DEPARTMENT OF

ELECTRONICS AND COMMUNICATION

ENGINEERING

COLLEGE OF TECHNOLGY AND ENGINEERING

MAHARANA PRATAP UNIVERSITY OF AGRICULTURE & TECHNOLGY, UDAIPUR

LABORATORY E-MANUAL
Third Year B.Tech. VI SEM (ECE)
APRIL 2020

DEPARTMENT OF ELECTRONICS AND COMMUNICATION ENGINEERING

CTAE, MPUAT, UDAIPUR

Prepared By

- **Dr. Navneet Agarwal**
- **Mr Vishwapriya**
- **Dr. P C Bapna**
- **Mr. Himanshu Mohan**

COLLEGE OF TECHNOLGY AND ENGINEERING

MAHARANA PRATAP UNIVERSITY OF AGRICULTURE & TECHNOLGY, UDAIPUR

DEPARTMENT OF ELECTRONICS AND COMMUNICATION ENGINEERING

LAB MANUAL

EC321- MICROWAVE ENGINEERING-II

PREPARED BY

DR. NAVNEET AGRAWAL

COLLEGE OF TECHNOLOGY AND ENGINEERING MPUAT, UDAIPUR

ELECTRONICS AND COMMUNICATION ENGINEERING

E-Manuals for Micowave Engineering -2(EC321)

LIST OF EXPERIMENTS

1 Development of an experimental setup for measurement of VSWR using double minima methods and cross verification of the experimental result.
2 Development of an experimental setup to measure unknown impedance of rectangular waveguide and verify the results using smith chart.
3 Design an experiment to develop I-V characteristics of microwave Gunn diode   and to measure output power and frequency as a function of bias voltage using microwave test bench.
4 Development of an experimental setup to carry out Square wave modulation through PIN diode.
5 Study of network analyzer.
6 Development of test setup for measurement of Scattering parameters of various microwave components.
7 Study of microwave development and software simulation tools like ADS, IE3D, FEKO etc.
8 Designing of MMIC structure (MOSFET/Frequency doubler/mixer) using Advance Designing Software (ADS).
9 Design of micro strip filters (Low Pass/ High Pass / Band Pass) using Advance Designing Software (ADS)/ IE3D Software.
10 Development of Polar Plot and Cartesian plot of Azimuth and Elevation planes on Log/Linear scales for radiation pattern of micro strip Antenna.
11 To measure coupling factor Isolation & Directivity of Hybrid ring rat race coupler.
   Additional experiments:-

a) To study the microwave components in detail.
b) To develop an experimental setup for power characterstics of wilkinson power divider.

EXPERIMENT:-1

Development of an experimental setup for measurement of VSWR using double minima methods and cross verification of the experimental result.

II.) EQUIPMENT ANDCOMPONENTS:

1. **APPARATUS**

1	Klystron power supply SKPS –610
2	Klystron tube2k25
3	Klystron mountxm-251
4	Isolator XI-621
5	Frequency meterXF-710
6	Variable attenuator XA-520
7	Detector mountXD-451
8	Waveguide standsX4-535
9	VSWR meter SW-215
10.	Movable shortXT-481
11.	Matched terminationxl-400
12.	Slotted sectionXS-651
13.	Tunable probeXP-655

(ii) DESCRIPTION OF EQUIPMENT:

1. Klystron power supply SKPS –610

The model Klystron Power Supply SKPS-610 is general purpose laboratory power supply which is specially designed to use for reflex klystron tubes of S to X band frequency range. It is reliable power source with very high regulation and low ripple contents.

The klystron power supply SKPS-610 has built in modulation facilities of amplitude and frequency modulation. Amplitude modulation can be applied with 0-110V (p.p.) Square wave and with frequency of 500 Hz to 2.5 KHz. This amplitude modulation is generally used along with VSWR measurements in slotted line technique. Frequency modulation is used for direct study of klystron modes on the Oscilloscope. Pure carrier wave operation and in external modulation facilities is also provided with the instrument for generalized use.

The klystron power supply also provides all the other D.C. Voltages required for operation of reflex klystron tube such as beam, heater and reflector voltage. The ranges of all these voltages are given in the specification data sheet.

1. Klystron tube2k25

The klystron tube 2k25 is a single cavity variable frequency microwave generator of low power and low efficiency. It consists of an electron gun, a filament surrounded by cathode and a focusing electrode at cathode potential. The electrons emitted by the cathode travel towards the reflector through an anode kept at higher potential compared to the cathode. When they approach the anode, the electrons form bunches and the bunches ultimately return towards the anode cavity after traveling a small distance towards the reflector. The power is taken from the anode reentrantcavity.

2. *IsolatorXI-621*

It is a two port device which provides very small amount of attenuation for transmission from port 1 to port 2 and provides maximum attenuation for transmission from port 2 to port 1. It is very much desirable when we want to match a source with variable load. It allows power flow only from the generator towards the load and suppresses any reflectedpower.

3. *Frequency meterXF-710*

It is also called wave meter. Usual construction of it consist a cylindrical cavity mounted on a shaft. By rotating the shaft the volume in the cavity is changed and it becomes resonant and gives minimum impedance at the resonant frequency. The scale calibrated and the resonant frequency can be directly read from the scale after observing a dip in the output meter.

4. *Variable attenuatorXA-520*

This is a wave guide piece having a groove on the lateral side. By rotating the screw the depth of penetration of resistive pad changes, there by introducing some attenuation. Attenuators are commonly used for measuring power game or loss in dB for providing isolation between instruments for reducing the power input to a particular stage to prevent over loading and also for providing the signal generators with means of calibrating there outputs accurately. Variable attenuator provide continuous or step wise variableattenuation.

5. *VSWR meterSW-215*

AVSWR meter basically consists of a high gain; high Q low noise voltage amplifier normally tuned at a fixed frequency at which the microwaves signal is modulated. The VSWR meter uses the detector signal out of the microwave

detector as its input, amplifies the same and provides the output on the calibrated volt meter. The meter itself can be calibrated in terms of VSWR

III. THEORY:

Microwave frequency can be measured by either electronic or mechanic techniques. Electronic Technique: These techniques are more accurate but expensive. Frequency counters are used. The unknown frequency is compared with harmonics of a known lower frequency, by use of a low frequency generator, as harmonic generator and a mixer.

Mechanical Technique: These include slotted line and cavity meter techniques whose operation and accuracy depends on the physical dimensions of mechanical devices.

Slotted-Line Technique: A slotted line is a piece of transmission line so constructed that the voltage and current along it can be measured continuously over its length.

For measuring the frequency, the distance between maxima (or) minima is measured on the slotted line horizontal scale from the above setup.

$$\frac{\lambda g}{2} = d2 - d1 \ cm$$

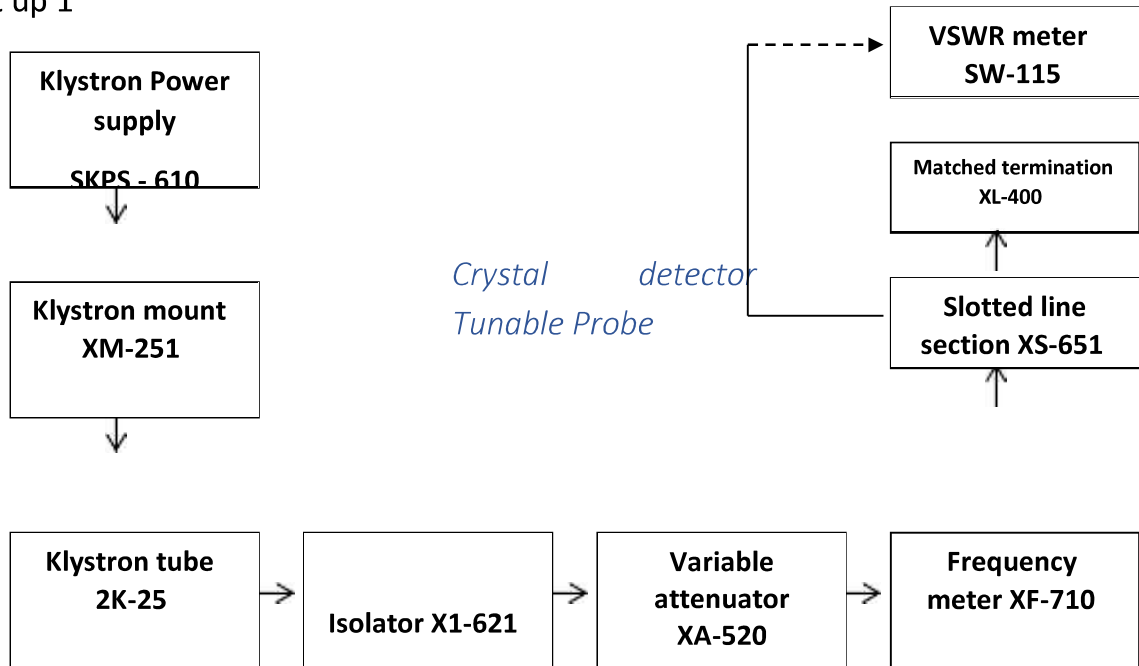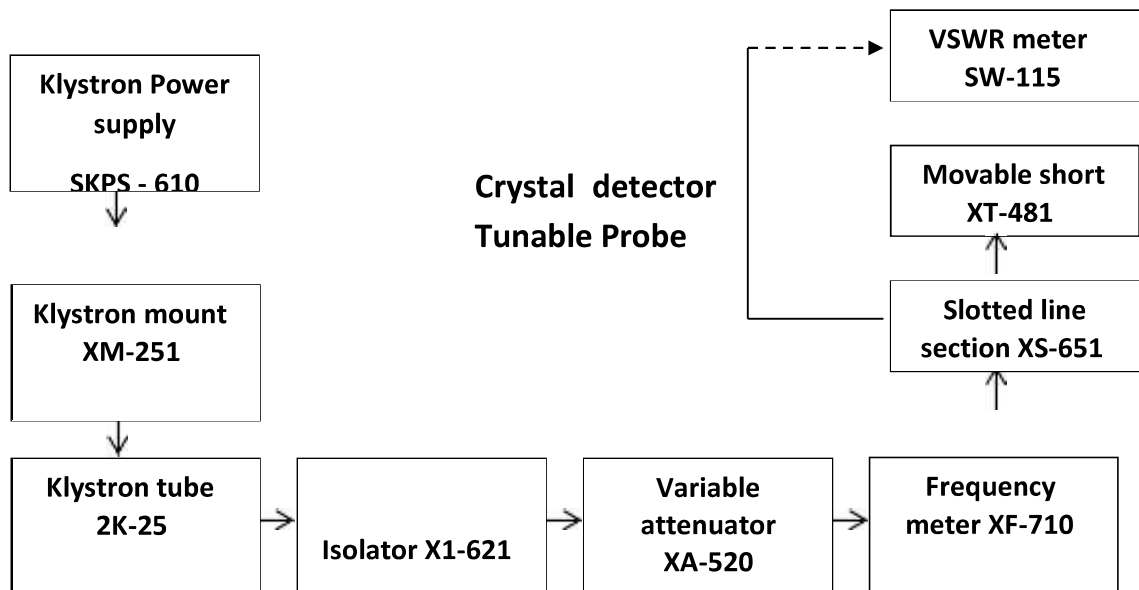$$\lambda g = 2(d2 - d1) \ cm$$

$$\lambda g = \frac{\lambda o}{\sqrt{1-(\lambda o/\lambda o)^2}}$$

For TE10 Mode, λC = 2a where **'a'** is the waveguide dimension **(22.86mm)**. The frequency so measured is not very accurate.

IV. BLOCKDIAGRAM:

Set up 1

```
┌──────────────────┐                              ┌──────────────────┐
│ Klystron Power   │         ┌ ─ ─ ─ ─ ─ ─ ─ ─ ─ →│ VSWR meter       │
│ supply           │         ╎                    │ SW-115           │
│                  │         ╎                    └──────────────────┘
│ SKPS - 610       │         ╎
└──────────────────┘         ╎                    ┌──────────────────┐
         ↓                   ╎                    │ Matched termination│
                             ╎                    │ XL-400           │
┌──────────────────┐         ╎                    └──────────────────┘
│ Klystron mount   │  *Crystal      detector*              ↑
│ XM-251           │  *Tunable Probe*           ┌──────────────────┐
│                  │         └──────────────────│ Slotted line     │
└──────────────────┘                            │ section XS-651   │
         ↓                                      └──────────────────┘
                                                         ↑
┌──────────────┐   ┌──────────────┐   ┌──────────────┐   ┌──────────────┐
│ Klystron tube│ → │              │ → │ Variable     │ → │ Frequency    │
│ 2K-25        │   │ Isolator X1-621│ │ attenuator   │   │ meter XF-710 │
│              │   │              │   │ XA-520       │   │              │
└──────────────┘   └──────────────┘   └──────────────┘   └──────────────┘
```

Set up 2

```
┌──────────────────┐                              ┌──────────────────┐
│ Klystron Power   │         ┌ ─ ─ ─ ─ ─ ─ ─ ─ ─ →│ VSWR meter       │
│ supply           │         ╎                    │ SW-115           │
│                  │         ╎                    └──────────────────┘
│ SKPS - 610       │         ╎
└──────────────────┘  **Crystal  detector**       ┌──────────────────┐
         ↓            **Tunable Probe**            │ Movable short    │
                             ╎                    │ XT-481           │
┌──────────────────┐         ╎                    └──────────────────┘
│ Klystron mount   │         ╎                             ↑
│ XM-251           │         └──────────────────┌──────────────────┐
│                  │                            │ Slotted line     │
└──────────────────┘                            │ section XS-651   │
         ↓                                      └──────────────────┘
                                                         ↑
┌──────────────┐   ┌──────────────┐   ┌──────────────┐   ┌──────────────┐
│ Klystron tube│ → │              │ → │ Variable     │ → │ Frequency    │
│ 2K-25        │   │ Isolator X1-621│ │ attenuator   │   │ meter XF-710 │
│              │   │              │   │ XA-520       │   │              │
└──────────────┘   └──────────────┘   └──────────────┘   └──────────────┘
```

V.PROCEDURE:

i.   Setup the components and equipments as shown in thefigure.

ii.   Setup the variable attenuator at minimum attenuationposition.

iii.   Keep the control knobs of VSWR meter as shownbelow:
       a) Range 40dB
       b) Input switch – crystal lowimpedance
       c) Meter switch – normalposition
       d) Gain (coarse and fine) = mid positions

iv.   Keep the control knobs of klystron power supplyas:
       a) Beam voltage –off
       b) Mod switch –AM
       c) Beam voltage knob – fullyanticlockwise.
       d) Repeller voltage = fullyclockwise
       e) AM amplitude knob = around fullyclockwise.
       f) AM frequency knob = around midposition.

v.   Switch the klystron power supply, VSWR meter and cooling fanswitch.
vi.   Switch 'ON' the beam voltage switch to set beam voltage at **300V** with the help of beam voltageknob.
vii.   Adjust the repeller voltage to get some deflection in VSWRmeter.
viii. Maximize the deflector with AM amplitude and frequency control knob of power supply.
ix.   Tune the plunger of klystron mount for maximumdeflection.
x.   Tune the reflector voltage knob for maximumdeflection.
xi.   Tune the probe for maximum deflection in VSWRmeter.
xii.   Tune the probe frequency meter knob to get a dip on the VSWR scale and note down the frequencymeter.
xiii. Replace the termination with movable short and detune the frequencymeter.
xiv.   Move the probe along the slottedline.
xv.   Move the probe to next minimum position and record the probe positionagain.
xvi.   Calculate the guide wavelength as twice the distance between two successive minimum positions obtained asabove.
xvii. Measure the waveguide inner broad dimension **'a'** which will be around **22.86mm**for
       x-band.
xviii. Calculate   the   frequency   by   following
equation Xix.   Verify   the   frequency
obtained by frequencymeter.

VI OBSERVATIONS:

Distance (in cm)	Power (in dB)

VII. CALCULATIONS:

$$f = \frac{c}{\lambda_0} = c \sqrt{\frac{1}{\lambda^2} + \frac{1}{\lambda_c^2}}$$

$$\lambda g = 2(d2 - d1) \text{cm}$$

$\lambda C = 2a$ where a = 2.286cm.

$$\lambda_g = \frac{\lambda_0}{\sqrt{1 - (\frac{\lambda_0}{\lambda_c})^2}}$$

VIII. GRAPH:

$$\frac{1}{\lambda_c^2}$$ (vertical axis)

$$\frac{1}{\lambda_g^2}$$ (horizontal axis)
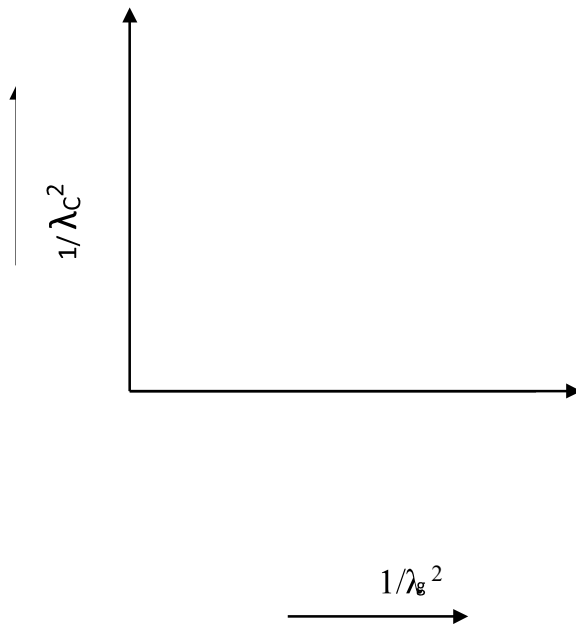
IX. RESULT:

Thus the frequency and wavelength of rectangular waveguide has been determined.

Frequency =_____

wave length= _____

X. INFERENCES:

The frequency observed from the frequency meter and the measured frequency by the slotted line technique is almost equal.

XI. PRECAUTIONS:

i.   Keep all the knobs in minimum position before going to switch 'ON' the power supply of VSWR / Klystron powersupplies.
     Note: For klystron power supply "HT" should be 'OFF' before switching 'ON' the main supply.

ii.   Beam knob should be completely in anticlockwise direction and repeller voltage knob should be completely clockwisedirection.

iii. Switch on the main supply and give some warm up time to get current / accurate reading.

iv.   After the completion of experiment, before going to switch off the mains keep all the knobs in minimum position (i.e.) as those are in rule1.

v.   If the main supply failed in the middle of the experiment, come to 1st

condition (i.e.) keep all the knobs in minimum positions and switch off mainswitches.

vi. Don'tincreasetherepellervoltagemorethan**-70V**(i.e.)itshouldbebetween**-70V** to **-270V.**

XII. APPLICATIONS:

It is used for Measurement of unknown impedance and Measurement of reflection coefficient.

XIII. EXTENSIONS:

Determination of VSWR for different conditions: open circuit and short circuit

XIV. TROUBLESHOOTING:

FAULT	DIAGNOSIS
Meterreadingfluctuating :	keep it in low dB De tune the frequency meter.
Nodip observe :	adjust it very slowly around 9.5GHz.

EXPERIMENT:-2

Development of an experimental setup to measure unknown impedance of rectangular waveguide and verify the results using smith chart.

1. **APPARATUS**

1	Klystron power supply SKPS –610
2	Klystron tube2k25
3	Klystron mountxm-251
4	Isolator XI-621
5	Frequency meterXF-710
6	Variable attenuator XA-520
7	Detector mountXD-451
8	Waveguide standsX4-535
9	VSWR meter SW-215
10	Movable shortXT-481
11	Matched terminationxl-400
12	Slotted sectionXS-651
13	Tunable probeXP-655

(ii) DESCRIPTION OF THE EQUIPMENT:

1. Klystron power supply SKPS – 610
The model Klystron Power Supply SKPS-610 is general purpose laboratory power supply which is specially designed to use for reflex klystron tubes of S to X band frequency range. It is reliable power source with very high regulation and low ripple contents.

The klystron power supply SKPS-610 has built in modulation facilities of amplitude and frequency modulation. Amplitude modulation can be applied with 0-110V (p.p.) Square wave and with frequency of 500 Hz to 2.5 KHz. This amplitude modulation is generally used along with VSWR measurements in slotted line technique. Frequency modulation is used for direct study of klystron modes on the Oscilloscope. Pure carrier wave operation and in external modulation facilities is also provided with the instrument for generalized use.

The klystron power supply also provides all the other D.C. Voltages required for operation of reflex klystron tube such as beam, heater and reflector voltage. The ranges of all these voltages are given in the specification data sheet.

2. Klystron tube2k25

The klystron tube 2k25 is a single cavity variable frequency microwave generator of low power and low efficiency. It consists of an electron gun, a filament surrounded by cathode and a focusing electrode at cathode potential. The electrons emitted by the cathode travel towards the reflector through an anode kept at higher potential compared to the cathode. When they approach the anode, the electrons form bunches and the bunches ultimately return towards the anode cavity after traveling a small distance towards the reflector. The power is taken from the anode reentrant cavity.

3. IsolatorXl-621

It is a two port device which provides very small amount of attenuation for transmission from port 1 to port 2 and provides maximum attenuation for transmission from port 2 to port 1. It is very much desirable when we want to match a source with variable load. It allows power flow only from the generator towards the load and suppresses any reflectedpower.

4. Frequency meterXF-710

It is also called wave meter. Usual construction of it consist a cylindrical cavity mounted on a shaft. By rotating the shaft the volume in the cavity is changed and it becomes resonant and gives minimum impedance at the resonant frequency. The scale calibrated and the resonant frequency can be directly read from the scale after observing a dip in the output meter.

5. Variable attenuatorXA-520

This is a wave guide piece having a groove on the lateral side. By rotating the screw the depth of penetration of resistive pad changes, there by introducing some attenuation. Attenuators are commonly used for measuring power game or loss in dB for providing isolation between instruments for reducing the power input to a particular stage to prevent over loading and also for providing the signal generators with means of calibrating there outputs accurately. Variable attenuator provide continuous or step wise variableattenuation.

6. VSWR meterSW-215

AVSWR meter basically consists of a high gain; high Q low noise voltage amplifier normally tuned at a fixed frequency at which the microwave signals is modulated. The VSWR meter uses the detector signal out of the microwave detector as its input, amplifies the same and provides the output on the calibrated volt meter. The meter itself can be calibrated in terms of VSWR

II. THEORY:

The impedance at any point on a transmission line can be written in the form R+jx. For comparison SWR can be calculated as:

$$S = \frac{1 + |R1|}{1 - |R1|} \quad \text{where}$$

Reflection coefficient

$$R = \frac{Z - Zo}{Z + Zo}$$

Where Zo = characteristic impedance of waveguide at operating frequency. Z = Load Impedance.

The measurement is performed in the following way:

The unknown device is connected to the slotted line and the position of one minima is determined. The unknown device is replaced by movable short to the slotted line. Two successive minima positions are noted. The twice of the difference between minima positions will be guide wavelength. One of the minima is used as reference for impedance measurement. Find the difference of reference minima and maxima position obtained from unknown load. Let it be 'd'. Take a smith chart, taking 'I' as centre draw a circle of radius equal to 'SO'. Mark a point on the circumference of smith chart towards load side at a distance equal to d/1g. Join the centre with this point. Find the point where it cuts the drawn circle. The co-ordinates of this point will show normalized impedance ofload.

III. BLOCKDIAGRAM:

For determination of 'λg'
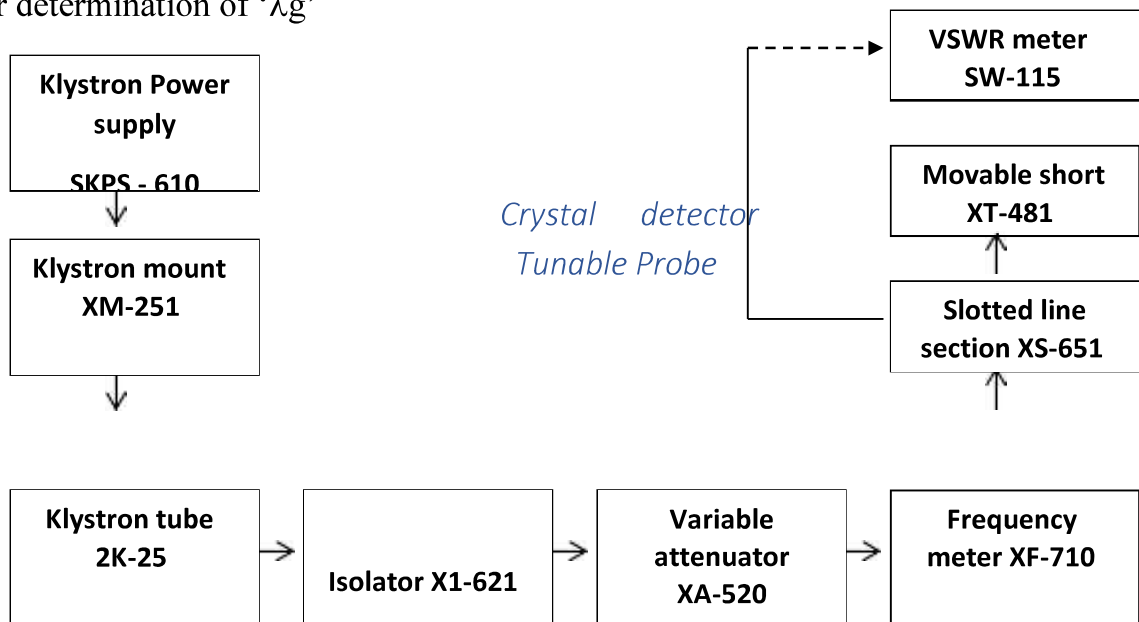
Set up 2: For measuring SWR and maximas and minimas
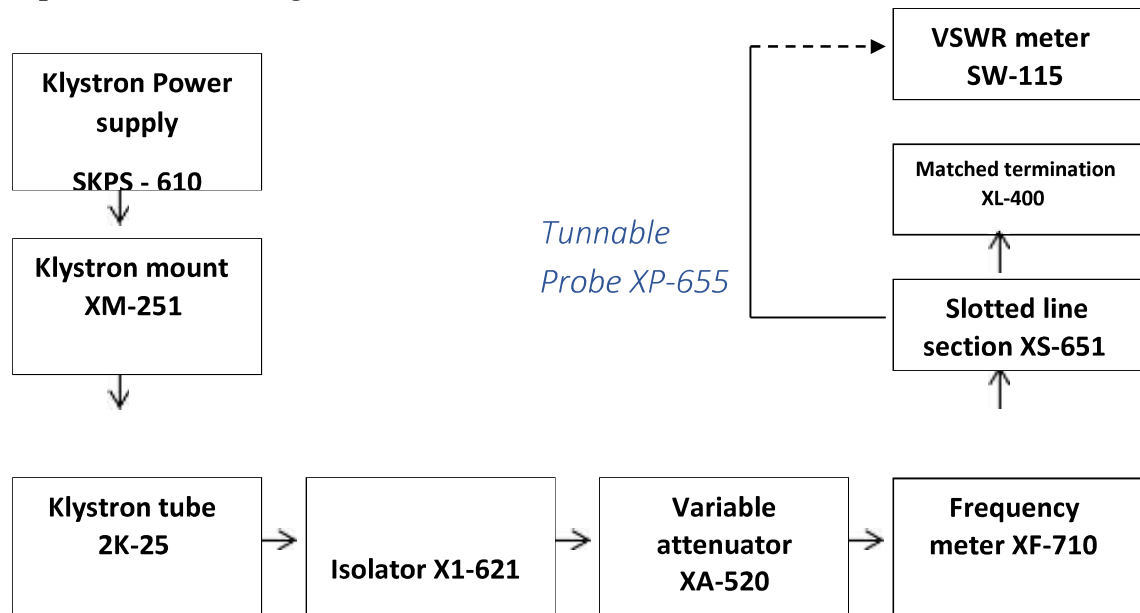
Klystron Power supply **SKPS - 610**				**VSWR meter SW-115**
↓				**Matched termination XL-400**
Klystron mount XM-251		*Tunnable Probe XP-655*		↑ **Slotted line section XS-651**
↓				↑
Klystron tube 2K-25 →	**Isolator X1-621** →	**Variable attenuator XA-520** →	**Frequency meter XF-710**	

PROCEDURE:

i.   Setup the components and equipments as shown in thefigure.

ii.   Setup the variable attenuator at minimum attenuationposition.

iii.   Keep the control knobs of VSWR meter as shownbelow:
a) Range 40dB
b) i/P switch – crystal lowimpedance
c) meter switch – normalposition
d) Gain (coarse and fine) = midpositions

iv.   Keep the control knobs of klystron power supplyas:
a) Beam voltage –off
b) Mod switch –AM
c) Beam voltage knob – fullyanticlockwise.
d) Repeller voltage = fullyclockwise
e) AM amplitude knob = around fullyclockwise.
f) AM frequency knob = around midposition.

v.   Switch the klystron power supply, VSWR meter and cooling fanswitch.

vi.   Switch 'ON' the beam voltage switch to set beam voltage at 300v with the help of beam voltageknob.

vii.   Adjust the repeller voltage to get some deflection in VSWRmeter.

viii.   Maximize the deflector with AM amplitude and frequency control knob of

power supply.

ix.   Tune the plunger of klystron mount for maximumdeflection.

x.   Tune the reflector voltage knob for maximumdeflection.

XI.   Tune the probe for maximum deflection in VSWRmeter.

xii.   Tune the frequency meter knob to get a 'dip' on the VSWR scale and note down the frequency directly from frequencymeter.

xiii.   Move the probe along the slotted line to get maximumdeflection.

xiv.   Adjust VSWR meter gain control knob and variable attenuator until the meter indicates 10 on the normal dB SWRscale.

xv.   Move the probe to next minima position and note down the SWR 'S$_O$' on the scale. Also note down the probe position. Let it bed.

xvi.   Note the position of 2 successive (maxima) or minima positional Let it be as d$_1$ and d$_2$.
Hence λg = 2(d2-d1)

xvii.   Calculateλg

xviii.   Find out normalized impedance asdescribed.

xix.   Repeat above for differentfrequencies.

VI. OBSERVATIONS:

i. Movableshort:

Distance (cm)	Power (dB)
	(min-1)
	(max-1)
	(min-2)
	(max-2)

ii. MatchedTermination

Distance (cm)	Power (dB)
	(min-1)
	(max-1)
	(min-2)
	(max-2)

VII. CALCULATIONS:

λg = 2(d2-d1)

where d2 – is distance at min-2 or

max-2 d1 - is distance at min-1 or

max-1

VIII. GRAPH:

IX. RESULT:

The unknown impedance has been determined using smith chart

XI. PRECAUTIONS:

i.   Keep all the knobs in minimum position before going to switch 'ON' the power supply of VSWR / Klystron powersupplies.
Note: For klystron power supply "HT" should be 'OFF' before switching 'ON' the main supply.

ii.   Beam knob should be completely in anticlockwise direction and repeller voltage knob should be completely clockwisedirection.

iii.   Switch on the main supply and give some warm up time to get current / accuratereading.

iv.   After the completion of experiment, before going to switch off the mains keep all the knobs in minimum position (i.e.) as those are in rule1.

v.   If the main supply failed in the middle of the experiment, come to 1stcondition (i.e.) keep all the knobs in minimum positions and switch off mainswitches.

vi.   Don't increase the repeller voltage more than -70V(i.e.) it should be between -70Vto-270V.

Experiment -3

Design an experiment to develop I-V characteristics of microwave Gunn diode   and to measure output power and frequency as a function of bias voltage using microwave test bench.

I EQUIPMENTS AND COMPONENTS:

i. APPARATUS

1. Gunn Power supplyGS-610
2. Gunn oscillatorXG-11
3. Isolator XL-621
4. Frequency meterXF-710
5. Pin Modulator
6. Matched terminationXL-400

(ii) DESCRIPTION OF EQUIPMENT:

1. Gunn Power Supply (GS-610)

The type GS-610 Gunn Power supply comprises of an electronically regulated power supply and a square wave generator designed to operate a Gunn Oscillator type XG-11 and PIN Modulator XM-55. The DC Voltage is variable from 0 to - 12 volts. However, the output voltage will not exceed +11 Volts because of over voltage Zener protection (Max. operating voltage for Gunn Oscillator is +12 Volts). The frequency of the square wave modulation can be continuously varied from 800 to 1100 Hz. The front panel meter indicates the Gunn voltage and the current drawn by the Gunn diode. The power supply has been designed to protect the Gunn diode in followingconditions:-

1. Reverse Voltageapplication
2. Over voltagetransients
3. **Low frequency oscillations generated by the negative resistance of the GunnDiode.**

VoltageRange   :   0   to   12   Volts
(Positive) Current   : 750 mA (max.)
Stability   :   0.2% for ±10% variations in the mains
voltage. Ripple   :   1 mVrms.
ModulationVoltage   :   0 - ±10 Volts
(P-P) Frequency   :   800 – 1100Hz.
Outputconnector   :   BNC Female for GunnOscillator
                                     TNC Female for Pin Modulator

2. Gunn Oscillator(XG-11)

The Gunn Oscillator XG-11 is stable and low noise microwave source. The Gunn diode is mounted in waveguide cavity, and source frequency can be tunable over the range 8.5 – 12.0 GHz by a micrometer controlled tuning plunger. Maximum power output is 25 mW, but it varies with frequency, minimum about 5 mW.

3. IsolatorXl-621

It is a two port device which provides very small amount of attenuation for transmission from port 1 to port 2 and provides maximum attenuation for transmission from port 2 to port 1. It is very much desirable when we want to match a source with variable load. It allows power flow only from the generator towards the load and suppresses any reflectedpower.

4. Frequency meterXF-710

It is also called wave meter. Usual construction of it consist a cylindrical cavity mounted on a shaft. By rotating the shaft the volume in the cavity is changed and it becomes resonant and gives minimum impedance at the resonant frequency. The scale calibrated and the resonant frequency can be directly read from the scale after observing a dip in the output meter.

5. Pin Modulator(XM-55)

The Pin Modulator XM-55 has been designed to amplitude modulate the CW output of the Gunn Oscillator XG-11. Modulating Voltage of 1 KHz, obtained from the Gunn Power Supply GS-610 to drives the modulator. It has built in 6 db attenuation to avoid any loading on the Pin Diode.

III. THEORY:

Transferred Electron Devices (TED's) are bulk devices that do not have any junctions or gates. They are fabricated with the compounds like GaAs, InP, CdTe. These operate on hot electrons. The Gunn diode is one such example. This also exhibits property of –ve resistance. Gunn observed that periodic fluctuations of current passing through n-type GaAs specimen, when the applied voltage exceeded a certain critical value (2.4 kV/ cm).
Basic mechanism involved in the operation of bulk n-type GaAs devices is the transfer electrons from low conduction valley to upper subsidiary valley the u-valley.
The current increases till a certain value and falls off after crossing a certain voltage level and increases further linearly.

IV. CIRCUITDIAGRAM:

Gunn Power Supply GS-610	Gunn Oscillator XG-11	Isolator XL-621	Pin Modulator

↓

↓

Matched termination XL-400
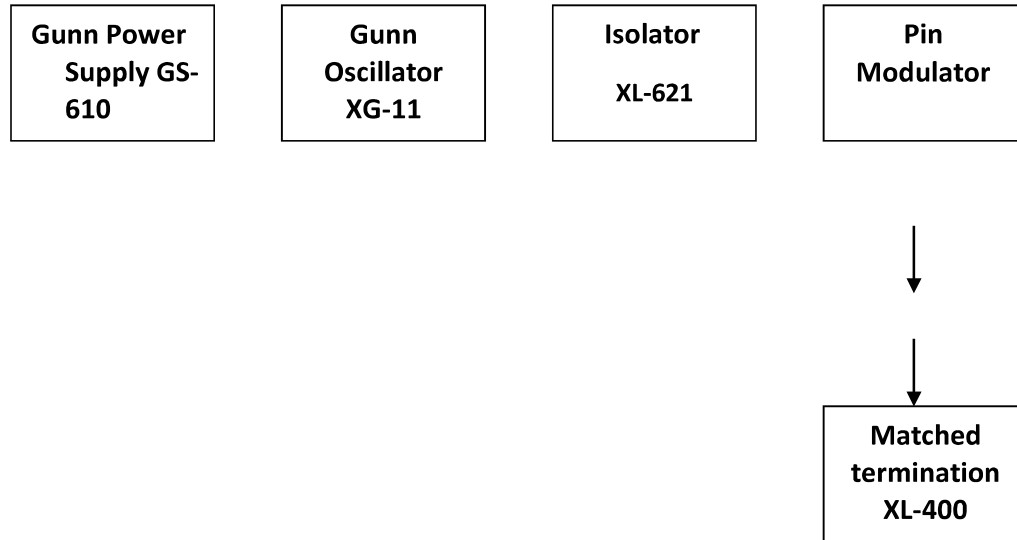
V. PROCEDURE:

1. Set the components as shown infigure.
2. Keep the control knobs of Gunn power supply asbelow
   - Meter switch should beoff
   - Gunn bias knob-fullyanticlockwise
   - Pin bias knob (mod amp) – fullyanticlockwise
   - Pin mode frequency – anyposition
3. Set the micrometer of Gunn oscillator for required frequency ofoperation.
4. Switch on the Gunn powersupply.
5. Measure the Gunn diode current corresponding to various Gunn bias voltages through the digital panel meter and meter switch. Do not exceed the bias voltage above10V.
6. Plot the voltage and current reading on the graph and compare with expected graph.
7. Measure the threshold voltage which corresponds to maximumcurrent.

VI. OBSERVATIONS:

S. No	Voltage (V)	Current (mA)

VII. CALCULATIONS:

V_T (Thresholdvoltage) =

$$\frac{\qquad\qquad\qquad}{=}\; \text{Imax}$$

VIII. GRAPH:
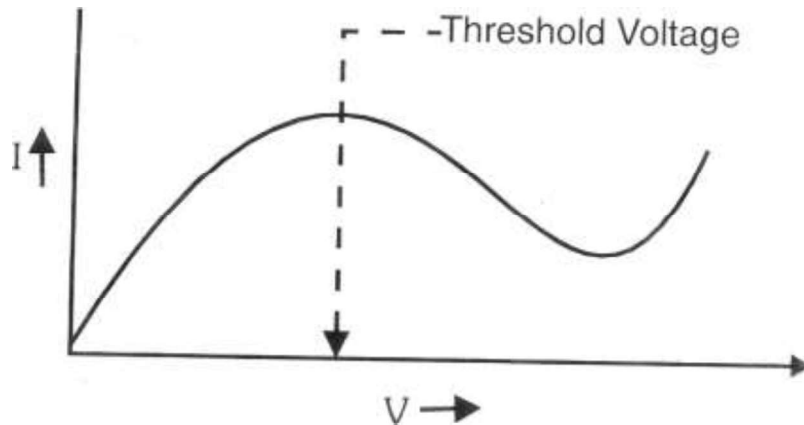
-Threshold Voltage

IX: RESULT:

The V-I characteristics of Gunn diode has been observed. The threshold voltageis ____

X: INFERENCE:

1. Thus the characteristics of Gunn diode had beenverified.
2. At the threshold voltage maximum current is observed
3. Negative resistance region isachieved

XI:PRECAUTIONS:

i.   Do not keep Gunn bias knob position at the threshold position for more than 10-15 seconds
ii.   Reading should be obtained as fast as possible otherwise due to excessive heat Gunn diode mayburn
iii. Care should be taken such that the bias voltage should not exceed above10V

XII. APPLICATIONS:

i.   In radar transmitters
ii.   Broadband linear amplifiers

iii.   As pump sources in
        paramp.
iv.   Low and medium power oscillator in microwave receivers.
v.   Fast combinational and sequential logiccircuits.

XIII. EXTENSION:

i.   The Experiment can be carried out from the determination ofTransconductance.
ii.   The experiment can be carried our from the determination of negative resistance region.

XIV. TROUBLESHOOTING:

FAULT   **DIAGNOSIS**

1. No readinginmeter   :   wave guidealignment

2. No variation inthecurrent   :   Vary the pin modulatorslowly

EXPERIMENT:-4

To develop an experimental setup for power characterstics of wilkinson power divider.

Required appratus and Components:-
1.Wilkinson power divider
2.connecting cables and probes
3.MPT

THEORY:-

The Wilkinson power divider was for the first time introduced by Ernest Wilkinson in 1960. His proposed configuration was able to either split an input signal in two in-phase output signals or combine two equal phase input signals. The technique used to match the split ports to the common port is the quarter wave transformer. To match all three ports simultaneously, there is also a resistor between the second and third the port. This resistor basically isolates these two ports at the operating frequency of the splitter. In this example, a Printed Wilkinson RF Power Splitter operating at 9 GHz, has been modeled with HFWorks and the reflection and transmission coefficients along with other parameters are presented.

Figure 1 - 3D view of the modeled Wilkinson divider

The model has been optimized: the best dimensions were used to draw the example in 3D. the dimensions are described within this figure.

Simulation
The simulation of this power divider was run on an Scattering Parameters study from 4 GHz to 12 GHz. The circuit was proven efficient in equally splitting the power into two in-phase outputs.

Solids and Materials
Taking into account the very narrow depth of the skin effect, we can model several conductors by assigning air to them. The material of the substrate is freely chosen from the library of the materials; We can apply materials that we define ourselves. Along with that material, we can specify a Signal boundary condition for the RF carrier and a PEC boundary condition for the ground metal beneath the substrate.

Load/ Restraint
The ports are applied to the pad of the main excitation path. The conductor that carries the RF is assigned a signal boundary condition.

Meshing

The model doesn't have very particular shapes. A uniform mesh of the *assembly* should be sufficient without applying particular mesh controls.

