COMMUNICATIONS CIRCUIT DESIGN LAB (CCD LAB REPORT)

NAME: VEDANT MILIND ATHAVALE

ID NO: 181090071

CLASS: TY BTECH EXTC

SUBJECT: COMMUNICATIONS CIRCUIT DESIGN LAB

FACULTY: MS. SHWETA GHODESWAR

DATE OF SUBMISSION: 20-11-2020

Experiment - 1

Aim: To design a Butterworth analog nth order low pass filter.

Software used: Matlab R2020b

Theory:

A Butterworth Filter is a type of Active Filter, where the frequency response of the across its pass band is relatively flat. Because of this frequency response, Butterworth Filters are also known as Maximally Flat Filters or Flat-Flat Filters.

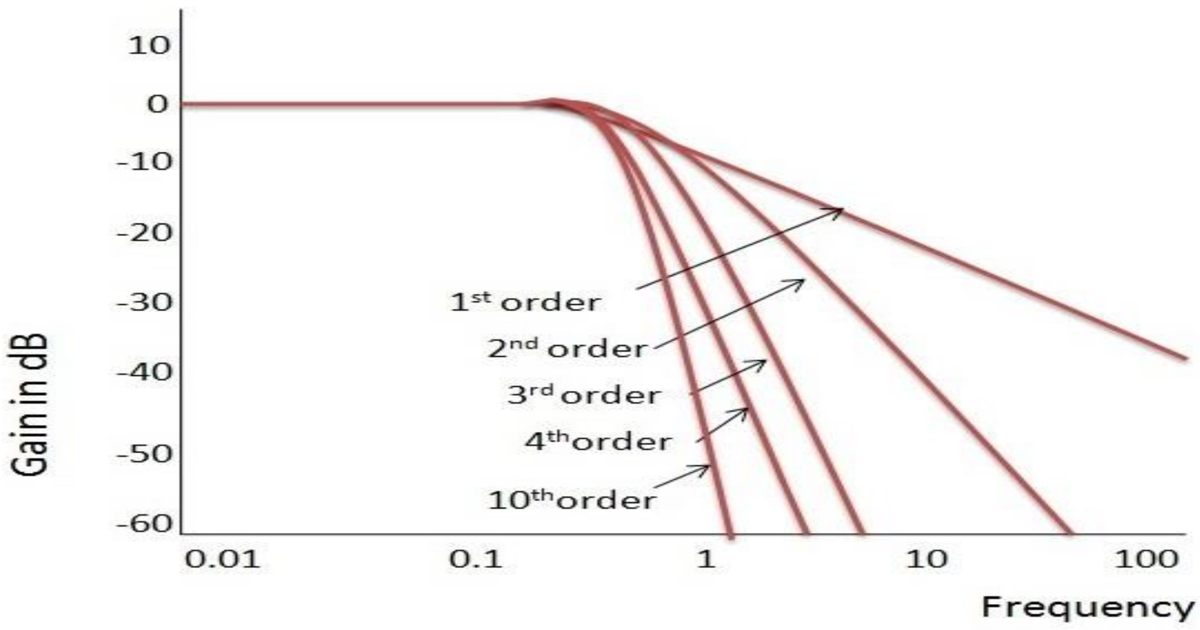
Using Butterworth Filter technique, you can design all types of filters i.e. High Pass, Low Pass, Band Pass etc. In this tutorial we will concentrate on Low Pass Filter Design using Butterworth Filter Technique.

The amplitude response of nth order Butterworth filter is given as follows:

$$V_{out} / V_{in} = 1 / \sqrt{\{1 + (f / f_c)^{2n}\}}$$

Where 'n' is the number of poles in the circuit. As the value of the 'n' increases the flatness of the filter response also increases.

'f' = operating frequency of the circuit and 'fc'= center frequency or cut off frequency of the circuit.



Normalized Low Pass Butterworth Filter

Normalization is a process in which voltage, current or impedance is divided by the quantity of the same unit of measure. This process is used to make a dimensionless range or level of particular value.

The denominator polynomial of the filter transfer function gives us the Butterworth polynomial. If we consider the s-plane on a circle with equal radius whose centre is at origin, then all the poles of the Butterworth filter are located in the left half of that s-plane.

For any order filter the co-efficient of the highest power of 's' should be always 1 and for any order filter the constant term is always 1. For even order filters all the polynomial factors are quadratic in nature. For odd order filters all the polynomials are quadratic except 1st order, for 1st order filter the polynomial is 1+s.

n (order)	Normalized Denominator Polynomials in Factored Form
1	$(1+s)$
2	$(1+1.414s+s^2)$
3	$(1+s)(1+s+s^2)$
4	$(1+0.765s+s^2)(1+1.848s+s^2)$
5	$(1+s)(1+0.618s+s^2)(1+1.618s+s^2)$
6	$(1+0.518s+s^2)(1+1.414s+s^2)(1+1.932s+s^2)$
7	$(1+s)(1+0.445s+s^2)(1+1.247s+s^2)(1+1.802s+s^2)$
8	$(1+0.390s+s^2)(1+1.111s+s^2)(1+1.663s+s^2)(1+1.962s+s^2)$
9	$(1+s)(1+0.347s+s^2)(1+s+s^2)(1+1.532s+s^2)(1+1.879s+s^2)$
10	$(1+0.313s+s^2)(1+0.908s+s^2)(1+1.414s+s^2)(1+1.782s+s^2)(1+1.975s+s^2)$

Butterworth polynomials in coefficients form is tabulated as given below.

Denominator coefficients for polynomials of the form $S_n + a_{n-1}s^{n-1} + a_{n-2}s^{n-2} + \ldots\ldots\ldots\ldots + a_1 s + a_0$

n	a_0	a_1	a_2	a_3	a_4	a_5	a_6	a_7	a_8	a_9
1	1									
2	1	1.414								
3	1	2.000	2.000							
4	1	2.613	3.414	2.613						
5	1	3.236	5.236	5.236	3.236					
6	1	3.864	7.464	9.142	7.464	3.864				
7	1	4.494	10.098	14.592	14.592	10.098	4.494			
8	1	5.126	13.137	21.846	25.688	21.846	13.137	5.126		
9	1	5.759	16.582	31.163	41.986	41.986	31.163	16.582	5.759	
10	1	6.392	20.432	42.802	64.882	74.233	64.882	42.802	20.432	6.392

The transfer function of the nth order Butterworth filter is given as follows

$$H(j\omega) = 1/\sqrt{\{1 + \varepsilon^2 (\omega/\omega_c)^{2n}\}}$$

Where n is the order of the filter

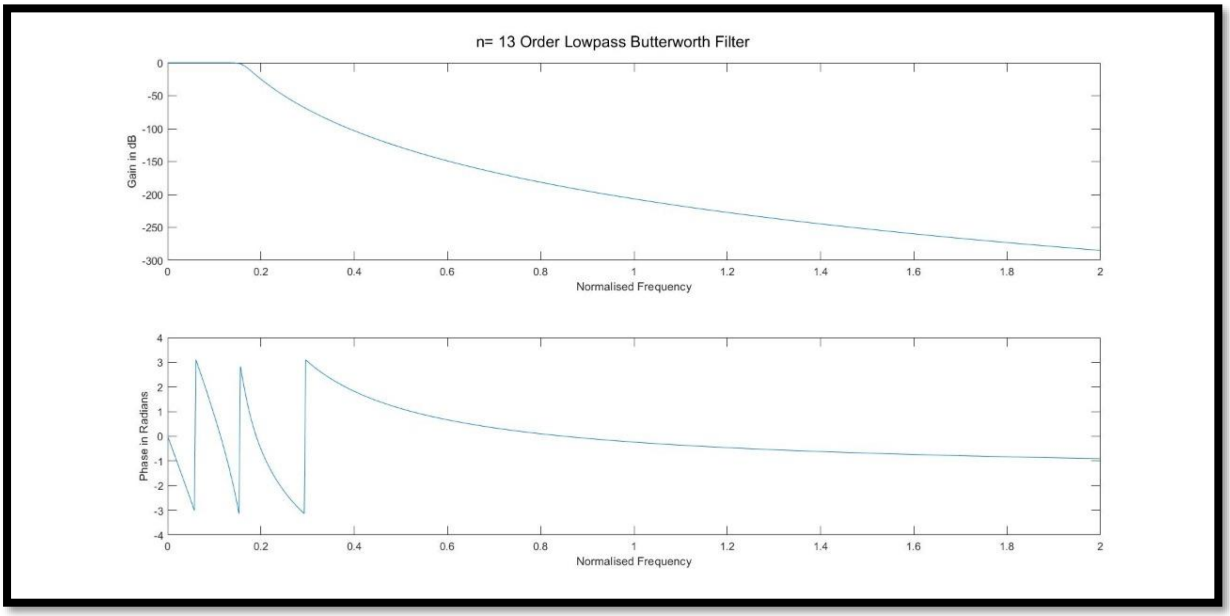
ω is the radian frequency and it is equal to $2\pi f$

And ε is the maximum pass band gain, Amax

Code:

```
% Lowpass filter
rp=0.15;
rs=60;
p=1500;
s=3000;
fs=7000;
wp=(p*2)./fs;
ws=(s*2)./fs;
[nl,wl]=buttord(wp,ws,rp,rs,'s');
[zl,pl,kl]=butter(nl,wl);
[b,a]=butter(nl,wl,'s');
w=0:0.01:2*pi;
[h,wo]=freqs(b,a,w);
m=20*log10(abs(h));
an=angle(h);
subplot(2,1,1);
plot(wo/pi,m);
xlabel("Normalised Frequency");
ylabel("Gain in dB");
subplot(2,1,2);
plot(wo/pi,an);
xlabel("Normalised Frequency");
ylabel("Phase in Radians");
sgtitle(sprintf('n= %d Order Lowpass Butterworth
Filter',nl));
figure;
```

Output:



Conclusion:

The order of Low Pass Filter problem is n=13 which we determined by using the buttord().The function butter() is used to plot the frequency and phase response and get the output plots in MATLAB for the given specifications to implement Butterworth Low-pass filter.

