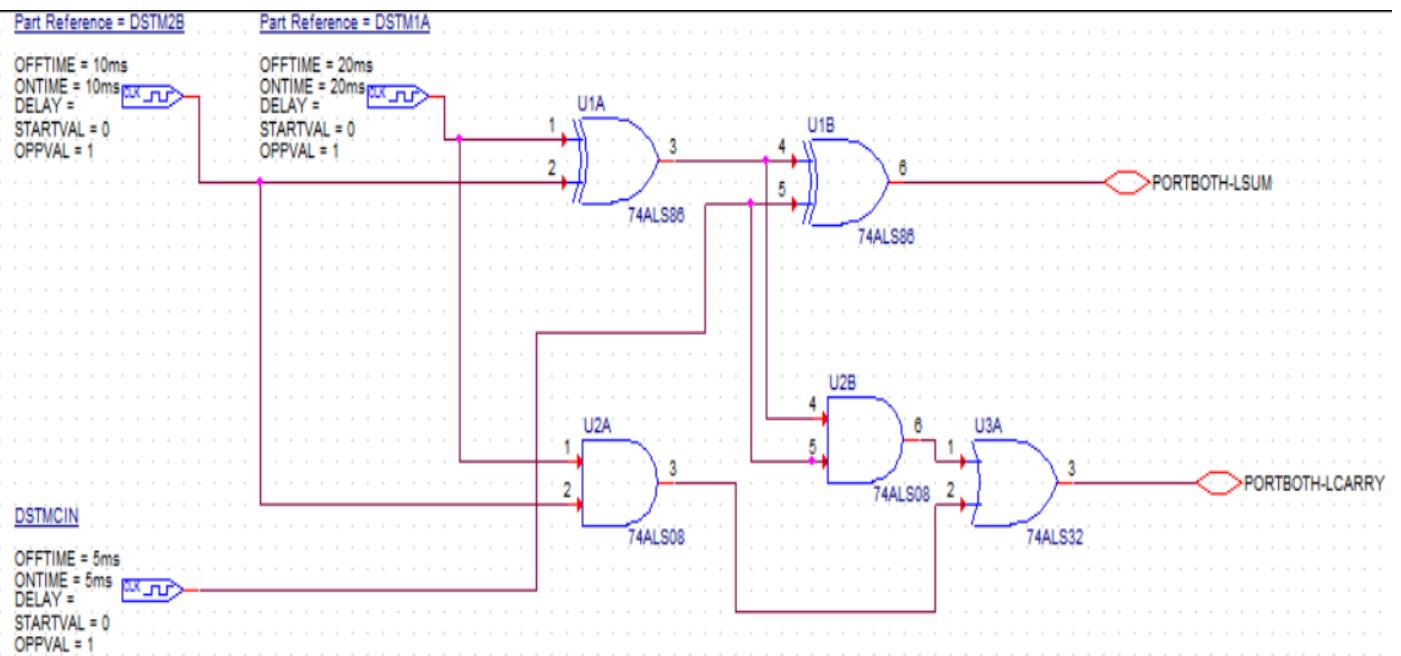


Figure: 10.6 Full Adder Circuit



### **3. RESULTS/OBSERVATIONS**



#### **4. POST LAB QUESTION**

1. Implement the truth table of full adder (with only NAND Gates) using PSPICE software.
2. Implement the truth table of full adder (with only NAND Gates) using PSPICE software.

#### **5. SAMPLE QUESTIONS**

1. What are the applications of adders?
2. What is the major difference between half-adders and full-adders?
3. What is basic drawback of n- bit parallel adder? How is it overcome?
4. Explain how an adder can be converted in to Sub tractor?
5. In Half adder how many types of gates are required?



## Experiment: 11

Implement and test the functionality of SR, JK and D Flip-Flops.

### 1. OBJECTIVES

To study and verify the truth tables for SR JK and D Flip-Flops.

### 2. THEORY

In electronics, a flip-flop or latch is a circuit that has two stable states and can be used to store state information. A flip-flop is a bistable multivibrator. The circuit can be made to change state by signals applied to one or more control inputs and will have one or two outputs. It is the basic storage element in sequential logic. Flip-flops and latches are a fundamental building block of digital electronics systems used in computers, communications, and many other types of systems.

Flip-flops and latches are used as data storage elements. Such data storage can be used for storage of state, and such a circuit is described as sequential logic. When used in a finite-state machine, the output and next state depend not only on its current input, but also on its current state (and hence, previous inputs). It can also be used for counting of pulses, and for synchronizing variably-timed input signals to some reference timing signal.

Flip-flops can be either simple (transparent or opaque) or clocked (synchronous or edge-triggered); the simple ones are commonly called latches. The word latch is mainly used for storage elements, while clocked devices are described as flip-flops. A latch is level-sensitive, whereas a flip-flop is edge-sensitive.

**SR Flip Flop:** This simple flip flop circuit has a set input (S) and a reset input (R). In this circuit when you Set “S” as active the output “Q” would be high and “Q<sup>“”</sup> will be low. Once the outputs are established, the wiring of the circuit is maintained until “S” or “R” go high, or power is turned off. As shown above, it is the simplest and easiest to understand. The two outputs, as shown above, are the inverse of each other. The truth table of SR Flip Flop is highlighted below.

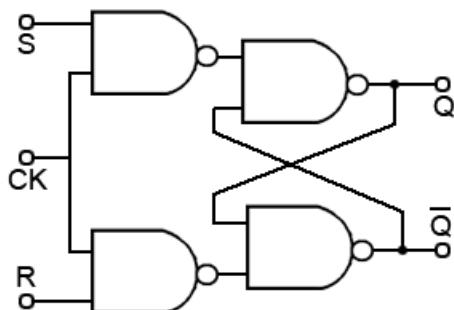
**JK Flip-Flop:** The JK flip flop is basically a gated SR flip-flop with the addition of a clock input circuitry that prevents the illegal or invalid output condition that can occur when both inputs S and R are equal to logic level “1”. Due to this additional clocked input, a JK flip-flop

has four possible input combinations, “logic 1”, “logic 0”, “no change” and “toggle”. The symbol for a JK flip flop is similar to that of an SR Bastable Latch as seen in the previous tutorial except for the addition of a clock input.

**D Flip Flop:** D flip-flop operates with only positive clock transitions or negative clock transitions. Whereas, D latch operates with enable signal. That means, the output of D flip-flop is insensitive to the changes in the input, D except for active transition of the clock signal.

### 3. CIRCUIT DIAGRAM

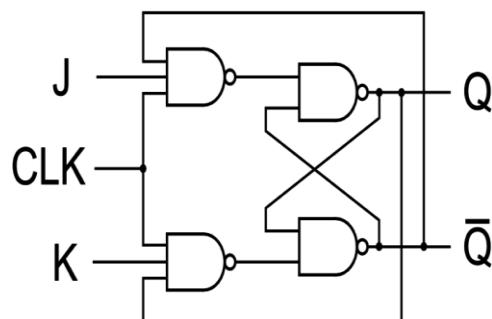
Table: 11.1 Truth Table for SR Flip Flop



S	R	Q	Q'
0	0	0	1
0	1	0	1
1	0	1	0
1	1	Undefined	Undefined

Figure: 11.1 SR Flip Flop

Table: 11.2 Truth Table for JK Flip Flop



CLK	J	K	Q	Q'
1	0	0	No change	
1	0	1	0	1
1	1	0	1	0
1	1	1	Toggle	

Figure: 11.2 J K Flip Flop

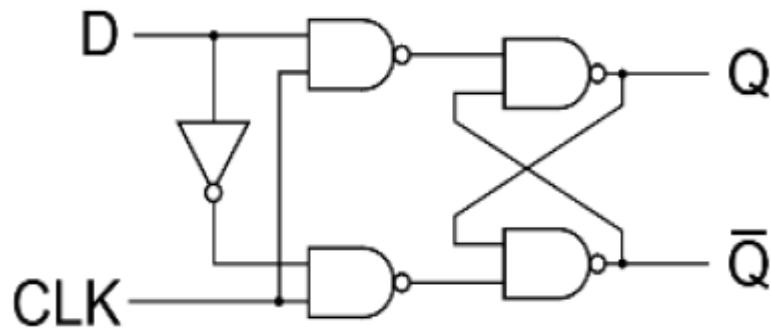


Figure: 11.3 D Flip Flop

Table: 11.3 Truth Table for D Flip Flop

INPUT			OUTPUT	
D	RESET	CLOCK	Q	Q'
0	0	0	0	1
0	0	1	0	1
0	1	0	0	1
0	1	1	0	1
1	0	0	0	1
1	0	1	1	0
1	1	0	0	1
1	1	1	0	1

#### 4. HARDWARE REQUIRED

Sl no	Particulars	Quantity
1	IC 7400, IC 7408, IC 7486, and IC 7432	
2	Trainer kit	
3	Patch cords	

#### 5. PROCEDURE

1. Connect the trainer kit to ac power supply.
2. Make the connections as shown in the circuit diagram.
3. Apply various input combinations and observe output for each one.
4. Verify the truth table for each input/ output combination.
5. Switch off the ac power supply.



