TE0222-Electronic Circuits & DSP Lab
(2011-2012)
# CONTENTS

<table>
<thead>
<tr>
<th>S.No</th>
<th>Name of the Experiment</th>
<th>Page No</th>
</tr>
</thead>
<tbody>
<tr>
<td>1.</td>
<td>Frequency Response Of a BJT Amplifier with And without Feedback Amplifier</td>
<td>4</td>
</tr>
<tr>
<td>2.</td>
<td>Single Tuned Amplifier</td>
<td>12</td>
</tr>
<tr>
<td>3.</td>
<td>Colpitts Oscillator</td>
<td>18</td>
</tr>
<tr>
<td>4.</td>
<td>Class-AB Complimentary Symmetry Push Pull Amplifier</td>
<td>23</td>
</tr>
<tr>
<td>5.</td>
<td>RC Phase Shift Oscillator</td>
<td>27</td>
</tr>
<tr>
<td>6.</td>
<td>Generation of Discrete Waveforms</td>
<td>35</td>
</tr>
<tr>
<td>7.</td>
<td>Generation of Sinusoidal and Triangular Waveforms</td>
<td>39</td>
</tr>
<tr>
<td>8.</td>
<td>Generation of DFT Using Matlab</td>
<td>41</td>
</tr>
<tr>
<td>9.</td>
<td>Generation of Linear and Circular Convolution</td>
<td>44</td>
</tr>
<tr>
<td>10.</td>
<td>Design of Finite Impulse Response Filters (FIR)</td>
<td>50</td>
</tr>
<tr>
<td>11.</td>
<td>Design of Infinite Impulse Response Filters (IIR)</td>
<td>58</td>
</tr>
</tbody>
</table>
FREQUENCY RESPONSE OF A BJT AMPLIFIER WITH AND WITHOUT FEEDBACK AMPLIFIER

AIM
1. To design and construct a current series feedback amplifier for a gain of 30 dB in the Audio frequency range and to measure its frequency response (assume the stability factor between 3 and 10)
2. To measure the effect of negative feedback on the frequency response.

APPARATUS REQUIRED
Transistor (BC107), Resistor, Capacitor, AFO, CRO and RPS

THEORY
The circuit diagram of CE Amplifier with current series feedback is shown below. The resistor $R_F$ in emitter is the feedback element. The voltage drop $V_f$ across $R_F$ constitutes the feedback signal while the current $I_c$ forms the sampled signal. Hence, this forms a current series feedback. Due to negative feedback, though the voltage gain of the amplifier is decreased, it improves stability and increases the bandwidth. This is the advantage of negative feedback. Using h-parameter model for ac analysis the amplifier parameters such as the voltage gain, bandwidth can be calculated. For this, following steps have to be followed.

i) To find the input circuit, set $I_0=0$, ie open the output loop. Hence $R_E$ appears in input side.

ii) To find the output circuit set $I_1=0$, i.e. open the input loop. Hence $R_E$ appears in output loop.
CIRCUIT DIAGRAM OF CURRENT-SERIES FEEDBACK AMPLIFIER

MODEL GRAPH
**PRELAB QUESTIONS:**

1. What is an amplifier?
2. What do you understand by feedback in amplifiers?
3. Explain the terms feedback factor and open loop gain?
4. What are the types of feedback?
5. Explain the basic concept of feedback?
6. Compare the negative feedback and positive feedback?
7. Explain the stability of feedback amplifier?
8. Define Nyquist criterion?
9. Define gain and phase margin?

**DESIGN**

**GIVEN SPECIFICATION:**

\[ V_{CC} = 12 \text{ V}, \ hfe = 150, \ hie = 1.2 \text{ K} \Omega, \ R_S = 800 \Omega, \ R_L = 1 \text{ K} \Omega, \ \text{Gain} \ A_V = 30 \text{ dB}, \ R_f = 100K \]

\[ A_V = \frac{31.63}{C_C + C_e} = 0.1 \mu f \]

\[ = - \frac{h_r \text{ Reff}}{(R_S + h_{ie})} \]

\[ = R_{eff} = 421.6 \Omega \]

\[ = R_L = R_C \]

\[ R_f = 730 \Omega \]

\[ V_{CEQ} = 50\% \ of \ V_{CC} = 6 \text{ V} \]

\[ V_E = 10\% \ of \ V_{CC} = 1.2 \text{ V} \]

Now, \[ V_{CC} = I_{CQ} \cdot R_C + V_{CEQ} + V_{EQ} \]

\[ I_{CQ} = 6.5 \text{ mA} \]

\[ R_E = \frac{V_E}{I_{CQ}} = 183 \Omega \]

\[ R_E = 183 \Omega \]

Now, \[ V_B = \frac{R_2 \cdot V_{CC}}{R_1 + R_2} = 1.9 \text{ V} \]

\[ R_2 = 0.16 \times R_1 + 0.16 \times R_2 \]
\[ R_2 = 0.19 \, R_1 \]

Also, the stability factor is given by 
\[ S = 1 + \left( \frac{R_B}{R_E} \right) \]

\[ = 1 + \left( \frac{R_1 \parallel R_2}{R_E} \right) \]

Assume \( S = 5 \)

\[ R_1 \parallel R_2 = 740 \, \Omega \]

\[ R_1 \left( \frac{0.19}{1+0.19} \right) = 740 \, \Omega \]

\[
\begin{array}{|c|c|}
\hline
R_1 & 4.6 \, K\Omega \\
R_2 & 865 \, \Omega \\
\hline
\end{array}
\]

**PROCEDURE**

**Frequency Response of Current- Series Feedback Amplifier**

1. Connect the circuit as shown in the figure.
2. Connect a sine- wave generator set at 1000Hz frequency and 50mV peak-to-peak signal voltage at the input of the amplifier circuit.
3. Connect an oscilloscope across the output nodes. Observe the sine wave output on the oscilloscope. Adjust the output of the sine-wave generator until undistorted. Maximum signal output is obtained.
4. Observe and measure the peak-to-peak amplitude of input and output signal and record the values in the tabulation provided.
5. Now, sweep the input signal frequency in the range 30Hz to 1 MHz by adjusting the sine wave generator output.
6. For each setting of input frequency, measure the output signal voltage.
7. Draw the frequency response curve on a semi-log graph sheet. From this plot, obtain the values of mid-band voltage gain, upper and lower cut-off frequency and BW \( (f_h-f_l) \).