Experiment -2

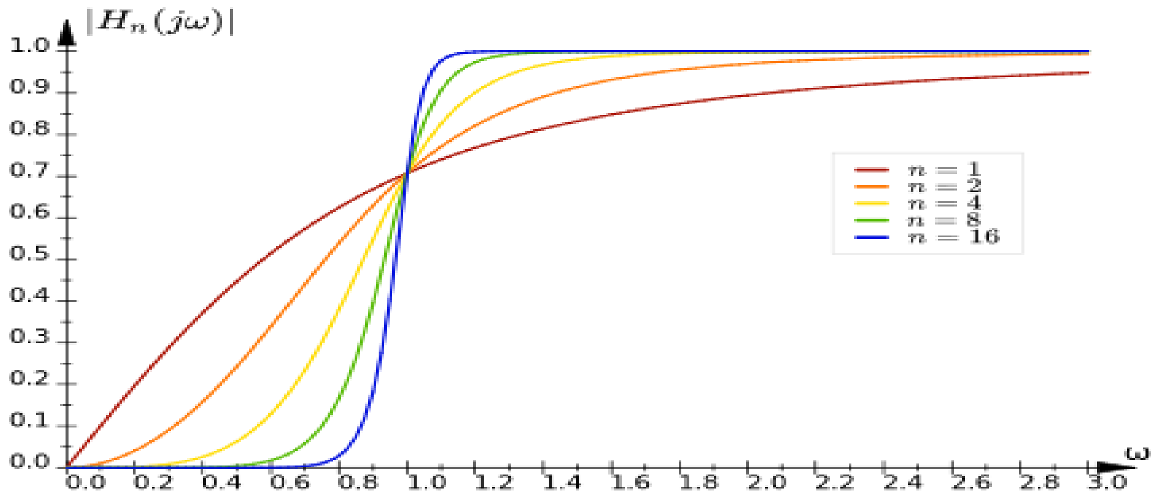
Aim : To design a Butterworth analog nth order High pass filter.

Software used: Matlab R2020b

Theory:

A Butterworth Filter is a type of Active Filter, where the frequency response of the across its pass band is relatively flat. Because of this frequency response, Butterworth Filters are also known as Maximally Flat Filters or Flat-Flat Filters.

$$|H_{n,hp}(j\omega)| \triangleq \frac{1}{\sqrt{1+\omega^{-2n}}}$$



We can just multiply the numerator and the denominator by ωn to get a more familiar form:

$$|H_{n,hp}(j\omega)| = \frac{\omega^n}{\sqrt{1+\omega^{2n}}}$$

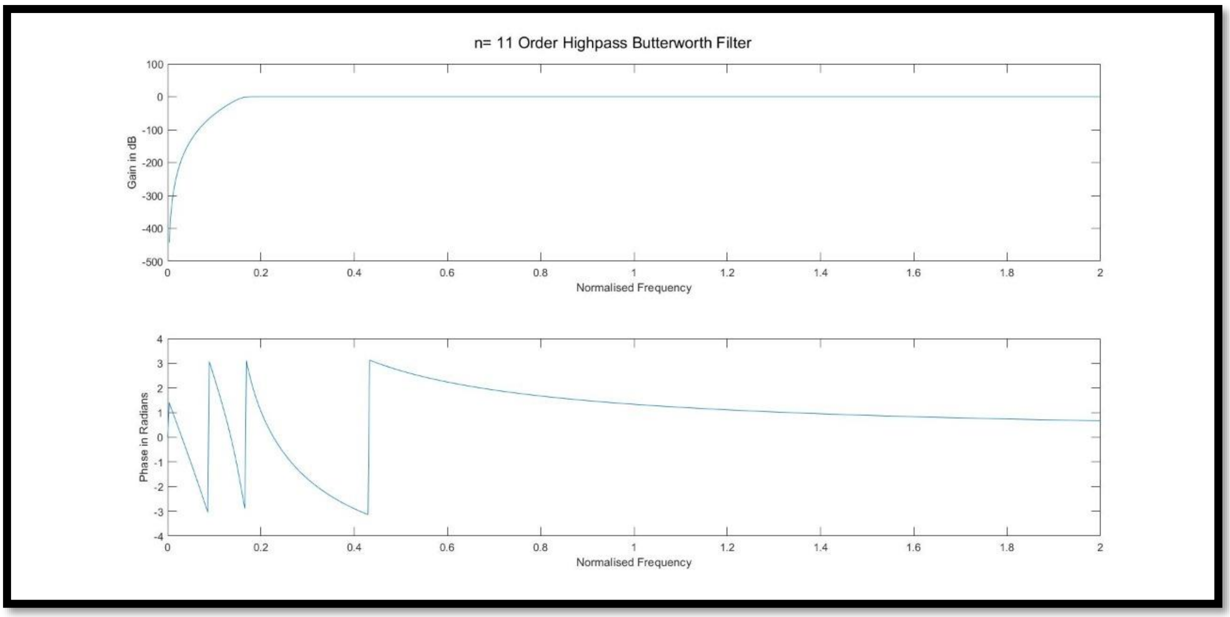
As you can see, the poles will be the same as for the low-pass version. On top of that, there now are n zeros for $s=0$.
So, the transfer function becomes:

$$H_{n,hp}(s) = \frac{s^n}{B_n(s)}$$

Code:

```
% Highpass filter
rp=0.2;
rs=40;
p=2000;
s=3500;
fs=6000;
wp=(p*2)./fs;
ws=(s*2)./fs;
[nh,wh]=buttord(wp,ws,rp,rs,'s');
[zh,ph,kh]=butter(nh,wh,'high','s');
[b,a]=butter(nl,wl,'high','s');
w=0:0.01:2*pi;
[h,wo]=freqs(b,a,w);
m=20*log10(abs(h));
an=angle(h);
subplot(2,1,1);
plot(wo/pi,m);
xlabel("Normalised Frequency");
ylabel("Gain in dB");
subplot(2,1,2);
plot(wo/pi,an);
xlabel("Normalised Frequency");
ylabel("Phase in Radians");
sgtitle(sprintf('n= %d Order Highpass Butterworth
Filter',nh));
```

<u>Output:</u>



Conclusion:

The order of High Pass Filter problem is n=11 which we determined by using the buttord().The function butter() is used to plot the frequency and phase response and get the output plots in MATLAB for the given specifications to implement Butterworth High-pass filter.

Experiment-3

Aim: To Design a two op-amp narrow-band, RC notch filter with a center notch frequency, fN of 1kHz and a -3dB bandwidth of 100 Hz by using 0.1uF capacitors in the design and calculating the expected notch depth in decibels. Plot the frequency response.

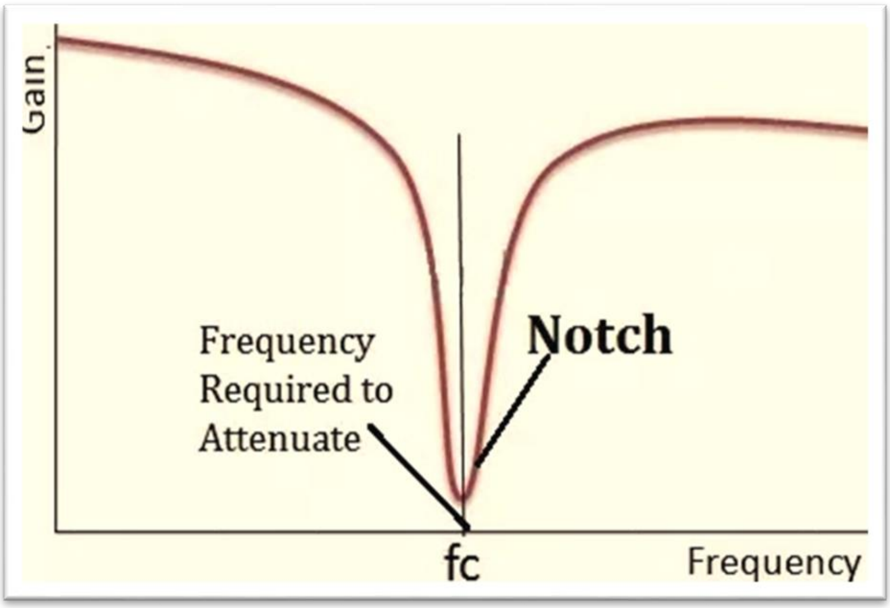
Software Used: Multisim v14.2

Theory:

A Notch filter is nothing but a narrow band-stop filter. If the stopband of band-stop filter is very narrow and highly attenuated over a few hertz, then that special type of band-stop filter is known as Notch filter. Since the stop band of Notch filter is narrow upto few Hz so the Notch filter is also known as Narrow band stop filter in some cases. It is a highly selective (High Q value) form of band-stop filter which can be used to block a single or very small band of frequencies rather than whole bandwidth of different frequencies.
It acts as a gain for one frequency component and attenuator for all other frequencies.
Notch filter by design has a very narrow stop band around their center frequency. The width of the notch is being calculated by its selectivity Q in exactly the same way as the resonance frequency peak is calculated in the RLC circuit.

