



INSTITUTE OF AERONAUTICAL ENGINEERING

(Autonomous)

Dundigal - 500 043, Hyderabad, Telangana

COURSE CONTENT

IC APPLICATIONS LABORATORY								
IV Semester: ECE								
Course Code	Category	Hours / Week			Credits	Maximum Marks		
AECC15	Core	L	T	P	C	CIA	SEE	Total
		0	0	2	1	30	70	100
Contact Classes: Nil	Tutorial Classes: Nil	Practical Classes: 45			Total Classes:45			
Prerequisite: Electronic Devices and Circuits Laboratory (AECC05)								

I. COURSE OVERVIEW:

This course enables design and testing of digital and analog integrated circuits. It focuses on design of advanced filters, phase locked loops, multi-vibrators and data converters. Develops the capability to construct and validate the Linear & digital using hardware and simulation tools.

II. COURSES OBJECTIVES:

The students will try to learn

- I. The concepts of waveform generators and filters for transmitter and receiver applications.
- II. The theory and operation of converters including closed-loop feedback control circuits.
- III. The IC based real-time applications in the fields of communication systems and automation systems.

III. COURSE OUTCOMES:

At the end of the course students should be able to:

CO1	Demonstrate the arithmetic, integration and differentiation operations using general purpose operational amplifiers.
CO2	Examine characteristics of active filters for determining gain, bandwidth and frequency response.
CO3	Build a stable and monostable multi vibrators using operational amplifier based timer integrated circuits.
CO4	Characterize phase locked loop circuits for estimating the capture range and lock-in range
CO5	Construct the low and high voltage regulators to find the percentage of regulation using operation amplifier based voltage regulators.
CO6	Implement combinational and sequential circuits using digital integrated circuits.

IV. COURSE CONTENT

EXERCISES FOR IC APPLICATIONS LABORATORY

Note: Students are encouraged to bring their own laptops for laboratory practice session

This laboratory enables design and testing of digital and analog integrated circuits. Initially Op-Amp based Inverting & Non inverting amplifiers, Integrators and Differentiators laboratory exercises are covered. It focuses on design of advanced filters, phase locked loops, multi-vibrators and data converters. Develops the capability to construct and validate the Linear & digital using hardware and multisim simulation tools.

1. INVERTING AND NON-INVERTING OPERATIONAL AMPLIFIERS

Inverting amplifiers are used in a number of applications like phase shifter, integration, signal balancing, mixer circuits, etc. Non-inverting amplifiers are used in circuits where high input impedance is required. They are used as voltage followers, isolation of cascaded circuits, to perform mathematical simulations.

1.1 Design an inverting amplifier circuit with OP AMP 741C for gain of 10.

Plot the waveforms, observe the phase reversal, measure the gain for the circuit shown in Figure.1.1.1.

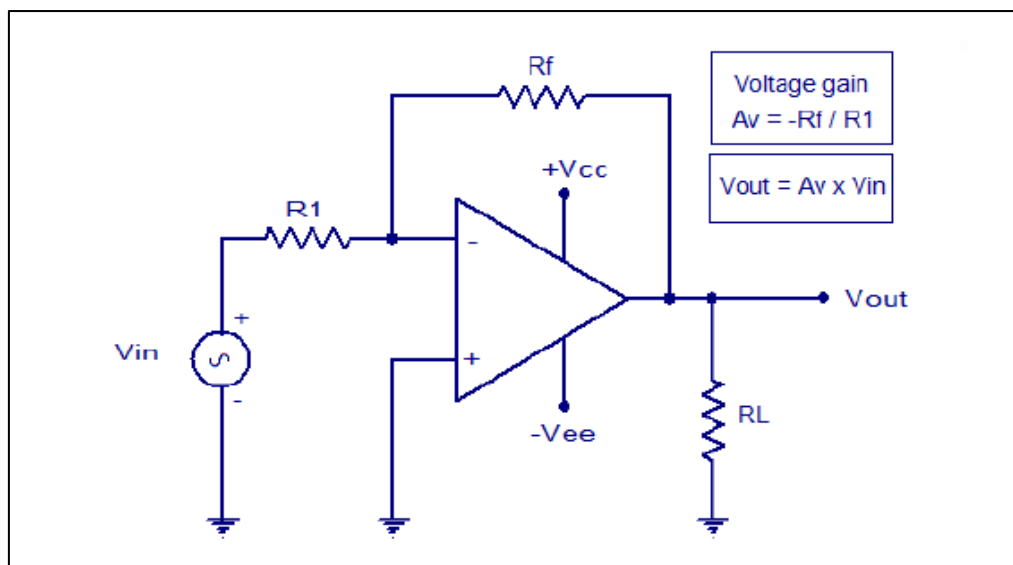


Figure 1.1.1: Inverting Amplifier

1.1 . Design of Non-Inverting amplifier with OP AMP741C for gain of 11.

Plot the waveforms, observe the phase reversal, measure the gain for the circuit shown in Figure.1.2.1.

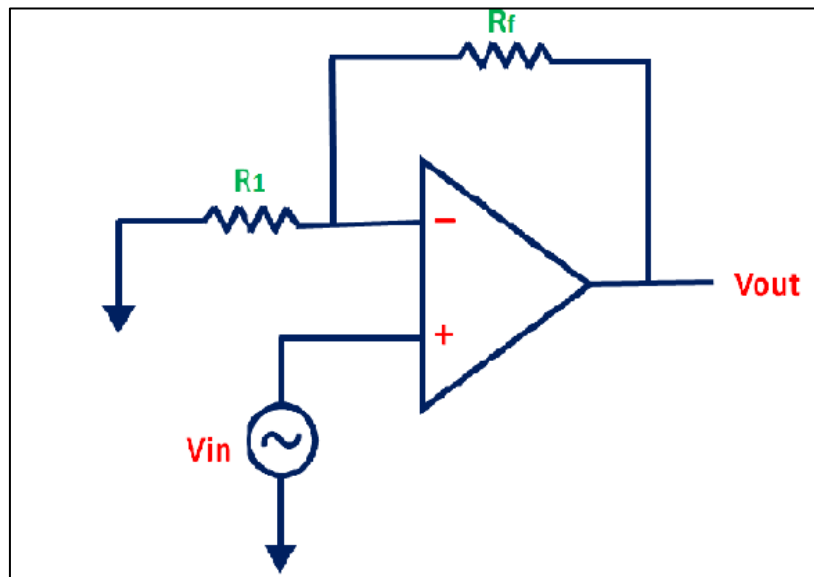


Figure 1.2.1: Inverting Amplifier

1.3. What is the output of the summing amplifier in Figure 1.2.2, with the given DC input voltages? Compare the values obtained from Simulation and Practical values.