**Frequency Response of Amplifier without Negative Feedback**

8. Remove \( R_f \) from the circuit and connect \( R_E \) and \( C_E \) directly to the emitter terminal.
9. Measure and record in the table, the frequency response of this circuit without \( R_f \) by repeating steps 5 through 6.
10. Draw the response curve on the same graph as before. Obtain the values of mid-band voltage gain, lower and upper cut-off frequency and BW. Comment on the differences between this response curve and the previous curve.

**TABULATION 1**

Measurement of frequency response of current series feedback amplifiers

\[ V_{in} = 50 \text{ mV} \]

<table>
<thead>
<tr>
<th>Frequency (in Hz)</th>
<th>( V_0 ) (Volts)</th>
<th>Gain = ( V_0/V_{in} )</th>
<th>Gain (dB) = ( 20 \log(V_o/V_{in}) )</th>
</tr>
</thead>
<tbody>
<tr>
<td>30</td>
<td></td>
<td></td>
<td></td>
</tr>
<tr>
<td>50</td>
<td></td>
<td></td>
<td></td>
</tr>
<tr>
<td>100</td>
<td></td>
<td></td>
<td></td>
</tr>
<tr>
<td>200</td>
<td></td>
<td></td>
<td></td>
</tr>
<tr>
<td>500</td>
<td></td>
<td></td>
<td></td>
</tr>
<tr>
<td>1K</td>
<td></td>
<td></td>
<td></td>
</tr>
<tr>
<td>2K</td>
<td></td>
<td></td>
<td></td>
</tr>
<tr>
<td>5K</td>
<td></td>
<td></td>
<td></td>
</tr>
<tr>
<td>10K</td>
<td></td>
<td></td>
<td></td>
</tr>
<tr>
<td>20K</td>
<td></td>
<td></td>
<td></td>
</tr>
<tr>
<td>50K</td>
<td></td>
<td></td>
<td></td>
</tr>
<tr>
<td>100K</td>
<td></td>
<td></td>
<td></td>
</tr>
<tr>
<td>200K</td>
<td></td>
<td></td>
<td></td>
</tr>
<tr>
<td>300K</td>
<td></td>
<td></td>
<td></td>
</tr>
<tr>
<td>500K</td>
<td></td>
<td></td>
<td></td>
</tr>
<tr>
<td>1M</td>
<td></td>
<td></td>
<td></td>
</tr>
<tr>
<td>2M</td>
<td></td>
<td></td>
<td></td>
</tr>
</tbody>
</table>
**TABULATION 2**

Measurement of frequency response of amplifier without feedback

\[ V_{in} = 50 \text{ mV} \]

<table>
<thead>
<tr>
<th>Frequency (in Hz)</th>
<th>( V_0 ) (Volts)</th>
<th>( \text{Gain} = \frac{V_0}{V_{in}} )</th>
<th>( \text{Gain dB} = 20 \log\left(\frac{V_0}{V_{in}}\right) )</th>
</tr>
</thead>
<tbody>
<tr>
<td>30</td>
<td></td>
<td></td>
<td></td>
</tr>
<tr>
<td>50</td>
<td></td>
<td></td>
<td></td>
</tr>
<tr>
<td>100</td>
<td></td>
<td></td>
<td></td>
</tr>
<tr>
<td>200</td>
<td></td>
<td></td>
<td></td>
</tr>
<tr>
<td>500</td>
<td></td>
<td></td>
<td></td>
</tr>
<tr>
<td>1K</td>
<td></td>
<td></td>
<td></td>
</tr>
<tr>
<td>2K</td>
<td></td>
<td></td>
<td></td>
</tr>
<tr>
<td>5K</td>
<td></td>
<td></td>
<td></td>
</tr>
<tr>
<td>10K</td>
<td></td>
<td></td>
<td></td>
</tr>
<tr>
<td>20K</td>
<td></td>
<td></td>
<td></td>
</tr>
<tr>
<td>50K</td>
<td></td>
<td></td>
<td></td>
</tr>
<tr>
<td>100K</td>
<td></td>
<td></td>
<td></td>
</tr>
<tr>
<td>200K</td>
<td></td>
<td></td>
<td></td>
</tr>
<tr>
<td>300K</td>
<td></td>
<td></td>
<td></td>
</tr>
<tr>
<td>500K</td>
<td></td>
<td></td>
<td></td>
</tr>
<tr>
<td>1M</td>
<td></td>
<td></td>
<td></td>
</tr>
<tr>
<td>2M</td>
<td></td>
<td></td>
<td></td>
</tr>
</tbody>
</table>

**POSTLAB QUESTIONS:**

**RESULT**
1. The current series feedback amplifier was designed, constructed and its frequency response was plotted.

2. The following parameters were observed.

<table>
<thead>
<tr>
<th>Frequency response data</th>
<th>Current series feedback amplifier</th>
<th>Amplifier without feedback</th>
</tr>
</thead>
<tbody>
<tr>
<td>Mid- band Voltage gain</td>
<td></td>
<td></td>
</tr>
<tr>
<td>Bandwidth</td>
<td></td>
<td></td>
</tr>
</tbody>
</table>
SINGLE TUNED AMPLIFIER

AIM

To design and construct a single tuned amplifier to amplify a 5 KHz signal and to plot the frequency response.

APPARATUS REQUIRED

1. Resistors - 1KΩ, 22 KΩ, 100 KΩ (1 each)
2. Capacitor - 0.1µF (3)
3. Inductors - 10mH
4. Transistor - BC147
5. AFO (0 – 1MHz)
6. RPS (0 – 30V)
7. CRO

THEORY

Sometimes it is desired that an amplifier should select a desired frequency or a narrow band of frequencies and amplify it to desired levels.

In order to pick up and amplify the desired radio frequency signal, the resistive load in the audio amplifier is replaced by a tuned circuit (Parallel resonant circuit). The tuned circuit is capable of selecting a particular frequency and rejecting others.

The circuit shown is a capacitively coupled tuned amplifier. The values of the capacitance (c) and inductance (L) of the tuned circuit are selected in such a way that the resonant frequency of the tuned circuit is equal to the frequency to be selected and amplified.
CIRCUIT DIAGRAM OF SINGLE TUNED AMPLIFIER

MODEL GRAPH

Gain (dB) vs. Frequency (Hz)

Band Width = \( f_u - f_l \)

Quality factor = \( \frac{f_0}{Bandwidth} \)
Resistors $R_1$, $R_2$, $R_E$ are biasing resistors used to provide the DC operating currents voltages for the transistor.