Notch Filter Frequency Response

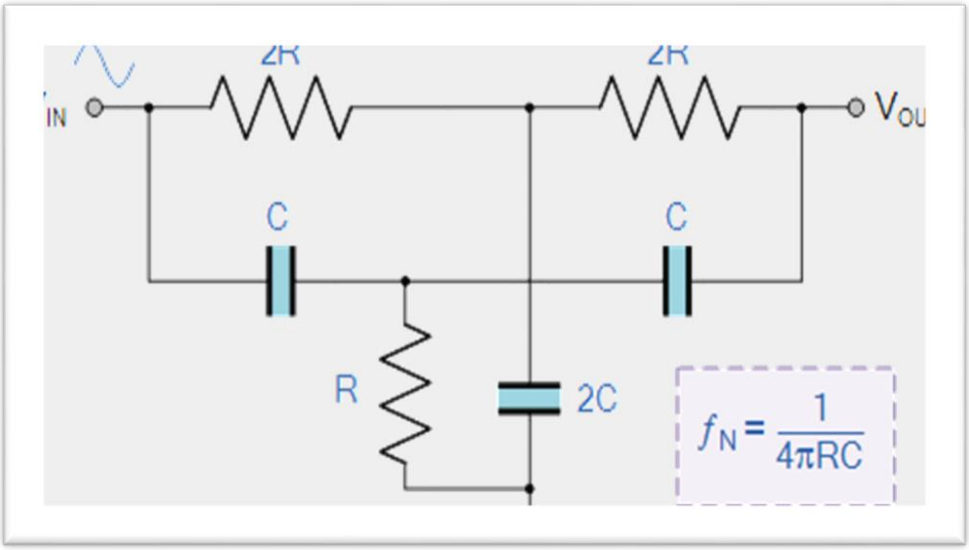
The ideal response of any notch filter would be a completely flat response over the usable range with the exception of notch frequency as shown in the figure below.



Practically perfection is very tuff to achieve. But by using the operational amplifier circuit, a high level of attenuation and narrow notch can be achieved.

Notch Filter Circuit

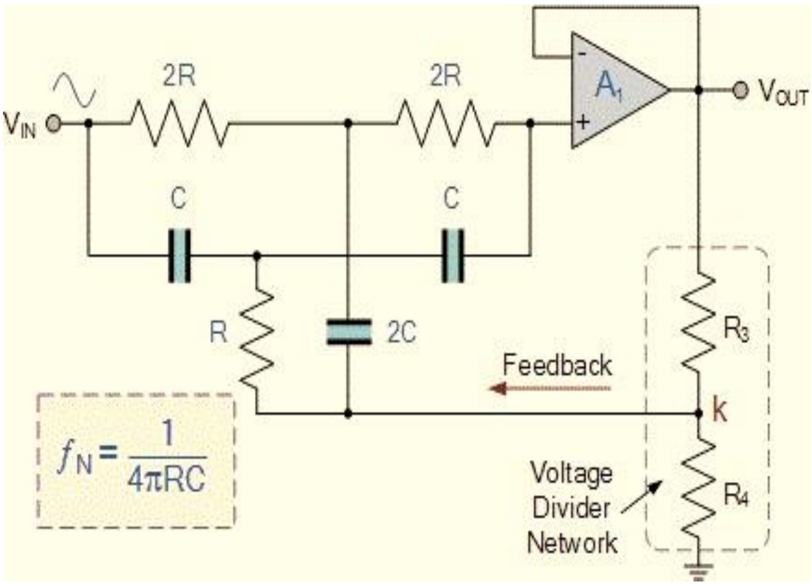
The most common Notch filter design is the twin-T notch filter network. In its basic form, the twin-T also called a parallel-T configuration. It consists of two RC branches in the form of two tee sections connected in parallel.
The basic twin-T Notch filter circuit is shown in the figure below as:



The upper T-configuration of resistor 2R and capacitor 2C form the low pass filter section, whereas the lower T-configuration of resistor R and capacitor C form the high pass filter section of the design.
The frequency at which this basic twin-T notch filter design offers maximum attenuation is called **Notch frequency**. So, notch frequency is formulated as:

$$f_N = \frac{1}{4\pi RC}$$

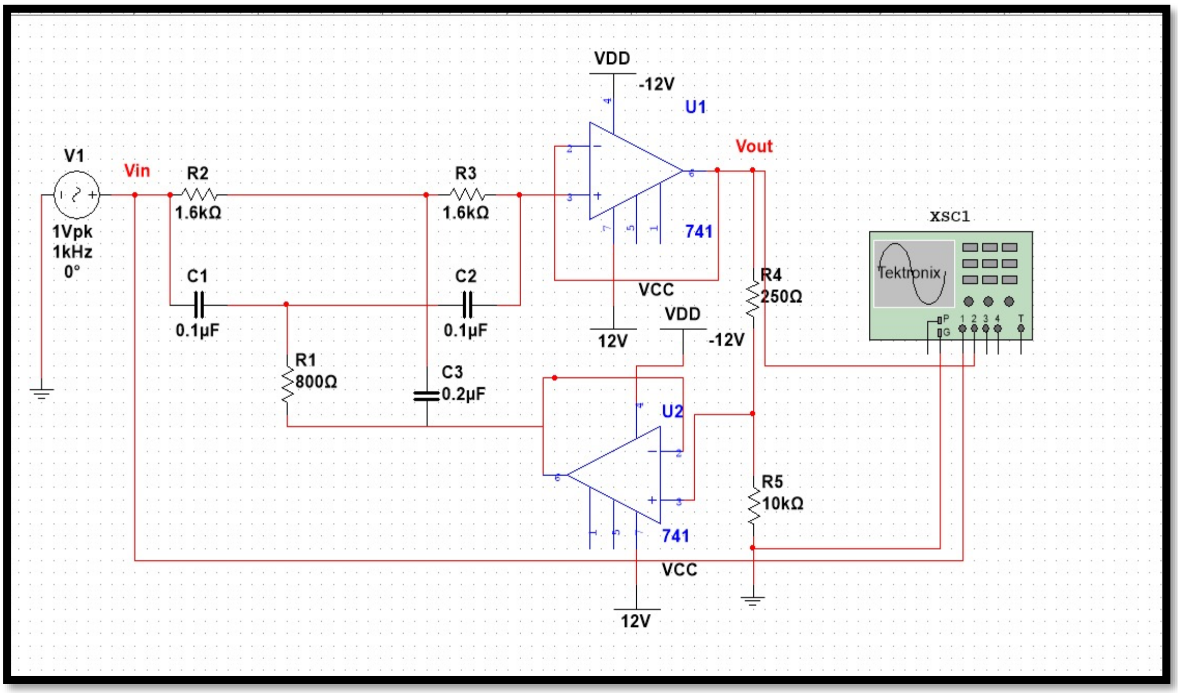
For obtaining a high level of attenuation and narrow notch, an operational amplifier is used to design a single Op-Amp twin-T notch filter circuit. The single Op-Amp twin-T notch filter circuit is shown in the below figure as:



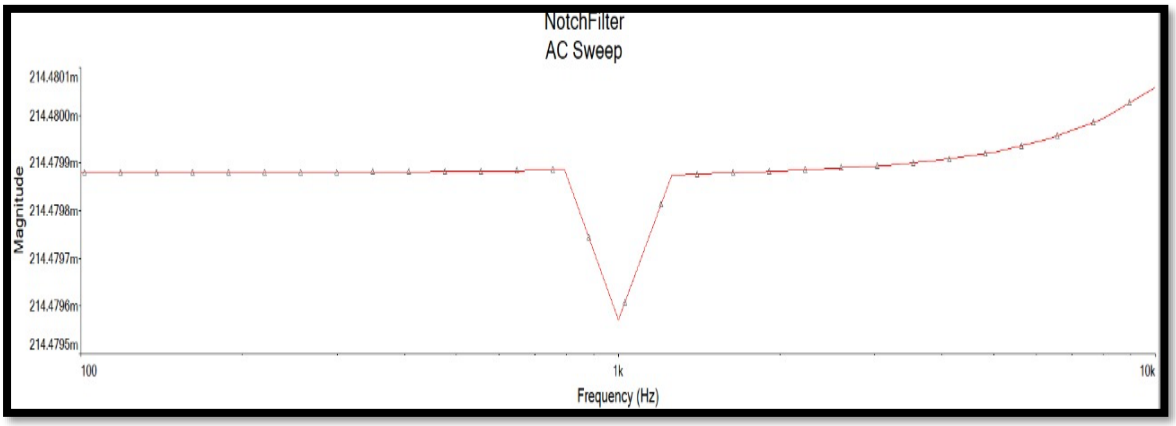
Notch Filter Design

For designing a two Op-Amp narrow band RC notch filter with center notch frequency of 1kHz and 3-dB bandwidth of 100 Hz. Consider the 0.1uF capacitor and calculate all the value of the required components as explained in the below steps.

Circuit Design:



Frequency Response:



Calculation:

Step 1: Calculate the value of R for the given capacitance of 0.1uF.

$$R = \frac{1}{4\pi f n C} = \frac{1}{4\pi(1000)(0.1)(10^{-6})} = 795\Omega \approx 800\Omega$$

Step 2: Calculate the value of Q.

$$Q = \frac{fn}{BW} = \frac{1000}{100} = 10$$

Step 3: Calculate the value of feedback fraction K.