Observations:-

FREQUENCY	READING AT DIRECT LEVEL	READING AT SERIAL PORT LEVEL	Remarks

Results

Animating the 3D electric field's distribution by varying the omega-T angle gives us a hint on how the wave propagates into the port and gets 'splitted' within the two branches of the divider. We can first have a look at the curve of the reflection coefficient at the port, to decide which frequency yields the best matching and thus the best propagation into the two ports, but it is possible to view the near field even for frequencies where no propagation takes place.
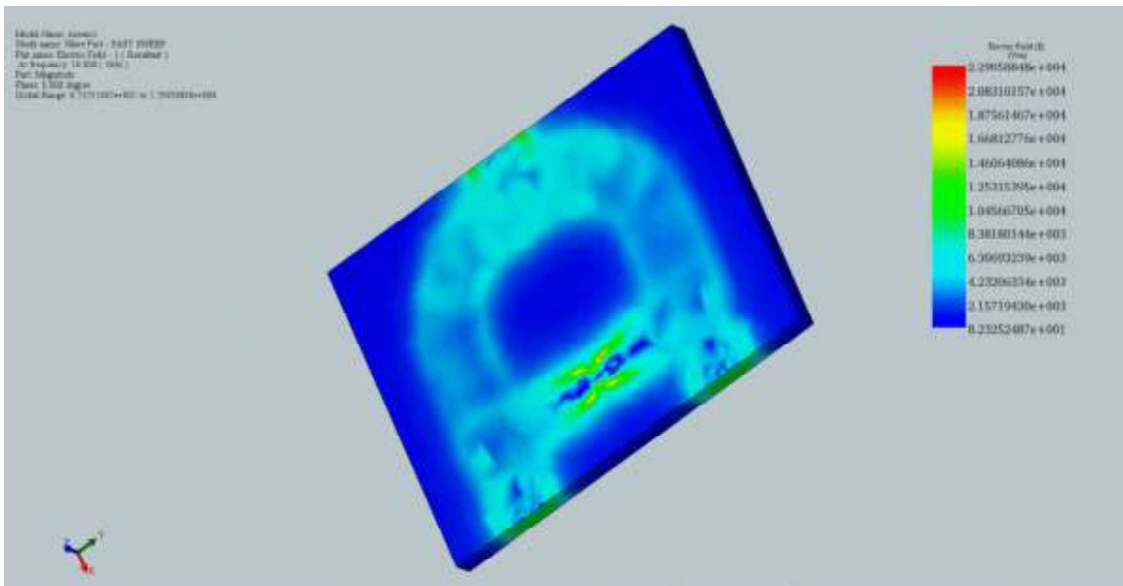
Figure 3 - Wave propagation in the power divider at 10 GHz

This figure shows plots of the variation of the reflection coefficient and insertion loss of the divider.
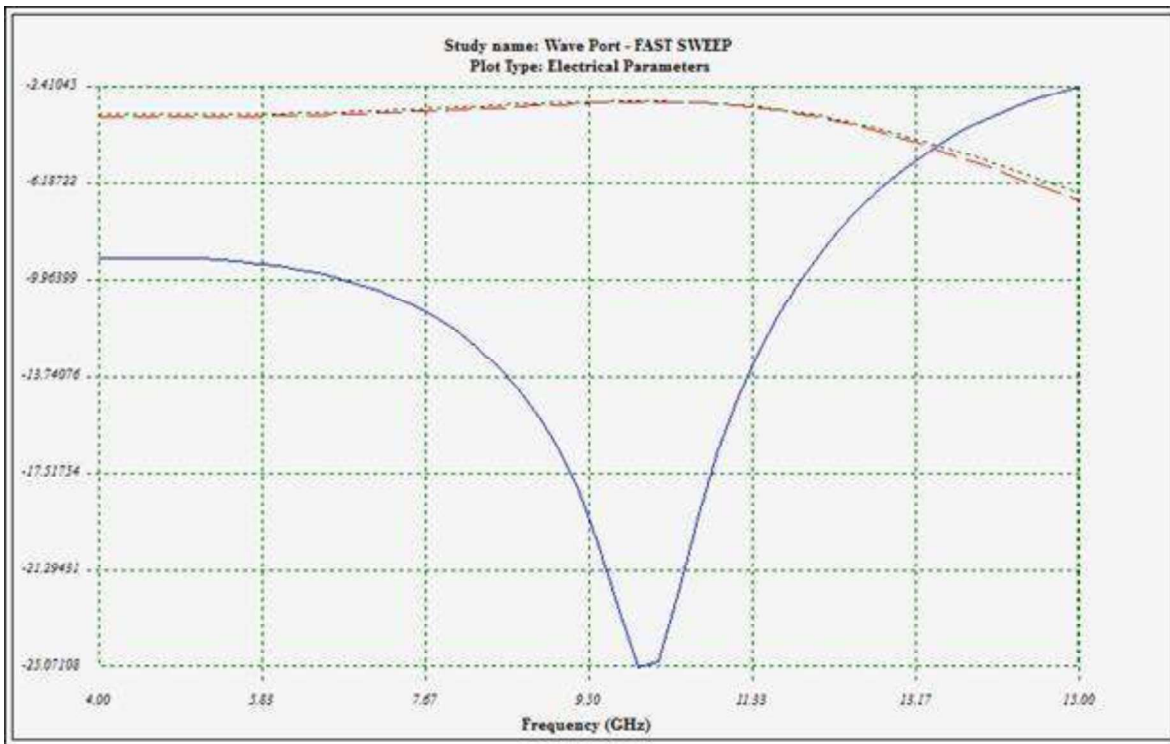
Blue= S11; Red=S21; Green=S31

Figure 4 - Variations of reflection coefficient and insertion losses at the outputs

We can see that the two outputs present a 3 dB power loss relatively to the input: the power is well split into halves. In this figure, we show generalized Scattering Parameters matrix shown in dB/Phase format (polar with magnitude expressed in dB); we can see that S21 and S31 are almost the same (-3 dB=0.5) and both isolation and return loss are better than -14 dB.
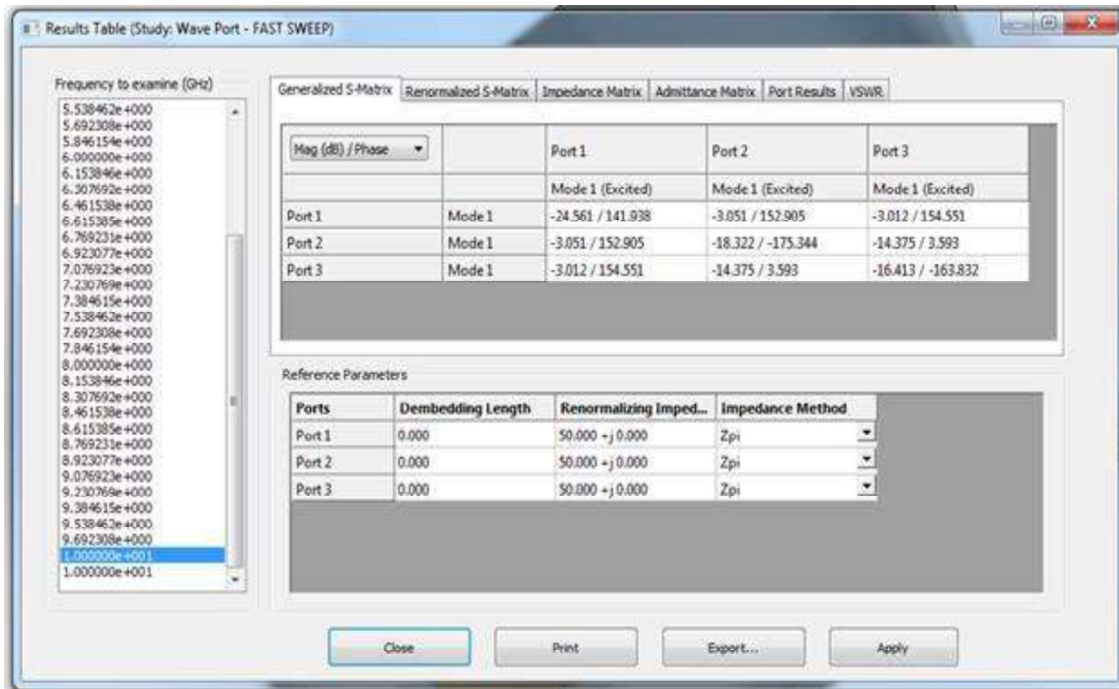
Figure 5 - Results table with various calculated parameters (within the tabs)

EXPERIMENT :-5

To Study the vector network analyzer.

Theory:-

VECTOR NETWORK ANALYZER-

A vector network analyzer is an instrument that measures the frequency response of a component or a network composed of many components, which can be both passive and active. A VNA measures the power of a high-speed signal going into and coming back from a component or a network, because power, in contrast to voltage and current, can be measured accurately at high frequencies. Both amplitude and phase of the high-frequency signal are captured at each frequency point. The built-in computer in the VNA calculates key parameters such as return loss and insertion loss of the network under test. It is also capable of visualizing the results in different formats—for example, real/imaginary, magnitude/phase, Smith chart, etc. In high-speed system tests, VNA is often used to characterize multi-port networks consisting of components such as connectors, filters, amplifiers, and transmission line/coaxial channels. VNA can be used for networks with an arbitrary number of ports—for example, the four-port differential pairs of a PCIe serial link. Figure 5.16 shows a contemporary four-port VNA with two ports connected to a device under test.
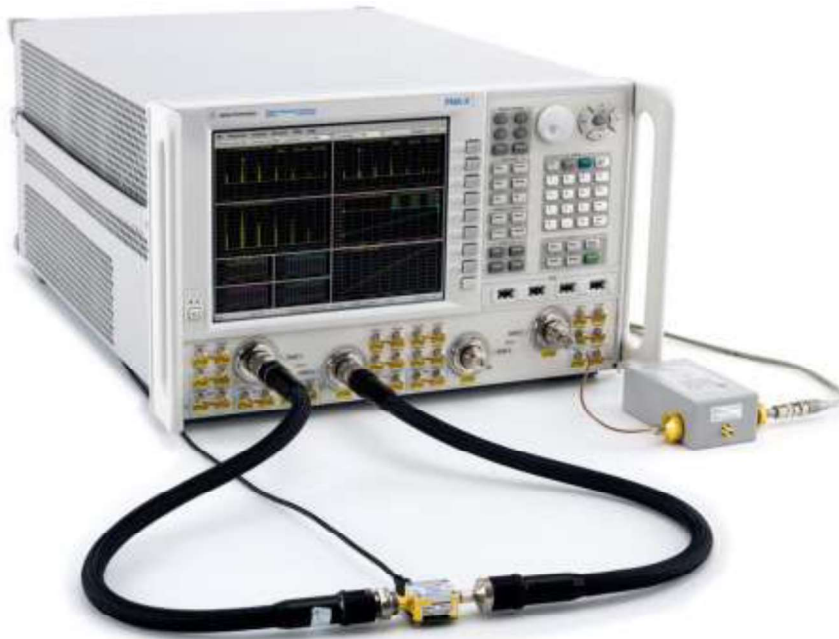
The VNA measurement frequency bandwidth and the number of frequency points across the selected frequency range are input options for a given VNA measurement. The VNA measures the high-speed signal vector response to a component or a network, one frequency at a time, by applying a continuous wave at that frequency. The magnitude of the continuous wave can also be adjusted. To faithfully characterize the measurement fixtures and generate an accurate filter for the scope to compensate for the measurement fixture losses to minimize the timing margin error, a minimum of all frequencies up to the second harmonic of the fundamental frequency of the high-speed digital/analog signal under test have to be covered in the VNA measurement bandwidth. To reduce the voltage margin error, all frequencies up to the fourth harmonic must be taken into account [5]. This translates to a minimum measurement bandwidth of 16 GHz for the PCIe Gen3 signal.

A simplified VNA architecture block diagram is shown in Figure 5.17. It involves RF sources, directional couplers (DC), dual-conversion frequency converters, reflectometers/power detectors, A/D converters, and an internal computer. The continuous wave signal used for testing is generated by the continuous wave source. The signal is swept over the specified measurement bandwidth in steps from one frequency to the next in small intervals defined by the user. A sample of the signal directly passes through the intermediate frequency (IF) processing unit and then to the reflectometers/power detector as a reference signal. The rest of the signal passes through a directional coupler (DC1) to the device/network under test (DUT). This directional coupler should not couple any of the incident power into its corresponding IF unit port. It is used to measure the reflected power from the DUT. The IF processing unit converts the incident and reflected signals to the IF frequency defined by the user, and the converted signals are subsequently detected by the reflectometers and converted to a digital signal. The ratio of the reverse reflected power to the incident power in digital format is then calculated by the internal computer and displayed as the mismatch at the input port of the DUT, also

known as the return loss. The rest of the incident power is sent through the DUT and sampled by the second directional coupler (DC2). The ratio of the transmitted power to the incident power is the gain or loss of the DUT, also known as the insertion loss. Other imported network parameters, such as phase, standing wave ratio, group delay, and impedance, can be calculated by a VNA.
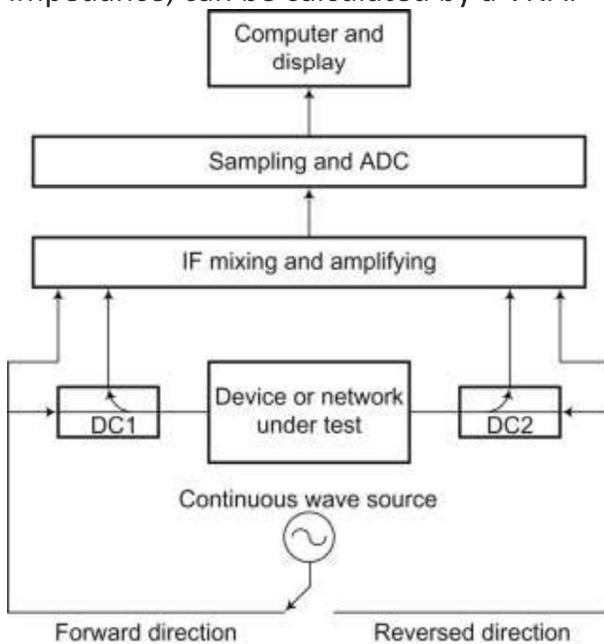
Propagation and Channel Modeling :-

In contrast to CW measurements, a vector network analyzer (VNA) allows maximum possible measurement bandwidths. The basic principle is to perform a frequency swept sampling of the channel over a predefined bandwidth. For providing corresponding channel responses in the delay domain, it is convenient to use discrete Fourier transformation (DFT) methods. As the VNA measurement principle is based on measuring fully coherent ratios between the transmitted and received signal both the transmitter and the receiver antennas have to be connected to the VNA with RF cables. The advantage is that the received signal is fully synchronized with the transmitted signal, which enables absolute delay measurements and coherent averaging over long times for the suppression of noise. A substantial drawback is that the mobility is limited by the RF cabling. Moreover, a single frequency sweep may take several seconds depending on the SNR at the receiver location which requires that both ends of the link are stationary. Another drawback is that the range is limited due to signal attenuation by the RF cables, which may be several dB per meter depending on the radio frequency. The range may be extended considerably to many kilometers, however, by replacing the RF cables by optical fibers using RF-to-opto and opto-to-RF converters. In Fig. 3.10 the measured channel response at 58.7 GHz using 2 GHz bandwidth is shown for a

NLoS microcellular street scenario [22]. In this specific measurement an optical fiber was used to extend the range to more than 100 m. The large bandwidth results in a large fraction of the rich multipath of this channel being resolved.
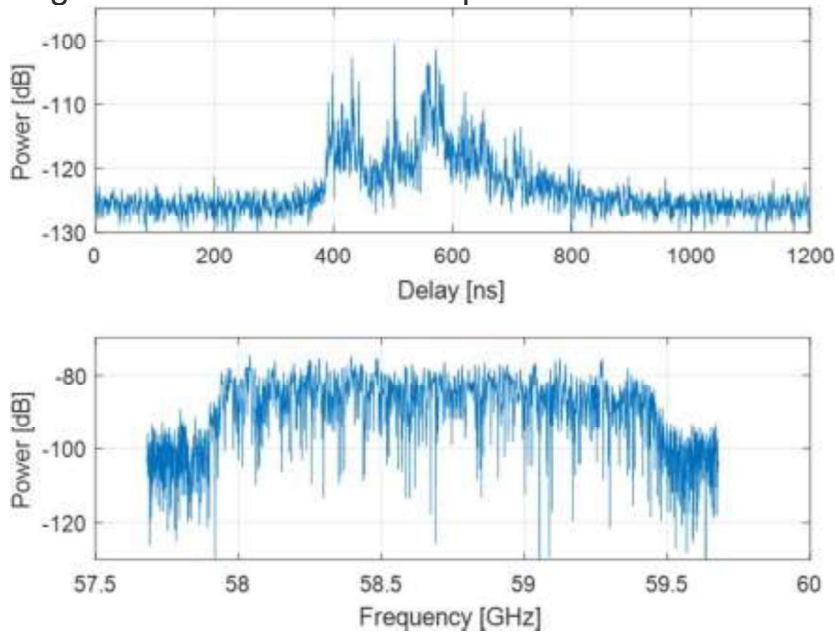
RESULT:- Thus we have studied the vector network analyzer .

EXPERIMENT :-6

To study the properties of magic tee and determine the scattering parameters of magic tee.

II. (i) EQUIPMENT AND COMPONENTS:

1.APPARATUS

1 Klystron power supply SKPS
– 610 2 Klystron tube 2k25
3 Klystron mount xm-
251 4 Isolator XI-621
5 Frequency meterXF-710
6 Variable attenuator XA-
520   7   Detector
mountXD-451
8 Waveguide stands X4-

535 9 VSWR meter SW-215
10 Movable shortXT-481
11 Matched   termination
XL-400   12   Slotted
sectionXS-651
13 Tunable probe XP-
655   14.Magic   Tee
XE-520

(ii) DESCRIPTION OF EQUIPMENT:

1. Klystron power supply SKPS –610

The model Klystron Power Supply SKPS-610 is general purpose laboratory power supply which is specially designed to use for reflex klystron tubes of S to X band frequency range. It is reliable power source with very high regulation and low ripple contents.

The klystron power supply SKPS-610 has built in modulation facilities of amplitude and frequency modulation. Amplitude modulation can be applied with 0-110V (p.p.) Square wave and with frequency of 500 Hz to 2.5 KHz. This amplitude modulation is generally used along with VSWR measurements in slotted line technique. Frequency modulation is used for direct study of klystron modes on the Oscilloscope. Pure carrier wave operation and in external modulation facilities is also provided with the instrument for generalized use.

The klystron power supply also provides all the other D.C. Voltages required for operation of reflex klystron tube such as beam, heater and reflector voltage. The ranges of all these voltages are given in the specification data sheet.

2. Klystron tube2k25

The klystron tube 2k25 is a single cavity variable frequency microwave generator of low power and low efficiency. It consists of an electron gun, a filament surrounded by cathode and a focusing electrode at cathode potential. The electrons emitted by the cathode travel towards the reflector through an anode kept at higher potential compared to the cathode. When they approach the anode, the electrons form bunches and the bunches ultimately return towards the anode cavity after traveling a small distance towards the reflector. The power is taken from the anode reentrantcavity.

3. IsolatorXI-621

It is a two port device which provides very small amount of attenuation for transmission from port 1 to port 2 and provides maximum attenuation for transmission from port 2 to port 1. It is very much desirable when we want to match a source with variable load. It allows power flow only from the generator towards the load and suppresses any reflectedpower.

4. Frequency meterXF-710

It is also called wave meter. Usual construction of it consist a cylindrical cavity mounted on a shaft. By rotating the shaft the volume in the cavity is changed and it becomes resonant and gives minimum impedance at the resonant frequency. The scale calibrated and the resonant frequency can be directly read from the scale after observing a dip in the output meter.

5. Variable attenuatorXA-520

This is a wave guide piece having a groove on the lateral side. By rotating the screw the depth of penetration of resistive pad changes, there by introducing some attenuation. Attenuators are commonly used for measuring power game or loss in dBs for providing isolation between instruments for reducing the power input to a particular stage to prevent over loading and also for providing the signal generators with means of calibrating there outputs accurately. Variable attenuator provide continuous or step wise variableattenuation.

6. VSWR meterSW-215

AVSWR meter basically consists of a high gain, high Q low noise voltage amplifier normally tuned at a fixed frequency at which the microwave signal is modulated. The VSWR meter uses the detector signal out of the microwave detector as its input, amplifies the same and provides the output on the calibrated volt meter. The meter itself can be calibrated in terms of VSWR

7. MagicTee

It is a four port device port one and port two are collinear arms port 3 is H-arm and port 4 is E-arm in this magic tee if any two ports are perfectly match to the junction then the remaining two ports are automatically match to thejunction.

III. THEORY:

The device magic tee is a combination of E and H-plane tee as shown in the figure. Arm 3 is the H-arm and arm-4 is the E-arm. If the power is fed into arm 3 (H-arm), the electric field divides equally between 1 and 2 arms with the same phase and no electric field exists in arm 4 (E-arm), it divides equally into arm-1 and arm-2 but out of phase with no power to arm-3, further if the power is fed in arm-1 and arm-2 simultaneously it is added in arm-3 (H-arm) and it is subtracted in E-arm i.e.arm-4.

The basic parameters to be measured for magic tee are defined below:

i.   **Input VSWR:** Value of SWR corresponding to each ort as a load to the line while other ports are terminated in matchedload.

ii.   **Isolation:** The isolation between E & H arms is defined as the ratio of the power supplied by the generators connected to the E-arm (port 4) to the power detected at H- arm (port 3) when side arms-1 and 2 are terminated in matched load.Hence

$Isolation\ I(dB) = 10\ log10 \left[\dfrac{P4}{P3}\right]$, similarly isolation between other ports may be defined.

iii. **Coupling Factor:** It is defined as $\dfrac{a}{20}$ where '**a**' is attenuation / isolationin **Cij=10-**

dB when 'i' is the input arm and j is the output arm.

$Thus\ \mathbf{a = 10\ log10} \left[\dfrac{P4}{P3}\right]$

Where, P3 = power delivered to
        arm-i P4 = power detected
        at j-arm

IV. BLOCKDIAGRAM:

Set up 1: for 0 dB setting

Klystron tube			Variable		Frequency
Klystron Power 2K-25 supply	Isolator X1-621	7	attenuator XA-520		meter XF-710 VSWR meter SW-115
SKPS - 610					
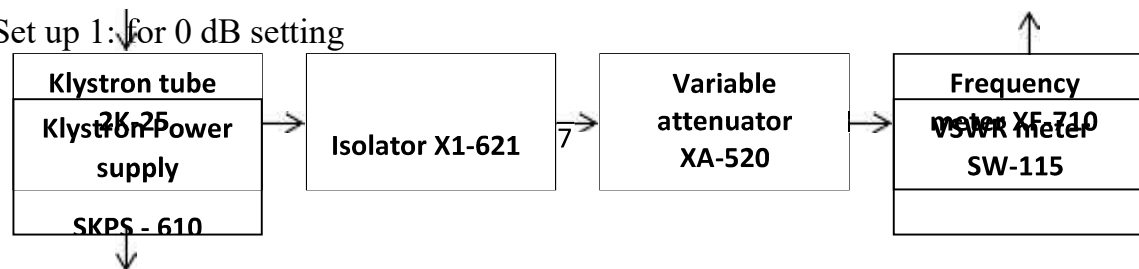
Set up 2: "Measurement set-up for S-parameters"

```
┌─────────────────┐
│   Klystron      │
│  power supply   │
│   SKPS-610      │
└─────────────────┘
         ↓
┌─────────────────┐
│   Klystron      │
│    Mount        │
│   XM-251        │
└─────────────────┘
         ↓
┌─────────────────┐
│  Klystron tube  │
│     2K25        │
│                 │
└─────────────────┘
         ↓
┌─────────────────┐
│    Isolator     │
│    X1-621       │
│                 │
└─────────────────┘
         ↓
```

| Variable attenuator XA-520 | → | Frequency meter XF-710 | → | $_1$Magic Tee 2 (4) | → | Detector Mount XD-451 |

| Matched Termination XL-400 | | VSWR meter SW-115 |

(arrows going up from Magic Tee to Matched Termination XL-400, and from Detector Mount to VSWR meter SW-115)

Magic Tee → ↓ → Matched Termination XL-400
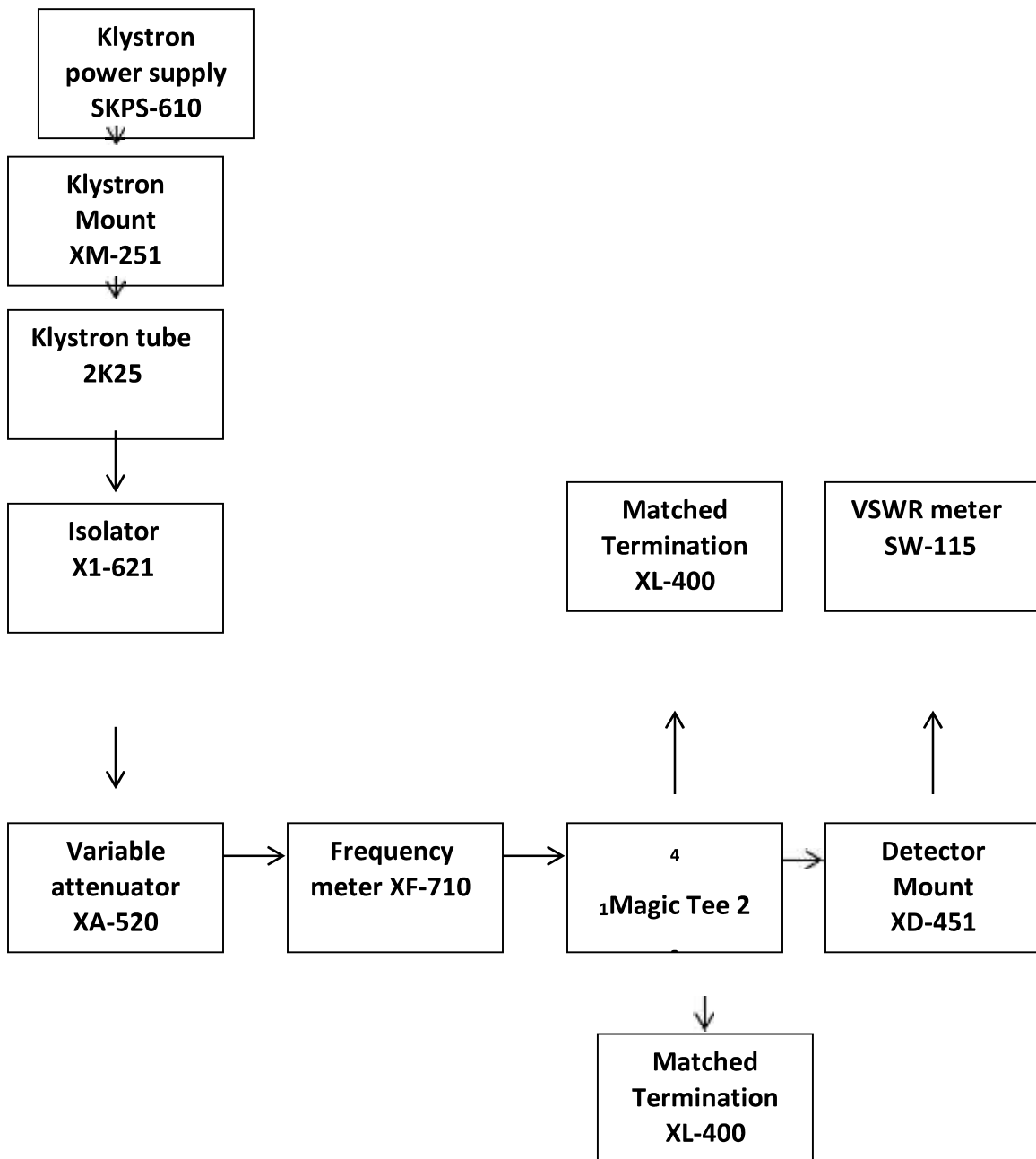
V. PROCEDURE:

i. Remove the tunable probe and magic tee from the slotted line and connect the detector mount to slottedline.

ii. Energise he microwave source for particular frequency of operation and tune the detector mount for maximumoutput.

iii. With the help of variable attenuator and gain control knob of VSWR meter, set any power level in the VSWR meter and note down. Let it be P3.

iv. Without disturbing the position of variable attenuator and gain

control knob, carefully place the magic tee after slotted line keeping H-arm connected to slotted line. Detector to E-arm and matched termination to arm-1 and arm-2. Note down the reading of VSWR meter. Let it beP4.

v.   Determine the isolation between port 3 and 4 as P3-P4(dB).
vi.   Determine coupling coefficient from equation given in theoryport.
vii. The same experiment may be repeated for other portsalso.
viii. Repeat the above experiment for otherfrequencies.

VI. OBSERVATIONS:

Input Port	Output Port	Matched Termination	Matched Termination	Power output
1	2	3	4	P2
1	3	2	4	P3
1	4	2	3	P4
2	1	3	4	P5
2	3	1	4	P6
2	4	1	3	P7
3	1	2	4	P8
3	2	1	4	P9
3	4	1	2	P10
4	1	2	3	P11
4	2	1	3	P12
4	3	1	2	P13

RESULT:-

Thus, we have studied the functions of magic tee by measuring its parameters and compared with its properties.

Experiment -7

Study of microwave development and software simulation tools like ADS, IE3D, FEKO etc.

Theory :-

ADS(ADVANCED DESIGN SYSTEM) Software;-

Keysight Technologies Inc. (NYSE: KEYS) introduced the latest release of its powerful advanced design system (ADS) software, ADS 2016. The software improves design productivity and speeds circuit and electromagnetic simulation performance. ADS 2016 provides new RF printed circuit board, laminate, module, and silicon RFIC products, as well as technologies, capabilities and other enhancements.

"ADS users represent some of the best and brightest engineers in the industry, with their designs in almost every communication device in the world," said Joe Civello, ADS product planning and marketing manager for KeysightEEsof EDA. "ADS 2016 represents a next major step forward and demonstrates our commitment to provide these engineers with software that increases their design productivity and efficiency through design flow integration."

ADS is the industry's leading electronic design automation software for RF, microwave and signal-integrity applications. ADS 2016 provides users with continued advances for the following.

Circuit Simulation

ADS 2016 enhanced the harmonic balance and circuit envelope simulators to improve speed, convergence and accuracy. Enhanced DC annotation speeds development of even the most complex designs. Windows and Linux support for the Electro-Thermal simulator now make it accessible to more designers.

Layout and Layout Verification

Numerous enhancements and new capabilities make ADS layout faster, more intuitive and manufacturing robust. Enhancements include manufacturing grid checking and control, improvements to ground nets, a

faster 3D viewer, improved highlighting and connectivity, and mapping of layout nets to schematic net names, as well as importing .brd files for Momentum electromagnetic (EM) simulation.

Silicon RFIC

ADS 2016 adds functionality to benefit the OpenAccess-based interoperable and native ADS silicon RFIC design flows. Designers can now use ADS/Virtuoso interoperable PDKs and iPDKs from foundries supporting the IPL Alliance standard from within the ADS platform, making industry leading simulation and analysis capabilities readily available to more silicon foundry customers and designers. Enhancements to the ADS Interoperable flow include the buildout of EM-cosimulation between circuit Spice models and interconnect EM, enhanced analogLib and VerilogA support, improved DC annotation, schematic Pcells, and added wire editing features. Layout and Momentum have also been enhanced for Silicon applications including support for TSMC iRCX, Momentum via simplification and metal fill simulation, as well as parameterized EM substrates. And, a new metal fill generation utility for silicon has been added to ADS Advanced Layout.

Signal and Power Integrity

ADS 2016 delivers SIPro, a signal integrity focused EM simulator that delivers high-accuracy for high-complexity PCB applications and PIPro, a power integrity focused suite of EM simulators for analysis of power delivery networks.

In addition to these advances, ADS 2016 also introduces a number of new products and improved capabilities.

- Direct layout import from Cadence Allegro .BRD file
- Pulsed amplitude modulation (PAM-4) signaling support in the IBIS-AMI Channel Simulator
- DDR bus simulator support for receiver equalization for DDR4
- Channel simuator support for the mix and match IBIS-AMI models with IBIS, SPICE and built-in transmitter/receiver models.
- New S-parameter checker/viewer

FEKO Software-

FEKO is a Method of Moments (MoM) tool that can be used to calculate the radiation pattern, impedance and gain of an antenna while mounted on some defined geometry. In addition, it can calculate the isolation or

mutual coupling (S12) between pairs of antennas, the near fields around an antenna and the electric currents that flow on an antenna or the surrounding structure.

The basic flow of performing a FEKO analysis consists of 1) Building a geometry for the antenna (example �a wire to represent the antenna) in CADFEKO or EDITFEKO 2) Building a geometry to represent surrounding geometry (for instance, can model an antenna on top of an airplane or an antenna on a supporting structure which will affect the antenna�s performance) in CADFEKO, EDITFEKO or an external tool 3) Meshing the Created Antenna and Surrounding Geometries (CADFEKO or EDITFEKO) 4) Requesting Solution Types and Setting Solution Parameters (CADFEKO or EDITFEKO) 5) Running the FEKO solver (FEKO) 6) Read in and interpret results using PostFEKO.

See whole tutorial at www.antennatheory.com.

IE3D SOFTWARE:-

IE3D is the first SCALABLE EM design and verification platform that delivers the modeling accuracy for the combined needs of high-frequency circuit design and signal integrity engineers across multiple design domains. For many companies, there is no longer just one EM problem at hand, but several different ones each presenting a unique bottleneck and delaying overall design closure. IE3D's multi-threaded and distributed simulation architecture and high-design capacity is the most cost-effective EM simulation and modeling solution for component-level and circuit-level applications. IE3D offers the highest simulation capacities and fastest turnaround times for the broadest number of applications making it the best choice for improving your design team productivity and meeting design schedules on time.

IE3D for Antenna Design

Today's high performance antenna array design requires both, large capacity EM simulation and unit array cell EM design and optimization capabilities. Getting the array unit cell right from the start is essential before replicating into a larger antenna array structure. And if accuracy is sought, then the designer needs to be extremely careful with approaches

that formulate estimated boundary conditions for unit cells used within larger arrays. These approaches typically suffer from poor capacity limits and do not accurately model the EM behavior between unit cells, especially for cells on the antenna array periphery. IE3D-SSD offers FASTEM to thoroughly explore the relevant design space and optimize the geometry for each unique unit cell. In addition, IE3D-SSD's superior capacity and run-time enable even the largest antenna arrays to be solved in least amount of time. IE3D-SSD is the best solution for your antenna design.

IE3D for RFID Design

Determining the primary and secondary characteristics of any RFID tag is always challenging. And ensuring the target layout will indeed meet the overall design objectives including a correct conjugate match is often a lengthy trial and error process. RFID design and EM modeling is a strength of IE3D-SSD where combining both disciplines of antenna and embedded passive design are best addressed. Critical geometries and dielectric impact on overall tag performance can be easily isolated and optimized accelerating design closure and improving the overall quality of the design. IE3D-SSD is the sure bet for RFID design.

IE3D For MMIC Design

Today's MMIC designer now require "circuit-level" EM simulation and modeling to support the accuracy and throughput demands to meet aggressive project schedule deadlines. The days of using flat planar 3D EM simulations solutions for MMIC design are over. Now full 3D geometry modeling of bond wires, via structures, and other 3D structures are a must-have for designers pushing performance. Circuit-level EM simulation captures the true electrical behavior of a designs passive structures including coupling. IE3D-SSD is the perfect fit for MMIC designers requiring full 3D geometry modeling and high simulation capacities.

IE3D for IC Package Modeling

Increasing operating frequencies and the rapid move to multi-chip system architectures have sparking a new parasitic extraction paradigm shift for IC packaging that now requires the accuracy of full 3D EM simulation and modeling. Package models built upon simplistic 2D approximations can no longer be relied upon to secure successful design closure. IE3D-SI is the perfect answer for the circuit design and signal integrity engineer

delivering the fastest throughput and capacity of any true 3D EM simulation commercially available. IE3D-SI delivers the most accurate models of packages housing stacked die, multiple die, and embedded passives with multi-height bond wires, solder balls, bumps, vias, and through silicon vias. IE3D-SI scalable computing architecture leverages your multi-core computers and server farms to solve the largest packaging problems with the fastest turnaround ensuring a fully optimized design where all end cases have been investigated prior to final tape out. IE3D-SI reduces the risk an undetected EM design flaw slips through the final verification saving time and money.

RESULT:- We have studied all the simulation tools like IE3D, FEKO, ADS etc.

Experiment :- 8

Designing of MMIC structure (MOSFET/Frequency doubler/mixer) using Advance Designing Software (ADS).

Theory:-

SIMULATION OF MMIC STRUCTURE FOR MOSFET:-

The following figure illustrates a setup for performing an idealized, swept DC bias simulation of a MOSFET. Vdd and Vgg are swept across a range of voltages. The result is a family of curves representing drain current versus drain voltage, for varying values of gate voltage.

To perform an idealized, swept DC bias simulation:
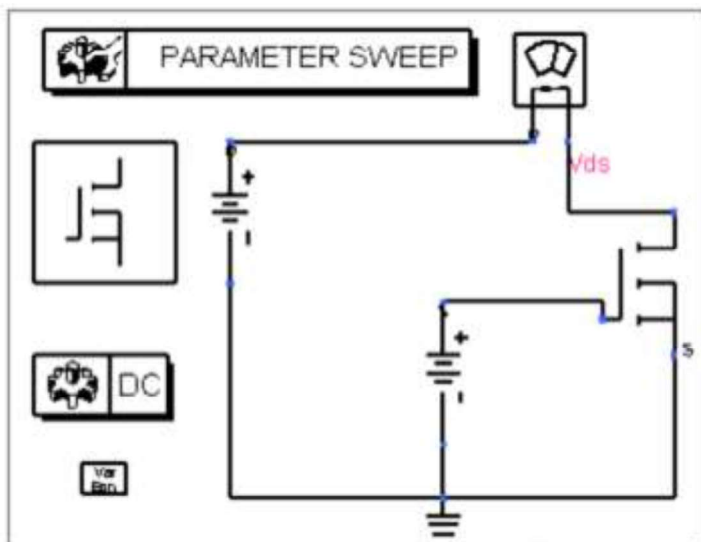
1. Place the MOSFET instance and model on the schematic.

2. From the Sources-Freq Domain palette, select V_DC. Place this V_DC component

(SRC1) between the base of the device and ground. Edit this component and define

Vdc = Vgg1, rather than a numerical value.

3. Place another V_DC component (SRC2) between the drain of the device and ground.

Open this component and define Vdc = Vdd1, rather than a numerical value.

This makes it possible to sweep these values. Any swept parameter must be

initialized by means of an equation, as will be demonstrated later.

4. From the Probe Components palette, select I_Probe. Place the probe between the

drain and SRC2, and rename it if you like (in this case, it has been called Probe1).

5. From the Simulation-DC palette, select and place the DC simulation component on

the schematic.

6. Edit the DC component by selecting the Sweep tab and modifying the sweep

parameters as follows:

Parameter to sweep = Vdd1. This appears as SweepVar on the schematic if this

parameter is displayed on the schematic.

Note

Variables entered in this field will appear in quotes on the schematic. If you enter a variable

directly on the schematic (in this case, as the right-hand side of the SweepVar statement of

the DC component), you must surround the variable with double quotes. This applies to the

other user-defined variables shown in subsequent steps.

Start = 0

Stop = 6

Step = 0.1

7. Click OK to accept changes and close the dialog box.

8. Choose Simulation-DC palette, select PrmSwp (ParamSweep). Place this

component on the schematic and edit it as follows:

On the Sweep tab:

Parameter to sweep = Vgg1

Start = 2

Stop = 3

Step = 0.25

On the Simulations tab:

Simulation 1 = DC1. This appears as SimInstanceName[1] on the

schematic if this parameter is displayed on the schematic.

9. Click OK to accept changes and close the dialog box.

10. Choose the Data Items palette, select Vareqn (Variables and equations). Place this

component on the schematic and enter the following equations:

Vdd1 = 4

Vgg1 = 2.5

11. Click OK to accept changes and close the dialog box.

12. Simulate. When the simulation is finished, plot the probe current (Probe1.i).

RESULT:- The resulting family of curves representing drain current versus drain voltage, for varying

values of gate voltage, is shown next:
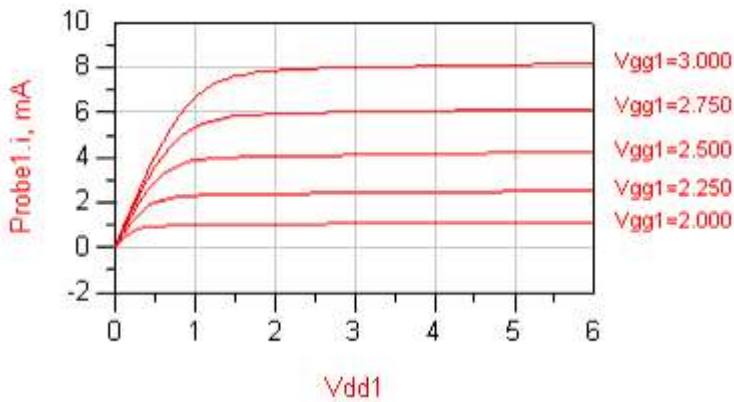
EXPERIMENT :- 9

Design of micro strip filters (Low Pass/ High Pass / Band Pass) using Advance Designing Software (ADS)/ IE3D Software.

THEORY:-

DESIGN OF LOW PASS FILTER USING ADS:-

Follow the procedure below to design the low pass filter.