## **6. RESULTS/OBSERVATIONS**

## Using Pspice Software:

### 1. OBJECTIVES

To study and verify the truth tables for SR JK and D Flip-Flops.

### 2. CIRCUIT DIAGRAM

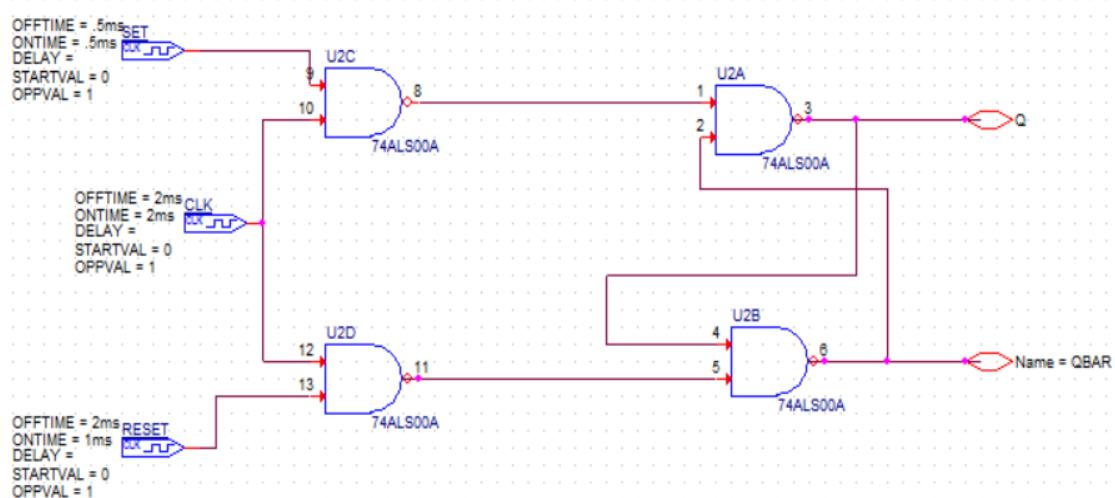


Figure: 11.3 SR Flip Flop

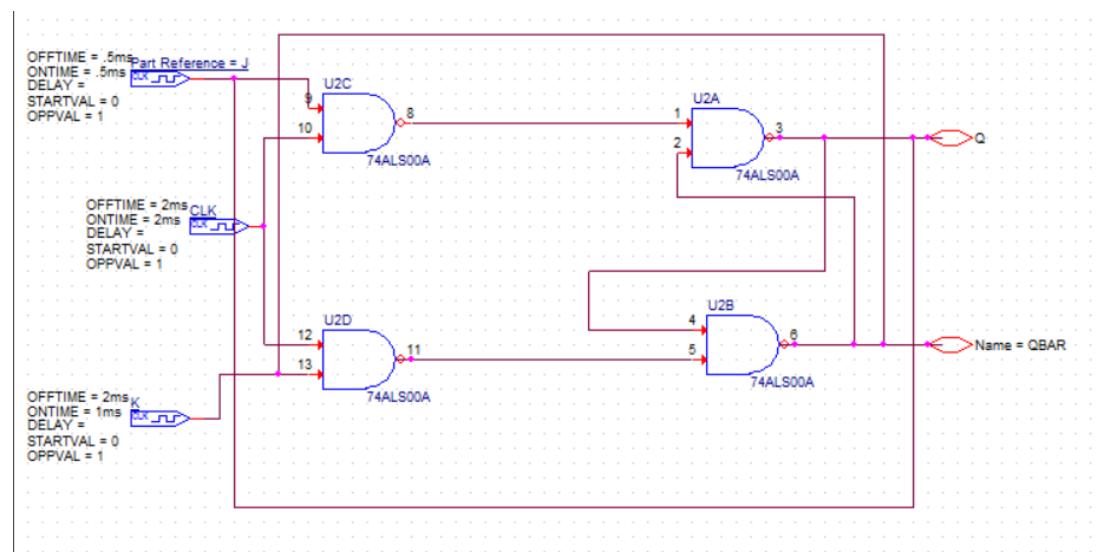


Figure: 11.4 JK Flip Flop

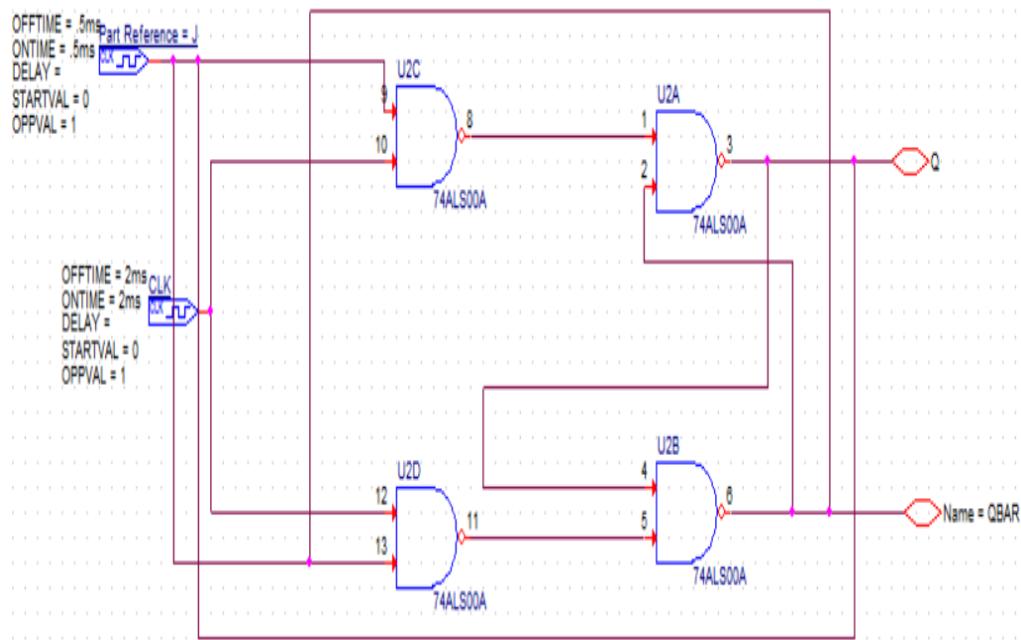


Figure: 11.3 D Flip Flop

### 3. RESULTS/OBSERVATION



#### **4. SAMPLE QUESTIONS**

1. Define Latches?
2. Differentiate between latch and Flip Flops?
3. What is edge and level triggering?
4. What is toggle? Which Flip Flop is used for this phenomenon?
5. What is race around condition and how to eliminate it?
6. Explain the advantage of JK flip-flop over SR flip-flop?