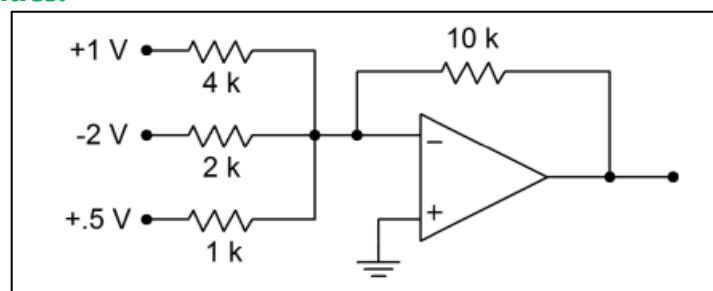


Figure.1.2.2

Hint: The easy way to approach this is to just treat the circuit as three inverting voltage amplifiers, and then add the results to get the final output.

Try

1. Design a simple difference amplifier with an input impedance of 10 k per leg, and a voltage gain of 26 Db.
2. Design Adder/ Subtractor Circuits

Hint: If inverting and non- inverting summing amplifiers are combined using the differential amplifier topology, an adder/subtractor results. Normally, all resistors in an adder/subtractor are the same value.

2. INTEGRATORS

The integrator circuit is mostly used in analog computers, analog-to-digital converters and wave-shaping circuits. A common wave-shaping use is as a charge amplifier and they are

usually constructed using an operational amplifier though they can use high gain discrete transistor configurations.

2.1. Design of integrator circuit for periodic signal with a frequency of 5 kHz

Draw the input and output waveforms for different time constants for the basic integrator circuit shown in Figure.2.1.

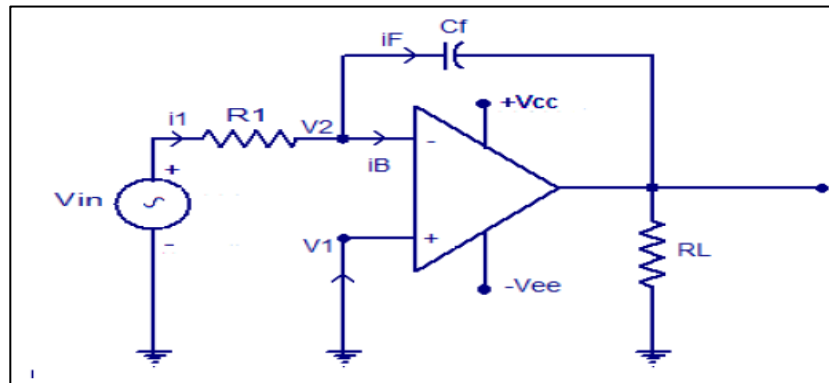


Figure 2.1: Basic Integrator

2.2. Design lossy integrator so that the peak gain is 20dB and the gain is 3dB down from its peak when $\omega = 10,000$ rad/sec. use a capacitance of $0.01\mu\text{F}$ in hardware and multisim.

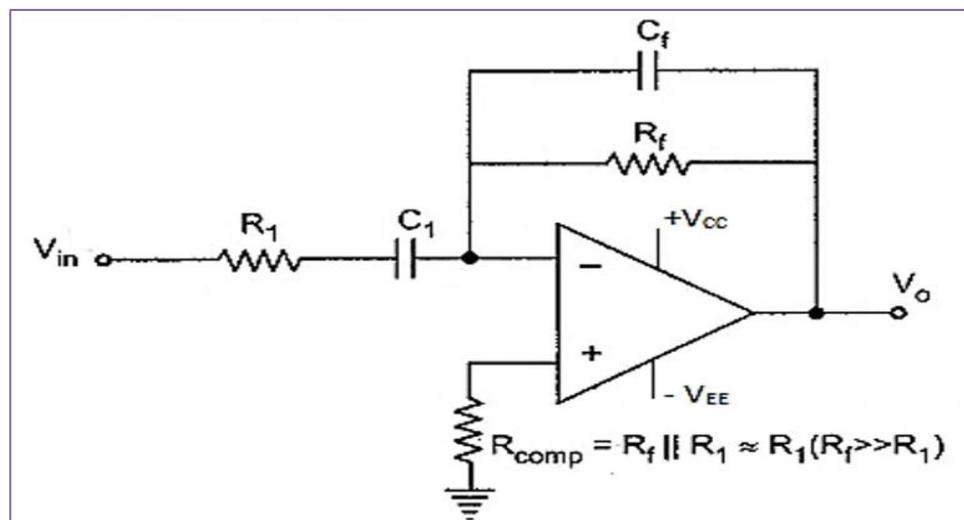


Figure 2.2: Practical Integrator

2.3. Design Summing integrator to give output as per the following equation using Multisim.

$$v_o(t) = -\frac{1}{C_f} \int_0^t \left(\frac{v_1(t)}{R_1} + \frac{v_2(t)}{R_2} + \frac{v_3(t)}{R_3} \right) dt + v_o(0)$$

Try

1. The circuit that integrates the input signal twice is called double integrator. The design of double integrator requires two reactive portions for obtaining double integration. Design double integrator in mutisim /pspice to produce output as per the equation as shown below.

$$v_o(t) = -\frac{4}{(RC)^2} \iint v_i(t) dt$$

2. Elaborate practical integrator circuit and state the advantages of practical integrator over ideal integrator. Using simulation with multisim determine R1 and Rf for a given lossy integrator for a peak gain of 30dB and the gain is 3dB down from its peak when $\omega = 5000$ rad/sec with a capacitance of 0.01micro farads.

3. DIFFERENTIATORS

Differentiating amplifiers are most commonly designed to operate on triangular and rectangular signals. Differentiators also find application as wave shaping circuits, to detect high frequency components in the input signal.

3.1. Design of Differentiator circuit for periodic signal with a frequency of 5 kHz

Draw the input and output waveforms for different time constants for the basic Differentiator circuit shown in Figure.3.1.

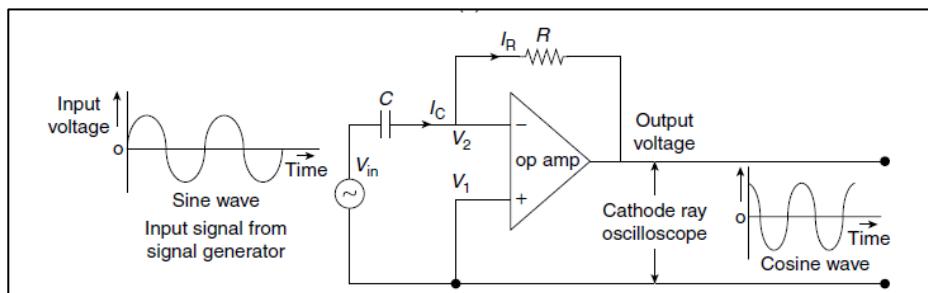


Figure 3.1: Basic Differentiator

3.2. Design a differentiator using op-amp to differentiate an input signal with $f_{max} = 200$ Hz. Also draw the output waveforms for a sine-wave and a square-wave input of 1 V peak at 200 Hz. Compare the values obtained from Simulation and Practical values.