- From the schematic window, click DesignGuide> Filter > Filter Control Window to open the Filter DesignGuide Control Window, as shown in Figure4-31.
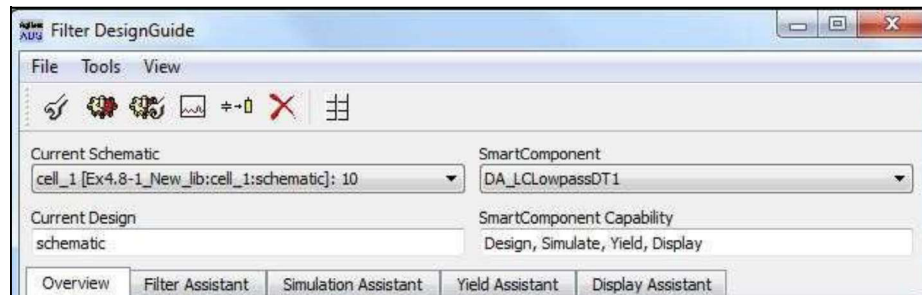
Figure 4-31 Filter DesignGuide control window

- In the Filter DesignGuide Control Window, click View > Component Palette-All to place the SmartComponent Palettes in the schematic window.

- From the list of Filter DG - All Palette in the schematic window, select the lowpass filter SmartComponenet,DA_LCLowpassDT.
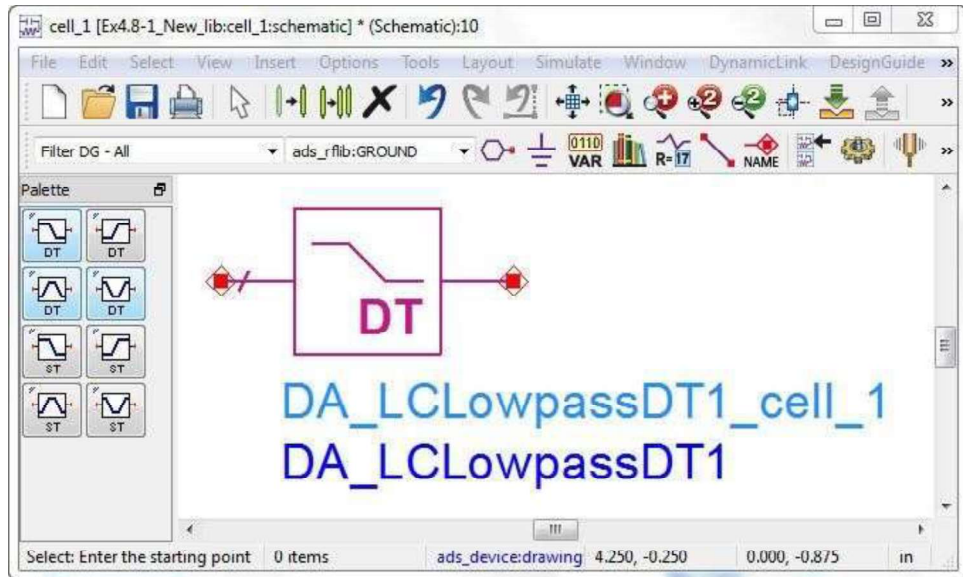
Figure 4-32 SmartComponent Palette in the schematic window

- Click anywhere within the schematic window to place thecomponent.

- Double-click the SmartComponent to open a dialog box containing all parameters.

- Modify the parameters to meet the design specifications, as shown in Figure4-33.
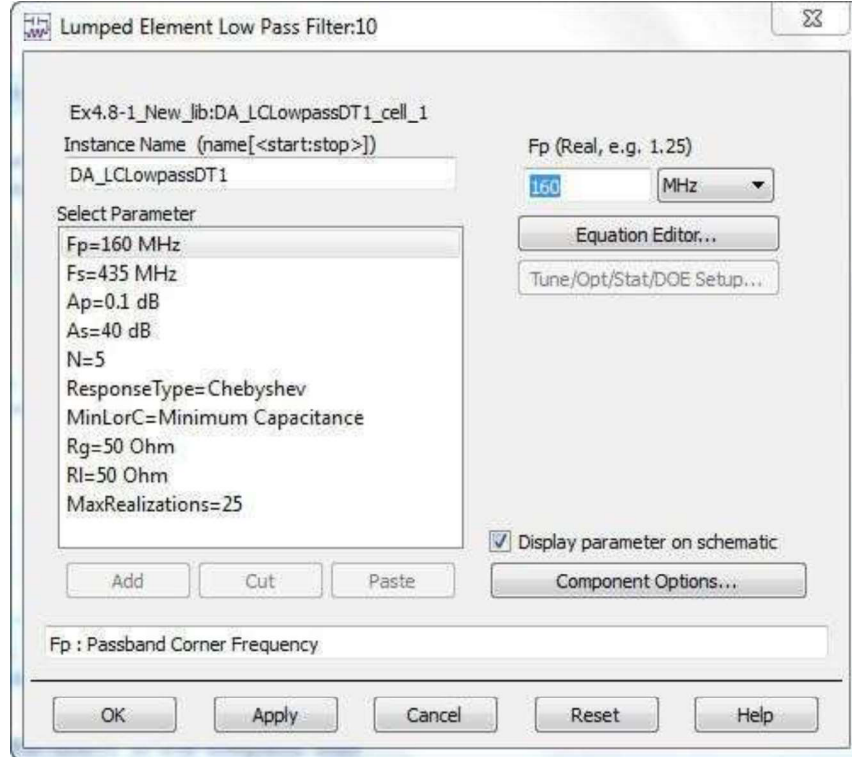
Figure 4-33 Modified parameters of the lowpass filter

- In the Filter DesignGuide control window select the Filter Assistant tab and click Design to start a simulation and generate the filter sub- network, as shown in Figure4-34.
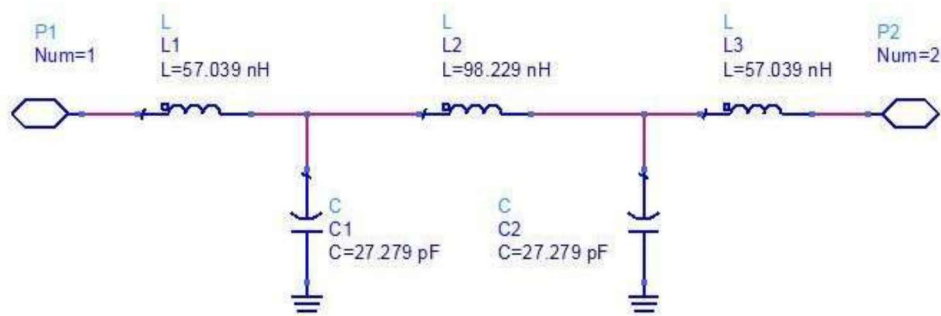
Figure 4-34 Lowpass filter sub-network (N=5)

The required filter order is determined by increasing the order until the specification of -40 dBc attenuation at 435 MHz is achieved. As the filter response curve in Fig.4-36 shows, this Low Pass filter must be of fifth order to achieve the required attenuation.

After SmartComponents are designed and tested, they can be used as standalone components. The Filter DesignGuide is not needed to use them in new designs unless you want to modify or analyze them. In Figure 4-35 the lowpass filter SmartComponent is used in S parameter simulation circuit.
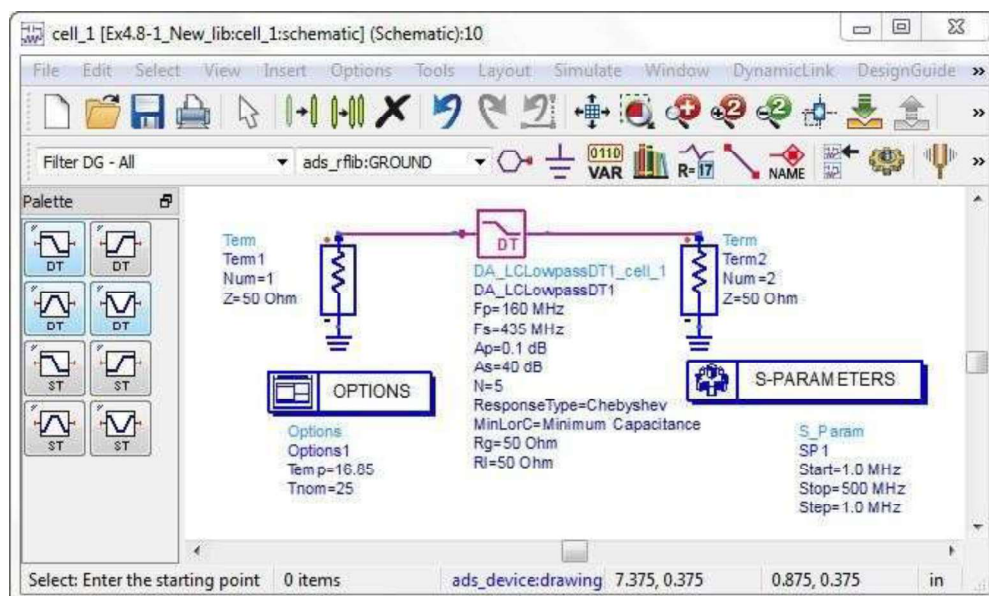
Figure 4-35 SmartComponent used in S parameter simulation circuit

RESULT:- Simulated Response of the low pass filter is shown in Figure 4-36.
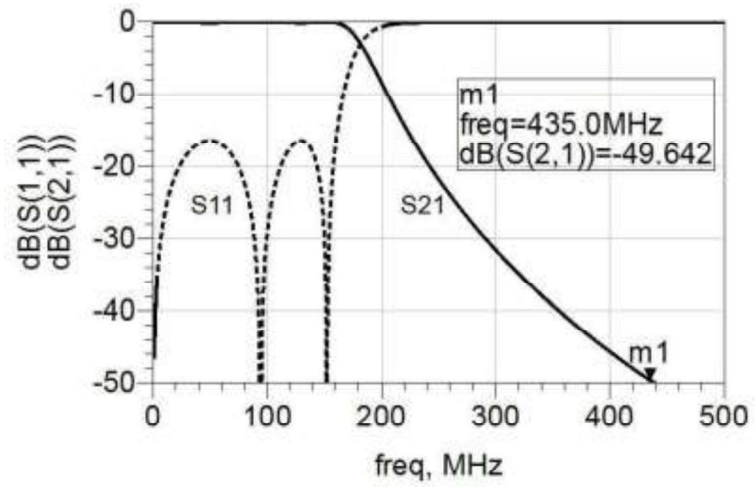
Experiment:-10

To study the microwave components in detail.

Theory:-

The followings microwave components are:

RECTANGULAR WAVE GUIDE

Wave guides are manufactured to the highest mechanical and electrical standards and mechanical tolerances.

L and S band wave guides are fabricated by precision brazing of brass-plates and all other wave guides are in extrusion quality.

W.G. sections of specified length can be supplied with flanges, painted outside and silver or gold plated in side.
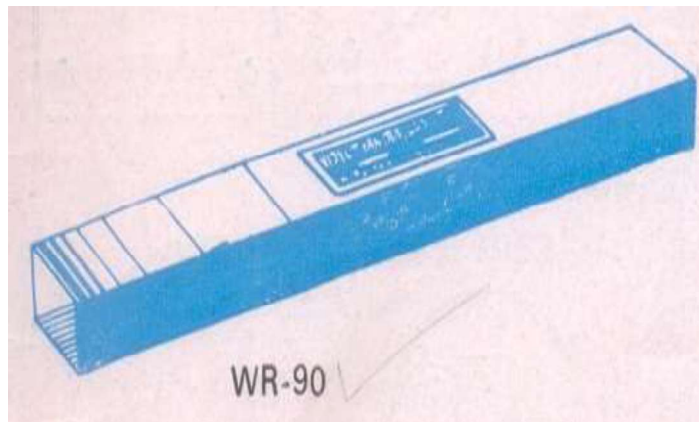
WR-90

SPECIFICATIONS X   Band

EIA No.   :   WR -90

Frequency   :   8.2 - 12.4GHZ

Width   :   2.286cm   Height : 1.1016cm Width : 2.54cm

Height   :   1.27cm   ± Tol. (μm):7.6   Material :Brass/Copper.

TUNABLE PROBE

Model 6055 Tunable probe is designed for use with model 6051 slotted sections. These are meant for exploring the energy of the EF in a suitably fabricated section of wave guide.

The depth of penetration into a wave guide - section is adjustable by the knob of the probe. The tip pick up the RF power from the line and this power is rectified by crystal detector, which is then fed to the VSWR meter or indicating instrument.

/Model No. : X6055 /Freq (Ghz) : 8.2 - 12.4 /output Connector : BNC(F) /Detector : IN23
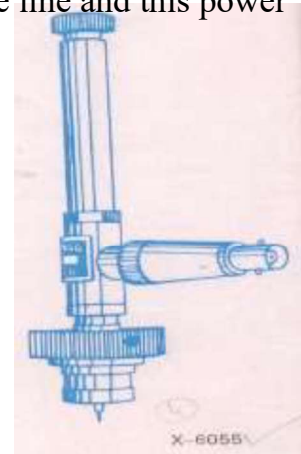
WAVE GUIDE DETECTOR MOUNT (TUNABLE)

Model 4051 Tunable Detector Mount is simple and easy to use instrument for detecting microwave power thro'a suitable detector. It consists of a detector crystal mounted in a section of a Wave guide and shorting plunger for matching purpose. The output from the crystal may be fed to an indicating instrument. In K and R bands detector mounts the plunger is driven by a micrometer.

Model No.	:	X - 4051
Freq.Range(Ghz)	:	8.2 - 12.4
O/PConnector	:	BNC (F)
Wave guidetype (WR-)	:	90
FlangeType(UG/U)	:	39
Detector	:	IN23
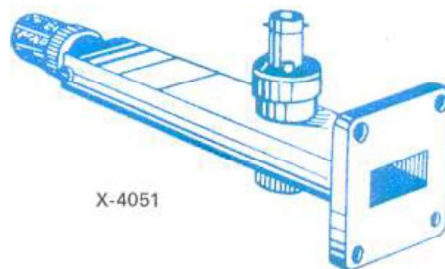
Model 2051 Klystron mounts are meant for mounting corresponding KLYSTRON MOUNT 2K25, 723A/B, 726A or RK - 5976 etc.

These consists of a section of wave guide flanged on one end and terminated with a movable short on the other end. An octal
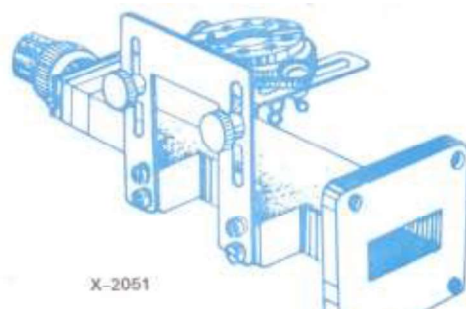
base with cable is provided for Klystron.

Model No. : X – 2051/ Freq. Range (GHz)
8.2 - 12.4/ WG Type (WR-) : 90
Flange Type (UG-/U): 39

CIRCULATORS

Model 6021 and 6022 are T and Y types of three port circulators respectively. These are precisely machined and assembled to get the desired specifications. Circulators are matched three port devices and these are meant for allowing Microwave energy to flow in clockwise direction with negligible loss but almost no transmission in the anti-clockwise direction.

Model No.   :   X - 6021
FrequencyRange(Ghz)   : 8.6 - 10.6 or 10.2 -
12.2 Min.Isolation(dB)   : 20
Max. Insertion Loss (dB):
                   0.4
Max.VSWR   :
                   1.20
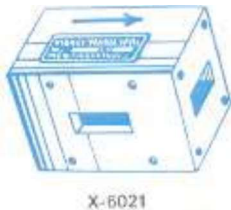
X-6021

X-6022

SLIDE SCREW TUNERS

Model 4041 slide screw tuners are used for matching purposes by changing the penetration and position of a screw in the slot provided in the centre of the wave guide.

These consists of a section of wave guide flanged on both ends and a thin slot is provided in the broad wall of the Wave guide. A carriage carrying the screw, is provided over the slot. A VSWR upto 20 can be tuned to a value less than 1.02 at certain frequency.
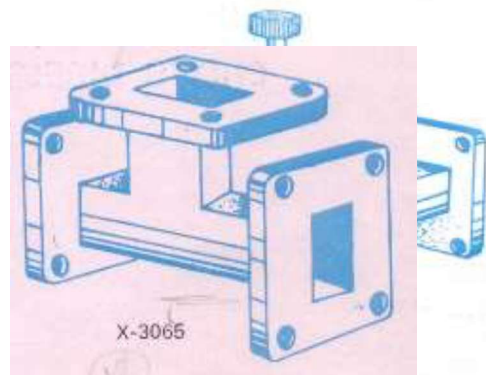
X-3065

Model No. : X – 4041/ Freq. Range (Ghz) :
8.2 - 12.4/WG Type (WR-) : 90
Flange type (UG/U) : 39

MULTIHOLE DIRECTIONAL COUPLERS

Model 6000 series Multihole directional couplers are useful for sampling a part of Microwave energy for monitoring purposes and for measuring reflections and impedance. These consists of a section of Wave guide with addition of a second parallel section of wave guide thus making it a four port network. However the fourth port is terminated with a matched load. These two parallel sections are coupled to each other through many holes, almost to give uniform coupling; minimum frequency sensitivity and high directivity. These are available in 3,6,10,20 and 40dB coupling.

Model No.	:	X - 6003
FrequencyRange(Ghz)	:	8.2 -12.4
Coupling (dB)	:	3,10,20,40
Directivity(dB)	:	35
Wave guidetype (WR-)	:	90
Flangetype(UG/U)	:	39

Model 3061 E - plane tee are series type T - junction and consists of three section of wave guide joined together in order to divide or compare power levels. The signal entering the first port of this T - junction will be equally dividing at second and third ports of the same magnitude but in opp.phase

E-PLANE TEE

Model No.	:	X - 3061
FrequencyRange(Ghz)	:	8.2 - 12.4
WG Type (WR-)	:	90
FlangeType(UG/U)	:	39

H - PLANT TEE

Model 3065 H - Plane Tee are shunt type T - junction for use in conjunction with VSWR meters, frequency - meters and other detector devices. Like in E-plane tee, the signal fed through first port of H - plane Tee will be equally divided in magnitude at second and third ports but in samephase.

Model No.	:	X - 3065
Frequency Range (GHz) :	8.2 - 12.4	
WG Type (WR-)	:	90
Flange Type (UG-/U)	:	39
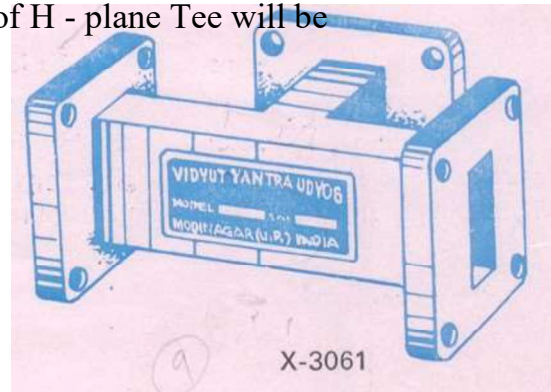
MAGIC TEE

Model 3045 E - H Tee consists of a section of wave guide in both series and shunt wave guide arms, mounted at the exact midpoint of main arm. Both ends of the section of wave guide and both arms are flanged on their ends. These Tees are employed in balanced mixers, AFC circuits and impedance measurement circuits etc. This becomes a four terminal device where one terminal is isolated from the input terminal.

Model No.	:	X - 3045
FrequencyRange(Ghz) :	8.2 - 12.4	
WG Type (WR-)	:	90
Flange Type (UR-/U) :	39	
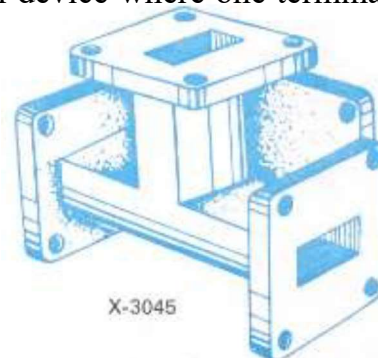
MOVABLE SHORT

Model 4081 movable shortsconsistsof a section of waveguide, flanged on one end and terminated with a movable shorting plunger on the other end. By means of this non contacting type plunger, a reflection co-efficient of almost unity

Model No.	:	X - 4081
Frequency Range (GHz) :	8.2 - 12.4	
WG Type (WR-)	:	90
Flange Type (UG-/U)	:	39
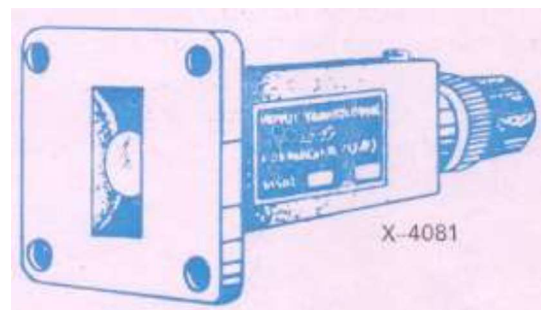
may beobtained.

MATCHED TERMINATION

Model 4000 are low power and non-reflective type of terminations. It consists of a small and highly dissapative taper flap mounted inside the centre of a section of wave guide. Matched Terminations are useful for USWR measurement of various waveguide components. These are also employed as dummy and as a precise reference loads with Tee junctions, directional couplers and other similar dividing devices.

Model No. : X - 4000, Freq. Range (Ghz) : 8.2 - 12.4 Max VSWR : 1.04
A$_V$ Power : 2W, WG Type (WR-) 90,
Flange Type (UG-/U) : 39
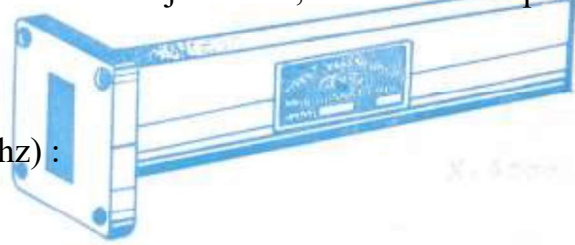
PYRAMIDAL WAVEGUIDE HORN ANTENNA

Model 5041 pyramidal Wave guide Horn antenna consists of waveguide joined to pyramidal section fabricated from brass sheet. The pyramidal section shapes the energy to concentrate in a specified beam. Wave guide horns are used as feed horns as radiators for reflectors and lenses and as a pickup antenna for receiving microwave power.

Model No.	:	X - 5041
FrequencyRange(Ghz)		8.2 -
	:	12.4
MaxVSWR	:	1.20
WG Type (WR-)	:	90
Flange Type (UG-/U) :		39
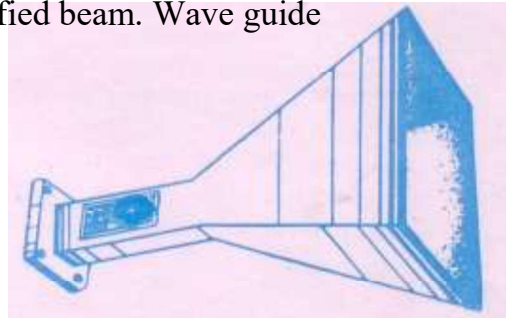
GUNN OSCILLATORS

Model 2151 Gunn Oscillators are solid state microwave energy generators. These consists of waveguide cavity flanged on one end and micrometer driven plunger fitted on the other end. A gunn-diode is mounted inside the Wave guide with BNC (F) connector for DC bias. Each Gunn osciallator is supplied with calibration certificate giving frequency vs
Model No. : X - 2152, Freq : 8.2 - 12.4 Ghz, Min output power : 10 MW
WG Type (WR-) : 90 Flange Type (UG-/U) : 39
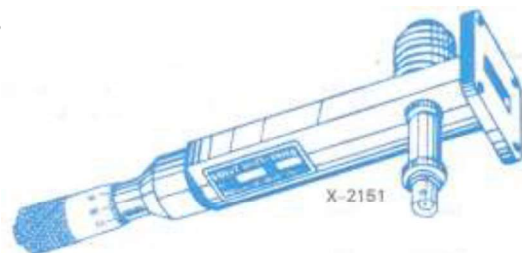
PIN MODULATORS

Model 451 pin modulators are designed to modulate the cw output of Gunn Oscillators. It is operated by the square pulses derived from the UHF(F) connector of the Gunn power supply. These consists of a pin diode mounted inside a section of Wave guide flanged on it's both end. A fixed attenuation vane is mounted inside at the input to protect the oscillator.

Model No.   :   X -
451 FrequencyRange(Ghz):
                     8.3   -
12.4 Max RF Power   :   1W
WG Type (WR-)   :   90
FlangeType(GHz)   :   39

X-451
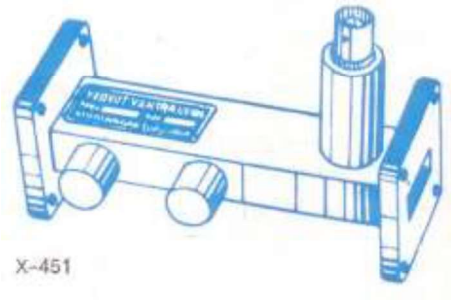
GUNN POWER SUPPLY

Model X-110 Gunn Power supply comprises of an regulated DC power supply and a square wave generator, designed to operate Gunn-Oscillator model 2151 or 2152, and pin modulators model 451 respectively. The DC voltage is variable from 0 - 10V. The front panel meter monitors the gunn voltage and the current drawn by the Gunn diode. The square wave of generator is variable from 0 - 10V. in amplitude and 900 - 1100 Hz in frequency. The power supply has been so designed to protect Gunn diode from reverse voltage application over transient and low frequency oscillations by the negative resistance of the Gunn-diode.

SPECIFICATIONS

AmplifierType   :   High gain tuned at onefrequency
Frequency   :   1000 Hz ±2%
Sensitivity   :   0.1 microvolt at 200 for fullscale
Bandwidth   :   25 - 30 cps
Range   :   70dB min in 10 dBsteps
Scale selector   :   NormalExpand
Gaincontrol   :   'Coarse' &'Fine'
Mainspower   :   230V, 50Hz

X-110
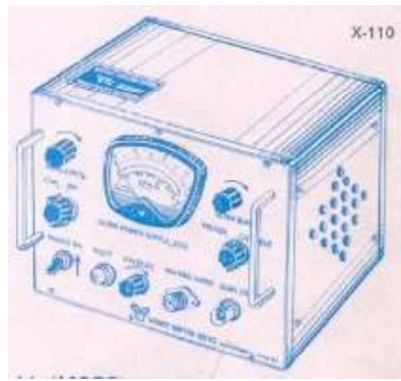
ISOLATORS

The three port circulators Model 6021 may be converted into isolators by terminating one of its port into matched load. these will work over the frequency range of circulators. These are well matched devices offering low forward insertion loss and high reverse isolation.

Model No.   :   X - 6022
Frequency Range(GHz) : 8.6 -   10.6   or
10.2 - 12.2 MinIsolation (dB)   :
                                            20
Max Insertion Loss (dB)
:
                                            0.
4
MaxVSWR   :
                                            1.
20

RESULT

Thus all the microwave components were studied in detail.

EXPERIMENT: - 11

To study the characteristics of Hybrid Rat Race Coupler by measuring the following parameters: Coupling factor and directivity ofcoupler.

I. *(i) EQUIPMENT ANDCOMPONENTS:*

1. APPARATUS

1. Klystron power supply SKPS – 610
2. Klystron tube2k25
3. Klystron mountxm-251
4. IsolatorXI-621
5. Frequency meterXF-710
6. Variable attenuator XA-520
7. Detector mountXD-451
8. Waveguide stands X4-535
9. VSWR meter SW-215
10. Movable short XT-481
11. Matched termination XL-400
12. Slotted line XS-651
13. Tunable probe XP-655
14. Hybrid Rat Race coupler

(ii) DESCRIPTION OF THE EQUIPMENT:

1. Klystron power supply SKPS – 610

The model Klystron Power Supply SKPS-610 is general purpose laboratory power supply which is specially designed to use for reflex klystron tubes of S to X band frequency range. It is reliable power source with very high regulation and low ripple contents.

The klystron power supply SKPS-610 has built in modulation facilities of amplitude and frequency modulation. Amplitude modulation can be applied with 0-110V (p.p.) Square wave and with frequency of 500 Hz to 2.5 KHz. This amplitude modulation is generally used along with VSWR measurements in

slotted line technique. Frequency modulation is used for direct study of klystron modes on the Oscilloscope. Pure carrier wave operation and in external modulation facilities is also provided with the instrument for generalized use.

The klystron power supply also provides all the other D.C. Voltages required for operation of reflex klystron tube such as beam, heater and reflector voltage. The ranges of all these voltages are given in the specification data sheet.

2. Klystron tube2k25

The klystron tube 2k25 is a single cavity variable frequency microwave generator of low power and low efficiency. It consists of an electron gun, a filament surrounded by cathode and a focusing electrode at cathode potential. The electrons emitted by the cathode travel towards the reflector through an anode kept at higher potential compared to the cathode. When they approach the anode, the electrons form bunches and the bunches ultimately return towards the anode cavity after traveling a small distance towards the reflector. The power is taken from the anode reentrantcavity.

3. IsolatorXl-621

It is a two port device which provides very small amount of attenuation for transmission from port 1 to port 2 and provides maximum attenuation for transmission from port 2 to port 1. It is very much desirable when we want to match a source with variable load. It allows power flow only from the generator towards the load and suppresses any reflectedpower.

4. Frequency meterXF-710

It is also called wave meter. Usual construction of it consist a cylindrical cavity mounted on a shaft. By rotating the shaft the volume in the cavity is changed and it becomes resonant and gives minimum impedance at the resonant frequency. The scale calibrated and the resonant frequency can be directly read from the scale after observing a dip in the output meter.

5. Variable attenuatorXA-520

This is a wave guide piece having a groove on the lateral side. By rotating the screw the depth of penetration of resistive pad changes, there by introducing some attenuation. Attenuators are commonly used for measuring power game or loss in dB for providing isolation between instruments for reducing the power input to a particular stage to prevent over loading and also for providing the signal generators with means of calibrating there outputs accurately. Variable attenuator

provide continuous or step wise variableattenuation.

6. VSWR meterSW-215

AVSWR meter basically consists of a high gain, high Q low noise voltage amplifier normally tuned at a fixed frequency at which the microwave signal is modulated. The VSWR meter uses the detector signal out of the microwave detector as its input, amplifies the same and provides the output on the calibrated volt meter. The meter itself can be calibrated in terms of VSWR

7. MHD CouplerXK-620

It is a wave guide used for the measurement of low standing wave ratios, to sample a small amount of powers. It consists of two transmission lines the main arm and auxiliary arm, electro-magnetically coupled to each other. Here, for a two hole directional coupler the two holes are at a distance of $\lambda g/4$.

II. THEORY:

A directional coupler is a device with which it is possible to measure the incident and reflected wave separately.

i. It consists of two transmission lines the main arm and auxiliary arm, electro-magnetically coupled to each other. The power entering the main arm gets divided between port 2 and 3 and almost no power comes out in port 4. Power entering at port 2 is divided between port 1 and 4.

The coupling factor is defined as:

$$Coupling\ (dB)\ =\ 10\ log10 \left| \frac{P1}{\overline{P3}} \right| \text{ where port 2 is terminated}$$

$$Isolation\ (dB)\ =\ 10\ log10 \left| \frac{P2}{P3} \right| \text{ where Port1 is matched.}$$

ii. With built in termination and power entering at port 1, the directivity of coupler is a measure of separation between incident and reflected wave. Directivity is measured as follows.

$$Hence\ \textbf{Directivity D (dB)} = 10\ log10 \left| \frac{P2}{P1} \right| = I\text{-}C$$

Main line VSWR is SWR measured, looking into the main line input terminal when the matched loads are placed at all other ports.

iii. Auxiliary line VSWR is SWR measured in the auxiliary line looking into the output terminal when the matched loads are placed on other terminals. Main line insertion loss is the attenuation introduced in the transmission line by insertion of coupler, it is definedas:

$$Insertion\ loss\ (dB)\ =\ 10\ log10\left\lceil\left|\frac{P1}{P2}\right|\right\rceil$$

III. BLOCKDIAGRAM:

Set up 1: Reference level **"x dB"**

```
┌─────────────┐
│ Klystron    │
│ power       │
│ supply      │
│  SKPS-610   │
└─────────────┘
       │
       ▼
┌─────────────┐                                    ┌─────────────┐
│ Klystro     │                                    │ VSWR        │
│ n           │                                    │ meter       │
│ Mount       │                                    │ SW-115      │
│ XM-         │                                    │             │
│ 251         │                                    └─────────────┘
└─────────────┘                                          ▲
       │                                                 │
       ▼                                                 │
┌───────────┐   ┌───────────┐   ┌───────────┐   ┌───────────┐
│ Klystro   │   │ Isolat    │   │ Variable  │   │ Detect    │
│ n         │ → │ or X1-    │ → │ Attenuat  │ → │ or        │
│ tube2K    │   │ 621       │   │ or   XA-  │   │ Mount     │
│ 25        │   │           │   │ 520       │   │ XD-       │
│           │   │           │   │           │   │ 451       │
└───────────┘   └───────────┘   └───────────┘   └───────────┘
```
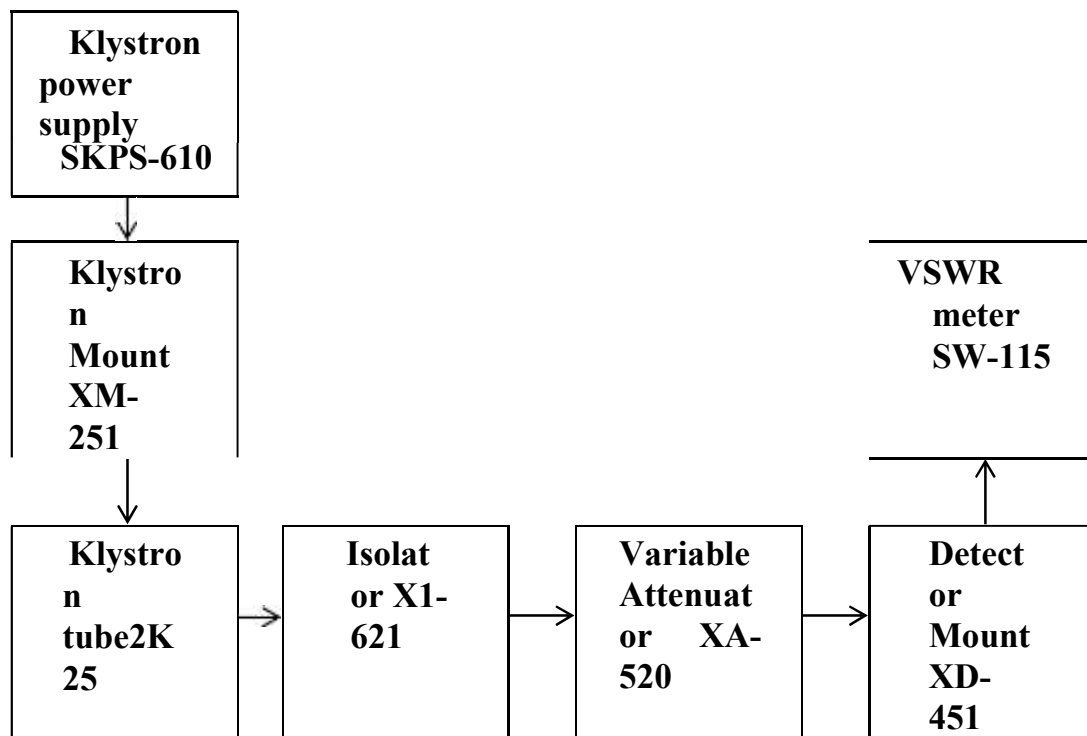
Set up 2: To determine the coupling factor 'X-Y dB"

Klystron power supply SKPS-610

↓

Klystron Mount XM-251

↓

Klystron tube2K25

Matched Termination XL-400

VSWR meter SW-115

Isolator X1-621	→	Variable Attenuator XA-520	→	Hybrid rat race coupler	→	Detector Mount XD-451
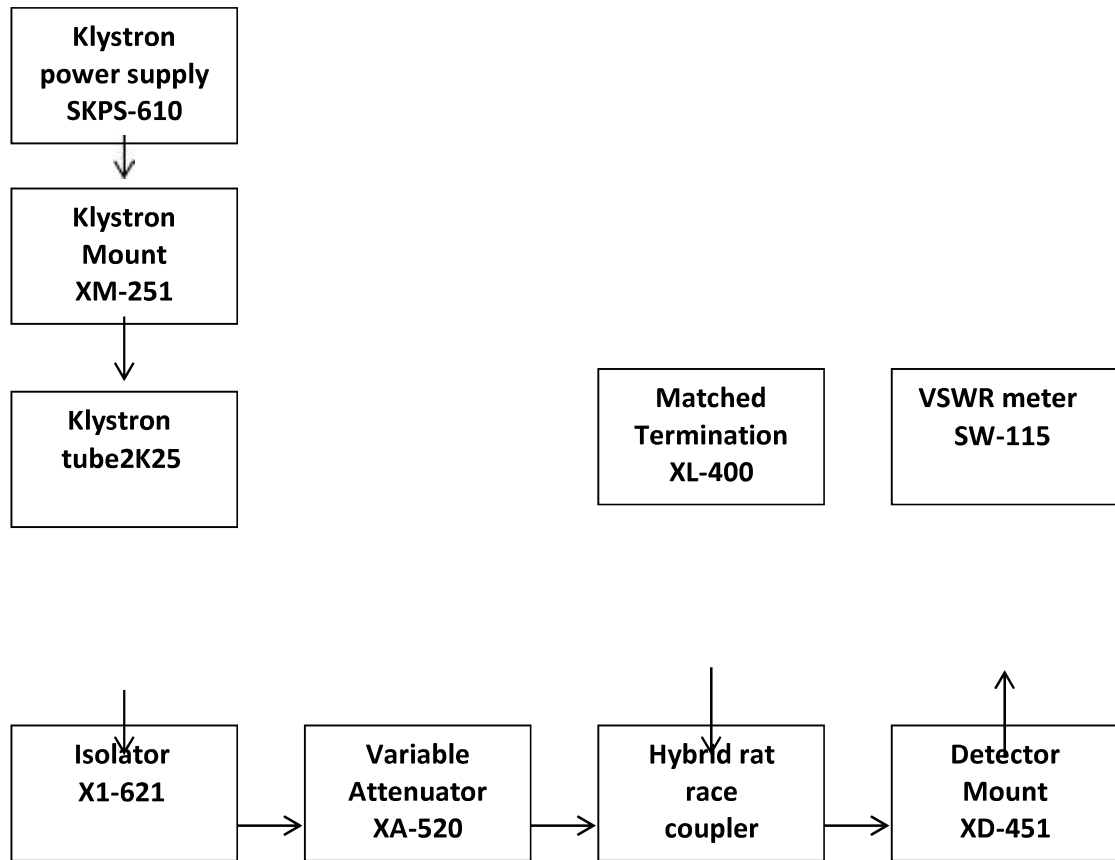
Set up 3: To determine insertion loss **"X-Z dB"**

Klystron power supply SKPS-610

↓

Klystron Mount XM-251

↓

| Klystron tube2K25 | →

→

→ | VSWR meter SW-115 |
|---|

Isolator X1-621	Variable Attenuator XA-520	Matched Termination XL-400	Detector Mount XD-451
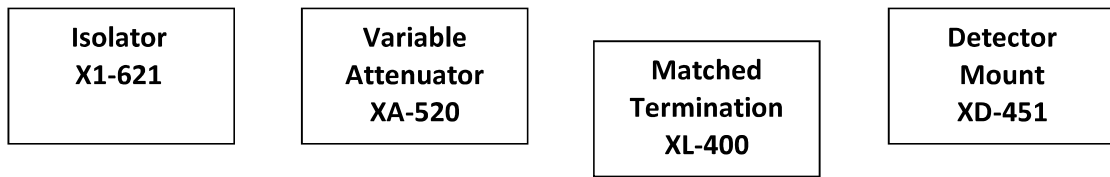
Set up 4: To determine isolation "X-Y dB"

```
Klystron
power supply
SKPS-610
    |
    v
Klystron
Mount
XM-251
    |
    v
```

Matched Termination XL-400	VSWR meter SW-115

```
Klystron
tube2K25
    |
    v
```

Isolator X1-621	→	Variable attenuator XA-520	→	Rat Race coupler	→	Detector Mount XD-451
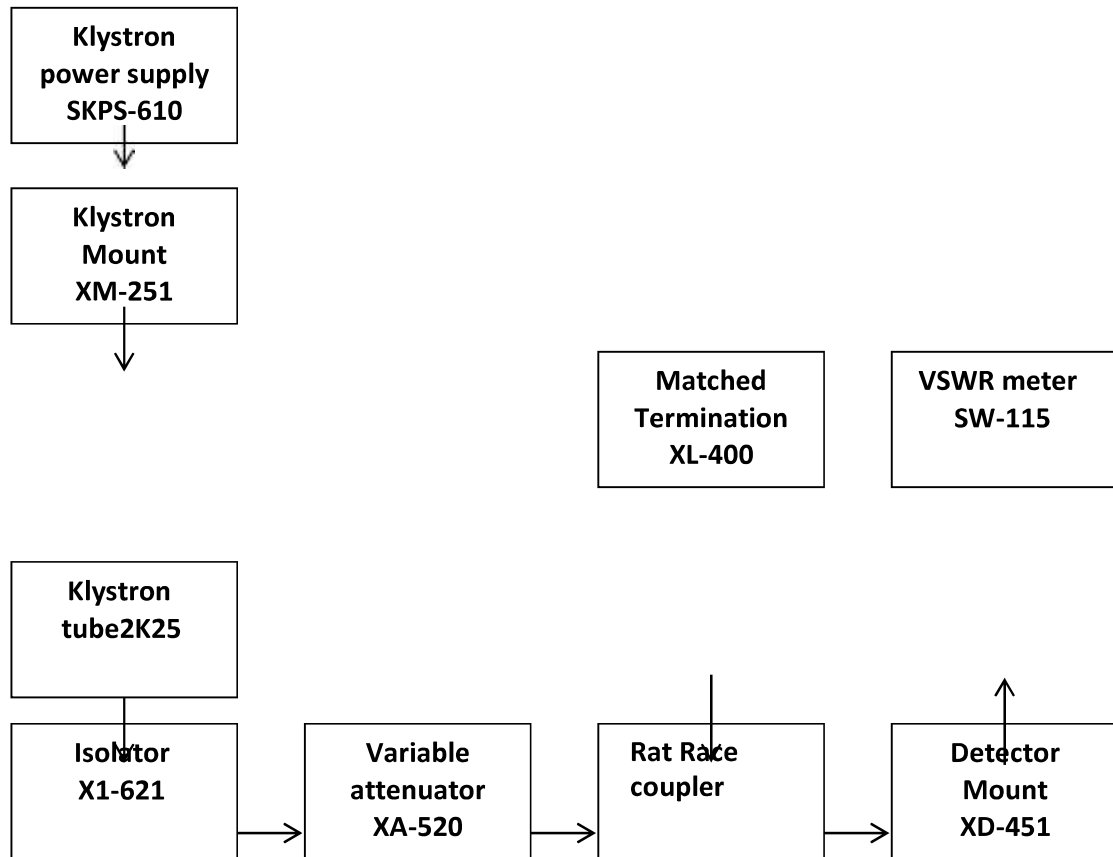
IV. PROCEDURE:

i. Setup the components and equipments as shown infigure.

ii. Energize the microwave source for particular frequency ofoperation.

iii. Remove the multi-hole directional coupler and connect the detector mount to the frequency meter. Tune the detector for maximumoutput.

iv. Set any reference level of power on VSWR meter with the help of variable attenuator, gain control knob of VSWR meter and note-down the reading.

v. Insert the directional coupler as shown in figure with detector to the auxiliary port 3 and matched termination to port 2, without changing the position of variable attenuator and gain control knob of VSWRmeter.

vi. Note down the reading on VSWR meter on the scale with the help of range

–db switch if required. Let it be'**Y**'

vii. Calculate coupling factor which willbe

$C (dB) = X – Y$

viii. Now carefully disconnect the detector from auxiliary port 3 and match termination from port 2 without disturbing thesetup.

ix. Connect the matched termination to auxiliary port 3 and detector to port 2 and measure the reading on VSWR meter. Suppose it is**Z.**

x. Computeinsertion loss **X-Z indB**.