Formula

$$f_0 = \frac{1}{(2\pi \sqrt{LC})}$$

Quality factor $= \frac{f_0}{BW}$
Bandwidth $= f_H - f_L$

$f_0$ – Resonant frequency
$f_L$ – Lower cutoff frequency
$f_H$ – Upper cutoff frequency

**DESIGN**

**Tank Circuit**

$$f_0 = \frac{1}{(2\pi \sqrt{LC})}$$

Given $f_0 = 5$ KHz, Assume $C = 0.1\mu F$

$$L = 10.14 \text{ mH}$$

**Amplifier Design**

Transistor BC 147

$h_{fe\,\text{min}} = 200$, $I_C = 1mA$

Assume $V_{CC} = 10V$

**Selection of $R_E$, $R_2$ and $R_1$**

$$V_{RE} = \frac{1}{10} \times V_{CC} = \frac{1}{10} \times 10 = 1V$$

$$I_E = I_C = 1mA$$

$$R_E = \frac{V_{RE}}{I_E} = \frac{1V}{1mA} = 1K\Omega$$

Select $R_E = 1K\Omega$

$$R_2 = h_{fe\,\text{min}} \frac{R_E}{10} = 20K\Omega$$

Select $R_2 = 22K\Omega$
\[ V_{R1} = V_{CC} - V_{R2} \]
\[ V_{R1} = V_{CC} - (V_{BE} - V_{RL}) \]
\[ V_{R1} = 10 - (0.6 + 1) = 8.4V \]
\[ \frac{V_{R1}}{V_{R2}} = \frac{R_1}{R_2} \]
\[ R_1 = \frac{(V_{R1} \times R_2)}{V_{R2}} = 8.4 \times 20K/1.6 = 105K\Omega \]

Select

\[ R_1 = 100K\Omega \]

**Coupling capacitor \( C_C = 0.1\mu F \)**

**Bypass capacitor \( C_E \)**

\[ X_{CE} = \frac{1}{10} \times R_E = 100\Omega \]
\[ X_{CE} = \frac{1}{2\pi f C_E} \]

Let \( f = 20Hz \) (lowest frequency of input signal)

\[ C_E = 79.6\mu F \]

**PROCEDURE**

1. Connect the circuit as shown in the figure.
2. Connect a sine-wave generator set at 1000Hz frequency and 50mV (p-p) signal voltage at the input of the amplifier circuit.
3. Connect an oscilloscope across the output nodes. Observe the sine wave output on the oscilloscope. Adjust the output of the sine-wave generator until undistorted. Maximum signal output is obtained.
4. Observe and measure the peak-to-peak amplitude of input and output signal and record the values in the tabulation provided.
5. Now, sweep the input signal frequency in the range 30HZ to 1 MHZ by adjusting the sine wave generator output.
6. For each setting of input frequency, measure and record the output signal voltage.
7. Draw the frequency response curve on a semi-log graph sheet. From this plot, obtain the values of resonant frequency, upper and lower cut-off frequency and BW.
### TABULATION

$$V_{in} = 50 \text{ mV}$$

<table>
<thead>
<tr>
<th>Frequency (in Hz)</th>
<th>$$V_0$$ (Volts)</th>
<th>Gain $$= \frac{V_o}{V_{in}}$$</th>
<th>Gain dB = 20 log($$\frac{V_o}{V_{in}}$$)</th>
</tr>
</thead>
<tbody>
<tr>
<td>30</td>
<td></td>
<td></td>
<td></td>
</tr>
<tr>
<td>50</td>
<td></td>
<td></td>
<td></td>
</tr>
<tr>
<td>100</td>
<td></td>
<td></td>
<td></td>
</tr>
<tr>
<td>200</td>
<td></td>
<td></td>
<td></td>
</tr>
<tr>
<td>500</td>
<td></td>
<td></td>
<td></td>
</tr>
<tr>
<td>1K</td>
<td></td>
<td></td>
<td></td>
</tr>
<tr>
<td>2K</td>
<td></td>
<td></td>
<td></td>
</tr>
<tr>
<td>5K</td>
<td></td>
<td></td>
<td></td>
</tr>
<tr>
<td>10K</td>
<td></td>
<td></td>
<td></td>
</tr>
<tr>
<td>20K</td>
<td></td>
<td></td>
<td></td>
</tr>
<tr>
<td>50K</td>
<td></td>
<td></td>
<td></td>
</tr>
<tr>
<td>100K</td>
<td></td>
<td></td>
<td></td>
</tr>
<tr>
<td>200K</td>
<td></td>
<td></td>
<td></td>
</tr>
<tr>
<td>300K</td>
<td></td>
<td></td>
<td></td>
</tr>
<tr>
<td>500K</td>
<td></td>
<td></td>
<td></td>
</tr>
<tr>
<td>1M</td>
<td></td>
<td></td>
<td></td>
</tr>
<tr>
<td>2M</td>
<td></td>
<td></td>
<td></td>
</tr>
</tbody>
</table>

### RESULT

Single tuned amplifier is designed and constructed and its frequency response is plotted.

The tuned frequency was found to be

- Theoretical ($$f_t$$) :
- Practical ($$f_p$$) :

Quality factor (Q) ($$f_o$$/BW) :
Review Questions:

1. Define tuned amplifier? What are the various types of tuned amplifier?
2. What are small signal tuned amplifiers?
3. What are the types of single tuned amplifier?
4. Discuss the effect of cascading tuned amplifiers on bandwidth?
5. Derive the equation for the 3dB bandwidth of capacitance coupled single tuned amplifier?
6. What are doubled tuned amplifiers?
COLPITTS OSCILLATOR

AIM
To design, construct and test a Colpitts oscillator to generate a sine wave of frequency $f_0 = 100$ KHz.

APPARATUS REQUIRED
1. Resistors: 470Ω, 2.2KΩ, 10 KΩ, 47KΩ (1 each)
2. Capacitors: 50.7nF (2), 0.1µF (2)
3. Inductor: 0.1 mH
4. Transistor: BC 107 (1)
5. R.P.S.: (0 – 30V)
6. C.R.O

THEORY
The Colpitt’s oscillator uses tapped capacitance. The tank circuit is made up of two capacitors $C_1$ and $C_2$ connected in series with each other across a fixed inductance $L$.

The resistors $R_1$, $R_2$ and $R_E$ are used to provide D.C. bias for the transistor. The feedback between the output and input circuit is accomplished by the voltage developed across the capacitor $C_2$.

