College of Engineering, Design and Computing
UNIVERSITY OF COLORADO DENVER

DEPARTMENT OF ELECTRICAL ENGINEERING

ELEC 3900: CIRCUIT DESIGN AND FABRICATION

SUMMER 2025

LAB#2

OP-AMP APPLICATIONS

<u>Objectives</u>:

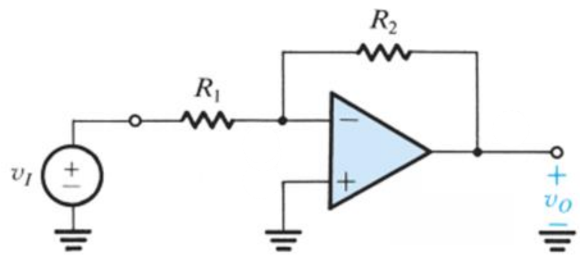
This laboratory exercise will focus on developing practical skills and theoretical understanding related to operational amplifiers. Our specific objectives are to:

- Introduce and gain proficiency in using standard laboratory equipment essential for electronic circuit design and analysis.
- Thoroughly investigate the fundamental properties of **operational amplifiers** and analyze the behavior of their basic configurations, specifically the **inverting, non-inverting** circuits.
- Examine the principles and practical implementation of op-amps in two important filter applications: **low-pass filters** and **high-pass filters**.
- Acquire hands-on experience in design, physical construction on a breadboard, and subsequent performance analysis of various op-amp circuits.

EXPERIMENT#1: OP-AMP as an Inverting Amplifier

Implement the circuit of an inverting amplifier for a gain of 10. The input to this circuit should be a sinewave of 1 V peak to peak and 100 Hz frequency. Display the waveforms you obtain separately for the input and output. Select the appropriate thickness of the waveforms so that the examiner can evaluate the results you get.

CIRCUIT DIAGRAM:



PROCEDURE:

Circuit Assembly:

1. Connect the positive terminal of your dual DC power supply (e.g. +12V) to pin 7 of the IC741.
2. Connect the negative terminal of your dual DC power supply (e.g. -12V) to pin 4 of the IC741.
3. Connect the common ground of the DC power supply to a designated ground rail on your breadboard. This will serve as your circuit's ground reference.
4. Connect one end of the input resistor (R_1) to pin 2 of the IC741.
5. The other end of R_1 will be connected to the function generator's output.
6. Connect one end of the feedback resistor (R_2) to pin 6 of the IC741.
7. Connect the other end of R_2 to pin 2 of the IC741, sharing the connection point with R_1.
8. Connect pin 3 of the IC741 directly to the common ground rail on your breadboard.
9. Connect the positive output terminal of the function generator to the free end of R_1.
10. Connect the ground terminal of the function generator to the common ground rail on your breadboard.

Function Generator Setup:

1. Configure the function generator to output a sine wave.
2. Set the frequency to 100 Hz.
3. Adjust the amplitude to 1 V peak-to-peak (Vp-p). Most function generators will have a Vp-p setting. If not, set the amplitude to 0.5 V peak (Vp).

Oscilloscope Setup and Measurement:

1. Connect the probe of oscilloscope **Channel 1** across the input signal. Place the probe tip at the connection point between the function generator output and R_1.
2. Connect the ground clip of the **Channel 1** probe to the common ground rail on your breadboard.
3. Adjust the vertical scale (Volts/Div) and horizontal scale (Time/Div) on **Channel 1** to clearly display the 1 Vp-p, 100 Hz input sine wave.
4. Use the oscilloscope's measurement functions to confirm the input voltage and frequency.
5. Connect the probe of oscilloscope **Channel 2** to pin 6 of the IC741.
6. Connect the ground clip of the **Channel 2** probe to the common ground rail on your breadboard.
7. Adjust the vertical scale (Volts/Div) and horizontal scale (Time/Div) on **Channel 2** to clearly display the output sine wave.
8. Observe the phase relationship between the input and output signals. Use the oscilloscope's measurement functions to determine the output voltage.

