EE 233 Circuit Theory

Lab 3: Simple Filters

Instructor: Tai-Chang Chen

Contents

1. Introduction

The objectives of this lab are to understand the Bode plots of electronic circuits, such as integrators and differentiators, learn SPICE simulation to design electronic circuits, and analyze and measure characteristics of simple analog amplifiers built with op-amps.

The circuits that are built in this lab are also part of the audio mixer system, so you can keep all of them for the later lab. To be more specific, the preamplifier and output summing amplifier in the equalizer is going to be built in this lab and you need to carefully arrange them on your breadboard. Please read Supplemental Material Audio Mixer for more information of the whole project.

2. Precautions

None of the devices used in this set of experiment are particularly static sensitive; nevertheless, you should pay close attention to the circuit connections and to the polarity of the power supplies, operational amplifier and oscilloscope inputs.

3. Prelab

In this procedure, the default power supply voltage to the op-amp is ± 12 V.

3.1 Inverting Amplifier

1. Calculate the gain $V_{out}(s)/V_{in}(s)$ for the circuit in Figure 1, and explain the reason why this circuit is called an inverting amplifier.

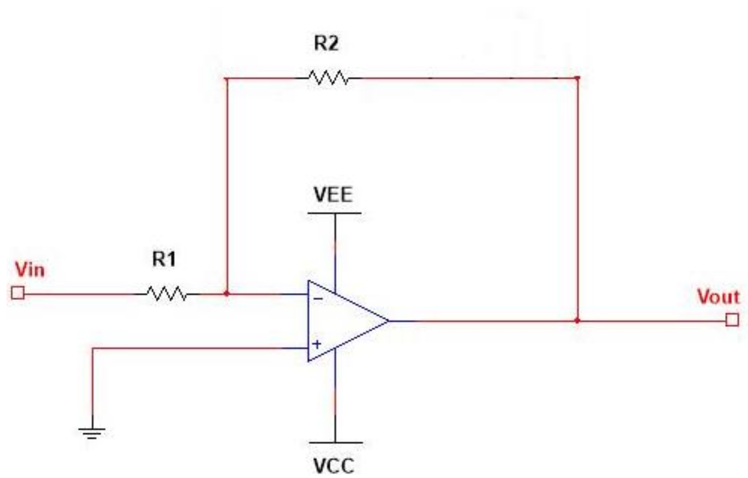

Figure 1 Inverting Amplifier

2. Design an inverting amplifier that has a gain of -47 (this gain is negative). Pick resistor values that you have in the lab kit. Include a schematic of this circuit with the component values labeled with your completed prelab.

3. Plot a Bode plot from 20 Hz to 20 kHz for your inverting amplifier with chosen resistor values in item 2. Read Reference 5.1 Bode Plot for more information.

4. Simulate this inverting amplifier circuit with MultiSim to make sure the circuit works as designed. Include the Bode plot (AC analysis) generated by MultiSim in your completed prelab. Comment on any differences from the results in item 3 and explain which result is closer to a practical result.

Hint: Read the Bode plots in the datasheet of the op-amp and comment on which result is closer to a practical result.

5. Design a non-inverting amplifier so that it has a gain of +48 (this gain is positive). Pick resistor values that you have in the lab kit. Include a schematic of this circuit with the component values labeled with your completed prelab assignment.

6. Simulate this non-inverting amplifier circuit with MultiSim to make sure the circuit works as designed. Include the Bode plot generated by MultiSim in you completed prelab.

3.2 Integrator

The resistors in the inverting amplifiers could be substituted by any type of impedance to achieve other functions rather than simple constant positive or negative gain. Now change the resistor in the feedback loop to a capacitor, as that in Figure 2, and it becomes a simple integrator.

1. Write the equation for output voltage $V_{out}(s)$ in Figure 2, in terms of $V_{in}(s), R_1$, and C. Then write the time-domain equation for $v_{out}(t)$ with $v_{in}(t), R_1$, and C; then show that the circuit performs the function of an integrator.