$K = 1 - \frac{1}{4Q} = 1 – 0.025 = 0.975$

Step 4: Calculate the value of R3 and R4.

$K = \frac{R3}{R3+R4} = 0.975$

Assume R4 = 10 kΩ,

R3 = 10000 (1 - 0.975) = 250Ω

Step 5: Finally Calculate the value of notch depth in dB as:

f_N(in db) = 20.log (1/Q) = 20.log (1/10)

= -20 db

Conclusion:

Designed a circuit with two narrow-band op-amps and a RC notch filter. Plotted the frequency response using Multisim software and the notch depth was achieved and calculated in decibels from the designed circuit.

<u>Experiment – 4</u>

<u>Aim:</u> To find out the output waveforms for the AM modulator and demodulator circuit design using IC 1496.

<u>Software Used:</u> Multisim v14.2.

<u>Theory:</u>

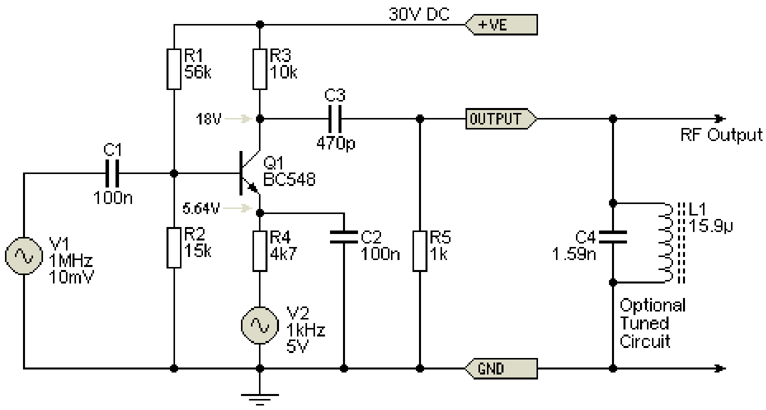


Figure 1 - Simple Transistor Modulator

The transistor circuit works because the gain of Q1 is changed as its emitter current changes, caused by the audio waveform appearing at the emitter. The amplitude of the carrier waveform is modulated by the transistor's non-linearity. However, the circuit - whether simulated or built with real parts - has poor distortion performance, so the audio and RF waveforms are both distorted. If one does a FFT (Fast Fourier Transform) of the waveform, there are countless harmonics, and it's not really a viable option if you need a nice clean AM waveform. It's obviously pointless trying to determine the distortion from a detector if the audio waveform is already distorted. The tuned circuit is optional and is described below.

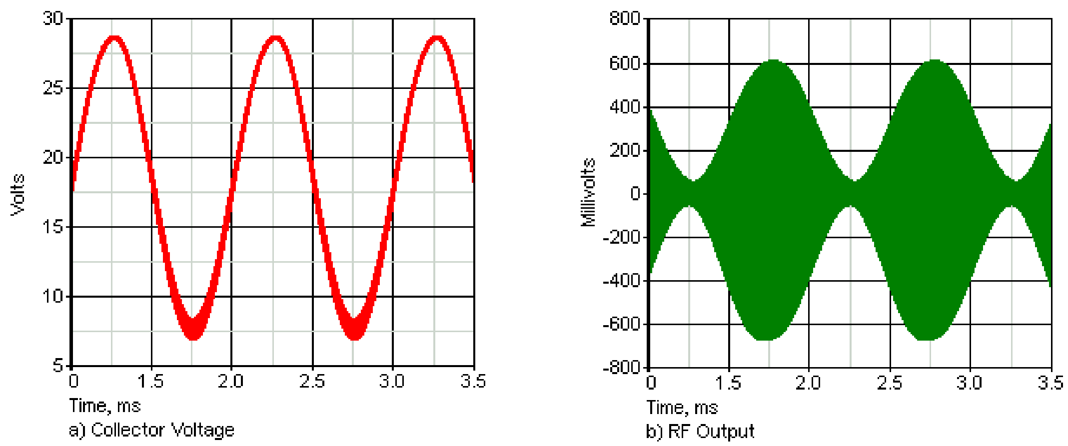


Figure 2- Transistor Modulator Waveforms (No Tuned Circuit)

In the above, a) shows the waveform at the collector of Q1. The 1MHz RF carrier is at a low level, and only shows up as 'fuzz' on the audio signal, with the amplitude of the fuzz varying over the audio cycle. C3 and R5 are used to filter out the low frequency (audio) component so only the RF gets through to the output. The AM output is shown in b) and you can see that it is distorted - note that this is *without* the tuned circuit. The distortion is subtle, but the modulated waveform isn't as clean as it should be. Note that the positive and negative peaks are offset slightly. This doesn't matter, because only one sideband is normally detected, but it still demonstrates imperfect modulation.

The missing link is the tuned circuit (a bandpass filter), and when that's added the RF waveform is improved (the symmetry of the RF envelope is greatly improved), but it's still far from ideal. While the tuned circuit makes the RF waveform a lot cleaner, it doesn't help the audio component, so the distortion after detection won't be as low as you need to be able to accurately measure the results from the detector you are working with.

AM Modulation using MC1496:

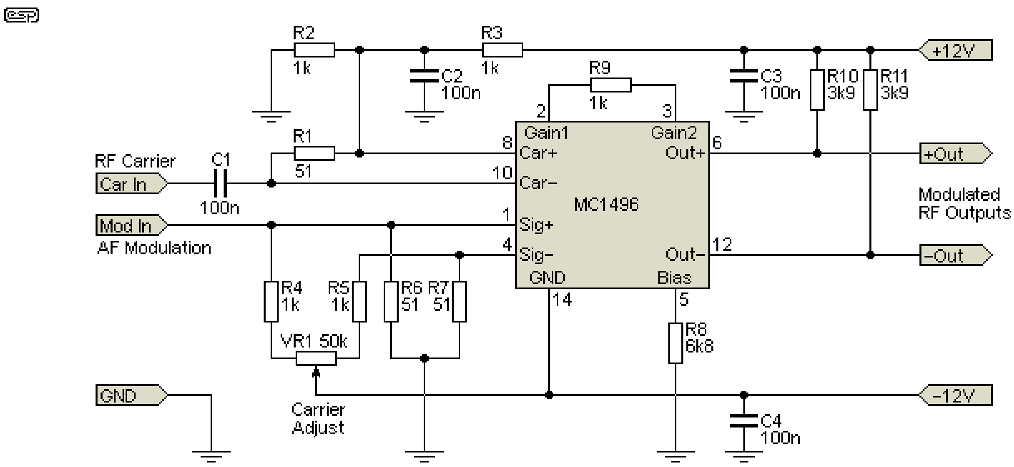


Figure 3- MC1496 Amplitude Modulator

The levels of both RF (carrier) and AF (audio modulation) must be well within the maxima that the IC can handle, or the output will be distorted. Note that the modulation input has a *very* low input impedance, set by R6, and is 51 ohms as shown. An input resistor will generally be necessary to reduce the signal level to a few millivolts at most - an initial value of around 1k is suggested. This will provide 100mV at the IC with an input voltage of around 2.5V RMS. The RF level needs to be around 300mV RMS (according to the datasheet). The output level will be very small without additional amplification - expect no more than around 500µV peak between +Out and -Out.

The AF and RF levels need to be set carefully, using an oscilloscope and (ideally) a frequency analyzer. The latter is a serious piece of kit, and the FFT function of a digital oscilloscope will probably be sufficient. The output is monitored using an AM radio. You'll probably need to include a (very) small 'power amplifier' to feed the aerial, which should include a broadly tuned circuit if you need to tune the carrier frequency, or a high Q filter for a fixed frequency.

AM Detection

The recovery of the original audio modulating frequency. This type of detector was one of the very first ever used to detect RF, and although there were other, earlier, detectors they weren't linear and were often insensitive. The circuit below shows a simple Schottky detector, with 800mV of forward bias applied to improve linearity. The tuned circuit and antenna shown are for the sake of completeness, but would not normally be included in a simulation. Note that C2 is essential if the source (your modulator) is DC coupled. If you leave out C2, the diode detector won't have any forward bias, and that increases distortion dramatically. The anode of D1 *must* have a DC return path or it won't work at all.