When the circuit is energized by switching on the supply, the capacitors $C_1$ and $C_2$ are charged. These capacitors discharge through the coil $L$, which sets up the oscillations of frequency

$$f_0 = \frac{1}{(2\pi\sqrt{LC})}$$

Where $C = \frac{C_1C_2}{(C_1+C_2)}$

The oscillations across the capacitor $C_2$ are feedback to the base emitter junction and appear in an amplified form at the collector. Because of the positive feedback, oscillations of constant amplitude are produced.
CIRCUIT DIAGRAM OF COLPITTS OSCILLATOR

MODEL GRAPH
DESIGN EQUATIONS

\[ f_0 = \frac{1}{2\pi\sqrt{LC}} \]
\[ C = \frac{C_1C_2}{C_1+C_2} \]

DESIGN

**Feedback Network Design**

Given \( f_0 = 100 \text{ KHz} \)

Oscillation frequency \( f_0 = \frac{1}{2\pi\sqrt{LC}} \)

Assume

\[
\begin{align*}
L &= 0.1\text{mH} \\
C &= 25.3\text{ nF}
\end{align*}
\]

\[ C = \frac{C_1C_2}{C_1+C_2} \]

Let \( C_1 = C_2 \)

\[
\begin{align*}
C_1 = C_2 &= 2C = 50.7\text{ nF}
\end{align*}
\]

**Amplifier Design**

Given: \( V_{CC} = 10\text{V}; I_C = 2\text{ mA} \), \( \beta = 200 \)

\( \beta \) – DC Current gain

\[
\begin{align*}
V_{CE} &= \frac{V_{CC}}{2} = 5\text{V} \\
V_{RE} &= \frac{1}{10} \times V_{CC} = 1\text{V} \\
R_E &= \frac{V_{RE}}{I_C} = 500\ \Omega
\end{align*}
\]

\[
\begin{align*}
R_E &= 470\ \Omega \\
V_{CC} &= V_{RE} + V_{CE} + I_C R_C
\end{align*}
\]

\[
\begin{align*}
R_C &= (V_{CC} - V_{CE} - V_{RE})/I_C \\
Select \quad R_C &= 2.2\ \text{K}\Omega
\end{align*}
\]

\[
\begin{align*}
VR_2 &= V_{BE} + V_{RE} = 0.7 + 1 = 1.7\ \text{V} \\
R_2 &= \beta R_E/10 = 10\ \text{K}\Omega
\end{align*}
\]

\[
\begin{align*}
R_2 &= 10\ \text{K}\Omega
\end{align*}
\]
Select

\[ V_{R1} + V_{R2} = V_{CC} \]

\[ V_{R1} = V_{CC} - V_{R2} = 8.3 \text{ V} \]

We have,

\[ \frac{V_{R1}}{V_{R2}} = \frac{R_1}{R_2} \]

\[ R_1 = (\frac{V_{R1}}{V_{R2}}) \times R_2 = 48 \text{ K} \Omega \]

Select

\[ R_1 = 47 \text{ K} \Omega \]

Let

\[ C_E = 0.1 \mu \text{F} \]

PROCEDURE

1. The circuit connections are made as shown in the circuit diagram.
2. Setting \( V_{CC} = 10 \text{V} \), the amplitude and frequency of the output waveform is noted.
3. A graph of the output waveform is drawn.

OBSERVATIONS

Peak to Peak output Amplitude (\( V_{P-P} \)) =

Frequency = \( 1/ \text{time period} \) =

RESULT

Thus a Colpitts oscillator is designed, constructed and tested.

Theoretical frequency \( f_{TH} \) =

Practical frequency \( f_P \) =
Review Questions:

1. What is an oscillator? What are the types of oscillator?
2. Explain the main difference between an amplifier and an oscillator?
3. What are the constituent parts of an oscillator?
4. State and briefly explain Barkhausen criterion for oscillation?
5. What is the frequency of oscillation for Colpitt’s oscillator?
CLASS-AB COMPLIMENTARY SYMMETRY PUSH PULL AMPLIFIER

AIM:

To design and construct a class-AB tuned amplifier and a frequency multiplier.

APPARATUS REQUIRED:

SN 3055
Resistors
Capacitors
AFO
RPS
CRO

THEORY:

An electronic circuit in which two transistors (or vacuum tubes) are used, one as a source of current and one as a sink, to amplify a signal. One device “pushes” current out into the load, while the other “pulls” current from it when necessary. A common example is the complementary-symmetry push-pull output stage widely used to drive loudspeakers, where an NPN transistor can source (push) current from a positive power supply into the load, or a PnP transistor can sink (pull) it into the negative power supply. The circuit functions as an amplifier in that the current levels at the output are larger than those at the input.

A so-called bias network in a complementary-symmetry push-pull output stage functions to maintain a constant voltage difference between the bases of the two transistors. It can be designed either by setting a bias current and diode sizes or by replacing it with a different network for class B, class A, or the common compromise, class AB mode of operation. In class B operation, where the bases of the transistors might simply be shorted together, only one transistor is “on” at a time and each is on average “on” for only 50% of the time; when the output current is zero, no current at all flows in the circuit. In class A operation a large voltage is maintained between the bases so that both devices stay “on” at all times, although their currents vary so that the difference flows into the load; and even when the output is zero, a large quiescent current flows from the power supplies. Class B operation is much more efficient than class A, which wastes a large amount of power when the signal is small. However, class B suffers from zero-crossing distortion as the output current passes through zero, because there is generally a delay involved as the input swings far enough to turn one transistor entirely off and then turn the other on.
In class AB operation, some intermediate quiescent current is chosen to compromise between power and distortion. Class AB amplifiers are conventionally used as loudspeaker drivers.
in audio systems because they are efficient enough to be able to drive the required maximum output power, often on the order of 100 W, without dissipating excessive heat, but can be biased to have acceptable distortion. Audio signals tend to be near zero most of the time, so good performance near zero output current is critical, and that is where class A amplifiers waste power and class B amplifiers suffer zero-crossing distortion. A class AB push-pull amplifier is also conventionally used as the output stage of a commercial operational amplifier.

PROCEDURE:

1. Give circuit connection as shown in the circuit diagram.
2. Check the dc conditions. (if necessary)
3. Switch on the AFO after setting the input signal amplitude at $|V_i|=1$ volts as peak value.
4. Measure the output signal using CRO. Note the amplitude and frequency of it.
5. Sketch the output wave in a graph sheet.