XI. Repeat steps from 1 to4.

Xii. Connect the directional coupler in reverse directions i.e. port 2 to frequency meter   side. Matched termination to port 1 and detector mount to port 3. Without disturbing the position of the variable attenuator and gain control knob of VSWRmeter.

Xiii. Measure and note down the reading on VSWR meter. Let it be**Yd.**

X-Y gives isolation**I(dB).**

Xiv. Compute the directivityas

xv. *Repeat the same for otherfrequencies.*

VI. OBSERVATIONS:

Input Port	Output Port	Matched Terminati on	S-parameters
1	2	3	S12
1	3	2	S13
2	1	3	S21
2	3	1	S23
3	1	2	S31
3	2	1	S32

VII.CALCULATIONS:

Coupling (dB) = 10 log10 $\left| \frac{P1}{} \right|$ where port 2 is terminated

$\lfloor P3 \rfloor$

C (dB)= X-Y

Isolation (dB) = 10 log10 $\left| \frac{P2}{} \right|$ where Port1 is matched.

$\lfloor P3 \rfloor$

I (dB) = X – Yd

Hence Directivity D (dB) = $10 \log_{10} \left| \left[\dfrac{P2}{\overline{P1}} \right] \right|$ = I-C

VIII.   RESULT:

TheHybrid rat race coupler characteristic has been studied by measuring its scattering parameters.

i. Couplingfactor
ii. Isolation
iii. Insertionloss
iv. Directivity

IX. *INFERENCES:*

By knowing the power output at various ports we can measure various loses in directional couplers.

X. *PRECAUTIONS:*

i. Keep all the knobs in minimum position before going to switch 'ON' the power supply of VSWR / Klystron powersupplies.
Note: For klystron power supply "HT" should be 'OFF' before switching 'ON' the main supply.
ii. Beam knob should be completely in anticlockwise direction and repeller voltage knob should be completely clockwisedirection.
iii. Switch on the main supply and give some warm up time to get current / accurate reading.
iv. After the completion of experiment, before going to switch off the mains keep all the knobs in minimum position (i.e.) as those are in rule1.
v. If the main supply failed in the middle of the experiment, come to 1st condition (i.e.) keep all the knobs in minimum positions and switch off mainswitches.
vi. Don'tincreasetherepellervoltagemorethan-**70V**(i.e.)itshouldbebetween-**70Vto**
-270V.

XI. APPLICATIONS:

i. It is used to measure incident and reflectedpowers

ii.   It can sample a small amount of micro power for measurementpurposes.
iii.   Provides signal path to a receiver.

XII. EXTENSIONS:

This experiment can be extended to four port directional couplers.

XIII. TROUBLESHOOTING:

FAULT   **DIAGNOSIS**

1. No outputat ports   :   Vary the repeller
                                         voltage slowly
                                         Check for
                                         properalignment

2. Meterreadingfluctuating   :   Place the power dB knob to 20 or 30dB

COLLEGE OF TECHNOLGY AND ENGINEERING
MAHARANA PRATAP UNIVERSITY OF AGRICULTURE & TECHNOLGY, UDAIPUR

DEPARTMENT OF ELECTRONICS AND COMMUNICATION ENGINEERING

LAB MANUAL

EC323 – Digital Communication

PREPARED BY

Mr. VISHWAPRIYA

EC323 Digital Communication Engineering Lab Manual

TABLE OF CONTENT

EXPERIMENT NO. -1

AIM/OBJECTIVE: Demonstrate the time division multiplexing of signals and de-multiplexing of signals

APPARATUS:

1. Time division multiplexer and de-multiplexer trainer kits.
2. Coaxial cable.
3. Patch cords.

PROCEDURE:

MULTIPLEXER:
1. Observe the AF generator-1 output and note down the amplifier and frequency.
2. Observe the AF generator-2 output and note down the amplitude and frequency.
3. Connect the AF generator and 2 outputs to CH1 and CH2 of TDM multiplexer.
4. Observe and connect the clock generator output to the control input of the TDM multiplexer (it acts like selection line for MUX).
5. Observe the TDM output in storage oscilloscope.

DEMULTIPLEXER:
1. Using coaxial cable connect the TDM de-multiplexer.
2. Connect the clock generator output in de-multiplexer trainer to the control input of the TDM de-multiplexer.
3. Observe the de-multiplexed signals at CH1 and CH2.
4. Connected the CH1 and CH2 outputs to low pass filter and amplifier and note down the outputs.

Circuit Diagram:

MULTIPLEXER:

CH1 INPUT — 14

CD 4052   13 • → TDM-PAM OUTPUT

CH2 INPUT • — 12

10   9

CONTROL INPUT •

500HZ – 15KHZ
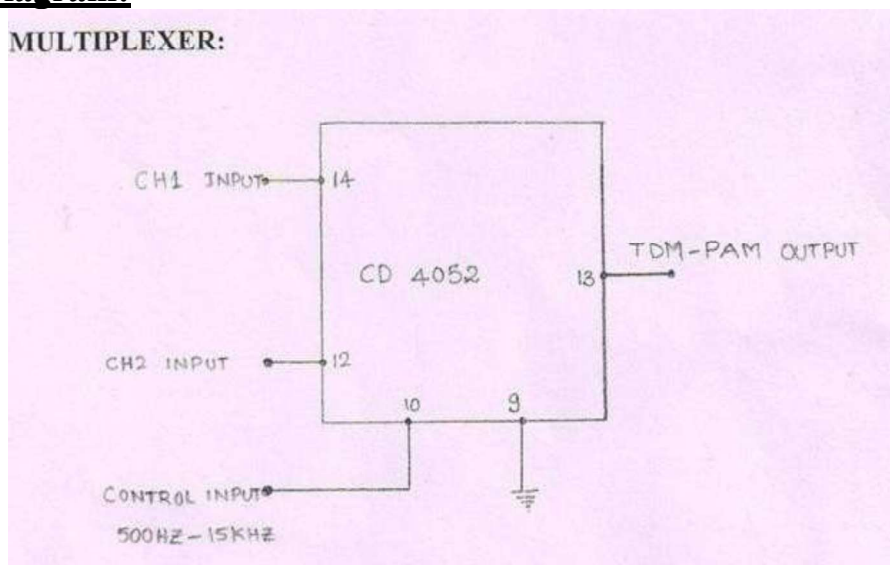
DEMULTIPLEXER:
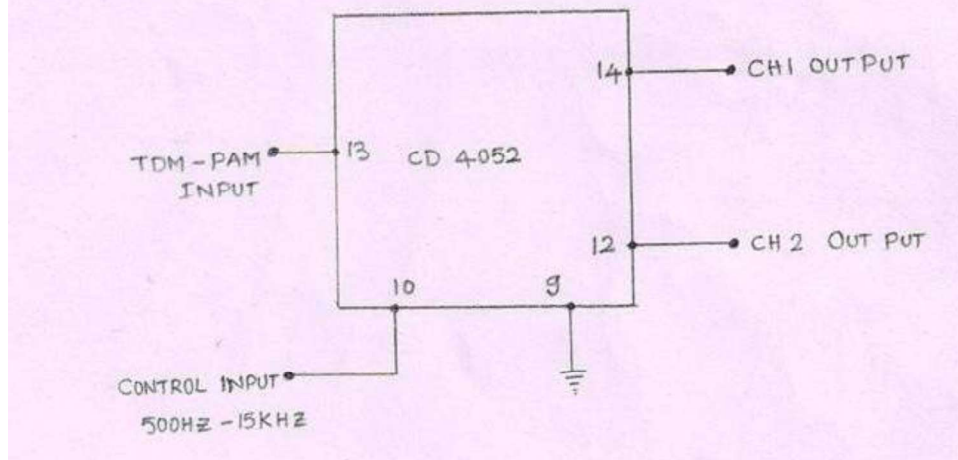
Wave form :
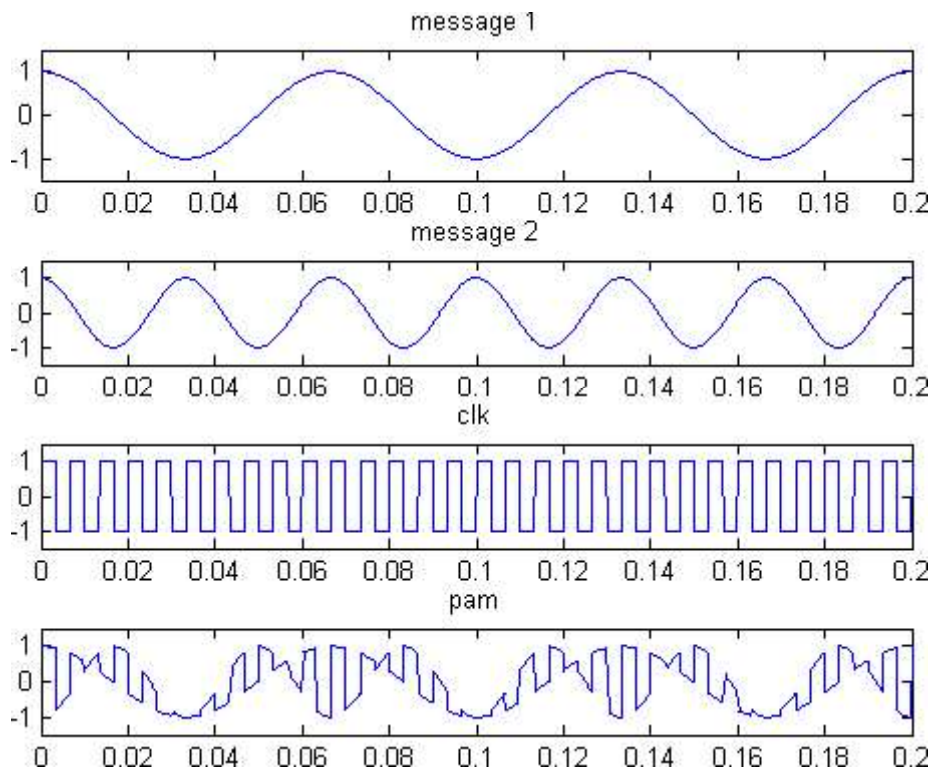
Result & Discussion:

The waveform was traced and compared with the designed theoretical one.

Experiment No. -2

Aim/Objective: Demonstrate the generation and detection of PCM signal and to measure the output parameters.

APPARATUS:

1. PCM Modulator trainer
2. PCM Demodulator trainer
3. Storage Oscilloscope/ Dual Trace Oscilloscope
4. Digital multi-meter.
5. Co-axial cables (standard accessories with trainer)
6. Patch chords

CIRCUIT DIAGRAM:

PCM Modulator

PCM Demodulator
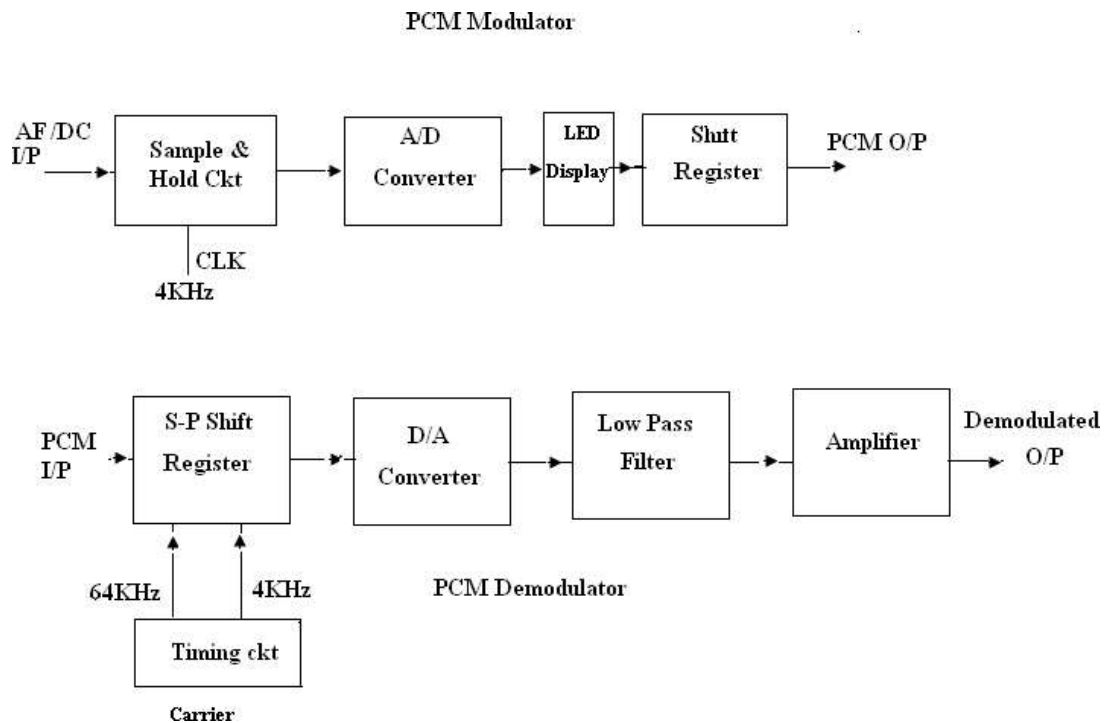
THEORY:

PCM: In pulse code modulation (PCM) only certain discrete values are allowed for the modulating signals. The modulating signal sampled, as in other forms of pulse modulation. But any sample falling within a specified range of values is assigned a discrete value. Each value is assigned a pattern of pulses and the signal transmitted by means of this code. The electronic circuit that produces the coded pulse train from the modulating waveform is termed a coder or encoder. A suitable decoder must be used at the receiver in order to extract the original information from the transmitted pulse train.

PROCEDURE:

1. Study the theory of operation thoroughly.
2. Connect the trainer (Modulator) to the mains and switch on the power supply.
3. Observe the output of the AF generator using CRO; it should be a Sine wave of 200Hz frequency with 3Vpp amplitude.
4. Verify the output of the DC source with multi-meter / scope, output should vary 0 to +5v.
5. Observe the output of the Clock generator using CRO, they should be 64 KHz and 8KHz frequency of square wave with 5Vpp amplitude.
6. Note: These clock signals are internally connected the circuit so no external connections are required.
7. Connect the trainer (De Modulator) to the mains and switch on the power supply.
8. Observe the output of the clock generator using CRO, it should be 64 KHz square wave with 5Vpp amplitude.

PCM Operation (with DC input)

Modulation:

1. Set DC source to some value say 1 V with the help of multi-meter and connect it to the A/D converter input and observe the output LED's.
2. Note down the digital code i.e. output of the A/D converter and compare with the theoretical value

---

Department of Electronics and Communication Engineering, CTAE, Udaipur

Theoretical value can be obtained by:

A/D input voltage

$$\frac{\text{A/D input voltage}}{\text{1 LSB value}} = X_{(10)} = Y_{(2)}$$

1 LSB value

Where , 1 LSB value = $V_{ref}/ 2^n$, Since V_{ref}= 5V and n= 8

1 LSB Value = 0.01953

Example:

A/D input voltage = 1 V

$$= 51.2_{(10)}$$

$$= 00110011_{(2)}$$

So digital output is 00110011

3. Keep CRO in dual mode. Connect one channel to 8KHz signal (which is connected to the shift register) and another channel to the PCM out put

4. Observe the PCM output with respect to the 8 KHz signal and sketch the waveforms. Compare them with the given waveforms

    Note: From this wave form you can observe that the LSB bit enters the output first.

Demodulation:
1. Connect PCM signal to the demodulators (S-P Shift register) from the PCM modulator (AET- 68M) with help of coaxial cable (supplied with the trainer)

2. Connect clock signal (64 KHz) from the transmitter to the receiver using coaxial cable.

3. Connect transmitter clock to the timing circuit

4. Observe and note down the S-P shift register output data and compare it with the transmitted data (i.e. output A/D converter at transmitter). It will be noticed that the output of the S-P shift register is following the A/D converter output in the modulator. Observe D/A converter output (demodulated output) using multi-meter /scope and compare it with the original signal and you can observe that there is no loss in information in process of conversion and transmission.

5. Similarly you can try for different values of modulating signal voltage.

Sample work sheet:

    a.Modulating signal   : 1V

    b. A/D output (theoretical)   : 00 11 00 $11_{(2)}$

    c. A/D output (practical)   : 00 11 00 $11_{(2)}$

    4. S-P output   : 00 11 00 $11_{(2)}$

    5. D/A Converter output   : 1V

PCM Operation (with AC input)

Modulation:

1. Connect AC signal of $2V_{PP}$ amplitude to Sample & Hold circuit.
2. Keep the CRO in dual mode. Connect one channel to the AF signal and another channel to the sample & hold output. Observe and sketch the sample & hold output.
3. Connect the sample and hold output to the A/D converter and observe the PCM output using storage oscilloscope/ DTO
4. Observe PCM output by varying AF signal voltage.

Demodulation:

1. Connect PCM signal to the demodulator input (S-P shift register) from the PCM modulator with the help of coaxial cable (supplied with the trainer)
2. Connect clock signal (64KHz) from the transmitter to the receiver using coaxial cable
3. Connect transmitter clock to the timing circuit.
4. Keep CRO in dual mode. Connect CH 1 input to the sample and hold output and CH 2 input ot the D/A converter output.
5. Observe and sketch the D/A output.
6. Connect D/A output to the LPF input.
7. Observe output of the LPF/Amplifier and compare it with the original modulating signal.
8. From above observation you can verify that there is no loss in information (modulating signal) in conversion and transmission process.
9. Disconnect clock from transmitter and connect to local oscillator (i.e. clock generator output from De-Modulator) with remaining setup as it is. Observe D/A output and compare it with the previous result. This signal is little bit distorted in shape. This is because lack of synchronization between clock at

transmitter and clock at receiver.

Note: You can take modulating signals from external sources. Maximum amplitude should not exceed 4V in case of DC and $3V_{PP}$ in case of AC (AF) signals.

EXPECTED WAVEFORMS:

PCM wave forms / timing diagram of DC input

Input signal

+1 v

Clock 2 (4 KHz)

+ 5v

Gnd

1 1   1 1   1 1   1 1

0 0   0 0   0 0   0 0

A/D Output   0101 (Led Indication MSB to LSB)

DPCM signal

S – P : Register output 0101 (Led Indication MSB to LSB)

D/A output (demodulated signal)

RESULT:

Department of Electronics and Communication Engineering, CTAE, Udaipur
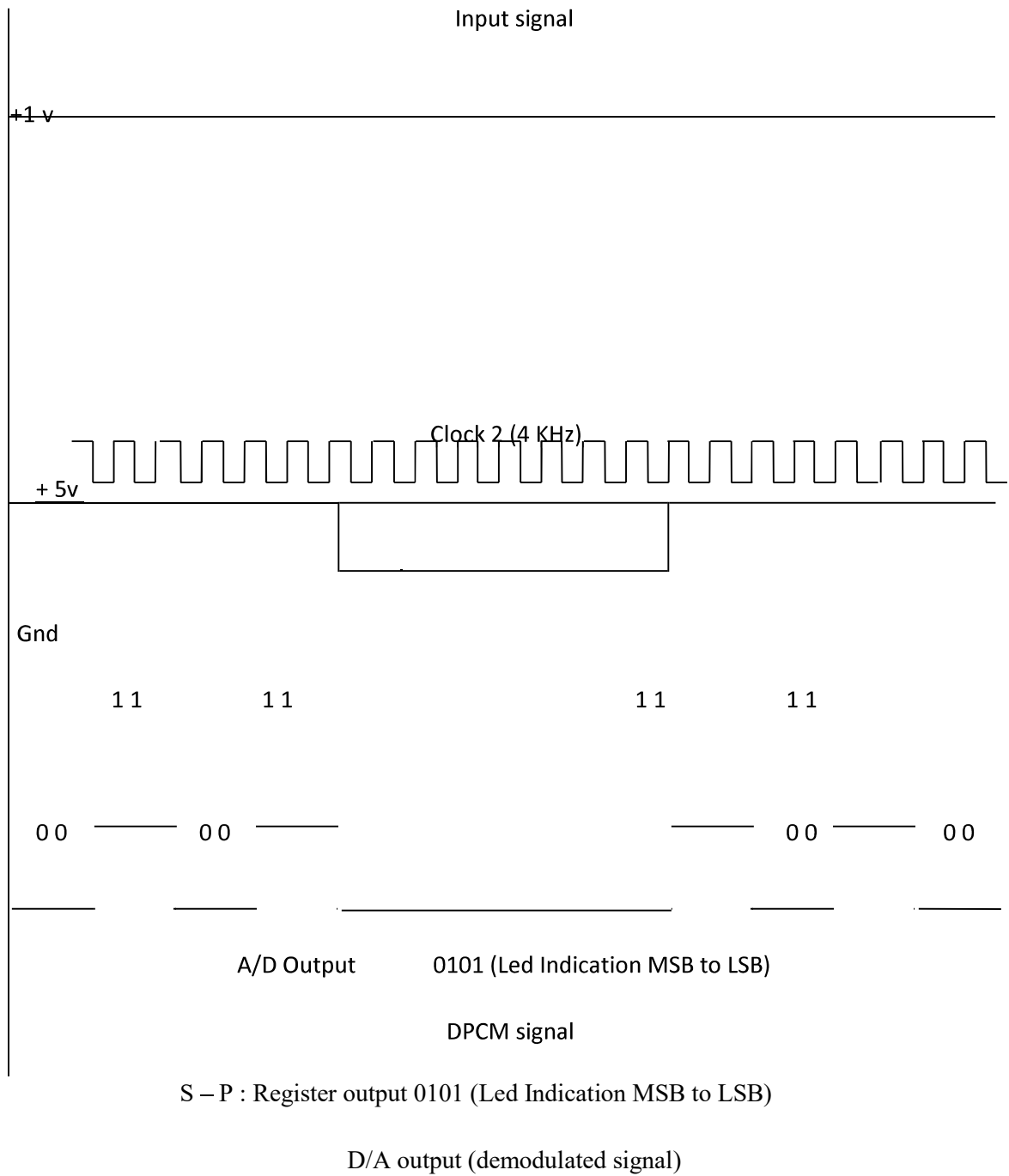
Experiment No. -3

AIM: Demonstrate the generation of Delta modulator and demodulator signal and to measure the output parameters

APPARATUS:

1. Trainer Kit of Delta Modulation & Demodulation.
2. CRO
3. Function Generator
4. Patch cords, Probes.

PROCEDURE:

1. Switch ON the experimental board
2. Connect Clock Signal to the Delta Modulator circuit.
3. Connect Modulating Signal to the Modulating signal input of the Delta Modulator and observe the same on channel 1 of a Dual Trace Oscilloscope.
4. Observe the Delta Modulator output on channel II.
5. Observe Slope Over Load noise by increasing the amplitude of Modulating signal and also observe granular noise.
6. Connect this Delta Modulator output to the Demodulator
7. Also connect the clock signal to the demodulator.
8. Observe the demodulator output with and without RF filter on CRO.

CIRCUIT DIAGRAM
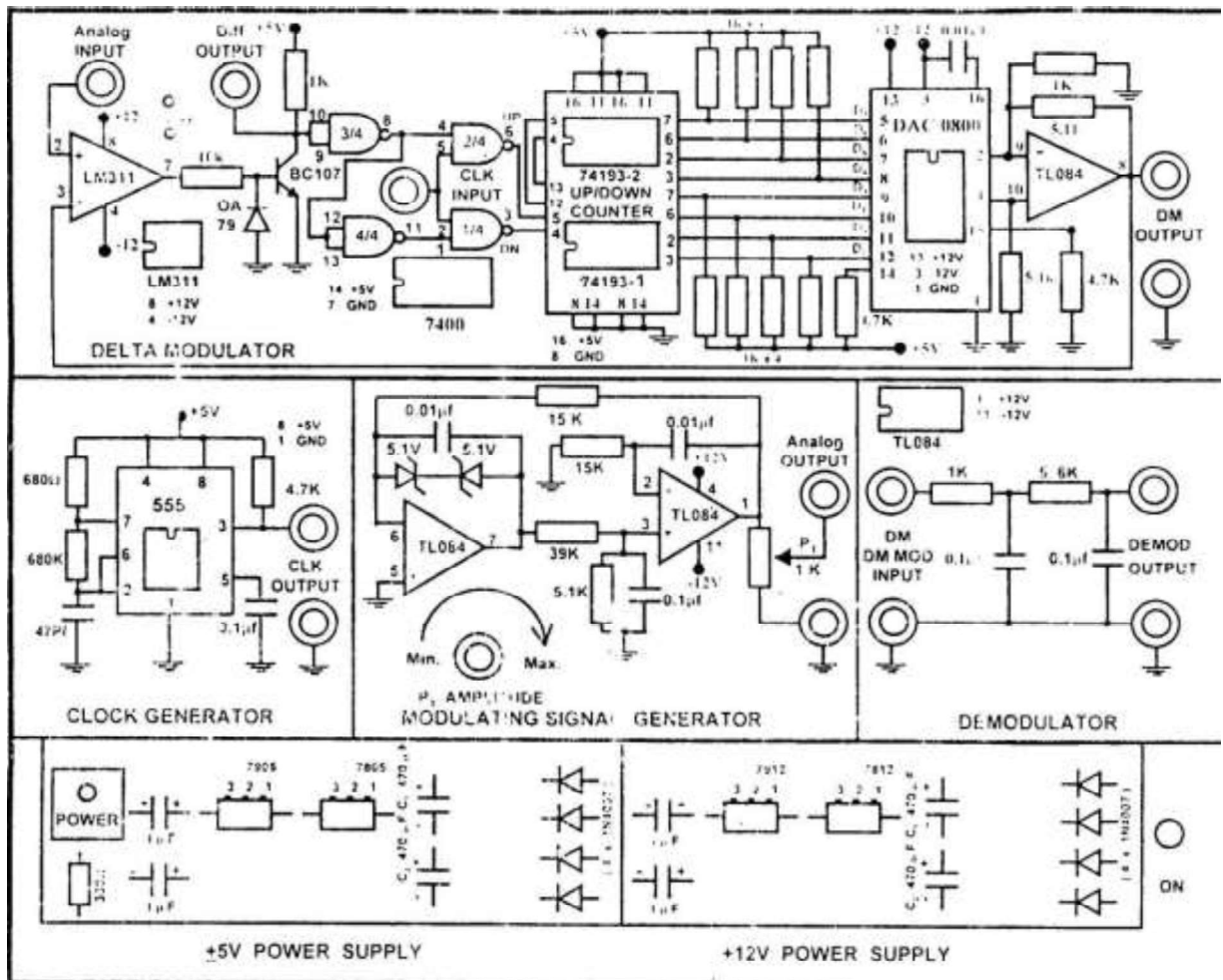
WAVEFORM:
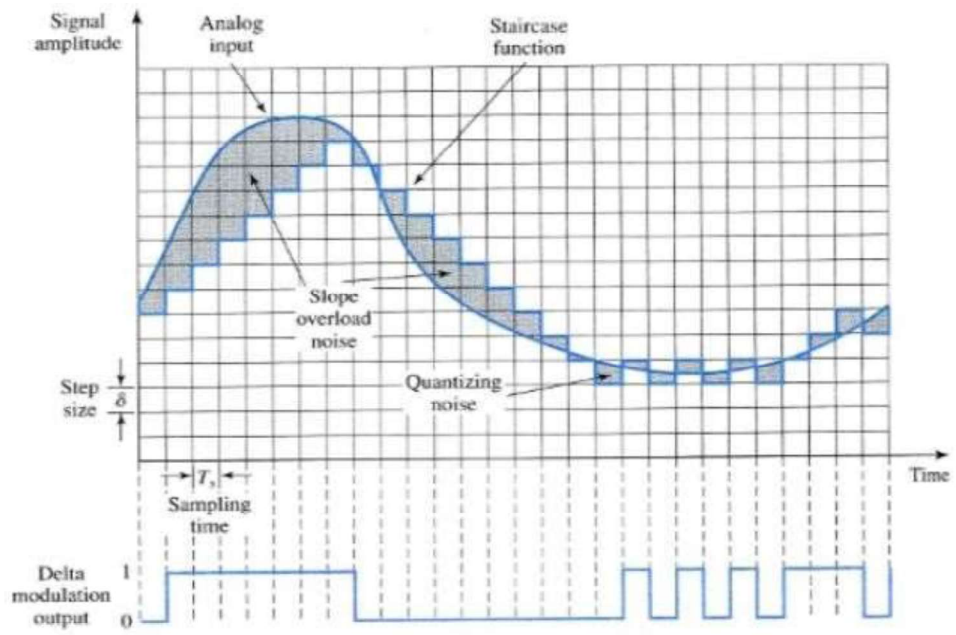
RESULT: The waveform was traced and compared with the designed theoretical one

Experiment No. -4

AIM: To Demonstrate the generation and detection of ASK, PSK, FSK, signal and to measure the output parameters.

Amplitude Shift Keying

APPARATUS:
1. ASK Modulation & Demodulation Trainer kit
2. C.R.O. 30 MHZ Dual Channel
3. Patch cords

PROCEDURE:

Modulation:

1. Connect the sine wave 500 KHz from the carrier generator to the carrier input of the modulator.
2. And also connect the Data clock D1 i.e., modulation signal to the modulation input.
3. Switch ON the power supply.
4. Observe the output.
5. By varying the gain pot P3 observe the ASK output.
6. Adjusting the carrier offset and modulation offset we can observe the ASK output.
7. By changing the carrier signal 1MHz and different data clocks D2, D3, D4 observe the output.

Demodulation:

1. Connect ASK output to the rectifier input and observe the waveform.
2. Now connect rectifier output to the low pass filter input and observe the output.
3. Connect LPF output to the data squaring circuit input and observe the demodulation output waveform.
4. By changing the different data clocks and observe the demodulation output.

BLOCK DIAGRAM

MODULATOR:
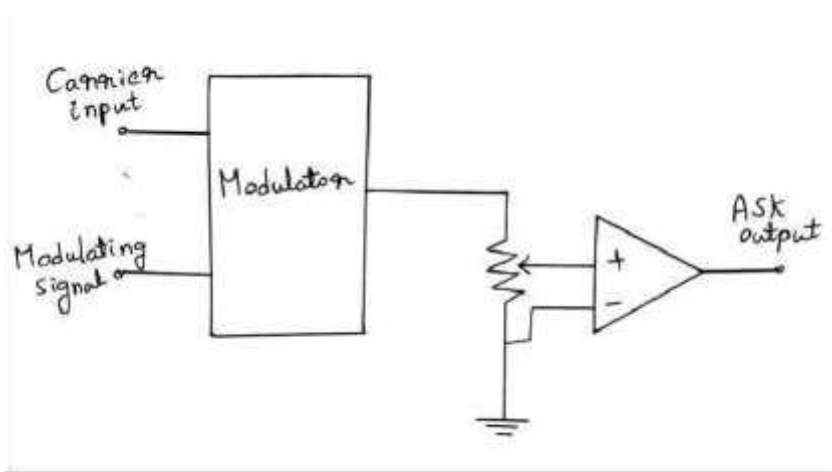
DEMODULATOR:

EXPECTED GRAPHS

RECTIFICATION at TP13   0V

LOW PASS FILTER IN at TP15   0V

DATA SQUARING at TP17   0V
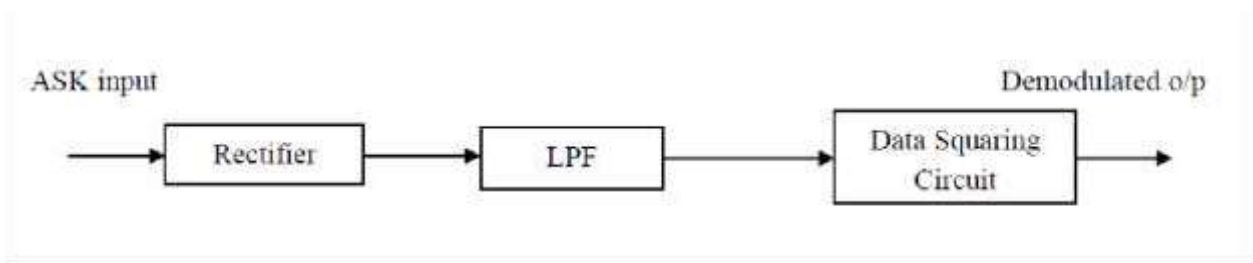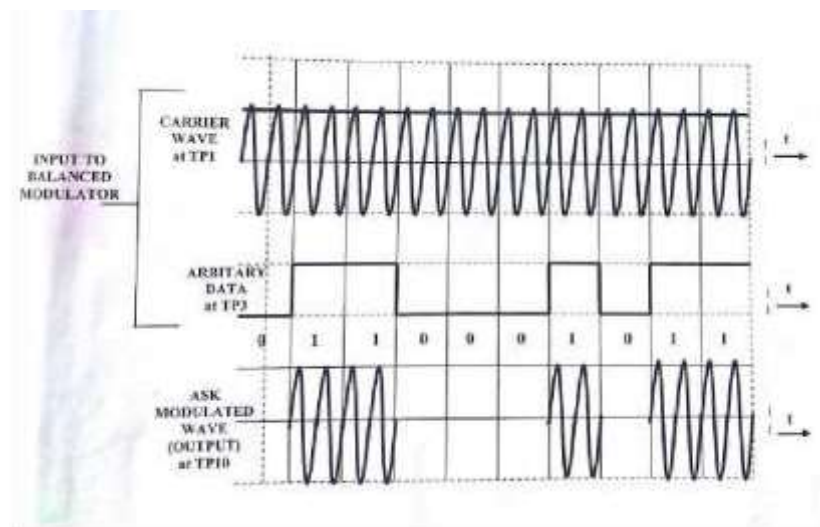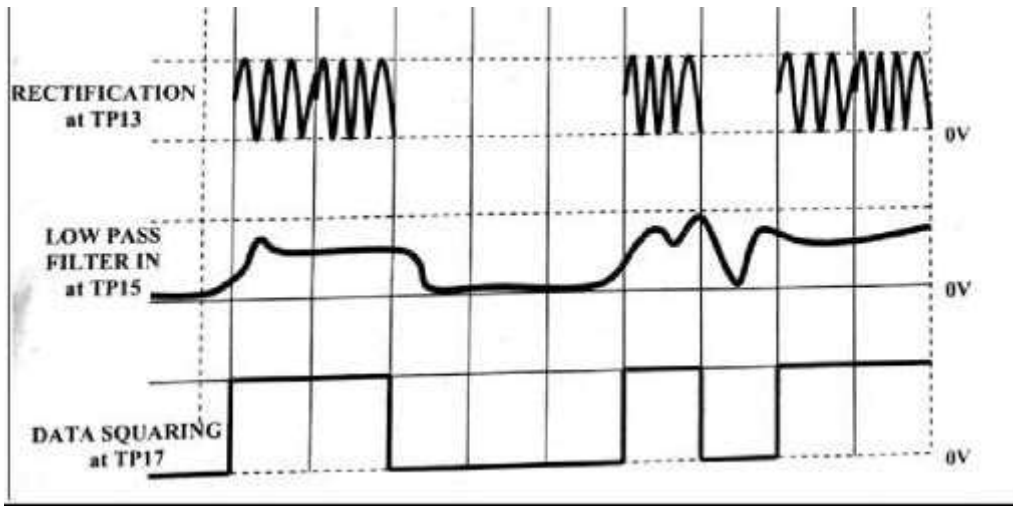
FREQUENCY SHIFT KEYING GENERATION AND DETECTION

APPARATUS:

1. Frequency Shift Keying kit

2. Dual trace C.R.O (30MHz)

3. Digital frequency counter & DMM

CIRCUIT DIAGRAM:
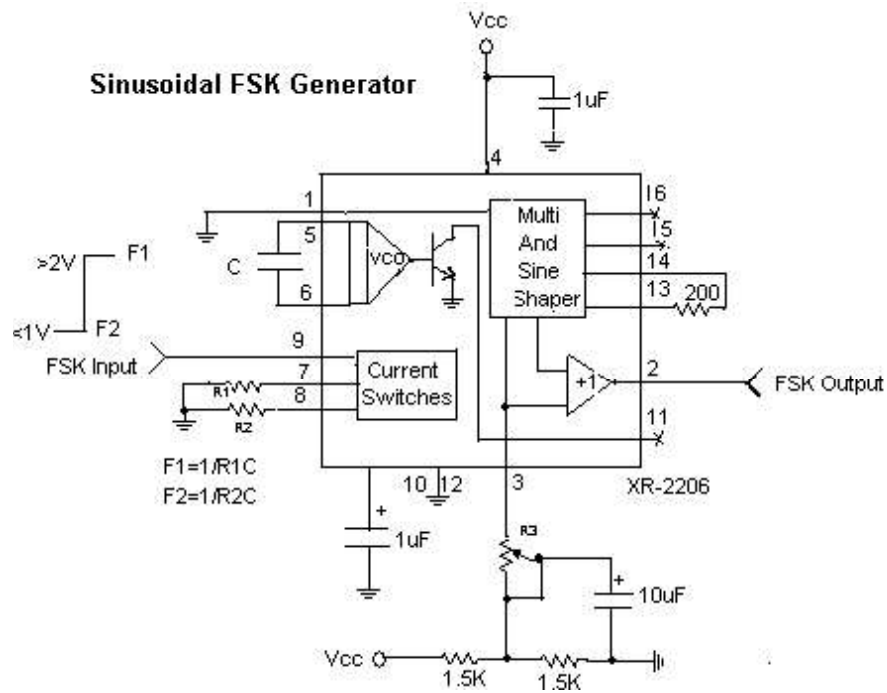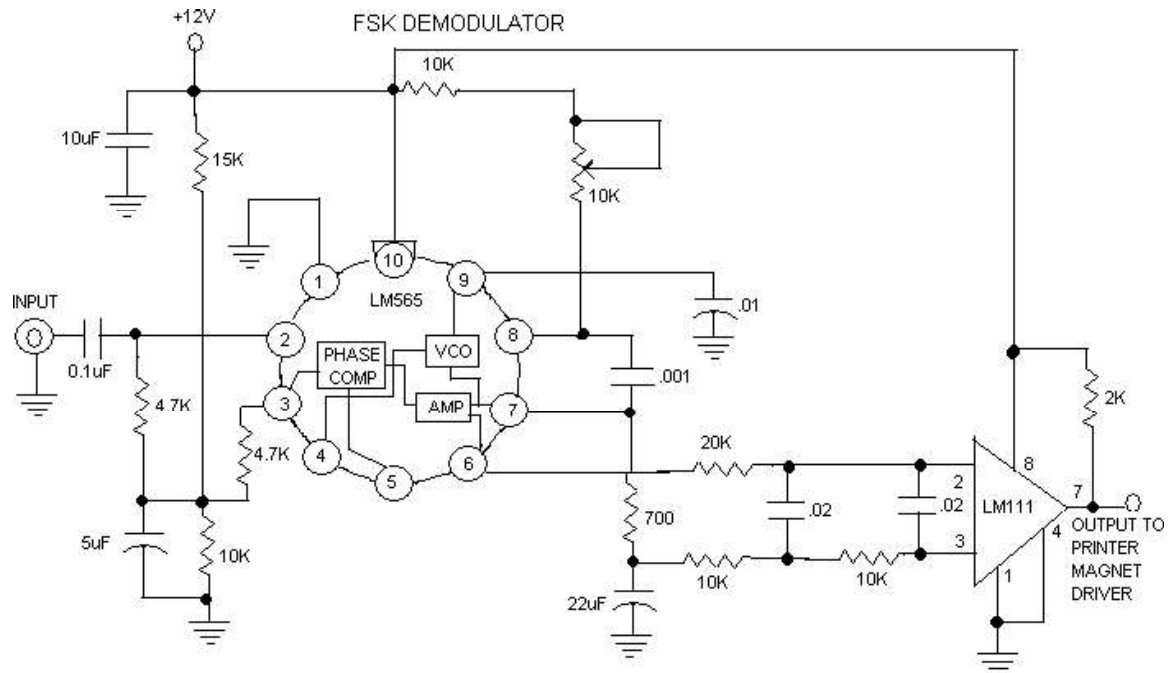
THEORY:

Frequency shift keying (FSK) is a modulation / Data transmitting technique in which carrier frequency is shifted between two distinct fixed frequencies to represent logic 1 and logic 0. The low carrier frequency represents a digital 0 (space) and higher carrier frequency is a 1 (mark). FSK system has a wide range of applications in low speed digital data transmission systems. Waveforms are shown in figure. FSK modulating & demodulating circuitry can be developed in number of ways; familiar VCO and PLL circuits are used in this trainer.