## APPENDIX A

### Introduction to PSpice

In the past, students traditionally verified their laboratory electronic circuits by building them on breadboards and measuring the various nodes with the appropriate laboratory equipment. By using a computer simulation program, such as PSpice, students can obtain results before they come to lab. Hence the laboratory experiments become reinforcement to the subject matter at hand. The use of a computer simulation program allows the student to easily subject the circuit to various stimuli (such as input signals and power supply variations) and to see the results in either a tabular format or plotted out graphically using PSpice's post processor called Probe.

### An Outline of PSpice

PSpice simulates the behaviour of electronic circuits on a digital computer and tries to emulate both the signal generators and measurement equipment such as multimeters, oscilloscopes, curve tracers, and frequency spectrum analyzers.

### Types of Analysis Performed by PSpice

PSpice is a general-purpose circuit simulator capable of performing four main types of analysis: Bias Point, DC Sweep, AC Sweep/Noise, and Time Domain (transient).

#### Bias Point

The **Bias Point** analysis is the starting point for all analysis. In this mode, the simulator calculates the DC operating point of the circuit. Options include calculating the detailed bias points for all nonlinear controlled sources and semiconductors (.OP), performing sensitivity analysis (.SENS), and calculating the small signal DC gain. (.TF)

#### DC Sweep

The **DC Sweep** analysis varies a voltage source over a range of voltages in an assigned number of increments in a linear or logarithmic fashion.

#### AC Sweep/Noise

The **AC Sweep/Noise** analysis varies the operating frequency in a linear or logarithmic manner. It linearizes the circuit around the DC operating point and then calculates the network variables as functions of frequency. The start and stop frequencies as well as the number of points can be assigned. Spice will compute the effective noise voltage spectral density that appears at the *Output Voltage* node because of internal noise sources (.NOISE). In this analysis



the detailed bias points for all non-linear controlled sources and semiconductors (.OP) can also be performed.

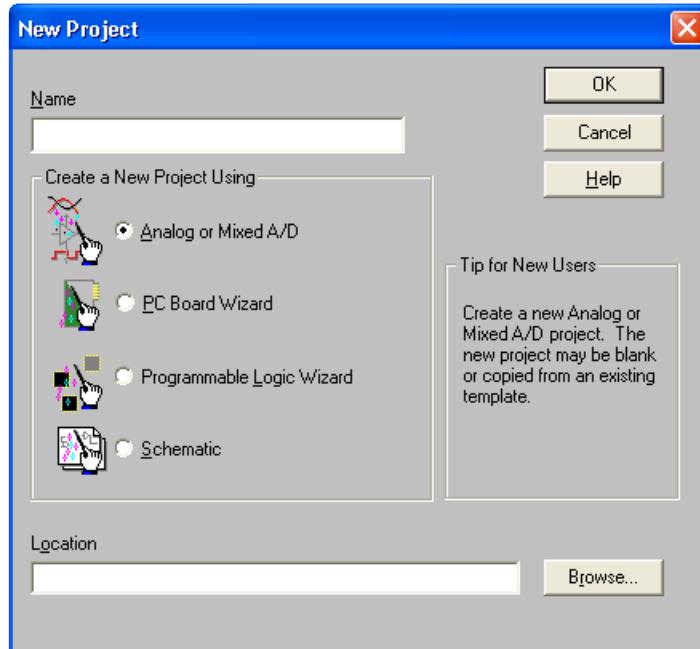
### Time Domain (transient)

The **Time Domain (transient)** analysis is probably the most popular analysis. In this mode, you can plot the various outputs as a function of time. The starting and ending times for the various plots can be input. The accuracy (smoothness) of the output plots can also be controlled by regulating the maximum (time) step size.

## Getting Started with Orcad Capture

You start a new project (program) by going to the **File** menu in the upper left corner, then **New**, and then **Project**.

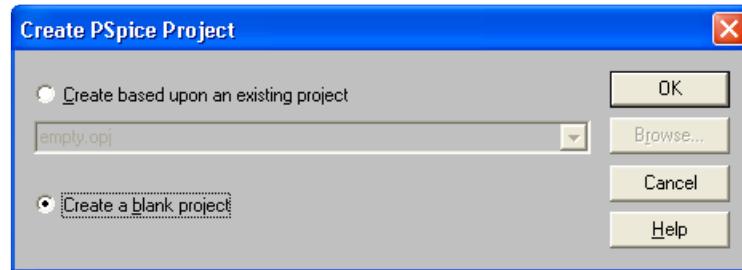
The following screen will appear. Be sure that the **Analog or Mixed A/D** button is activated. (see figure below) Change it if necessary. This is **VERY** important.



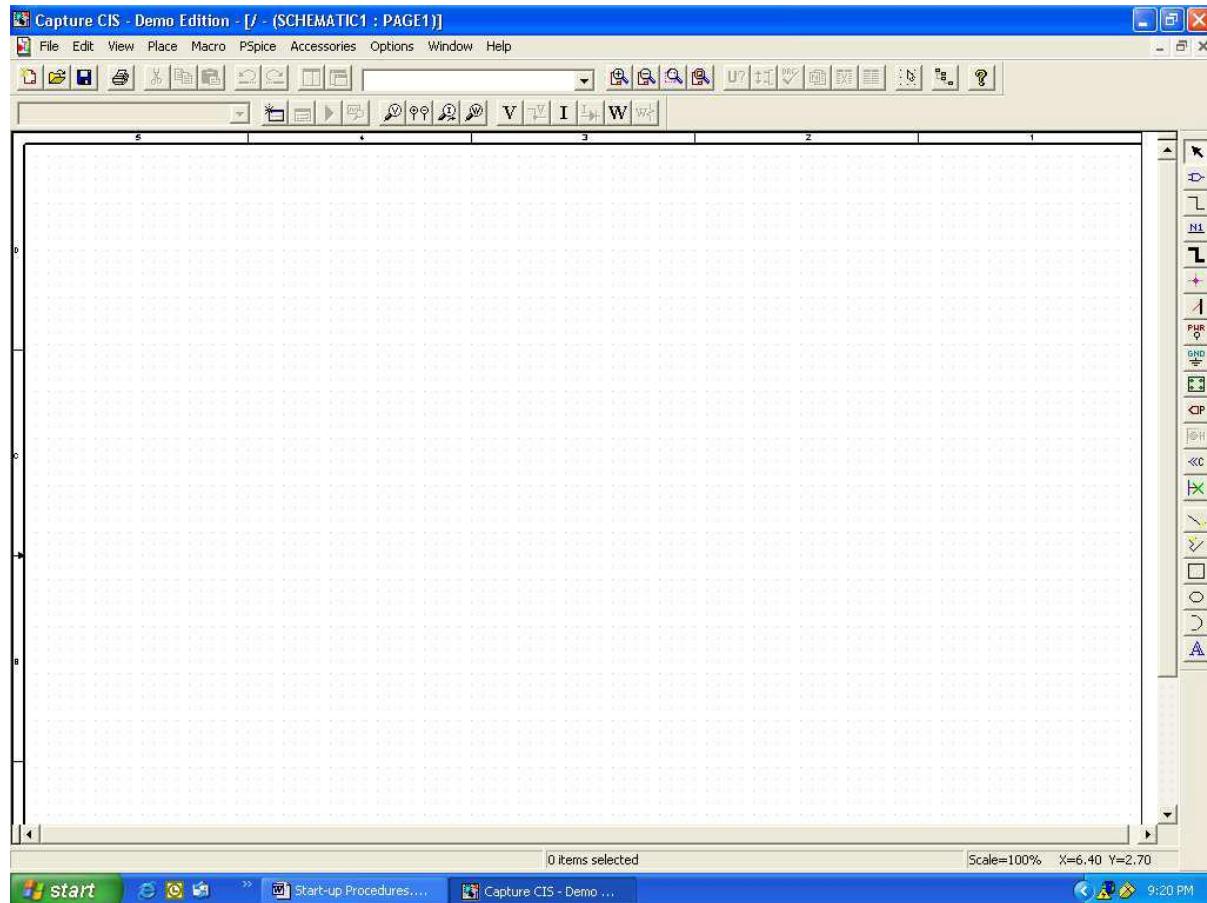
You will need to fill in the top line **Name** with a file name (use Start-Up Example) and then the bottom line **Location** with the path name. This is the directory where you will be storing your “Project”.



