BAPATLA ENGINEERING COLLEGE

ELECTRONIC CIRCUITS -1LAB (EC261)

PREPARED BY

M. BABY M.Tech

LECTURER

&

S. ANUSHA B.Tech

LECTURER

Department of Electronics and Communications Engineering
Bapatla Engineering College
(Affiliated to Acharya Nagarjuna University)
Bapatla-522101

2008-2009

1. FULL WAVE RECTIFIER

Aim:

01. To observe the output waveform of full wave rectifier with and without filter

02. To find ripple factor and percentage regulation of FWR with & without filter

Apparatus:

Silicon Diodes BY126

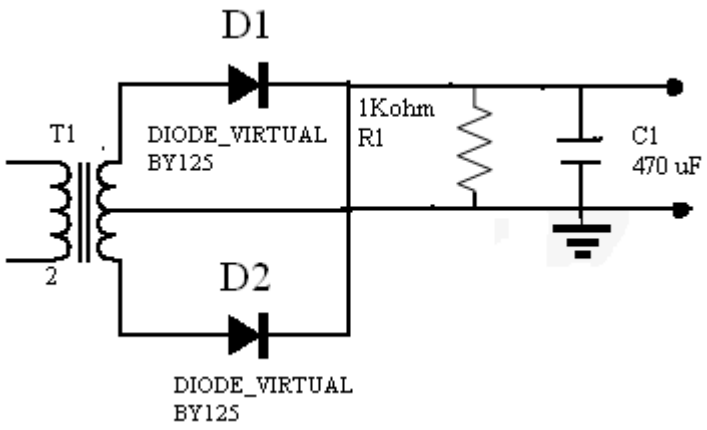
Resistance 1 kΩ

Capacitor 470 uF

CRO

Digital Multimeter

Circuit Diagram:



Theory:

The full wave rectifier consists of two half wave rectifier circuits with common load. These are connected in such a way that conduction takes place through two diodes in alternate half-cycles and current through the load is sum of two currents. Thus, the output voltage waveform contains two half sinusoids in the two half-cycles of the AC input signal.

The output of a rectifier is a pulsating DC consisting of a DC component and superimposed ripple. A way to eliminate or reduce the ripple to the required level is to use a filter.

PROCEDURE:

Without filter:

01. Connect the circuit as per the circuit diagram
02. Connect CRO across the load
03. Note down the peak value VM of the signal observed on the CRO
04. Switch the CRO into DC mode and observe the waveform. Note down the DC shift
05. Calculate $Vrm_s$and Vd_c values by using the formulae

    Calculate V_{rms} & Vdc by using the formulas

    Vrms=Vm/√2, Irms=Im/√2

    $Vdc=2V_M/\pi$, $Idc=2I_M/\pi$,

    Where V_r is the peak to peak amplitude of filter output
06. Calculate the ripple factor by using the formulae

    Ripple factor=Vac/Vdc=$\sqrt{Vrms^2 - Vdc^2}$ /Vdc
07. Remove the load and measure the voltage across the circuit. Take down the value as V_{NL}; calculate the percentage of voltage regulation using the formulae

    % Regulation = $(V_{NL}-V_{FL})/V_{FL}*_{100}$

With filter:

01. Connect the capacitor filler across the load in the above circuit diagram
02. Proceed with the same procedure mentioned above to measure V_r value from the CRO and also dc shift from CRO
03. Calculate Vr_{rms} & Vdc by using the formulas

    Vr, rms=Vdc/4√3fCR_L

    $Vdc=2V_M/\pi$

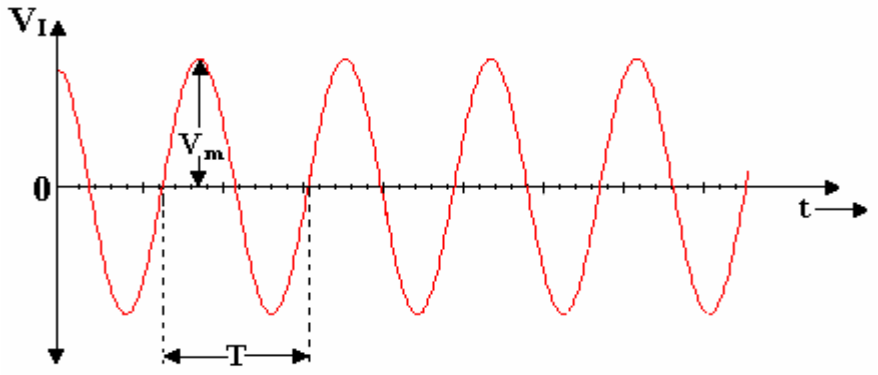
    Where V_r is the peak to peak amplitude of filter output

04. Calculate ripple factor and % regulation by using the formulae.

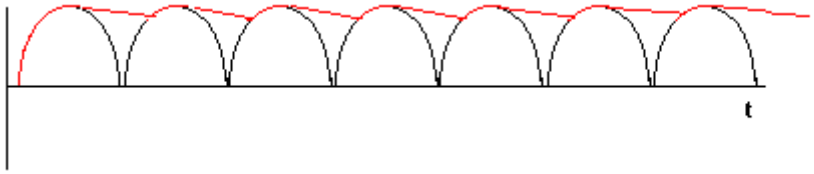
Ripple factor=Vrms/Vdc=$1/4\sqrt{3}fCR_L$

%Regulation = $(V_{NL}-V_{FL})/V_{FL}*_{100}$

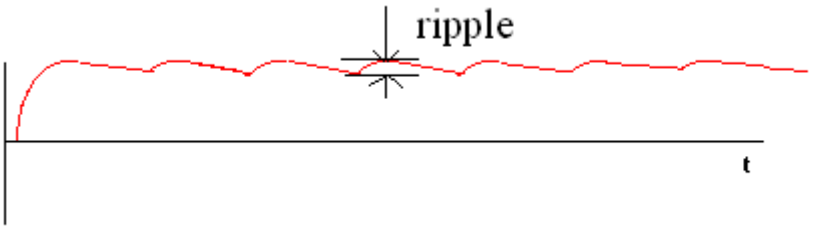
EXPECTED WAVE FORMS:



RECTIFIED OUTPUT:



FILTEROUTPUT



Precautions:

1. Wires should be checked for good continuity.

2. Carefully note down the readings with out any errors.

Result:

2. BRIDGE RECTIFIER

Aim:

1. To observe the output waveform of bridge rectifier with and without filter
2. To find ripple factor and percentage of regulation of bridge rectifier with and without filter

Apparatus:

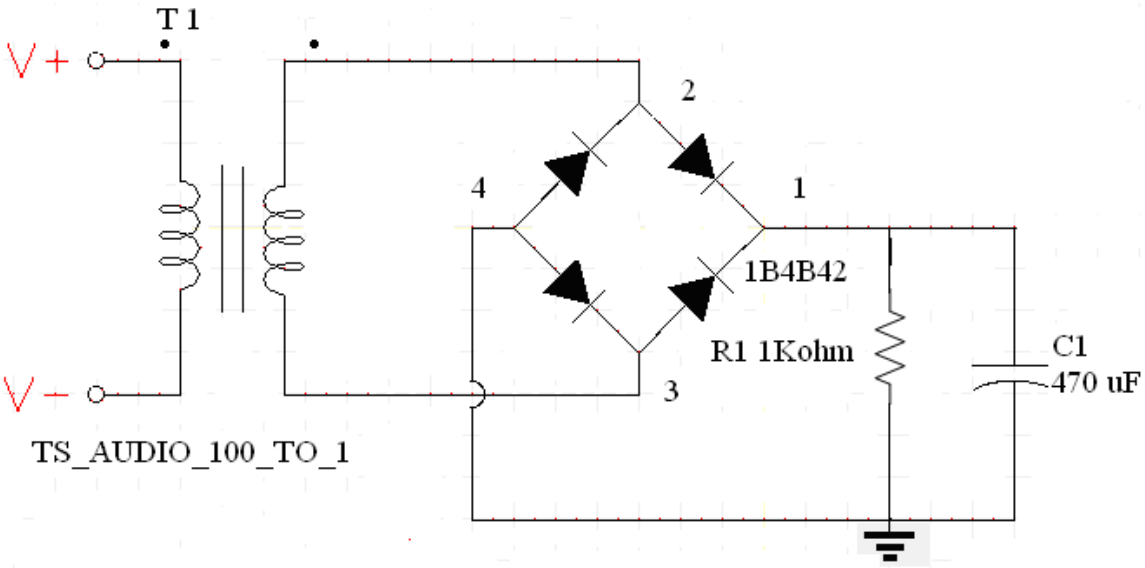
Silicon Diodes BY126

Resistance 1 KΩ

Capacitor 470 µF

CRO Multimeter

Circuit Diagram:



Theory:

The bridge is seen to consist of four diodes connected with their arrowhead symbols all pointing toward the positive output terminal of the circuit.

During the positive half cycle of input voltage. Thus load current flows from the positive input terminal through D1 to R_L and then through RL and D4 back to the negative input terminal. During this time, the positive input terminal is applied to the cathode of D2 so it is reversed biased and similarly D3 is also reverse biased. These two diodes are forward biased during negative half cycle; D1 & D4 are reverse biased during this cycle. And finally both half cycles are rectified.