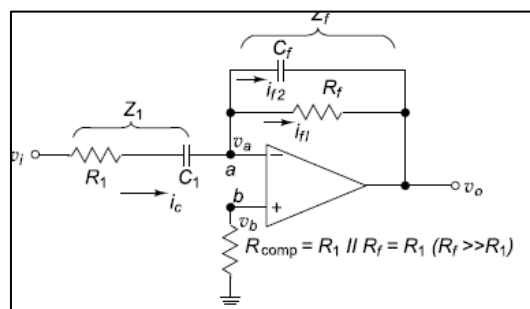


Figure 3.2: Practical Differentiator

3.3. Construct a differentiator for a given differential input signal that varies in frequency from 100 Hz to about 5 KHz with a sine wave of 10V peak at 2k Hz is applied to this differentiator. Also draw the required waveforms.

Try

1. Sketch the output waveform of a differentiator for a sinusoidal signal of 3V peak at 250Hz. Compare the values obtained from Simulation and Practical values.
2. Determine the useful range for differentiation in the circuit of Figure 3.3. Also determine the output voltage if the input signal is a 2 V peak sine wave at 3 kHz.

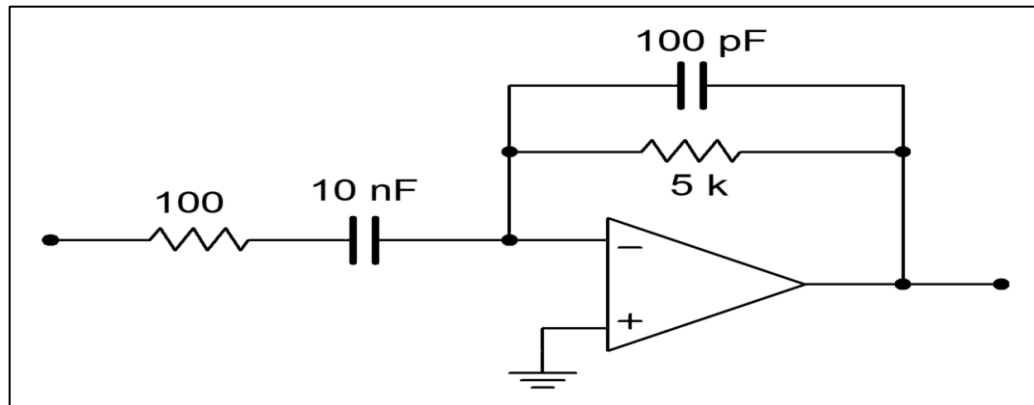


Figure 3.3

3. Given the circuit of Figure 3.4 carryout the simulation and analyse the output waveforms if the input is a 100 Hz, 1 V peak triangle wave.

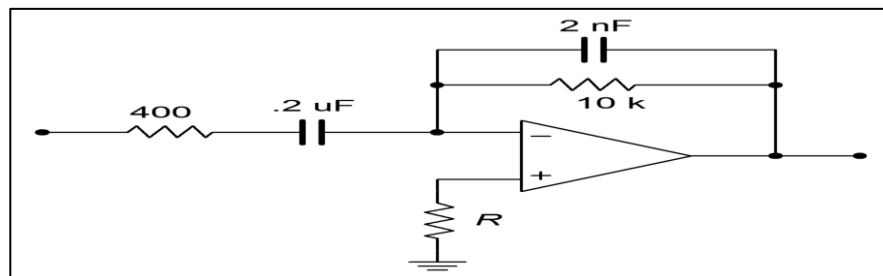


Figure 3.4

4. INSTRUMENTATION AMPLIFIER

The Instrumentation amplifiers are extensively used in biomedical instrumentation, for example, where they amplify signals from devices like ECG and EEG monitors. These amplifiers are also prevalent in telecommunications for extracting weak signals.

4.1. Build and verify the operation of instrumentation amplifier using IC 741 as per Figure 4.1.

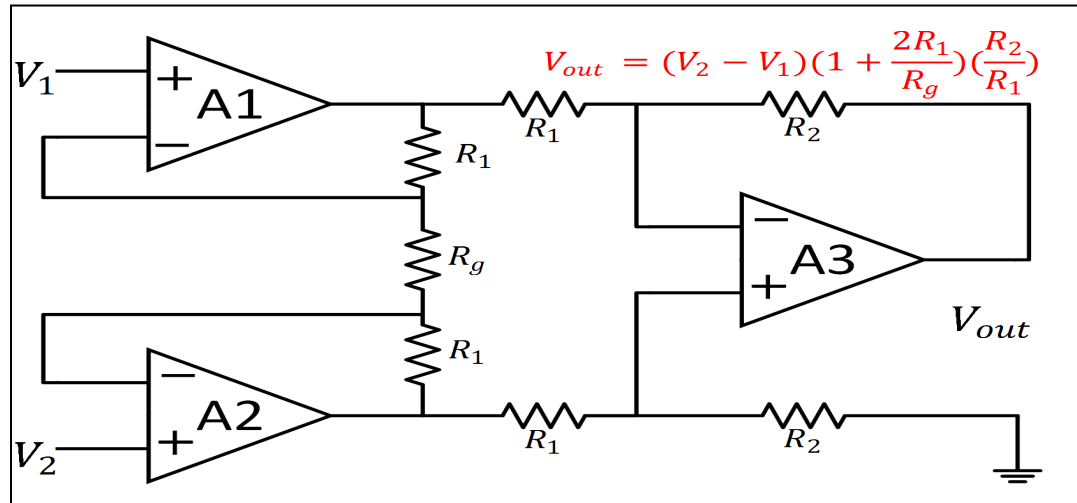


Figure 4.1. Instrumentation Amplifier

4.2 Design a differential instrumentation amplifier using a transducer bridge. Given data $R_1=1k$ ohms, $R_f=4.7 k$ ohms, $R_A = R_B = R_C = 100 k$ ohms, $V_{DC}=5V$.

Try

- Design an instrumentation amplifier with following specifications: $R_1=10K$ ohms, $R_2=5K$ ohms, $R_3=10K$ ohms.