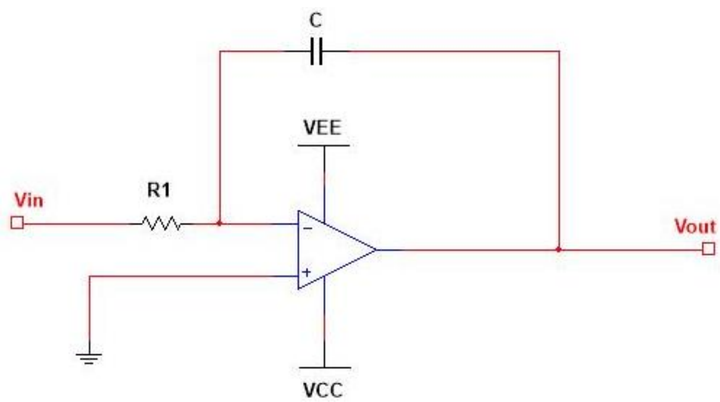

Figure 2 Simple Integrator

2. Use MultiSim transient analysis to simulate the circuit in the time domain using a sine wave input with an amplitude of 5 V and a frequency of 10 kHz, with capacitor $C = 47$ pF and the resistor you chose in 3.1 item 2. From the MultiSim output plot of the input and output waveforms, confirm that this circuit is an integrator.

   Hint: The output signal takes time to reach steady state, so you might see unexpected waveforms in the beginning. To find the steady state output, you could use the oscilloscope in the software or set the time in transient analysis much later than zero. The integration of a sine wave should be a cosine wave.

3. Use MultiSim transient analysis to simulate the circuit in the time domain using a sine wave input with an amplitude of 100 mV and a frequency of 1 kHz, with the same capacitor and resistor you chose in 3.1 item 2. From the SPICE output plot of the input and output waveforms, what does the output voltage look like? What is the maximum and minimum voltage of the output signal? If you change the power supply voltage V_{CC} and V_{EE} to $\pm 100\ V$, what is the maximum and minimum voltage of the output signal? Explain the reason for why the output waveform is distorted.

Hint: The output signal takes time to reach steady state, so you might see unexpected waveforms in the beginning. To find the steady state output, you could use the oscilloscope in the software or set the time in transient analysis much later than zero.

4. Use MultiSim AC analysis to simulate this simple integrator circuit with the same resistor and capacitor, and include the Bode plot generated by MultiSim in your completed prelab.

5. Now add another resistor parallel to the capacitor in the circuit and it becomes Figure 3. Write the equation for output voltage $V_{out}(s)$ with $V_{in}(s), R_1, R_2$, and C. Then write the time-domain equation for $v_{out}(t)$ with $v_{in}(t)$, R_1, R_2, and C; then show that the circuit performs the function of an integrator.

   Hint: The inverse Laplace Transform $\boldsymbol{L^{-1}\left\{\frac{F(s)}{s+a}\right\} = e^{-at}\int_0^t f(\tau)e^{a\tau}d\tau}$, where $\boldsymbol{L}\{f(t)\} = F(s)$.

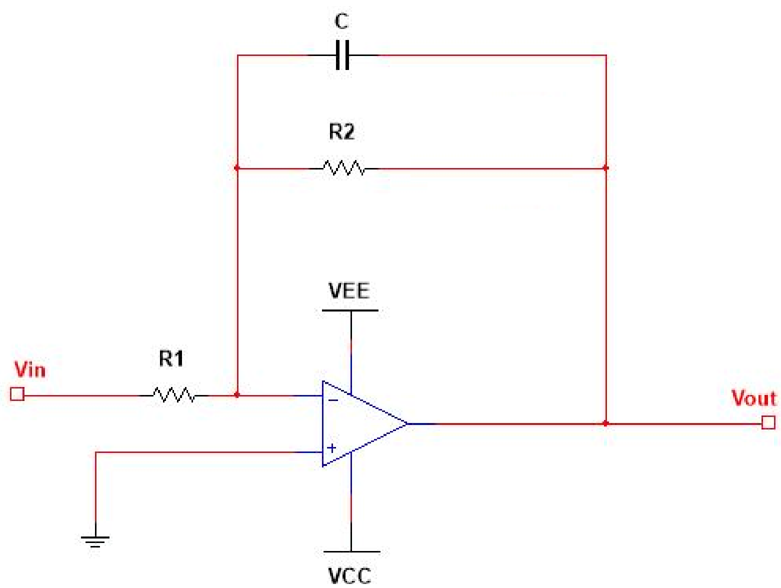

Figure 3 Integrator with Shunt Resistance

6. Now analyze the circuit in Figure 3 in the frequency domain. What is the magnitude of the gain of this circuit? Draw the Bode plot from 20 Hz to 20 kHz with capacitor $C = 47$ pF and the resistors you chose in 3.1 item 2.

