

BEE 233 Circuits

Fall 2015

Lab 4: Opamp integrators, differentiators and simple filters

1 Objectives

1. Analyze several simple opamp circuits.
2. Simulate the circuits and create Bode plots of the expected frequency response.
3. Build the circuits and observe and measure their actual behavior.
4. Compare expected versus actual results.

2 Circuits

You'll be analyzing these five opamp circuits in this lab. In all cases, the opamp is the same LM348 you worked with in Lab 3, $V_{CC} = +12\text{ V}$ and $V_{EE} = -12\text{ V}$.

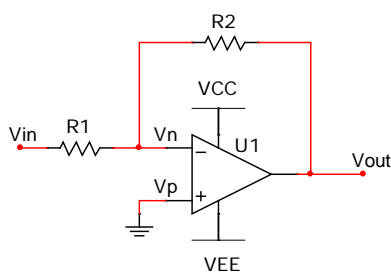


Figure 1. Inverting amplifier.

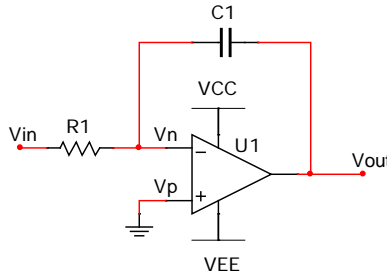


Figure 2. Simple integrator.

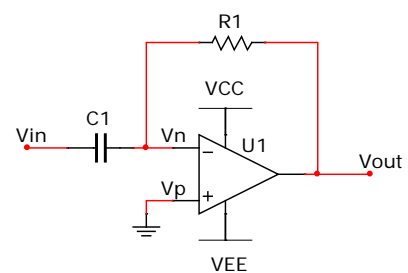


Figure 3. Simple differentiator.

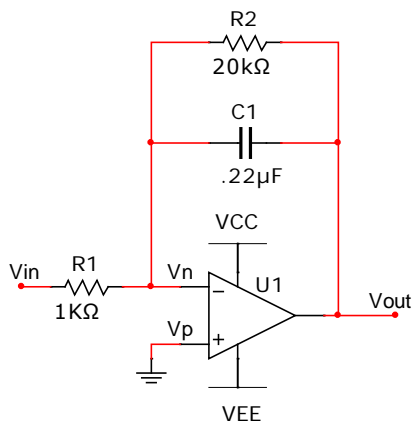


Figure 4. Integrator with shunt.

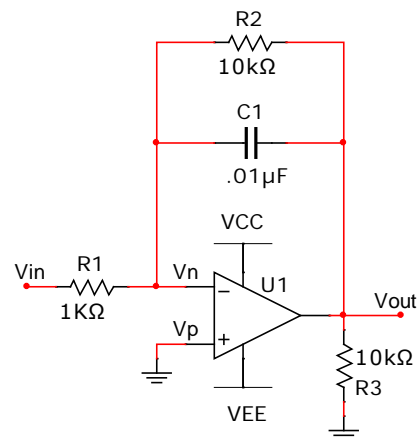


Figure 5. Low-pass filter.

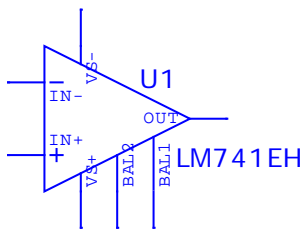
3 Components

To avoid having to tear one circuit apart to build the next, these are the components you'll need. You should record the measured values of your resistors.

Quantity	Description
2	LM348 opamps
3	1 K Ω resistors
3	10 K Ω resistors
1	20 K Ω resistor
3	0.01 μ F capacitors
1	0.22 μ F capacitor

The LM348 is a common part made by several manufacturers. Their datasheets describing this opamp are easily found on the web.