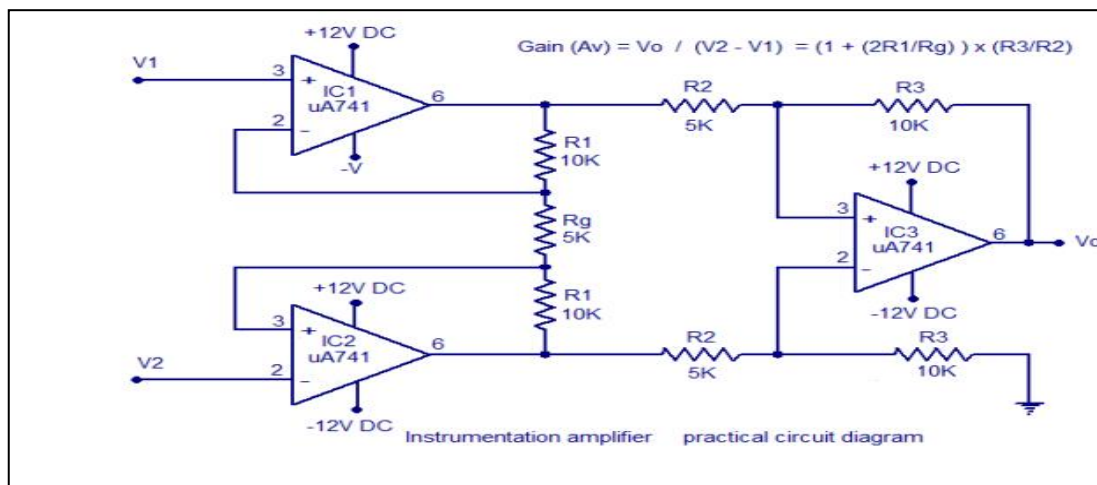
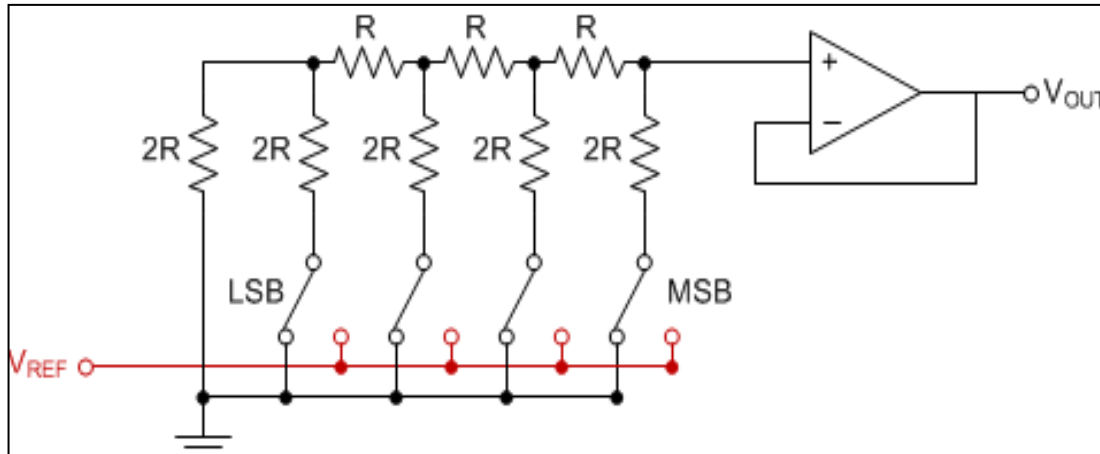


Figure 4.2. Instrumentation Amplifier Practical Circuit Diagram

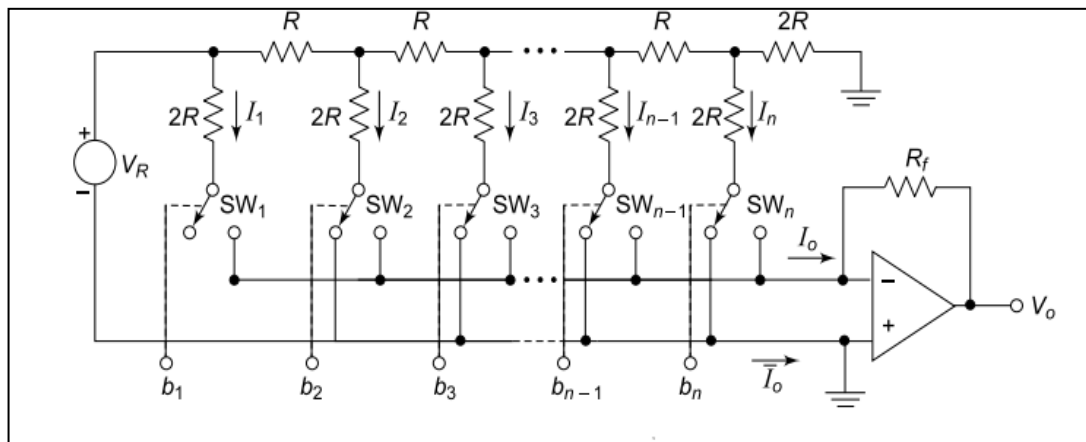
5. DIGITAL TO ANALOG CONVERTER

DACs are commonly used in music players to convert digital data streams into analog audio signals. They are also used in televisions and mobile phones to convert digital video data into analog video signals. These two applications use DACs at opposite ends of the frequency/resolution trade-off.

5.1. To design and verify the operation of R-2R and Inverted R-2R DAC Converter as per Figure 5.1



(a)



(b)

Figure 5.1. DAC (a) Non-inverted (b) Inverted

4.2. Construct a 4 bit R-2R ladder type DAC. Plot the transfer characteristics between binary input and output. Let $R = 10\text{ k}$.

4.3. Experimentally design inverted R-2R ladder has $R = R_f = 10\text{ k}\Omega$ and $V_R = 10\text{ V}$. Calculate the total current delivered to the op-amp and the output voltage when the binary input is 1110.

Try

1. Design the 4-bit digital to analog converter with the following input digital words, when 4-bit D/A converter with $V_R = 10\text{ V}$, $R_f = 10\text{ k}\Omega$ is considered i) 0001 ii) 0110 iii) 1010.
2. Construct a 4-bit R-2R ladder type DAC. Plot the transfer characteristics between binary input and output. Let $R = 10\text{ k}$.

6. ASTABLE MULTIVIBRATORS USING 555

An astable multivibrator is a type of electronic oscillator circuit that generates a continuous square wave output. The 555 timer IC (integrated circuit) is commonly used to build a stable multivibrators due to its versatility and ease of use.

6.1. Design a stable multi-vibrators using timer IC 555 shown in figure below:

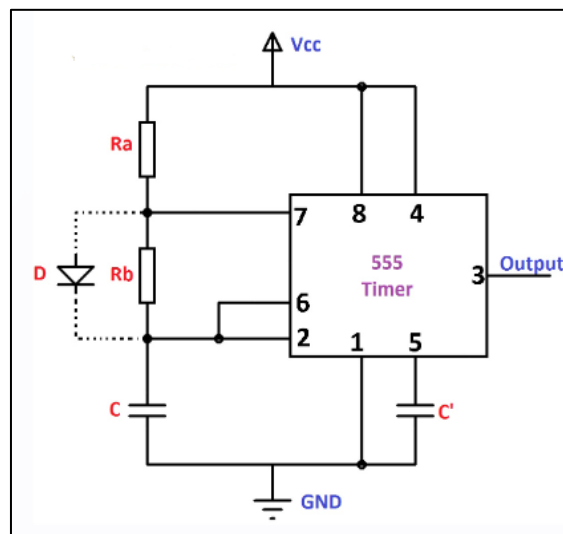


Figure. 6.1.A stable Multivibrator with 555 timer

Hint: The easy way to approach this is to select the appropriate resistor and capacitor values.