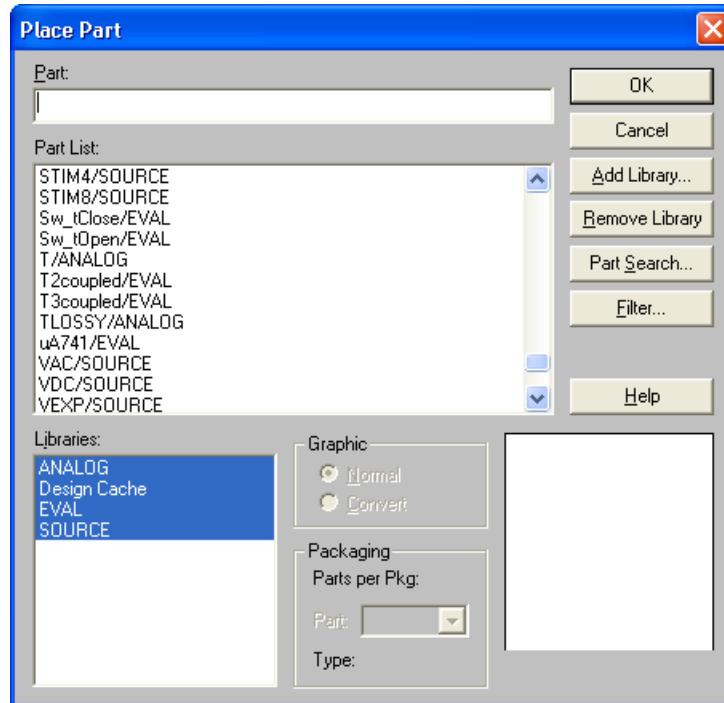
Now the following screen will appear. Since you are starting a new project, change the button settings as shown below. Activate the **Create a blank project** button and left-click OK.



Now you should come up to a blank schematic entry screen.



You can now start adding components and symbols to your schematic, by using the **Place**, **Part...**menu sequence, or the special icon (the uppermost one) on the right hand toolbar. The following screen will appear.



If all of the Libraries shown do not appear on your screen, and they probably won't, go to **Add Library**. There you will find a list of available libraries. For this first example, you will need the analog.olb, the eval.olb, and the source.olb libraries. Add them now.

**Note:** that only parts from the Libraries that are highlighted are shown in the **Part List** window. At this time, highlight all of the libraries. Then start entering your parts. When you have found the required part, either by entering its name in the **Part** window or by highlighting its name in the **Part List** window, left-click OK to place the part onto the schematic. You can continue left-clicking to place multiple copies of the same part or right click to end this selection.

Practice now by entering the schematic shown below. Change the default values and orientations to those shown below.

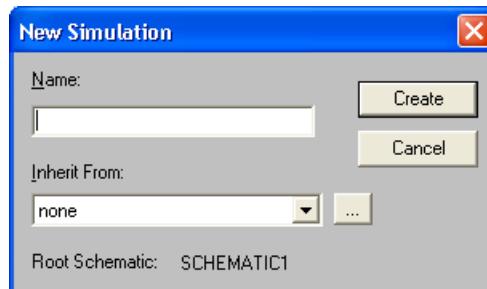


To change a value, or a reference, highlight the appropriate value (left-click) and then double leftclicking. When you have added the resistor (R), and the power supply (Vdc) symbols, enter the ground symbol labeled “0”, which is located in the “..../PSpice/source.olb” library. Recall that every circuit has to have a node “0”. Left-click Apply and close the page.

You can rotate parts by highlighting the part (left-click) and bringing up the part menu (right click), or by pressing the “r” key on the keyboard. See **PSpice Component Layout** description on the next page. Now it’s time to add the connecting wires. Use the **Place, Wire** menu sequence or the icon on the right hand side toolbar. (second one from the top) Connecting wires requires that you drag the “cross hair” over the end of the part and left-click. This “solders” one end of the wire. Drag the wire to another connecting point and left-click again. You have now “soldered” the other end. You are now ready to simulate your circuit.

### DC Bias Simulation

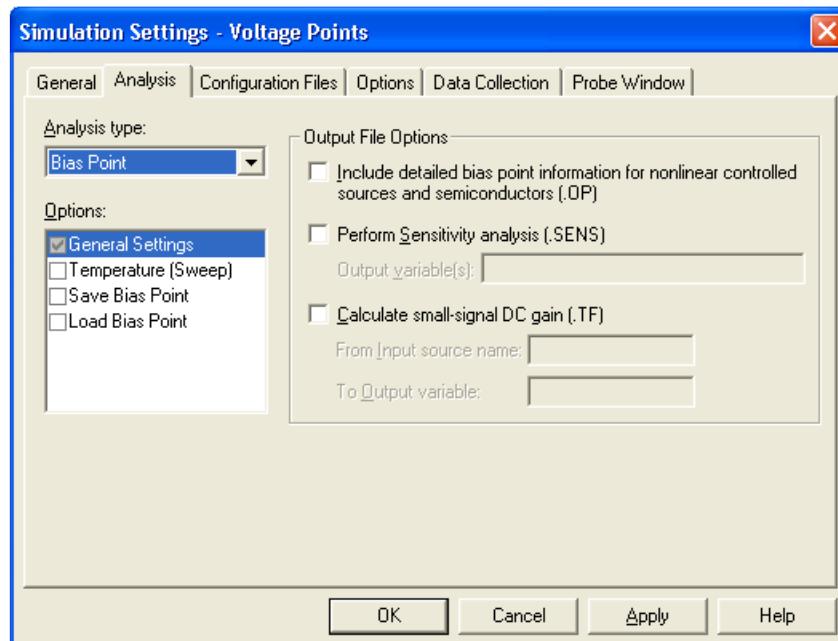
To start the simulation process, open the **PSpice** menu. The first choice available is **New Simulation Profile**. Left-click on it and the following window will appear.



Give the New Simulation a **Name**. For now use “*Voltage Points*”

*Note: You could have done the same thing by left-clicking on the button on the toolbar.*

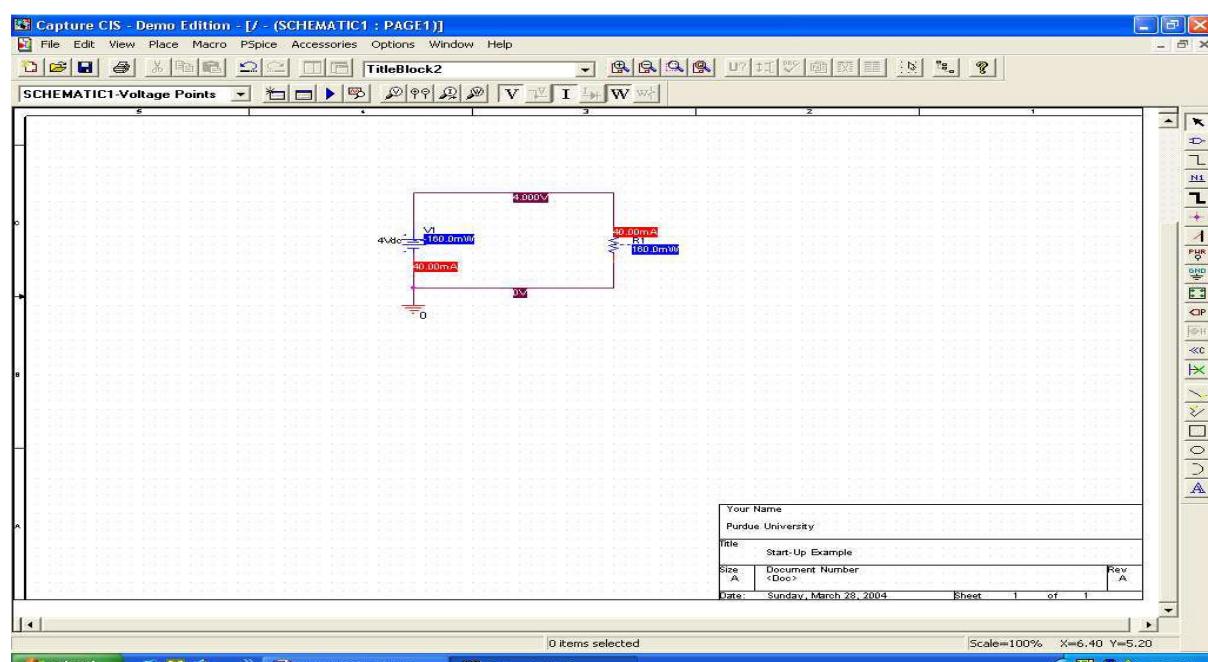
Left-click Create and the next screen will appear.