Procedure:

Without filter:

01. Connect the circuit as per the circuit diagram.
02. Connect CRO across the load
03. Note down the peak value VM of the signal observed on the CRO
04. Switch the CRO into DC mode and observe the waveform. Note down the DC shift.
05. Calculate V_{mis} and V_{dc} values by using the formulae

    Vrms=Vm/√2, Irms=Im/√2,

    $Vdc=2V_M/\pi$, $Idc=2I_M/\pi$

06. Calculate the ripple factor by using the formulae

    $R=\sqrt{(Vrms/Vdc)^2}-1$

07. Remove the load and measure the voltage across the circuit take down the value as V_{NL}. Calculate the percentage of voltage regulation using the formula

    $Regulation = (V_{NL}-V_{FL})/V_{FL}*_{100}$

With Filter:

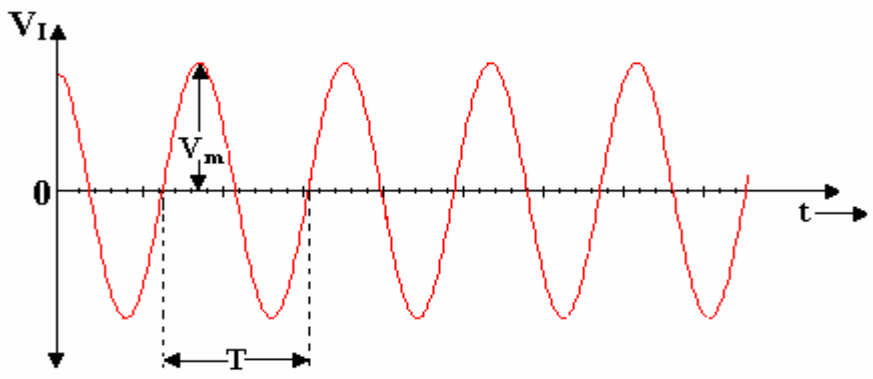
01. Connect the capacitor filter across the load in the above circuit diagram.
02. Procedure mentioned above to measure Vr value and also dc shift from CRO
03. Calculate V_{rms} by using the formula

    Where V_r is the peak to peak amplitude of filter output

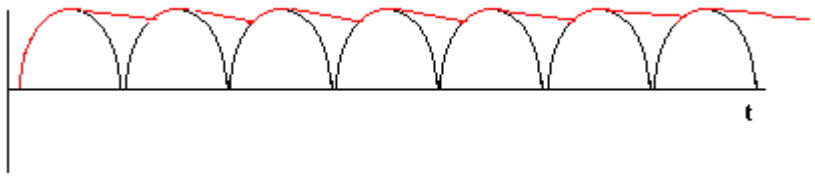
04. Calculate ripple factor and % regulation by using the formulae

Regulation = $(V_{NL}-V_{FL})/V_{FL}*_{100}$

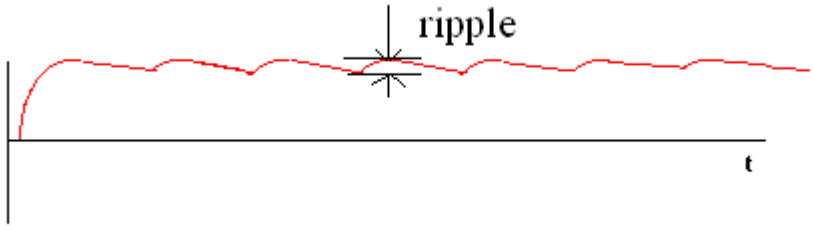
EXPECTED WAVE FORMS:



RECTIFIED OUTPUT:



FILTER OUTPUT



Precautions:

01. Wires should be checked for good continuity.

02. Carefully note down the readings with out any errors.

Result:

3. COMMON EMITTER AMPLIFIER

AIM: To find the voltage gain of a CE amplifier and to find its frequency response

APPARATUS:

Transistor BC107

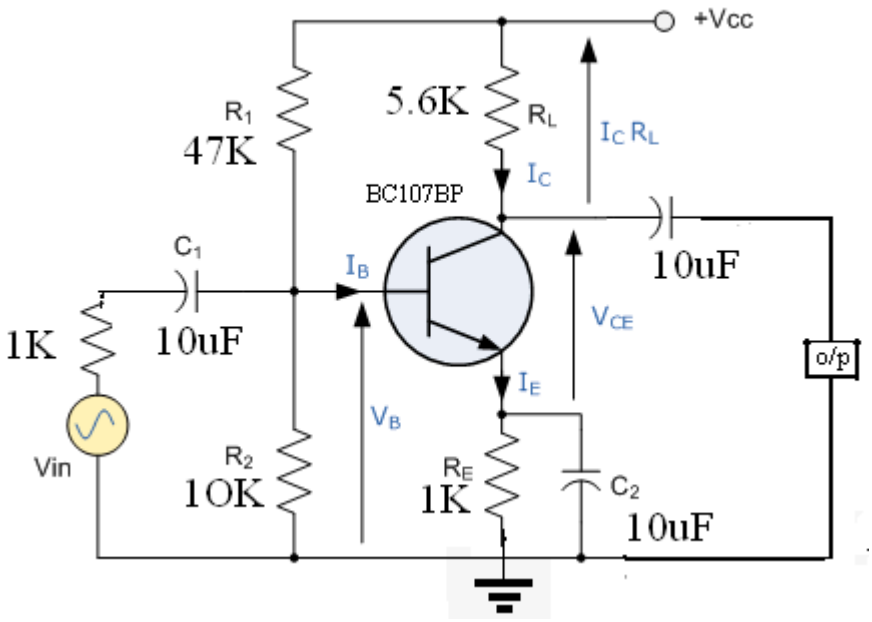
Resistors

Capacitors

CRO

Signal generator

CIRCUIT DIAGRAM:



THEORY:

The CE amplifier is a small signal amplifier. This small signal amplifier accepts low voltage ac inputs and produces amplified outputs. A single stage BJT circuit may be employed as a small signal amplifier; has two cascaded stages give much more amplification.

Designing for a particular voltage gain requires the use of a ac negative feedback to stabilize the gain. For good bias stability, the emitter resistor voltage drop should be much larger than the base -emitter voltage. And $_{Re}$ resistor will provide the required negative feedback to the circuit. C_E is provided to provide necessary gain to the circuit. All bypass capacitors should be selected to have the smallest possible capacitance value, both to minimize the physical size of the circuit for economy. The coupling capacitors should have a negligible effect on the frequency response of the circuit.

PROCEDURE:

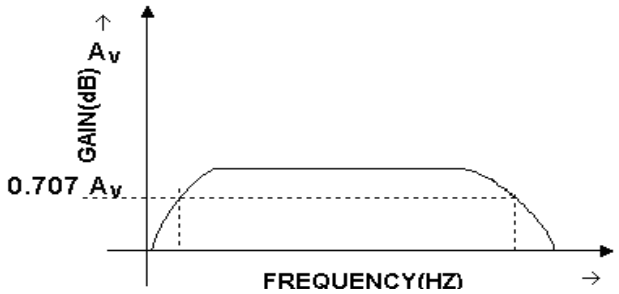
1. Connect the circuit as per the circuit diagram.
2. Give l00Hz signal and 20mv p-p as Vs from the signal generator
3. Observe the output on CRO and note down the output voltage.
4. Keeping input voltage constant and by varying the frequency in steps 100Hz-1MHz, note down the corresponding output voltages.
5. Calculate gain in dB and plot the frequency response on semi log sheet.

TABULAR FORM

Input voltage (Vi)=

.NO	FREQUENCY	OUTPUT VOLTAGE(Vo)	GAIN Av=Vo/Vi	GAIN IN dB 20 log gain

MODEL GRAPH:



Precautions:01. Wires should be checked for good continuity.

02. Transistor terminals must be identified and connected carefully.

Result:

4. COMMON SOURCE AMPLIFIER

AIM:

To find the voltage gain of a CS amplifier and to find its frequency response

APPARATUS:

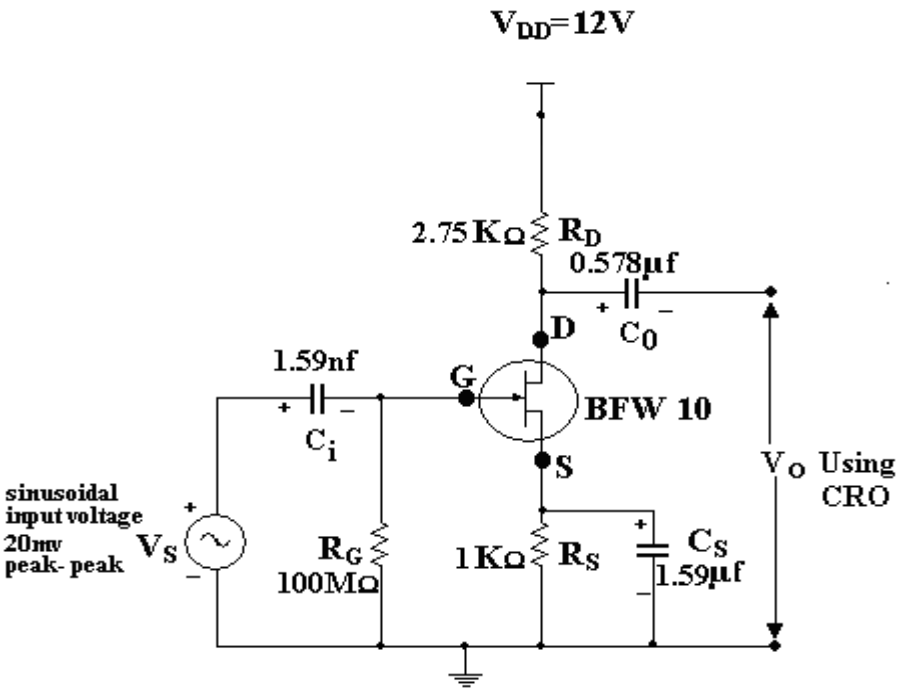
FETBFW10

RESISTORS

Capacitors

Signal generator &CRO

CIRCUIT DIAGRAM:



THEORY:

The CS amplifier is a small signal amplifier. For good bias stability, the source resistor voltage drop should be as large as possible. Where the supply voltage is small, Vs may be reduced to a minimum to allow for the minimum level of Vds.R2 is usually selected as 1MΩ or less as for BJT capacitor coupled circuit,

coupling and bypass capacitors should be selected to have the smallest possible capacitance values. The largest capacitor in the circuit sets the circuit low 3dB frequency (capacitor C2). Generally to have high input impedance FET is used. As in BJT circuit RL is usually much larger than Zo and Zi is often much larger than Rs

PROCEDURE:

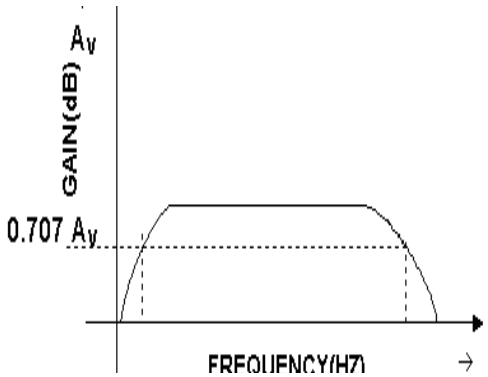
1. Connect the circuit as per the circuit diagram
2. Give 1 KHz signal and 25 mv (P-P) as Vs from signal generator.
3 Observe the output on CRO for proper working of the amplifier.
4 After ensuring the amplifier function, vary signal frequency from 50 Hz to 600 Hz in proper steps for 15-20 readings keeping Vs =25mv(P-P) at every frequency ,note down the resulting output voltage and tabulate in a table
5. Calculate gain in dB and plot on semi log graph paper for frequency Vs gain in dB

TABULAR FORM:

Input voltage (Vi)=

S.NO	FREQUENCY	OUTPUT VOLTAGE(Vo)	GAIN Av=Vo/Vi	GAIN IN dB 20 log gain

MODEL GRAPH:



PRECAUTIONS: 1. Wires should be checked for good continuity

2 FET terminals must be identified and connected carefully.

RESULT:

5. MEASUREMENT OF PARAMETERS OF EMITTER FOLLOWER AND SOURCE FOLLOWER

Aim: To calculate the Voltage gain, Current gain, input resistance and output resistance of Emitter follower& source follower.