6.2. Design 2 kHz clock with 50 percent duty cycle using 555 timer:

6.3. Measure the output frequency and duty cycle of a 555 timer which is configured to run in astable mode with $R_1 = 10\text{ k}$, $R_2 = 60$ and $C = 0.5\text{ }\mu\text{F}$.

Compare the values in hardware and Simulation using Multisim.

Try

1. Find the output frequency and duty cycle of a 555 timer which is configured to run in a stable mode with $R_1 = 10k$, $R_2 = 60$ and $C = 0.5 \mu F$.
2. Design a 555 timer an astable multivibrator with an output signal frequency of 800hz and 60% duty cycle.

7. MONOSTABLE MULTIVIBRATORS USING 555

A monostable multivibrator, also known as a one-shot or a pulse generator, can be easily created using a 555 timer IC. In a monostable configuration, the 555 timer produces a single output pulse in response to an external trigger signal.

7.1 To design and construct mono stable multi-vibrators using IC 555.

The circuit template is given below.

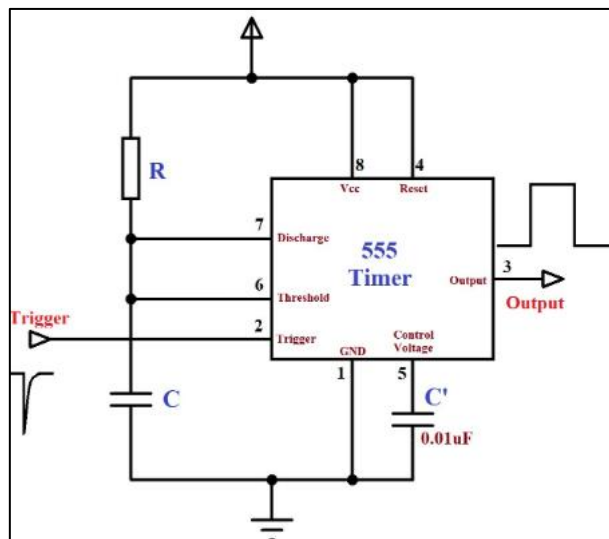


Figure 7.1. Monostable Multivibrator with 555 timer

Hint: To design this multivibrator adjust R_1 , R_2 , and C values based on your specific requirements. Adjust R_1 , R_2 , and C values based on your specific requirements. The RESET pin can be connected to V_{cc} if not used.

7.2 Construct monostable multivibrator to produce output pulse 15ms wide with hardware and multisim software.

7.3 Design monostable multivibrator to produce output pulse 10 ms wide using hardware.

Try

- 1 IC555 timer used as a monostable has $R = 20k\Omega$, and $C = 0.01 \mu F$. What is the duration of output pulse in simulation and compare it with theoretical values.

8. SCHMITT TRIGGER USING 555

Schmitt triggers can be used to change a sine wave into a square wave, clean up noisy signals, and convert slow edges to fast edges. A Schmitt trigger is a type of comparator circuit with hysteresis, meaning it has two different voltage thresholds for rising and falling input signals. The 555 timer IC (integrated circuit) can be configured as a Schmitt trigger, providing a stable and reliable solution for signal conditioning.

8.1 Design Schmitt trigger using NE555 Timer

Construct the circuit as per schematic 8.1.(a). The expected input and output waveforms are shown in Figure. 8.1(b).

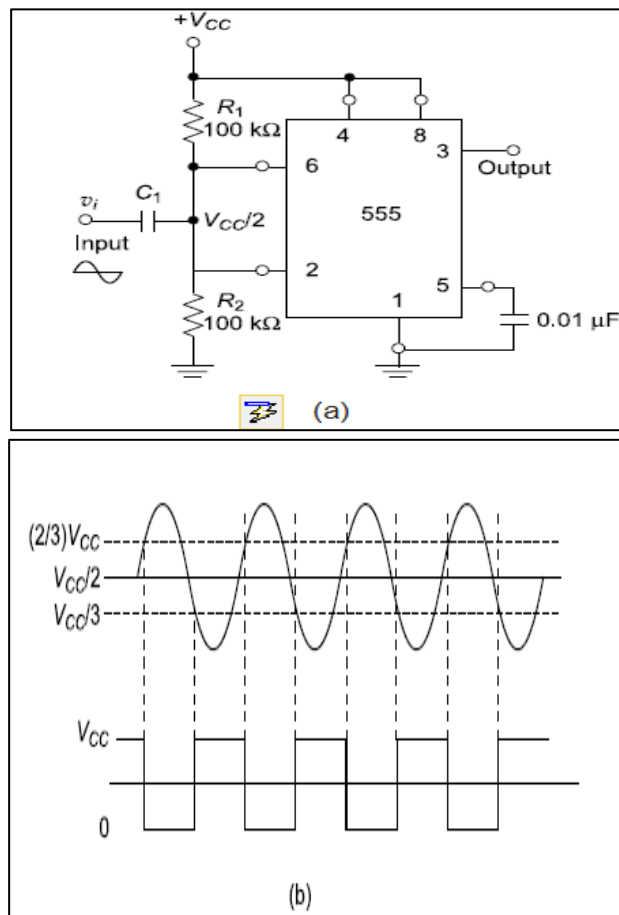


Figure 8.1.Schmitt Trigger (a) Connection Diagram (b) Wave Forms

8.1 Design Schmitt trigger circuit to achieve upper and lower threshold voltage as 1.5V.

Hint: The formula for the threshold voltages in an astable multivibrator configuration (which can be used as a Schmitt trigger) is:

$$V_{TH} = \frac{R_1}{R_1 + R_2} \times V_{CC}$$

8.2. Design Schmitt trigger circuit to achieve upper and lower threshold voltage as 3 volts. Hint: Adjust R1, R2, and C1 values as needed to achieve the desired

upper and lower threshold voltages and frequency of oscillation.

Try

1. Design a Schmitt trigger with upper and lower threshold voltages of 2.5 volts and 1.5 volts, respectively, and a frequency of oscillation around 1 kHz.
2. Hint: Calculate resistor ratios, choose proper capacitor value and calculate the frequency.
3. Design a Schmitt trigger using a 555 timer IC with the following specifications:
Upper Threshold Voltage (VUP): 3.0 volts Lower Threshold Voltage (V_{LOW}): 1.0 volts
Oscillation Frequency (f): 2 kHz.

9. SECOND ORDER ACTIVE LOW-PASS FILTERS

Low-pass filters have applications such as anti-aliasing, reconstruction, and speech processing, and can be used in audio amplifiers, equalizers, and speakers. Low-pass filters can also be used in conjunction with high-pass filters to form band pass, band-stop, and notch filters.

9.1. Design of First Order Low pass Filter with roll of rate 20 db /decade

Determine the roll of rate for the first order Low pass filter through simulation and hardware for the circuit shown in Figure. 9.1.