For a DC analysis, select the **Bias Point** setting in the **Analysis type:** window. Since we do not need that process in this part of our example, go to the **Probe Window** tab, uncheck the box next to the **Display Probe Window** setting and then left-click OK.

Now you are ready to **Run** a simulation. Go to the **PSpice** tab and select **Run**.

The simulation window will appear. When the simulation has completed, close this window and the schematic will reappear. When the V, I, and W tool buttons are activated, the results of the voltage, the current, and the power dissipated in that component will be shown. The tool buttons alongside the V, I, and W buttons allow you to alternately toggle a highlighted value OFF and ON.



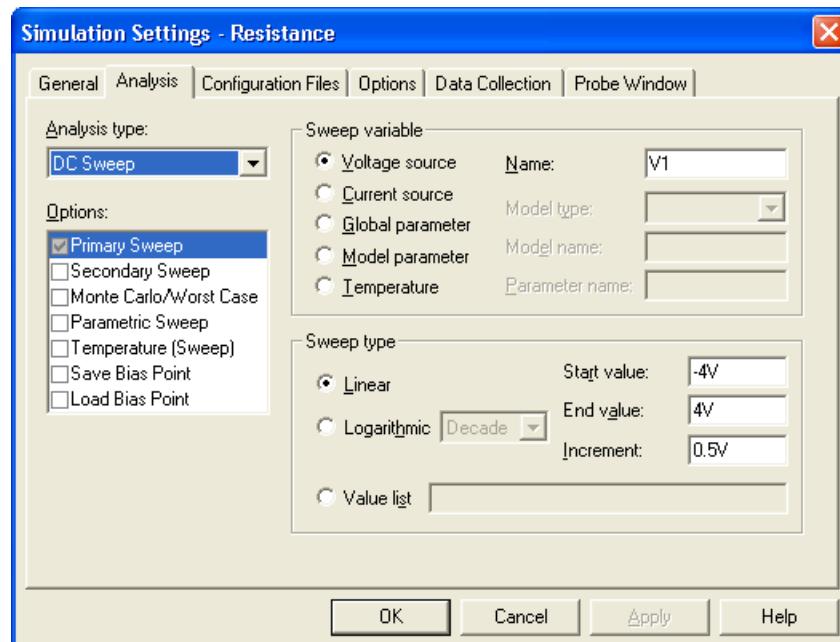
## Linear Resistance

In this segment you will plot out the current vs. voltage characteristics of a linear resistor. The resistor network is repeated below.



As before, go to the **PSpice** tab, or the button, and create a **New Simulation Profile**. This time give it the **Name “Resistance”**

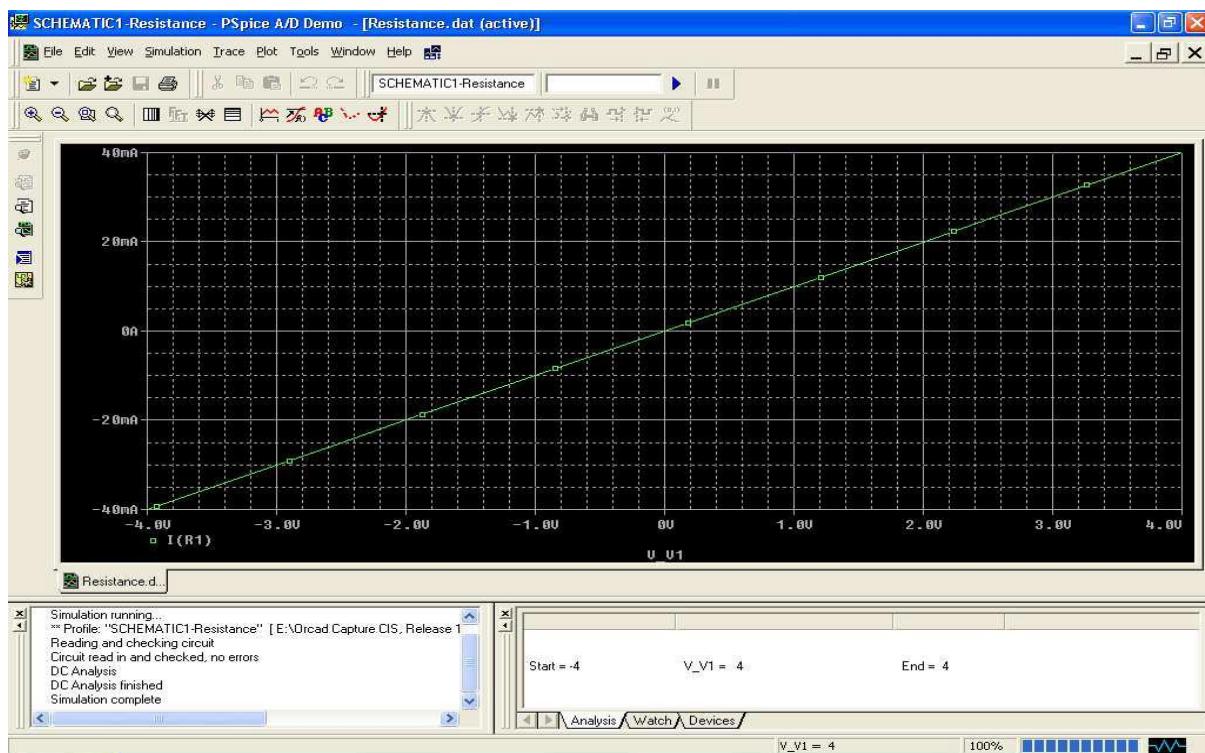
The simulation profile setup for a DC Sweep is shown below.



Click OK to apply the values and close this window. Now re-run PSpice. When the circuit is finished simulating, the Probe window will appear. The X-AXIS will already be set up with a scale of -4V to +4V in increments of 1V. Left-click on the **Trace** menu or on the toolbar menu. This brings up the **Add Traces** window. Highlight and left-click I(R1) to add it to the **Trace**



**Expression** line at the bottom of the screen. Left-clicking OK brings up the plot of the resistor current vs. the resistor voltage.



If your resistor “curve” is negative that means your resistor is “backwards” in the circuit. Disconnect the resistor, rotate it  $180^\circ$  and reconnect the wires. Re-simulate the circuit to get an image similar to that shown above.



## Appendix: B

### Circuit Simulation Software: Logisim

Logisim Evolution is a free software which can be downloaded from the source:

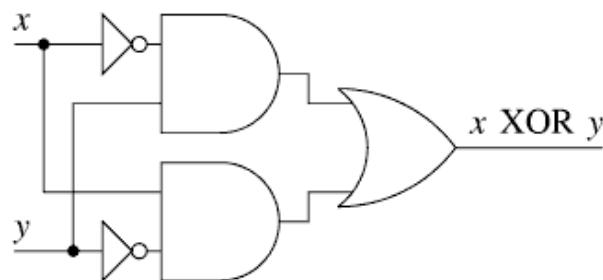
<https://sourceforge.net/projects/logisimevolution/>

Logisim allows you to design and simulate digital circuits. It is intended as an educational tool, to help you learn how circuits work.