7. (optional) What is the low-frequency gain of this circuit in the frequency domain? Explain why it is the same as the gain of an inverting amplifier. Compare this Bode plot with the plot for the inverting amplifier and explain any differences between them.

   Hint: Low frequency means $\omega \to 0$ Hz.

8. (optional) What is the high frequency gain for this circuit in the frequency domain? Show that the circuit performs the function of an integrator for high frequency and explain why its gain is the same as the gain of a simple integrator at high frequency.

   Hint: High frequency means $\omega \gg 1/(R_2C)$.

9. (optional)Explain the function of the resistor R_2 in the circuit, comparing to the simple integrator.

Hint: Compare the difference of gain with and without the shunt resistor R_2, especially at low frequency.

10. Use MultiSim to simulate the frequency response of the integrator with shunt resistance. If the input signal has a low frequency what is the expected gain? If the input signal has a high frequency what is the expected gain from these plots? Comment on whether this circuit is a low-pass, high-pass or band-pass filter. Explain the reason why this circuit is a better choice for preamplifier than a simple inverting or non-inverting amplifier.

Hint: Consider noise from the input signal. Read Reference 5.2 Noise for more details about noise.

3.3 Simple Differentiator

1. Change the resistor that is connecting the input source to the inverting amplifier into a capacitor. Calculate the output $V_{out}(s)$ for the circuit in Figure 4 with $V_{in}(s)$ and R_1.

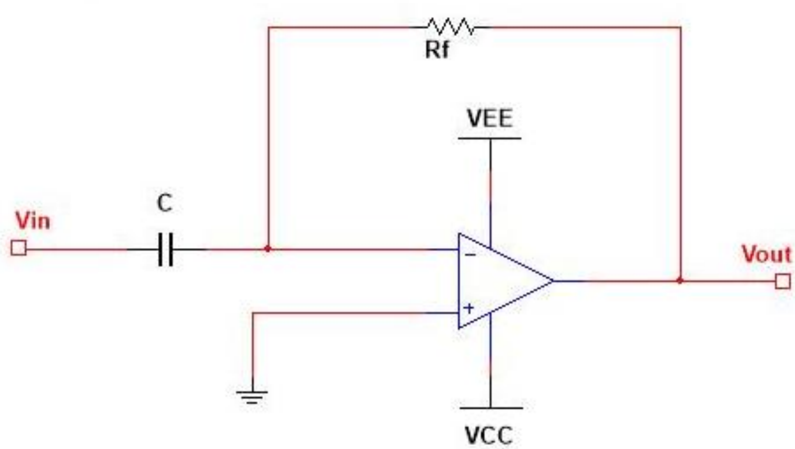

Figure 4 Simple Differentiator

2. Derive the time-domain equation for $v_{out}(t)$ with $v_{in}(t), R_f$, and C. Show that the circuit performs the function of a differentiator.

3. Draw the Bode plot for this circuit from 20 Hz to 20 kHz with $R_f = 50$ kΩ and $C = 0.1$ μF and include the plot in your prelab.

4. Use SPICE AC analysis to simulate this circuit with the same resistor and capacitor in 3.3 item 3 from 20 Hz to 20 kHz and include the Bode plot in your completed prelab.

5. Use MultiSim transient analysis to simulate this circuit in the time domain using a sine wave input with an amplitude of 100 mV and a frequency of 1 kHz, with the same resistor and capacitor. From the SPICE output plot of the input and output waveforms, confirm that this circuit is an integrator.

Hint: The output signal takes time to reach steady state, so you might see unexpected waveforms in the beginning. To find the steady state output, you could use the oscilloscope in the software or set the time in transient analysis much later than zero.

4. Experimental Procedure and Data Analysis

In this procedure, the preamplifier and summing amplifier for your equalizer are going to be built. Keep what is built on the breadboard for later experiments. Before moving on to the next lab, let your TA check your circuit to make sure that it is working well.

4.1 Preamplifier for the Equalizer

1. Build the circuit in Figure 4.1 with power supplies $V_{CC} = 12\ V, V_{EE} = -12\ V$. Set $R_1 = 1$ kΩ, $R_2 =$ 47 kΩ, and $C = 47$ pF; then choose a 50 kΩ potentiometer and set it to zero in the circuit. Sweep the frequency of V_{in} starting at 10 Hz, then vary it using the 1-2-5 sequence up to 1 MHz while keeping the amplitude at 100 mV. Record the amplitude and phase change of the output signal.