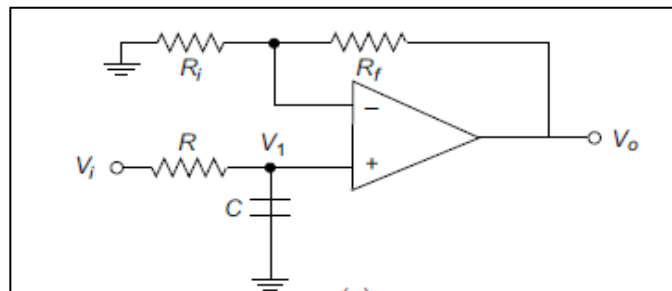


Figure 9.1: Low Pass Filter First order

9.2. Design of Second Low pass Filter with roll of rate 40 db/decade. Also design a second-order low-pass Butterworth filter with a cut-off frequency of 10 kHz and unity gain at low frequency. Hint. The circuit is shown in Figure. 9.2.

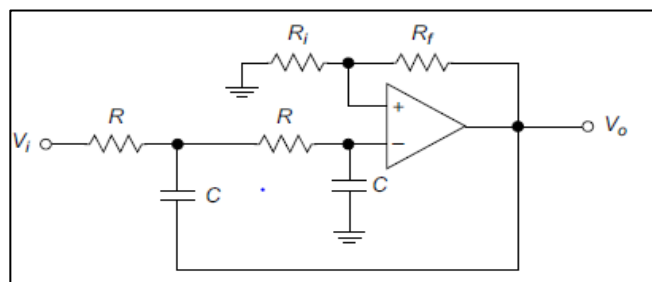


Figure 9.2: Second Order Low Pass Filter

9.3 Design a second order butter-worth low-pass filter having upper cut-off frequency 1 kHz. Plot the frequency response. Given parameters: $f_h=1$ kHz, $C=0.1F$, $R=1.6Kohms$ and damping factor $=1.414$.

Try

1. Design a second order Butterworth low-pass filter having upper cut-off frequency of 2 kHz.
2. A third order and a fourth order low-pass Butterworth filters have a cut-off frequency of 10 kHz and unity gain at low frequency. Determine the magnitude of the voltage transfer function in dB at 12 kHz for each filter.
3. Design a Butterworth second-order LPF with following specifications.
 - I. Sallen–Key equal component configuration
 - II. Critical frequency $\omega_c = 1$ radian/s.
 - III. Cut-off frequency f_c of 1.6 kHz
 - IV. Damping factor $a = 1.0$

10. SECOND ORDER ACTIVE HIGH-PASS FILTERS

High pass filters are used in audio system as part of an audio crossover to direct high frequencies to a tweeter while attenuating bass signals which could interfere with, or damage, the speaker. High pass filters are also used in digital image processing to perform image modifications, enhancements, noise reduction etc.

10.1. Design of First Order High pass Filter with roll of rate 20 db/decade at a cut-off frequency of 2 kHz with a pass band gain of 2. Also, plot its frequency response for the circuit shown in Figure 10.1.

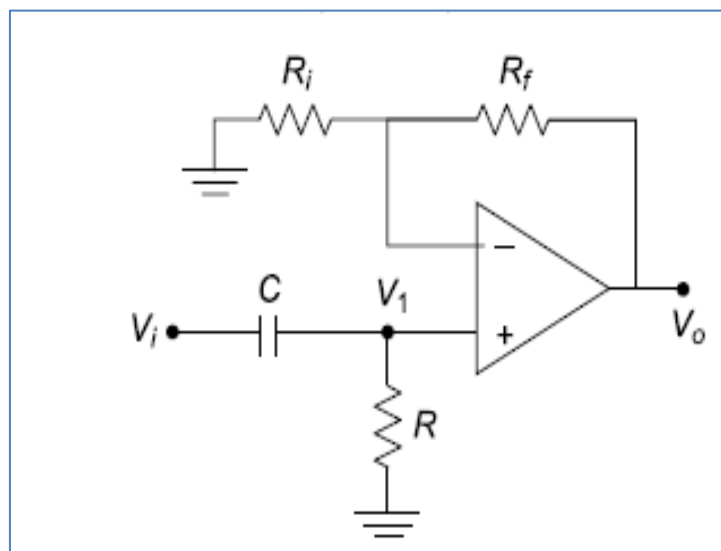


Figure 10.1: High Pass Filter First order

10.2. Construct a butter-worth high-pass filter of second order with lower cut-off frequency 5 kHz. Given parameters: $f_l=12$ kHz, $C=0.1\mu\text{F}$, $R=3\text{k}$ Ohms and damping factor $=1.625$. Determine R_f , R_i and frequency response. Also determine the roll of rate for the Second order high pass filter through simulation and hardware. The standard circuit is shown in Figure. 10.2.

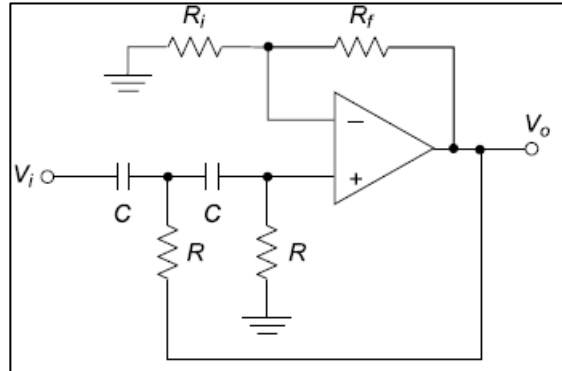


Figure 10.2: Second Order High Pass Filter

10.3. Find the lower cut-off frequency f_L for the second order high-pass Butterworth filter shown in Figure. 5.3. Also, find the pass band gain of the filter, and plot the frequency response of the filter through simulation.

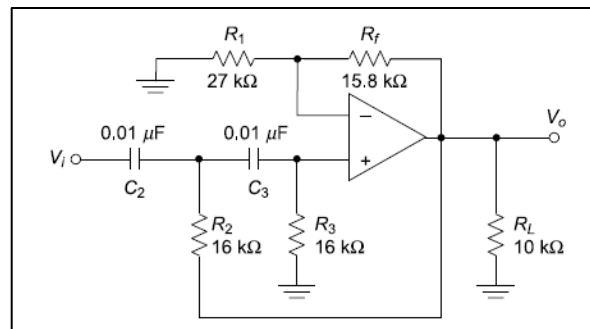


Figure 10.3: Second Order High Pass Filter

Try

1. Design a fourth order high-pass Butterworth filter with unity gain having a cut-off frequency of 50 kHz. Determine the frequency at which the voltage transfer function magnitude is 2% of its maximum value.
2. Design a second-order, high-pass Bessel filter, with a break frequency (f_{3dB}) of 5 kHz through simulation.

11. PLL USING IC 565

11.1. For the phase locked loop circuit of Figure 11.1. estimate the capture range and lock-in range using simulation and compare theoretical and practical values.