Use the following formula to calculate the voltage gain of the circuit and select appropriate values for R1 and R2:

$$\mathbf{A_V = -\frac{R_2}{R_1} * Vin}$$

PSpice Simulation:

A. Schematic implementation:

1. Open the "**Cadence Capture CIS**" application and follow the instructions given in LAB#1 Software Tutorial section.
2. Select "**Place**" → "**PSpice Components**" → "**Discrete**" → "**Op-Amp**".
3. Connect the circuit as shown in the circuit diagram above.
4. Select the resistors in the circuit from "**Place**" → "**PSpice Components**" → "**Passives**" → "**R**" and change the values as per the requirements of the circuit.
5. For the input signal to the op-amp, Select the sinewave parameter from the **Function Generator Setup given above.** For this, Select "**Place**" → "**PSpice Components**" → "**Sources**" → "**Voltage Sources**" → "**Sine**".
6. Input the following parameters for the "Sine": VOFF = 0, VAMPL = 0.5V, FREQ = 100 Hz, AC = 0.
7. Select the ground for the circuit from "**Place**" → "**GND**" → "**0/CAPSYM**".

B. Simulation Implementation:

1. Select "**PSpice**" → "**New Simulation Profile**" and give a name to the simulation profile as "**OPAMP_Inverting_Amplifier**", then click "**Create**".
2. A new window of "**Simulation Settings**" will open.
3. Under the "**Analysis**" tab, Select "**Time Domain (Transient)**".
4. For the Transient analysis, Use the following parametric values:
   Run to time = 50 ms, Start = 0 seconds, Maximum Step Size = 0.01seconds.
5. Select "**Apply**" and then select "**Ok**".
6. Select the "**Run PSpice**" button on the schematic page to begin the simulation.
7. You will observe that the simulation is a blank black screen.
8. In the Schematic page, select a voltage probe from the top panel and place it on the sinewave positive terminal. Observe the waveform for the signal input.
9. Going back to the schematic page again, select another voltage probe from the panel and place it on the output of the op-amp. Observe the signal output.
10. Adjust the width/thickness and color of both the waveforms so that they are visible to the instructor clearly using the "**Trace Property**" option.

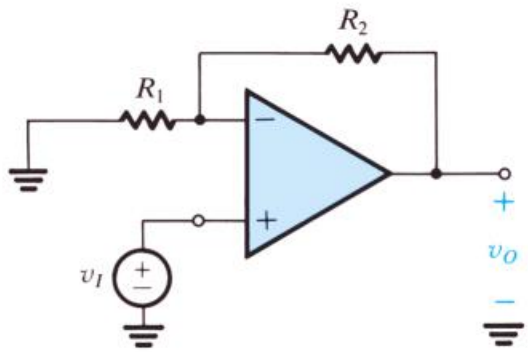
Questions: **Total points - 10 points**

1. Draw the circuit diagram with the pin numbers from the IC 741 on the circuit and components values by hand and submit this diagram in the report. **(2 points)**
2. As part of your analysis of the inverting amplifier, present the oscilloscope capture of the output signal. The image should clearly demonstrate the phase relationship between input and output (if both are displayed), the amplitude of the output, and the frequency. Include the oscilloscope's settings for both amplitude and time bases. **(4 points)**
3. Present the schematic, simulated output waveform of the inverting amplifier from Cadence PSpice. In your report, you directly compare the simulated results (e.g., gain, phase shift) to your manual calculations and the observations from the physical oscilloscope experiment. **(4 points)**

EXPERIMENT#2: OP-AMP as a Non-Inverting Amplifier

Implement the circuit of a non-inverting amplifier for a gain of 11. The input to this circuit should be a sinewave of 1 V peak to peak and 100 Hz frequency. Display the waveforms you obtain separately for the input and output. Select the appropriate thickness of the waveforms so that the examiner can evaluate the results you get.

CIRCUIT DIAGRAM:



PROCEDURE:

Circuit Assembly:

1. Connect the positive terminal of your dual DC power supply (e.g. +12V) to pin 7 of the IC741.
2. Connect the negative terminal of your dual DC power supply (e.g. -12V) to pin 4 of the IC741.
3. Connect the common ground of the DC power supply to a designated ground rail on your breadboard. This will serve as your circuit's ground reference.
4. Connect one end of the input resistor (R_1) to pin 2 of the IC741.
5. The other end of R_1 will be connected to GND.
6. Connect one end of the feedback resistor (R_2) to pin 6 of the IC741.
7. Connect the other end of R_2 to pin 2 of the IC741, sharing the connection point with R_1.
8. Connect pin 3 of the IC741 to the positive output terminal of the function generator.
9. Connect the ground terminal of the function generator to the common ground rail on your breadboard.