RESULT:

Thus a class-AB amplifier was designed and constructed and tested for 10KHZ signal.
Review Questions:

1. Write the classifications of amplifier based on biasing condition?
2. What is a push pull amplifier?
3. What are the disadvantages of a push pull amplifier?
4. Define complementary symmetry push pull amplifier?
5. Define class-AB amplifier?
6. What are the applications of class-AB amplifier?
RC PHASE SHIFT OSCILLATOR

AIM

To design and construct the RC phase shift Oscillator to generate a sine wave of frequency 1.5 KHz.

COMPONENTS REQUIRED

<table>
<thead>
<tr>
<th>S.No</th>
<th>Components</th>
<th>Range</th>
<th>Type</th>
<th>Quantity</th>
</tr>
</thead>
<tbody>
<tr>
<td>1.</td>
<td>Transistor</td>
<td>BC 147</td>
<td></td>
<td>1</td>
</tr>
<tr>
<td>2.</td>
<td>Resistor</td>
<td>3.3KΩ</td>
<td>470Ω</td>
<td>4</td>
</tr>
<tr>
<td>3.</td>
<td>Capacitor</td>
<td>0.01µF</td>
<td></td>
<td>4.</td>
</tr>
<tr>
<td></td>
<td>CRO</td>
<td></td>
<td></td>
<td>1</td>
</tr>
</tbody>
</table>

THEORY

Definition

An Oscillator is an amplifier, which uses positive feedback and without any external input signal, generates an output waveform by energizing the DC signal. An Oscillator is a source of AC voltage. Oscillations are produced in the circuit when Barkhausen Criterion is satisfied.

Barkhausen Criterion

(1) The total phase shift in the closed loop is 0 or 360°.
(2) The magnitude of the loop gain of the amplifier (A) and the feedback factor β is unity. 

\[ A\beta = 1 \]

The frequency of Oscillation is determined by the frequency selective feedback network (R-C, L-C). In the RC phase shift Oscillator, the cascaded RC networks determine the frequency. It is an Audio frequency Oscillator capable of
CIRCUIT DIAGRAM OF RC PHASE SHIFT OSCILLATOR

MODEL GRAPH
Generating signals from 15Hz to 20 KHz. The frequency of Oscillations can be varied over a wide range by ganged tuning the capacitor.

The frequency of oscillation is given by the relation

\[
f_0 = f_{\text{TH}} = \frac{1}{2\pi RC \sqrt{(6+(4RC/R))}}
\]

Where \( R \) = value of resistor in the phase shift network

\( C \) = value of capacitor in the phase shift network

For the loop gain to be greater than unity, the current gain of the transistor,

\[
h_{\text{fe min}} > 23 + 29 \left( \frac{R}{R_C} \right) + 4 \left( \frac{R_C}{R} \right)
\]

**DESIGN**

Given \( f = 1.5 \) KHz

If \( R = R_C \), \( h_{\text{fe min}} > 56 \)

BC 147 with \( h_{\text{fe}} = 200 \) is suitable.

**Feedback circuit design**

Since \( R = R_C \)

\[
F = \frac{1}{2\pi RC \sqrt{10}}
\]

Assume \( C = 0.01 \mu F \)

1.5 KHz = \( 1/(2\pi x R x 0.01 \mu F \sqrt{10}) \)

\[
R = 3.3 K\Omega = R_C
\]

**Biasing circuit design**

For BC147 \( \beta = 200 \)

Assume \( V_{CC} = 10V \)

\( I_C = 2mA \)

In the active region,

\[
V_{CE} = \frac{1}{2} V_{CC} = \frac{1}{2} \times 10 = 5V
\]

\[
I_B = I_C/\beta = 10 \mu A
\]

\( \beta \) - Feedback factor

\[
V_{CE} = I_B R_B + V_{BE}
\]

\[
R_B = (V_{CE} - V_{BE})/I_B = (5-0.6)/10\mu = 440 K\Omega
\]
Select $R_B = 470 \, \text{K\Omega}$

**PROCEDURE**

1. Connections are made as shown in the circuit diagram
2. The DC power supply is switched ON
3. The output waveform is displayed on the CRO.
4. The peak to peak Amplitude and time period of the sine wave is noted.
5. The graph of output waveform is drawn

**OBSERVATION**

\[
\text{Time period } T = \]
\[
\text{Amplitude } V_{P-P} =
\]
\[
f_0 = 1/T =
\]

**RESULT**

Thus an RC phase shift oscillator is designed, constructed and tested.

Frequency of oscillation

Theoretical $f_T = $

Practical $f_P = $
Review Questions:

1. What is an oscillator? List its types?
2. What are RC oscillators? What are its types?
3. What is the phase shift given by each RC section?
1. Introduction

1.1 WHAT IS MATLAB?

- It stands for MATrix LABoratory
- It is developed by The Mathworks, Inc. (http://www.mathworks.com)
- It is an interactive, integrated, environment
  - for numerical computations
  - for symbolic computations
  - for scientific visualizations
- It is a high-level programming language
- Program runs in interpreted, as opposed to compiled, mode

1.2 Characteristics of MATLAB

- Programming language based (principally) on matrices.
  - Slow (compared with fortran or C) because it is an interpreted language, *i.e.* not pre-compiled. Avoid for loops; instead use vector form (see section on vector technique below) whenever possible.
  - Automatic memory management, i.e., you don't have to declare arrays in advance.
  - Intuitive, easy to use.
  - Compact (array handling is fortran90-like).
  - Shorter program development time than traditional programming languages such as Fortran and C.
  - Can be converted into C code via MATLAB compiler for better efficiency.
- Many application-specific toolboxes available.
- Coupled with Maple for symbolic computations.
- On shared-memory parallel computers such as the SGI Origin2000, certain operations processed in parallel autonomously -- when computation load warrants

1.3 Learning Matlab

Matlab is one of the fastest and most enjoyable ways to solve problems numerically. The computational problems arising in most undergraduate courses can be solved much more quickly with Matlab, than with the standard programming languages (Fortran, C, Java, etc.). It is particularly easy to generate some results, draw graphs to look at the interesting features, and then explore the problem further.

This introduction gives a quick way to become familiar with the most important parts of Matlab. The first five sections emphasize simple arithmetic, matrix-vector operations (including solving systems of equations), and graphing functions and data.

The best way to used this introduction is to sit down at a computer and type in the commands as they are described. Look at Matlab's response, and check that the answers are what you expect. It is also a good idea to do the small exercises.
1.4 Further References

More information on Matlab can be found in the books

GENERATION OF DISCRETE WAVEFORMS

AIM:

To write a program in MATLAB to generate time signal waveforms.