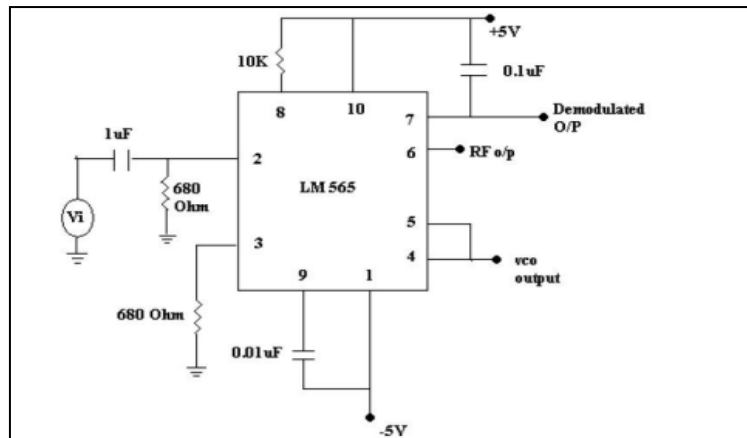


Figure 11.1.PLL with LM 565

11.2. Design phase locked loop circuit for estimating the capture range and lock-in range.

11.3. Measure the output frequency, lock range and capture range of a 565 PLL if $R = 10\text{ K ohms}$, $C_T = 0.01\mu\text{f}$ and $C = 10\mu\text{f}$.

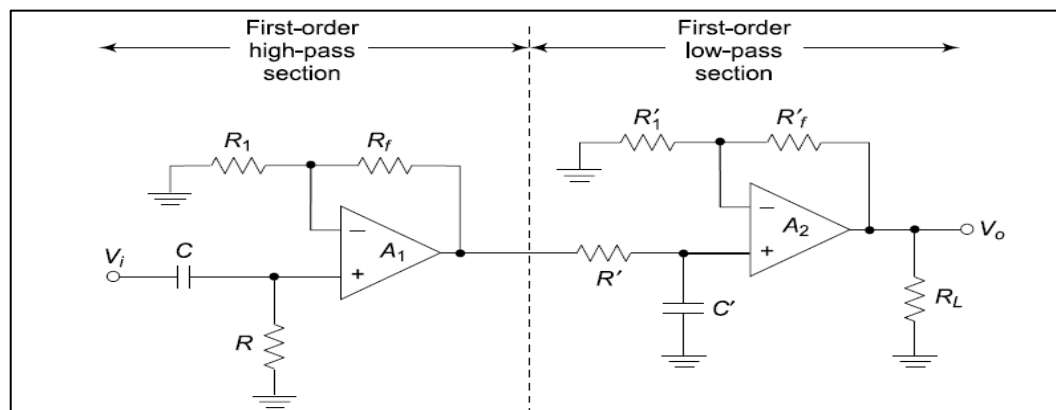
Try

1. Design the phase locked loop circuit with the basic operating frequency of 2kHz and cutoff frequency of 1kHz?
2. Measure the output frequency, lock range and capture range of a 565 PLL if $R=10K\Omega$ ohms, $C_1 = 0.01\mu F$ and $C = 10 \mu F$.
3. For PLL 565, given the free-running frequency as 100 kHz, the demodulation capacitor of $2 \mu F$ and supply voltage is $\pm 6 V$, determine the lock and capture frequencies and identify the component values.

12 SECOND ORDER ACTIVE BAND PASS FILTERS

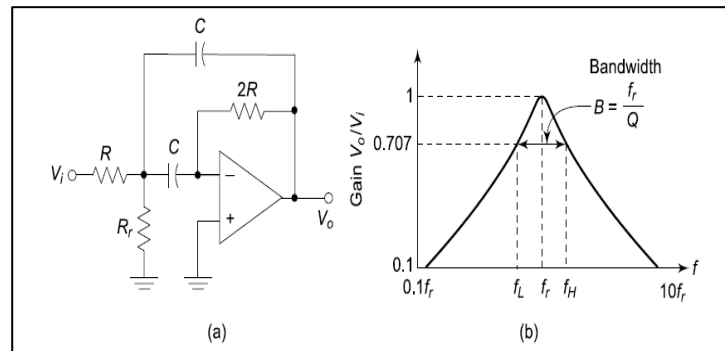
Band pass filters are widely used in wireless transmitters and receivers. The main function of such a filter in a transmitter is to limit the bandwidth of the output signal to the band allocated for the transmission. This prevents the transmitter from interfering with other stations.

12.1. Design a wide band pass filter using operational amplifier 741 as per Figure 12.1



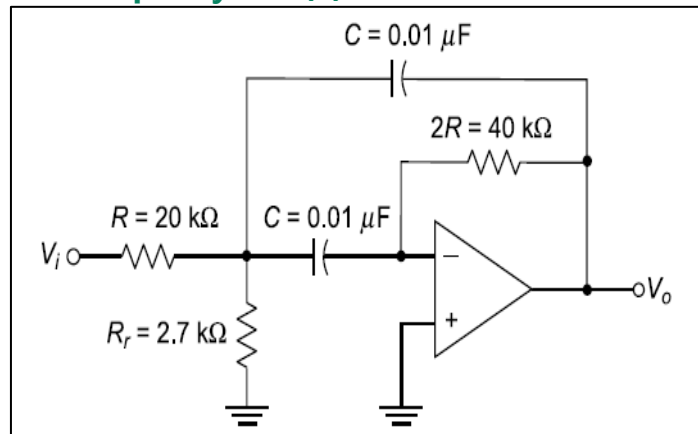
12.1. Band pass Filter

12.2. Design a narrow band pass filter using operational amplifier 741 as per the Figure 12.2



12.2. Narrow Band pass Filter and it's response

12.3. Given a band pass filter with the component values shown in Figure. 9.3, find its (a) resonant frequency and (b) bandwidth.



12.3 Band pass Filter

Try

1. Given a band pass filter with resonant frequency f_r of 1000 Hz and a bandwidth (B) of 3000 Hz, find its (a) quality factor, (b) lower cut-off frequency and (c) higher cut-off frequency by hardware.
2. Construct band pass filter with resonant frequency f_r of 1500 Hz and a bandwidth (B) of 3500 Hz, find its (a) lower cut-off frequency and (b) higher cut-off frequency using multisim.

13. SECOND ORDER WIDE BAND REJECT FILTERS

The Band reject Filters are used as an output sound source from musical instruments. These are used in such amplifiers to reduce a specific humming sound that may occur after the instruments are plugged

13.1. To design and verify the operation of the Band reject filters:

The Circuit and expected wave forms are shown in Figure 13.1(a) & 13.1(b) respectively.