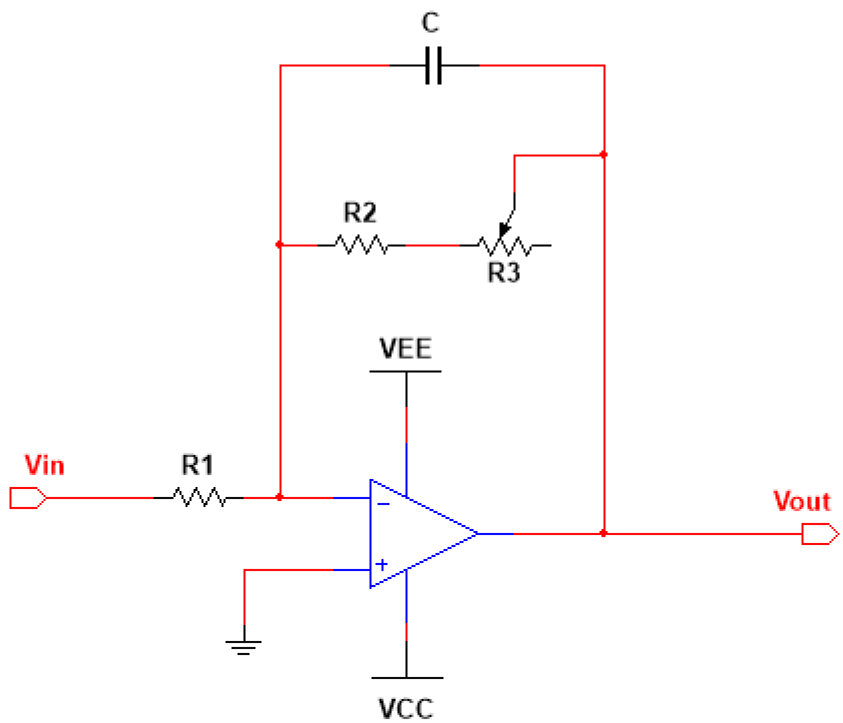

Figure 4.1 Preamplifier Circuit

Question-1: Draw the Bode plot from 20 Hz to 20 kHz and compare it to the results in 3.2 item 6.

2. Set the potentiometer to 50 kΩ and sweep the frequency of V_{in} starting at 10 Hz, then vary it using the 1-2-5 sequence up to 1 MHz while keeping the amplitude the same. Record the amplitude and phase change of the output signal.

   Question-2: Draw the Bode plot from 20 Hz to 20 kHz. Compare the plot with that in 4.1 item 1 and comment on the difference caused by changing the potentiometer.

3. Apply a sine wave input signal with an amplitude of 300 mV and a frequency of 300 Hz. Display the input signal on Channel 1 of the oscilloscope and the output signal on Channel 2. Adjust the time base to display 2-3 complete cycles of the signals.

Question-3: Get a hardcopy of output from the scope display with both waveforms to confirm that the circuit is an integrator. Turn this hardcopy in as part of your lab report.

4.2 Summing Amplifier for the Equalizer

1. Build the circuit in Figure 4.2 with power supplies $V_{CC} = 12$ V, $V_{EE} = -12$ V. Set the capacitor to 0.1 μF and all resistors to 100 kΩ potentiometers. Set all potentiometers to 100 kΩ. Apply the function generator to V_1 (V_2 and V_3 are disconnected) with an amplitude of 500 mV. Sweep the frequency of V_{in} starting at 10 Hz, then vary it using the 1-2-5 sequence up to 1 MHz while keeping the amplitude the same. Record the amplitude and phase change of the output signal.