To practice using Logisim, let's build a XOR circuit - that is, a circuit that takes two inputs (which we'll call  $x$  and  $y$ ) and outputs 0 if the inputs are the same and 1 if they are different. The following truth table illustrates.

$x$	$y$	$x \text{ XOR } y$
0	0	0
0	1	1
1	0	1
1	1	0

We might design such a circuit on paper.

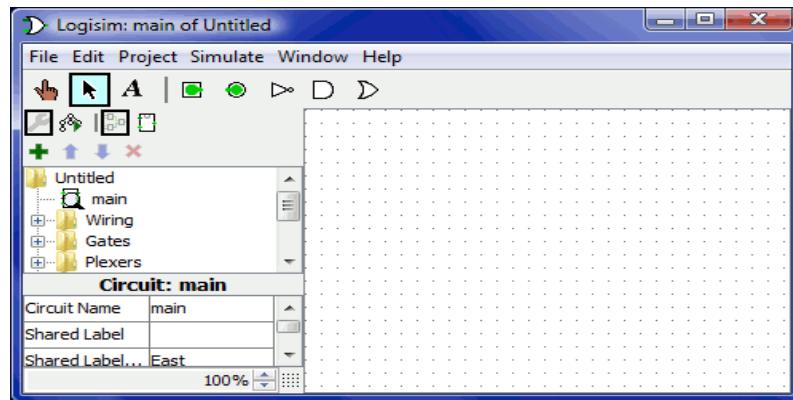


But just because it's on paper doesn't mean it's right. To verify our work, we'll draw it in Logisim and test it.

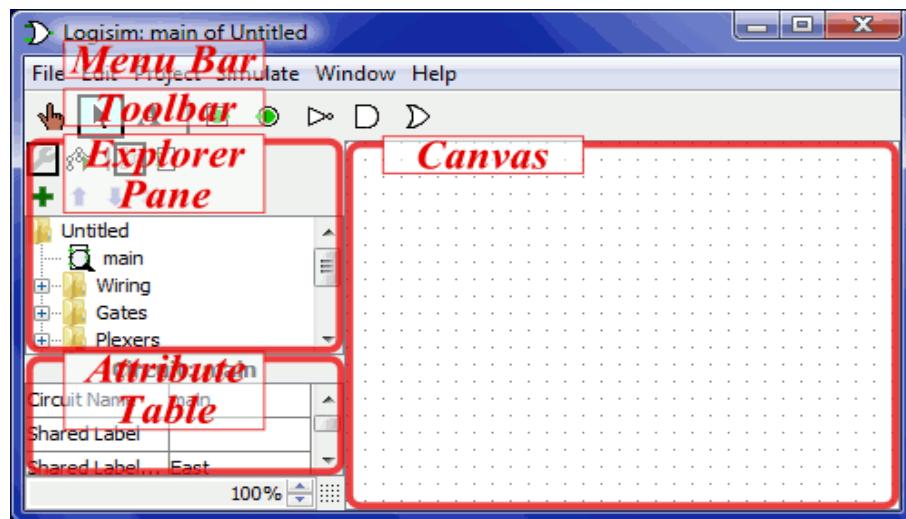
### Orienting yourself



When you start Logisim, you'll see a window similar to the following.



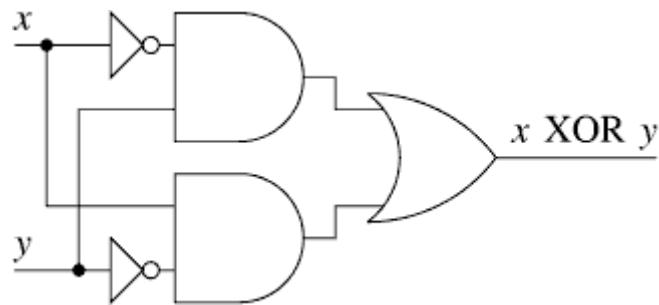
All Logisim is divided into three parts, called the *explorer pane*, the *attribute table*, and the *canvas*. Above these parts are the *menu bar* and the *toolbar*.



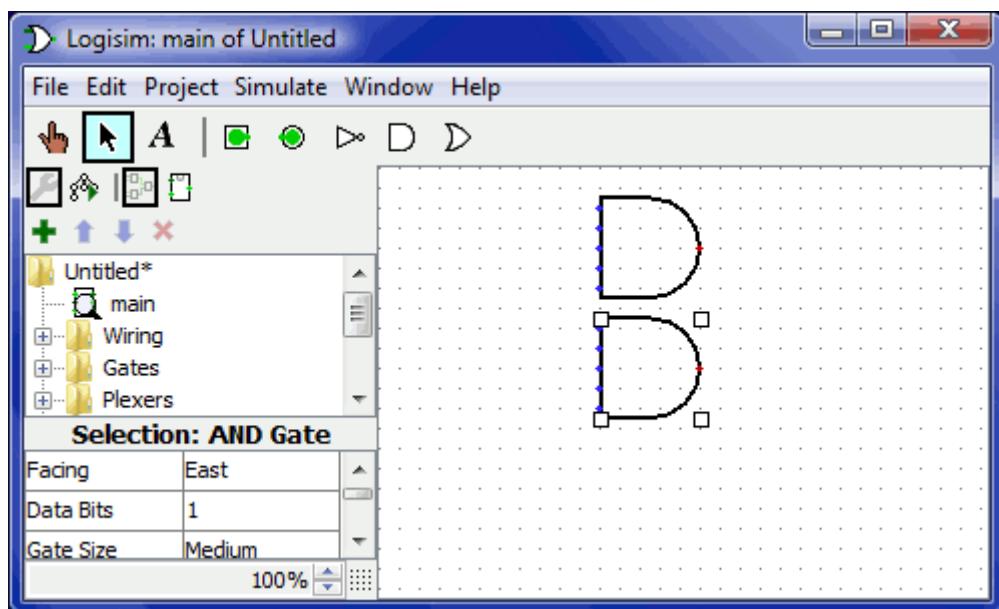
The canvas is where you'll draw your circuit; and the toolbar contains the tools that you'll use to accomplish this. Also, the menu bar is self-explanatory

## Step 1: Adding gates

Recall that we're trying to build the following circuit in Logisim.

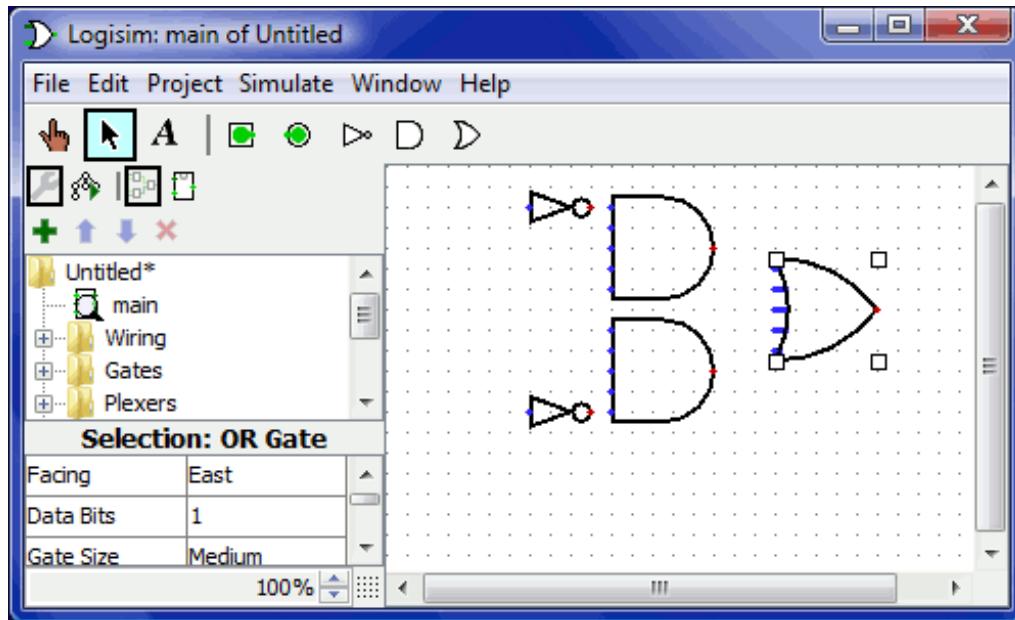


Click on the AND tool in the toolbar ( $\square$ , the next-to-last tool listed). Then click in the editing area where you want the first AND gate to go. Be sure to leave plenty of room for stuff on the left. Then click the AND tool again and place the second AND gate below it.