FSK Modulator:

Figure shows the FSK modulator using IC XR 2206, IC XR 2206 is a VCO based monolithic function generator capable of producing Sine, Square, Triangle signals with AM and FM facility. In this trainer XR2206 is used generate FSK signal. Mark (Logic 1) and space (Logic 0) frequencies can be independently adjusted by the choice of timing potentiometers. The output is phase continuous during transitions. The keying signal i.e. data signal is applied to pin 9.

FSK Demodulator:

Figure 2 shows FSK Demodulator is a combination of PLL (LM565) and comparator (Op-amp). The frequency changing signal at the input to the PLL drives the phase detector to result in rapid change in the error voltage, which is applied to the input of the comparator. At the space frequency, the error voltage out of the phase detector is below the comparison voltage of the comparator. The comparator is a non inverting circuit, so its output level is also low. As the phase detector input frequency shifts low (to the mark frequency), the error voltage steps to a high level, passing through the comparison level, causing the comparator output voltage between its two output levels in manner that duplicates the data signal input to the XR2206 modulator.

The free running frequency of the PLL (no input signal) is set midway between the mark and space frequencies. A space at 2025 HZ and mark at 2225 Hz will have a free running VCO frequency of 21125Hz.

PROCEDURE:

1. Study the theory of operation.
2. Connect the trainer to mains and switch on the power supply.
3. Measure the output voltage of the regulated power supply i.e +12V with the help of digital multi- meter.
4. Verify the operation of the logic source using digital multi-meter. Output should be zero volts in logic 0 position and 12V in logic 1 position.
5. Observe the output of the data signal using oscilloscope. It should be a square wave of 20Hz to 180Hz @ 10Vpp. Select 125Hz for getting proper demodulated output. (For frequency variation potentiometer is provided).

FSK Modulation:

1. Connect output of the logic source to data input of the FSK modulator.

2. Set logic source switch in 0 positions.
3. Connect FSK modulator output to oscilloscope as well as frequency counter.
4. Set the output frequency of the FSK modulator as per your desire(say 2KHz) with the help of control F0 which represents logic 0
5. Set logic source switch in 1 position.
6. Set the output frequency of the FSK modulator as per your desire (say 2.5 KHz) with the help of control F1 which represents logic 1.
7. Now connect data input of the FSK modulator to the output of the data signal generator.
8. Keep CRO in dual mode connect CH1 input of the oscilloscope to the input of the FSK modulator and CH2 input to the output of the FSK modulator.
9. Observe the FSK signal for different data signal frequencies and plot them. By this we can observe that the carrier frequency shifting between two predetermined frequencies as per the data signal i.e. 2 KHz when data signal is 0 and 2.5KHz when data input is 1 in this case.
10. Compare these plotted wave forms with the theoretically drawn in figure.

Note: we have chosen F0 as 2 KHz and F1 as 2.5 KHz for ease of operation in fact you may set any value.

FSK Demodulation:

1. Now connect the FSK modulator output to the FSK input of the demodulator.
2. Connect CH1 input of the Oscilloscope to the data signal at modulator and CH2 input to the output of the FSK demodulator (keep CRO in dual mode)
3. Observe and plot the output of the FSK demodulator for different frequencies of data signal. Compare the original data signal and demodulated signal; by this we can observe that there is no loss in process of FSK modulation and demodulation.

EXPECTED WAVEFORMS:
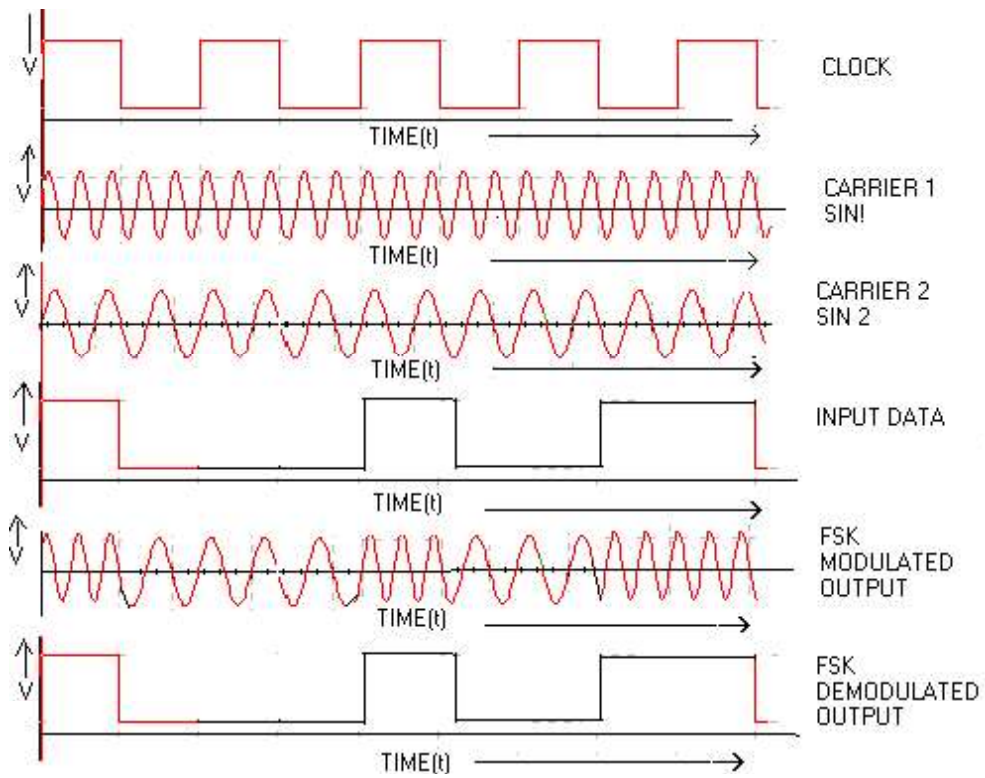
PHASE SHIFT KEYING GENERATION AND DETECTION

APPARATUS:

1. Phase Shift keying trainer-kit
2. Dual trace Oscilloscope
3. Digital multi-meter
4. Patch chords

BLOCK DIAGRAM:

PSK Modulator

Carrier

10KE

10KE

TL 084

CH 1

14

CH 2

12   CD 4052   13   PSK O/P

10   9   1 KHz

Data Signal

PSK Demodulator

PSK Signal   TL 084

Data signal

74LS86D

Carrier   TL 084
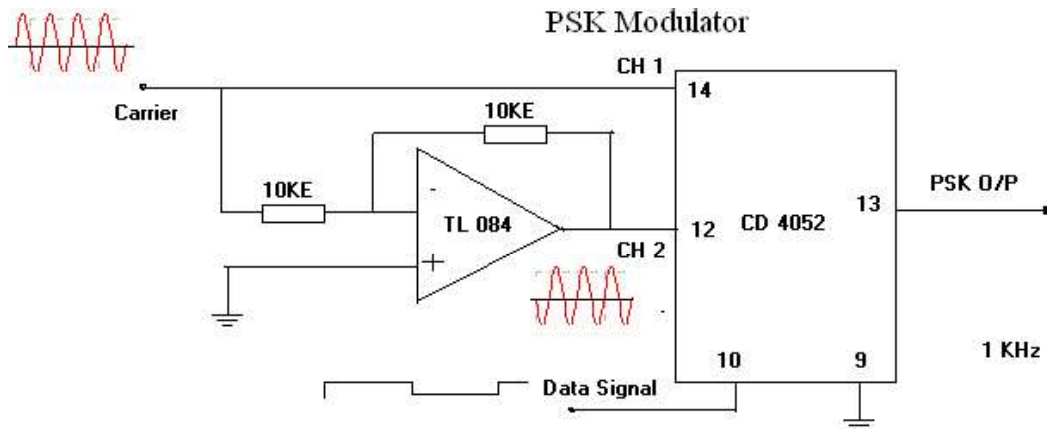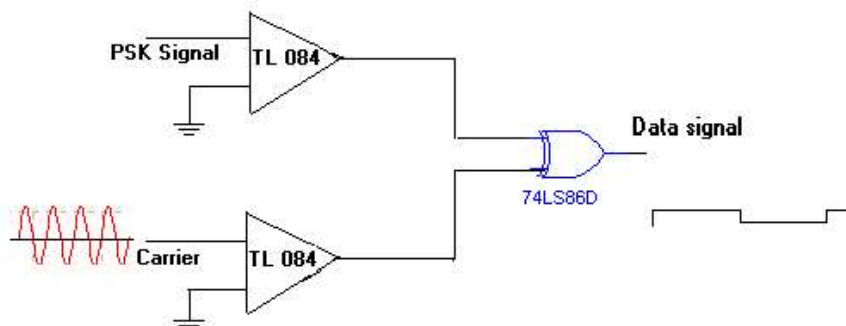
THEORY:

Phase Shifting Keying (PSK) is a modulating / Data transmitting technique in which phase of the carrier signal is shifted between two distinct levels. In a simple PSK (i.e Binary PSK) un-shifted carrier $V\cos W_o t$ is transmitted to indicate a 1 condition, and the carrier shifted by 180° i.e $-V\cos W_o t$ is transmitted to indicate a 0 condition. Wave forms are shown in Figure PSK Modulating & Demodulating circuitry can be developed in number of ways; one of the simple circuits is used in this trainer.

PROCEDURE:

1. Study the theory of operation.
2. Connect the trainer to mains and switch on the power supply.
3. Measure the output of the regulated power supply i.e +5V and -5V with the help of digital multi-meter.
4. Observe the output of the carrier generator using CRO, it should be an 8 KHz Sine with 5 Vpp amplitude.
5. Observe the various data signals (1 KHz, 2 KHz and 4 KHz) using CRO.

Modulation

1. Connect carrier signal to carrier input of the PSK Modulator.
2. Connect data signal say 4 KHz from data source to data input of the modulator.
3. Keep CRO in dual mode.
4. Connect CH1 input of the CRO to data signal and CH2 to the output of the PSK modulator
5. Observe the PSK o/p Signal with respect to data signal and plot the wave forms
6. Compare the plotted waveforms with given wave forms.

Demodulation:

1. Connect the PSK output to the PSK input of the demodulator.
2. Connect carrier to the carrier input of the PSK demodulator
3. Keep CRO in dual mode.
4. Connect CH1 to the data signal (at Modulator) and CH2 to the output of the demodulator.

5. Compare the demodulated signal with original data signal; by this we can notice that there is no loss in modulation and demodulation process.

6. Repeat the steps 7 to 15 with different data signals i.e. 2 KHz and 1 KHz.

Note: In actual communication system reference carrier is generated at receiver.

EXPECTEDWAVEFORMS:
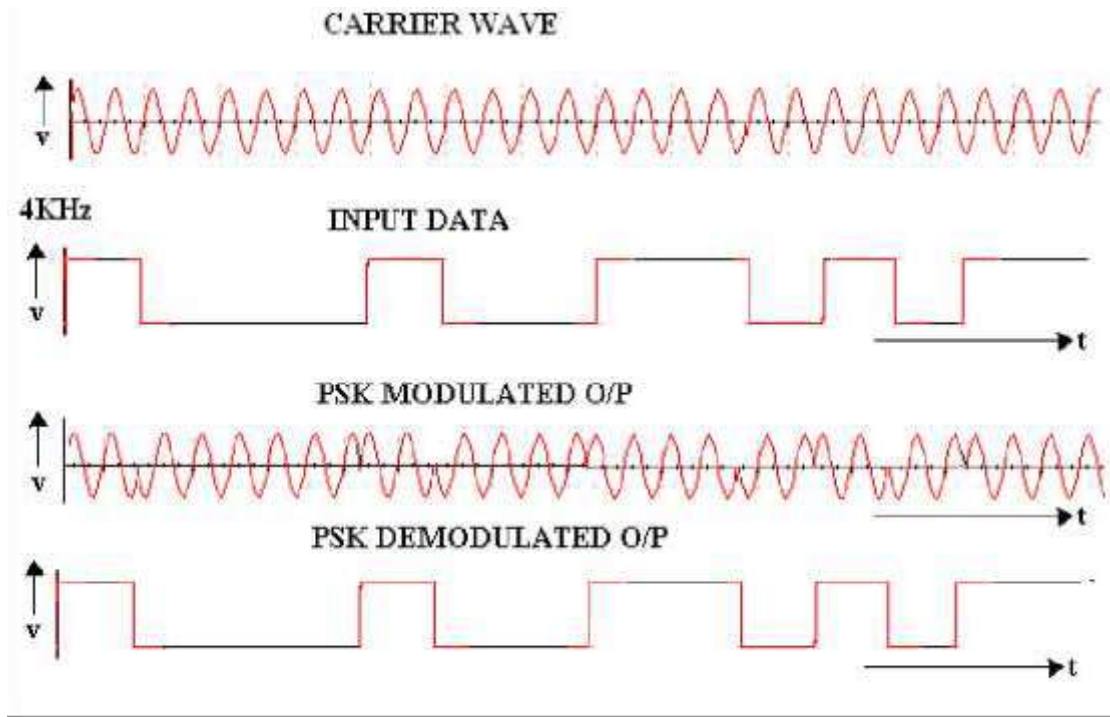
RESULT:
Generation and detection of ASK, PSK, FSK is successfully obtained.

Experiment No. -5

AIM: Demonstrate the generation and detection of DPSK signal and to measure the output parameters.

APPARATUS:

1. Differential Phase Shift Keying Kits
2. C.R.O
3. Digital multi-meter.
4. coaxial cables (standard accessories with trainer)

BLOCK DIAGRAM:
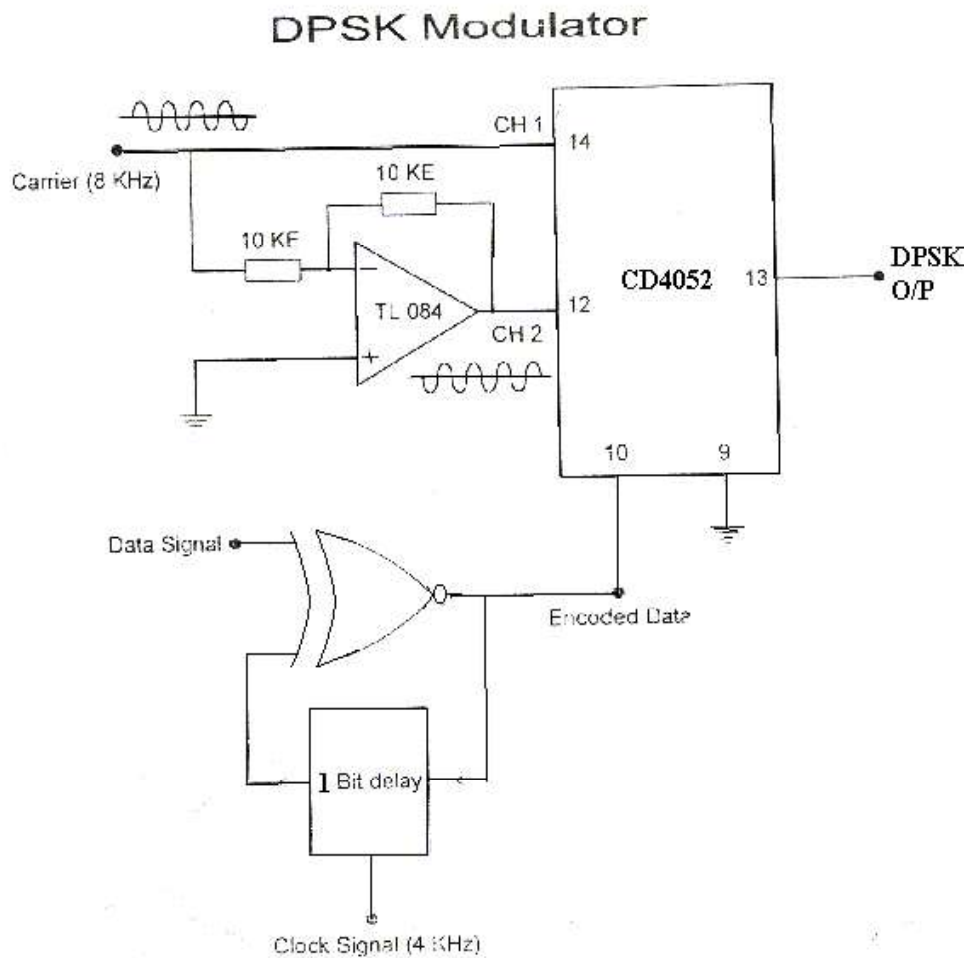
DPSK Modulator

DPSK Demodulator
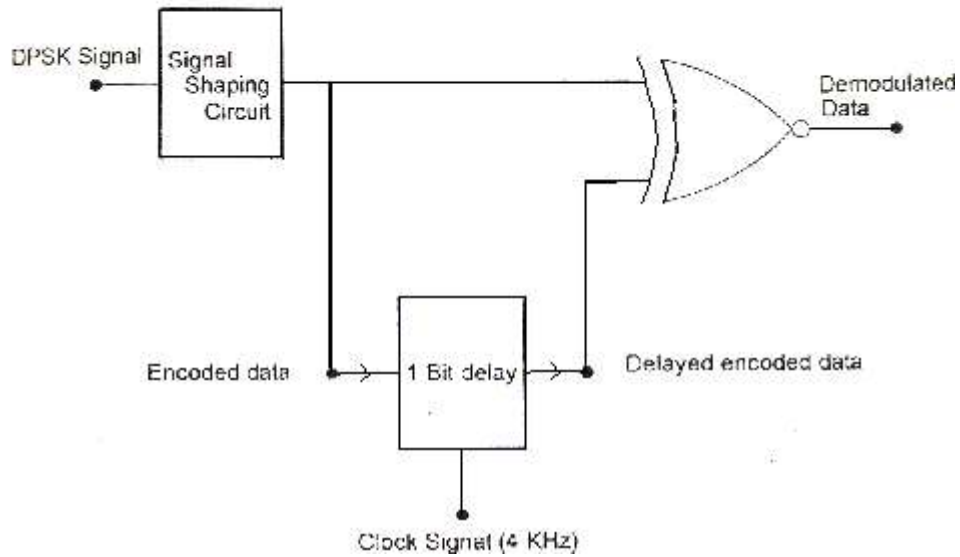
THEORY:

DPSK: Phase Shift Keying requires a local oscillator at the receiver which is accurately synchronized in phase with the un-modulated transmitted carrier, and in practice this can be difficult to achieve. **Differential Phase Shift Keying (DPSK)** over comes the difficult by combining two basic operations at the transmitter (1) differential encoding of the input binary wave and (2) phase shift keying – hence the name differential phase shift keying. In other words DPSK is a non-coherent version of the PSK.

The differential encoding operation performed by the modulator is explained below Let b(t) be the binary message to be transmitted. An encoded message stream b(t) is generated from b'(t) by using a logic circuit The first bit in b(t) is arbitrary which may be chosen as 1 or 0 . The subsequent bits in b(t) are determined on the basis of the rule that when b'(t) is 1 b(t)does not change its value. In the first bit stream, the initial bit (arbitrary) is 1 and in the second bit stream, the initial bit is 0 EX-NOR gate can be used to perform this operation as its output is a 1 when both the input are same, and a 0 when the inputs are different.

b'(t)		0	1	1	0	0
b(t)	1	0	0	0	1	0
Phase	0^0	180^0	180^0	180^0	0^0	180^0

B(t)		0	1	1	1	0	1
Phase	180^0	0^0	0^0	0^0	180^0	0^0	

Example for Complete DPSK operation (with arbitrary bit as 0):

Message signal (to be transmitted) 0		1	1	0	0		
Encoded data (differential data)		0	1	1	1	0	1

Transmitted signal phase:		180^0	0^0	0^0	0^0	180^0	0^0
Received signal phase:		180^0	0^0	0^0	0^0	180^0	0^0
Encoded data (differential data)			0	1	1	1	0
Message signal (Demodulation)			0	1	1	0	0

DPSK Demodulator:

DPSK Modulator:

IC CD 4052 is a 4 channel analog multiplexer and is used as an active component in this circuit. One of the control signals of 4052 is grounded so that 4052 will act as a two channel multiplexer and other control is being connected to the binary signal i.e., encoded data. Un-shifted carrier signal is connected directly to CH1 and carrier shifted by 180^0 is connected to CH2. Phase shift network is a unity gain inverting amplifier using Op-Amp (TL084).

When control signal is at high voltage, output of the 4052 is connected to CH1 and un-shifted (or 0 phase) carrier is passed on to output. Similarly when control signal is at zero voltage output of 4052 is connected to CH2 and carrier shifted by 180^0 is passed on to output.

Differential encoder: This consists of 1 bit delay circuit and an X-NOR Gate. 1 bit delay circuit is formed by a D-Latch. Data signal i.e., signal to be transmitted is connected to one of the input of the X-NOR gate and other one being connected to out of the delay circuit. Output of the X-NOR gate and is connected to control input of the multiplexer (IC 4052) and as well as to input of the D- Latch. Output of the X-NOR gate is 1 when both the inputs are same and it is 0 when both the inputs are different.

DPSK Demodulator:

This consists of 1 bit delay circuit, X-NOR Gate and a signal shaping circuit. Signal shaping circuit consists of Op-amp based zero crossing detector followed by a D- latch. Receiver DPSK signal is converted to square wave with the help of zero crossing and this square wave will pass through the D-Latch. So output of the D-latch is an encoded data. This encoded data is applied to 1 bit delay circuit as well as to one of the inputs of X-NOR gate. And output of the delay circuit is connected to another input of the X-NOR gate. Output of the X-NOR gate is 1 when both the inputs are same and it is 0 when both the inputs are different.

PROCEDURE:

Modulation:

a. Connect carrier signal to carrier input of the PSK Modulator.
b. Connect data signal from data input of the X-NOR gate.
c. Keep CRO in dual mode.
d. Connect CH1 input of the CRO to data signal and CH2 input to the encoded data (which is nothing but the output of the X-NOR gate)
e. Observe the encoded data with respect to data input. The encoded data will be in a given sequence.
f. Actual data signal   :   10101101001010110100
g. Encoded data signal   :   01100011011001110010

h. Now connect CH2 input of the CRO to the DPSK output and CH1 input to the encoded data. Observe the input and output waveforms and plot the same.

i. Compare the plotted waveforms with the given waveforms in figure

Note: Observe and plot the waveforms after perfect triggering. Better to keep the encoded data more than 4 cycles for perfect triggering.

Demodulation:

1. Connect DPSK signal to the input of the signal shaping circuit from DPSK transmitter with the help of coaxial cable (supplied with trainer).

2. Connect clock from the transmitter (i.e. DPSK Modulator) to clock input of the 1 bit delay circuit using coaxial cable.

3. Keep CRO in dual mode. Connect CH1 input to the encoded data (at modulator) and CH2 input to the encoded data (at demodulator).

4. Observe and plot both the waveforms and compare it with the given waveforms. You will notice that both the signals are same with one bit delay.

5. Keep CRO in dual mode. Connect CH1 input to the data signal (at modulator) and CH2 input to the output of the demodulator.

6. Observe and plot both the waveforms and compare it with the given waveforms. You will notice that both the signals are same with one bit delay.

7. Disconnect clock from transmitter and connect to local oscillator clock (i.e., clock generator output from De Modulator) with remaining setup as it is. Observe demodulator output and compare it with the previous output. This signal is little bit distorted. This is because lack of synchronization between clock at modulator and clock at demodulator. You can get further perfection in output waveform by adjusting the   locally generated clock frequency by varying potentiometer.

Carrier signal(8KHz)

Data signal (M)

Encoded data (M)

DPSK signal (M)

Encoded data (D)

Delayed encoded data (D)

Demodulated Data signal (D)
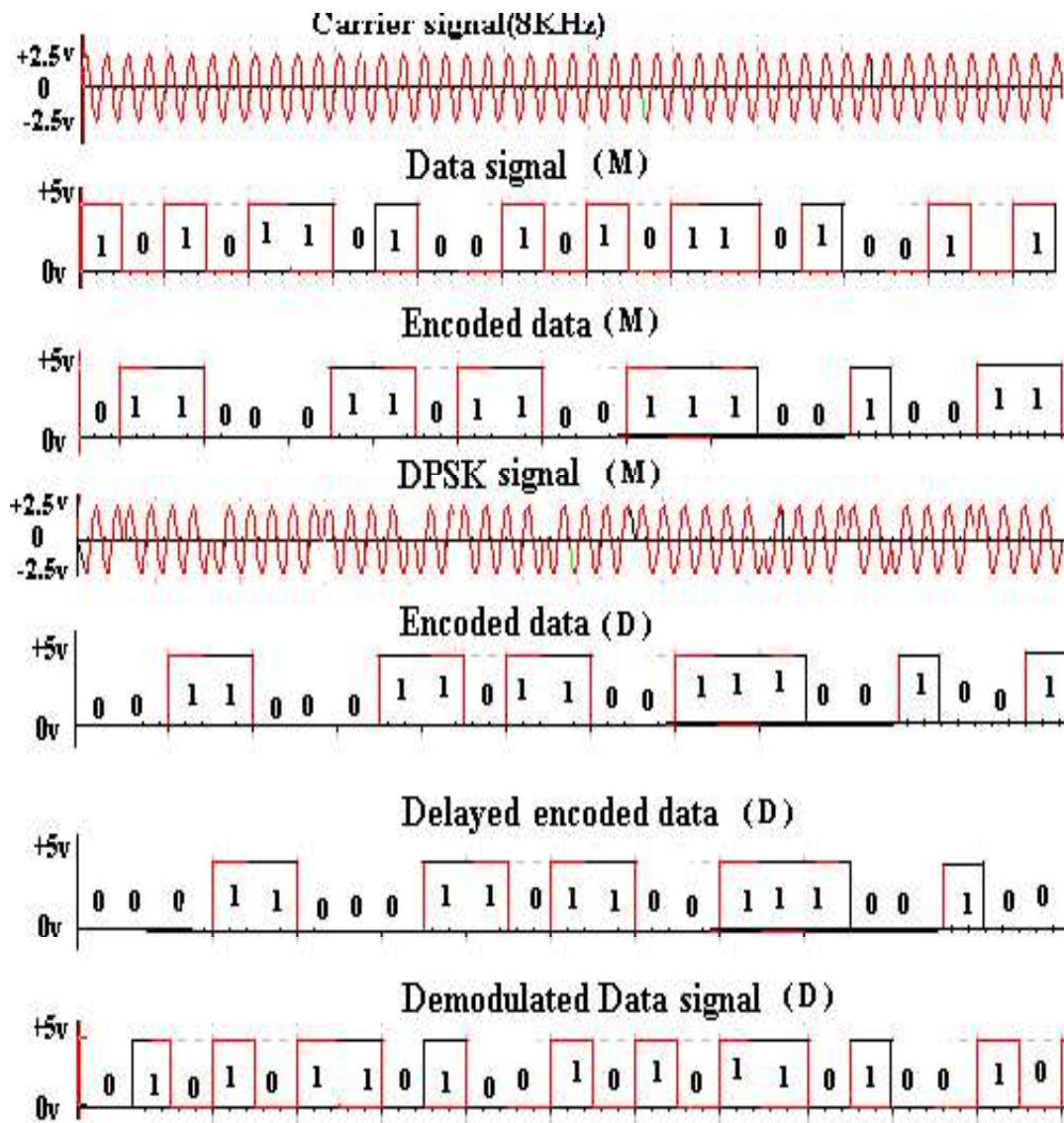
RESULT:
Generation and detection of DPSK signal is obtained successfully.

Experiment No. -6

AIM: Demonstrate the generation and detection of QAM signal and to measure the output parameters using SIMULINK model.

APPARATUS: MATLAB Simulink

THEORETICAL BACKGROUND:
Quadrature Amplitude Modulation (QAM) conveys two bit streams by changing the amplitude of two carrier waves that have the same frequency and a 90° shift. The most common type of QAM modulation is rectangular QAM, were the constellation points are arranged in a square grid. Depending on the desired number of bits per symbol (4, 5, 6 …), we have 16QAM, 32QAM, 64 QAM, etc.…

16 QAM Constellation Diagram
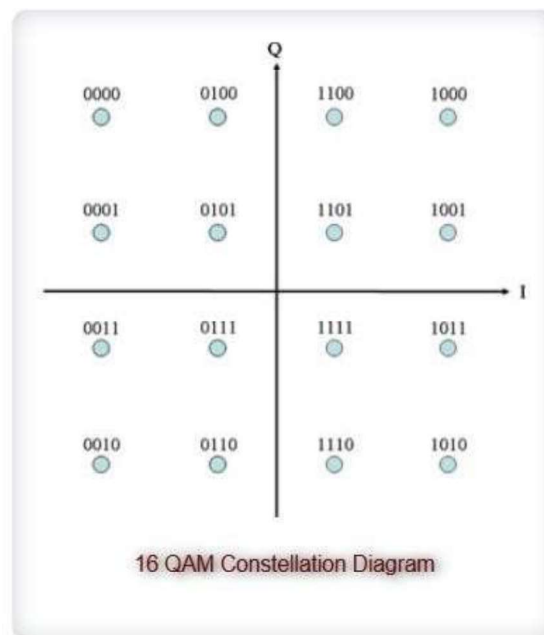
As in QPSK, there are two ways (codes) to map the symbols to the constellation points: Binary code and Gray code. In Gray code, two adjacent symbols differ in one bit, while in Binary code, two adjacent symbols may differ in 2 bits. Therefore, Gray code is preferable over Binary code, since if a receiver maps a symbol to one of its adjacent symbols (due to noise or errors), it will lead to 1 wrong bit instead of 2.

The demodulator maps the received signal (possibly distorted due to noise in the channel) back to bit streams.

For 16 QAM, the Bit Error Rate (BER) is the same as BPSK.

$$BER = \frac{3}{4} Q \left(\sqrt{\frac{4E_b}{N_o}} \right)$$

Since in QAM modulation two carriers are used, the Symbol Error Rate per carrier is given by:

$$P_{sc} = \frac{6}{4} Q \left(\sqrt{\frac{\overline{E_b}}{5N_o}}\right)$$

And the total Symbol Error Rate is given by:

$$P_s = 1 - (1 - P_{sc})^2$$

Where $N_0/2$ is the noise power spectral density, and $Q(.)$ is the Q function of the Gaussian distribution.

Building Simulink Model of 16 QAM Modulator and Demodulator

The Simulink model of 16 QAM modulator and demodulator is shown below
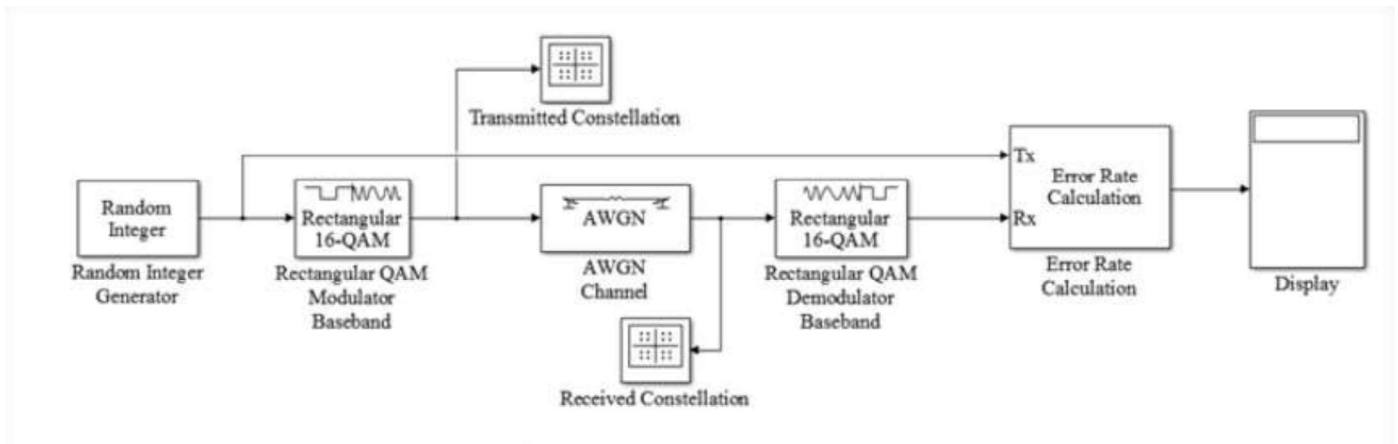
- Build the Simulink model shown
- Double-click on the Random Integer Generator and adjust the set size to a proper value (Remember that the input to the 16 QAM modulator should be from the set {0, 1, 2, …, 15}).
- In the Random Integer Generator block, set the Sample Time to 1e-6 (i.e. 1 μs) and the Samples per frame parameter to 1024.
- In the AWGN block, set the Symbol period parameter to 1e-6 (i.e. 1 μs) and the Number of bits per symbol parameter to 4 (since 16 QAM uses 4 bits per symbol).
- For the Error Rate Calculation block, set the Output data field to "port" so you can connect the Display block.
- The Display Block will show you three values. The first value is the BER, the second value is the number of incorrect bits, and the third value is the total number of bits received.
- Set the simulation time to 10 seconds.
- In both 16 QAM Modulator and Demodulator blocks, set the Constellation ordering to Gray, set the Normalization method to Peak Power, and set the value of the Peak power to 1 Watt.
- In this experiment, you will adjust the value of the E_b/N_0 in the AWGN block, starting from 3, incrementing by 1 every step, and ending at 15, and observe the error rate displayed in the Display block. Make a table recording the value of E_b/N_0 and the corresponding BER.
- Plot BER vs. E_b/N_0 and compare with the theoretical values. Comment on the results.
- Repeat for different values of the Peak power

RESULTS: Plot the results.

Experiment No. -7

AIM: To develop a code for the ASK, FSK, PSK digital modulation & demodulation technique using simulation tool

Software Used: MATLAB

MATLAB CODE:

RESULTS:

Experiment No. -8

AIM: To develop a code for the shannon fano and huffman coding technique to calculate the code efficiency and redundancy using simulation tool

Software Used: MATLAB

MATLAB CODE:

RESULTS:

Experiment No. -9

AIM: To develop a code for the linear block/Cyclic coding technique for user input data using simulation tool

Software Used: MATLAB

MATLAB CODE:

RESULTS:

COLLEGE OF TECHNOLGY AND ENGINEERING

MAHARANA PRATAP UNIVERSITY OF AGRICULTURE & TECHNOLGY, UDAIPUR

DEPARTMENT OF ELECTRONICS AND COMMUNICATION ENGINEERING

LAB MANUAL

ECE326- PULSE AND SWITCHING WAVEFORMS CIRCUITS

PREPARED BY

DR. P.C.BAPNA

INDEX

S.no.	DATE	Name of Experiment	Remarks

Exp.no.1   Date:

_____

Aim:

Design High Pass circuit using R-C components and determine the output for different input waveform e.g., pulse, step and square wave. Determine the effect of time constant and pulse width over output. Plot the output waveform for RC>> t_p, RC = t_p and RC << t_p.

COMPONENTS REQUIRED:

1. Resistors 1kΩ, 10kΩ, 100kΩ
2. Capacitor 0.1 μf (1No.)
3. Bread Board
4. Connecting wires
5. CRO & Probes
6. Function Generator

THEORY:

In a high pass RC circuit, at zero frequency the reactance of the capacitor is infinity and so it blocks the input and hence the output is zero. Hence this capacitor is called the blocking capacitor and this circuit also called the capacitive coupling circuit, is used to provide DC isolation between the input and output. As the frequency increases the reactance of the capacitor decreases and hence the output and gain increases. At very high frequencies the capacitive reactance is very small so a very small voltage appears across capacitor and so the output is almost equal to the input and the gain is equal to unity. Since this circuit attenuates low frequency signals and allows transmission of high frequency signals with little or no attenuation, it is called a high pass circuit. RC differentiator can be operated in three different modes i.e.

Large Time constant ($RC \gg T$), Medium Time constant ($RC = T$) & Short Time constant ($RC \ll T$).

CIRCUIT DIAGRAM

:
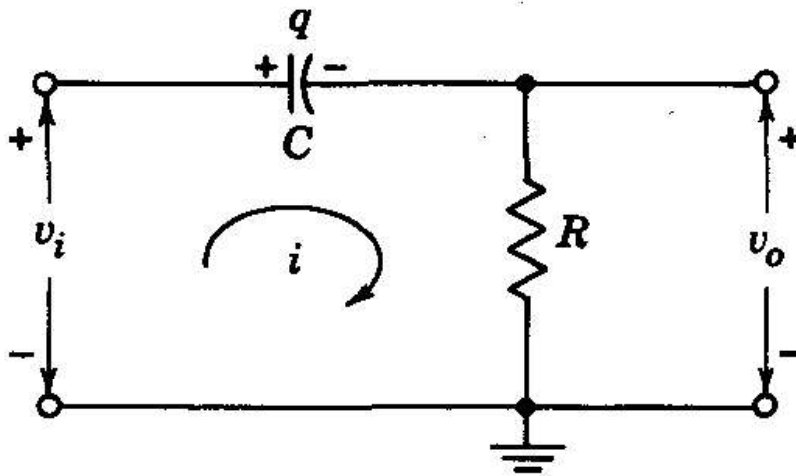
RC DIFFERENTIATOR (or) HIGH PASS FILTER:

C=0.1 µf

Vi (From function generator)

Vo(Output in CRO)

R = 1kΩ, 10kΩ, 100kΩ

PROCEDURE:

1. Connect the high pass circuit as per the circuit diagram.

2. Connect the function generator at the input terminals and CRO at the output terminals.

3. Apply a square wave signal of 10V amplitude and 1 KHz frequency at input.

4. Observe the output waveform of the circuit for different time constants. Large Time constant ($RC \gg T$), Medium Time constant ($RC = T$),Short Time constant ($RC \ll T$).

5. Draw the graph for high pass circuits for above three cases of time constants.

RESULT:

(a) When $\ll T$
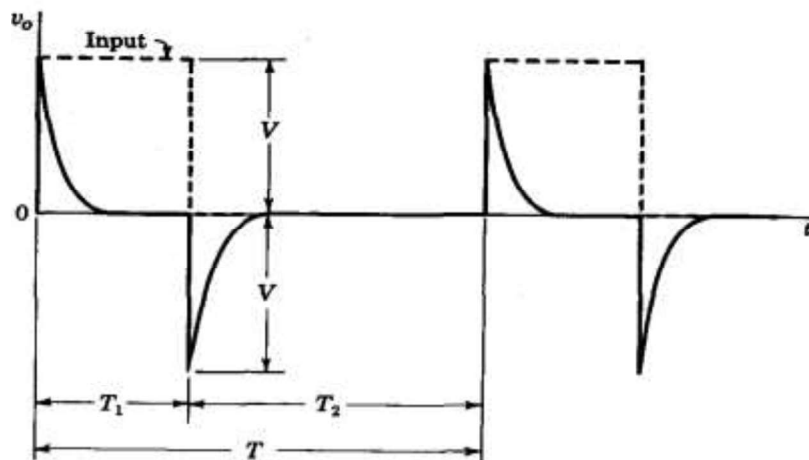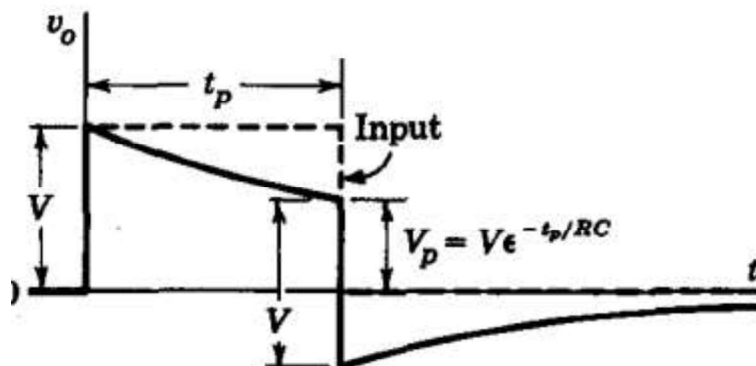
(b) When RC = T

(C) When RC<< T
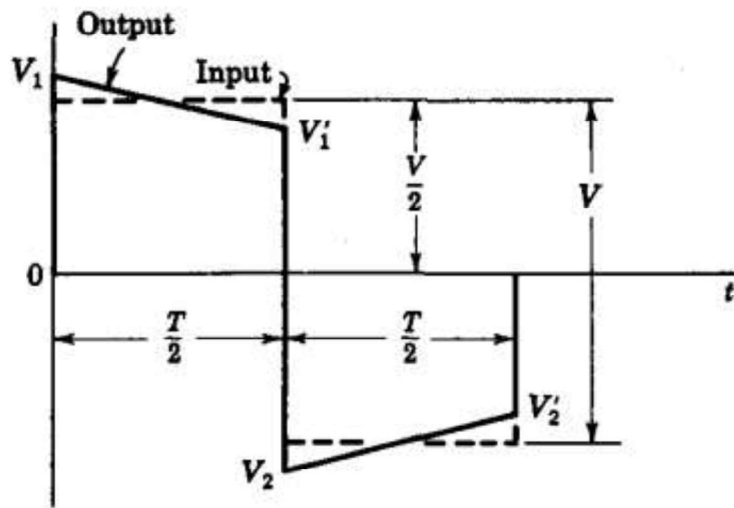
Graphs

Exp.no.2   Date:

_____

Aim:

Design Low Pass circuit using R-C components and determine the output for different input waveform e.g., pulse, step and square wave. Determine the effect of time constant and pulse width over output. Plot the output waveform for RC>> t_p , RC = t_p and RC << t_p.