Function Generator Setup:

1. Configure the function generator to output a sine wave.
2. Set the frequency to 100 Hz.
3. Adjust the amplitude to 1 V peak-to-peak (Vp-p). Most function generators will have a Vp-p setting. If not, set the amplitude to 0.5 V peak (Vp).

Oscilloscope Setup and Measurement:

1. Connect the probe of oscilloscope **Channel 1** across the input signal. Place the probe tip at the connection point at the function generator output positive terminal.
2. Connect the ground clip of the **Channel 1** probe to the common ground rail on your breadboard.
3. Adjust the vertical scale (Volts/Div) and horizontal scale (Time/Div) on **Channel 1** to clearly display the 1 Vp-p, 100 Hz input sine wave.
4. Use the oscilloscope's measurement functions to confirm the input voltage and frequency.
5. Connect the probe of oscilloscope **Channel 2** to pin 6 of the IC741.
6. Connect the ground clip of the **Channel 2** probe to the common ground rail on your breadboard.
7. Adjust the vertical scale (Volts/Div) and horizontal scale (Time/Div) on **Channel 2** to clearly display the output sine wave.
8. Observe the phase relationship between the input and output signals. Use the oscilloscope's measurement functions to determine the output voltage.

Use the following formula to calculate the gain of the circuit and select appropriate values for R1 and R2:

$$\mathbf{A_V = \{1 + \frac{R_2}{R_1}\}\ Vin}$$

PSpice Simulation:

A. Schematic implementation:

1. Open the "**Cadence Capture CIS**" application and follow the instructions given in LAB#1 Software Tutorial section.
2. Select "**Place**" → "**PSpice Components**" → "**Discrete**" → "**Op-Amp**".
3. Connect the circuit as shown in the circuit diagram above.
4. Select the resistors in the circuit from "**Place**" → "**PSpice Components**" → "**Passives**" → "**R**" and change the values as per the requirements of the circuit.
5. For the input signal to the op-amp, Select the sinewave parameter from the **Function Generator Setup given above.** For this, Select "**Place**" → "**PSpice Components**" → "**Sources**" → "**Voltage Sources**" → "**Sine**".
6. Input the following parameters for the "Sine": VOFF = 0, VAMPL = 0.5V, FREQ = 100 Hz, AC = 0.
7. Select the ground for the circuit from "**Place**" → "**GND**" → "**0/CAPSYM**".

B. Simulation Implementation:

1. Select "**PSpice**" → "**New Simulation Profile**" and give a name to the simulation profile as "**OPAMP_Non_Inverting_Amplifier**", then click "**Create**".
2. A new window of "**Simulation Settings**" will open.
3. Under the "**Analysis**" tab, Select "**Time Domain (Transient)**".
4. For the Transient analysis, Use the following parametric values:
   Run to time = 50 ms, Start = 0 seconds, Maximum Step Size = 0.01seconds.
5. Select "**Apply**" and then select "**Ok**".
6. Select the "**Run PSpice**" button on the schematic page to begin the simulation.
7. You will observe that the simulation is a blank black screen.
8. In the Schematic page, select a voltage probe from the top panel and place it on the sinewave positive terminal. Observe the waveform for the signal input.
9. Going back to the schematic page again, select another voltage probe from the panel and place it on the output of the op-amp. Observe the signal output.
10. Adjust the width/thickness and color of both the waveforms so that they are visible to the instructor clearly using the "**Trace Property**" option.