The LM348 may or may not be in the component database that came with your copy of Multisim. If it's not, the LM348 is simply a quad LM741. The various LM741xx parts are all basically the same except for packaging / power, which won't matter in your simulations. Here, e.g., is the Multisim LM741EH. Even though the real LM741EH has pins for power and balancing, they are not connected and don't need to be wired in the model that came with my copy of Multisim *but the models that came with your copy could require that VCC and VEE must be connected properly.*



Right-clicking properties:

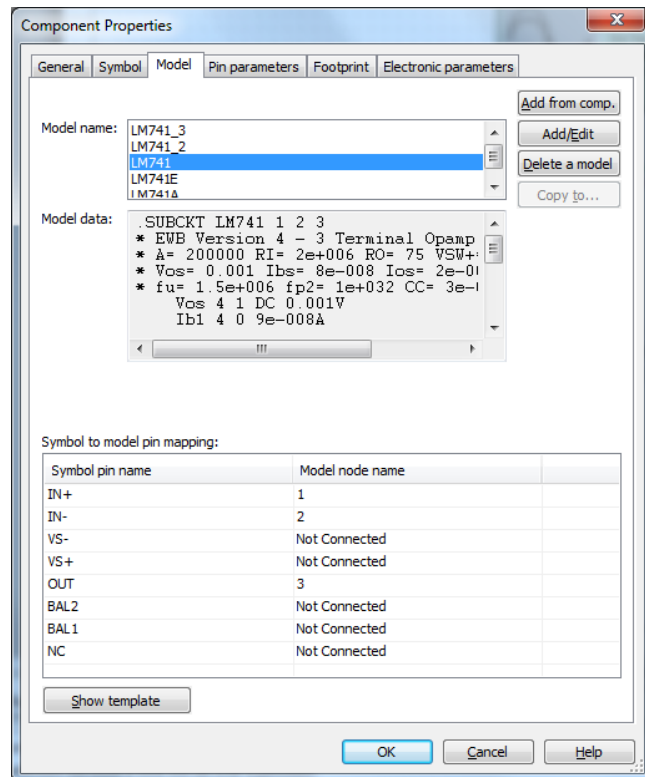
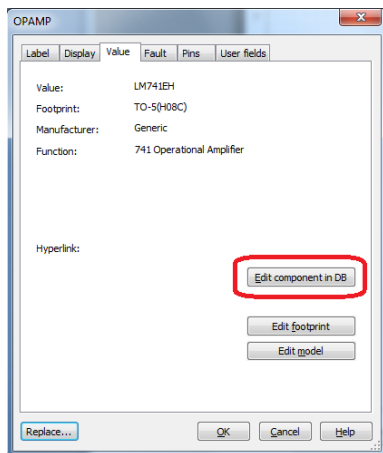


Figure 6. Only the input and output pins are connected my own LM741 model.

4 Discussion

In opamp circuits, the gain will often vary with frequency, either by design or because of the component limits. At any given frequency and with a sine wave input, we measure gain as either voltage gain, $G = \text{Gain}(A_v) = |A_v| = |V_{\text{out}}/V_{\text{in}}|$, or in dB (decibels), where $\text{Gain}(\text{dB}) = 20 \log_{10}(G)$. We can then plot this against frequency to show the bandwidth.

For example, here are the expected results for the low-pass filter, plotting gain versus frequency on a linear scale.

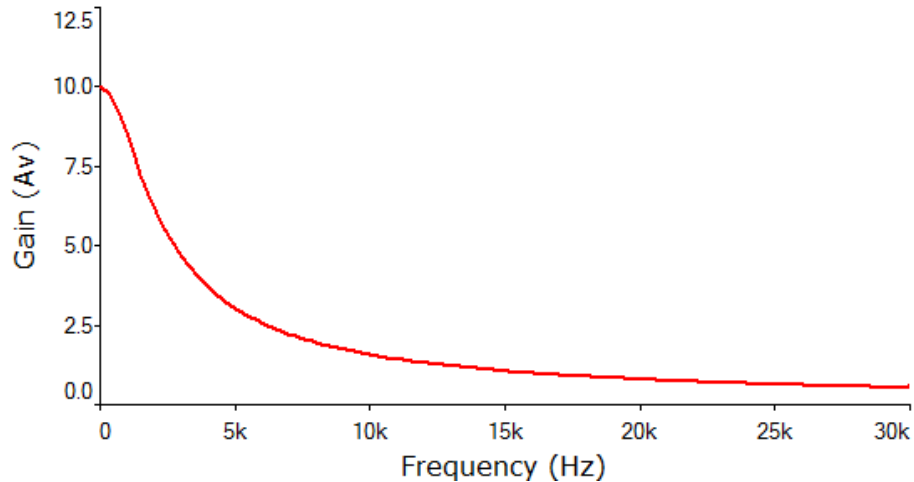


Figure 7. Linear plot of gain versus frequency.