Apparatus:

FETBFW10

Transistor BC107

Resistors

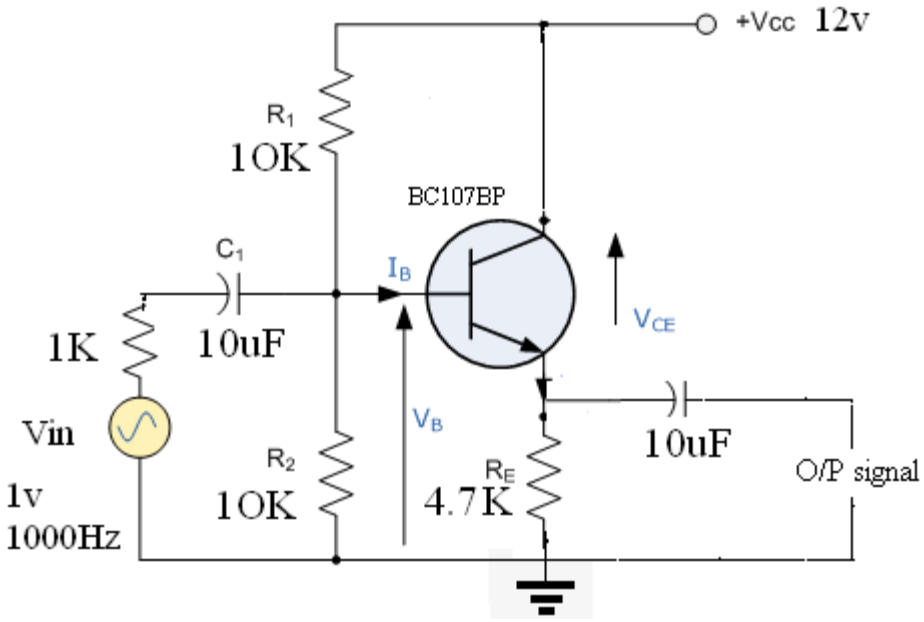
Capacitors

CRO

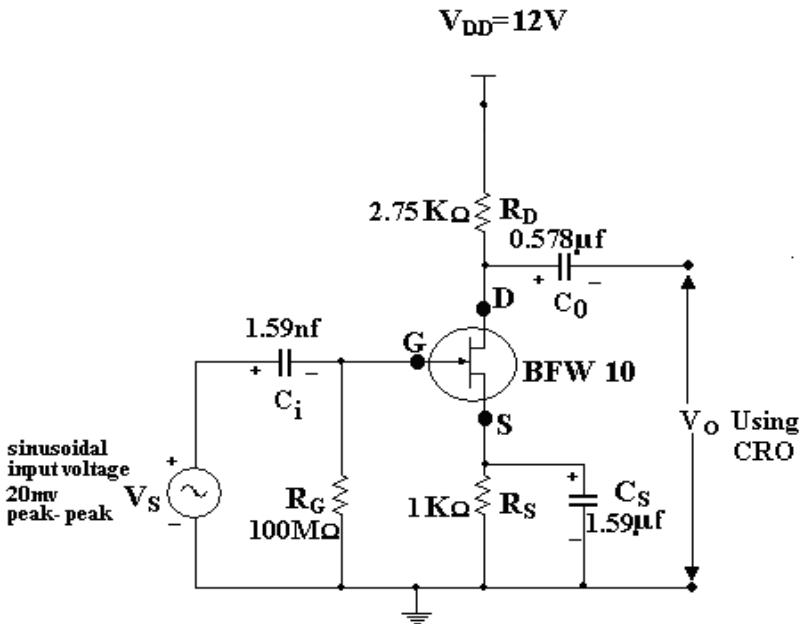
Function Generator.

Multi meter

CIRCUIT DIAGRAM F0R EMITTER FOLLOWER



CIRCUIT DIAGRAM F0R SOURCE FOLLOWER



THEORY: EMITTER FOLLOWER

The common collector circuit is also known as emitter follower. The ac output voltage from a CC circuit is essentially the same as the input voltage; there is no voltage gain or phase shift. Thus, the CC circuit can be said to have a voltage gain of 1. The fact that the CC output voltage follows the changes in signal voltage gives the circuit its other name emitter follower. The input impedance of a CC amplifier is high. Output impedance is low and the Voltage gain is almost unity. Because of these Characteristics the CC circuit is normally used as a buffer amplifier, placed between a high impedance signal source and a low impedance load.

SOURCE FOLLOWER

The FET common drain circuit has the output voltage developed across the source resistor Rs. Here the ac output voltage is closely equal to the ac input voltage, and the circuit can be said to have unity gain. Because the output voltage at the source terminal follows the signal voltage at the gate, the common drain circuit is also known as a source follower.

A common drain circuit has a voltage gain approximately equal to 1, no phase shift between input and output, very high input impedance and low output impedance. Because of its high Zi, low Zo and unity gain the CD circuit is used as a buffer amplifier between a high impedance signal source and a low impedance load.

PROCEDURE:

1. Connect the circuit as per the circuit diagram.
2. Apply Vslv 1 KHz signal from the signal generator.
3. Observe corresponding output from the CRO and then calculate voltage gain using the formula Av=Vo/Vi.
4. Measure voltage across AB terminals and then calculate input current by using the formula $Iin=V_{ab}/R_{ab}$.
5. Measure current flowing through resistor at Source (or Emitter) terminal and note down it as Iout.
6. Calculate Current gain using the formula $AI=Iin/I_{out}$.
7. Calculate input resistance using the formula Rin=Vin/Iin.
8. To calculate the output resistance, connect the pot at the output and vary the resistance of the pot up to half of the output with RL is equal to infinity. The resistance of pot is the output resistance.

PRECAUTIONS:

1. Wires should be checked for good continuity
2. FET terminals must be identified and connected carefully.

RESULT:

6. CASCODE AMPLIFIER

AIM:

To measure voltage gain, input resistance and output resistance of cascode amplifier

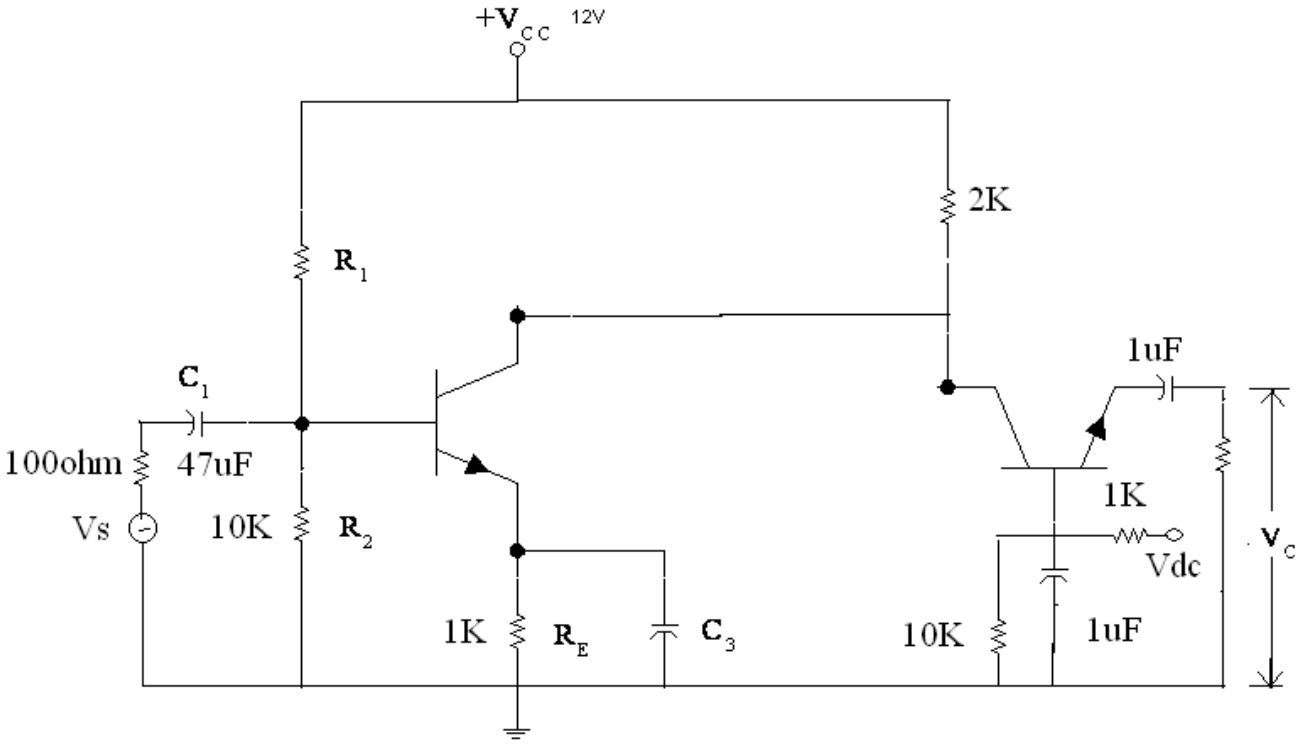
APPARATUS:

Transistor BC107 Resistors

Signal generator

Capacitors

CIRCUIT DIAGRAM:



DESIGN:

$I_{B1}=V_{CE}-V_{BE}/R_{B1}$

$I_{C1}=I_{E2}=I_{C2}=\beta I_{B1}$

$V_{C1}=V_{E2}=V_{B2}-V_{BE}$

$V_{C2}=V_{CC}-I_{C2}*R_{C2}$

$V_{CE2}=V_{C2}-V_{E2}$

$Rin=R_{B1}\,||\,\beta_1 R_{E1}$

$A_{v1}=-R_{L1}/R_{E1}=-1$

$Ro=R_{C2}$

$R_{L2}=R_{C2}\,||\,R_L$

$A_{V2}=R_{L2}/R_{E2}$

$Av=A_{V1}*A_{V2}$

THEORY: Cascode amplifier is a cascade connection of a common emitter and common base amplifiers. It is used for amplifying the input signals. The common application of cascade amplifier is for impedance matching. The low impedance of CE age is matched with the medium of the CB sage.