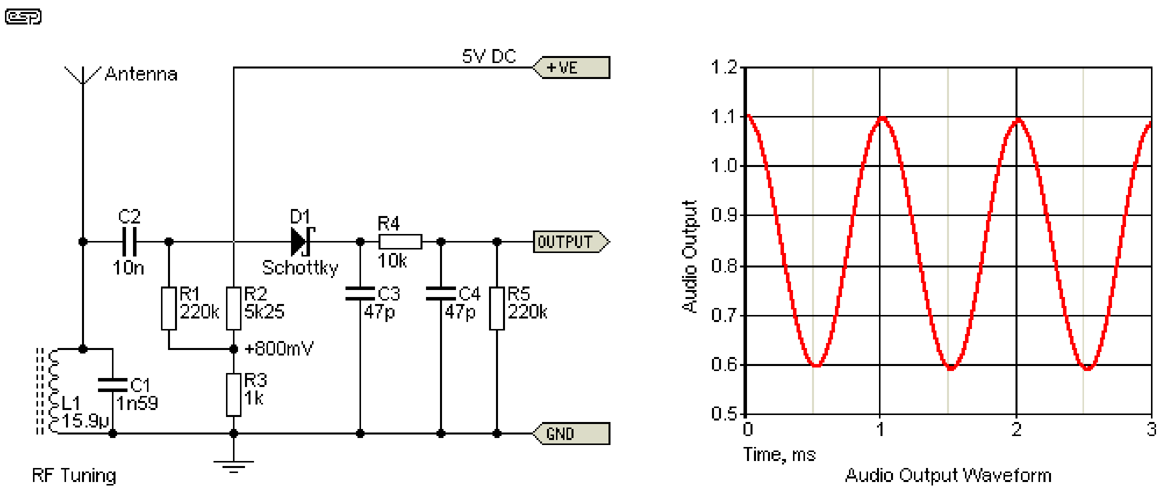


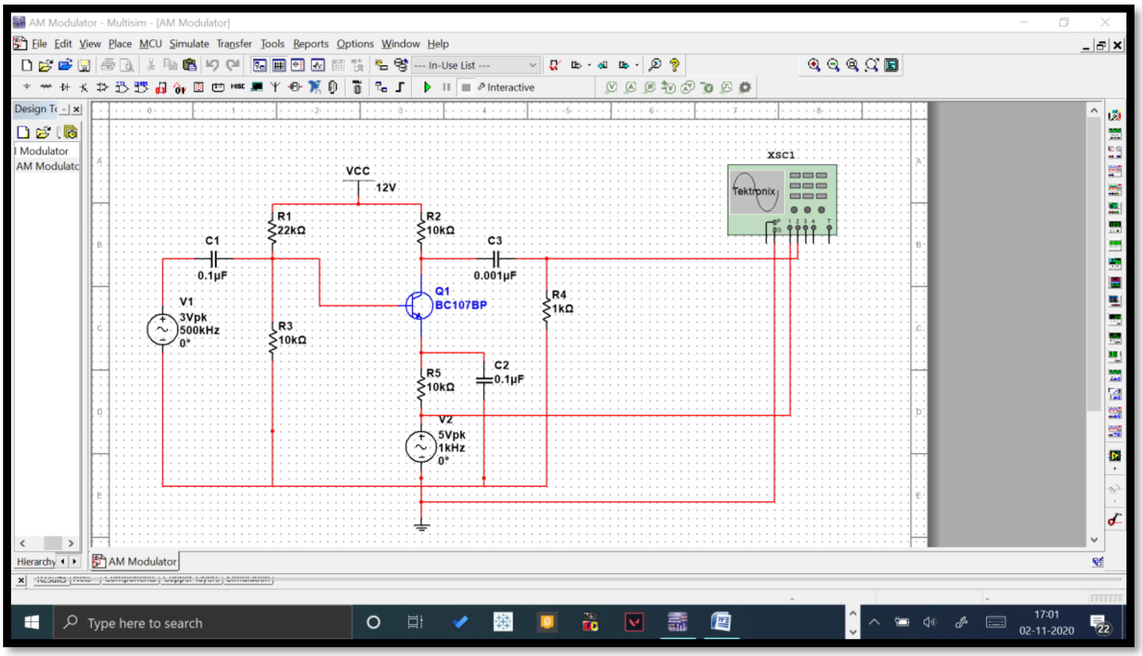
Figure 4 - Diode Based AM Detector/ Demodulator

In most radio receivers, the average DC level is used to activate the circuit's AGC (automatic gain control). This is designed to keep the intermediate frequency amplitude reasonably constant at the detector's input as different stations are tuned in, so that the audio level remains fairly steady. Without AGC, the audio level is entirely dependent on the strength of the received signal. The DC must be removed from the audio signal before being passed to the audio amplifier stage, and this is done simply with a coupling capacitor.

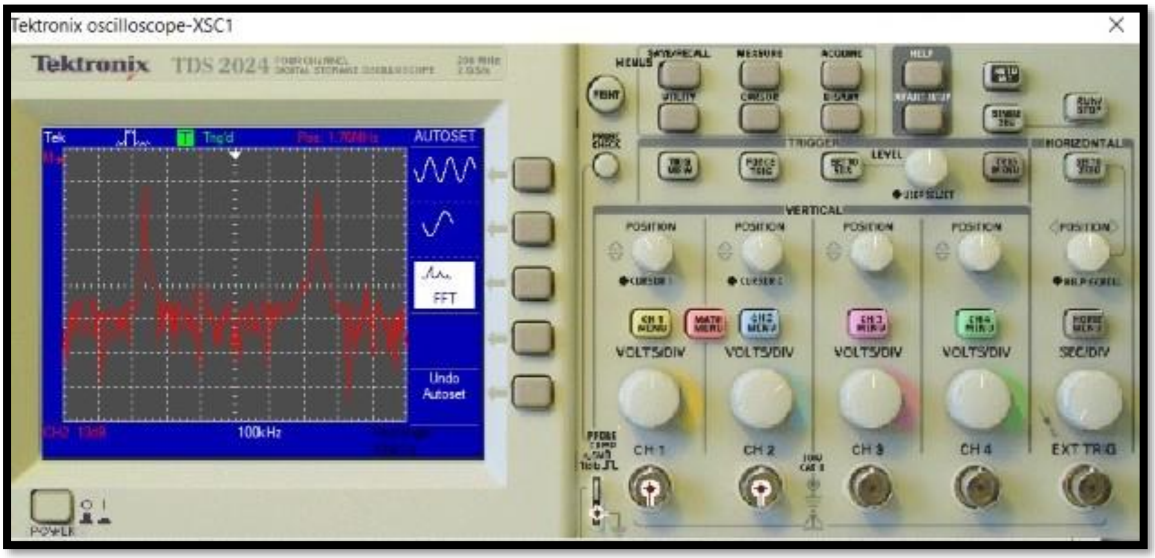
An ideal detector will *perfectly* half-wave rectify the RF envelope, so that the audio waveform is preserved intact. It makes no difference whether the positive or negative half cycles are demodulated, since the same audio information is present in both. The RF component is then removed with a low pass filter, leaving only the audio and a DC level that depends on the RF amplitude. The DC is easily removed with a capacitor, leaving only the audio, which will hopefully be an exact replica of the signal used to modulate the transmitter. While the concept is simple in theory, it is very difficult to achieve in practice, and there are many different solutions (including applying forward bias as shown above).

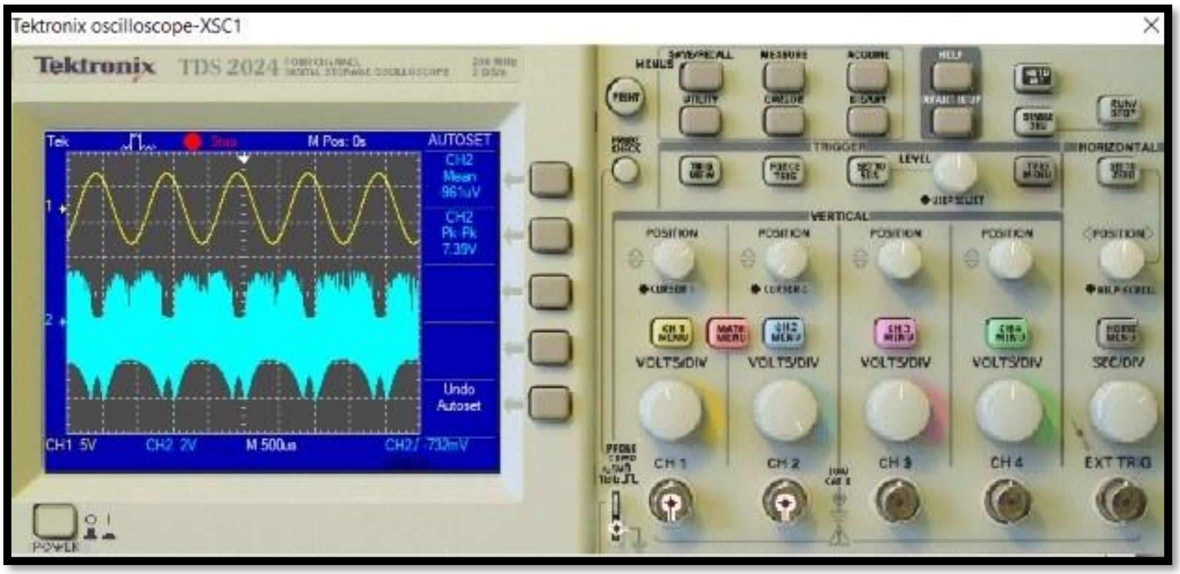
AM MODULATOR:

CIRCUIT:



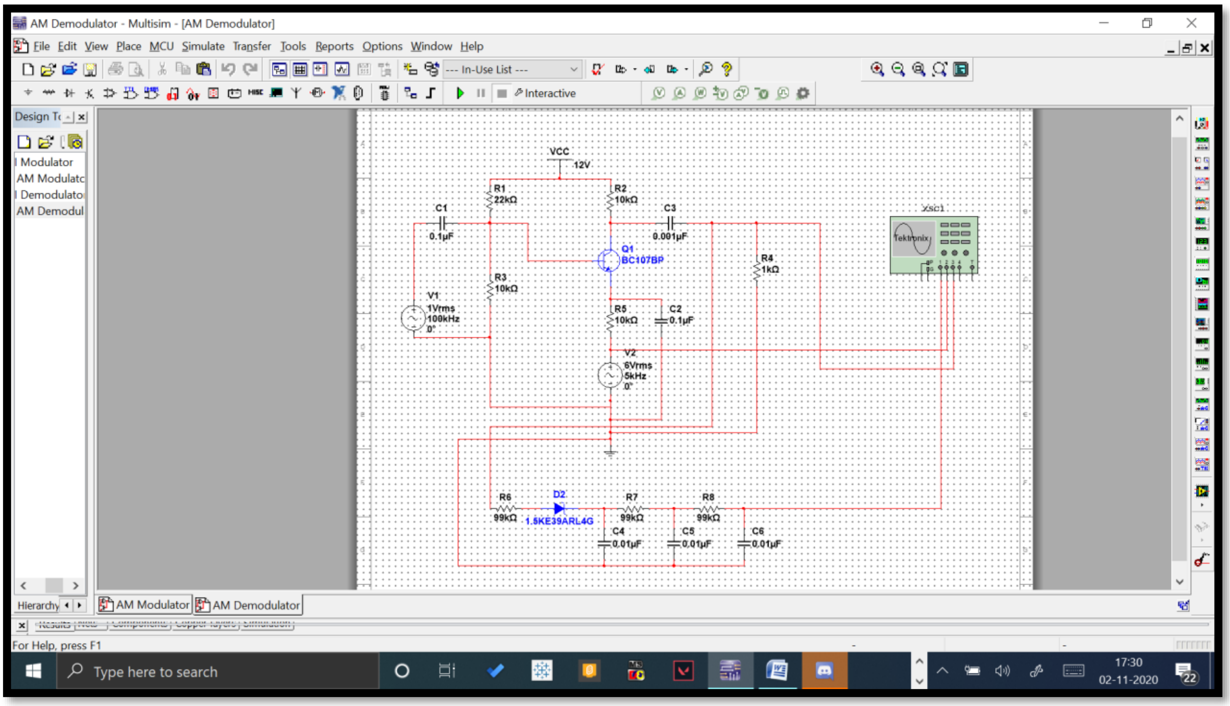
OUTPUT WAVEFORM OF AM MODULATOR ON THE OSCILLOSCOPE:



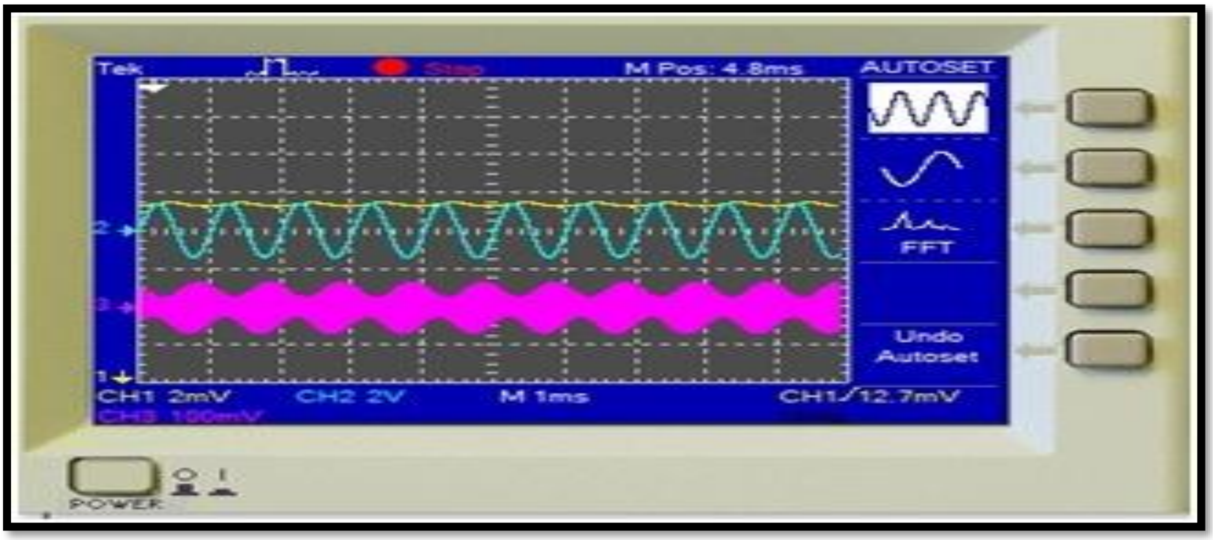


AM DEMODULATOR:

CIRCUIT:



OUTPUT WAVEFORM OF AM DEMODULATOR ON THE OSCILLOSCOPE:



Conclusion:

Observed the output waveforms for AM modulator and Demodulator circuit design using IC 1496 by simulating it in NI Multisim.

Experiment-5

Aim: To design a circuit for FM modulation and demodulation.

Software used: NI Multisim 14.2

Theory:

Frequency modulation (FM) is the encoding of information in a carrier wave by varying the instantaneous frequency of the wave. The technology is used in telecommunications, radio broadcasting, signal processing, and computing.

In analog frequency modulation, such as radio broadcasting, of an audio signal representing voice or music, the instantaneous frequency deviation, i.e. the difference between the frequency of the carrier and its center frequency, has a functional relation to the modulating signal amplitude.

Digital data can be encoded and transmitted with a type of frequency modulation known as frequency-shift keying (FSK), in which the instantaneous frequency of the carrier is shifted among a set of frequencies. The frequencies may represent digits, such as 0 and 1. FSK is widely used in computer modems, such as fax modems, telephone caller ID systems, garage door openers, and other low frequency transmissions.[1] Radioteletype also uses FSK.[2]

Frequency modulation is widely used for FM radio broadcasting. It is also used in telemetry, radar, seismic prospecting, and monitoring newborns for seizures via EEG,[3] two-way radio systems, sound synthesis, magnetic tape-recording systems and some video-transmission systems. In radio transmission, an advantage of frequency modulation is that it has a larger signal-to-noise ratio and therefore rejects radio frequency interference better than an equal power amplitude modulation (AM) signal. For this reason, most music is broadcast over FM radio.

Types of FM Demodulators:

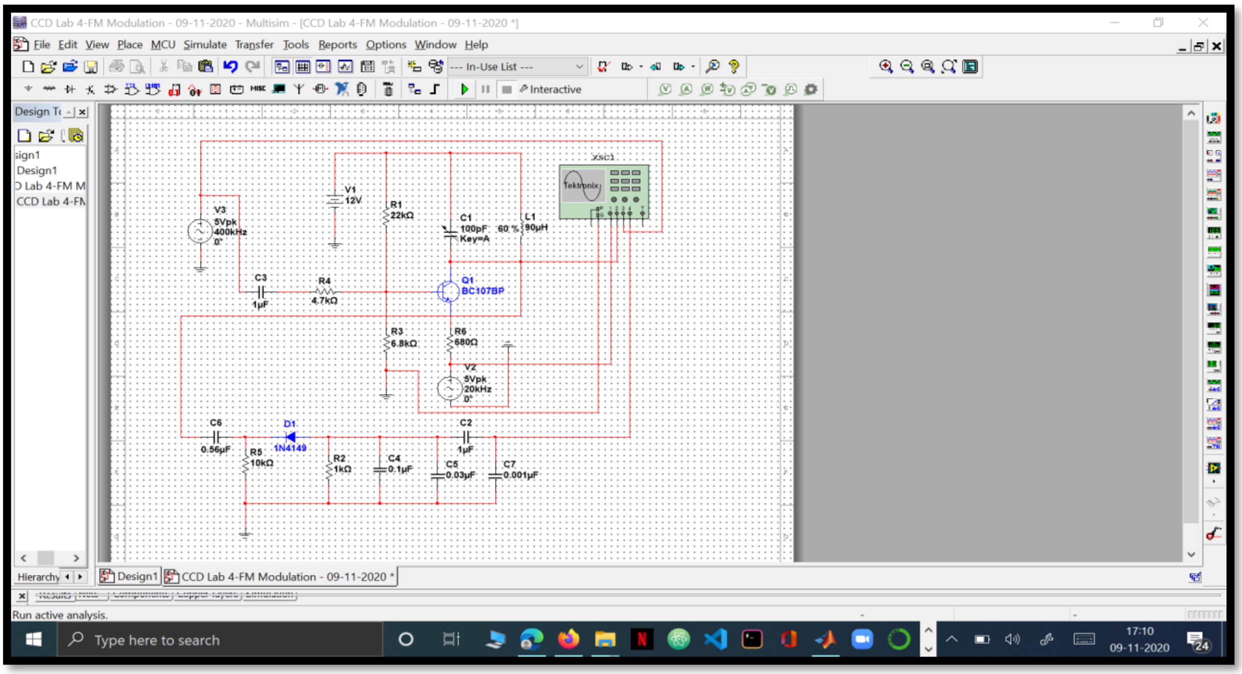
There are several types of FM detector / demodulator that can be used. Some types were more popular in the days when radios were made from discrete devices, but nowadays the PLL based detector and quadrature / coincidence detectors are the most widely used as they lend themselves to being incorporated into integrated circuits very easily and they do not require many, if any adjustments.

To improve the noise performance of the FM receiver, typically the IF stage may operate such that the IF amplifier is driven into limiting. This removes the amplitude variations, that will result in noise, and only allows through the frequency variations.