COMPONENTS REQUIRED:

1. Resistors 1kΩ, 10kΩ, 100kΩ
2. Capacitor 0.1 μf (1No.)
3. Bread Board
4. Connecting wires
5. CRO & Probes
6. Function Generator

THEORY:

The process in which non sinusoidal signal is altered by transmission through a linear network is called "Linear wave shaping".

 i) RC INTEGRATOR (or) LOW PASS FILTER: A Low pass circuit is a circuit which transmits only low frequency signals and alternates (or) stops high frequency signals at zero frequency, the reactance of the capacitor is infinity (i.e. the capacitor acts as a open circuit). So the entire input appears at the output i.e. the input is transmitted to the output with zero alternation. So the entire output is same as the input i.e. the gain is unity. As the frequency increase the capacitor reactance $XC = 1/2\pi f c$ decreases and so the output decreases. At high frequencies the capacitor virtually acts as a short circuit and the output falls to zero. RC Integrator can be operated in three different modes i.e. Large Time constant ($RC \gg T$), Medium Time constant ($RC = T$) & Short Time constant ($RC \ll T$).
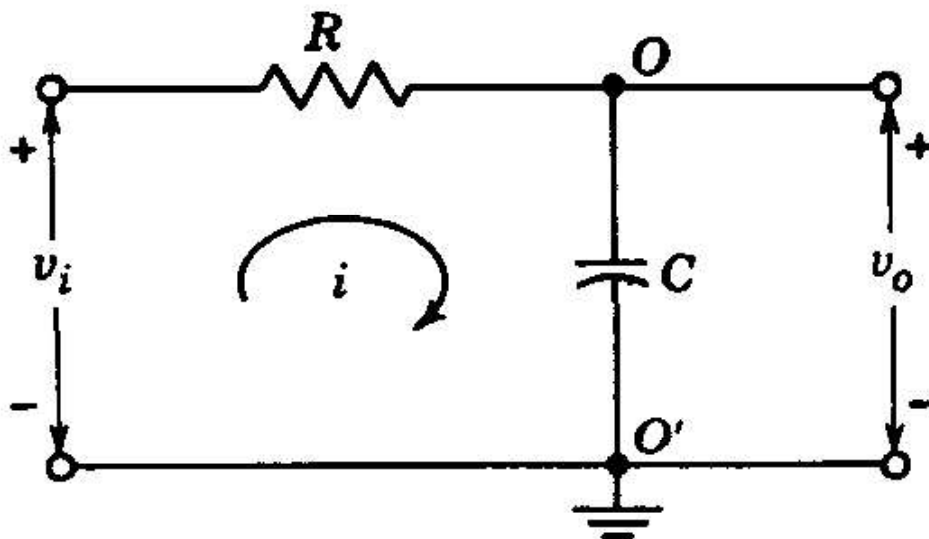
CIRCUIT DIAGRAM

RC INTEGRATOR (or) LOW PASS FILTER:

C=0.1 μf

Vi (From function generator)

 Vo(Output in CRO)

R = 1kΩ, 10kΩ, 100kΩ

PROCEDURE:

1. Connect the low pass circuit as per the circuit diagram.

2. Connect the function generator at the input terminals and CRO at the output terminals.

3. Apply a square wave signal of 10V amplitude and 1 KHz frequency at input.

4. Observe the output waveform of the circuit for different time constants. Large Time constant ($RC \gg T$), Medium Time constant ($RC = T$), Short Time constant ($RC \ll T$).

5. Draw the graph for low pass circuits for above three cases of time constants.

RESULT:

(a) When $\ll T$
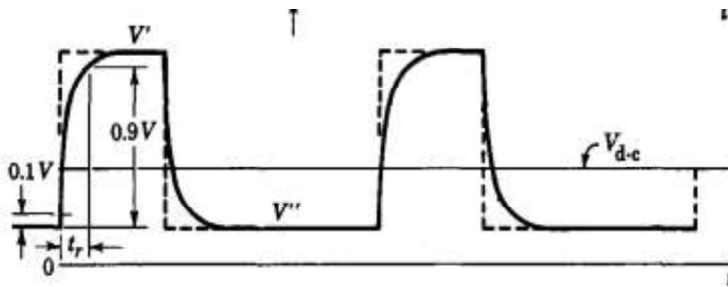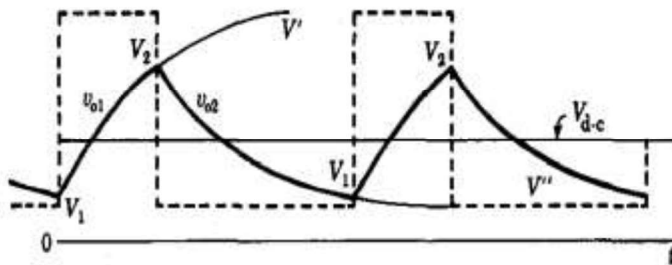
(b) When RC = T

(c) When RC >> T
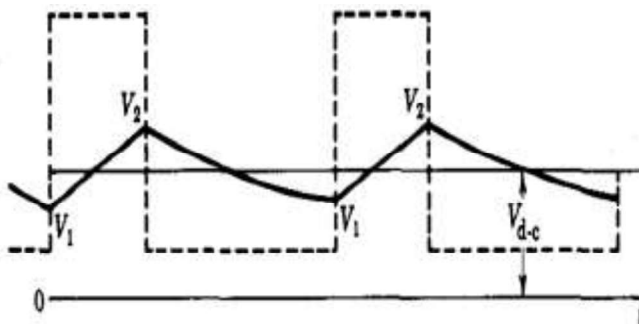
Graphs

Exp.no.3   Date:

_____

NON LINEAR WAVE SHAPING – CLIPPERS

AIM:

To design and verify waveforms of different clipping circuits with different reference voltage.

COMPONENTS REQUIRED:

1. Resistors 1kΩ (1No.)

2. Diode 1N4007 (2No.)

3. Bread Board

4. Connecting wires

5. CRO & Probes

6. Function Generator

7. Regulated Power Supply (0 - 30V)

THEORY:

The non-linear semiconductor diode in combination with resistor can function as clipper circuit. Energy storage components like capacitor etc. are not required in the basic process of clipping. These circuits will select part of an arbitrary waveform which lies above or below some particular reference voltage level and that selected part of the waveform is used for transmission. So they are referred as voltage limiters, current limiters, amplitude selectors or slicers. There are three different types of clipping circuits. 1) Positive Clipping circuit. 2) Negative Clipping. 3) Positive and Negative Clipping (slicer). In positive clipping circuit positive cycle of Sinusoidal signal is clipped and negative Portion of sinusoidal signal is obtained in the output. If reference voltage is added, instead of complete positive cycle that portion of the positive cycle which is above the reference voltage value is clipped.

In negative clipping circuit instead of positive portion of sinusoidal signal, negative Portion is clipped. In slicer both positive and negative portions of the sinusoidal signal are clipped.

Operation can be explained based on equations as shown below:

1. When Vi < VR + Vγ, Diode is reverse biased (OFF). Output follows the input.

2. When Vi > VR + Vγ, Diode is forward biased (ON). And the Output is equal to (VR + Vγ).

Here Vi is Supplied input voltage, VR is connected reference voltage, Vγ is diode cut-in voltage.

CIRCUIT DIAGRAMS

1) Positive clipper:

Model Graphs
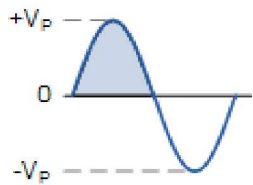
| Input waveform | Circuit diagram | Output waveform |
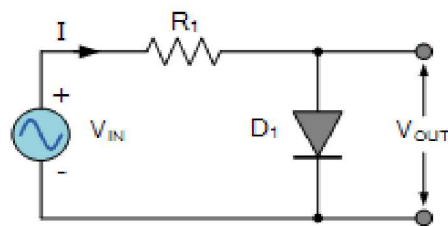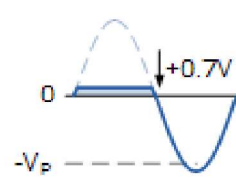
2) Negative clipper
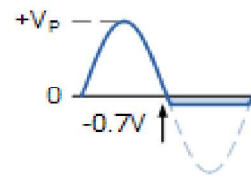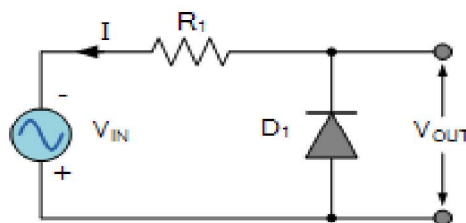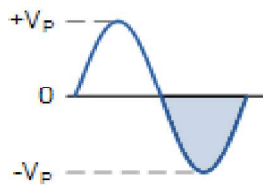
3) clipping at both half cycles
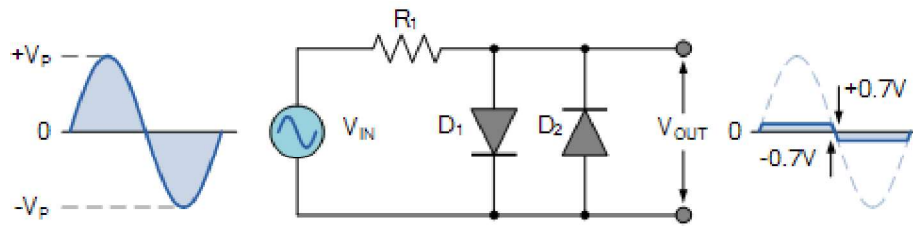
4) Positive bias diode clipper
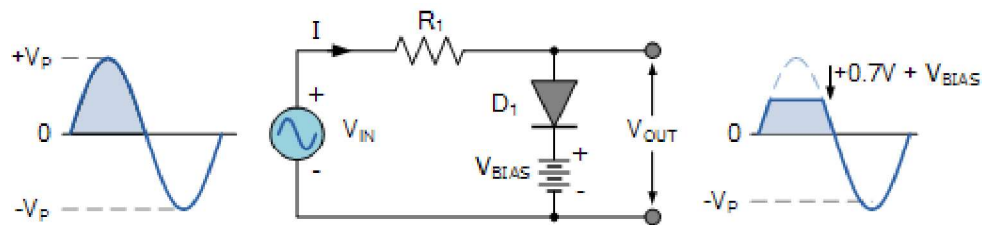
5) Negative bias diode clipper
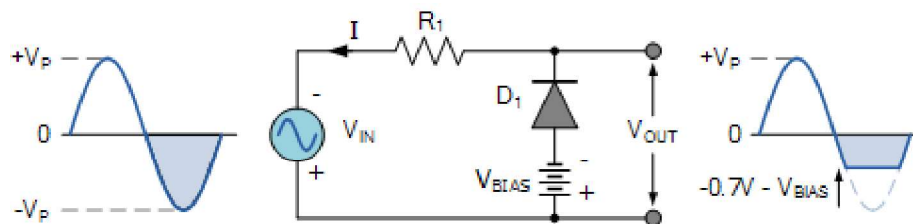
6) Diode clipping at different bias level
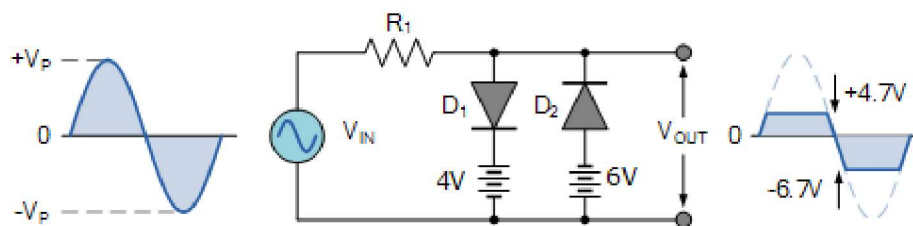
Table:

	Theoretical values	Practical values
1. Positive clipper	$V_\gamma = 0.7$	
2. Positive clipper with positive reference	$V_\gamma + VR = 0.7+$	
3. Negative clipper	$V_\gamma = -0.7$	
4. Negative clipper with negative reference	$VR - V_\gamma =$	
5. slicer	$V_\gamma + VR =$ $VR - V_\gamma =$	

PROCEDURE:

1. Connect the circuit elements as shown in the Circuit Diagram.

2. A Sinusoidal voltage of 10V and frequency of 1kHz is applied to the circuit as an input.

3. Note down the corresponding output wave forms from C.R.O and Enter the values in table.

4. Plot the graph from above readings.

Result:

Plot the graphs

Exp.no.4   Date:

_____
NON LINEAR WAVE SHAPING – CLAMPERS

AIM:

To design and verify the characteristics of different clamping circuits with different reference voltage. COMPONENTS REQUIRED:

1. Capacitor 10 μ F (1 No)

2. Diode IN4007 (1 No)

3. Bread Board

4. Connecting wires

5. CRO & Probes

6. Function Generator

THEORY:

"A clamping circuit is one that takes an input waveform and provides an output that is a faithful replica of its shape but has one edge tightly clamped to the zero voltage reference point".

There are various types of Clamping circuits, which are mentioned below:

1. Positive Clamping Circuit.
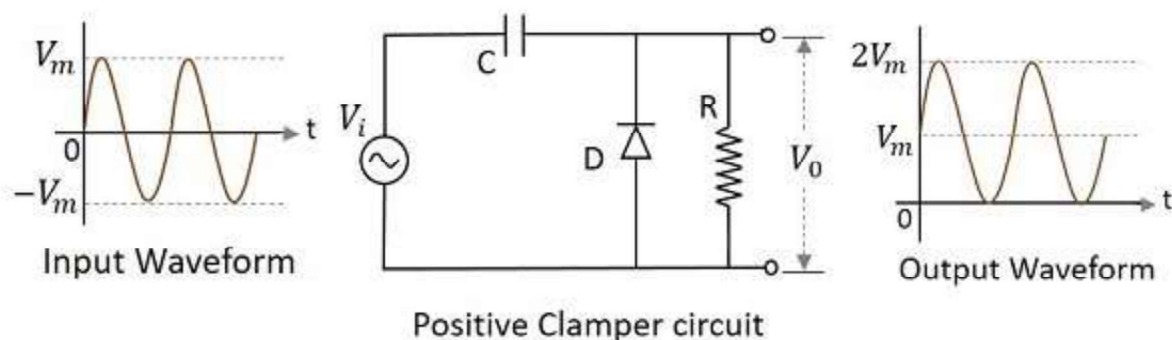
Positive Clamper circuit

2. Negative Clamping Circuit.
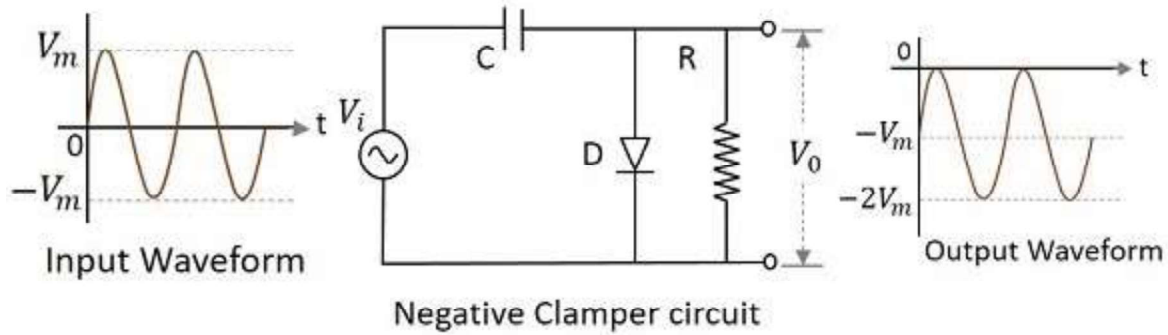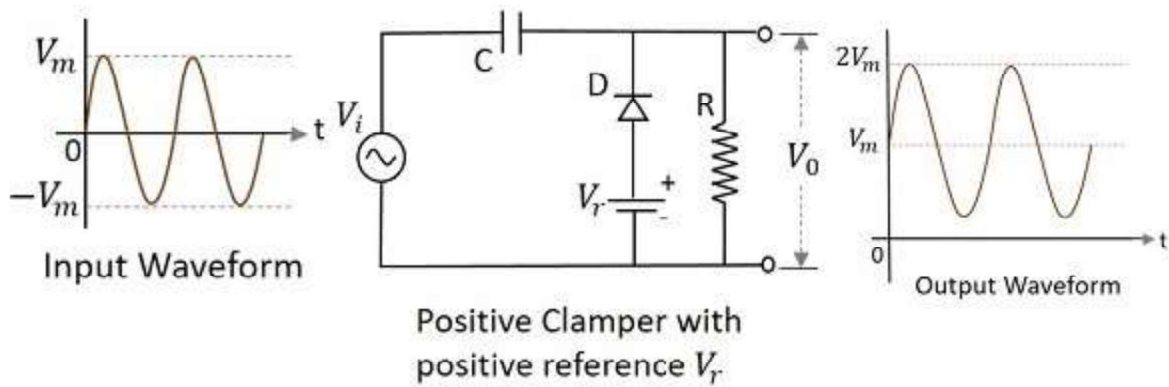
Input Waveform

Negative Clamper circuit

Output Waveform

3. Positive Clamping with positive reference voltage.

Input Waveform

Positive Clamper with positive reference V_r

Output Waveform

4. Negative Clamping with positive reference voltage.

Input Waveform

Negative Clamper with positive reference V_r

Output Waveform

5. Positive Clamping with negative reference voltage.

Input Waveform

Positive Clamper with negative reference V_r

Output Waveform

6. Negative Clamping with negative reference voltage.

Input Waveform

Negative Clamper with negative reference V_r

Output Waveform
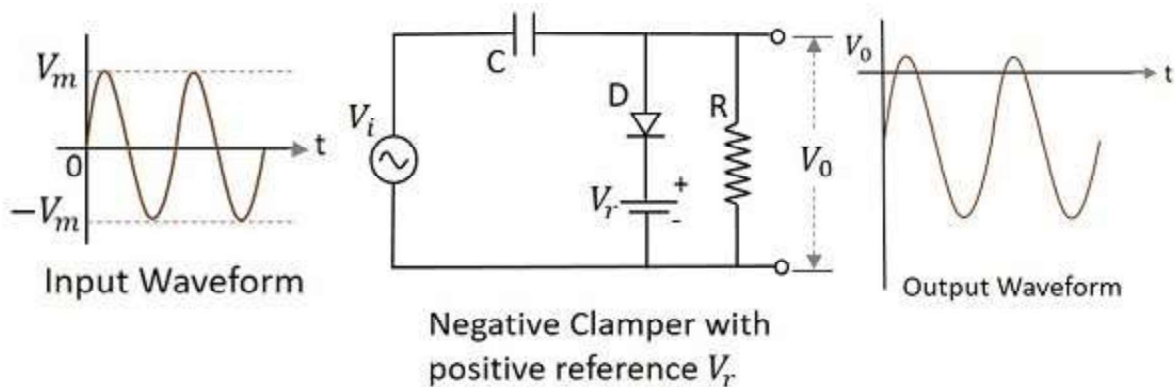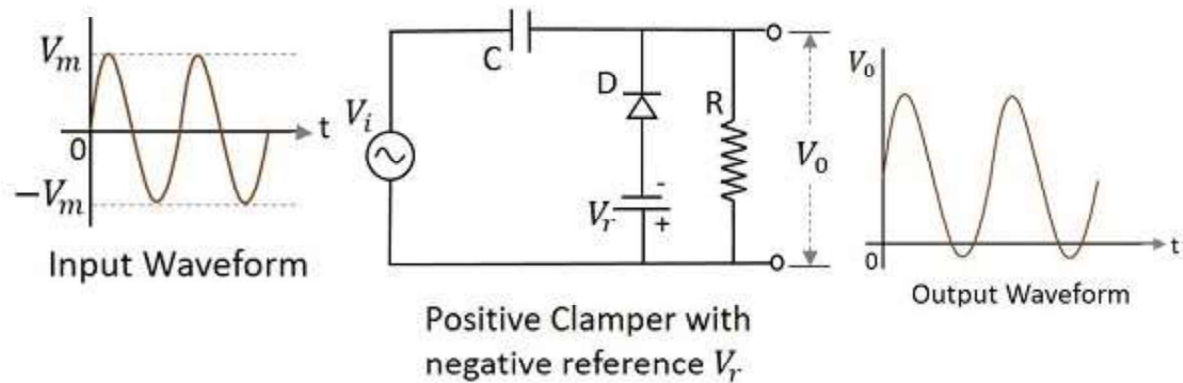
PROCEDURE:

1. Connect the circuit elements as shown in the Circuit Diagram.

2. A Sinusoidal voltage of 10V and frequency of 1kHz Hz is applied to the circuit as an input.

3. Note down the corresponding output wave forms in C.R.O and plot the graph.

RESULT: Hence different clamping circuits were designed and outputs were verified.

Graphs

Exp.no.5   Date:

ASTABLE MULTIVIBRATOR

AIM:

To study and verify the characteristics of an astable multivibrator.

COMPONENTS REQUIRED:

1. Transistor BC 547 -------------------- 2 No's

2. Capacitor 0.01 μ F --------------------2 No's

3. Resistors 1K Ώ, 33K Ώ ------- each 2No

4. Bread Board

5. Connecting wires as required

6. CRO & Probes

7. Function Generator

8. Regulated Power Supply (0 - 30V)

THEORY:

Astable multivibrator has two quasi – states and it keeps on vibrating between these two states by itself. No external signal is needed. The astable remains indefinitely in any of these two states. Assuming that the multivibrator is already in action and is switching between two states. Let it be further observed that at the instant considered Q1 is OFF and Q2 is ON. Since Q2 is ON, the capacitor is charged through RC1 and capacitor C1 discharges through R1 the voltage across C1 when it is about to start discharging in Vcc. As capacitor C1 discharges more and more the identical at the point A becomes more and more positive, and eventually VA = Vr the cut in voltage Q1 states conducting. When Q1 is ON Q2 becomes OFF. Similar operation repeats when Q1 becomes ON and Q2 becomes OFF and vice versa.

Circuit Diagram:
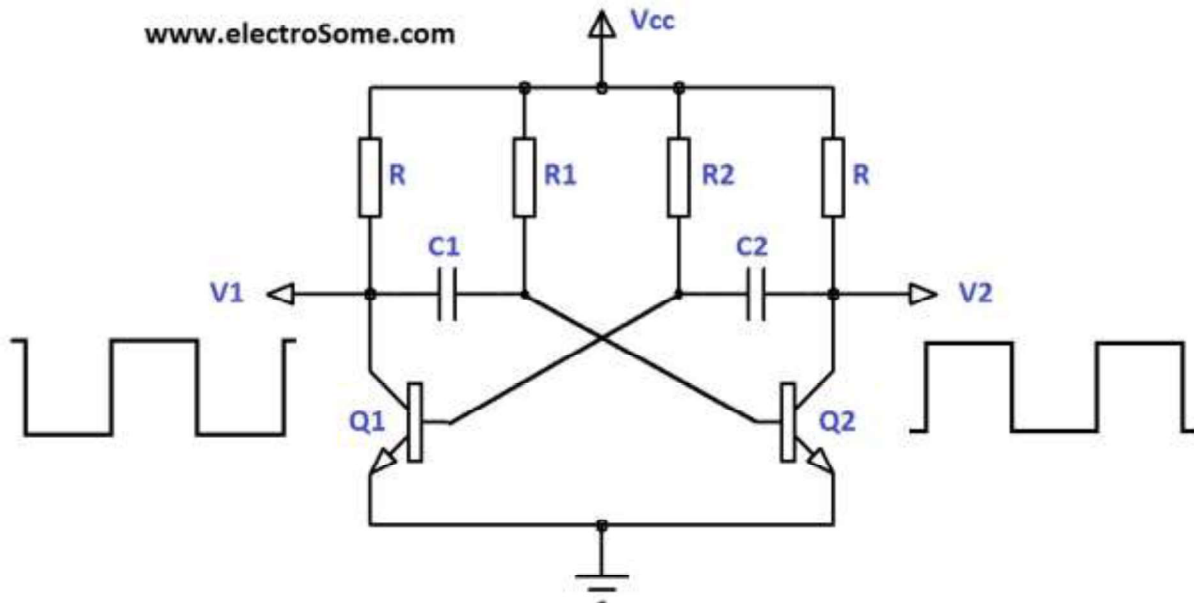
R=1k, R1 R2 = 33k and c1=c2= 0.01 μ F

Model graphs
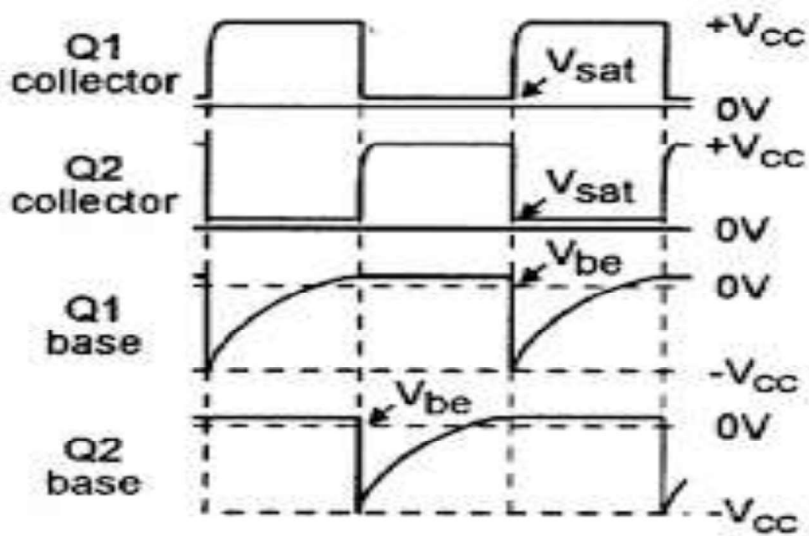
PROCEDURE:

1. Connect the circuit as shown in the circuit diagram.

2. Take the output across the collector considering Q1 is OFF and Q2 is ON, we get Vc2.

3. Now connect the wire across the base also and take the output VB2 and repeat with Q1 ON and Q2 OFF.

4. The required waveforms are taken from the CRO.

RESULT: The astable multivibrator is studied and its output waveforms were verified.

Draw graphs

Exp.no.6   Date:

MONOSTABLE MULTIVIBRATOR

AIM:

To study and verify the characteristics of an Monostable multivibrator.

COMPONENTS REQUIRED:

1. Transistor BC 547 ---------- 2 No's

2. Capacitor 0.1 μF ------------ 2 No's

3. Resistors 1 KΩ -------------- 2 No's 10 KΩ ------------ 4 No's

4. Diode 1N 4007 -------------- 1 No

5. Bread Board

6. Connecting wires as required

7. CRO & Probes

8. Function Generator

9. Regulated Power Supply (0 - 30V)

THEORY:

A monostable multivibrator has one stable state and the other state being quasi – stable state. Normally the multi is in the stable state and when an external triggering pulse is applied it switches back to its original state, without any triggering pulse. The output of the monostable multi while it remains in the quasi – stable is a part of duration T, whose value depends upon the circuit components. Hence the monostable multivibrator is called as 'pulse operator '. The monostable multivibrator is also referred to and 'one shot' or 'uni vibrator'. Since only one triggering signal is required to revert to the original state of stability, that is the stable state.
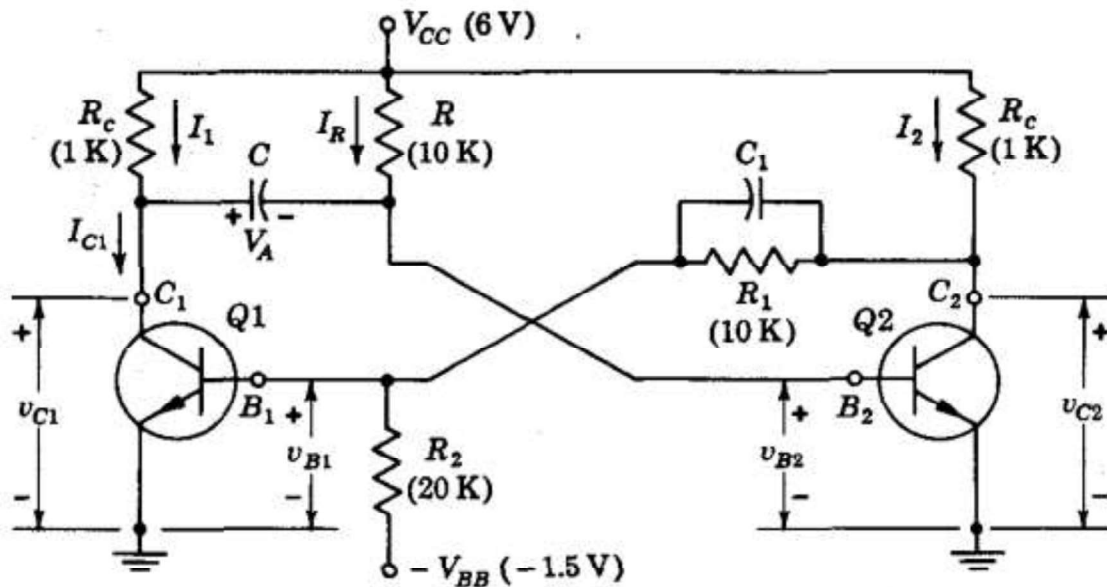
VC2 = Voltage at collector of Q2

VB2 = Voltage at base of Q2

VC1 = Voltage at collector of Q1

VB1 = Voltage at base of Q1

PROCEDURE:

1. Connect the circuit as shown in the circuit diagram.

2. Usually a +Ve supply voltage of about 10 V is applied as source voltage to Vcc

3. In the normal operation transistor Q2 is ON and output voltage across the base and collector are noted.

4. Then an external triggering is applied from the high-pass filter circuit.

5. Then the transistor Q2 is OFF state, and then the voltage across the junction of the transistor Q1 is noted.

6. The output waveforms are plotted for VC1 , VB1 , VC2 , VB2 which are observed from the CRO.

Waveforms :
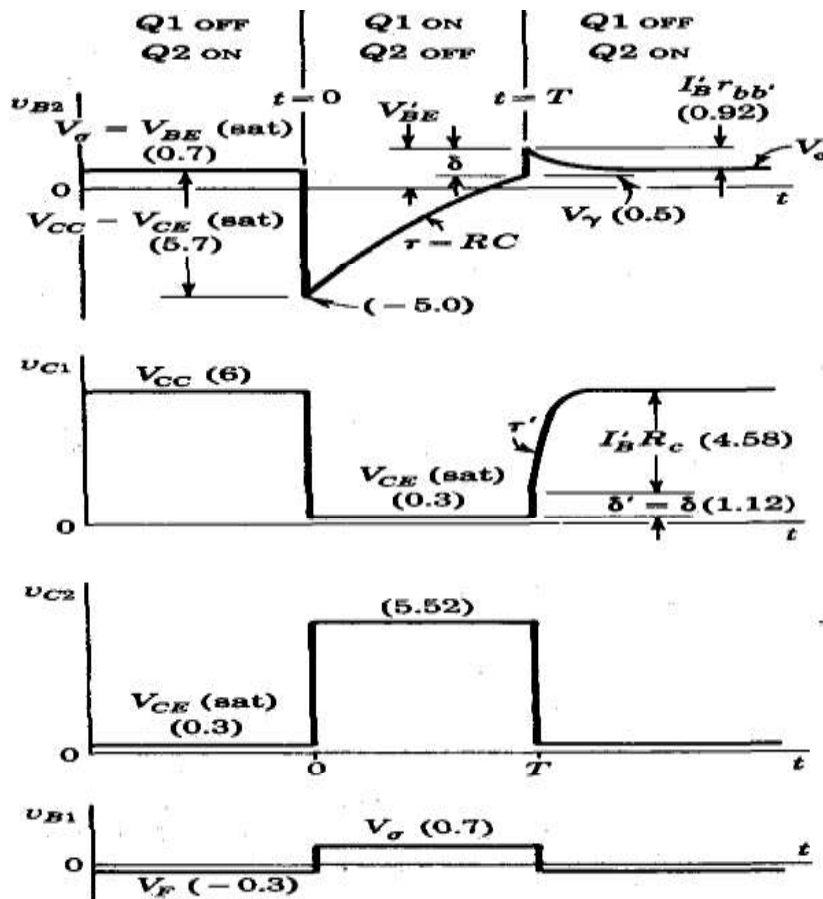
RESULT:

A monostable multivibrator is studied and output wave forms were verified.

Draw Graphs

Exp.no.7   Date:
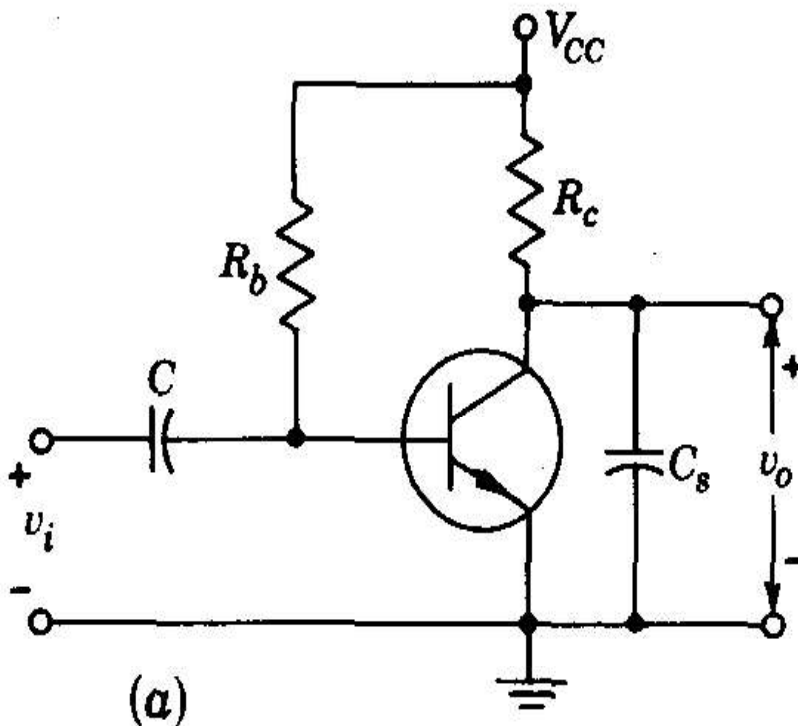
TRANSISTOR SWITCH

Aim:

To Design Transistor switch with Inductive load and draw output waveform for square wave input.

Components required:

1. Transistor BC 547 ---------- 1 No's
2. Capacitor 0.1 μF ------------ 1 No's
3. Resistors 1 KΩ -------------- 1 No's 10 KΩ ------------ 2No's
4. Bread Board
5. Inductor (Relay)
6. Connecting wires as required
7. CRO & Probes
8. Function Generator

Circuit Diagram

(a)
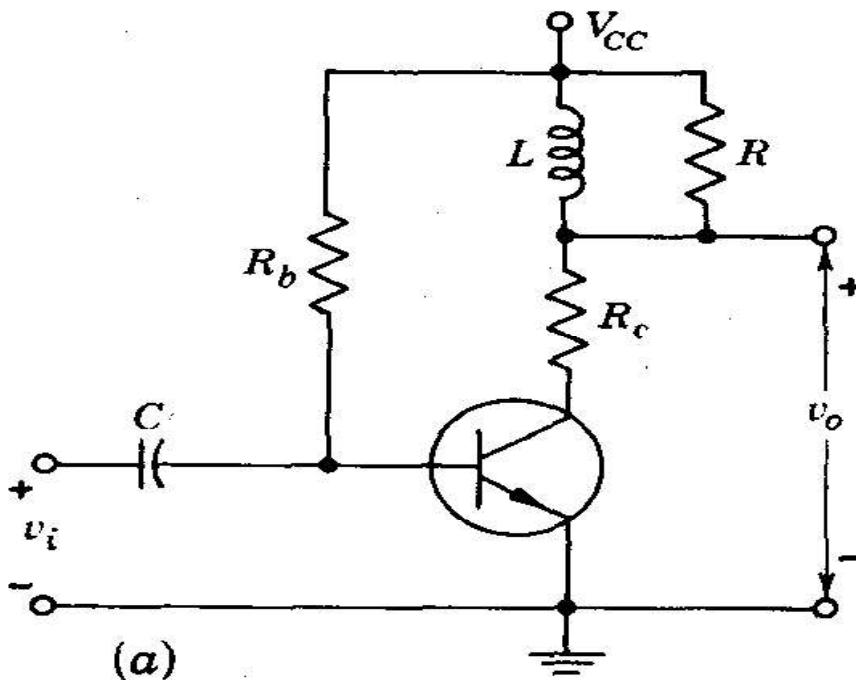
Theory:

In Transistor switch inductor is shunted by resistance R. The required bias is given then the base circuit will act as clamper and transistor will carried from saturation to cut-off. The additional resistance Rc is connected to collector circuit to control the saturation current. Thus driving the switch to cutoff at the collector a positive spike of amplitude I_0R superimposed on the supply voltage. The inductor current, which must flow through R, decays from I_0 to zero with time constant L/R. The output voltage is given by

$$v_0 = V_{cc} + I_0R\mathcal{E}^{-Rt/L}$$

When switch is returned to saturation at the time t = T_2 the inductor will acts as open circuit. The transistor current and hence current through R is

$$[V_{cc} - V_{CE}(\text{sat})]/(R + R_c)$$

Let I_0' represent the transistor current which flows when switch is turned ON.

The current will decay from I_0' to zero with time constant L/R˙ where R˙ is parallel combination of R and R_c.

The output for t > T_2 is

$$v_0 = V_{cc} - I_0\text{'}R\mathcal{E}^{-R\text{'}(t-T_2)}$$

and has the form of the negative spike superimposed on the supply voltage. The time constant of negative spike is larger thus decay slowly.

Thus circuit will act as Peaking circuit.

Waveforms:
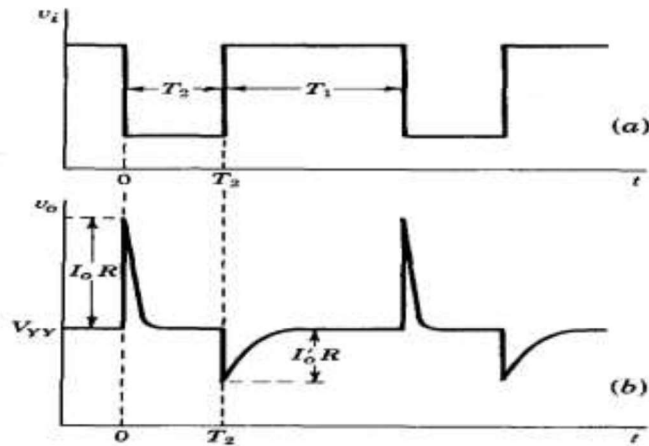
a) Input and b) output

PROCEDURE:

1. Connect the circuit as shown in the circuit diagram.

2. Usually a +Ve supply voltage of about 10 V is applied as source voltage to Vcc

3. In the normal operation transistor Q is ON and output voltage across the base and collector are noted.

4. Then an external input is applied from the function generator to the base circuit.

5. Then the transistor Q is OFF state, and then the voltage across the junction of the transistor Q is noted.

6. Draw the waveforms and calculate the output voltages at on and off switch.

Draw the results on graph.

Exp.no.8   Date:

TRANSISTOR SWITCH

Aim:

To Design Transistor switch with Capacitive load and draw output waveform for square wave input.

Components required:

1. Transistor BC 547 ----------- 1 No's
2. Capacitor 0.1 µF ------------ 2 No's
3. Resistors 1 KΩ -------------- 1 No's 10 KΩ ------------ 2No's
4. Bread Board
5. Connecting wires as required
6. CRO & Probes
7. Function Generator

 Circuit Diagram

(a)

Theory:

Initially at t = 0 - transistor is in saturation the output voltage v_0 and collector current is given by

$$v_0 = V_{ce(sat)} \text{ and } i_c = (V_{cc} - V_{ce(sat)})/ R_c = I_0$$

At t > 0+ when switch is driven to cutoff. The collector rises towards V_{cc} with time constant $R_c C_s$.

At t = T_2+ the input rises abruptly, the base emitter diode is forward biased by by V_c and base current $I_b = (V_{cc} - V_c)/ R_b$ flows. Since this voltage reverse biases the collector junction, the transistor remains in active region and collector current $I_c = h_{fe}$. $I_b = I_0'$.