1. Unit step function
2. Unit impulse wave
3. Unit ramp function
4. Exponentially decaying function
5. Exponentially growing function
6. Signum function

ALGORITHM:

1. Get the input “n” values.
2. Get the length of ramp or exponential signal.
3. Determine time duration for signal representation.
4. Determine the subplot of output figure.
5. Label X&Y-axes.
6. Stop the execution.

PROGRAM:

```matlab
% GENERATION OF DISCRETE WAVEFORMS
% IMPULSE RESPONSE
clear all;
close all;
clc;
y=zeros(1,11);
for n=0:1:10
    if (n==0)y(n+1)=1;
    else
        y(n+1)=0;
    end
end
m=0:1:10;
subplot(3,3,1),stem(m,y);
title('impulse');

% DELAYED IMPULSE FUNCTION
for n=0:1:10;
y(n+1)=0;
if(n==2)
y(n+1)=1;
end
```
%UNIT STEP FUNCTION
y=ones(1,11);
n=0:1:10;
subplot(3,3,3),stem(n,y);
title('unit step function');

%DELAYED STEP FUNCTION
y=zeros(1,21);
for n=1:1:21;
  if(n<11)
    y(n)=1;
  else
    y(n)=1;
  end
end
m=0:1:20;
subplot(3,3,4),stem(m,y);
title('delayed step function')

%RAMP FUNCTION
n=0:1:10;
x=n;
subplot(3,3,5),stem(n,x);
title('ramp function')

%EXponentially DECAYING FUNCTION
n=0:1:10;
x=0.8.^n;
subplot(3,3,6),stem(n,x);
title('exponentially decaying function');

%EXponentially GROWING FUNCTION
n=0:1:10;
subplot(3,3,7),stem(n,x);
title('exponentially growing function');

%SIGNUM FUNCTION
for n=1:1:21;
  if(n<10)
    y(n)=-1;
  else
  end
Thus the MATLAB program was successfully executed and the output was verified.
GENERATION OF SINUSOIDAL AND TRIANGULAR WAVEFORMS

AIM:

To write a program in MATLAB to generate time signal waveforms.

  i) Sinusoidal waveform
  ii) Triangular waveform

ALGORITHM:

1. Get the input “n” values.
2. Determine time duration for signal representation.
3. Determine the subplot of output figure.
5. Stop the execution.

PROGRAM:

%SINUSOIDAL & TRIANGULAR FUNCTION
n=0:1:10;
x=2*sin(0.2*pi*n);
x1=1:10;
triangle=mod(x1,2);
subplot(4,1,1),stem(n,x);
title('sinusoidal function')
subplot(4,1,2),plot(triangle);
title('triangular function');
RESULT:

Thus the MATLAB program was executed successfully for sinusoidal and triangular waveforms.
GENERATION OF DFT USING MATLAB

AIM:

To compute a discrete Fourier transform of a sequence using MATLAB.

ALGORITHM:

1. Get the input sequences.
2. Find the length of the sequence.
3. Define the computation of DFT.
4. Plot the magnitude and phase response.
5. Display the DFT sequences

THEORY:

THE DISCRETE FOURIER TRANSFORM

The discrete Fourier transform (DFT) is one of the specific forms of Fourier analysis. It transforms one function into another, which is called the frequency domain representation, or simply the DFT, of the original function (which is often a function in the time domain). But the DFT requires an input function that is discrete and whose non-zero values have a limited (finite) duration.

Therefore it is often said that the DFT is a transform for Fourier analysis of finite-domain discrete-time functions. The sinusoidal basis functions of the decomposition have the same properties. Since the input function is a finite sequence of real or complex numbers, the DFT is ideal for processing information stored in computers. In particular, the DFT is widely employed in signal processing and related fields to analyze the frequencies contained in a sampled signal, to solve partial differential equations, and to perform other operations such as convolutions.

The DFT can be computed efficiently in practice using a fast Fourier transform (FFT) algorithm.

Definition

Given a discrete set of real or complex numbers: \( x[n], \ n \in \mathbb{Z} \) (integers), the discrete-time Fourier transform (or DTFT) of \( x[n] \) is usually written

\[
X(\omega) = \sum_{n=-\infty}^{\infty} x[n] e^{-i\omega n}.
\]
**Inverse Fourier Transform**

The following inverse transforms recover the discrete-time sequence:

\[
x[n] = \frac{1}{2\pi} \int_{-\pi}^{\pi} X(\omega) \cdot e^{i\omega n} \, d\omega
\]

\[
= T \int_{-\frac{1}{2T}}^{\frac{1}{2T}} X_T(f) \cdot e^{i2\pi fnT} \, df.
\]

**PROGRAM**

```
%% DFT OF A SEQUENCE
%% TO COMPUTE N POINT DFT OF THE SEQUENCE x(n)

clear all
x1=[1,2,3,4,5]
N=length(x1);
for k=0:1:N-1
    for n=0:1:N-1
        p=exp(-i*2*pi*n*k/N);
        x2(k+1,n+1)=p
    end
end
X=x1*x2;
magX=abs(X)
angleX=angle(X)
k=0:1:N-1
subplot(2,1,1)
stem(k,magX)
xlabel('k')
ylabel('magnitude of X(k)')
title('DFT Magnitude Response')
subplot(2,1,2)
stem(k,angleX)
xlabel('k')
ylabel('argX(k)')
title('DFT Phase Response')
```
RESULT:

Thus the MATLAB program was successfully executed and output was verified.
GENERATION OF LINEAR AND CIRCULAR CONVOLUTION

AIM:

(i) To compute linear convolution of two given sequences using MATLAB program.
   \[ X(n) = \{1, 2\} \]
   \[ Y(n) = \{1, 2, 4\} \]

(ii) To compute circular convolution of two given sequences using MATLAB program.
    \[ X1(n) = \{1, 0, -1, 0, 1, 0\} \]
    \[ X2(n) = \{0, 1, 1, 1, 1\} \]

ALGORITHM:

Linear convolution:
1. Get the two input sequences.
2. Find their length.
3. Compute linear convolution.
4. Plot the input and output sequences.

Circular convolution:
1. Get the two input sequences.
2. Find their length.
3. Compare their length.
4. If they are not equal, equal them by zero padding.
5. Compute circular convolution.