A more useful and more common way to plot frequency response, however, is to plot gain in dB versus frequency on a log scale. This is called a Bode plot.

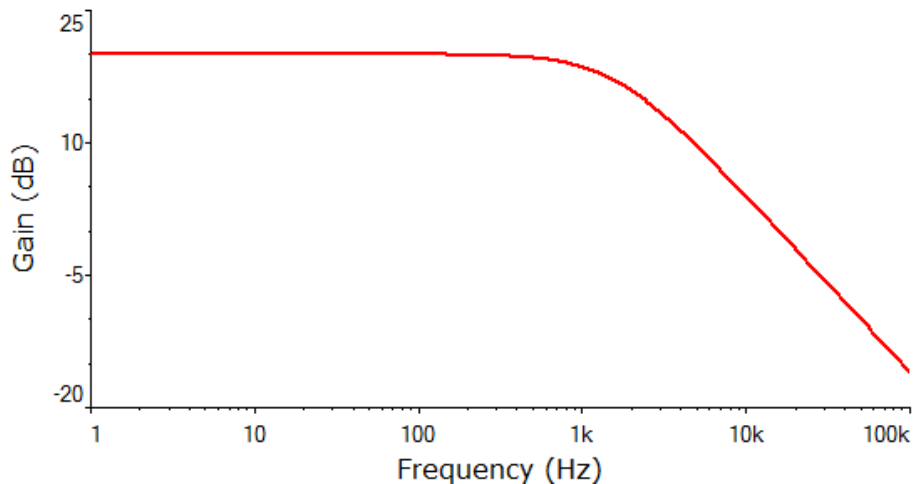


Figure 8. Bode plot of gain in dB versus frequency on a log scale.

The point at which the gain has fallen by 3 dB is referred to as the cutoff frequency.

4.1 Bode plots in Multisim

There are two ways to generate Bode plots in Multisim. Both require somewhat arbitrarily that you excite your circuit with V_{in} = a sine wave, but it doesn't matter much what amplitude and frequency you choose.

4.1.1 The Bode plotter

The easy way to generate a Bode plot is with the Bode plotter, chosen from the same instrument toolbar where the oscilloscope is found. The input is connected to V_{in} and the output to V_{out} . Notice how the cursor can be used to find the 3 dB point. The "I" and "F" settings refer to the initial and final values on the horizontal and vertical axes; you will have to adjust these to get a usable display.