The output voltage starts at a value V_{cc} and falls exponentially with time constant $R_c C_s$. towards steady state value of V_{cc} - $I_0' R_c$. For t > T_2 until transistor returns to saturation. The output is given by

$$v_o = V_{cc} - I_o' R_c + I_o' R_c \epsilon^{-(t-T_2)/R_c C_s}$$

The time constant are small in compare to T_1 or T_2 The circuit behaves like Integrator i.e., a step input is converted into ramp.

Wave forms
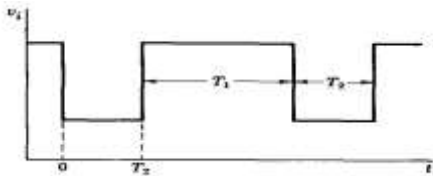
(a) Input

(b) Output

PROCEDURE:

1. Connect the circuit as shown in the circuit diagram.

2. Usually a +Ve supply voltage of about 10 V is applied as source voltage to Vcc

3. In the normal operation transistor Q is ON and output voltage across the base and collector are noted.

4. Then an external input is applied from the function generator to the base circuit.

5. Then the transistor Q is OFF state, and then the voltage across the junction of the transistor Q is noted.

6. Draw the waveforms and calculate the output voltages at on and off switch.

Draw the results on graph.

COLLEGE OF TECHNOLGY AND ENGINEERING

MAHARANA PRATAP UNIVERSITY OF AGRICULTURE & TECHNOLGY, UDAIPUR

DEPARTMENT OF ELECTRONICS AND COMMUNICATION ENGINEERING

LAB MANUAL

EC327 - System Design Lab

PREPARED BY

Mr. HIMANSHU MOHAN

INDEX

S.no.	DATE	Name of Experiment	Remarks

Exp.no.1   Date:

_____

Aim:

Study of Proteus 8.0 software.

COMPONENTS REQUIRED:

1. Proteous Software

THEORY:

About Proteus

It is a software suite containing **schematic**, **simulation** as well as **PCB designing**.

- **ISIS** is the software used to draw schematics and simulate the circuits in real time.The simulation allows human access during run time,thus providing real time simulation.
- **ARES** is used for PCB designing.It has the feature of viewing output in 3D view of the designed PCB along with components.
- The designer can also develop 2D drawings for the product.

Features

ISIS has wide range of components in its library. It has sources, signal generators, measurement and analysis tools like **oscilloscope**, voltmeter, ammeter etc., probes for real time monitoring of the parameters of the circuit, **switches**, **displays**, loads like motors and lamps, discrete components like resistors, capacitors, inductors, transformers, digital and analog Integrated circuits, semi-conductor switches, relays, microcontrollers, processors, sensors etc.

ARES offers PCB designing up to 14 inner layers, with surface mount and through hole packages. It is embedded with the foot prints of different category of components like ICs, transistors, headers, connectors and other discrete components. It offers Auto routing and manual routing options to the PCB Designer. The schematic drawn in the ISIS can be directly transferred ARES.

Starting New Design

Step 1: Open ISIS software and select New design in File menu
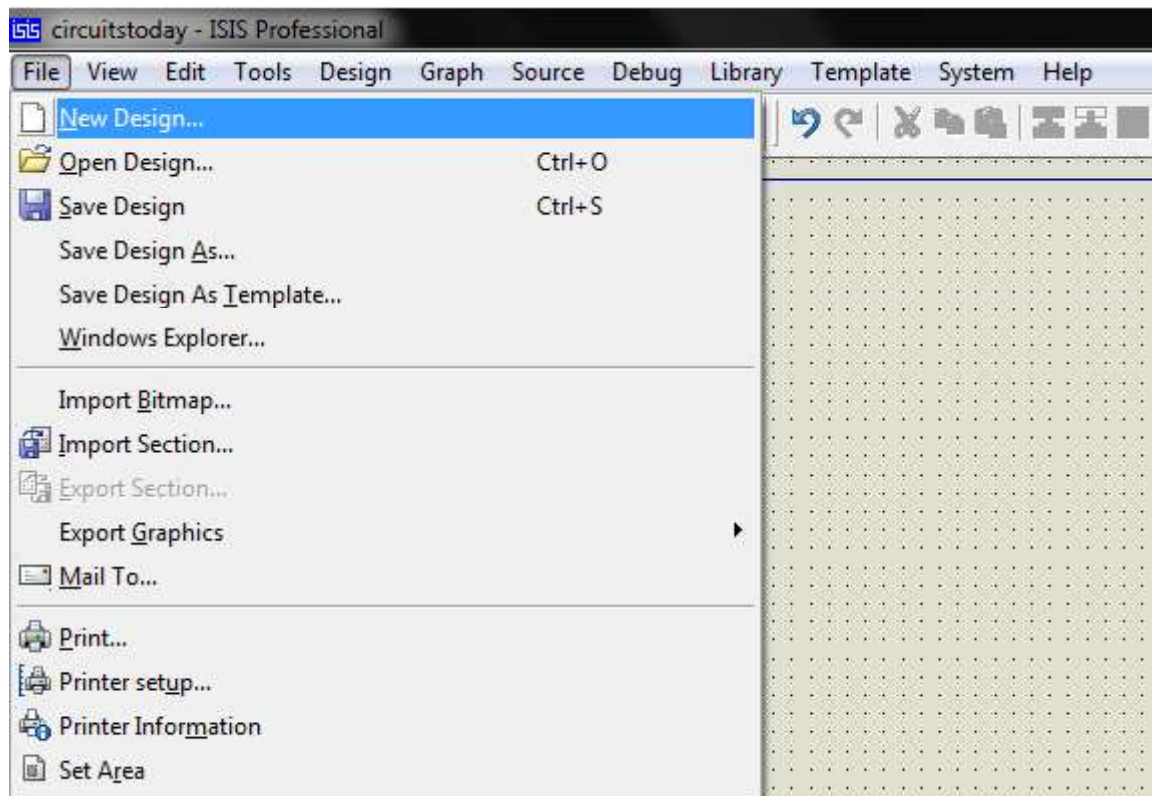
Proteus
File Menu

Step 2: A dialogue box appears to save the current design. However, we are creating a new design file so you can click Yes or No depending on the content of the present file. Then a Pop-Up appears asking to select the template. It is similar to selecting the paper size while printing. For now select default or according to the layout size of the circuit.
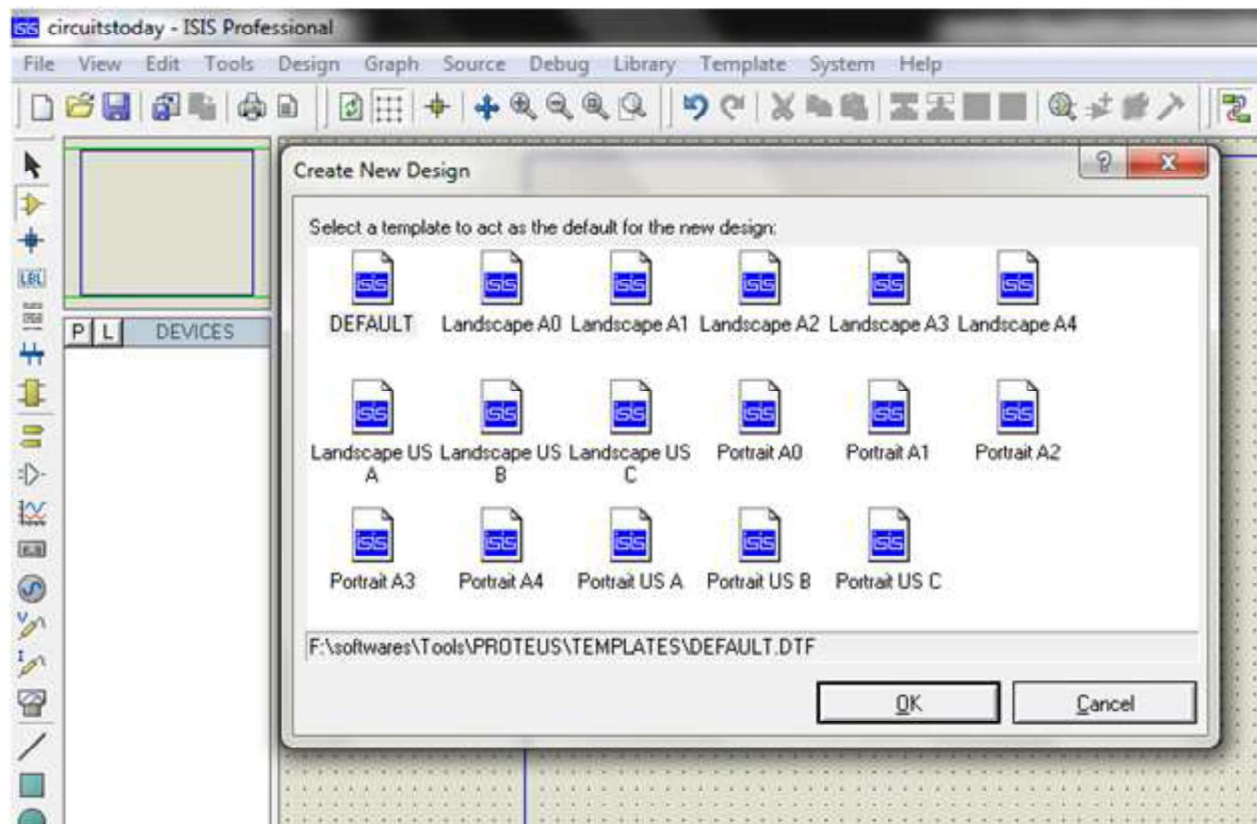
Proteus Default Template Select

Step 3:An untitled design sheet will be opened, save it according to your wish,it is better to create a new folder for every layout as it generates other files supporting your design. However,it is not mandatory.
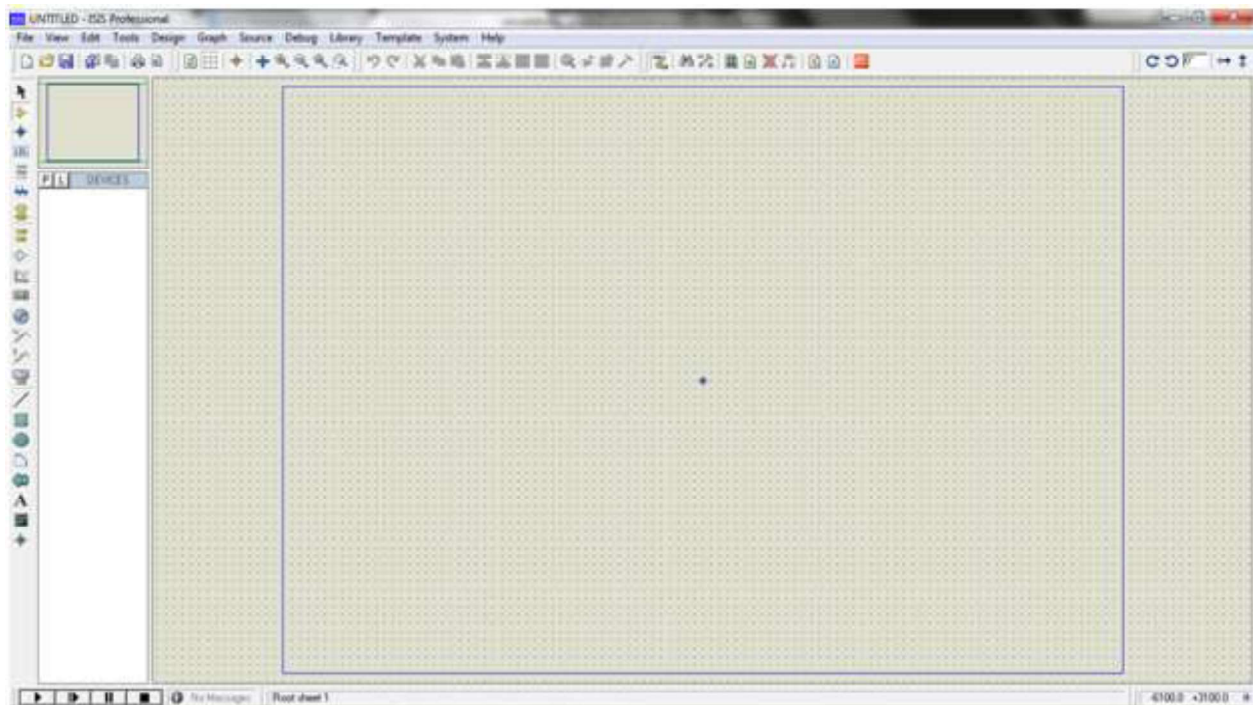
Proteus Design Sheet

Step 4:To Select components, Click on the component mode button.
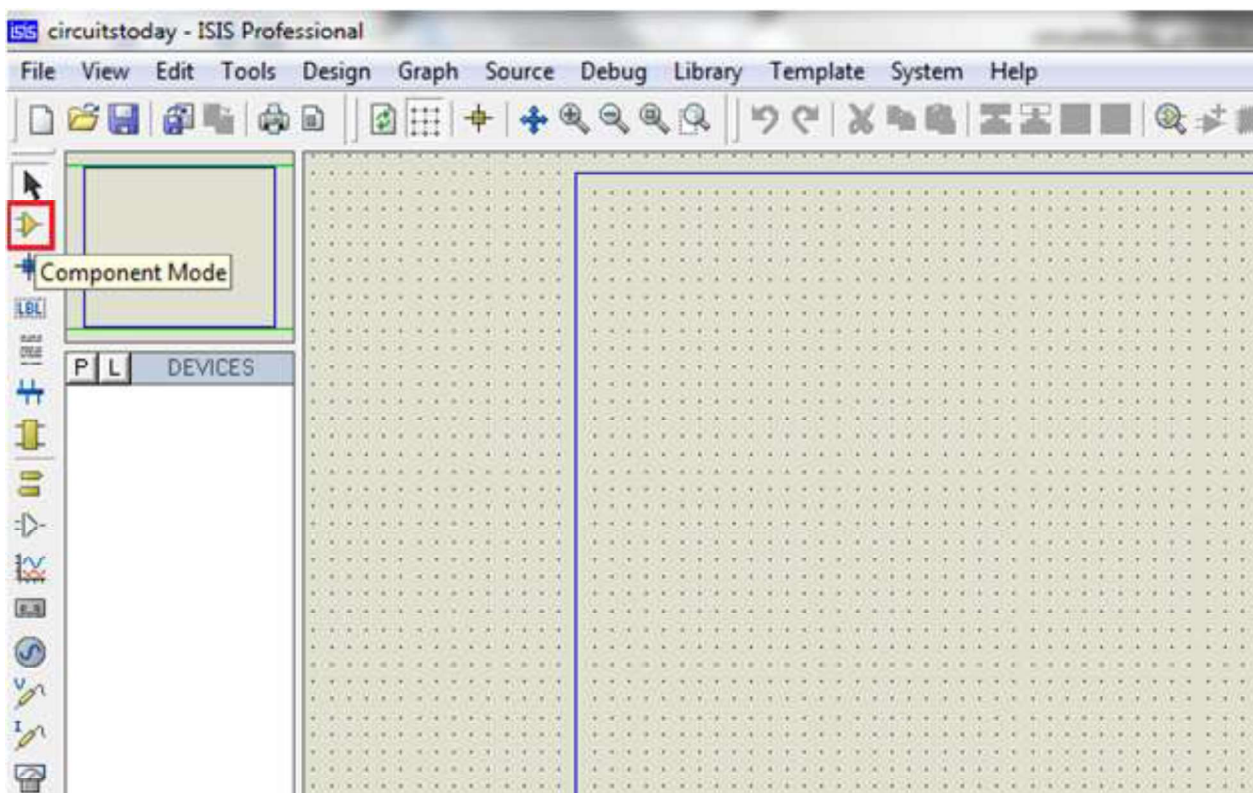
Component Mode

Step 5:Click On Pick from Libraries. It shows the categories of components available and a search option to enter the part name.
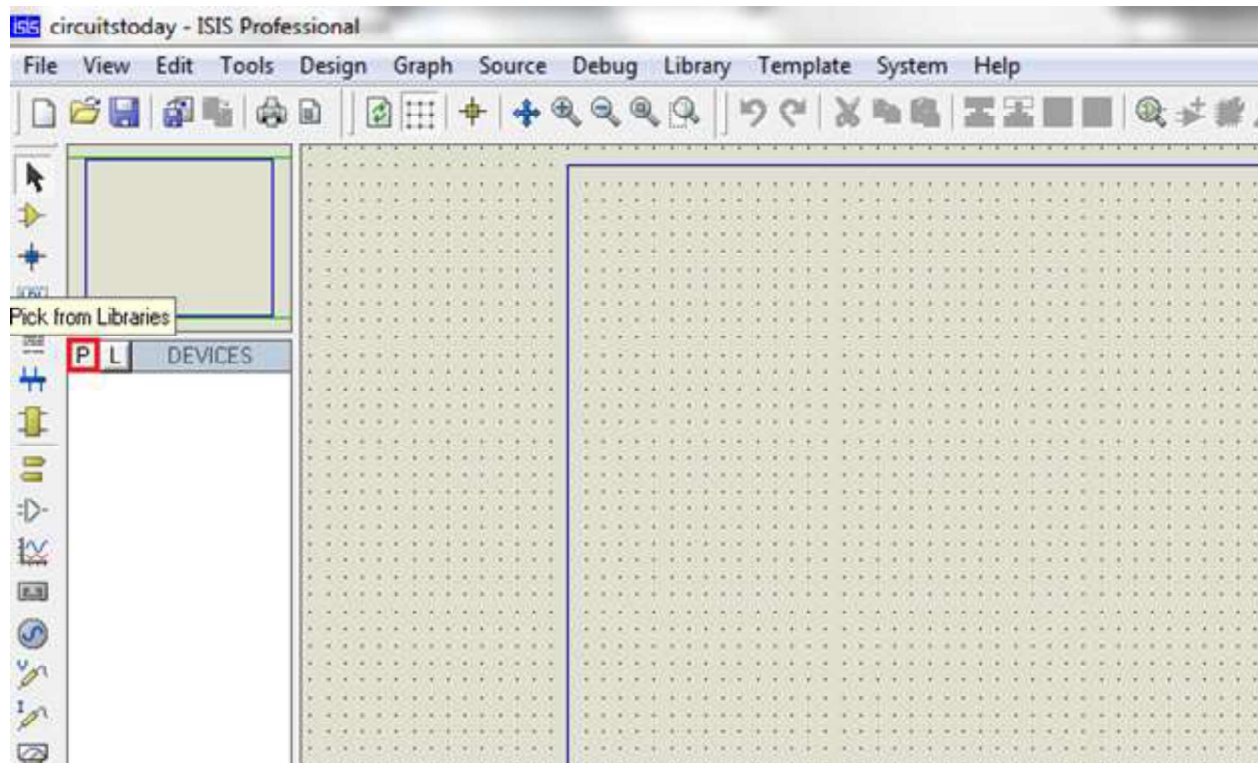
Pick from Libraries

Step 6: Select the components from categories or type the part name in Keywords text box.
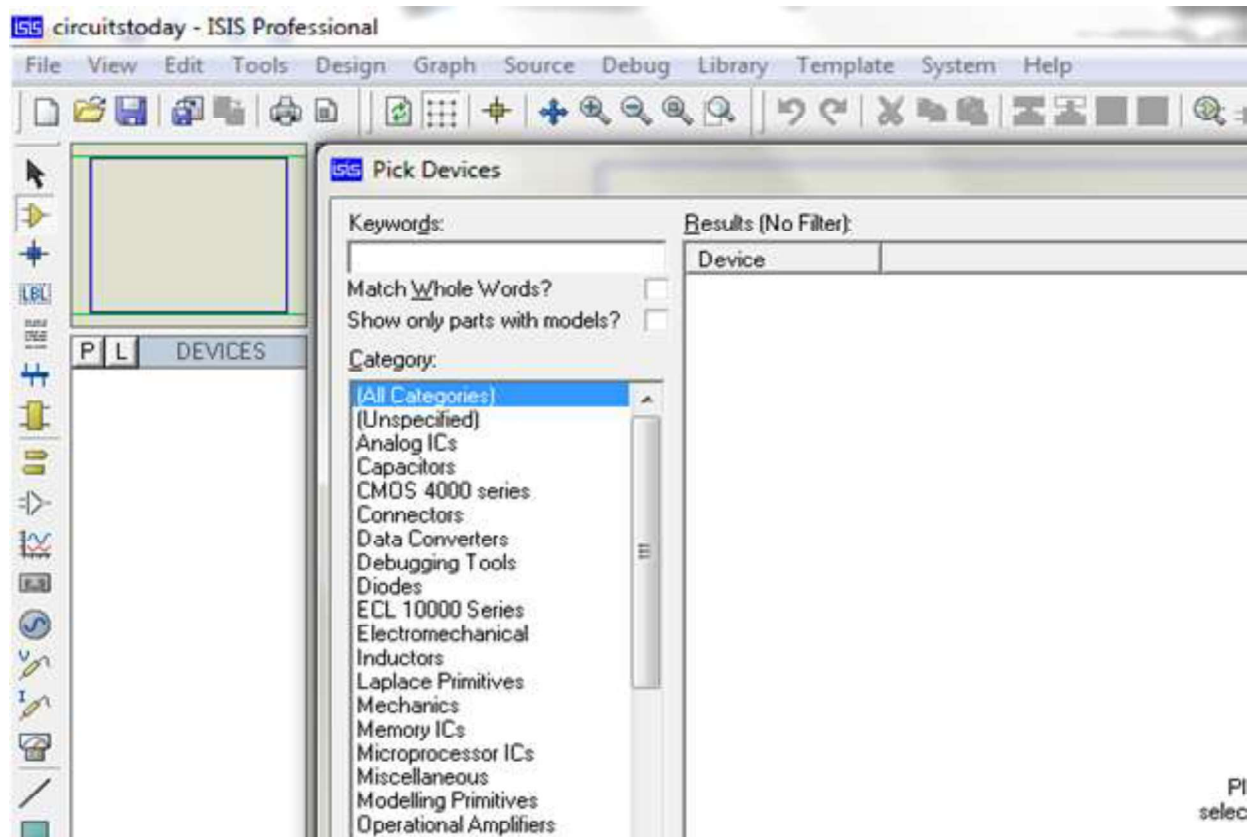
Keywords Textbox

Example shows selection of push button. Select the components accordingly.
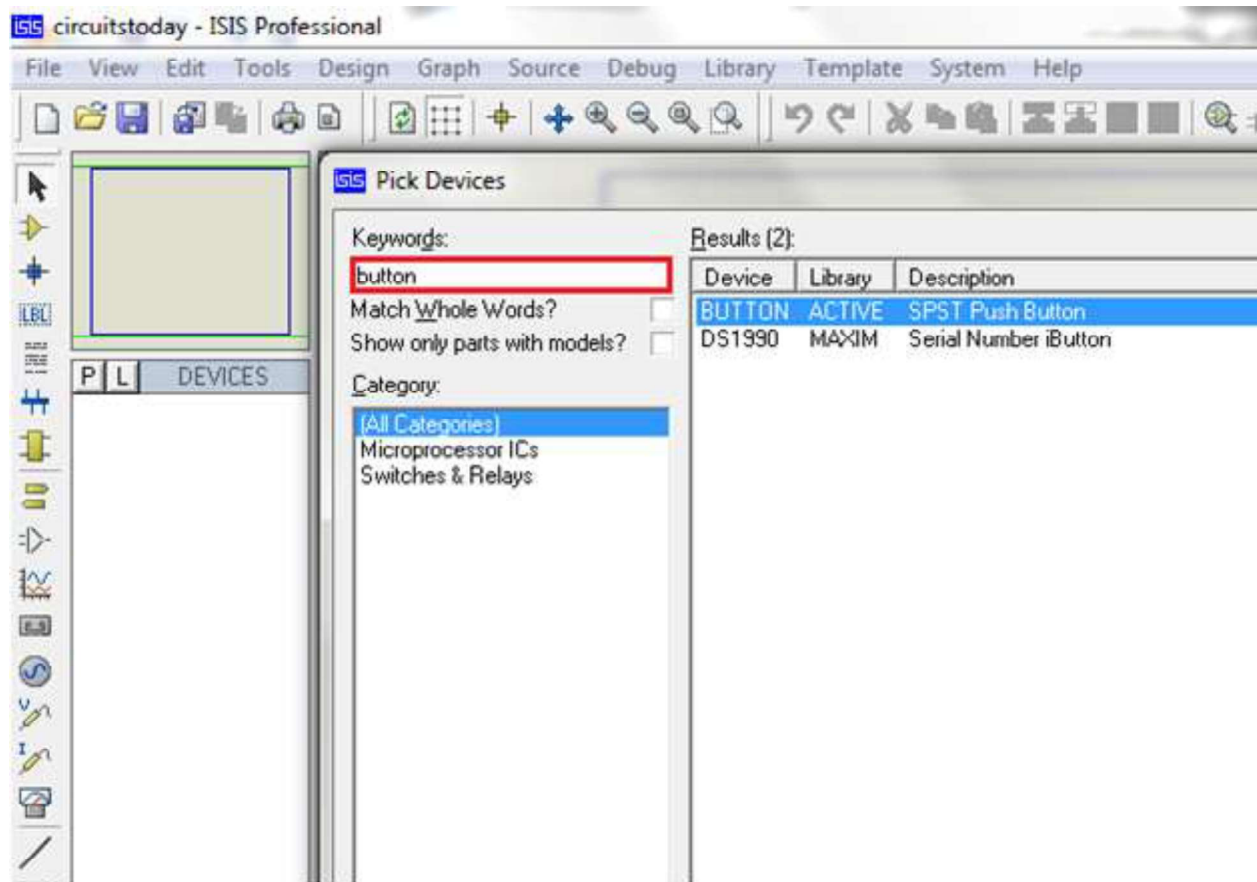
Push Button Selection

Step 7: The selected components will appear in the devices list. Select the component and place it in the design sheet by left-click.
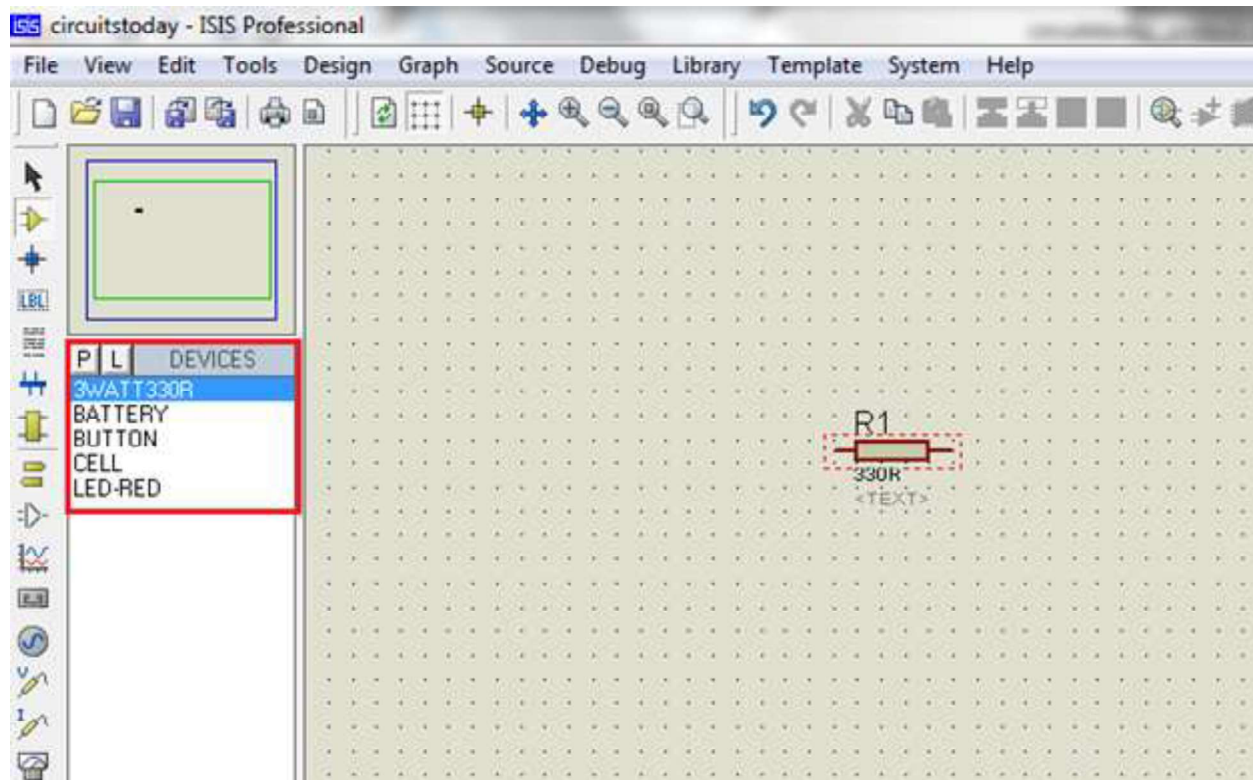
Component Selection

Place all the required components and route the wires i.e, make connections.

Either selection mode above the component mode or component mode allows to connect through wires. Left click from one terminal to other to make connection. Double right-click on the connected wire or the component to remove connection or the component respectively.
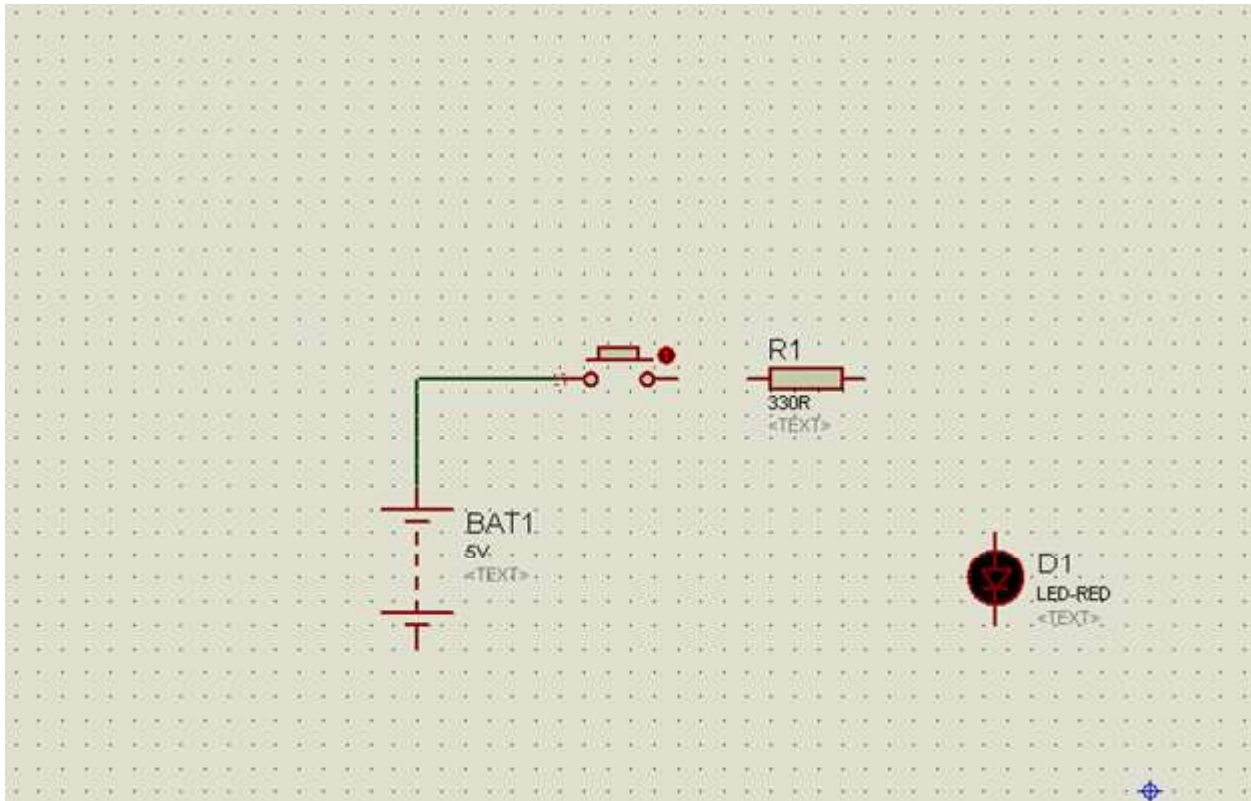
Component Properties Selection

Double click on the component to edit the properties of the components and click on Ok.
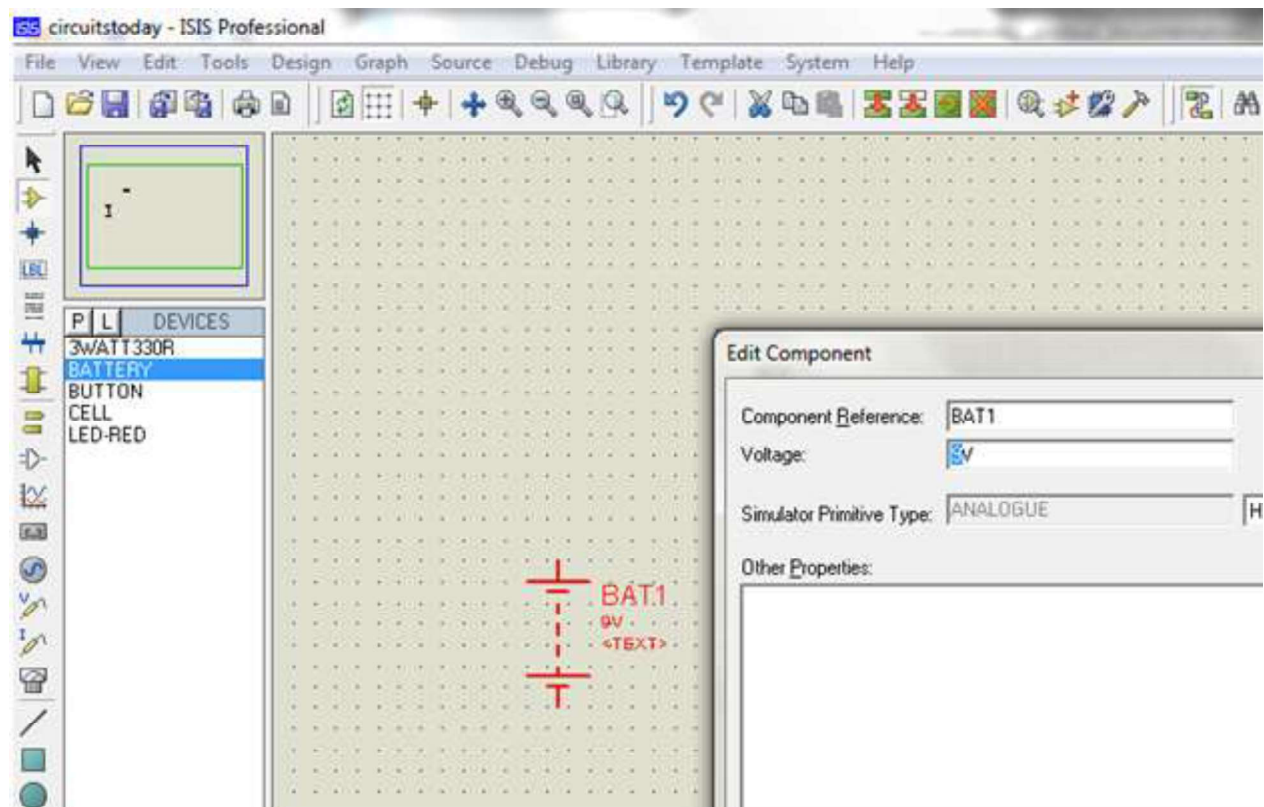
Component Properties Edit

Step 8: After connecting the circuit,click on the play button to run the simulation.
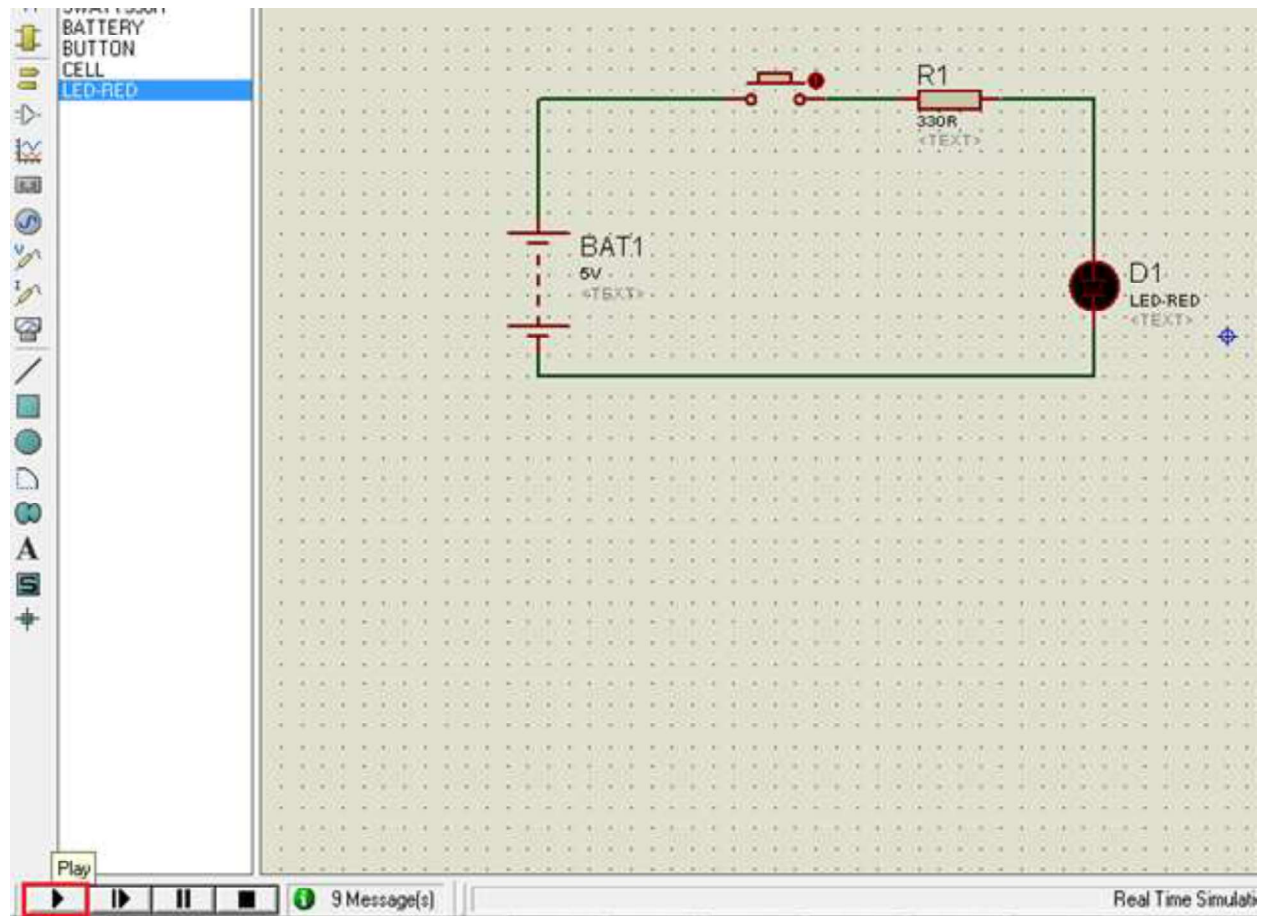
Simulation Run

In this example simulation, the button is depressed during simulation by clicking on it to make LED glow.
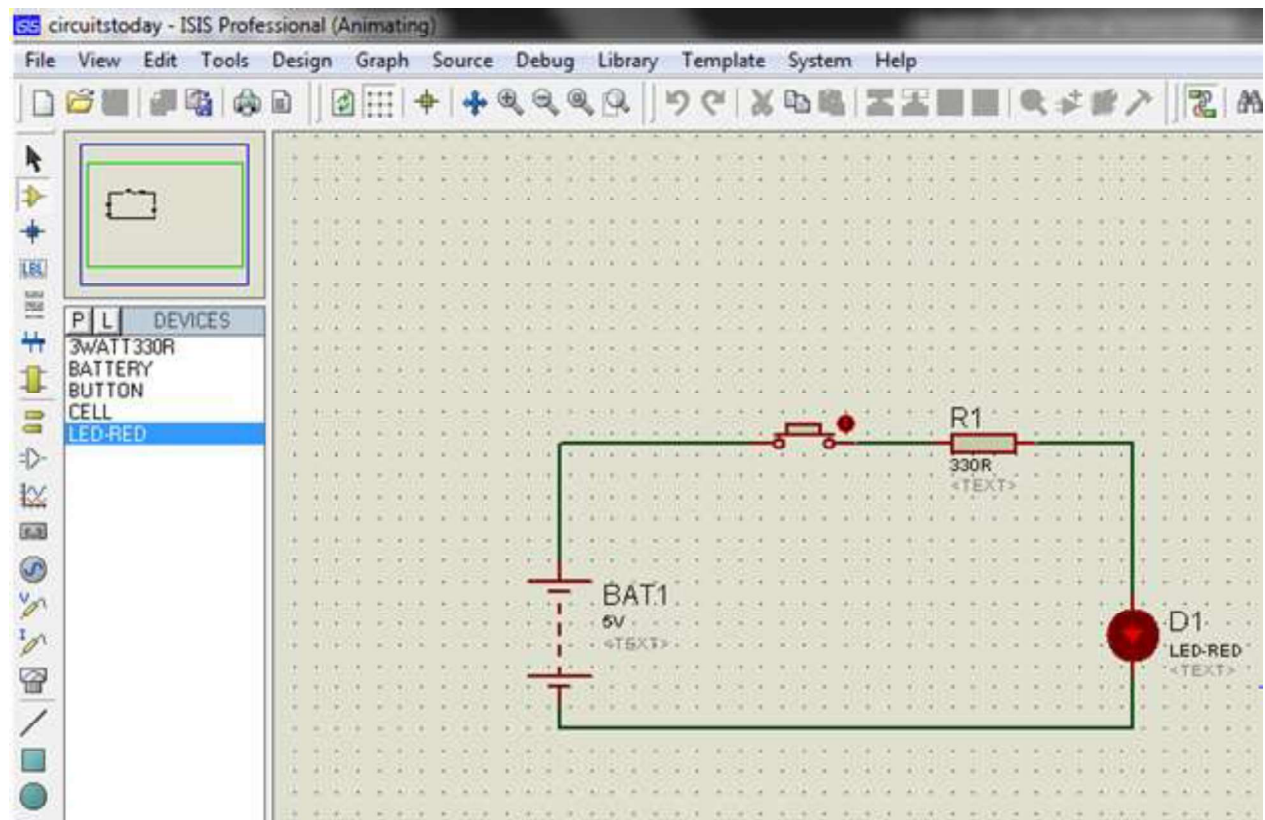
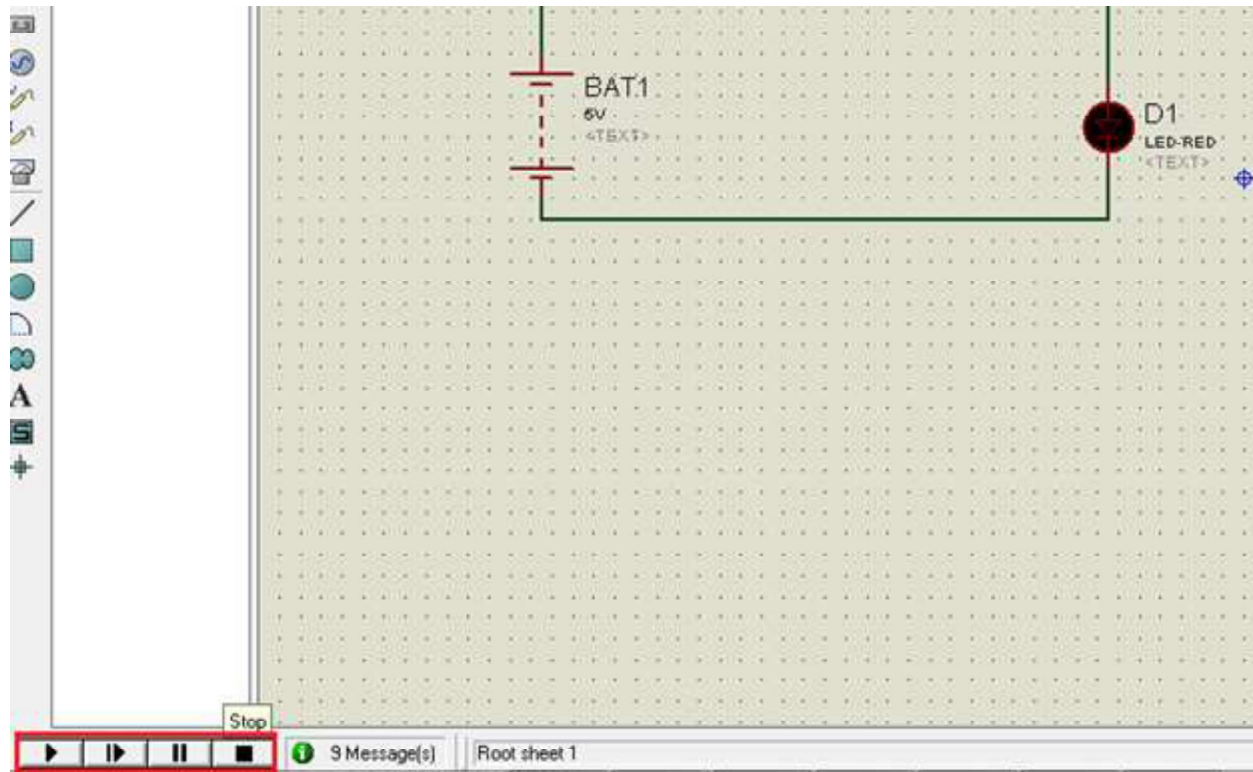Simulation Animating

Simulation can be stepped, paused or stopped at any time.