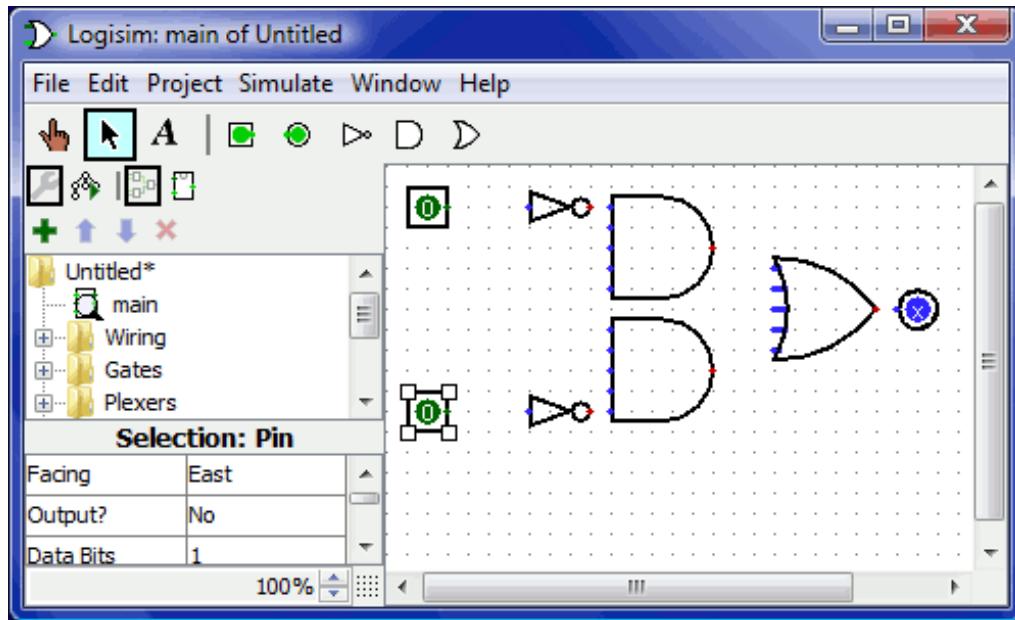


Notice the five dots on the left side of the AND gate. These are spots where wires can be attached. It happens that we'll just use two of them for our XOR circuit; but for other circuits, you may find that having more than two wires going to an AND gate is useful.

Now add the other gates. First click on the OR tool ( $\triangleright$ ); then click where you want it. And place the two NOT gates into the canvas using the NOT tool ( $\blacktriangleright\!\!\!$ ).



Now we want to add the two inputs  $x$  and  $y$  into the diagram. Select the Input tool (), and place the pins down. You should also place an output pin next to the OR gate's output using the Output tool (). (Again, I'm leaving a bit of space between the OR gate and the output pin, but you might choose to place them right next to each other.)



If you decide you don't like where you placed something, then you can select it using the Edit tool () and drag it to the desired spot. Or you can delete it altogether by selecting Delete from the Edit menu or pressing the Delete key.



As you place each component of the circuit, you'll notice that as soon as the component is placed, Logisim reverts to the Edit tool so that you can move the recently-placed component or (as we'll see soon) connect the component to others by creating wires. If you want to add a copy of the recently placed component, a shortcut is to press Control-D to duplicate the selection.

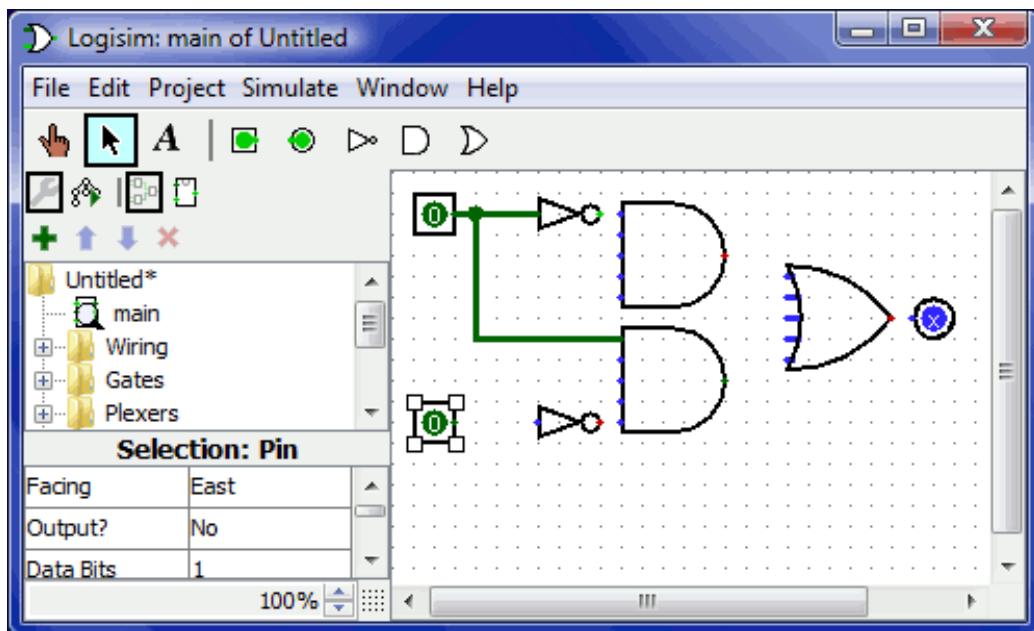
## Step 2: Adding wires

After you have all the components blocked out on the canvas, you're ready to start adding wires.

Select the Edit Tool (). When the cursor is over a point that receives a wire, a small green circle will be drawn around it. Press the mouse button there and drag as far as you want the wire to go.

Logisim is rather intelligent when adding wires: Whenever a wire ends at another wire, Logisim automatically connects them. You can also "extend" or "shorten" a wire by dragging one of its endpoints using the edit tool.

Wires in Logisim must be horizontal or vertical. To connect the upper input to the NOT gate and the AND gate, then, I added three different wires.



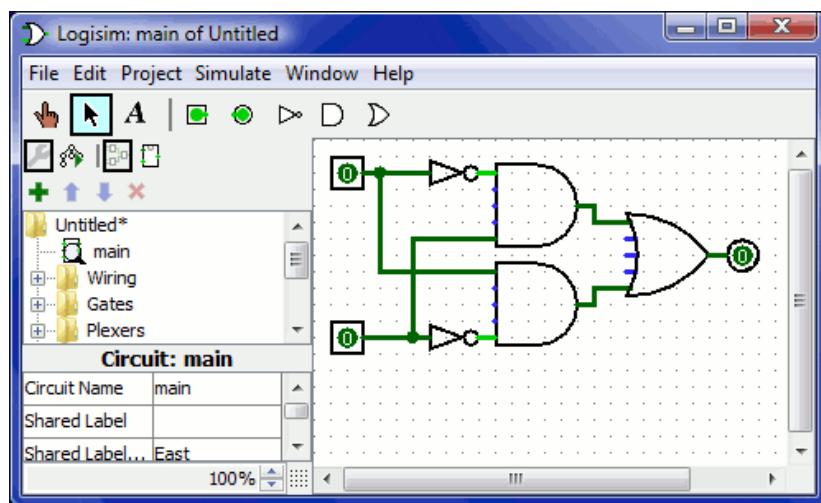
Logisim automatically connects wires to the gates and to each other. This includes automatically drawing the circle at a *T* intersection as above, indicating that the wires are connected.