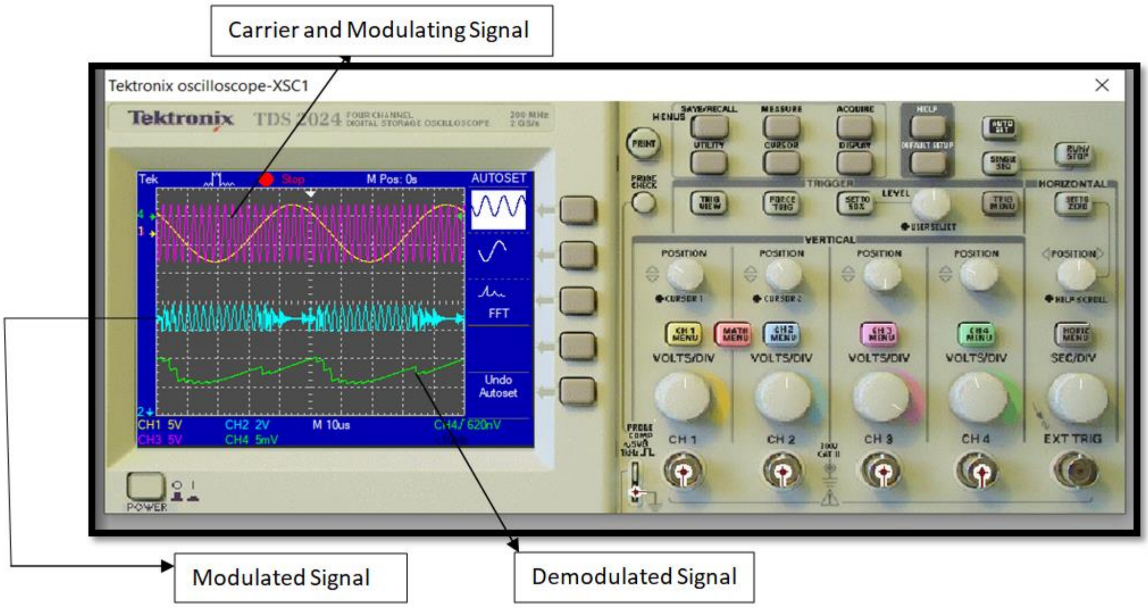
There main types of FM demodulator found in broadcast receivers, radio communication systems two way radios or walkie talkies / handheld radios, etc, are outlined below:

1) Slope detection.
2) Ratio detector.
3) Foster Seeley FM.
4) Phase Locked Loop Demodulator.
5) Quadrature Detector.

Circuit Diagram:



Output:



Conclusions:

We studied the FM modulation of a message signal with the help of multisim 14.2.The output graphs of the message signal, carrier signal , modulated and demodulated signal were observed.

Experiment-6

Aim: To Design a Sample and Hold Circuit

Software Used: NI Multisim 14.2

Theory:

In electronics, a sample and hold (also known as sample and follow) circuit is an analog device that samples (captures, takes) the voltage of a continuously varying analog signal and holds (locks, freezes) its value at a constant level for a specified minimum period of time. Sample and hold circuits and related peak detectors are the elementary analog memory devices. They are typically used in analog-to-digital converters to eliminate variations in input signal that can corrupt the conversion process.[1] They are also used in electronic music, for instance to impart a random quality to successively-played notes.

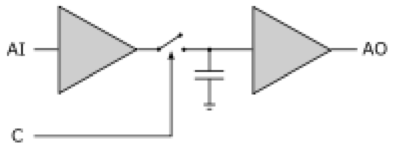


Figure 1: A simplified sample and hold circuit diagram.

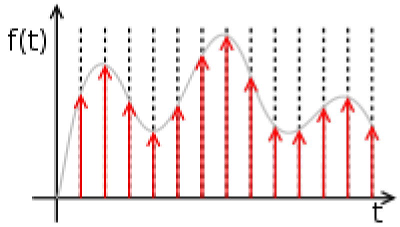


Figure 2: Sample times.

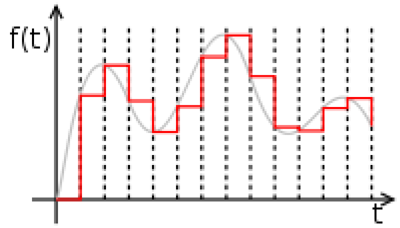
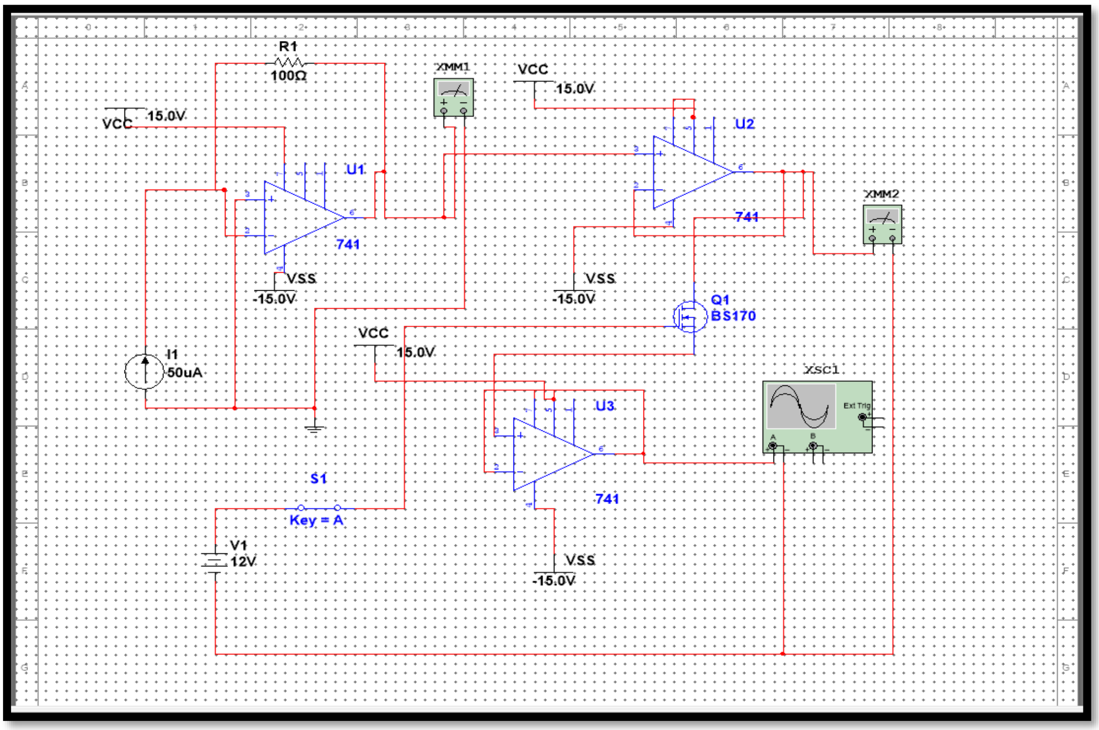


Figure 3: Sample and hold.

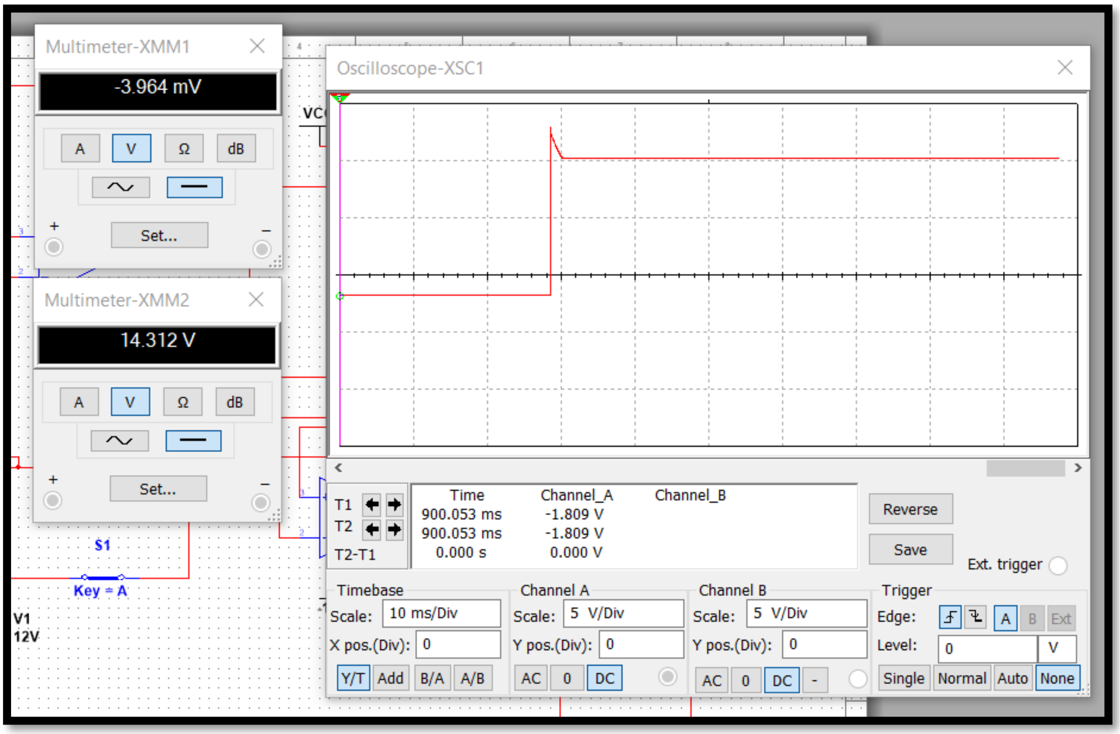
A typical sample and hold circuit stores electric charge in a capacitor and contains at least one switching device such as a FET (field effect transistor) switch and normally one operational amplifier.[2] To sample the input signal the switch connects the capacitor to the output of a buffer amplifier. The buffer amplifier charges or discharges the capacitor so that the voltage across the capacitor is practically equal, or proportional to, input voltage. In hold mode the switch disconnects the capacitor from the buffer. The capacitor is invariably discharged by its own leakage currents and useful load currents, which makes the circuit inherently volatile, but the loss of voltage (voltage drop) within a specified hold time remains within an acceptable error margin for all but the most demanding applications.