Simulation Step-Pause-Stop Buttons

Exp.no.2   Date:

_____

Aim:

Design LED Blinking Circuit using Proteus 8.0.

COMPONENTS REQUIRED:

1. Proteus Demonstration 8.0.

THEORY:

- So, get these components from Proteus components library and place it in your workspace, these components are shown in below figure:

- So, now I hope you have got all these components, now design a circuit in your Proteus software as shown in below figure:
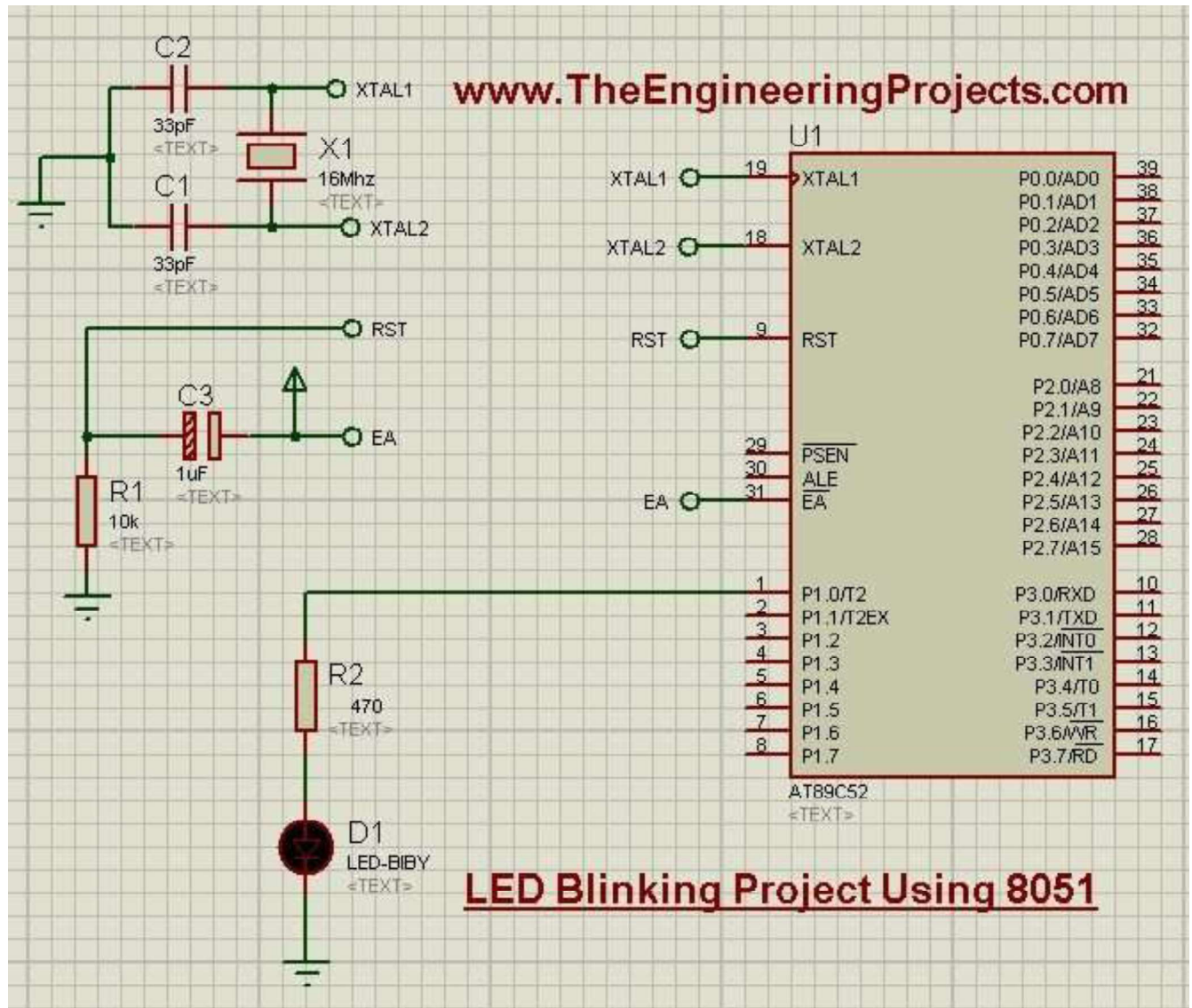
- Now you can see in the above image, I have used crystal oscillator of 16MHz which is used to provide frequency to 8051 Microcontroller.
- After that we have placed a 10k resistance in path of our Reset pin.
- LED is connected to first pin of Port 1 which is P1.0.
- So, now let's design the programming code for 8051 Microcontroller as we are done with the Proteus Simulation.

Keil Programming Code for LED Blinking Project

- Now as I have mentioned earlier, the compiler I have used for designing the programming code for LED Blinking Project is Keil micro vision 4.
- So I hope you have this installed on your computer and if not then must install it as otherwise you wont be able to compile this code, but I have also placed the hex file so that you can run the simulation easily.

- You can download them easily by clicking the Download buttons present at the end of this post.
- So now create a new project in your keil compiler and paste the below code in your c file.

```c
#include<reg51.h>

sbit LED = P1^0;

void cct_init(void);
void delay(int a);


int main(void)
{
  cct_init();

  while(1)
  {
     LED = 0;
     delay(30000);
     LED = 1;
     delay(30000);
  }
}

void cct_init(void)
{
   P1 = 0x00;
}

void delay(int a)
{
  int i;
  for(i=0;i<a;i++);
}
```

- Now let me explain this code a bit, first of all, I declare the pin1.0 as LED so that its easy to use it in our code in future.
- After that I declared two functions. One of them is the delay function which is just adding the delay, while the second function is for initialization of Port 1 as output port.
- While in the Main function, we have used the LED blinking code in which LED is ON and then OFF continuously and so that make it blink.
- Now after adding the code in your Keil software, compile it and get the hex file.
- Upload this hex file into your 8051 Microcontroller which I have used is AT89C52 and hit the RUN button.
- If everything's goes fine then you will get results as shown in below figure:
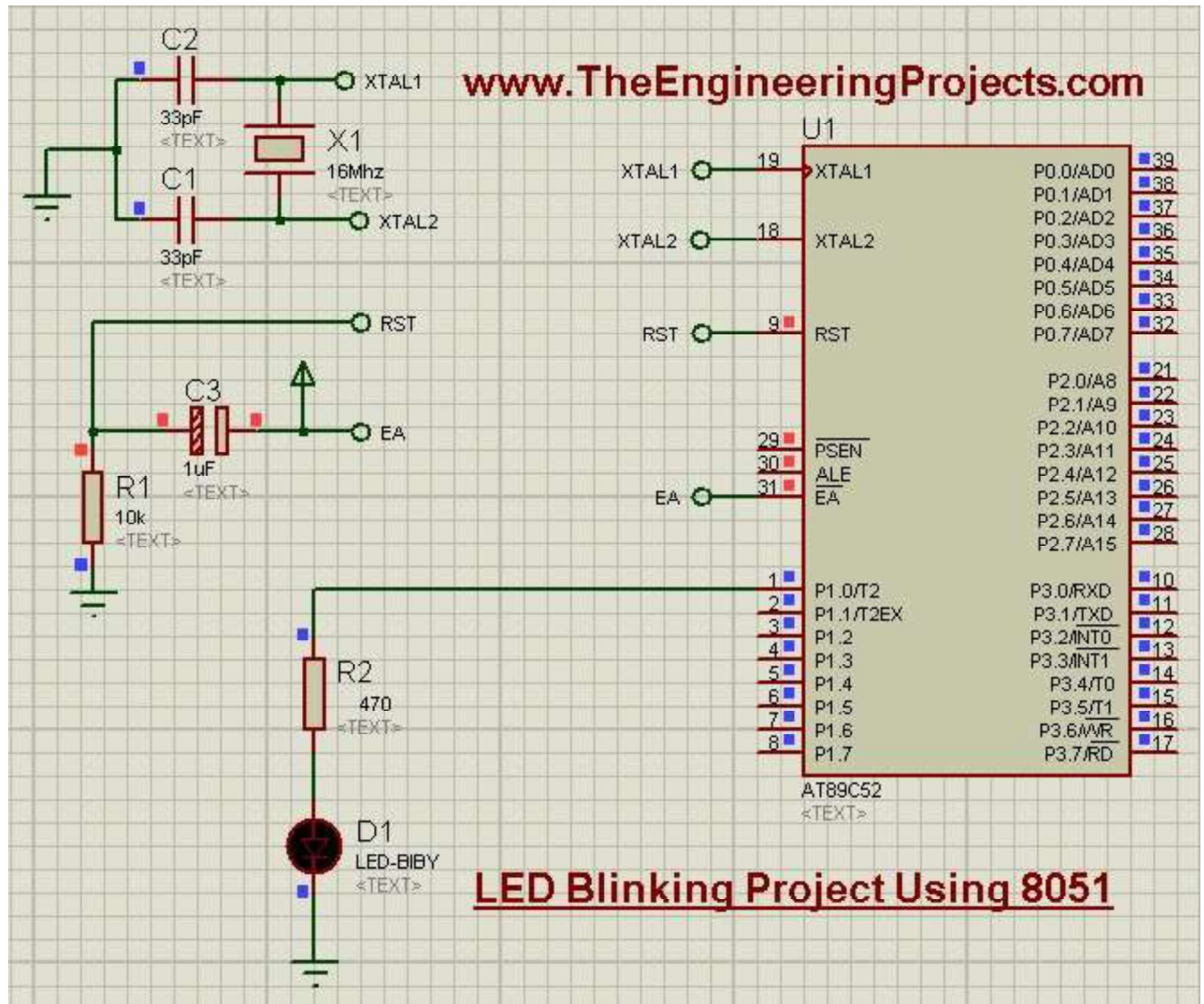
- Now click the below button to get the simulation and the programming code

RESULT:
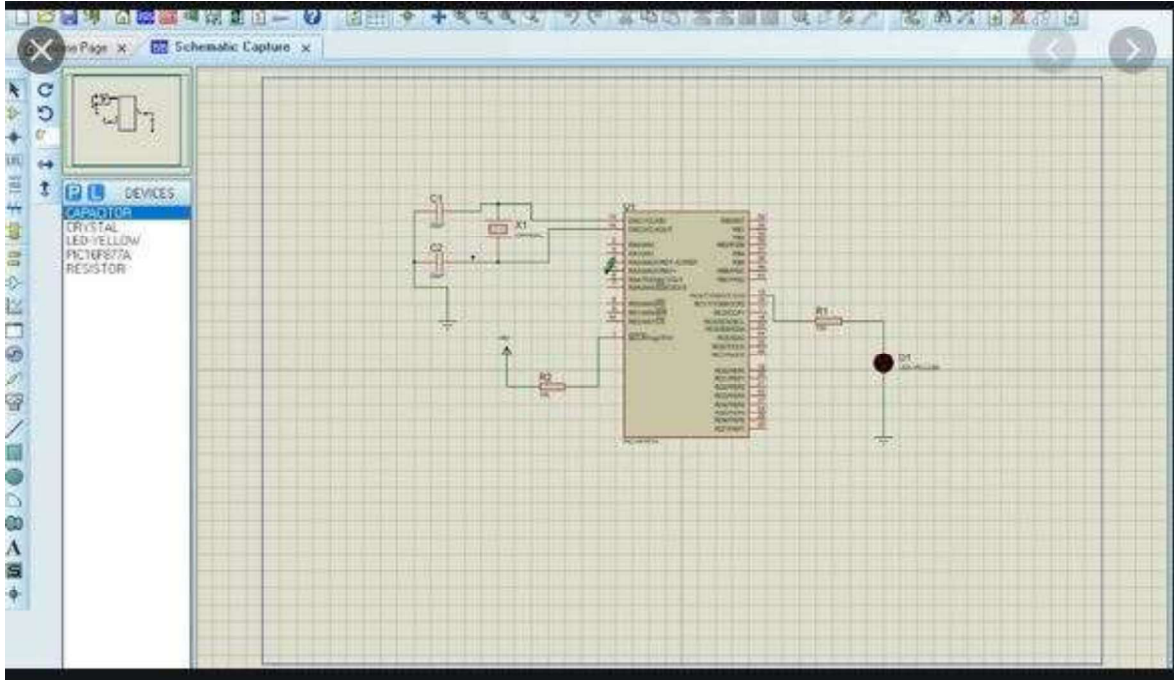
Exp.no.3   Date:

_____

AIM:

To design 555 timer circuit using Proteus 8.0.
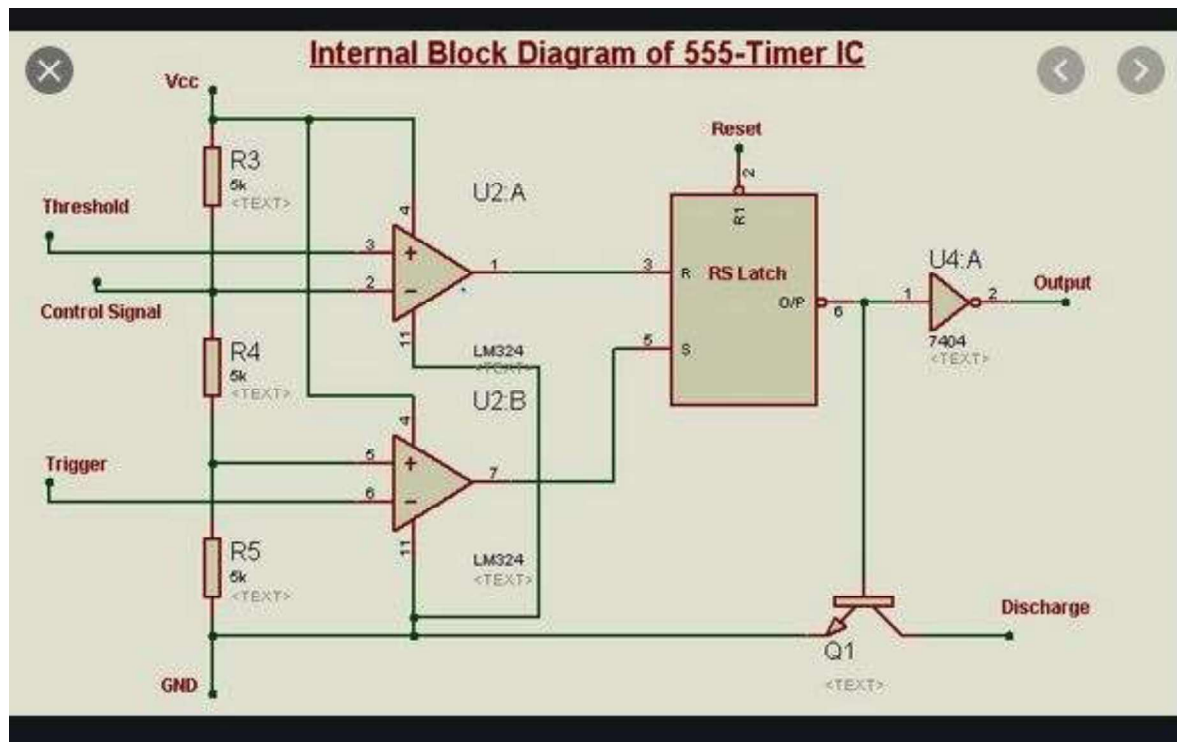
COMPONENTS REQUIRED:

1. Proteus 8.0

THEORY:

555-Timer is one of the most popular and mostly used ICs. It best suits for **timing/timekeeping related circuits**. It consists of two **operational amplifiers** operated in an open loop or comparator mode, RS Latch with additional Reset input, a discharge transistor, an inverting buffer and an amplifier in the output stage. It has a **voltage divider circuit** with three 5K Ohm resistors in series. 556 is a dual timer IC. The **internal block diagram of 555** is as follows.
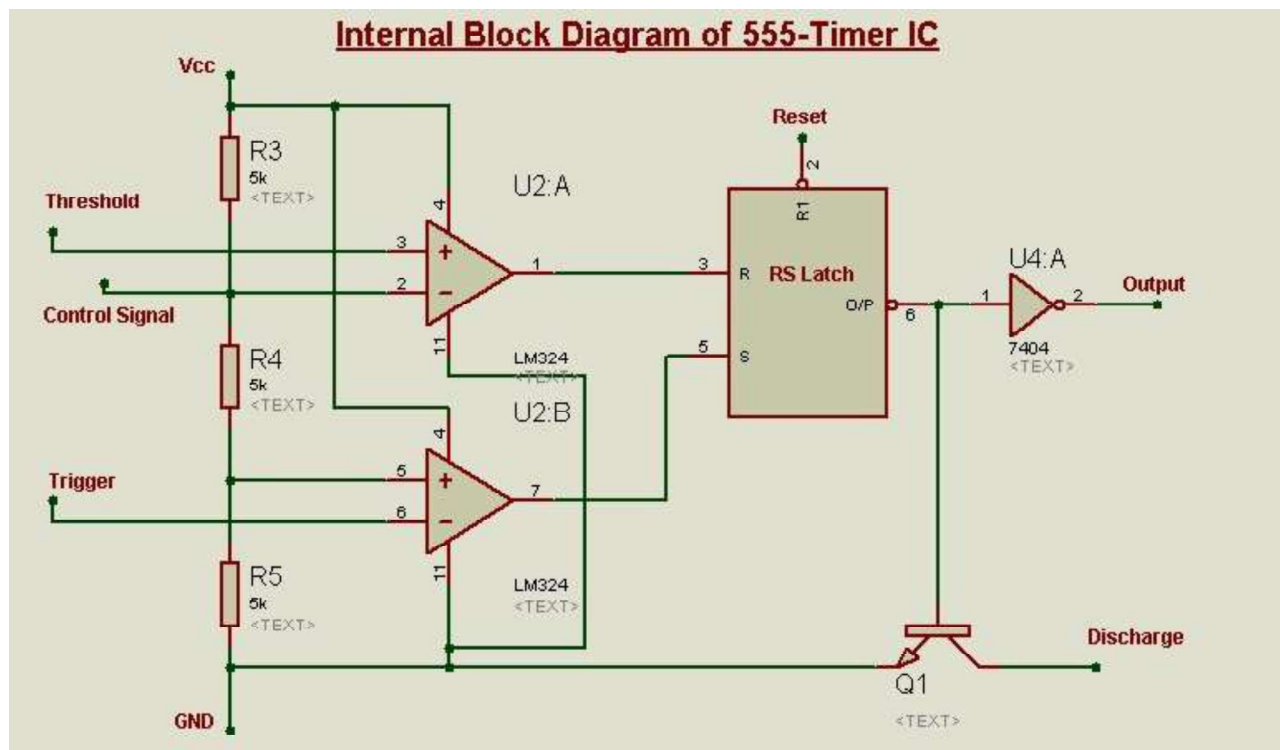
Internal Block diagram of 555-TimerCIRCUIT DIAGRAMS

Result:

Internal Block Diagram of 555-Timer IC

Exp.no.4   Date:

_____

AIM:

To design BCD to Seven Segment using Proteus 8.0.

COMPONENTS REQUIRED:

1. Proteus 8.0 Demonstration software.

THEORY:

However to display the characters and numbers (in order to produce the decimal readout), seven-segment displays are most commonly used. Mostly these displays are driven by the output stages of digital ICs (to which the visual indication of the output stages has to be performed) such as latches and decade counters, etc.

But these outputs are in the form of 4-bit binary coded decimal (BCD), and not suitable for directly driving the **seven-segment displays**.

A display decoder is used to convert a BCD or a binary code into a 7 segment code. It generally has 4 input lines and 7 output lines. Here we design a simple display decoder circuit using logic gates.

Even though commercial BCD to 7 segment decoders are available, designing a display decoder using logic gates may prove to be beneficial from economical as well as knowledge point of view.

Back to top

Principle of Display Decoder Circuit

The basic idea involves driving a common cathode 7-segment LED display using combinational logic circuit. The logic circuit is designed with 4 inputs and 7 outputs, each representing an input to the display IC. Using Karnough's map, logic circuitry for each input to the display is designed.

Related Post: Water Level Indicator using AVR Microcntroller

Theory Behind the Circuit:

The first and foremost aspect of this circuit is decoder. A decoder is a combinational circuit which is used to convert a binary or BCD (Binary Coded Decimal) number to the corresponding decimal number . It can be a simple binary to decimal decoder or a BCD to 7 segment decoder.

Another relevant section is the combinational logic circuitry. A combinational logic circuit is a system of logic gates consisting of only outputs and inputs. The output of a combinational logic circuit depends only on the present state of the inputs and nothing else. Best examples of such circuits are Encoders and Decoders, Multiplexers and De-multiplexers, Adders, Subtractors etc.

To understand the design and operation of these logic circuits, one needs to have a good knowledge about Boolean algebra and logic gates. For example few basic Boolean algebra rules to be followed are the complementary law, associative law, De-Morgan's law etc.

A 7 segment LED display consists of an arrangement of 8 LEDs such that either all the anodes are common or cathodes are common. A common cathode 7 segment display consists of 8 pins – 7 input pins labeled from 'a' to 'g' and 8th pin as common ground pin.

RESULT:
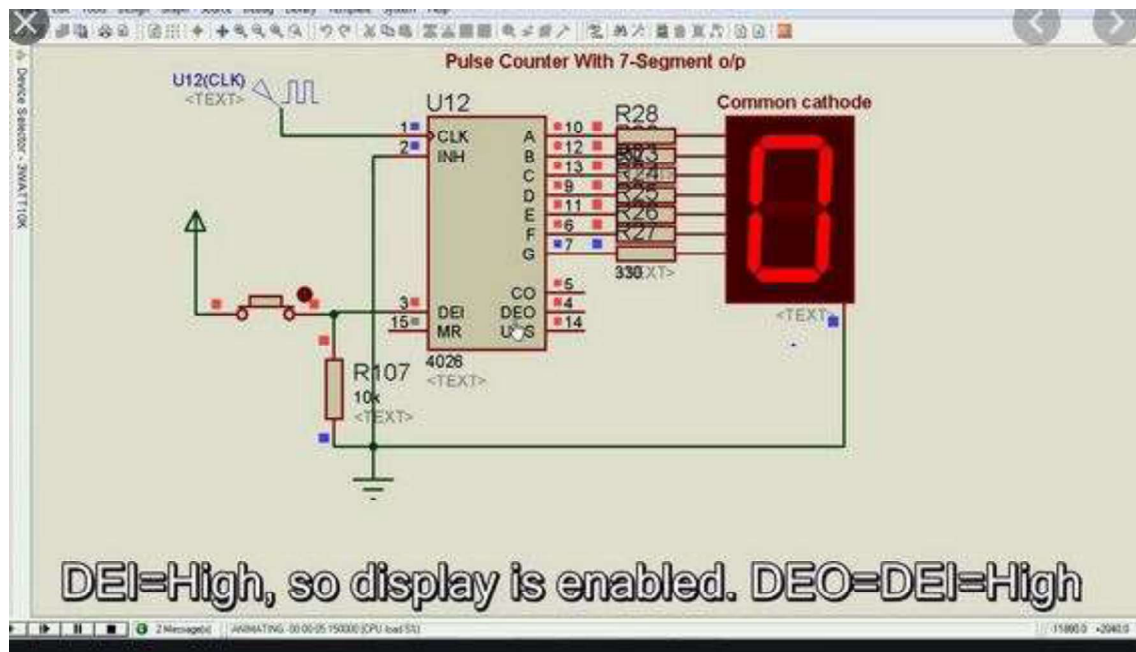
Exp.no.5   Date:

AIM:

Design Voltage Divider Circuit using Proteus 8.0.

COMPONENTS REQUIRED:

Proteus 8.0

THEORY:

In electronics, a **voltage divider** (also known as a **potential divider**) is a passive linear circuit that produces an output voltage (V_{out}) that is a fraction of its input voltage (V_{in}). **Voltage division** is the result of distributing the input voltage among the components of the divider. A simple example of a voltage divider is two resistors connected in series, with the input voltage applied across the resistor pair and the output voltage emerging from the connection between them.

Resistor voltage dividers are commonly used to create reference voltages, or to reduce the magnitude of a voltage so it can be measured, and may also be used as signal attenuators at low frequencies. For direct current and relatively low frequencies, a voltage divider may be sufficiently accurate if made only of resistors; where frequency response over a wide range is required (such as in an oscilloscope probe), a voltage divider may have capacitive elements added to compensate load capacitance. In electric power transmission, a capacitive voltage divider is used for measurement of high voltage
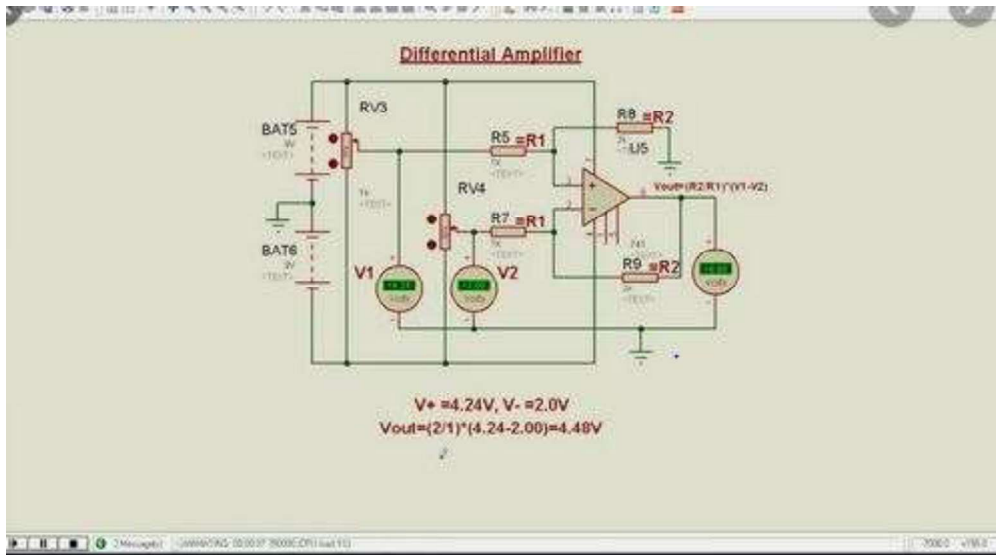
Exp.no.6   Date:

AIM:

Study of Arduino.

What is Arduino?

Arduino is an open-source electronics platform based on easy-to-use hardware and software. Arduino boards are able to read inputs - light on a sensor, a finger on a button, or a Twitter message - and turn it into an output - activating a motor, turning on an LED, publishing something online. You can tell your board what to do by sending a set of instructions to the microcontroller on the board. To do so you use the Arduino programming language (based on Wiring), and the Arduino Software (IDE), based on Processing.

Over the years Arduino has been the brain of thousands of projects, from everyday objects to complex scientific instruments. A worldwide community of makers - students, hobbyists, artists, programmers, and professionals - has gathered around this open-source platform, their contributions have added up to an incredible amount of accessible knowledge that can be of great help to novices and experts alike.

Arduino was born at the Ivrea Interaction Design Institute as an easy tool for fast prototyping, aimed at students without a background in electronics and programming. As soon as it reached a wider community, the Arduino board started changing to adapt to new needs and challenges, differentiating its offer from simple 8-bit boards to products for IoT applications, wearable, 3D printing, and embedded environments. All Arduino boards are completely open-source, empowering users to build them independently and eventually adapt them to their particular needs. The software, too, is open-source, and it is growing through the contributions of users worldwide.

Why Arduino?

Thanks to its simple and accessible user experience, Arduino has been used in thousands of different projects and applications. The Arduino software is easy-to-use for beginners, yet flexible enough for advanced users. It runs on Mac, Windows, and Linux. Teachers and students use it to build low cost scientific instruments, to prove chemistry and physics principles, or to get started with programming and robotics. Designers and architects build interactive prototypes, musicians and artists use it for installations and to experiment with new musical instruments. Makers, of course, use it to build many of the projects exhibited at the Maker Faire, for example. Arduino is a key tool to learn new things. Anyone - children, hobbyists, artists, programmers - can start tinkering just following the step by step instructions of a kit, or sharing ideas online with other members of the Arduino community.

There are many other microcontrollers and microcontroller platforms available for physical computing. Parallax Basic Stamp, Netmedia's BX-24, Phidgets, MIT's Handyboard, and many

others offer similar functionality. All of these tools take the messy details of microcontroller programming and wrap it up in an easy-to-use package. Arduino also simplifies the process of working with microcontrollers, but it offers some advantage for teachers, students, and interested amateurs over other systems:

- **Inexpensive** - Arduino boards are relatively inexpensive compared to other microcontroller platforms. The least expensive version of the Arduino module can be assembled by hand, and even the pre-assembled Arduino modules cost less than $50
- **Cross-platform** - The Arduino Software (IDE) runs on Windows, Macintosh OSX, and Linux operating systems. Most microcontroller systems are limited to Windows.
- **Simple, clear programming environment** - The Arduino Software (IDE) is easy-to-use for beginners, yet flexible enough for advanced users to take advantage of as well. For teachers, it's conveniently based on the Processing programming environment, so students learning to program in that environment will be familiar with how the Arduino IDE works.
- **Open source and extensible software** - The Arduino software is published as open source tools, available for extension by experienced programmers. The language can be expanded through C++ libraries, and people wanting to understand the technical details can make the leap from Arduino to the AVR C programming language on which it's based. Similarly, you can add AVR-C code directly into your Arduino programs if you want to.
- **Open source and extensible hardware** - The plans of the Arduino boards are published under a Creative Commons license, so experienced circuit designers can make their own version of the module, extending it and improving it. Even relatively inexperienced users can build the breadboard version of the module in order to understand how it works and save money.

How do I use Arduino?

See the getting started guide. If you are looking for inspiration you can find a great variety of Tutorials on Arduino Project Hub.

The text of the Arduino getting started guide is licensed under a Creative Commons Attribution-ShareAlike 3.0 License. Code samples in the guide are released into the public domain.

Exp.no.7   Date:

Aim:

To Design LED Blinking using Arduino UNO.

Components required:

Blinking LED

The first program every programmer learns consists in writing enough code to make their code show the sentence "Hello World!" on a screen. The blinking LED is the "Hello World!" of physical computing.

LEDs

An LED is a small light (it stands for "light emitting diode") that works with relatively little power. The Arduino board has one built-in on digital pin 13.

Code

To blink the LED takes only a few lines of code. The first thing we do is define a variable that will hold the number of the pin that the LED is connected to. We don't have to do this (we could just use the pin number throughout the code) but it makes it easier to change to a different pin. We use an integer variable (called an int).

```
int ledPin = 13;
```

The second thing we need to do is configure as an output the pin connected to the LED. We do this with a call to the pinMode() function, inside of the sketch's setup() function:

```
void setup()
{
  pinMode(ledPin, OUTPUT);
}
```

Finally, we have to turn the LED on and off with the sketch's loop() function. We do this with two calls to the digitalWrite() function, one with HIGH to turn the LED on and one with LOW to turn the LED off. If we simply alternated calls to these two functions, the LED would turn on and off too quickly for us to see, so we add two calls to delay() to slow things down. The delay function works with milliseconds, so we pass it 1000 to pause for a second.

```
void loop()
{
  digitalWrite(ledPin, HIGH);
  delay(1000);
  digitalWrite(ledPin, LOW);
  delay(1000);
}
```

full sketch code: digital_write

Upload the sketch to the board and you should see the on-board LED begin to blink: on for one second, off for the next.
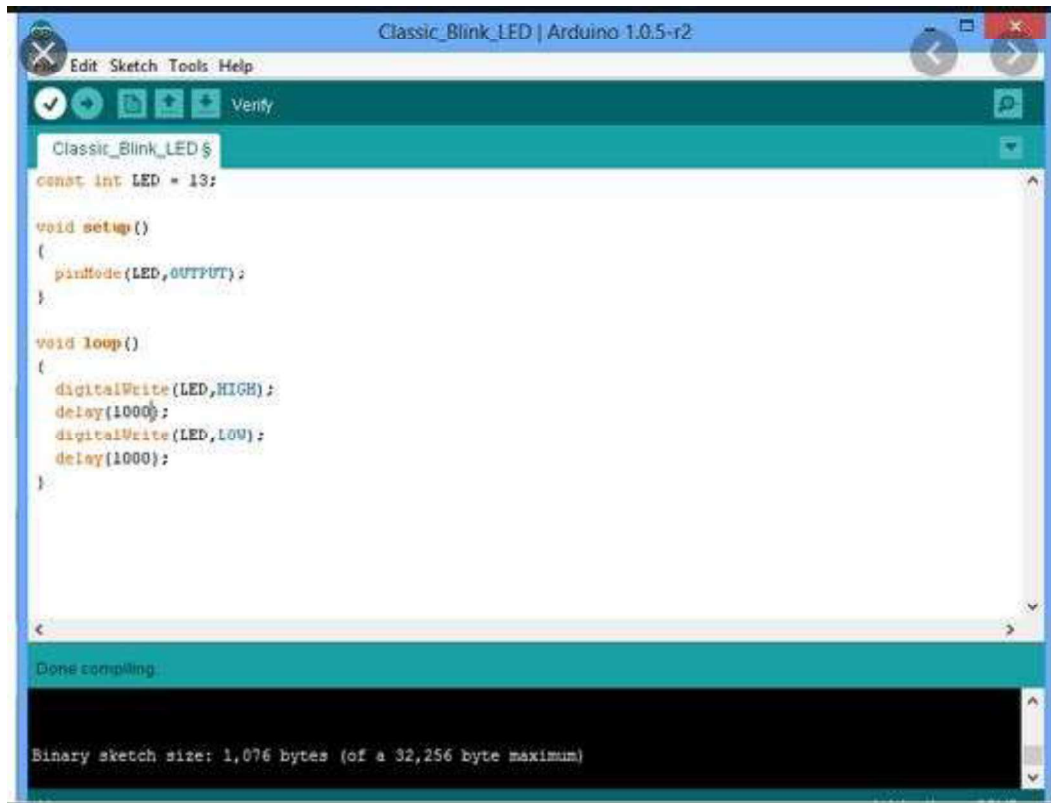
Connecting an LED

LEDs have polarity, which means they will only light up if you orient the legs properly. The long leg is typically positive, and should connect to a digital pin on the Arduino board. The short leg goes to GND; the bulb of the LED will also typically have a flat edge on this side.

In order to protect the LED, you will also need use a resistor "in series" with the LED.

If the LED doesn't light up, trying reversing the legs (you won't hurt the LED if you plug it in backwards for a short period of time).
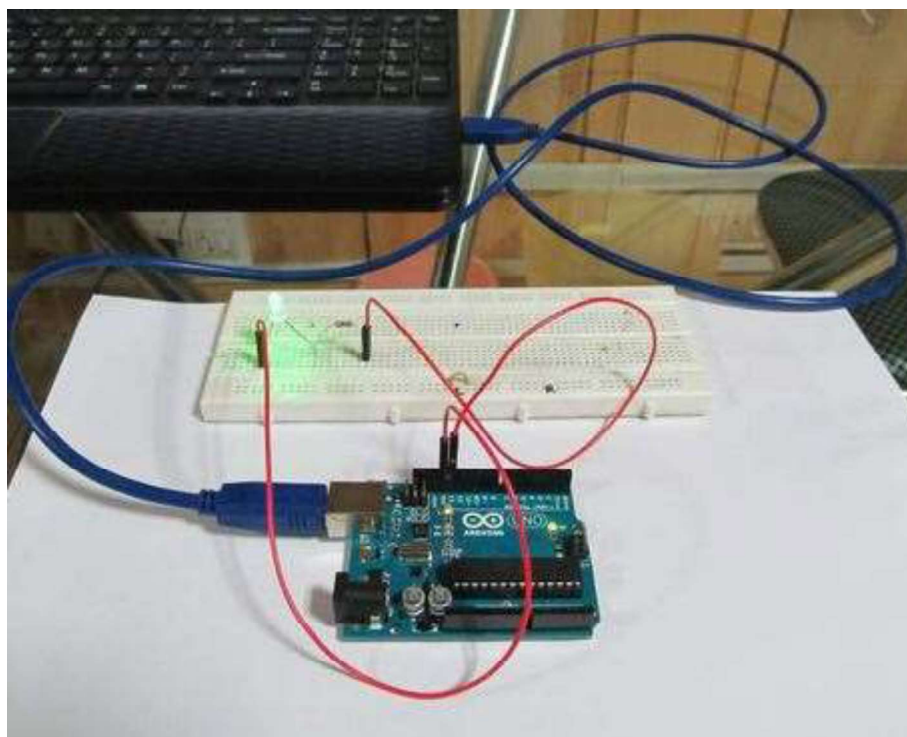
Exp.no.8   Date:

Aim:

To Design Power Supply circuit using Proteus 8.0.

Components required:

Proteus 8.0

Theory:

A **power supply** is an electrical device that supplies electric power to an electrical load. The primary function of a power supply is to convert electric current from a source to the correct voltage, current, and frequency to power the load. As a result, power supplies are sometimes referred to as electric power converters. Some power supplies are separate standalone pieces of equipment, while others are built into the load appliances that they power. Examples of the latter include power supplies found in desktop computers and consumer electronics devices. Other functions that power supplies may perform include limiting the current drawn by the load to safe levels, shutting off the current in the event of an electrical fault, power conditioning to prevent electronic noise or voltage surges on the input from reaching the load, power-factor correction, and storing energy so it can continue to power the load in the event of a temporary interruption in the source power (uninterruptible power supply).

All power supplies have a *power input* connection, which receives energy in the form of electric current from a source, and one or more *power output* connections that deliver current to the load. The source power may come from the electric power grid, such as an electrical outlet, energy storage devices such as batteries or fuel cells, generators or alternators, solar power converters, or another power supply. The input and output are usually hardwired circuit connections, though some power supplies employ wireless energy transfer to power their loads without wired connections. Some power supplies have other types of inputs and outputs as well, for functions such as external monitoring and control.