PRECAUTIONS:

1. Wires should be checked for good continuity
2. Take the readings carefully

RESULT:

7. TWO STAGE RCCOUPLED AMPLIFIERS

AIM:

To obtain the frequency response of' a two stage RC coupled amplifier

Apparatus:

Transistors

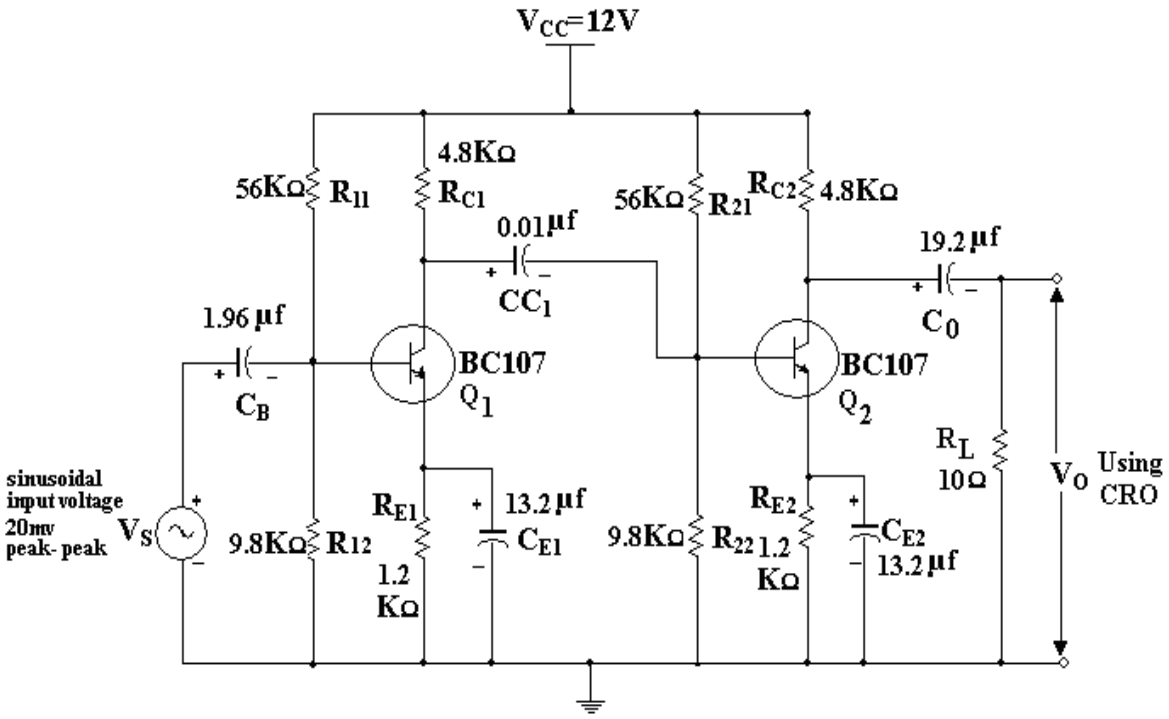
BC 107

Resistors

Capacitors

Signal Generators & CRO

Circuit Diagram:



Theory:

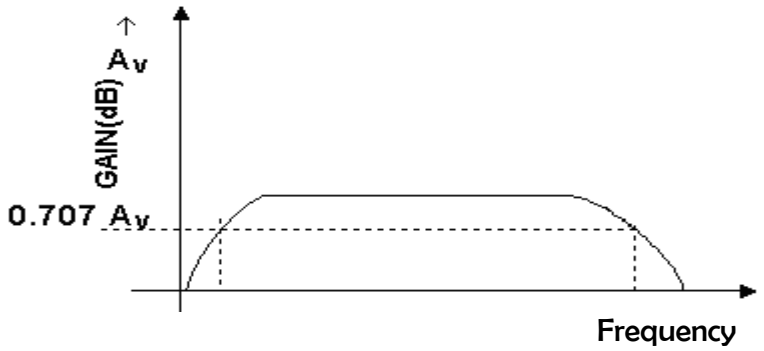
A RC coupled two stage amplifier is shown in the figure each stage is similar to the single stage circuit. Stage 1 is capacitor coupled to the input of stage 2. The signal is applied to the input of stage 1 and the load is coupled to the output of stage 2. The signal *κ* amplified by stage 1 and the output of stage 1 is amplified by stage 2. So that the overall voltage gain is much greater than the gain of a single stage. The signal voltage is phase shifted through 180° by stage 1 and through a further 180° by stage 2. Consequently the overall phase shift from input to output is zero or 360"

PROCEDURE

01. Connect the circuit as per the circuit diagram
02. Give 1 KHz signal, 25 mV (p-p) as V_s from signal generator
03. Observe the output on CRO for proper working of the amplifier
04. After ensuring the amplifier function, vary signal frequency from 50 Hz to 600 Hz in proper steps for 15 to 20 readings. Keeping V_s = 25 mV (p-p) at every frequency, note down the resetting output voltage and tabulate in a table.
05. Calculate gain db and plot on semi log graph paper for frequency VS gain db.

Expected waveforms:

Frequency Response



Tabular Form:

S.NO	FREQUENCY (HZ)	OUTPUT VOLTAGE(Vo)	GAIN Av=Vo/Vi	GAIN IN dB 20 log gain

PRECAUTIONS:

1. Wires should be checked for good continuity
2. FET terminals must be identified and connected carefully.

RESULT:

8. CONSTANT K LOW PASS & HIGH PASS FILTER

AIM: To design and verify Constant-K high pass& low pass niters and draw the frequency response.

APPARATUS:

Signal generator

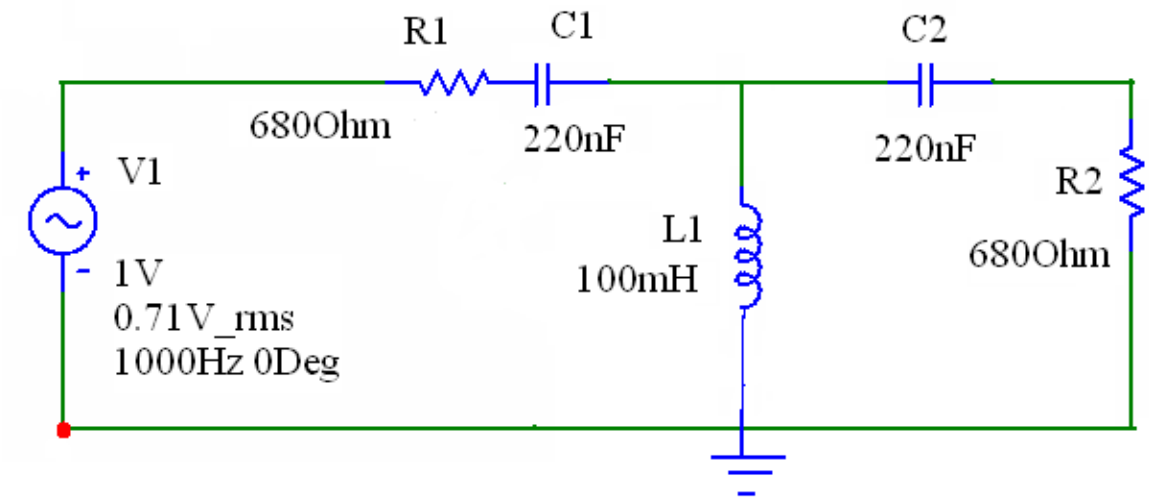
CRO

Capacitors

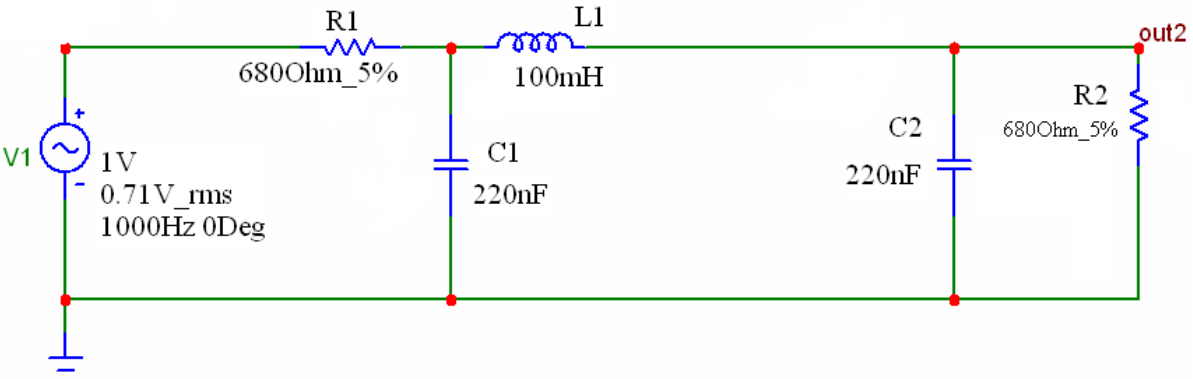
Inductors & Resistors

CIRCUIT DIAGRAMS:

CONSTANT-K HPF



CONSTANT-K LPF



DESIGN EQUATIONS:

ForLPF:

Given R_L-680Ω, Fc=2 KHz, Rs=680 Ω. Rl=Rs=Rk

C=1/ (πfcRk), R=Rk/fc

For HPF:

Given R_L=Rs=Rk

Given R_L-680Ω, Fc=2 KHz, Rs=680 Ω. Rl=Rs=Rk

C=1/ (πfcRk), L=Rk/4πfc

THEORY:

The low pass filter is a filter that transmits all frequencies from zero unto some designated frequency called the cut-oft frequency and offer great attenuation at all other higher frequencies. A Constant-K filter is a T or TT network in which the series and shunt impedances, Zl and Z2 are connected *by* the relationship Z.1Z2=Rk2. where Rk is a real constant or it is termed as design impedance or nominal impedance of Constant-K filter. A HPF is a filter that transmits all frequencies above a designated cut-off frequency but attenuates frequencies below this.