Questions: **Total points - 10 points**

1. Draw the circuit diagram with the pin numbers from the IC 741 on the circuit and components values by hand and submit this diagram in the report. **(2 points)**
2. As part of your analysis of the non-inverting amplifier, present the oscilloscope capture of the output signal. The image should clearly demonstrate the phase relationship between input and output (if both are displayed), the amplitude of the output, and the frequency. Include the oscilloscope's settings for both amplitude and time bases. **(4 points)**
3. Present the schematic, simulated output waveform of the non-inverting amplifier from Cadence PSpice. In your report, you directly compare the simulated results (e.g., gain, phase shift) to your manual calculations and the observations from the physical oscilloscope experiment. **(4 points)**

ACTIVE FILTER DESIGN

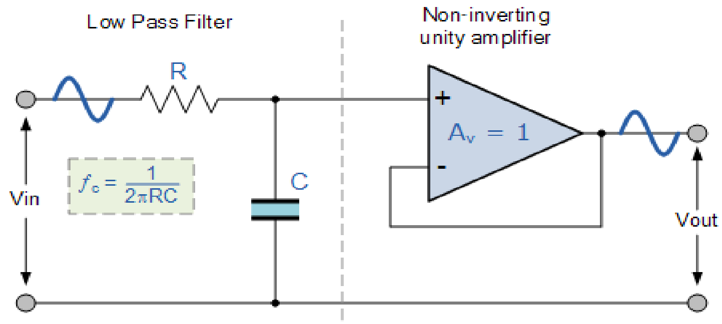
The objective of this section is to design several active filters meeting the specifications given. Operational amplifiers are used as an active element. The circuits are to be designed, simulated, built and tested.

INTRODUCTION:

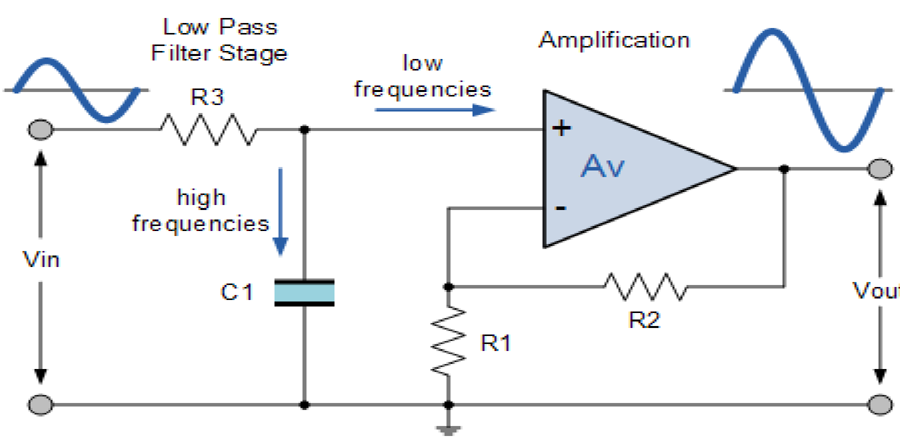
Filters can be placed in one of two categories: passive or active. Passive filters include only passive components—resistors, capacitors, and inductors. In contrast, active filters use active components, such as op-amps, in addition to resistors and capacitors, but not inductors.

There are 4 filters:

1. **Low Pass Filter** passes signals with a frequency lower than a selected cutoff frequency and attenuates signals with frequencies higher than the cutoff frequency. The image given below is a generalized