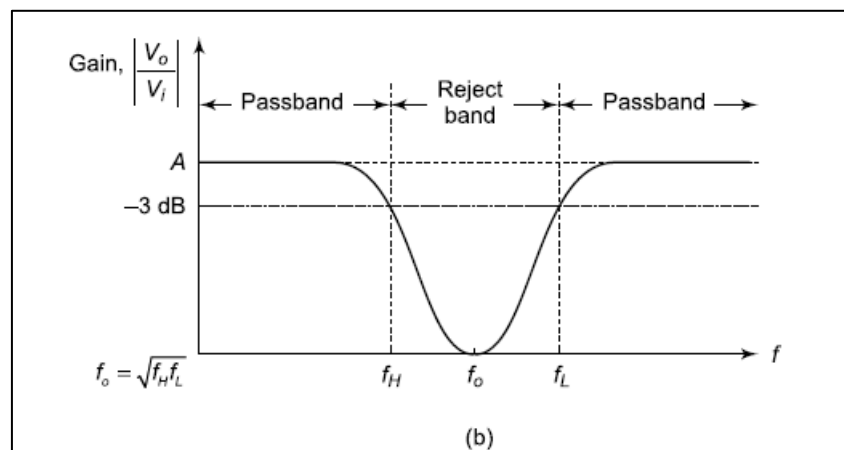
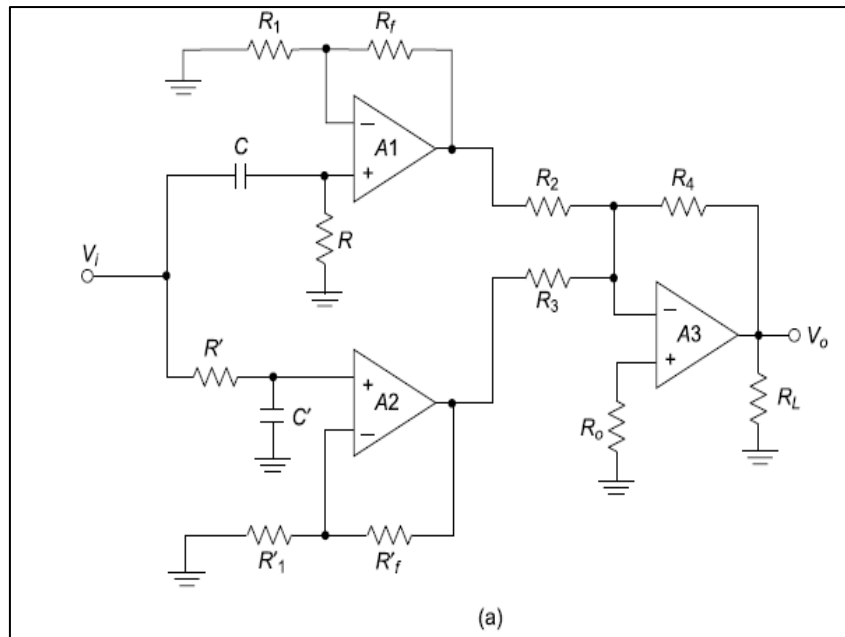


Figure 13.1. Wide band reject Filter Trigger (a) Connection Diagram
(b) Wave Forms

13.2. Design a wide band pass filter having $f_l=400\text{Hz}$, $f_h=2\text{kHz}$ and pass band gain of 4.

13.3. Construct a wide band reject filter with $f_h=500\text{Hz}$, $f_l=4\text{kHz}$ with a pass band gain of 2 using multisim

Try

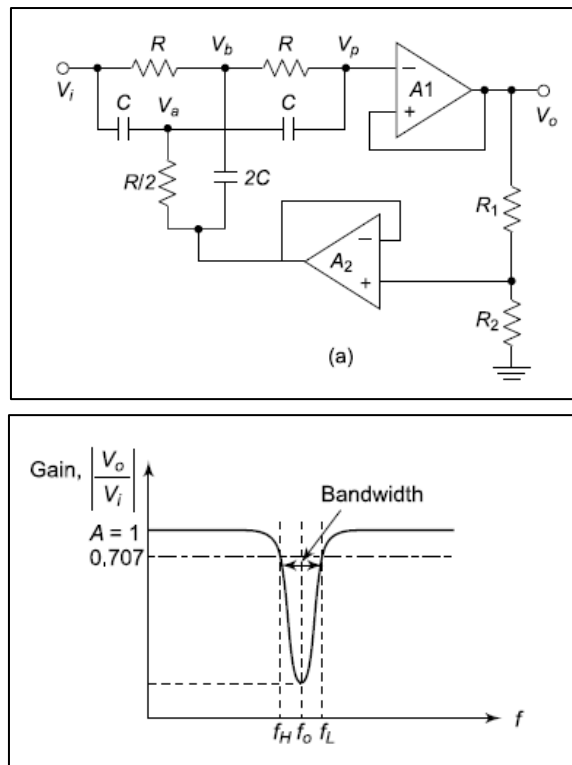
1. Design a wide band reject filter with $f_h=1000\text{Hz}$, $f_l=8\text{kHz}$ with a pass band gain of 4 by simulation.

14 SECOND ORDER NARROW BAND REJECT FILTERS

The Band reject Filters are used as an output sound source from musical instruments. These are used in such amplifiers to reduce a specific humming sound that may occur after the instruments are plugged

14.1. To design and verify the operation of the Narrow Band reject filters:

Hint: The narrowband band-reject filter, often called the notch filter, is the twin-T network cascaded with the voltage follower as shown in Figure. 14.1(a). The frequency response characteristic of the wideband band-reject filter is shown in Figure. 14.1(b).



(b)

Figure 14.1. Narrow band reject Filter Trigger (a) Connection Diagram (b) Wave Forms

14.2. Design a wide band pass filter having $f_L=1.5\text{kHz}$, $f_H=2\text{kHz}$ and pass band gain of 4 using multisim.

14.3 Construct a Narrow band reject filter with $f_H=1000\text{Hz}$, $f_L=1500\text{Hz}$ with a pass band gain of 2.

Try

1. Design a Narrow band reject filter with $f_H=4\text{kHz}$, $f_L=4500\text{Hz}$ with a pass band gain of 3 by simulation.
2. Using hardware build Narrow band reject filter with $f_H=5\text{kHz}$, $f_L=5500\text{Hz}$ with a pass band gain of 3 by simulation.

V. REFERENCE BOOKS:

1. D. Roy Chowdhury, “Linear Integrated Circuits”, New age international (p) Ltd, 2nd edition, 2003.
2. Ramakanth A.Gayakwad, “Op-Amps & Linear ICs”, PHI, 3rd edition, 2003.
3. John F.Wakerly, “ Digital Design Principles and practices”, Prentice Hall, 3rd edition, 2005.

VI. WEB REFERENCES:

1. Salivahanan, “Linear Integrated Circuits and Applications”, TMH, 1st edition, 2008.

VII. ELECTRONICS RESOURCES:

1. <https://nptel.ac.in/courses/108108111>
2. <https://archive.nptel.ac.in/courses/108/108/108108112/>