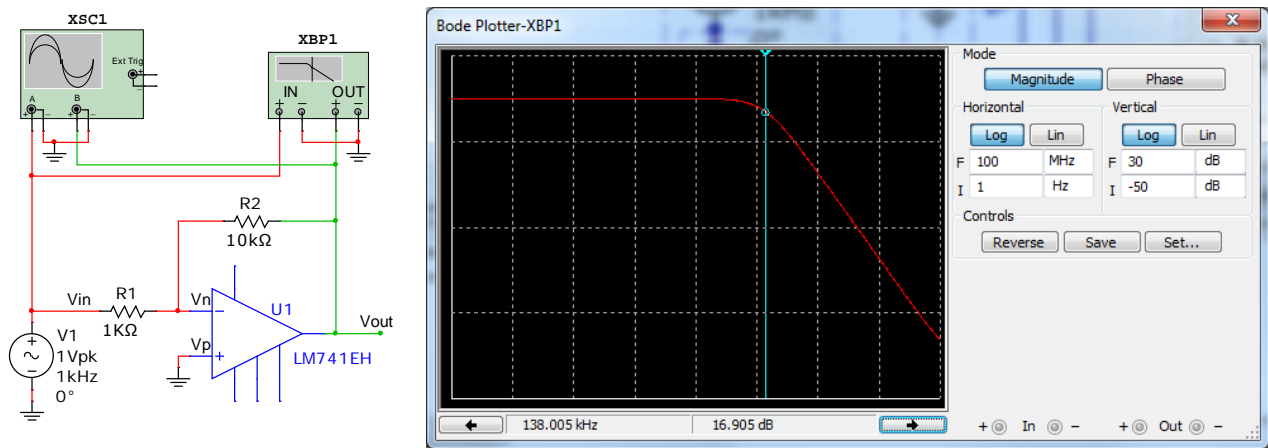
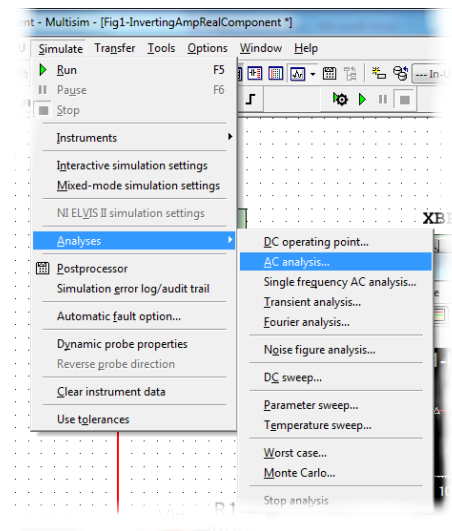


Figure 9. Using the Bode plotter.

4.1.2 AC analysis

The other way to generate a Bode plot is to via Simulate → Analyses → AC analysis from the main menu bar.

AC analysis is more complex to use but offers lots of options for formatting your plot. You can change the width and color of the trace, edit the labels on the axes, choose fonts, etc., options you don't get with the Bode plotter. This makes AC analysis more suitable if you'd like to produce graphs for a presentation-quality report. (This is how figures 7 and 8 were done.)



Here I've set AC Analysis to generate a sweep from 100 Hz to 1 MHz, plotting the output in dB versus frequency on a decade scale.