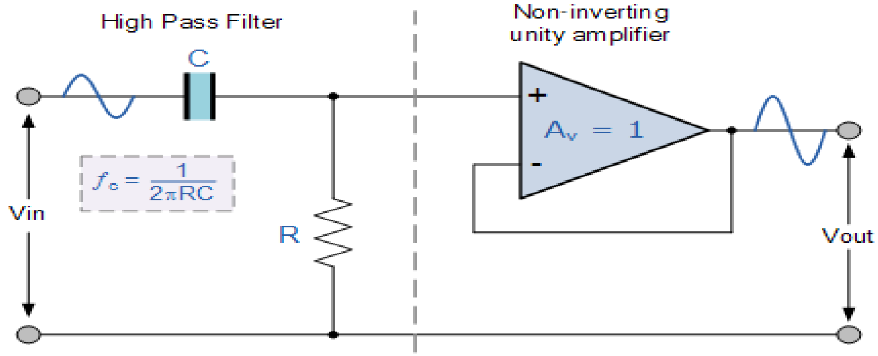
First order Low Pass Filter



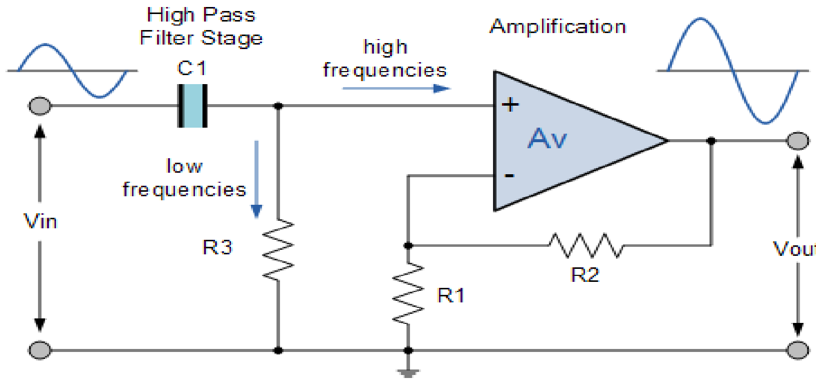
First order Low Pass Filter with voltage gain amplification

Image source: https://www.electronics-tutorials.ws/filter/filter_5.html

2. **High Pass Filter** that passes signals with a frequency higher than a certain cutoff frequency and attenuates signals with frequencies lower than the cutoff frequency.



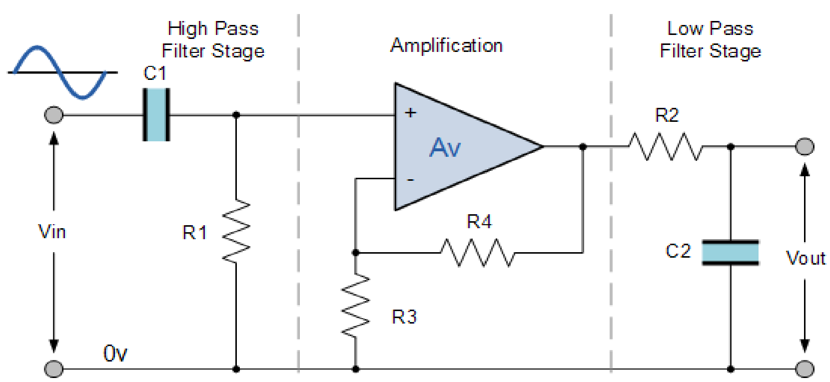
First Order High Pass Filter



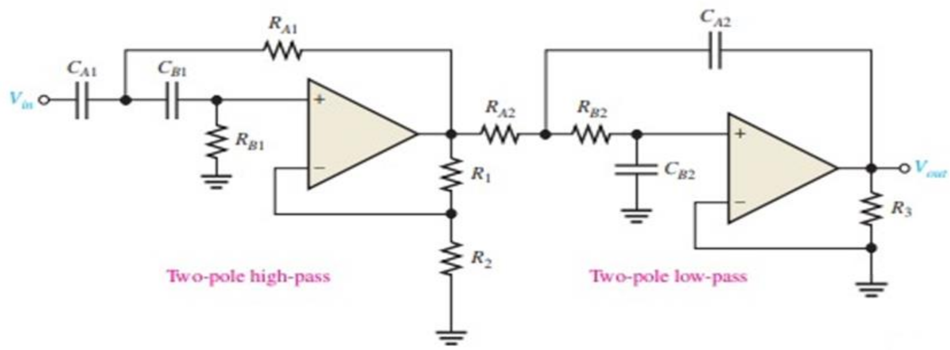
First order High Pass Filter with voltage gain amplification