THEORY:

LINEAR AND CIRCULAR CONVOLUTION:

Convolution of Signals

Continuous-Time Signals

The convolution of two signals is defined by

\[ f_1(t) * f_2(t) = \int_{-\infty}^{\infty} f_1(\tau)f_2(t - \tau)d\tau, \quad -\infty < t < \infty \]

Discrete-Time Signals

In the discrete-time domain, the convolution of two signals is defined by
Circular Convolution

An N point circular convolution is defined as follows

\[ f_1[k] \ast f_2[k] = \sum_{m=-\infty}^{\infty} f_1[m] f_2[k - m], \quad -\infty < k < \infty \]

There are two ways to write a function for circular convolution: 1) in the transform domain and 2) in the time domain. The Matlab function will require four inputs: the signal vectors x[n] and h[n], the length of the circular convolution, N, and a variable that indicates which method to use.

Relationship to Linear Convolution

The comparison of circular convolution output to a linear convolution output shows that they are not always the same. Some values may be correct, while others are corrupted because of time aliasing.

PROGRAM FOR CONVOLUTION

```matlab
% program for linear convolution with x=[1 2] and h=[1 2 4]
clc;
clear all;
close all;
x=input('1st sequence');
h=input('2nd sequence');
c=conv(x,h);
t=0:1:5;
subplot(3,1,1);
stem(t,conv(x,h));
xlabel('index');
ylabel('amplitude');
title('Linear Convolution');
t=0:1:5;
subplot(3,1,2);
stem(t,conv(x,h));
xlabel('index');
ylabel('amplitude');
title('Linear Convolution');
```

44
ylabel('amplitude');
title('second sequence');
t=0:1:15;
subplot(3,1,3);
stem(c);
xlabel('index');
ylabel('amplitude');
title('linear conv result');
disp('linear convolution of two sequence');
disp('c');
PROGRAM:

%%% Circular Convolution of two given sequences
%%% GIVEN x(n)=[1,0,-1,0,1,0]
%%% GIVEN h(n)=[0,1,1,1,1]

x=input('the first sequence');
y=input('the second sequence');
a=length(x)
b=length(y)
if a > b
    N=a;
else
    N=b;
end
x1=[x,zeros(1,N-a)];
y1=[y,zeros(1,N-b)];
for k=1:1:N
    for n=1:1:N
        q(n,k)=x1(n);
    end
    for r=2:1:N
        x2(r)=x1(r-1);
    end
    x2(1)=x1(n);
    x1=x2;
end
cc=y1*q;
n=0:1:N-1;
subplot(3,3,1),stem(n,x1);
xlabel('first sequence x(n)');
subplot(3,3,2),stem(n,y1);
xlabel('n'),ylabel('y(n)');
title('second sequence y(n)');
subplot(3,3,3),stem(n,cc);
xlabel('n'),ylabel('cc(n)');
title('circular convolution of two given sequence');
disp(cc);
RESULT:
Thus the MATLAB program was successfully executed and output was verified.
AIM:
To design the following FIR filters using rectangular, hamming, hanning window techniques
i) LPF with cut off frequency $\omega_c = \pi/2$ & $N=11$
ii) HPF with cut off frequency $\omega_c = \pi/4$ & $N=11$

ALGORITHM:
1. Get the passband and stopband attenuation & calculate $A_p$ and $A_s$
2. Get the passband frequencies & stopband frequencies.
3. Calculate $\omega_p$ and $\omega_s$
4. Compute Magnitude and phase response
5. Display the magnitude and phase response

THEORY:
INTRODUCTION TO FILTERS:

Analog and digital filters

In signal processing, the function of a filter is to remove unwanted parts of the signal, such as random noise, or to extract useful parts of the signal, such as the components lying within a certain frequency range.

The following block diagram illustrates the basic idea.

There are two main kinds of filter, analog and digital. They are quite different in their physical makeup and in how they work.

An analog filter uses analog electronic circuits made up from components such as resistors, capacitors and op amps to produce the required filtering effect. Such filter circuits are widely used in such applications as noise reduction, video signal enhancement, graphic equalisers in hi-fi systems, and many other areas.

The analog input signal must first be sampled and digitised using an ADC (analog to digital converter). The resulting binary numbers, representing successive sampled values of the input signal, are transferred to the processor, which carries out numerical calculations on them. These calculations typically involve multiplying the input values by constants and adding the products together. If necessary, the results of these calculations, which now represent sampled values of the filtered signal, are output through a DAC (digital to analog converter) to convert the signal back to analog form.
The following diagram shows the basic setup of such a system.

![Diagram of analog to digital conversion process]

**FIR FILTERS**

Finite impulse response (FIR) filters are the most popular type of filters implemented in software. This introduction will help you understand them both on a theoretical and a practical level.

A digital filter takes a digital input, gives a digital output, and consists of digital components. In a typical digital filtering application, software running on a digital signal processor (DSP) reads input samples from an A/D converter, performs the mathematical manipulations dictated by theory for the required filter type, and outputs the result via a D/A converter.

An analog filter, by contrast, operates directly on the analog inputs and is built entirely with analog components, such as resistors, capacitors, and inductors.

There are many filter types, but the most common are **lowpass, highpass, bandpass, and bandstop**. A lowpass filter allows only low frequency signals (below some specified cutoff) through to its output, so it can be used to eliminate high frequencies. A lowpass filter is handy, in that regard, for limiting the uppermost range of frequencies in an audio signal; it's the type of filter that a phone line resembles.

A **highpass filter** does just the opposite, by rejecting only frequency components below some threshold. An example highpass application is cutting out the audible 60Hz AC power "hum", which can be picked up as noise accompanying almost any signal in the U.S.
The designer of a cell phone or any other sort of wireless transmitter would typically place an analog **bandpass filter** in its output RF stage, to ensure that only output signals within its narrow, government-authorized range of the frequency spectrum are transmitted.

Engineers can use **bandstop filters**, which pass both low and high frequencies, to block a predefined range of frequencies in the middle.

**Frequency Response**

Simple filters are usually defined by their responses to the individual frequency components that constitute the input signal. There are three different types of responses. A filter's response to different frequencies is characterized as passband, transition band, or stopband. The passband response is the filter's effect on frequency components that are passed through (mostly) unchanged. Frequencies within a filter's stopband are, by contrast, highly attenuated. The transition band represents frequencies in the middle, which may receive some attenuation but are not removed completely from the output signal.

In Figure 1, which shows the frequency response of a lowpass filter, $\omega_p$ is the passband ending frequency, $\omega_s$ is the stopband beginning frequency, and $A_s$ is the amount of attenuation in the stopband. Frequencies between $\omega_p$ and $\omega_s$ fall within the transition band and are attenuated to some lesser degree.