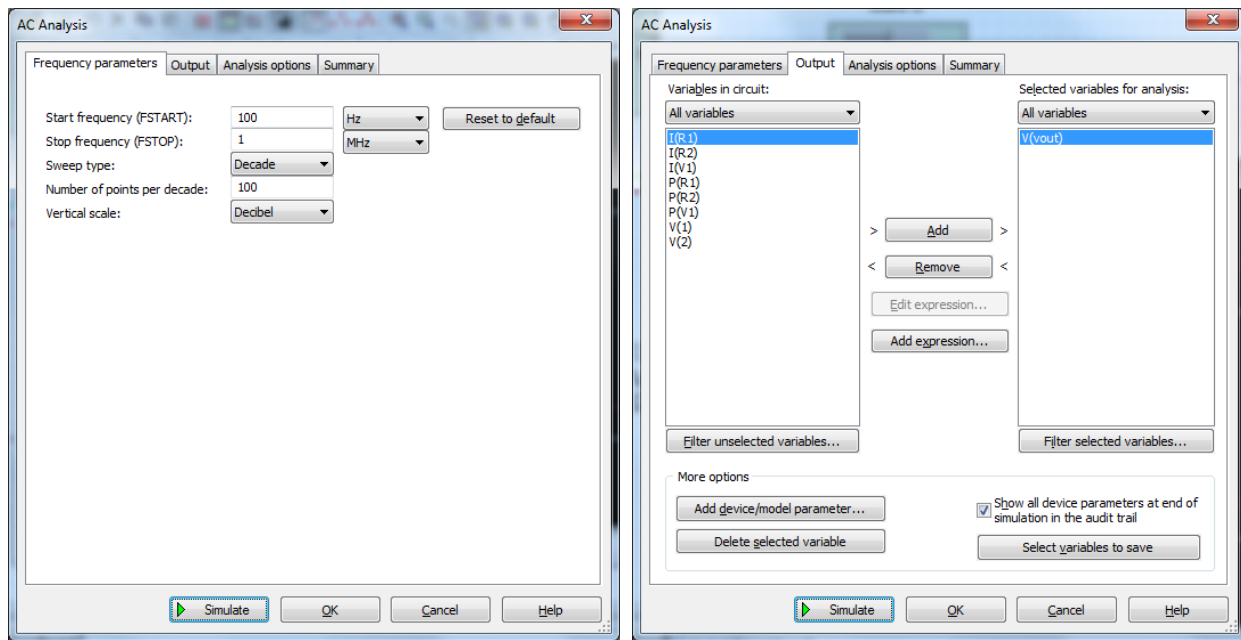


Figure 10. Configuring AC analysis.

Pressing Simulate, here is a typical result. From there, you can customize it any way you like.

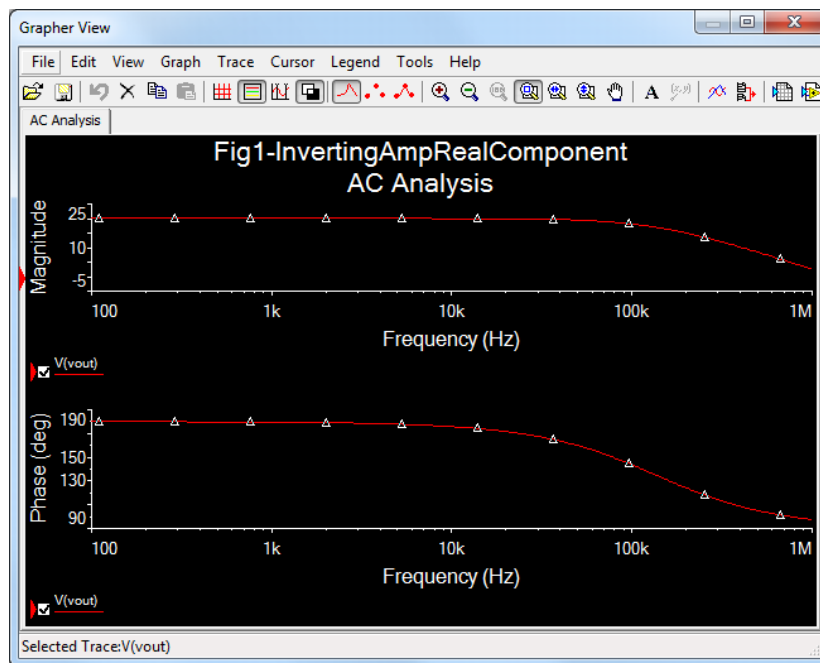


Figure 11. A typical initial AC analysis result before customizing the format.

5 Inverting amplifier

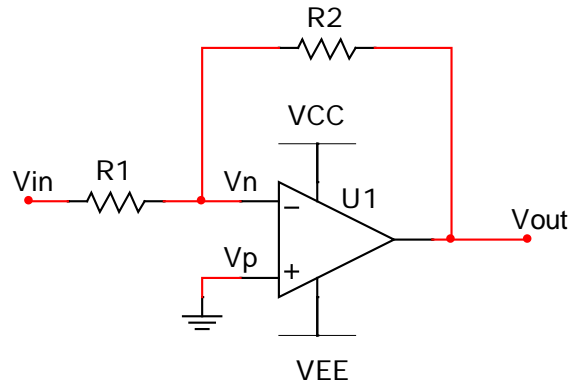


Figure 12. Inverting amplifier.

1. Analyze the inverting amplifier circuit in figure 12 (same as figure 1) to derive an equation for the gain, A_v , as a function of R_1 and R_2 .
2. Choose values for R_1 and R_2 to produce a gain, $A_v = -10$.
3. Simulate your circuit. Capture images of the following.
 - a. Your schematic.
 - b. An oscilloscope display of V_{in} and V_{out} for $V_{in} = 1.0$ Vpp, 1 KHz sine wave, 0 V DC offset.
 - c. A Bode plot of the frequency response with a cursor at the cutoff frequency.
4. Breadboard the circuit and capture screenshots with $V_{in} = 1.0$ Vpp sine wave, 0 V DC offset, at 1 KHz, 100 KHz and 1 MHz.
5. Find the cutoff frequency at which gain has dropped by 3 dB and capture a screenshot.
6. Create a table of measurements at increasing frequencies from 100 Hz to 1 MHz in a 1-2-5-10 sequence. Try to hold V_{in} fairly constant as you increase the frequency.