https://www.electronics-tutorials.ws/filter/filter_6.html

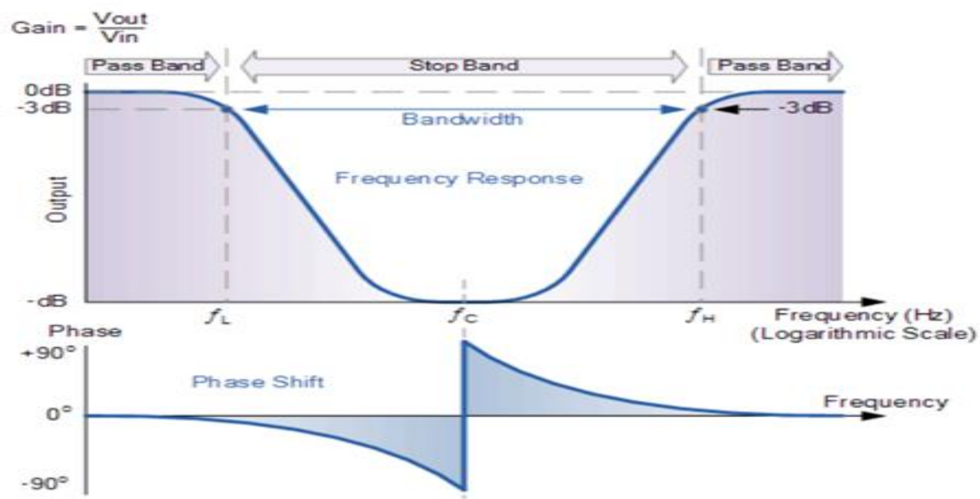
3. **Band Pass Filter** passes frequencies within a certain range and rejects (attenuates) frequencies outside that range. The band pass filter can be formed by cascading a low – pass filter with a high pass filter.



First Order Band Pass Filter

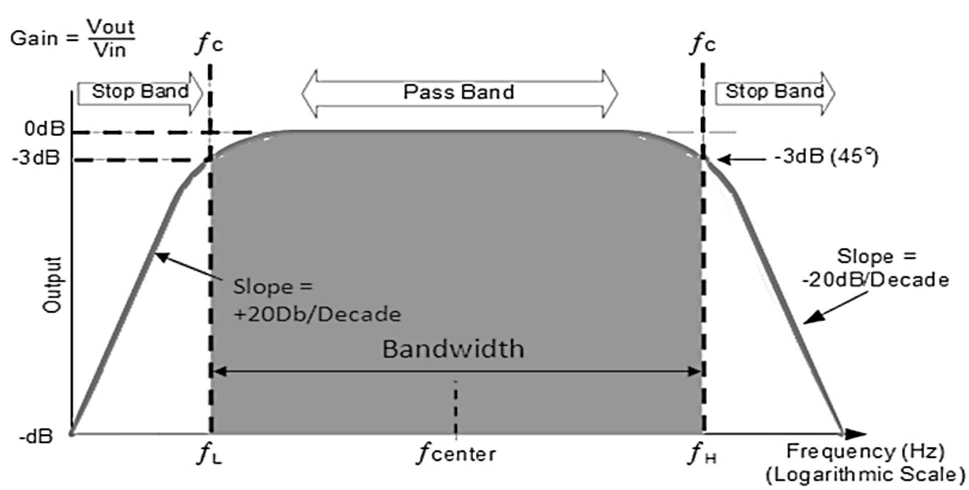


4. **Band Reject (Band Stop or notch) Filters** which pass all frequencies with the exception of those within a specified stop band which are greatly attenuated.



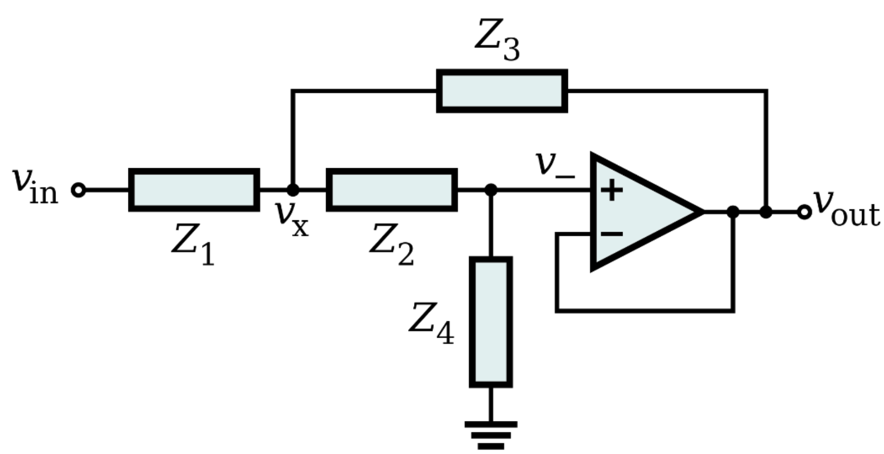
https://www.electronics-tutorials.ws/filter/band-stop-filter.html

To summarize all the filter responses, consider the following



The **Sallen and Key Filter** design is a second-order active filter topology which we can use as the basic building blocks for implementing higher order filter circuits, such as low-pass (LPF), high-pass (HPF) and band-pass (BPF) filter circuits. As we have seen in this filters section, electronic filters, either passive or active, are used in circuits where a signals amplitude is only required over a limited range of frequencies. The advantage of using *Sallen-Key Filter* designs is that they are simple to implement and understand.