As you draw wires, you may see some blue or gray wires. Blue in Logisim indicates that the value at that point is "unknown," and gray indicates that the wire is not connected to anything. This is not a big deal as you're in the process of building a circuit. But by the time you finish it, none of your wires should be blue or gray. (The unconnected legs of the OR gate will still be blue: That's fine.)

If you do have a blue or a gray wire after you think everything ought to be connected, then something is going wrong. It's important that you connect wires to the right places. Logisim draws little dots on the components to indicate where wires ought to connect. As you proceed, you'll see the dots turn from blue to light or dark green.

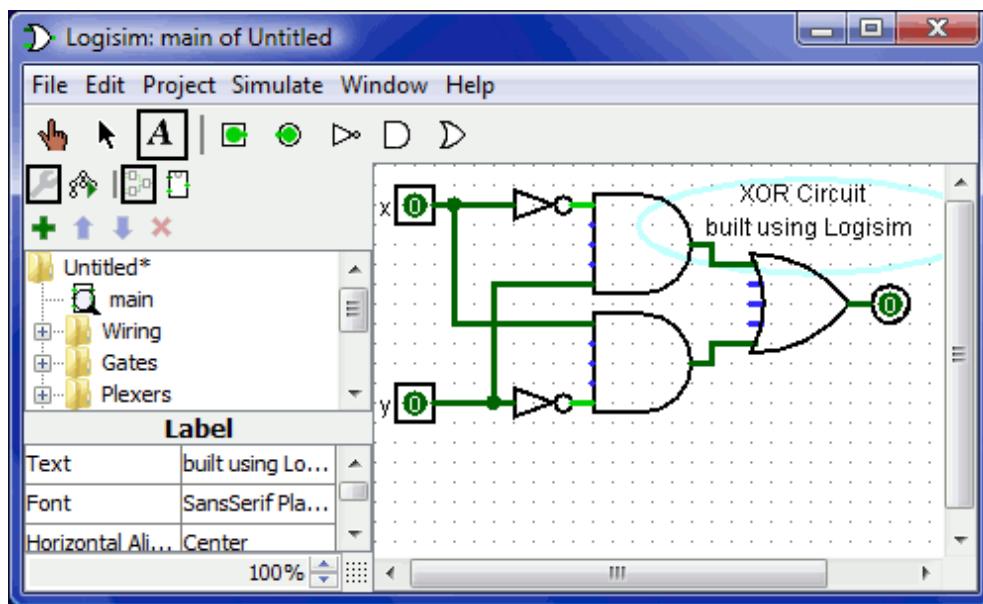
Once you have all the wires connected, all of the wires you inserted will themselves be light or dark green.



### Step 3: Adding text

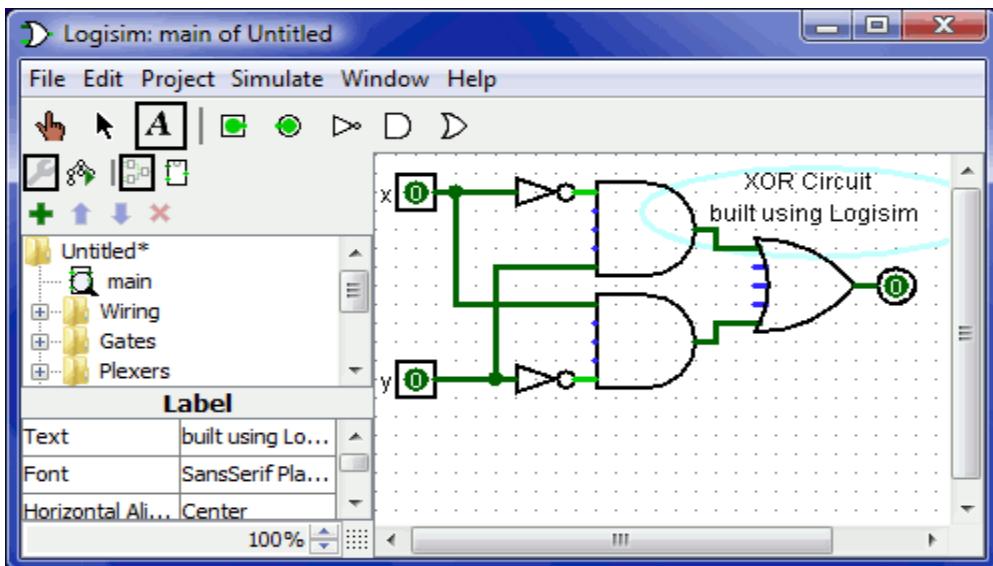
Adding text to the circuit isn't necessary to make it work; but if you want to show your circuit to somebody (like a teacher), then some labels help to communicate the purpose of the different pieces of your circuit.

Select the text tool (**A**). You can click on an input pin and start typing to give it a label. (It's better to click directly on the input pin than to click where you want the text to go, because then the label will move with the pin.) You can do the same for the output pin. Or you could just click any old place and start typing to put a label anywhere else.



## Step 4: Testing your circuit

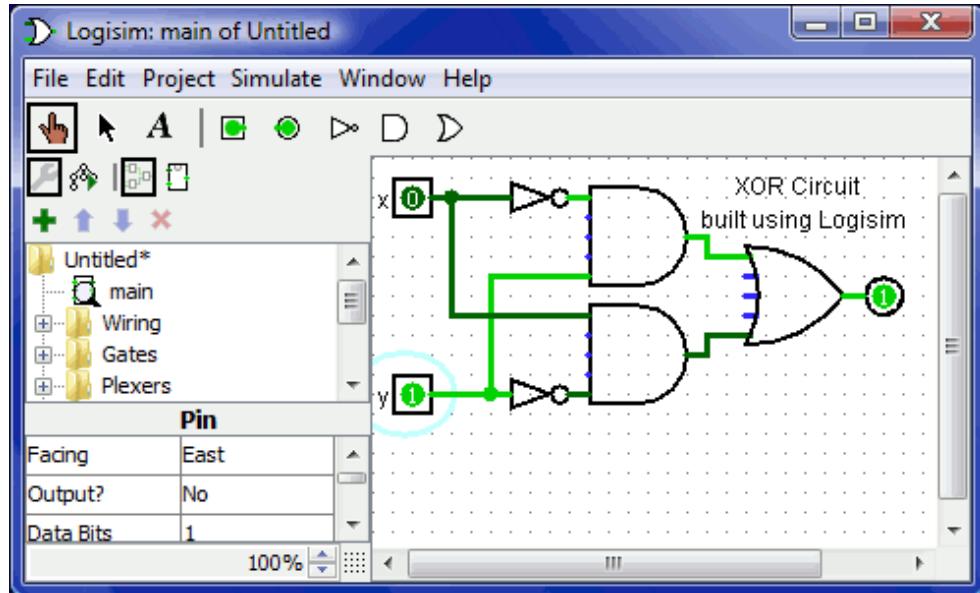
The final step is to test our circuit to ensure that it really does what we intended. Logisim is already simulating the circuit. Let's look again at where we were.



Note that the input pins both contain 0s; and so does the output pin. This already tells us that the circuit already computes a 0 when both inputs are 0.



Now try another combination of inputs. Select the poke tool ( ) and start poking the inputs by clicking on them. Each time you poke an input, its value will toggle. For example, we might first poke the bottom input.



When you change the input value, Logisim will show you what values travel down the wires by drawing them light green to indicate a 1 value or dark green (almost black) to indicate a 0 value. You can also see that the output value has changed to 1.

So far, we have tested the first two rows of our truth table, and the outputs (0 and 1) match the desired outputs.

x	y		x XOR y
0	0		0
0	1		1
1	0		1
1	1		0

By poking the switches through different combinations, we can verify the other two rows. If they all match, then we're done: The circuit works!

To archive your completed work, you might want to save or print your circuit. The File menu allows this, and of course it also allows you to exit Logisim.



Now that you are finished with tutorial, you can experiment with Logisim by building your own circuits. Logisim is a powerful program, allowing you to build up and test huge circuits; this step-by-step process just scratches the surface.