Frequency (Hz)	V_{in}	V_{out}	Gain (A_v)	Gain (dB)
100				
200				
500				
1K				
2K				
:				
500K				
1M				

7. Create a Bode plot of Gain(dB) versus frequency from your measurements.
8. Compare the Bode plots and cutoff frequencies based on your simulation with your experimental results and explain any differences.

6 Simple integrator

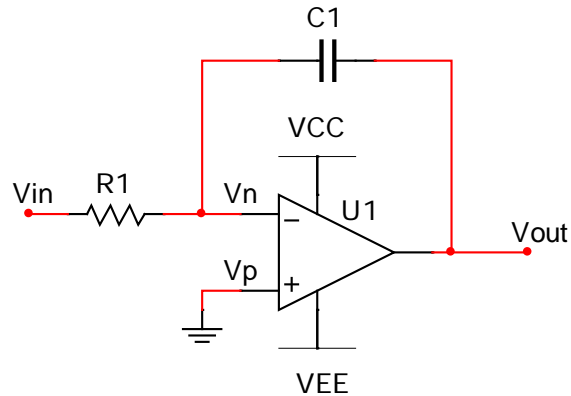


Figure 13. Simple integrator.

1. Analyze the integrator circuit in figure 13 (same as figure 2) to derive an equation for $V_{out}(t)$ as a function of $V_{in}(t)$. Show that the output is the integral of the input.

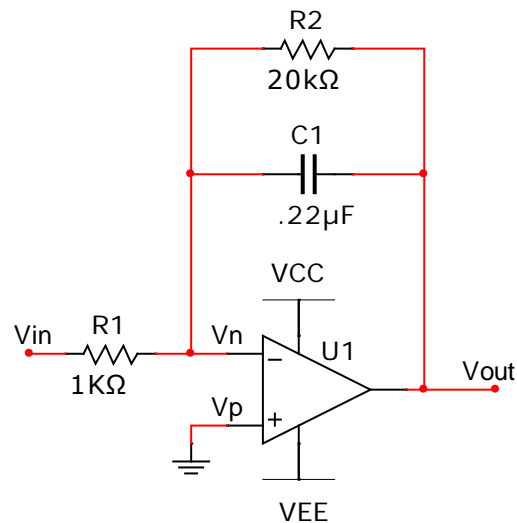


Figure 14. Integrator with shunt.

2. Analyze the circuit in figure 14 (same as figure 4) to calculate the low-frequency gain. Show that for $\omega \gg \frac{1}{R_2 C_1}$, this circuit also functions as integrator.
3. Simulate this circuit and capture images of your schematic and of oscilloscope displays of V_{in} and V_{out} with V_{in} set to be a sine wave, a square wave and a triangle wave. In each case, set $V_{in} = 1.0$ Vpp at 1 KHz, 0 V DC offset.
4. Build this circuit and capture screenshots with V_{in} set to be a sine wave, a square wave and a triangle wave = 1.0 Vpp at 1 KHz, 0 V DC offset.

- Reset $V_{in} = 0.5 \text{ Vpp}$ sine wave at 1 KHz, 0 V DC offset. While it's running, watch the oscilloscope as you remove R2 and then put it back. Describe what you observed. What is the purpose of R2?

7 Simple differentiator

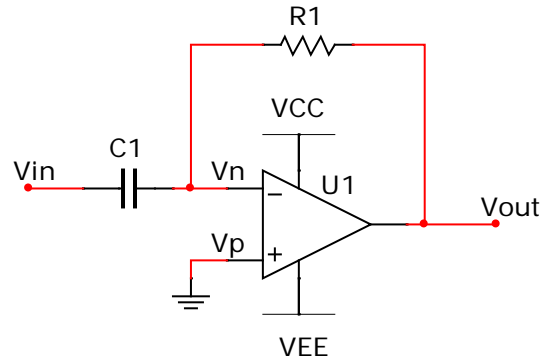


Figure 15. Simple differentiator.