The Sallen and Key topology are an active filter design based around a single non-inverting operational amplifier and two resistors, thus creating a voltage-controlled voltage-source (VCVS) design with filter characteristics of, high input impedance, low output impedance and good stability, and as such allows individual Sallen-key filter sections to be cascaded together to produce much higher order filters. A first order filter can be converted into a second order type by using an additional RC network.



General representations of a unity gain Sallen-Key topology active filter

FILTER DESIGN STEPS

A. Choose a value for the high cutoff frequency $\boldsymbol{f_H}$ using the formula

$$\boldsymbol{f_H = \frac{1}{2\pi * \sqrt{R3 * R4 * C1 * C2}}}$$

B. To simplify the design calculation, set R1 = R2 = R & C1 = C2 = C. Then choose a value of $C \leq 1\mu F$.

C. Calculate the value of R using the following equation:

$$\mathbf{R} = \frac{\mathbf{1}}{\mathbf{2\pi * f_H * C}}$$

D. Because of the equal resistor and capacitor values, the passband voltage gain $A_F = (1 + R_F / R_{IN})$ of the second order LPF/ HPF must be equal to 1.586 i.e. $R_F = 0.586 * R_{IN}$. This gain is necessary to guarantee Butterworth response. Hence, choose the value of $R_{IN} \leq 100\ k\Omega$ and calculate the value of R_F.

EXPERIMENT#3: OP-AMP as an Active Low Pass Filter

Design a second order Butterworth Low Pass Filter at a cutoff frequency of 2 kHz. Assume the power supply to the circuit such that the output waveform does not go in the saturation region. Perform the following stages:

- You will need to show the hand calculation for the assumptions taken for the components.
- Draw the circuit diagram for your design with the values available during the hand calculation stage.
- Draw the schematic diagram for the above circuit with labelled pin numbers on the IC LM358, the components you have selected and the power supply to the IC.
- Using the Cadence Capture CIS, simulate the circuit and obtain the transient response along with the frequency response of the circuit.
- Implement the circuit on a breadboard using IC LM358 and verify the output you obtain on the oscilloscope with the transient response graph from the simulation.

Questions: **Total points - 10 points**

1. Show the hand calculation done by you for the selection of the passive components in the design. (2 points)
2. Drawn by hand, show the circuit diagram for the designed circuit in detail. (2 points)
3. Drawn by hand, show the schematic diagram for the circuit. (2 points)
4. Present the schematic, simulated output waveform and the frequency response curve of the designed filter from Cadence PSpice. (2 points)
5. As part of your analysis, present the oscilloscope capture of the output signal and verify it with the simulated output. (2 points)

EXPERIMENT#4: OP-AMP as an Active High Pass Filter

Design a second order Butterworth High Pass Filter at a cutoff frequency of 5 kHz. Assume the power supply to the circuit such that the output waveform does not go in the saturation region. Perform the following stages:

- You will need to show the hand calculation for the assumptions taken for the components.
- Draw the circuit diagram for your design with the values available during the hand calculation stage.
- Draw the schematic diagram for the above circuit with labelled pin numbers on the IC LM358, the components you have selected and the power supply to the IC.
- Using the Cadence Capture CIS, simulate the circuit and obtain the transient response along with the frequency response of the circuit.

Questions: **Total points - 10 points**

1. Show the hand calculation done by you for the selection of the passive components in the design. (1 points)
2. Drawn by hand, show the circuit diagram for the designed circuit in detail. (3 points)
3. Drawn by hand, show the schematic diagram for the circuit. (3 points)
4. Present the schematic, simulated output waveform and the frequency response curve of the designed filter from Cadence PSpice. (3 points)