Circuit Diagram:

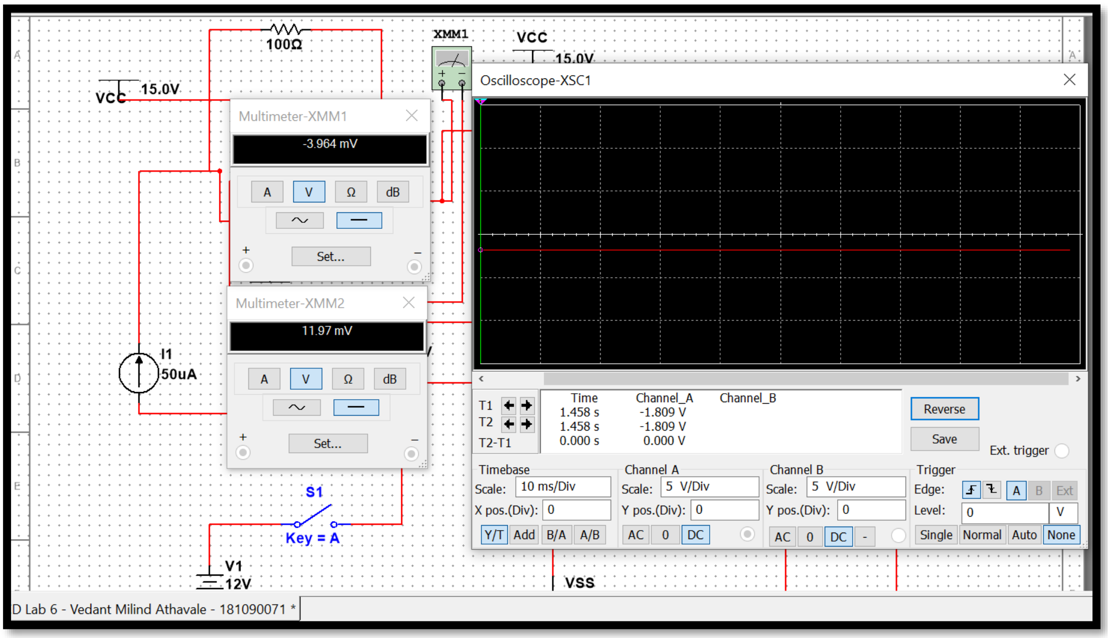


Output:

When S1 key is closed: (With Reversed Background Color for better visuals of plot)



When S1 key is open:



Conclusion:

Designed and observed the circuit diagram of Sample and Hold Circuit along with its working, output voltages and its plots in Multisim v14.2

Experiment-7

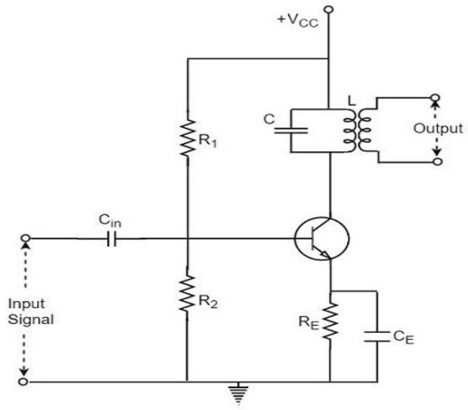
Aim: To design and observe the circuit diagram of Tuned Amplifier Circuit in NI Multisim.

Software Used: NI Multisim 14.2

Theory:

Tuned amplifiers are the amplifiers that are employed for the purpose of **tuning**. Tuning means selecting. Among a set of frequencies available, if there occurs a need to select a particular frequency, while rejecting all other frequencies, such a process is called **Selection**. This selection is done by using a circuit called as **Tuned circuit**.

When an amplifier circuit has its load replaced by a tuned circuit, such an amplifier can be called as a **Tuned amplifier circuit**. The basic tuned amplifier circuit looks as shown below.



The tuner circuit is nothing but a LC circuit which is also called as **resonant** or **tank circuit**. It selects the frequency. A tuned circuit is capable of amplifying a signal over a narrow band of frequencies that are centred at resonant frequency.

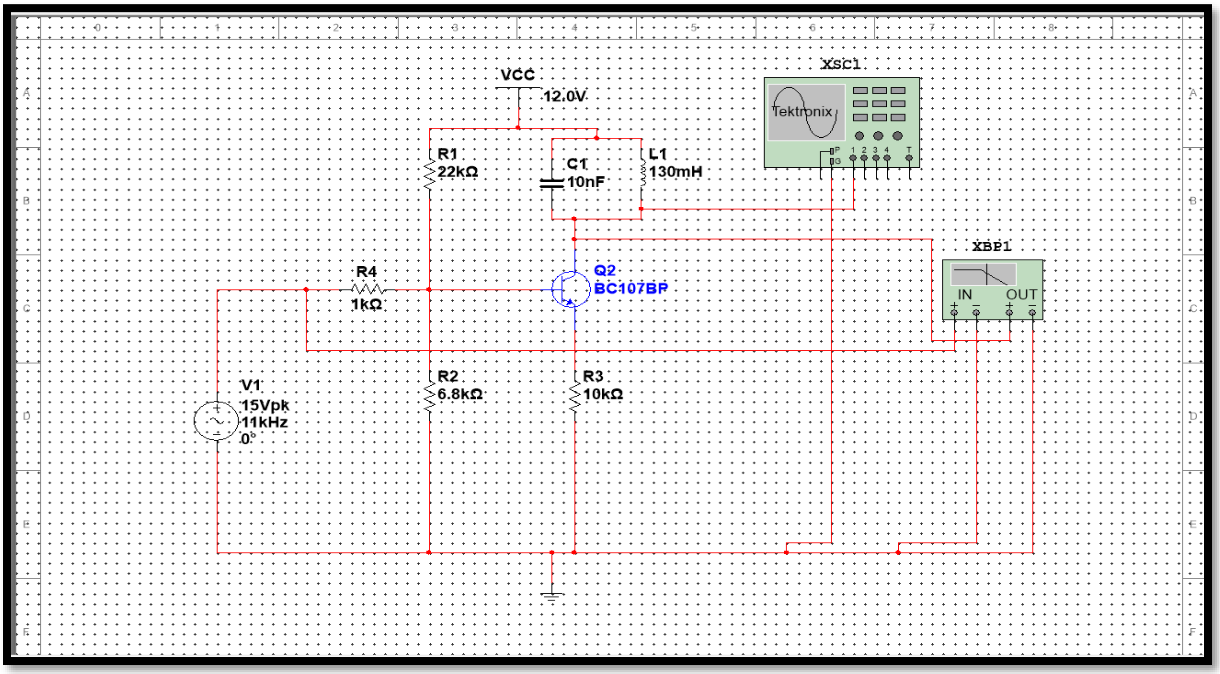
When the reactance of the inductor balances the reactance of the capacitor, in the tuned circuit at some frequency, such a frequency can be called as **resonant frequency**. It is denoted by $\mathbf{f_r}$.

The formula for resonance is

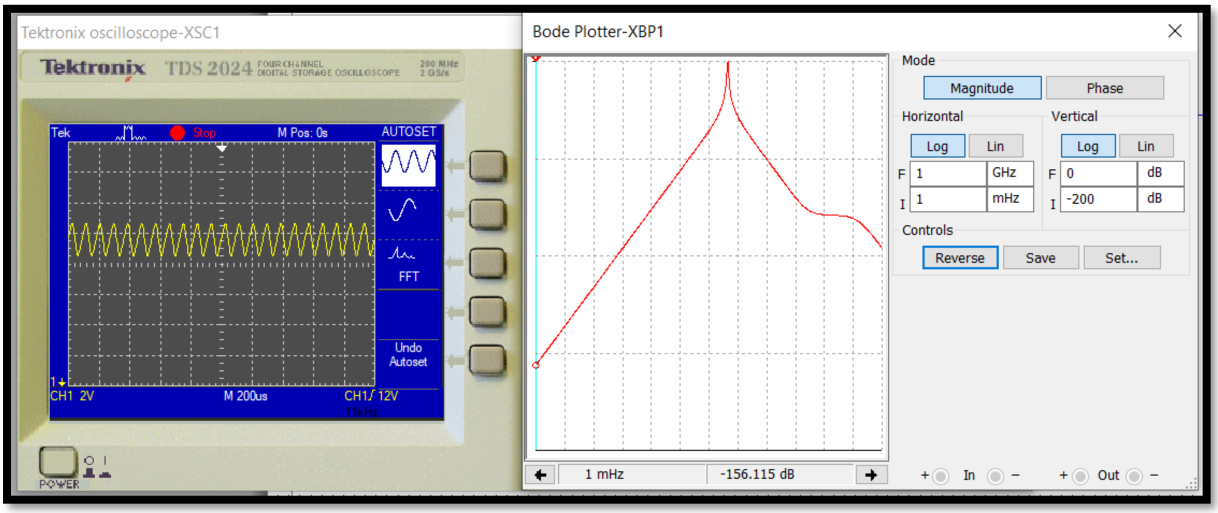
$$2\pi f_L = \frac{1}{2\pi f_c}$$

$$f_r = \frac{1}{2\pi\sqrt{LC}}$$

Circuit Diagram:



Output:



Conclusion:

Designed and Observed the circuit diagram, working and output voltages and plots of tuned amplifier circuit with Multisim 14.2 .

-------------x----------------x-------------