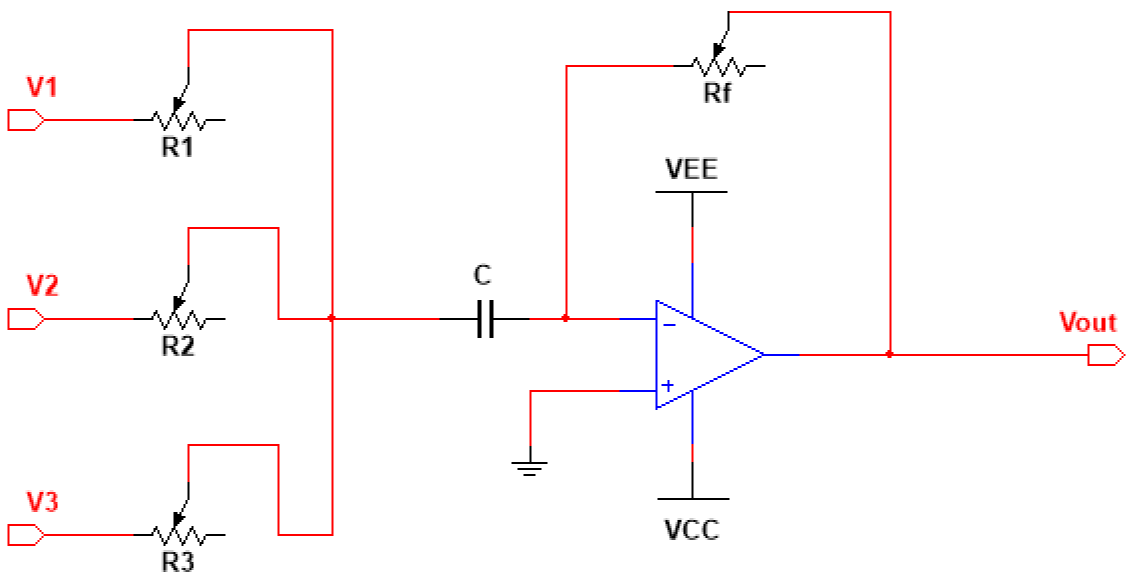

Figure 4.2 Output summing amplifier

Question-4: Draw the Bode plot from 10 Hz to 20 kHz and comment on whether it is a low-pass, high-pass or band-pass filter.

2. Set $R_1 = 0\ \Omega$ and $R_f = 50$ kΩ. Apply a sine wave input signal to V_1 with an amplitude of 300 mV and a frequency of 300 Hz. Display the input signal on Channel 1 of the oscilloscope and the output signal on Channel 2. Adjust the time base to display 2-3 complete cycles of the signals.

   Question-5: Get a hardcopy of the output from the scope display with both waveforms to confirm that the circuit is a differentiator. Turn this hardcopy in as part of your lab report.

3. Apply a sine wave input signal to V_1 with an amplitude of 1 V and a frequency of 1 kHz and display the output signal in the oscilloscope. Set $R_1 = 1$ kΩ and increase R_f until the output waveform becomes distorted.

   Question-6: Explain the reason why it becomes distorted and comment on how to avoid such distortion.

   Hint: The distorted sine waveform should become flat on top and bottom. Record the maximum and minimum value of the distorted waveform and compare them with V_{cc} and V_{EE}.

4. Apply a low frequency sine wave to V_1 and connect V_{out} to speaker. Listen to the sound in the speaker. Then remove the capacitor and listen to the sound again.

   Question-7: Explain the function of the capacitor in the output summing amplifier.

4.3 Mixer and Microphone

In this procedure you are ready to play with the mixing console to mix signals up. Connect the speaker to the output terminal of the output summing amplifier and use headphone jacks to provide three channels of signals from your laptops, PCs or cell phones. Listen to the sound of the speaker and tell whether the three signals are mixed.

Now use the potentiometers in the output summing amplifier to change the whole volume and the ratio of volumes between three channels. This is exactly the same thing as what happens in audio studios. Let your TA check your sounds to make sure your mixer works well.

You are also encouraged to build a microphone circuit as one of the input channels. Since the electronic signal from microphone is too small, the preamplifier would be utilized in this channel right after the microphone circuit. The microphone circuit is an extra credit assignment, and it will show you how a microphone works. After you build it, you can see how the acoustic signal could be amplified and hear your own sounds from the speaker.

Extra Credit

This is an extra credit assignment that involves building a microphone circuit on the breadboard. Read Reference 5.3 Microphone for more information on the microphone circuit. Build the circuit in Figure 4.3 and provide power source $V_s = 12$ V and $C = 1$ μF. Connect the output signal to preamplifier, and then to the speaker. Play a sound from a cell phone, harmonica, guitar or any other instrument into your microphone, or you can sing to the microphone and listen to the sound from the speaker.