PROCEDURE:

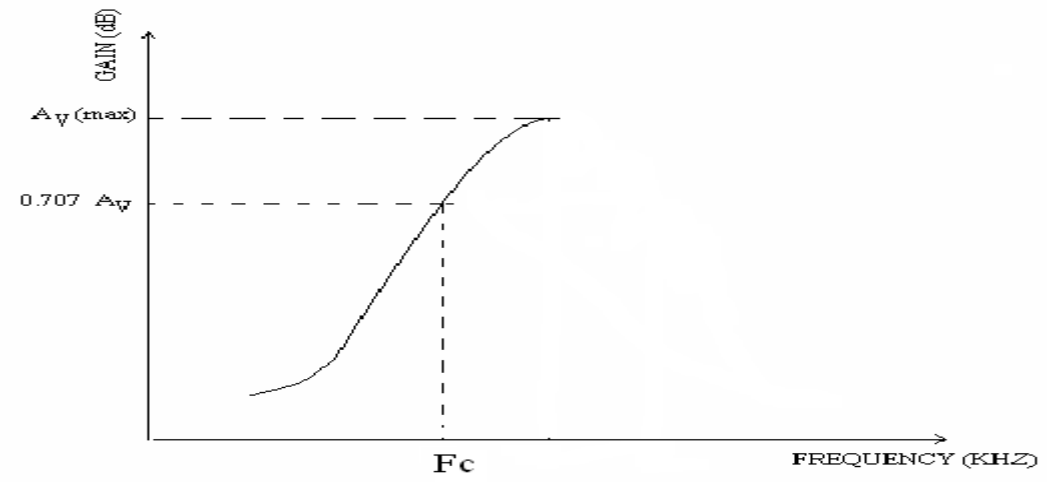
1. Design L and C values with the help of formulae and connect them in the circuit.
2. Set the input voltage Vi=5v using signal generator and vary the frequency from1Hz-lMHz in regular intervals.
3. Note down the corresponding output voltage.
4. Calculate gain in dB.
5. Plot the frequency response of HPF & LPF.

TABULAR COLUMN

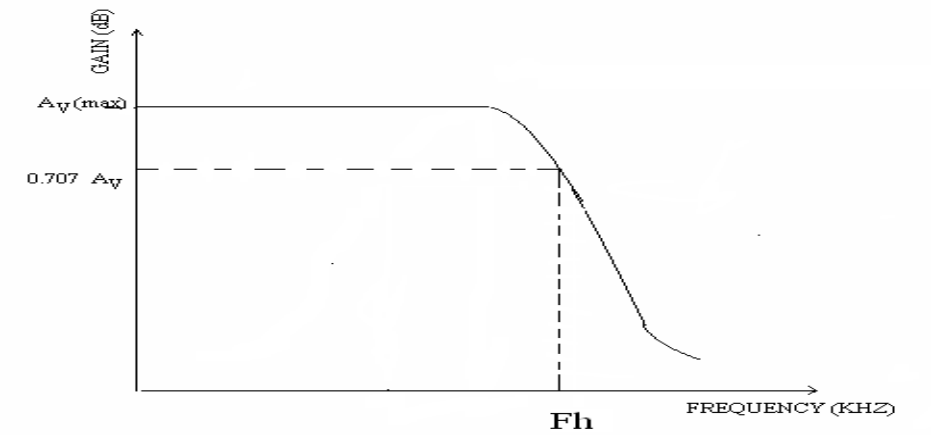
S.NO	FREQUENCY (HZ)	OUTPUT VOLTAGE(Vo)	GAIN Av=Vo/Vi	GAIN IN dB 20 log gain

MODEL GRAPHS:

CONSTANT-KHPF



CONSTANT-KLPF



PRECAUTIONS:

1. Wires should be checked for good continuity
2. Vary the frequency carefully.

RESULT:

9. CONSTANT K BAND PASS AND BAND ELIMINATION

FILTERS

AIM:

To design constant-k band pass and band elimination filters and measures their cut off frequencies.

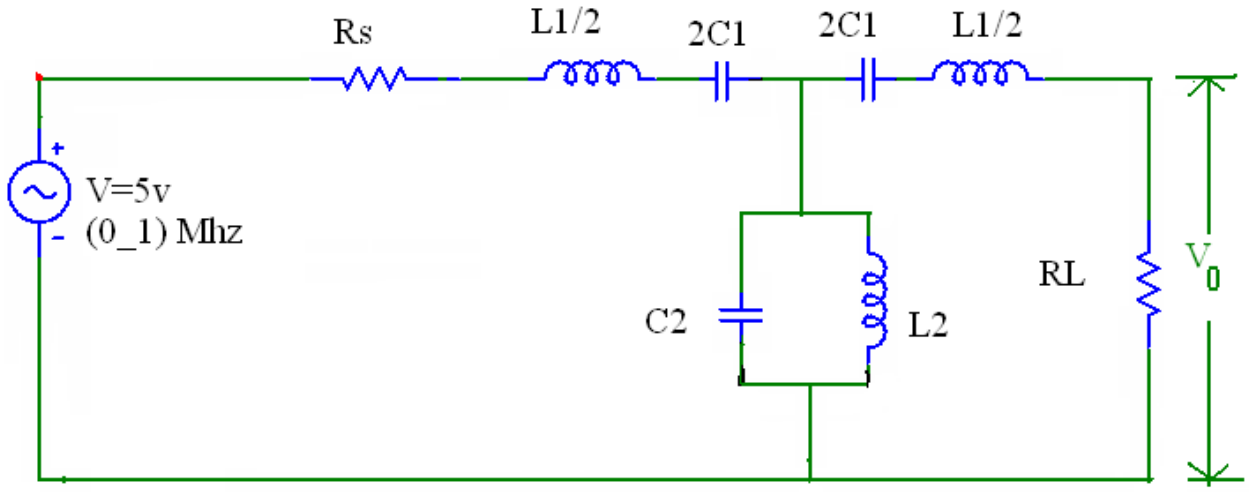
APPARATUS:

Signal generator (0-1) MHz

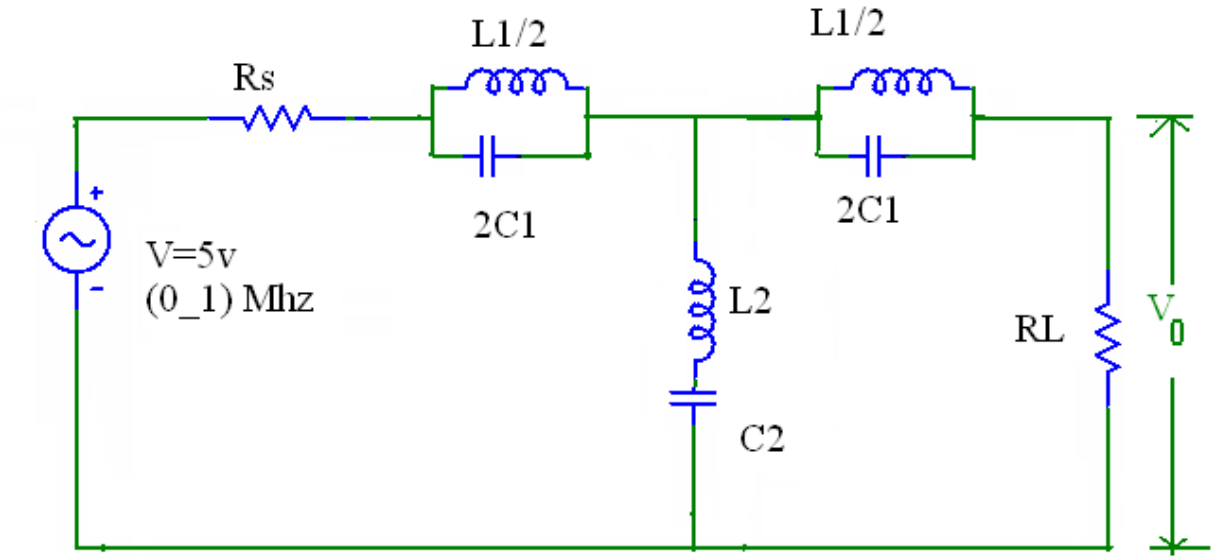
CRO

CIRCUIT DIAGRAM:

Band pass filter:



BAND ELIMINATION FILTER:



DESIGN:

For band pass:

To evaluate the values for the series arm,

$C_1 = (f_2-f_1)/4\pi f_1 f_2 R_k$

$L_1 = R_k/\pi f_1 f_2$

To evaluate the values for the shunt arm

$L_2 = R_k(f_2-f_1)/4\pi f_1 f_2$

$C_2 = 1/\pi R_k(f_2-f_1)$

For band elimination:

To evaluate the values for the series arm consider the equations,

$C2 = (f_2-f_1)/\pi f_1 f_2 R_k$

$L_2 = R_k(f_2-f_1)/4\pi f_1 f_2$

To evaluate the values for the shunt arm, consider

$L_1 = C_2 R_k * R_k = R_k(f_2-f_1)/\pi f_1 f_2$

$C_1 = L_2/ R_k * R_k = 1/4\pi R_k(f_2-f_1)$

THEORY:

Band passes and band elimination filters are designed to allow a particular band of frequencies or to eliminate a particular band of frequencies. A band pass filter is obtained by using a low pass filter followed by a high pass filter in which the cut off frequency of the LP Filter is above the cut off frequency of the HP hence allowing only a band of frequencies. A band stop filter can be realized by connecting a low pass filter in parallel with a high pass filter. The cutoff frequency of low pass is below the high pass.