*Figure 1. The response of a lowpass filter to various input frequencies*

Given these individual filter parameters, one of numerous filter design software packages can generate the required signal processing equations and coefficients for implementation on a DSP.
Before we can talk about specific implementations, however, some additional terms need to be introduced.

Ripple is usually specified as a peak-to-peak level in decibels. It describes how little or how much the filter's amplitude varies within a band. Smaller amounts of ripple represent more consistent response and are generally preferable.

Transition bandwidth describes how quickly a filter transitions from a passband to a stopband, or vice versa. The more rapid this transition, the higher the transition bandwidth; and the more difficult the filter is to achieve. Though an almost instantaneous transition to full attenuation is typically desired, real-world filters don't often have such ideal frequency response curves.

There is, however, a tradeoff between ripple and transition bandwidth, so that decreasing either will only serve to increase the other.

**Finite impulse response**

A finite impulse response (FIR) filter is a filter structure that can be used to implement almost any sort of frequency response digitally. An FIR filter is usually implemented by using a series of delays, multipliers, and adders to create the filter's output.

Figure 2 shows the basic block diagram for an FIR filter of length N. The delays result in operating on prior input samples. The $h_k$ values are the coefficients used for multiplication, so that the output at time $n$ is the summation of all the delayed samples multiplied by the appropriate coefficients.

![Figure 2. The logical structure of an FIR filter](image)
PROGRAM:

% Design of low pass digital FIR filter using Windowing technique

clc;
clear all;
close all;
wc=0.5*pi;
N=11;
alpha=(N-1)/2;
eps=0.001;
n=0:1:N-1;
hd=sin(wc*(n-alpha+eps))./(pi*(n-alpha+eps));
wr=boxcar(N);
hn=hd.*wr';
w=0:pi/12:pi;
h=freqz(hn,1,w);
m=20*log10(abs(h));
subplot(3,1,1),plot(w/pi,m);
grid;
title('rectangular window');
ylabel('magnitude');
wh=hamming(N);
hn=hd.*wh';
w=0:pi/12:pi;
h=freqz(hn,1,w);
m=20*log10(abs(h));
subplot(3,1,2);
plot(w/pi,m);
grid;
title('hamming window');
ylabel('magnitude');
wn=hanning(N);
hn=hd.*wn';
w=0:pi/12:pi;
h=freqz(hn,1,w);
m=20*log10(abs(h));
subplot(3,1,3),plot(w/pi,m);
grid;
title('hanning window');
ylabel('magnitude');
xlabel('normalised frequency');
PROGRAM:

% Design of high pass digital FIR filter using Windowing technique

clc;
close all;
clear all;
N=11;
wc=25*pi;
alpha=(N-1)/2;
eps=0.001;
N=0:1:N-1;
hd=(sin(pi*(n-alpha+eps))-sin(wc*(n-alpha+eps)))/(pi*(n-alpha+eps));
wr=boxcar(N);
hk=hd.*wr';
w=0:pi/12:pi;
h=freqz(hk,1,w);
m=20*log10(abs(h));
subplot(3,1,1),plot(w/pi,m);
title('rectangle window');
ylabel('magnitude');
wh=hamming(N);
hn=hd.*wh';
w=0:pi/12:pi;
h=freqz(hn,1,w);
m=20*log10(abs(h));
subplot(3,1,2);
plot(w/pi,m);
grid;
title('hamming window');
ylabel('magnitude');
wn=hanning(N);
hn=hd.*wh';
w=0:pi/12:pi;
h=freqz(hn,1,w);
m=20*log10(abs(h));
subplot(3,1,3),plot(w/pi,m);
grid;
title('hanning window');
ylabel('magnitude');
xlabel('normalized frequency');
RESULT:
Thus the MATLAB program was executed successfully.
AIM:
To design a Butterworth low pass and high pass IIR filters and to simulate it using MATLAB

ALGORITHM:
1. Get the pass band and stop band ripples.
2. Get the pass band frequencies & stop band edge frequencies.
3. Get the sampling frequencies
4. Calculate the order of the filter using specific equations
5. Find the filter coefficients.
6. Draw the magnitude and phase response

THEORY:

INTRODUCTION TO IIR FILTER

Infinite impulse response (IIR) is a property of signal processing systems. Systems with that property are known as IIR systems or when dealing with electronic filter systems as IIR filters. They have an impulse response function which is non-zero over an infinite length of time. This is in contrast to finite impulse response filters (FIR) which have fixed-duration impulse responses. The simplest analog IIR filter is an RC filter made up of a single resistor (R) feeding into a node shared with a single capacitor (C). This filter has an exponential impulse response characterized by an RC time constant.

IIR filters may be implemented as either analog or digital filters. In digital IIR filters, the output feedback is immediately apparent in the equations defining the output. Note that unlike with FIR filters, in designing IIR filters it is necessary to carefully consider "time zero" case in which the outputs of the filter have not yet been clearly defined.

Design of digital IIR filters is heavily dependent on that of their analog counterparts because there are plenty of resources, works and straightforward design methods concerning analog feedback filter design while there are hardly any for digital IIR filters. As a result, mostly, if a digital IIR filter is going to be implemented, first, an analog filter (e.g. Chebyshev filter, Butterworth filter, Elliptic filter) is designed and then it is converted to digital by applying discrimination techniques such as Bilinear transform or Impulse invariance.

Example IIR filters include the Chebyshev filter, Butterworth filter, and the Bessel filter.
PROGRAM:

```matlab
%% Butter worth digital IIR High pass filters
clc;
clear all;
close all;
rp=input('enter the passband ripple');
rs=input('enter the stopband ripple');
wp=input('enter the passband frequency');
ws=input('enter the stopband freq');
fs=input('enter the sampling freq');
w1=2*wp/fs;
w2=2*ws/fs;
[n,wn]=buttord(w1,w2,rp,rs);
[b,a]=butter(n,wn,'high');
w=0:.01:pi;
[h,om]=freqz(b,a,w);
m=20*log10(abs(h));
an=angle(h);
subplot(2,1,2);
plot(om/pi,m);
ylabel('gain in db--->');
xlabel('(a)Normalised frequency--->');
subplot(2,1,2);
plot(om/pi,an);
ylabel('phase in radians--->');
xlabel('(b)Normalised frequency--->');
```
RESULT:

The Butterworth digital IIR high pass filter is stimulated using Matlab and the results are:

Enter the passband ripple  0.5
Enter the stopband ripple  50
Enter the passband frequency  1200
Enter the stopband frequency  2400
Enter the sampling frequency  10000