- Analyze the differentiator circuit in figure 15 (same as figure 3) to derive an equation for $V_{out}(t)$ as a function of $V_{in}(t)$. Show that the output is the derivative of the input.
- Simulate this circuit with $R1 = 1 \text{ K}\Omega$ and $C1 = 0.22 \text{ }\mu\text{F}$ and capture images of your schematic and of oscilloscope displays of V_{in} and V_{out} with V_{in} set to be a sine wave, a square wave and a triangle wave. In each case, set $V_{in} = 1.0 \text{ Vpp}$ at 1 KHz, 0 V DC offset.
- Build this circuit with $R1 = 1 \text{ K}\Omega$ and $C1 = 0.22 \text{ }\mu\text{F}$ and capture screenshots with V_{in} set to be a sine wave, a square wave and a triangle wave = 1.0 Vpp at 1 KHz, 0 V DC offset.
- Describe what you observe. Does the circuit work as you expected? Can you explain the behavior?

8 Low pass filter

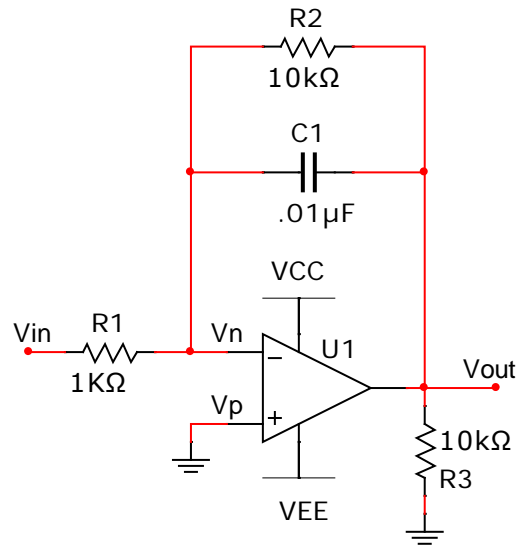


Figure 16. Low-pass filter.

1. Analyze the low-pass filter circuit in figure 16 (same as figure 5) to show that for $V_{in}(t) = A\cos(\omega t)$:

$$V_{out}(t) = -\frac{R_2}{R_1} A \cos(\omega t) \left(\frac{1 - j\omega R_2 C_1}{1 + \omega^2 R_2^2 C_1^2} \right)$$

2. Show that for the component values shown, the voltage gain, A_v , of this circuit is:

$$A_v = \frac{V_{out}}{V_{in}} = -\frac{10^5}{\sqrt{10^8 + \omega^2}}$$

3. Create a Bode plot of the expected gain from 1 Hz to 100 KHz and calculate the expected cutoff frequency.
4. Simulate your circuit. Capture images of the following.
 - a. Your schematic.
 - b. An oscilloscope display of V_{in} and V_{out} for $V_{in} = 1.0$ Vpp, 1 KHz sine wave, 0 V DC offset.
 - c. A Bode plot of the frequency response with a cursor at the 3 dB point.
5. Breadboard the circuit and capture screenshots with $V_{in} = 1.0$ Vpp sine wave, 0 V DC offset, at 100 Hz, 1 KHz and 10 KHz.
6. Find the frequency at which gain has dropped by 3 dB and capture a screenshot.

7. Create a table of measurements at increasing frequencies from 100 Hz to 100 KHz in a 1-2-5-10 sequence. Try to hold V_{in} fairly constant as you increase the frequency.

Frequency (Hz)	V_{in}	V_{out}	Gain (A_v)	Gain (dB)
100				
200				
500				
1K				
2K				
:				
50K				
100K				

8. Create a Bode plot of Gain(dB) versus frequency from your measurements.
9. Compare your three Bode plots and cutoff frequencies based on your calculations, your simulation and your experimental results and explain any differences.