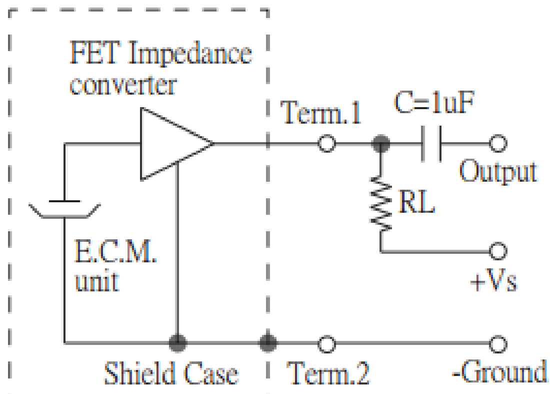

Figure 4.3 Microphone circuit

Question-E: Is there any noise in the speaker? Where does the noise come from? Find a simple method to remove the loud noise.

Hint: Consider the speaker as a resistor load R_L. The DC voltage or current will produce sound as well as signal voltage or current. Read Reference 5.3 Microphone for more information.

5. Reference

5.1 Bode Plot

The Bode plot is a graph of the transfer function of a system versus frequency. It is usually plotted with log-scale frequency axis to show the frequency response of the system. It is a very useful way to represent the gain and phase of a system as a function of frequency. This is referred as the frequency domain behavior of a system.

To be more specific, a linear, time-invariant electronic system could be represented by its transfer function $H(j\omega) = V_{out}(j\omega)/V_{in}(j\omega)$ in the frequency domain. The magnitude of the transfer function $|H(j\omega)|$ is the gain versus frequency while the angle of the transfer function on the complex plane $tan^{-1}(Im\{H(j\omega)\}/Re\{H(j\omega)\})$ represents the change of phase from input to output signal.

For example, the output voltage over the capacitor is $V_{out}(j\omega) = \frac{V_{in}(j\omega)}{1+j\omega RC}$ so that the transfer function is $H(j\omega) = \frac{V_{out}(j\omega)}{V_{in}(j\omega)} = \frac{1}{1+j\omega RC}$. The gain is $|H(j\omega)| = \left|\frac{1}{1+j\omega RC}\right| = \frac{1}{\sqrt{1+\omega^2R^2C^2}}$ and the phase is $\angle H(j\omega) = tan^{-1}(Im\{H(j\omega)\}/Re\{H(j\omega)\}) = tan^{-1}(\frac{-\frac{\omega RC}{1+\omega^2R^2C^2}}{\frac{1}{1+\omega^2R^2C^2}}) = tan^{-1}(-\omega RC)$. Then you could draw the Bode plot with these two equations. Notice log scale should be utilized in frequency axis.

5.2 Noise

Noise is a random fluctuation in an electrical signal that happens in all electronic circuits. There are three main types of noise in electronic systems: thermal noise, shot noise, and 1/f noise. Thermal noise is approximately white noise, meaning that its power spectral density is nearly equal throughout the frequency spectrum.

To relatively reduce the power of noise all input signals, including noise, could be amplified using the preamplifier. This process reduces the parts of the noise at the frequencies that are not going to be used, effectively quieting the noise. Thus, the signal will become more dominant in the system.

5.3 Microphone

The microphone in your lab kit has two pins, and those pins have polarity. To use the microphone a resistor, a capacitor and a power supply are usually used to provide power, as shown in Figure 5.1. The frequency response of microphone you are going to use a almost the same between 50Hz to 5kHz. The power supply is used to activate the microphone device. However, the output voltage has a DC bias, which will cause a loud noise to come from the speaker. Therefore, a capacitor is added before the output to remove DC noise.

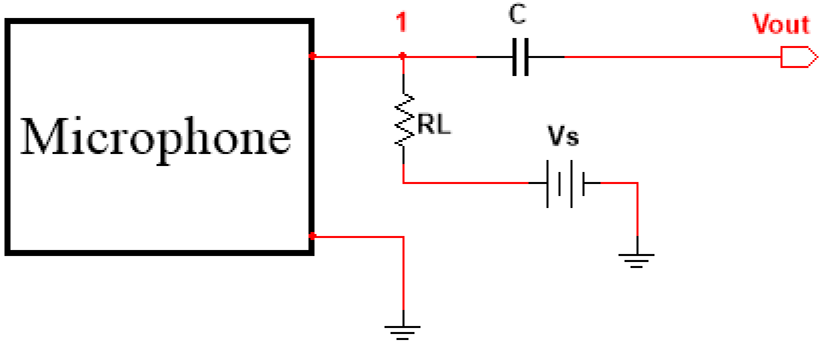

Figure 5.1 Microphone circuit scheme