PROCEDURE:

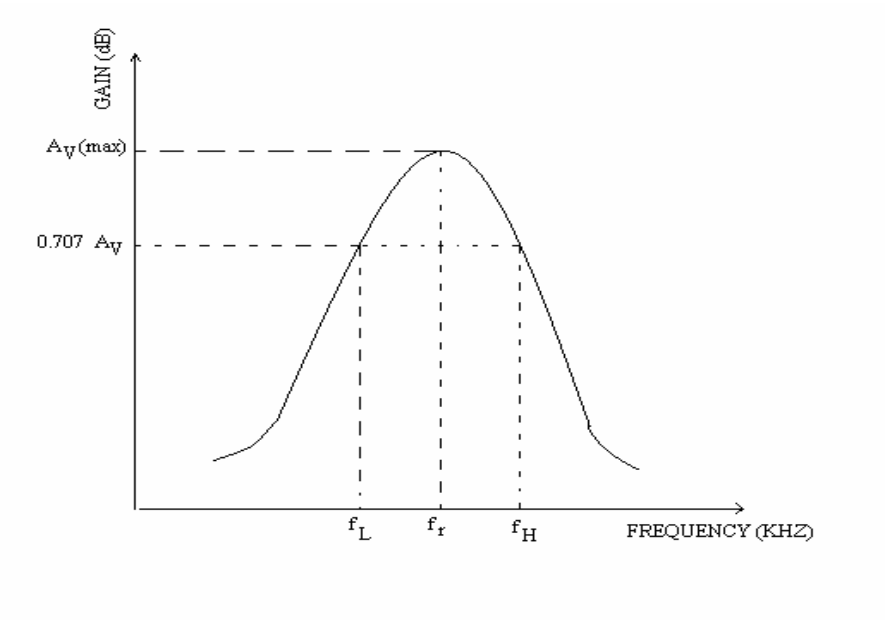
1. Connect the circuit as per the circuit diagram
2. Set i/p voltage Vi=5v using signal generator and vary the frequency from (0-l) MHz in regular intervals
3. Note down the corresponding output voltage
4. Plot the graph between gain Vs frequencies

TABULAR COLOUMN:

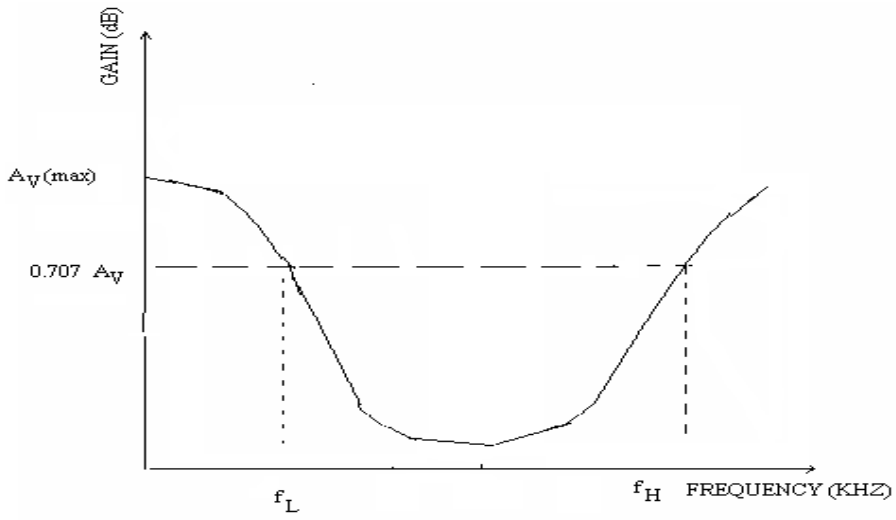
S.No	frequency(Hz)	Output Voltage (Volts)	Gain (Vo/Vin)	Gain in dB

MODEL GRAPH:

BAND PASS FILTER:



BAND ELIMINATION FILTER



PRECAUTIONS:

1. Wires should be checked for good continuity
2. Vary the frequency carefully.

RESULT:

10. M-DERIVED LOW-PASS AND HIGH-PASS FILTERS

AIM:

To design m derived high pass and low pass filters and lo measure its frequency.

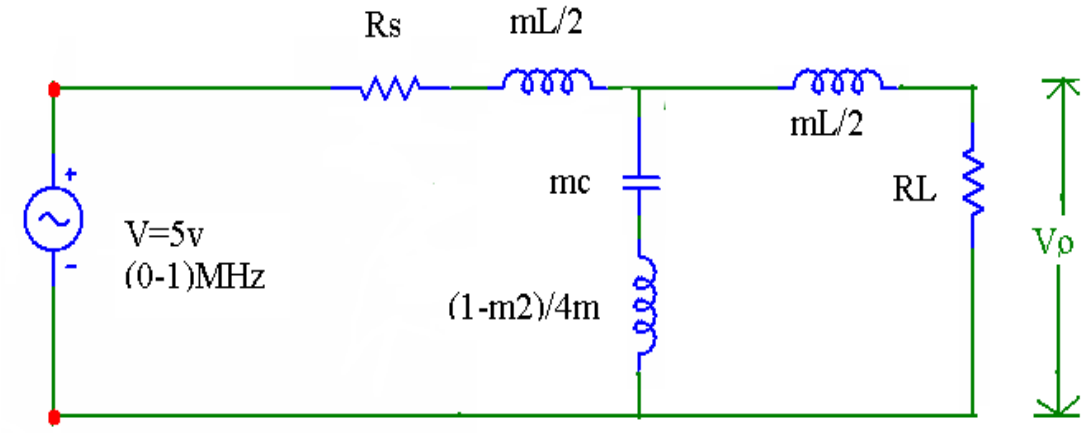
APPARATUS:

Signal generator

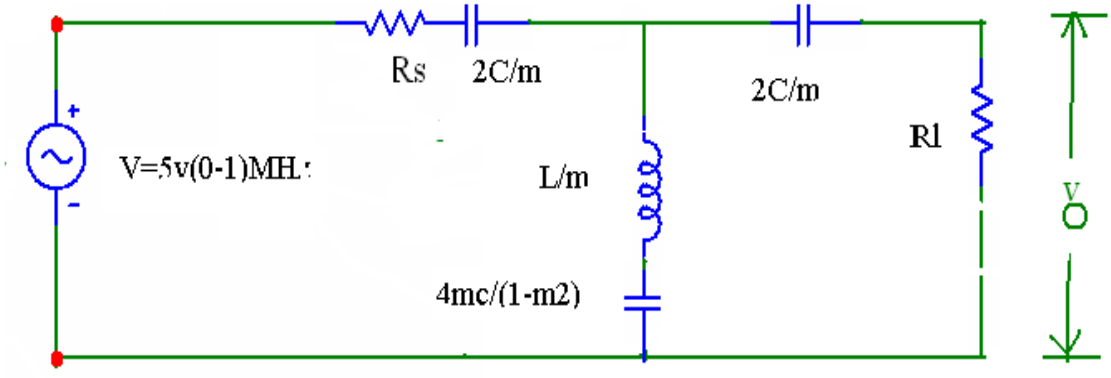
CRO

CIRCUIT DIAGRAM:

M-DERIVED Low-PASS FILTER



M-DERIVED HIGH-PASS FILTER



THEORY: For a constant k prototype filter section has two major disadvantages. The attenuation does not rise very rapidly at cutoff frequency .so that a satisfactory impedance match is not possible. So m derived filters used to raise the attenuation near cutoff. To have a sharp cut off frequency, m derived filter is used. A HPF is one which attenuates the lower frequencies and passes the higher frequencies above the cut off frequency. A LPF is one allows the frequencies below the cut off frequency and attenuates the frequencies above cut off frequency.

DESIGN:

FOR HPF

$0<m<1(m=0.5)$ f=2 kHz, C=0.01 $fr=1/4\pi\sqrt{LC}$ L=?

FOR LPF:

$0<m<1(m=0.5)$ f=2 kHz, C=0.01 uF

$fr=1/\pi\sqrt{LC}\ (1-m2)$ L=?

PROCEDURE:

1. Connect the circuit as per the circuit diagram
2. Set i/p voltage Vi=5v using signal generator and vary the frequency from (0-l) MHz in regular intervals
3. Note down the corresponding output voltage
4. Plot the graph between gain vs frequencies

TABULAR COLOUMN

M-DERIVED HIGH PASS FILTER

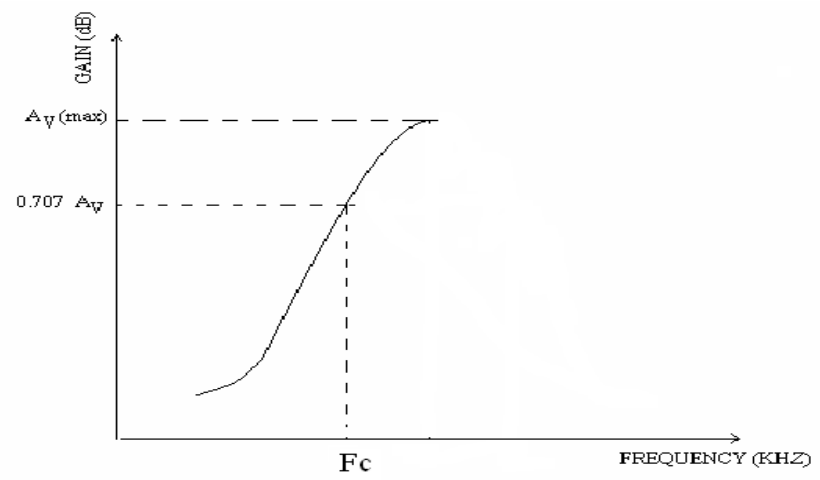
S.No	frequency(Hz)	Output Voltage (Volts)	Gain (Vo/Vin)	Gain in dB

TABULAR COLOUMN

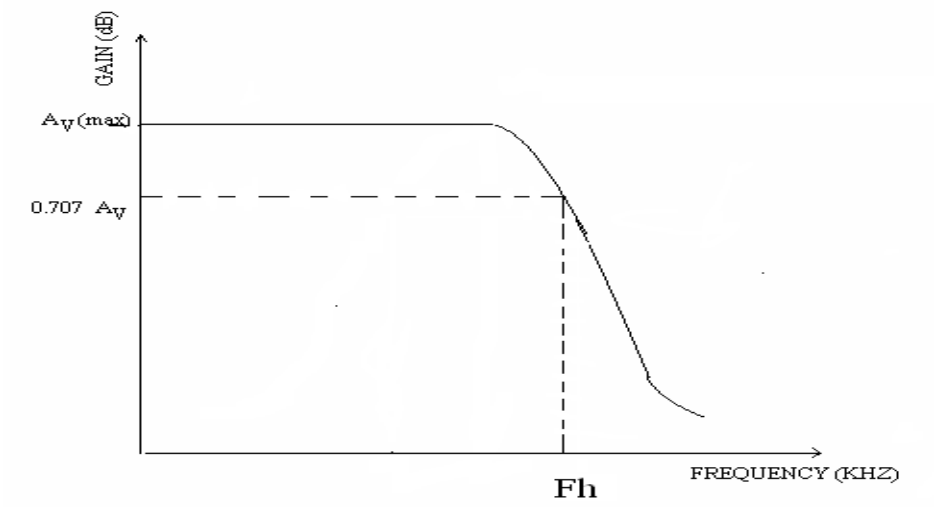
M-DERIVED LOW PASS FILTER

S.No	frequency(Hz)	Output Voltage (Volts)	Gain (Vo/Vin)	Gain in dB

MODEL GRAPH: m-Derived High Pas Filter:



m-Derived Low Pass Filter:



PRECAUTIONS:

1. Connections are made tight.
2. Vary the voltage properly.

RESULT:

11. ATTENUATORS

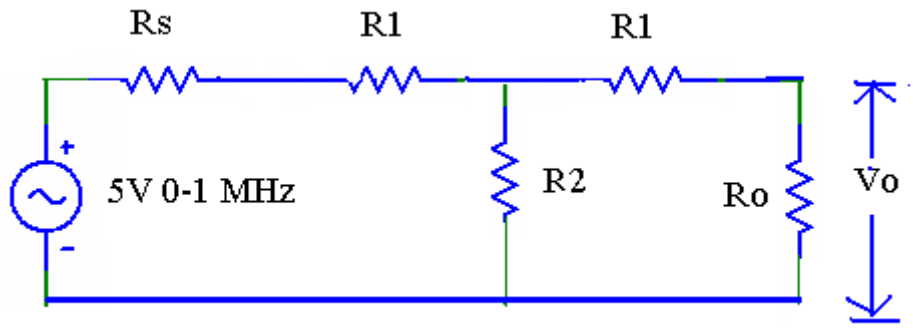
AIM: To design a T- attenuator and ∏-attenuator which attenuate given signal to the desired level

APPARATUS:

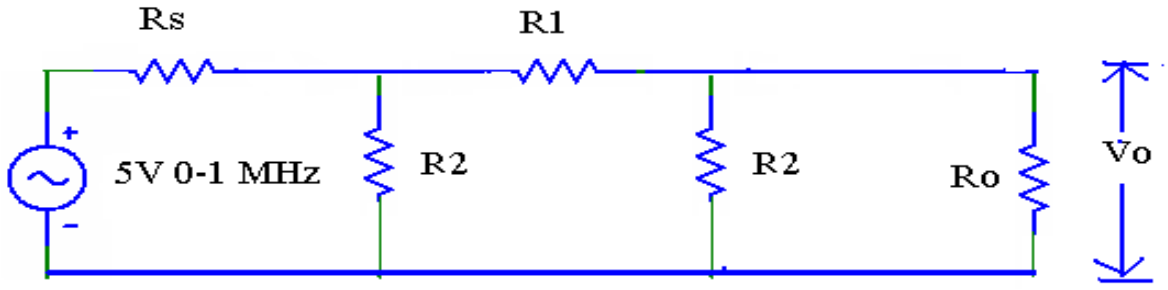
Signal generator (0-1) MHz

CRO (0-20) MHz

CIRCUIT DIAGRAM: T-ATTENUATOR



∏-ATTENUATOR



DESIGN: Given D=2dB, Rs=L680 Ω

Rl=Ro(N2-l/2N),R2=Ro(N+l/N-l)

N=Antilog(DdB/20)

PROCEDURE:

1. Connect the circuit as per the circuit diagram
2. Set input voltage, Vi=5v using signal generator and vary the frequency from (0--1) MHz in regular steps
3. Note down the corresponding output voltage
4. Plot the graph output voltage Vs frequencies

TABULAR COLOUMN

INPUT VOLTAGE= (Vi)

S.No	frequency(Hz)	Output Voltage (Volts)

MODEL GRAPH:

PRECAUTIONS:

1. Check for loose connections in the circuit.
2. Vary the frequency carefully.

RESULT:

12. MEASUREMENT OF IMPEDANCE ADMITTANCE AND TRANSMISSION PARAMETERS

AIM: To calculate and verify impedance parameter (Z), admittance parameters(Y) and transmission parameters (ABCD) of the given circuit.

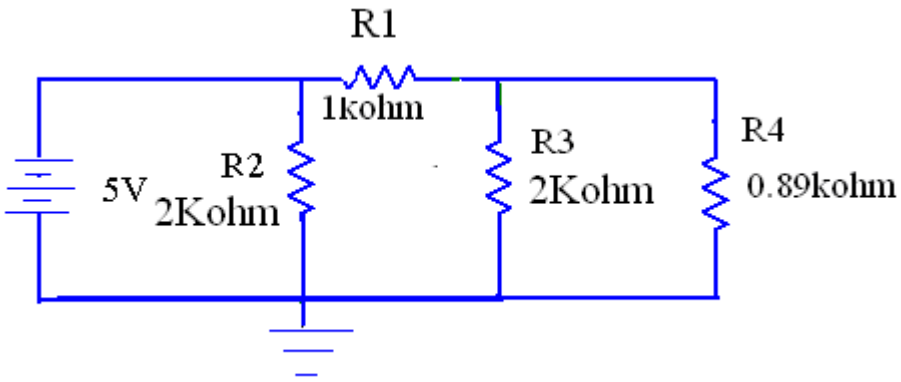
APPARATUS: Resistors

DC power supply

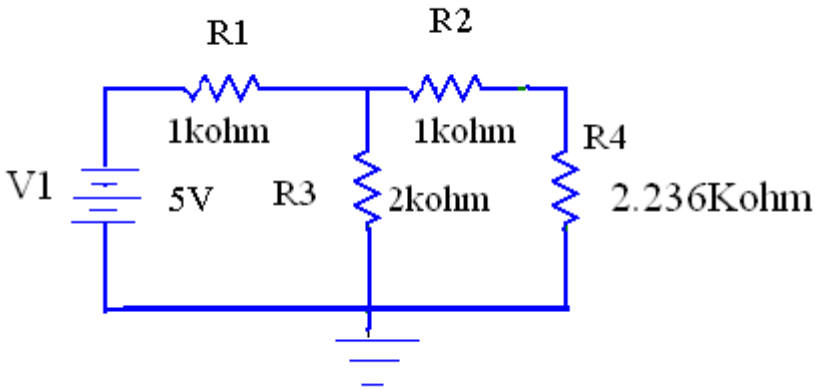
Multi meter.

CIRCUIT DIAGRAM:

Symmetrical π n/w



Asymmetrical T- n/w



THEORY:

A two-port network is described by a six possible sets of equations. So there are 6 parameters that could be described for a two-port network. They Are Z, Y, ABCD and h. Z-parameters are also known as open-circuit Parameters. In case of z-parameters, the two describing equations of a network are

$V1=Z_{11}I_1+Z_{12}I_2$-—(1)

$V2=Z_{12}I_1+Z_{22}I_2$ ——(2) where $Z_{l1}=V_1/I_1$ at$I_2=0$

$Z_{12}=V_1/I_2$at$I_l=0$; $Z21=V_2/I_l$at$I_2=0$;$Z_{22}=V_2/I_2$at$I_l=0$ Z_{11}, Z_{12}, $Z_{21}$and Z_{22} are the network functions and are called impedance parameters

The two describing equations to obtain y-parameters are$I_1=Y_{11}V1+Y_{12}V2$— (!)

$I2=Y_{21}V1+Y_{12}V2$— (2)

Where Y_{11}, Y_{12}, Y_{21} andY_{22} are known as short circuit parameters or admittance parameters are given as

$Y_{11}=I1/V1$ atV2=0;

$Y_{12}=I1/V2$ al V1=0;

$Y_{21}=I2/V1$ at V2=0 and $Y_{22}=I2/V2$atVl=0.

The two describing equations to obtain transmission parameters are

V1=AV2-BI2

I1=CV2-DI2

Where A=V1 /V2 at 12=0;

-B=V 1/12 atV2=0;

C=I1 /V2 at I2=0 and

-D=Il/I2at V2=0 A. *B,* C, D are known as transmission parameters.

PROCEDURE:

1. Connect the circuit as per the circuit diagram.
2. Calculate Z_{11}, Z_{12}, $Z_{21}$and Z_{22} & Y_{11}, Y_{12}, Y_{21} andY_{22} & A, B, C. D parameters theoretically.
3. Measure VI, V2, I1, I2 practically and finally obtain Z, Y and ABCD parameters practically.
4. Compare practical values with theoretical calculation.

TABULAR COLUMN

	Theoretical	Practical
Z parameters		
Y parameters		
ABCD Parameters		

PRECAUTIONS:

1. Check for loose connections in the circuit.

RESULT:

13. MEASUREMENT OF IMAGE AND ITERATIVE IMPEDANCE OF SYMMETRICAL AND ASYMMETRICAL NETWORKS

AIM:

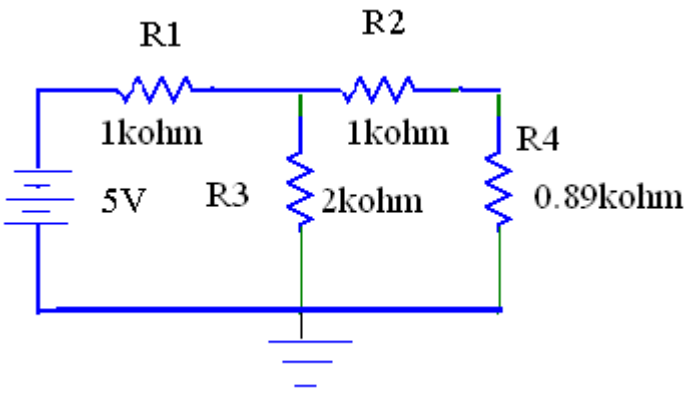
To measure the image and iterative impedances of symmetrical and asymmetrical networks

APPARATUS:

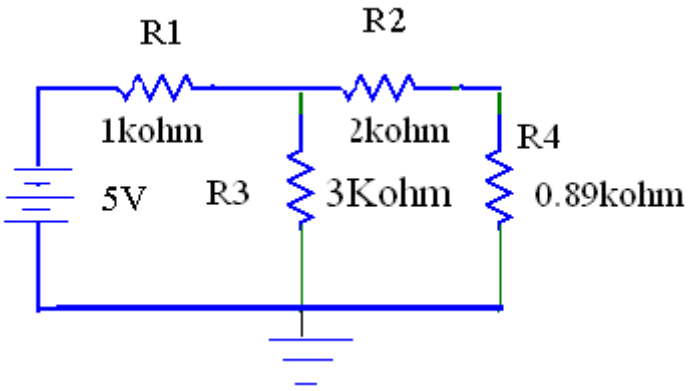
Resistors

CIRCUIT DIAGRAM:

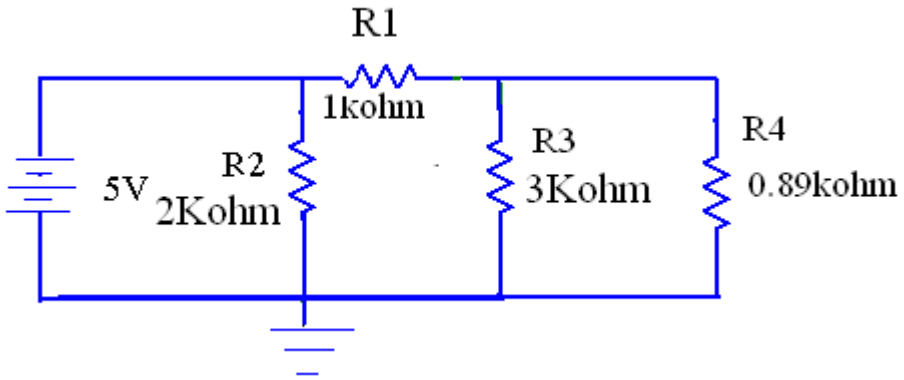
Symmetrical T-n/w:



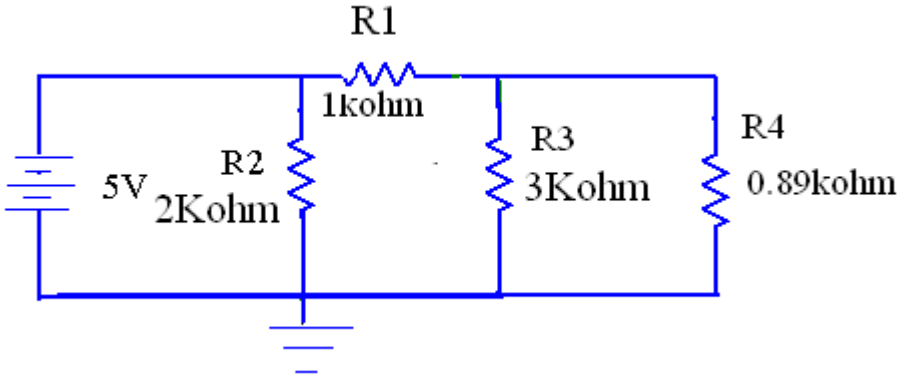
Asymmetrical T n/w



Symmetrical π n/w



Asymmetrical π n/w



THEORY:

Let Zil and Zi2 are two impedances of a two port network. If the port 1-1' is terminated in Zrl then the input impedance of the port is Z12 and its port 2-2" is terminated in Zi2 then the input impedance at port 1-1' is Zil. Then Zil and Zi2 are called age impedances of the two port network. If the network is symmetrical the image impedance Zil and Zi2 are equal to each other the image impedance is then called iterative impedance. Or characteristic impedance ie, if the symmetrical network is terminated in Z L, its input impedance will also be Z, L its impedance transformation ratio is unity

Formulas:

(i) For symmetrical T-n/w:

$Zi = [Z_{211} (Z_l/2+Zi)] + Z_l/2$

$= Z_1Z_2 (1+Z_1/4Z_2) = Z_{SC} * Z_{OC}$

(ii) For symmetrical π-n/w:

(iii) For Asymmetrical T-n/w

PROCEDURE:

1. Connect the circuit as per the circuit diagram
2. Connect Zo at the output and verily Zi value to set the same
3. Repeat this process by calculating the open circuit impedance at the o/p and also by short ckt the impedance at the i/p.
4. Tabulate the theoretical and experimental values.

TABULAR FORM

	Image impedance	Iterative impedance
Symmetrical T-n/w		
Symmetrical π-n/w		
Asymmetrical T n/w		
Symmetrical π-n/w		

PRECAUTIONS:

1. Make the connections properly.

RESULT:

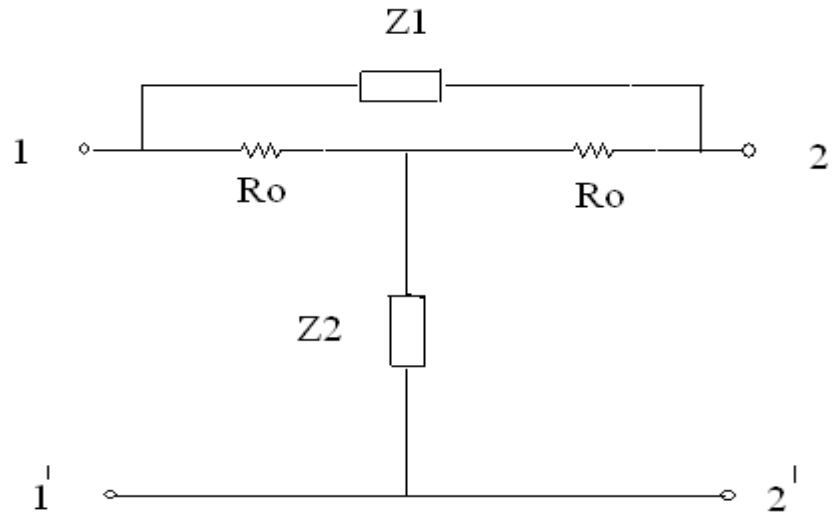
14. CONSTANT RESISTANCE EQUILIZERS

AIM: To observe the frequency response of constant resistance equalizers.

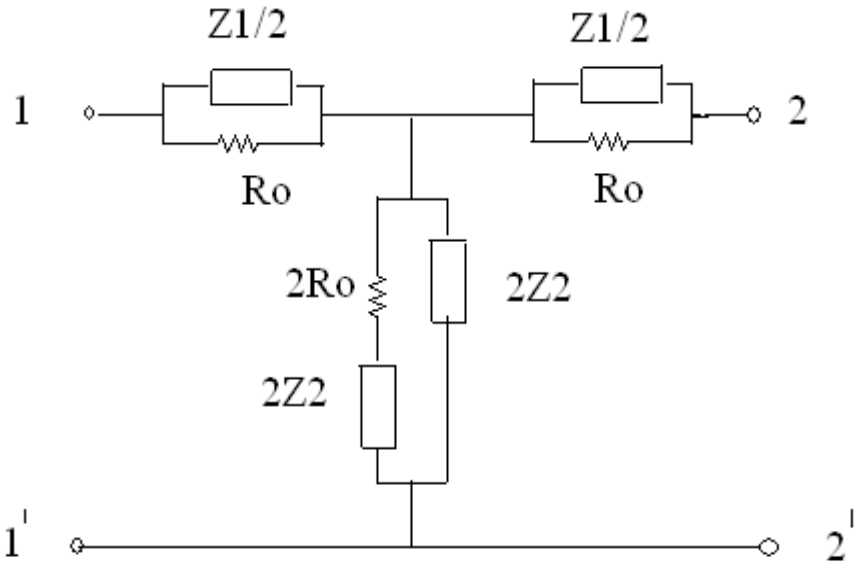
APPARATUS: CRO Resistors RPS

CIRCUIT DIAGRAM:

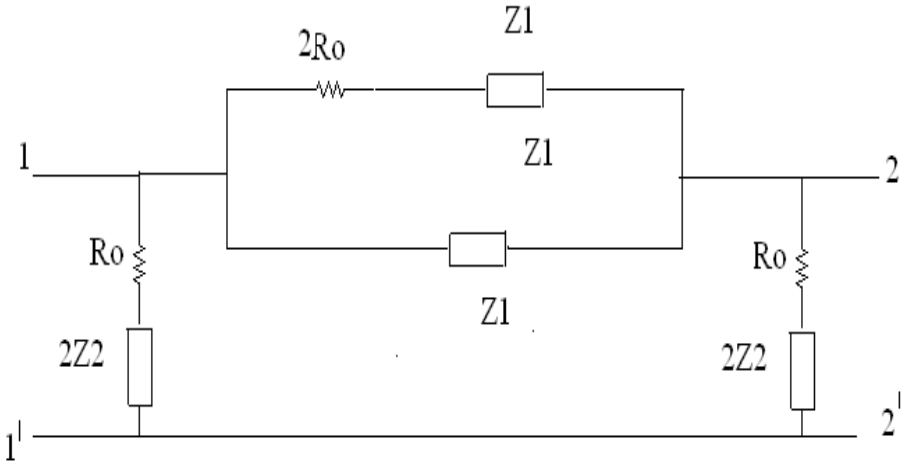
BRIDGE –T EQUALIZER



T-TYPE EQUALIZER



∏ -TYPE EQUALIZER



DESIGN EQUATIONS:

T- type Equalizer:

$Z_1Z_2=Ro^2$., M=16dB;

Ro, =600Ω, $Z_1=1/\omega C_1$, $Z2=\omega L_2$;

$L1=Ro\sqrt{m-1}/\omega$

$C1=\sqrt{m-1}/\omega Ro$

π type Equalizer:

$Z1Z_2=Ro^2$.,

$Z_{1=}\ \omega L1$

$Z_2 = 1/\ \omega c1$

$C1= Ro\sqrt{m-1}/\omega$

$L1=Ro\sqrt{m-1}/\omega$

BridgeT-Equalizer:

$R1R2=L1/C1=Ro^2$.

$L1=Ro\sqrt{m-1}/\omega$

$C3=L1/\ Ro^2$

PROCEDURE:

1. Connect the circuit as per the circuit diagram
2. Vary the frequency and observe the waveform on the CRO
3. Plot the attenuation Vs frequency graph.

TABULAR COLOUMN:

S NO	FREQUENCY	ATTENUATION

PRECAUTIONS:

1. **Check for loose connections**
2. **Vary the frequency carefully.**

RESULT: