

# *Experiment 4:*

## *AC Steady-State Circuit Analysis and Passive Analog Filters*

### **Objectives**

Students will:

- Gain experience in the application of AC steady-state circuit analysis techniques.
- Simulate some simple passive analog filters, and further investigate them by breadboard construction and testing.

### **Learning Outcomes**

Students will be able to:

- Use LTspice to simulate the transient response of  $RC$ ,  $RL$ , and  $RLC$  circuits with sinusoidal input.
- Use LTspice to simulate the AC frequency sweep response of  $RC$ ,  $RL$ , and  $RLC$  circuits, determining both the magnitude and the phase angle of input and output voltages.
- Determine the complex transfer functions for some simple  $RC$ ,  $RL$ , and  $RLC$  circuits and investigate their behavior for high and low-frequency sinusoidal signals.
- Define what is meant by high and low frequencies in the context of  $RC$ ,  $RL$ , and  $RLC$  circuits.
- Classify  $RC$ ,  $RL$ , or  $RLC$  circuits as low-pass, high-pass, band-pass, or band-stop filters.
- Determine the cutoff frequency for  $RC$  and  $RL$  circuits and the resonance frequency for  $RLC$  circuits.
- Understand a practical (non-ideal) model for a real inductor and determine the range of frequencies for which the real inductor behaves nearly like an ideal inductor.
- Measure capacitance using a digital multimeter.

### **Equipment Required**

- Computer with Windows operating system
- Fritzing software (optional)
- LTspice software
- Analog Devices ADALM2000 (M2K) Advanced Active Learning Module with Scopy software and probe cable
- DMM (Digital Multimeter) with Capacitance scale
- Breadboard
- Components:
  - $1\text{k}\Omega$  resistor
  - $1\mu\text{F}$  capacitor
  - $0.1\mu\text{F}$  (104) capacitor
  - $1\text{mH}$  inductor

## Prerequisites

Before doing this experiment, students should be able to:

- Determine the voltage transfer function of a two-resistor voltage divider.
- Determine the real and imaginary parts of a complex number, write complex numbers in polar form, and locate a complex number in the complex plane.
- Determine the impedance of capacitors and inductors from their component values and the operating frequency of the applied power source.
- Determine the values of capacitors and inductors from the information printed on them.

## Pre-Lab Exercises

- Read through this document so you know what is expected.
- Several circuits will be physically built and investigated during this experiment. Prepare a *breadboard layout* diagram for each one using Fritzing, or use the attached breadboard template to draw them by hand. Include these in your report.

## Table of Contents

Introduction

Part A: Series *RC* and *RL* Circuits

Background

Complex Numbers

Impedance

Experimental Procedure

Simulate a Series *RC* Circuit

Transient Analysis

Frequency Sweep Analysis

Simulate a Series *RL* Circuit

Transient Analysis

Frequency Sweep Analysis

Summary

Part B: *RC* and *RL* Circuit Transfer Functions and Filters

Background

Transfer Functions

Finding Transfer Functions

Filters

Using Transfer Functions to Determine Filter Behavior

Cutoff (or Corner) Frequency

Experimental Procedure

The Transfer Function Equation

The Transfer Function of an *RC* Circuit

The Transfer Function of an *RL* Circuit

Build and Test an *RC* Circuit

Summary

Part C: *RLC* Circuit Transfer Functions and Filters

Background

Phasors

RLC Circuits

More Complex Filters

Modeling Electrical Components

Experimental Procedure

Simulate a Series *RLC* Circuit

Build and Test a Series *RLC* Circuit

Summary

Part D: Equivalent Impedance

Background

Combined Impedance of Parallel Components

Experimental Procedure

Simulate a Parallel *RC* Circuit

Summary

What Should be in Your Report?

Breadboard Template

## Introduction

The key circuit concept to be investigated in this experiment is that of the voltage divider. Voltage dividers are encountered frequently in practice because almost all circuits involve some configuration with components connected in series.

Such a circuit is frequently used to provide a reference voltage for some measured characteristic and thereby control the operation of a physical system. For example, in a thermostat, a temperature sensor outputs a voltage proportional to temperature. This voltage is compared to a reference voltage that corresponds to the desired temperature in order to determine when a furnace or air conditioner should be turned off or on. The same principle is also used for lighting control based on dusk/dawn sensing systems, liquid-level control, and generally for control of anything that can be measured by a sensor that outputs a voltage.

The operation of an ideal voltage divider can be easily derived from Ohm's Law and simple circuit analysis. For the configuration shown here:

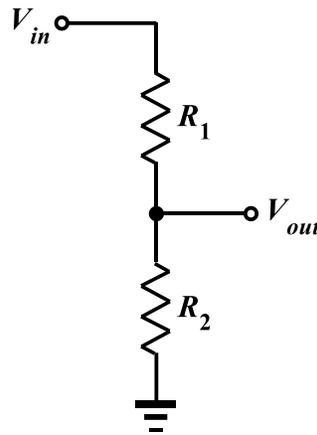


Figure 1.

the relationship between the input and output voltages is given by:

$$V_{out} = \frac{R_2}{R_1 + R_2} \cdot V_{in}$$

Clearly, given a voltage source  $V_{in}$ , a reference (output) voltage at any level in the range  $0 \leq V_{out} \leq V_{in}$  can be produced by this circuit.

Connecting an oscilloscope or meter across resistance  $R_2$  to view or measure  $V_{out}$  will add the input resistance ( $R_L$ ) of that instrument to the circuit in parallel with  $R_2$ . This consequence, shown in Figure 2, is referred to as “loading” the circuit.

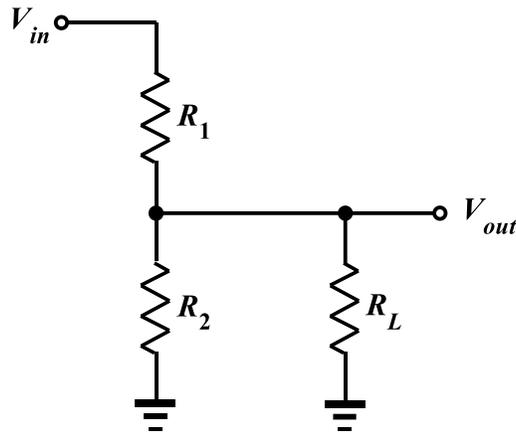


Figure 2.

Loading has the effect of changing the output voltage as shown below:

$$\begin{aligned}
 V_{out} &= \frac{(R_2 \parallel R_L)}{R_1 + (R_2 \parallel R_L)} \cdot V_{in} \\
 &= \frac{\left( \frac{R_2 R_L}{R_2 + R_L} \right)}{R_1 + \left( \frac{R_2 R_L}{R_2 + R_L} \right)} \cdot V_{in} \\
 &= \frac{R_2 R_L}{R_1 (R_2 + R_L) + R_2 R_L} \cdot V_{in} \\
 &= \frac{R_2 R_L}{R_1 R_2 + (R_1 + R_2) R_L} \cdot V_{in}
 \end{aligned}$$

To minimize this effect, the additional (load) resistance must be much larger than  $R_1 R_2 / (R_1 + R_2)$  so that

$$\lim_{R_L \rightarrow \infty} V_{out} \approx \frac{R_2 R_L}{(R_1 + R_2) R_L} \cdot V_{in} = \frac{R_2}{R_1 + R_2} \cdot V_{in}$$

Thus, we conclude that a complete model of a practical voltage divider circuit should include the load resistance unless it has been proven to have a negligible effect. That resistance must be much larger than the original divider resistors in order to avoid any problems resulting from loading.

And, the larger the load resistance, the closer the divider becomes to “ideal”.

More generally, the same result applies to any circuit like Figure 3:

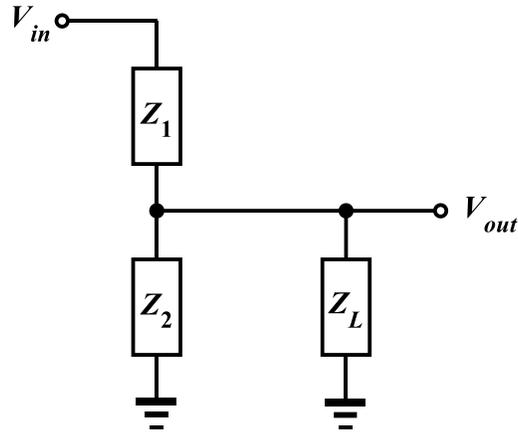


Figure 3.

When the input impedance of the connected device,  $Z_L$ , has a much larger magnitude than each of the impedances making up the voltage divider itself:

$$V_{out} \approx \frac{Z_2}{Z_1 + Z_2} \cdot V_{in}$$

## Part A: Series RC and RL Circuits

### Background

*Complex Numbers:* Complex numbers can be expressed in terms of their real and imaginary parts ( $x$  and  $y$ , respectively) as  $z = x + jy$  (called Cartesian or rectangular form), as  $z = A\angle\theta$  (polar form as shown in Figure A.1), or as  $z = Ae^{j\theta}$  (exponential form). The letter  $j$  (or  $i$  in mathematics) is used to represent  $\sqrt{-1}$ . The length ( $A$ ) of the line segment between the origin and point  $z$ , and the angle ( $\theta$ ) between this segment and the positive real axis, form the complex number:

$$z = A\cos\theta + jA\sin\theta \text{ where } A = \sqrt{x^2 + y^2} \text{ and } \theta = \tan^{-1}\left(\frac{y}{x}\right).$$

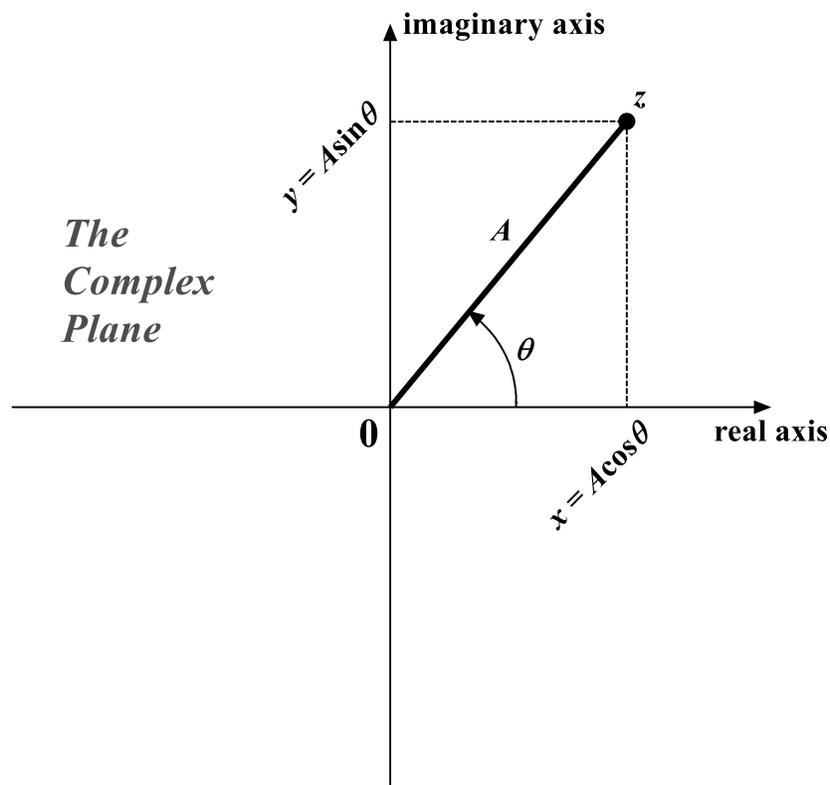


Figure A.1.

*Impedance:* Each basic circuit component ( $R$ ,  $L$  or  $C$ ) has an impedance that may affect the response of the circuit in two ways, i.e., it may change the magnitude (amplitude) and/or the phase angle (the relative position along the time axis) of the voltages and currents. Because of the mathematical similarities between the way sinusoids and complex numbers behave, we can conveniently use complex numbers to represent these relationships in circuits with sinusoidal signals. A resistor tends to affect only the amplitude because its impedance is purely real. Capacitors and inductors, on the other hand, may influence both amplitude and phase angle

because of their imaginary impedance.

We represent impedance by the letter  $Z$ , where:

- (a) for a resistor:  $Z_R = R$  is *real*,  
 (b) for an inductor:  $Z_L = j\omega L$  or  $\omega L \angle +90^\circ$  is *positive imaginary*, and  
 (c) for a capacitor:  $Z_C = \frac{1}{j\omega C} = -j\left(\frac{1}{\omega C}\right)$  or  $\frac{1}{\omega C} \angle -90^\circ$  is *negative imaginary*.

Recall that  $\omega$  is called angular frequency and is equal to  $2\pi f$ .

## Experimental Procedure

### Simulate a Series RC Circuit

*Transient Analysis:* In this section, we will use LTspice to perform a transient analysis of the RC circuit shown in Figure A.2.

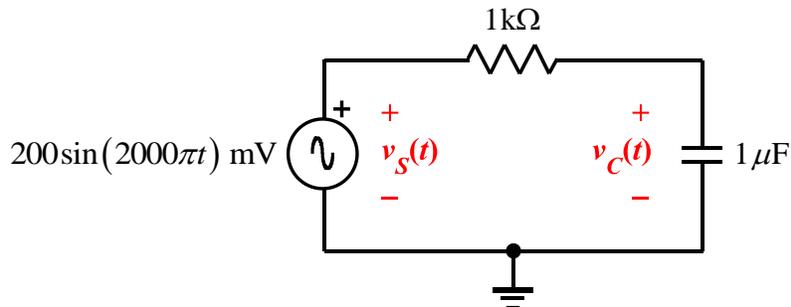


Figure A.2.

- Create the LTspice schematic diagram shown in Figure A.3.

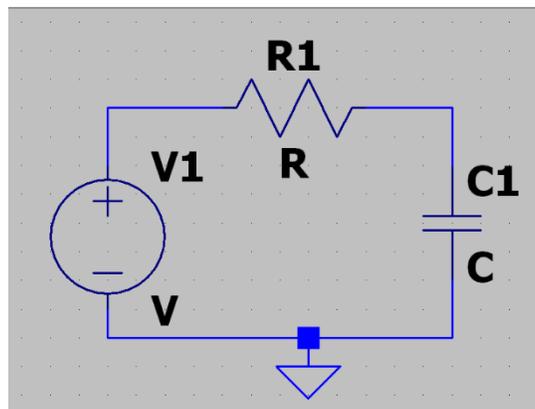


Figure A.3.

- Right-click on the resistor symbol, set its value to  $1k\Omega$  (just the “1k” part), and click “OK”.

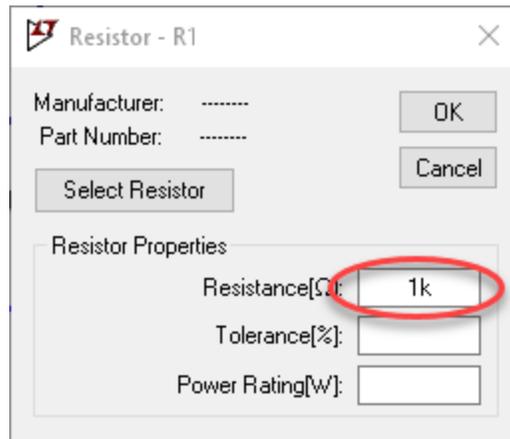


Figure A.4.

- Right-click on the capacitor symbol, set its value to  $1\mu\text{F}$  (just the “1u” part), and click “OK”.

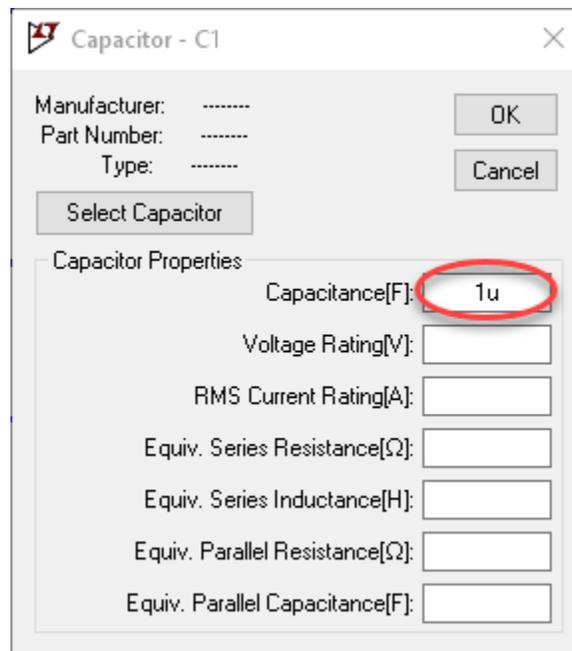


Figure A.5.

- Ctrl-right-click on the capacitor symbol to open the Component Attribute Editor. In the “SpiceLine2” row, double-click and then type “IC=0” in the Value column, then double-click in the Vis. Column to make it show an “X” in order to make that attribute visible on the schematic. Then click “OK”:

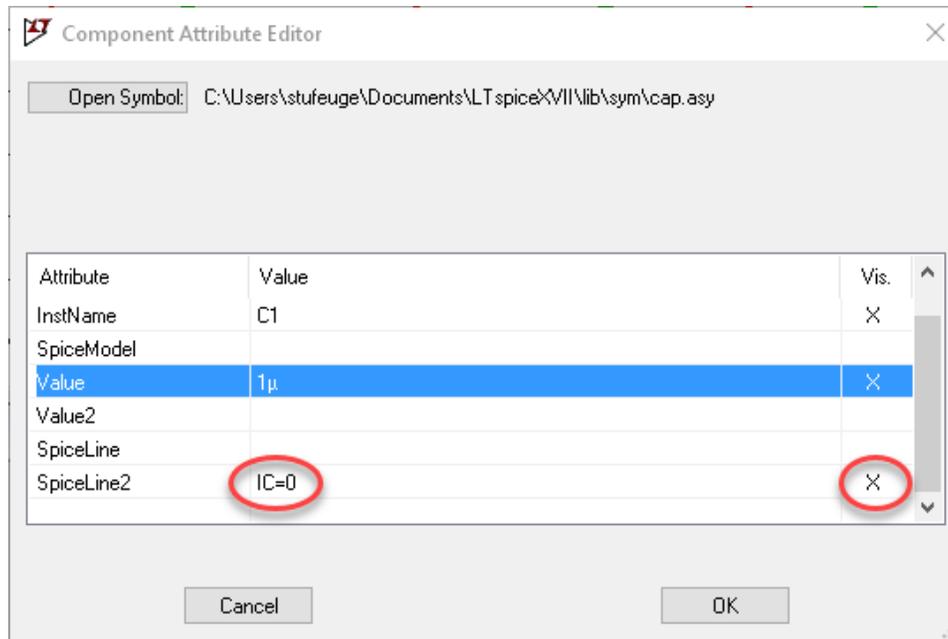


Figure A.6.

- Right-click on the voltage source symbol to open this pop-up menu:

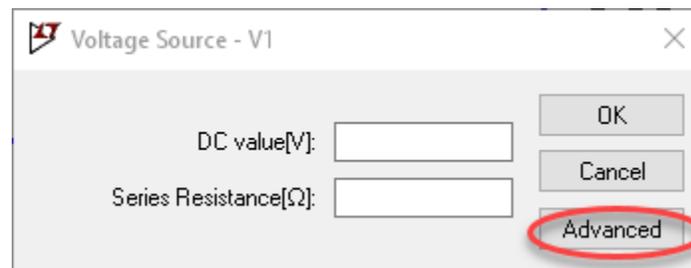


Figure A.7.

- Click on “Advanced” to open another pop-up menu to select the built-in SPICE transient SINE function, and enter the parameters necessary to describe the source you want:

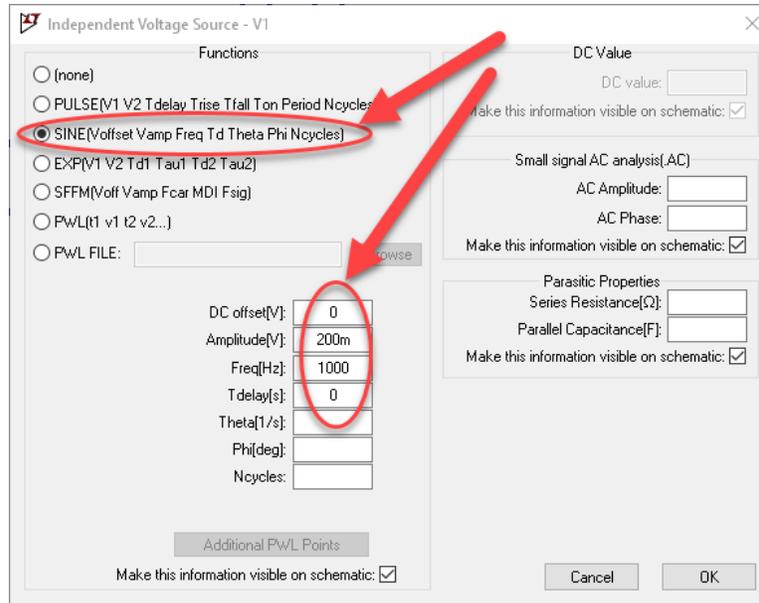


Figure A.8.

- Click “OK”. Your schematic display should now look similar to this:

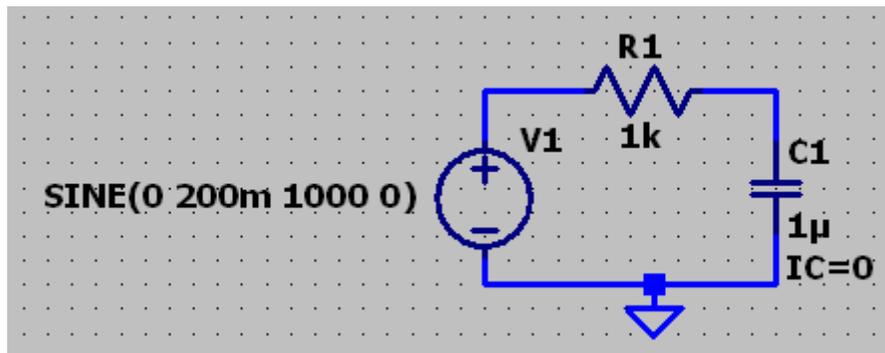


Figure A.9.

- On the upper right-hand side of the screen, in the toolbar, click on “.op” to open another pop-up box where you can enter the SPICE directive(s) necessary to accomplish the simulation. In this case we want to do a transient simulation. In the white box, type the “.TRAN” directive with parameters to run a transient analysis for 4 ms with a maximum step size of 4  $\mu$ s. Include the “UIC” option to force LTspice to use the specified initial voltage for the capacitor. (The initial capacitor voltage was set to 0 as a part of the capacitor description above, in Figure A.6.)

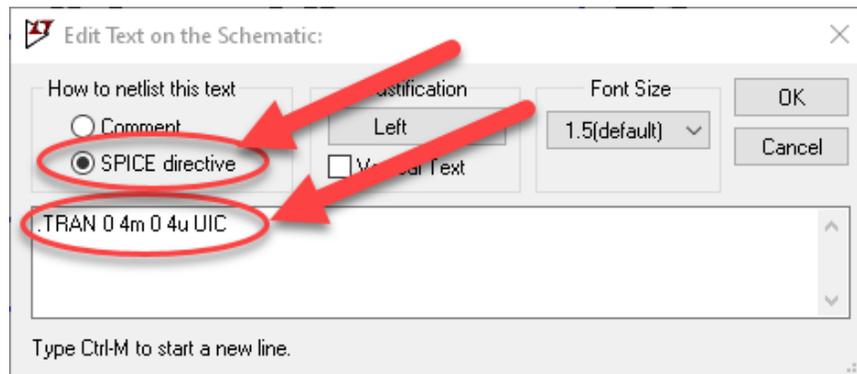


Figure A.10.

- Click “OK”. Your schematic display should now be similar to the following:

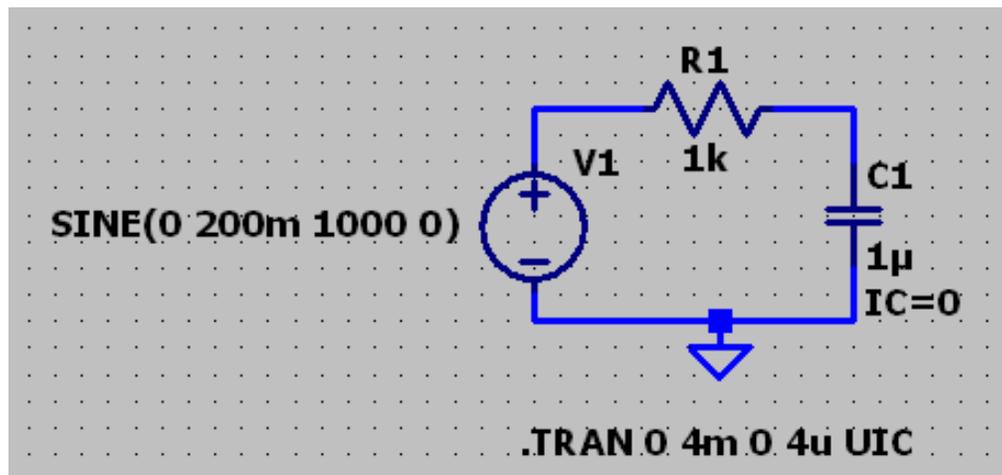


Figure A.11.

- Run the simulation (Click on  in the toolbar at the top of the screen.) and then plot  $v_s(t)$  and  $v_c(t)$  together in the plot window. You should see two sinusoids similar to this:

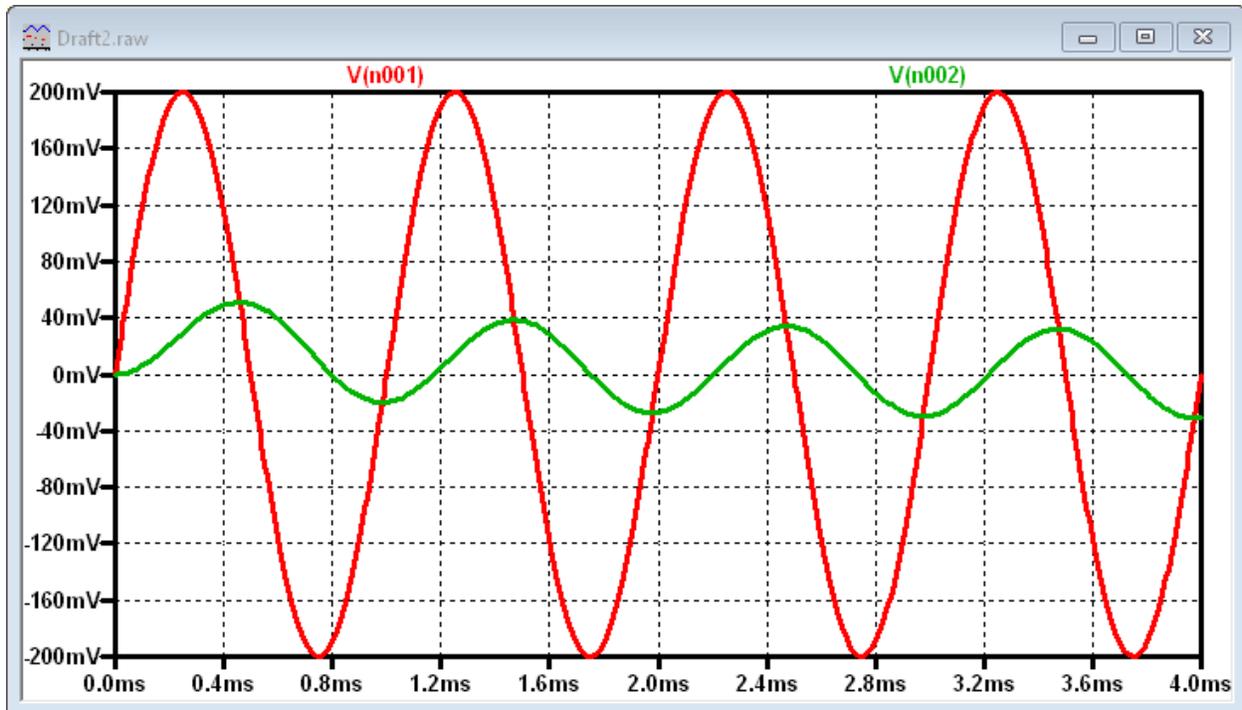


Figure A.12.

Note that the circuit has affected both the amplitude and the phase of the output. How close is the phase shift to  $-90^\circ$ ? (Note that the capacitor voltage is settling into the steady state throughout the duration of this simulation. Hence, you should *use one of the later cycles to answer this question* – See the hint discussion below.) Copy your schematic diagram and this plot and include them in your report.

*Hint/Suggestion:* The impedance expressions above for  $R$ ,  $L$  and  $C$  apply only to the AC steady-state. That is, they will describe the behavior of the voltages and currents only after the circuit has gone through its initial transients and settled into the state it will remain in forever after (its sinusoidal steady-state). Thus, when we are using transient analysis to help understand steady-state, we must ignore at least the first few cycles of the resulting sinusoids and focus on the later part of the simulation. At times the signal may not reach its steady-state in the number of cycles you are displaying. To be sure you have reached the steady-state, you could set up your simulation so that it delays showing the output until 10 ms and runs to 14 ms, or even delay showing results until 100 ms and run to 104 ms. Either of these would yield a 4-ms-long display of the resulting waveforms, and the longer the simulation the nearer the result will be to the steady-state. Because the circuits we analyze in this course are usually fairly small, the extra computer time required to run the simulation to 104 ms is not excessive.

- Try starting at 10 ms and see how things look.

```
.TRAN 0 14m 10m 4u
```

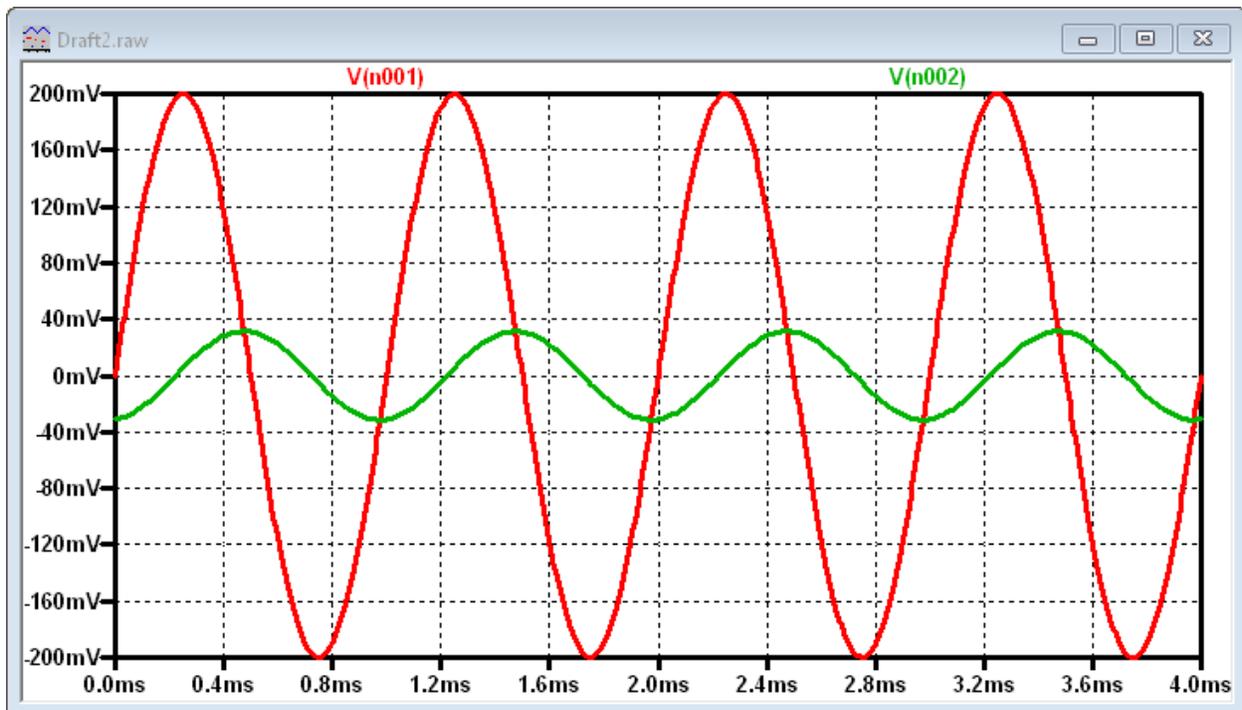


Figure A.13.

*Frequency Sweep Analysis:* LTspice has another type of analysis, called an AC sweep, that lets you look at the behavior of a circuit over a whole range of frequencies. We know that the impedance of the capacitor depends on  $\omega$ , the frequency of the input signal. Should the behavior of the circuit change at different frequencies? Let's set up an AC sweep and find out.

- Edit your simulation by clicking on “Edit Simulation Cmd” from the “Simulate” menu:

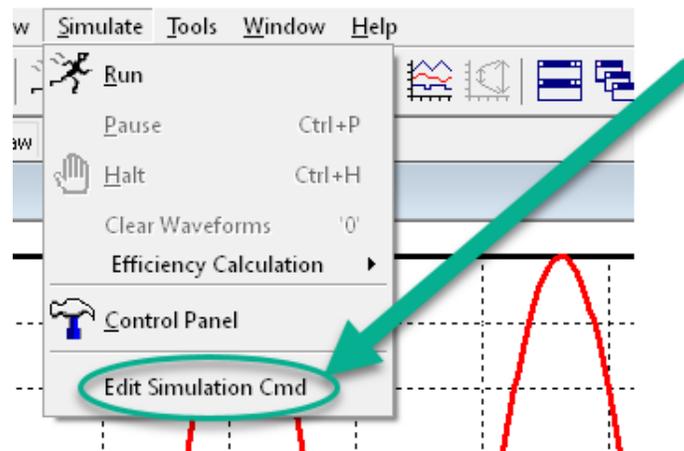


Figure A.14.

- Choose the “AC Analysis” tab from the new pop-up selection box:

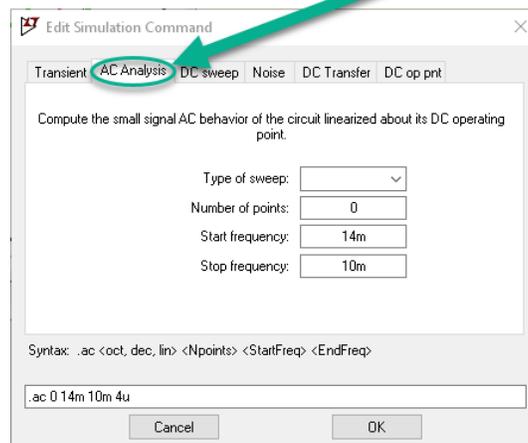


Figure A.15.

- Choose a logarithmic (Decade) sweep type with a start frequency of 1 Hz, an ending frequency of 1 MHz, and 100 points per decade. (This will give you plenty of points and the plot will be nice and smooth.)

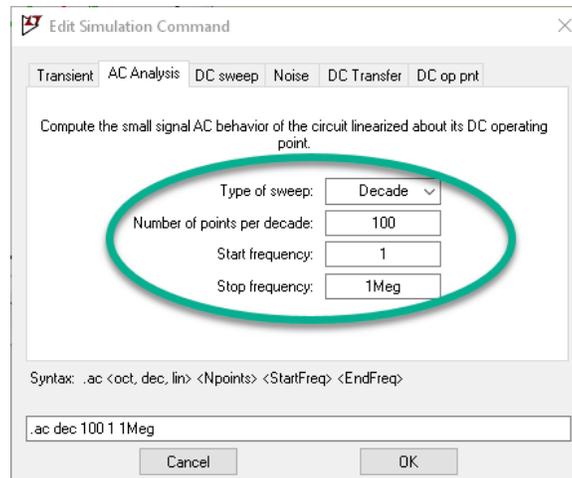


Figure A.16.

- Place the additional LTspice directive at a convenient location in the schematic window.

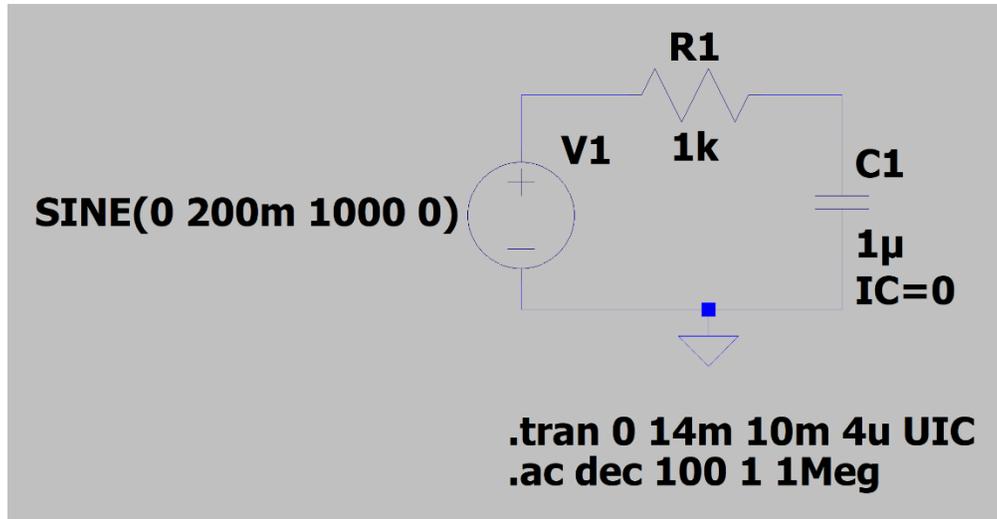


Figure A.17.

- You need to do one more important thing before you run the simulation. You cannot do an AC sweep without setting parameters for the sinusoidal voltage source. If you right-click on the source symbol in your schematic, the pop-up menu for the SINE signal will open again and parameter values can be changed. Set the amplitude parameter to 200 mV (400 mV<sub>p-p</sub>).

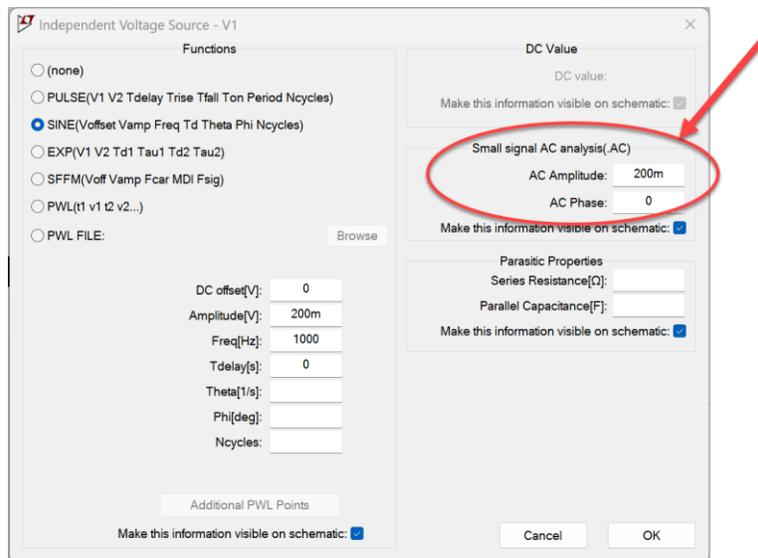


Figure A.18.

- Click “OK” and place the resulting LTspice directive at a convenient location in your schematic window.

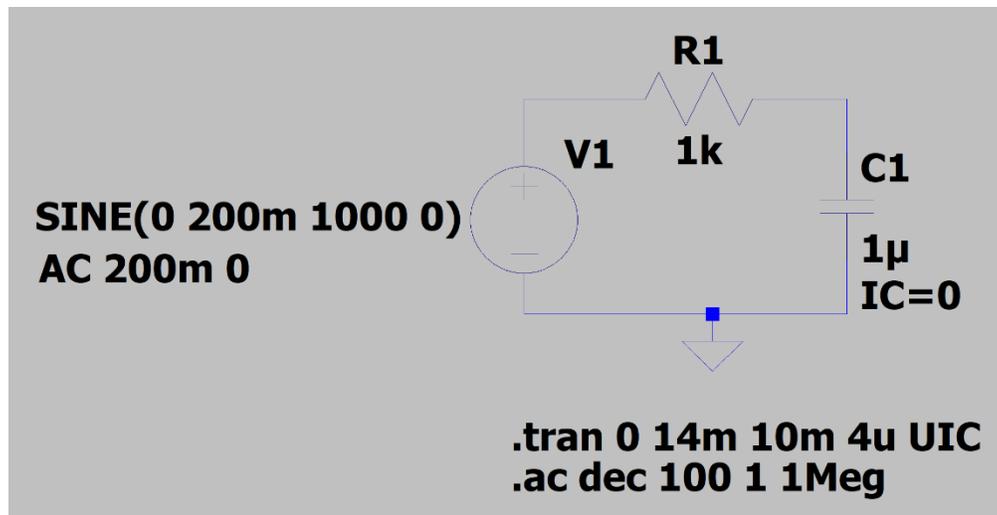


Figure A.19.

- Run the AC simulation.
- Right-click in the waveform display window and add traces for the amplitude of the input and the output. Note that the horizontal scale of the plot is now in Hertz, and the input trace is a horizontal line at 200 mV. This makes sense because the amplitude of the input is 200 mV at all frequencies.

The amplitude of the output, however, changes with frequency. For what range of frequencies is the amplitude of the output equal to the amplitude of the input? For what range of frequencies is it near zero? What is the amplitude at 1 kHz? Does this match the amplitude of the transient response you plotted at 1 kHz? Copy the AC sweep plot of your *RC* circuit and include it in your report.

- We now know that a capacitor will affect the amplitude of the circuit response, and that this effect varies with frequency. What about the phase? LTspice allows you also to plot the phase angle. Change your display to show the phase difference between the input and output voltages. Do this by plotting  $V(n002)/V(n001)$  over the frequency range  $1 \text{ Hz} \leq f \leq 1 \text{ MHz}$ .

Look up “Waveform Arithmetic” in the LTspice>Help>Index tab to see what other functions you can use in specifying a plot.

- Now you should be looking at a trace that represents the phase difference between the output and the input of the circuit at all frequencies between 1 Hz and 1 MHz. Note that the y axis is now in degrees. The input phase angle does not change. It is always zero because the sine wave in LTspice starts at zero by default. However, the phase angle for the output voltage across the capacitor does change with frequency. At what frequencies is the phase of the output the same as the phase of the input? At what frequencies is the phase shift  $-90^\circ$ ? What is the phase shift at 1 kHz? Does this correspond with the phase shift of the later cycles that you got in your transient analysis at 1 kHz? Copy the AC sweep plot of the phase of your *RC* circuit

and include it in your report.

### Simulate a Series *RL* Circuit

*Transient Analysis:* In this section, we will repeat the above procedure for the simple circuit with an inductor replacing the capacitor as shown in Figure A.20 below.

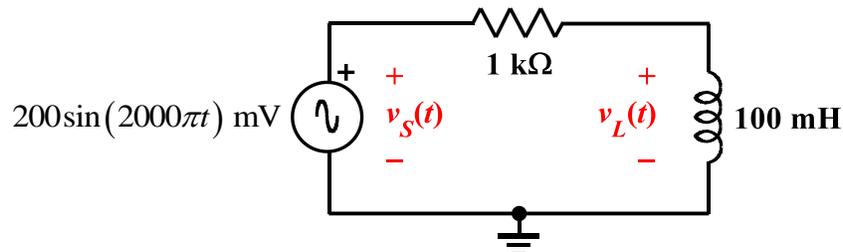


Figure A.20.

- Create a new schematic diagram for the *RL* circuit in LTspice.

**Note:** It is recommended that you create a new schematic diagram for each circuit, rather than just modifying the same one over and over. The individual files are very small and will not take up much space on your drive. Then it will be easier to re-run things, if necessary, when you are writing your report.

- Run the transient simulation. Again, the circuit has influenced both the amplitude and the phase of the input. How close is the phase shift to  $+90^\circ$ ? Note: As before, use one of the later cycles to determine this. Copy this plot and include it in your report.

*Frequency Sweep Analysis:* Now perform an AC sweep analysis.

- Change the values in the drop-down menu. Again, use the parameters you set before (from 1 Hz to 1 MHz with 100 points per decade), and run the simulation.
- You should see two traces. At what frequencies is the amplitude of the output equal to the amplitude of the input? At what frequencies is it near zero? What is the amplitude at 1 kHz? Does this match the amplitude of the transient you plotted at 1 kHz? How is this sweep different than the sweep you created using the circuit with the capacitor? Copy the AC sweep plot of the *RL* circuit and include it in your report.
- The final thing we need to do is plot the phase. Revise your display to show the input and output phase angles.
- At what frequencies is the phase of the output the same as the phase of the input? At what frequencies is the phase shift  $+90^\circ$ ? What is the phase shift at 1 kHz? Does this correspond with the phase shift you got in your transient analysis at 1 kHz? Copy the AC sweep plot of

the phase of your  $RL$  circuit and include it in your report.

## Summary

In this part of the experiment, you have seen that capacitors and inductors influence the behavior of a circuit. They change both the phase and the amplitude. The degree of influence depends on the frequency of the input source. You also learned how to examine the behavior of a circuit over a range of frequencies using the AC sweep feature in LTspice.

## Part B: RC and RL Circuit Transfer Functions and Filters

In this section, we will continue our analysis of the two simple circuits we created in part A and introduce the concepts of transfer functions and filters.

### Background

*Transfer Functions:* We know that a circuit with only resistors will behave the same at any frequency. A voltage divider with two 1 kΩ resistors divides a voltage in half at 10 Hz as well as it does at 100 kHz. We also know that circuits containing capacitors and/or inductors behave very differently at different frequencies. What if we could find a function that, when applied to any input signal, would give you the output signal? This cannot be done easily in the time domain, however, it is quite simple in the complex polar domain we introduced in part A. We can define a function  $H(\omega)$  for any circuit such that

$$V_{out} = [H(\omega)] \cdot V_{in}$$

*Finding Transfer Functions:* For a simple series circuit, we can find the transfer function using a concept similar to the voltage divider rule. We simply need to expand the idea of resistance to include complex impedance. We can combine impedances in the complex polar domain in just the same way that we combine resistances in the time domain. And we can also easily define our transfer function using the voltage divider rule, as illustrated below.

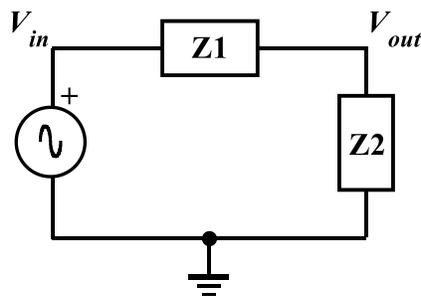
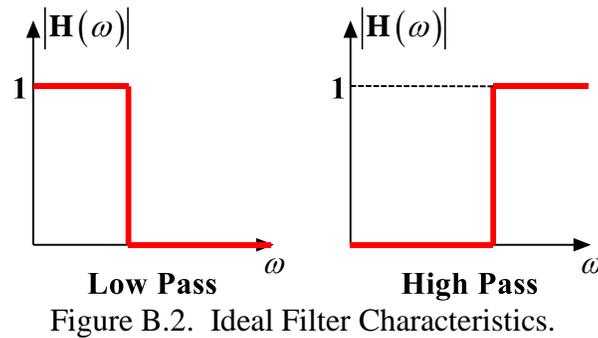


Figure B.1.

$$V_{out} = \frac{Z_2}{Z_1 + Z_2} \cdot V_{in} \quad \Rightarrow \quad H(\omega) = \frac{Z_2}{Z_1 + Z_2}$$

*Filters:* Most electrical signals are made up of many components having different frequencies. In Experiment 1, you learned that sound waves within human hearing range cover a range of frequencies from very low to very high. Sometimes we may not want to include all these frequencies in our signal. We may want to filter out the very high frequencies that sound like noise, for instance, so that we only hear the part of the sound that we want to. In electronics a circuit that filters out certain frequencies while allowing others to remain unchanged is called a

filter.



The two basic types of filters we will consider in this part are low-pass (LP) filters and high-pass (HP) filters. An idealized representation of these two types of filters is shown in Figure B.2. Low-pass filters filter out high frequencies while allowing low frequencies to pass through unchanged. High-pass filters block out low frequencies while allowing high frequencies to pass through unchanged. In an ideal world, the transfer function would be 1 for the frequencies that you want to remain unchanged (a signal multiplied by 1 is the same) and 0 for the frequencies you want to filter out (a signal multiplied by 0 is 0). In an ideal filter, the transition between 1 and 0 is instantaneous. In a real filter, this transition is less exact.

*Using Transfer Functions to Determine Filter Behavior:* We can determine what kind of filter the simple  $RC$  circuit pictured in Figure B.3 is by finding the transfer function and examining its behavior at low and high frequencies. First do the complex algebra to determine the capacitor voltage. You will notice that once again we have a voltage divider circuit, except that one of the impedances is imaginary and one is real. For an operating frequency  $\omega = 2\pi f$ , the impedance of

the capacitor  $C_1$  is equal to  $Z_{C_1} = \frac{1}{j\omega C_1}$ , while the impedance of the resistor  $R_1$  is  $Z_{R_1} = R_1$ .

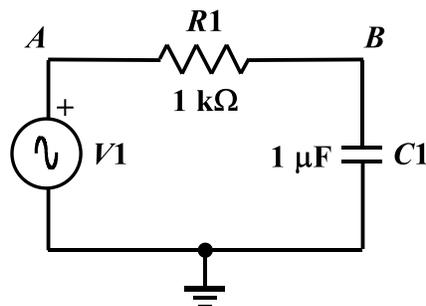


Figure B.3.

Applying the usual voltage divider relation for this series combination of two impedances gives us

$$V_B = \frac{Z_{C1}}{Z_{C1} + Z_{R1}} \cdot V_A = \frac{1/j\omega C_1}{1/j\omega C_1 + R_1} \cdot V_A$$

Note that the relationship between the input voltage  $V_A$  and the output voltage  $V_B$  is complex.

It is useful to be able to set up these expressions and then simplify them for very low and very high frequencies. We will be able to use the resulting expressions to see if our LTspice plots make any sense. We will first look at very low frequencies. We cannot set frequency equal to zero, since parts of our formula will blow up. Rather, we will assume that the frequency is very small, but not zero. Then the capacitive impedance  $Z_{C1}$  will be very much larger than the resistive impedance  $Z_{R1}$  and we can neglect the latter term. We have then, at low frequencies, that

$$V_{out} = V_B \approx \frac{Z_{C1}}{Z_{C1}} \cdot V_A = V_A = V_{in}$$

or, more simply, that  $\frac{V_{out}}{V_{in}} = 1$ , i.e., the input and output voltages are essentially identical. On the

other hand, at high frequencies, the capacitive impedance will be the small term, so we can neglect it in the denominator. We cannot neglect it in the numerator, since it is the only term there. Thus, at high frequencies,

$$V_{out} = V_B \approx \frac{1}{j\omega R_1 C_1} V_A = \frac{1}{j\omega R_1 C_1} V_{in}$$

or  $\frac{V_{out}}{V_{in}} = \frac{1}{j\omega R_1 C_1}$ . This ratio clearly goes to zero as the frequency goes to infinity. It is also negative imaginary and, thus, has a phase of  $-90^\circ$ .

*Note about “Low” and “High”:* It may seem that terms like *low* and *high*, when applied to something like frequency, are fuzzy, ill-defined terms. However, that is definitely not the case. Here, we mean something quite specific by the expression “low frequency”. A frequency is only low when the capacitive impedance is so much larger than the resistive impedance that we can neglect the resistive term. We usually need to define the required accuracy to make a statement like this. For example, we can say that a frequency is low as long as the approximate relationship we found between the input and output voltages is within 5% of the full expression. Most of the parts we use in circuits are no more accurate than this, so there is no particular need to do our calculations with better accuracy. In other applications we might need better accuracy.

*Cutoff (or Corner) Frequency:* When we design a high or low-pass filter, we only need to know three things: how its transfer function behaves at low frequencies (is it 0 or 1), how it behaves at high frequencies (0 or 1), and at what frequency it switches (from 0 to 1 or 1 to 0). For a simple *RC* or *RL* filter, the frequency at which it switches is called the corner or cutoff frequency. It is defined as the point at which the magnitude of the transfer function is equal to  $1/\sqrt{2}$ .

## Experimental Procedure

### The Transfer Function Equation

In this section we will explore the impedance of the capacitor circuit. If we want to apply the voltage divider equation to determine the transfer function, then the voltage drop across the two components must add up to the input voltage at all frequencies.

- In LTspice, reload the capacitor circuit (Figure A.3) or recreate it. To test the validity of applying a voltage divider to complex impedances, we want to show that, just like with the voltage divider made with resistors, the source voltage ( $V_1$ ) is equal to the sum of the voltages across the resistor and the capacitor ( $V_R + V_C$ ). To determine  $V_R$  and  $V_C$ , you may have to plot expressions relating the node voltages.
- Rerun the transient analysis from part A (4 ms with a step size of 4  $\mu$ s).
- When you have run your simulation, add a trace in the display window to show the input voltage. To display the sum of  $V_R$  and  $V_C$  for comparison, we must define a new trace.
  - Go to the add trace menu again, find the name for the  $V_R$  trace, and click on it. It should look something like V(R1). If your resistor has a different name or a different polarity relative to ground, the expression will not be exactly the same.
  - Type in + (plus).
  - Find the signal corresponding to  $V_C$  and click on it.
  - The expression to be plotted should then look something like: V(R1)+V(C1). (Since the names and polarities of your components may vary, your expression may not be identical to this example.)
  - Click on OK.
- If you have added the traces correctly, you should see that the sum will equal the source voltage. Copy a plot of your results, and include it in your report.
- Run a transient simulation for frequencies of 10 Hz (run time = 400 ms and step size = 400  $\mu$ s) and 10 kHz (run time = 0.4 ms and step size = 0.4  $\mu$ s). Place the trace of  $V_R$  and  $V_C$  on each plot. Verify that the voltages add as expected. Copy these plots and include them in your report.
- Since we have a series circuit, we know that the current,  $I$ , is the same through all the components. Therefore,  $V = IZ$  for each component and for the entire circuit. This means that the transfer function,  $H = V_{out}/V_{in}$ , is independent of current, and simplifies to an expression in terms of the component values only:

$$H(\omega) = \frac{V_{out}}{V_{in}} = \frac{IZ_C}{IZ_R + IZ_C} = \frac{Z_C}{Z_R + Z_C}$$

## The Transfer Function of an *RC* Circuit

In this part, we will find the transfer function of our capacitor circuit from part A and determine what type of filter it is.

- Rerun the AC sweep (from 1 Hz to 1 MHz at 100 points per decade).
- We know that  $H(\omega)$  is the ratio of the output voltage to the input voltage. We can observe this directly by adding a trace to take the ratio between the two traces we have on our plot.
  - Go to the Trace menu in the LTspice program and choose Add Trace.
  - Find the signal corresponding to your output in the list on the left-hand side. The name of this signal is written at the bottom of your plot next to a symbol indicating the color of the trace. The trace corresponding to the output is the one that is not constant. Click on the name of the output trace.
  - Type in / (divided by) or click on the / symbol in the functions menu.
  - Find the signal corresponding to your input and click on it.
  - The expression in the bottom box should look something like: V(R1:2)/V(V1:+)
    - Click on OK.
- Another trace should appear on your plot. This trace is the transfer function of the circuit. Is the transfer function 0 or 1 at low frequencies? Is it 0 or 1 at high frequencies? What kind of filter is this?
- The cutoff frequency of the circuit is the frequency at which the transfer function is  $1/\sqrt{2} \approx 0.707$ . Use the cursors to mark this location on your plot.
  - Click on the cursor button on the LTspice toolbar.  or 
  - Click on the colored symbol for the trace you added (H) at the bottom of your plot. This places the cursor on this trace.
  - Drag the cursor along the trace until the y coordinate is as close as possible to 0.707
  - Mark this point with the button on the toolbar.  or 
  - What is the cutoff frequency? Copy this plot and include it in your report.
- In class we derived an expression for the cutoff frequency of an *RC* circuit:  $\omega_c = \frac{1}{RC}$ . Calculate the cutoff frequency for this filter. Don't forget that  $f$  (in Hertz) is  $\omega/2\pi$ . How does this compare to the frequency you found with LTspice?
- The equation for the transfer function of a series circuit is  $H(\omega) = \frac{Z_2}{Z_1 + Z_2}$ . Use this expression to find the transfer function of the capacitor circuit. Take the limit of this function at high and low frequencies. Does the plot you made with LTspice behave the same way at low and high frequencies?

## The Transfer Function of an *RL* Circuit

In this part, we will find the transfer function of our inductor circuit from part A and determine what type of filter it is.

- In LTspice, recreate the inductor circuit (Figure A.3) by replacing the capacitor with a 1 mH inductor. Run the AC sweep.
- Create the transfer function by adding a trace that divides the output voltage by the input voltage.
- Is the transfer function 0 or 1 at low frequencies? Is it 0 or 1 at high frequencies? What kind of filter is this?
- Use the cursors to mark the location of the cutoff frequency on your plot. What is the cutoff frequency? Copy this plot and include it in your report.
- Can you derive the equation for the cutoff frequency of this circuit? Calculate the cutoff frequency using the equation you derived. Don't forget that  $f$  (in Hertz) is  $\omega/2\pi$ . How does this compare to the frequency you found with LTspice?
- Find the transfer function of the inductor circuit using the equation for  $H(\omega)$ . Take the limit of this function at high and low frequencies. Does the plot you made with LTspice behave the same at low and high frequencies?

### Build and Test an RC Circuit

Now build the circuit on your breadboard, connect it to the Signal Generator (as described below) and compare its behavior to the simulation. You may find the cursors and the Measurements view to be useful.

- Build the capacitor filter circuit (Figure A.2) on your breadboard using a 1  $\mu\text{F}$  capacitor and a 1  $\text{k}\Omega$  resistor.
- Set Signal Generator Channel 1 to a 1 kHz sinewave signal with an amplitude of 200 mV, (400 mV<sub>p-p</sub>).
- Display the input signal on channel 1 of the Oscilloscope (the orange wires and the yellow trace) and the output signal on channel 2 of the Oscilloscope (the blue wires and the blue trace). Copy the image of this signal with the edit function in the software and include it in your report. (We will call this the Camera Function of the software.) How does the output of this circuit compare (amplitude and phase) with the transient analysis you made of this circuit at this frequency?
- You should notice that the amplitude and phase of this circuit changes as you change the frequency. At what frequencies does the output look roughly the same as the input? At what frequencies does the output essentially disappear (the amplitude is so small it looks like a 'flat'?

line?

- At the cutoff frequency, the transfer function is equal to 0.707. The transfer function is defined as  $V_{out}/V_{in}$ . If  $V_{in}$  is 200 mV, what should the amplitude of the output be at the cutoff frequency? Adjust the frequency of the input signal until the output amplitude is 70.7% of the input amplitude. Record this frequency and take a picture of the signal with the camera function in the software and include it in your report.
- The cutoff frequency for an  $RC$  circuit is given by  $f_c = 1/(2\pi RC)$ . How close is the cutoff frequency of the circuit you built to the calculated one?

## Summary

Transfer functions relate the output voltage to the input voltage of a circuit for all frequencies. Transfer functions can be used to determine the type of filter a circuit represents. One can derive the transfer function for a series circuit by using circuit analysis rules to combine the complex impedances into a ratio  $H(\omega) = Z_{out}/Z_{in}$ .

## Part C: RLC Circuit Transfer Functions and Filters

### Background

*Phasors:* Circuits, like most engineering systems, can be divided into basic building blocks, each with its own function. If we assume that the function of such a building block is to change a voltage in some way (i.e. filter it, amplify it, etc.), then we label the input voltage as  $V_{in}$  and the output voltage as  $V_{out}$ . Since it is easiest to reference the phase of the input to zero, we write the sinusoidal input voltage as  $V_{in} = V_o \cos(\omega t)$  and the output voltage as  $V_{out} = V_1 \cos(\omega t + \phi_o)$  where  $\omega = 2\pi f$

When working exclusively with AC steady state conditions, it is generally easier to analyze circuits using phasor notation. Euler's identity tells us that exponentials with imaginary arguments can be related to sines and cosines by  $e^{j(\omega t + \phi)} = \cos(\omega t + \phi) + j \sin(\omega t + \phi)$ , and that permits us to write the output voltage, for example, in the equivalent form  $V_{out} = \Re\{V_1 e^{j(\omega t + \phi)}\} = \Re\{V_{out} e^{j\omega t}\}$ , where  $V_{out} = V_1 e^{j\phi}$  is a complex number and is called the phasor form of  $V_{out}$ . Much of the time we find it convenient to consider the real and imaginary parts of this complex representation separately  $V_{out} = V_{\Re} + jV_{\Im}$  where  $V_{\Re} = V_1 \cos \phi$  and  $V_{\Im} = V_1 \sin(\phi)$ .

Thus, we will also be keeping track of the real and imaginary parts of voltages and currents. When we run LTspice simulations, we will be able plot expressions that do this for us.

If we apply the phasor form to the equations that characterize voltage and current for capacitors and inductors, we obtain the following

$$V_L = \Re\{V_o e^{j\omega t}\} = L \frac{dI_L}{dt} = L \frac{d}{dt} [\Re\{I_o e^{j\omega t}\}] = j\omega L [\Re\{I_o e^{j\omega t}\}] = j\omega L I_L, \text{ or more simply, } V_L = j\omega L I_L \text{ and, following a similar analysis, } I_C = j\omega C V_C.$$

Thus, instead of a differential equation, we obtain the same kind of algebraic relationship we had for resistors, just like Ohm's Law, except that the impedances (a generalization of resistance) of inductors and capacitors are given by  $Z_L = j\omega L$  and  $Z_C = 1/(j\omega C)$ . For resistors,  $Z_R = R$ , while in general  $Z = R + jX$ , where  $R$  is resistance and  $X$  is reactance.

*RLC Circuits:* Circuits with resistors, inductors and capacitors are called resonant circuits. They find wide application in electronics because there are many circumstances in which we wish to produce or block a single frequency. There are two common types of resonant circuits: parallel or series combinations of a resistor, an inductor and a capacitor. Figure C.1 shows series RLC circuits with 6 possible input-output choices.

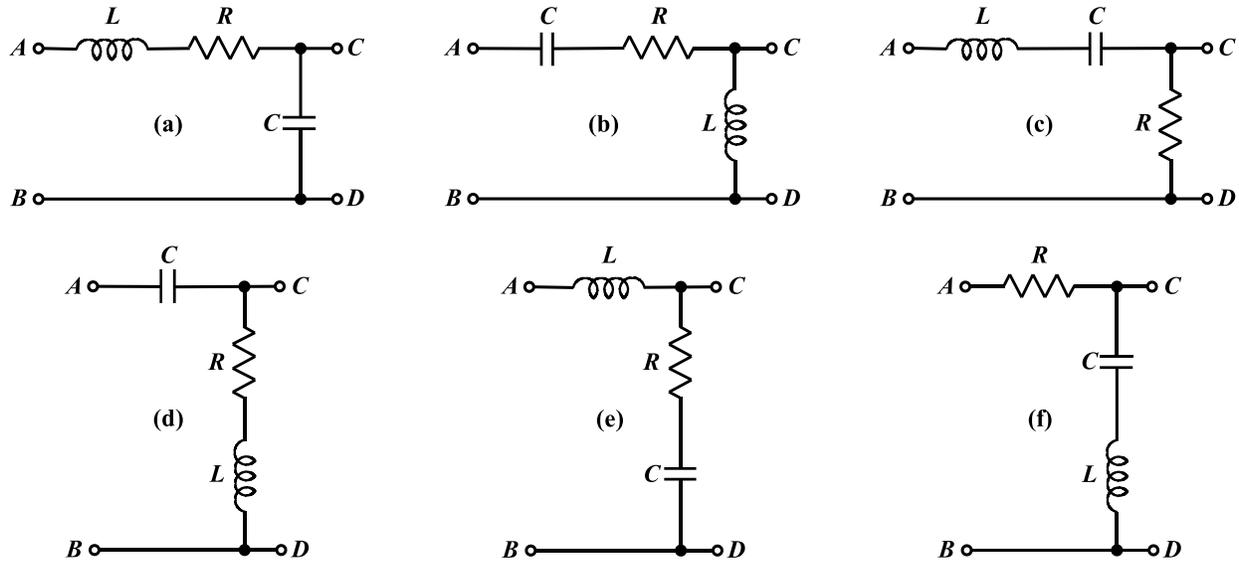


Figure C.1. Examples of Series Resonant Circuits.

Such systems have a natural resonance frequency -- the frequency where the inductive and capacitive impedances cancel. The expression for the resonant frequency is  $\omega_0 = 1/\sqrt{LC}$  as seen in class.

*More Complex Filters:* RLC circuits can be used to create more types of filters than could be accomplished with simple RC or RL circuits. Any of the four generic filter types shown in Figure C.2 can be constructed.

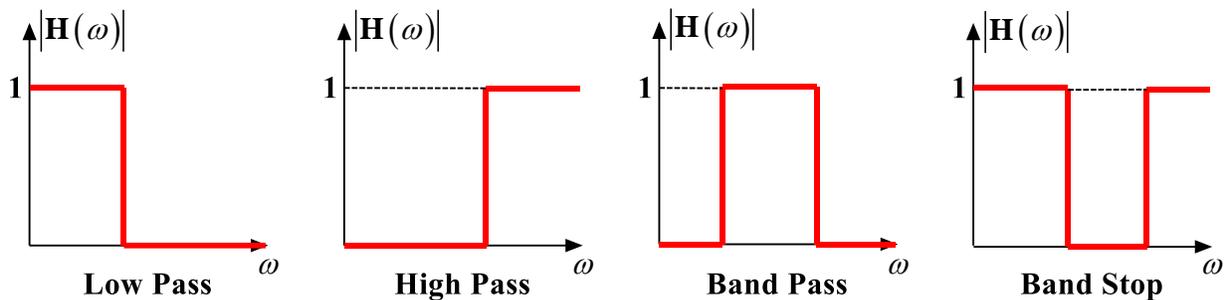


Figure C.2. Ideal RLC Filter Characteristics.

Each circuit in Figure C.1 has a low frequency and a high frequency approximation, found in the same manner as we have seen for RC and RL circuits. Looking at the limiting cases of  $H(\omega)$  for frequencies near zero and for very large frequencies, we can determine which type of filter each is. For example, note that at both high and low frequencies, the transfer function for case C.1(f) is equal to one, and thus the input voltage appears unchanged at the output. The circuit is “passing” very low and very high frequencies.] Also, since the impedance of a capacitor is negative

imaginary and that of an inductor is positive imaginary, there will be a frequency  $\omega_0$  where the net impedance between output terminals  $C$  and  $D$  will be zero. Near this frequency, the output voltage will be zero or at least very small. These frequencies are “rejected” by this filter. It must be a band-stop (also called band-reject or notch) filter.

*Modeling Electrical Components:* When we build circuits, we use a Signal Generator to produce the input voltage, together with a resistor, an inductor and a capacitor. However, we cannot use just a voltage source and three other components when we model them in LTspice, since real devices usually cannot be modeled by a single parameter. Let us consider each of these four components and see what is necessary for realistic analysis or simulation:

- *Signal Generator:* Most commercial Signal Generators are designed to have an internal impedance of  $50\ \Omega$ . Thus, in a realistic circuit, we must use an ideal sinusoidal voltage source and a series resistor to represent the Signal Generator. (In the case of the Signal Generator on the M2K this impedance can be neglected.)
- *Resistor:* Except at very high frequencies, resistors behave in close to an ideal manner. Thus, it is almost always reasonable to represent the resistor in a circuit as ideal.
- *Inductor:* Since inductors are made with a long piece of wire, they usually have a significant impedance, in addition to their inductance. Thus, we must include an additional series resistor when modeling a real inductor.
- *Capacitor:* A capacitor typically consists of two large metal plates separated by a thin insulator. If the insulator is very good, almost no current will flow between the plates. Then, like the resistor, the capacitor will behave in an essentially ideal manner and we don't need to add any extra components in a circuit that represents a real capacitor.

None of the components we can manufacture are really ideal. Resistors also have inductance and, sometimes, capacitance. Inductors have capacitance. Capacitors have resistance and inductance. Fortunately, we have learned how to make these devices so that they behave in a nearly ideal manner for quite a broad range of frequencies. However, when we push the limits of a circuit, we have to remember that it may behave as if it consists of more components that we can see when we build it. When the  $RLC$  circuit is built, we will only see the three components  $R_1$ ,  $L_1$  and  $C_1$ , as shown in Figure C.3.

## Experimental Procedure

### Simulate a Series $RLC$ Circuit

In this section, we will use LTspice to simulate an  $RLC$  circuit and plot the magnitude and phase of its transfer function.

- Set up the circuit as shown in Figure C.3 below.

The LTspice model of the inductor must include both L1 and R1.

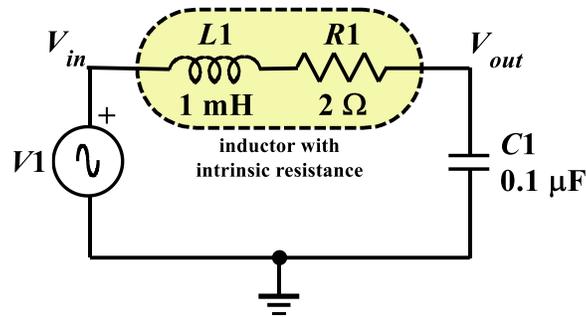


Figure C.3.

- Assume that the amplitude of  $V1$  is 200 mV, ( $400 \text{ mV}_{p-p}$ ). Note that there is nothing to represent the impedance of the Signal Generator (assumed zero) and  $R1$  represents the resistance of the 1mH inductor.
- Perform an AC sweep from 100 Hz to 1 MHz and display the input and output voltages. Since we have now done several AC sweeps, can you explain why we do logarithmic decade (DEC) sweeps rather than linear (LIN) sweeps?
- Create a double plot (two plots in the same window) of the magnitude and phase of the transfer function as follows:
  - Use the Add Plot to Window menu under the Plot menu in Probe.
  - Add a trace of the absolute value of the ratio of  $V_{out}$  to  $V_{in}$ , to the top plot. You can use the `abs()` function for this purpose. [The trace expression should look something like `abs(V(C1:1)/V(R1:1))`. To find the actual names of your input and output voltage traces, look at the bottom left hand corner of your plot.] This is the magnitude of the transfer function.
  - On the bottom plot display the trace of the phase angle of the transfer function  $V_{out}/V_{in}$ : Add a trace of the phase of the ratio of  $V_{out}$  to  $V_{in}$ . You can use the phase function (`p`) for this purpose. [The trace expression should look something like `p(V(C1:1)/V(R1:1))`. To find the actual names of your input and output voltage traces, look at the bottom left hand corner of your plot.] What is the phase shift between the output and input at low and high frequencies?
  - Copy this plot. Label the resonant frequency (extreme point) with the cursors.
  - Also calculate the theoretical resonant frequency and mark it on the plot.
  - You will also mark the actual resonant frequency of the circuit you build in the next section on this plot.
  - Note that this calculation isn't exact. It gives you a simple mathematical way to get close to the resonant frequency of a circuit. The exact value of the resonant frequency can be found by examining when denominator of the transfer function goes to zero.
- What type of filter is this?

### Build and Test a Series *RLC* Circuit

Now build the circuit in Figure C.3 on your breadboard, connect it to the Signal Generator and compare its behavior to the simulation. The inductor is labeled 102,  $10 \times 10^2$  or 1000 $\mu$ H, 1mH

- Before you build the circuit, measure the exact values of the components.
  - Measure the DC resistance of the inductor's winding using the DMM.
  - Measure the capacitance of the capacitor using the DMM.
- Once you have actual measurements of the parameters, change your LTspice simulation. Run the AC sweep again to see what it looks like now. Copy the AC sweep plot and include it in your report. Are there any significant differences between the two simulations?

Build the circuit on your breadboard. Remember that you need only two components: the 1mH inductor (with an internal resistance of about  $2\Omega$ ) and the 0.1 $\mu$ F capacitor. The 0.1 $\mu$ F capacitor will have 104 written on it. Remember that R1 represents the resistance of the 1 mH inductor. DO NOT add a separate resistor to your circuit for this.

- Find the resonant frequency of the circuit. This is the point where the voltage of the output is at a maximum. It will exceed the input voltage at this point. You should find that it is close to the resonant frequency you found using LTspice and the one you calculated using the theoretical equation. It may not fall in exactly the same place because of errors introduced by the tolerances of the components and the influence of other parts of the circuit. Mark the actual resonant frequency on the AC plot you generated using the theoretical component values. Note that, depending upon how you found the resonance frequency, the same circuit gave you three similar, but not exact, values.
- Use 5 frequencies related to the resonant frequency, set at close to 10%, 90%, 100%, 110% and 1000% of the resonant frequency. These represent low frequency, a point between low frequency and the resonant frequency, the resonant frequency, a point between the resonant frequency and the high frequency, and high frequency.
- At each frequency, determine the ratio of the input and output amplitudes (magnitude of the transfer function) and the phase shift. There is no magic button for determining either the transfer function or the phase shift between two signals. You will have to do this by displaying the input and output voltages on the Oscilloscope and manually determining the ratio of the amplitudes and the phase shift. Cursors will be useful.
- Put the values that you measured on the LTspice AC sweep plot you made using the actual measured component values.

## Summary

In this section, you extended your knowledge of transfer functions and filters to include circuits with all three component types:  $R$ ,  $L$  and  $C$ . These circuits are called resonant circuits. For these circuits the center of the band for band-pass and band-reject filters occurs at the resonant frequency. In high and low-pass filters, the resonant frequency can be used to determine the

location of the transition between passed and rejected frequencies.

## Part D: Equivalent Impedance

### Background

*Combined Impedance of Parallel Components:* We have examined several series circuits with a variety of different component combinations. What about parallel circuits? We know that we cannot use a simple voltage divider analysis on a resistive circuit with parallel components. We have to combine parallel components until we have a series circuit and then we can apply the voltage divider method. The same is true of complex impedance. If we combine the impedances until we have a series circuit, then we can find the transfer function for the simplified circuit using the same method already described in this experiment. The rules for combining complex impedances are the same as those for combining resistors:

(a) Impedances in series:  $Z_{eq} = Z_1 + Z_2 + \dots + Z_n$

(b) Impedances in parallel:  $\frac{1}{Z_{eq}} = \frac{1}{Z_1} + \frac{1}{Z_2} + \dots + \frac{1}{Z_n}$

### Experimental Procedure

#### Simulate a Parallel RC Circuit

In this part of the experiment, we will consider what happens when we combine two impedances in parallel.

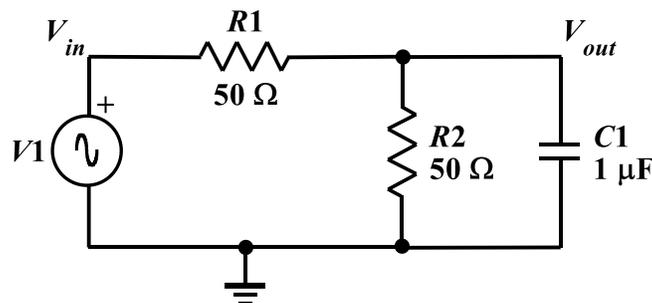


Figure D.1.

- Set up a voltage divider in LTspice with two 50Ω resistors and a sinusoidal source with 200mV amplitude, 1 kHz frequency and no DC offset. Add a 1μF capacitor in parallel with the second resistor, as shown in Figure D.1. Since we are going to do an AC sweep, don't forget to set the AC parameter to the amplitude of your signal, 200mV. Again, you can do this directly from the parameter displayed or using the spreadsheet.
- Set up an AC sweep of this circuit. Do a logarithmic sweep with a start frequency of 1 Hz and an end frequency of 15 MHz. You should use about 100 points per decade. The AC Sweep

Type has been chosen as Logarithmic - Decade (DEC) since frequency effects usually only become obvious when we change orders of magnitude. This generates a logarithmic scale for frequency. The start frequencies and end frequencies are chosen to cover an interesting range. Usually this range is selected from some knowledge of the expected performance of the circuit. However, since we are assuming that we know very little about this circuit, we can set the range to be roughly that covered by a typical Signal Generator.

- For what range of frequencies does the capacitor change the voltage across  $R_2$  by less than 5%? Use the cursor to find a reasonably precise answer to this question. Mark the location on your plot. Copy this plot and include it in your report.
- Let us see if we can determine at least the magnitude of the equivalent impedance of the combination of  $R_2$  and  $C_1$  at the frequency of 1 MHz.
  - Go back to your schematic diagram and change the value of the source ( $V_1$ ) frequency to 1 MHz.
  - Set up a transient analysis. Click on the Edit Simulation Settings button and set up for transient analysis using smaller times. Since 1 MHz is 1000 times larger than 1 kHz, you will have to make the run time and step size 1000 times smaller to produce three cycles of the oscillating voltage signal.
  - Run the transient analysis.  
In the circuit you are analyzing,  $R_2$  and  $C_1$  together have a different impedance at different input frequencies. This means that at any one given frequency, we could replace the combination by a single resistor (that we will call  $Z$ ). Note that at the higher frequencies the voltage is very small and the combination of the capacitor and  $R_2$  should look like a very small resistor indeed.
- You should see that the source oscillates as it did before, although at a much higher rate and that the voltage across  $R_2$  and the capacitor seems not to change with time at all. Actually, the latter voltage is still oscillating, but at such a small amplitude that you cannot see it. Delete the trace that shows the source voltage.
  - Click on the scissors icon in the toolbar at the top of the screen.
  - Move the scissors cursor to the name of the trace you wish to delete, and left-click. The trace should be erased.
- Using the cursors, determine the amplitude of the sine wave oscillation across  $R_2$  and the capacitor. The voltage will also have a DC level, but we only want to determine the sine wave amplitude. The peak-to-peak amplitude can be determined by subtracting the voltage at a minimum from the voltage at a maximum. The actual amplitude of the sine wave will be half the peak-to-peak value. Write down the amplitude you determined here.
- From this amplitude and your knowledge of how voltage dividers work, determine the magnitude of the equivalent impedance of the  $R_2 \parallel C_1$  combination, that we are calling  $Z$ . [Hint: Figure D.2 shows a simplified version of the voltage divider. You can use the voltage divider equation:

$$V_z = \frac{Z}{R1 + Z} \cdot V_1$$

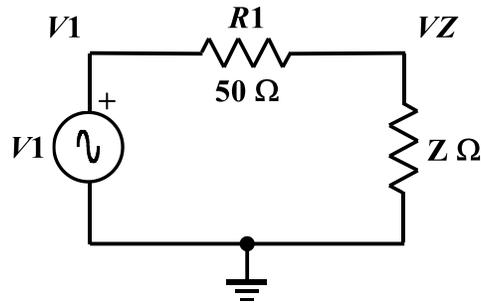


Figure D.2.

- Check your answer by replacing the capacitor/resistor combination with a single resistor having value equal to the value of  $Z$  you calculated. Rerun the simulation. Does the output have the same amplitude as the combination?
- Now check to see if the transfer function gives the same amplitude. Use the rules for parallel impedance to combine  $1/j\omega C$  and  $R$ . Then set up the transfer function for the circuit. Determine its magnitude at 1 MHz. (Don't forget that  $\omega = 2\pi f$ .) Multiply the value of  $H(\omega)$  at this frequency by the input voltage amplitude (200mV). How does the output amplitude of your plot compare to what you calculated?

## Summary

In this section we explored what happens when two impedances are connected in parallel. You have seen that it is possible to find transfer functions for and analyze circuits with components in parallel as well as the series circuits considered earlier. You simply need to combine the impedance of the parallel components into an equivalent impedance using rules similar to those used to combine resistors.

## What Should be in Your Report?

The following should be included in your report for this experiment. Everything should be labeled and easy to find. Partial credit will be deducted for poor labeling or unclear presentation. ALL PLOTS SHOULD INDICATE WHICH TRACE CORRESPONDS TO WHICH SIGNAL AT WHICH POINT.

### Pre-Lab Exercises

Fritzing or hand-drawn breadboard layout diagrams for all circuits that are to be analyzed using LTspice or physically built and characterized using your M2K board.

### Part A: *RC* and *RL* Circuits

Include the following plots:

1. LTspice transient plot of *RC* circuit response (for Figure A.2).
2. LTspice AC sweep plot of the *RC* circuit voltage.
3. LTspice AC sweep plot of the *RC* circuit phase angle.
4. LTspice transient plot of *RL* circuit response (for Figure A.3).
5. LTspice AC sweep plot of the *RL* circuit current.
6. LTspice AC sweep plot of *RL* circuit phase angle.

Answer the following questions:

1. What is the amplitude and phase of the output of the *RC* circuit at 1 kHz?
2. In what frequency range is the amplitude of the output of the *RC* circuit about equal to the input amplitude? In what frequency range is the amplitude of the output about zero?
3. In what frequency range is the phase of the output of the *RC* circuit about equal to the input phase? In what frequency range is the phase of the output about  $-90^\circ$ ?
4. What is the amplitude and phase of the output of the *RL* circuit at 1 kHz?
5. In what frequency range is the amplitude of the output of the *RL* circuit about equal to the input amplitude? In what frequency range is the amplitude of the output about zero?
6. In what frequency range is the phase of the output of the *RL* circuit about equal to the input phase? In what frequency range is the phase of the output about  $+90^\circ$ ?

### Part B: *RC* and *RL* Transfer Functions and Filters

Include the following plots:

1. LTspice plot of capacitor and resistor voltage sum adding to the input voltage at 1 kHz
2. LTspice plot of capacitor and resistor voltage sum adding to the input voltage at 10 Hz.
3. LTspice plot of capacitor and resistor voltage sum adding to the input voltage at 10 kHz.
4. LTspice plot of transfer function of *RC* circuit (Figure A.2) with corner frequency marked.
5. LTspice plot of transfer function of *RL* circuit (Figure A.3) with corner frequency marked.
6. Scopy picture of *RC* circuit at 1 kHz.
7. Scopy picture of *RC* circuit at the corner frequency.

Answer the following questions:

1. Write out the mathematical expressions for the output voltage of the capacitor for the first *RC* circuit case you considered (plot 1). Write it in the form  $v(t) = A \sin(\omega t + \phi)$ .
2. What kind of filter is the *RC* circuit?
3. At what frequency did you find the corner on the LTspice plot of the transfer function of the *RC* circuit? What frequency did you calculate using  $f = 1/(2\pi RC)$ ? How do the two compare?
4. Find the transfer function of the *RC* circuit in terms of R, C and  $j\omega$ . Take the limit of this function at very low and very high frequencies. Show that these results are consistent with the LTspice plot of the transfer function.
5. What kind of filter is the *RL* circuit (Figure A.3)?
6. Derive the equation for the corner frequency of the *RL* circuit.
7. At what frequency did you find the corner on the LTspice plot of the transfer function of the *RL* circuit? What frequency did you calculate using the equation you derived in the previous question? How do the two compare?
8. At what frequencies did the output of the *RC* circuit you built look roughly the same as the input? At what frequencies did the output disappear into the noise?

### Part C: *RLC* Transfer Functions and Filters

Include the following plots:

1. LTspice plot of the transfer function of the *RLC* circuit in Figure C.3 (magnitude and phase), with three resonant frequency values marked [LTspice value, calculated value, experimental value] (original component values).
2. LTspice plot of the transfer function of the *RLC* circuit in Figure C.3 (magnitude and phase), with the resonant frequency marked (real component values). Also put the 5 experimental points on this plot.

Answer the following questions:

1. Why is it necessary to plot the phase angle and the magnitude of the transfer function separately, rather than on the same plot?
2. For each of the *RLC* circuits in Figure C.1 indicate what type of filter the circuit is (high-pass, low-pass, band-reject, or band-pass filter) and explain why each is the filter it is. [Filter (f) is discussed in the Background section for part C.] Recall that a capacitor can be modeled as an open circuit at low frequencies and a short circuit at high frequencies. Also, recall that an inductor can be modeled as a short circuit at low frequencies and an open circuit at high frequencies. Redraw the circuits at low and high frequencies and consider the value of the output between C and D for each case. You can check your answers in LTspice if you want.
3. What is the phase shift between the output and input of plot 1 above at low and high frequencies? Does the phase shift change when you adjust the values of the components to create plot 2? Why or why not?
4. Determine the resonant frequency of the *RLC* circuit you analyzed with LTspice. (This occurs at the extreme point.) Calculate the resonant frequency with the equation  $f = 1/(2\pi\sqrt{LC})$ . How similar are they? What factors do you think account for the discrepancy?
5. Why do you suppose it is that, in practice, we generally use filters designed with capacitors and not inductors?

## Part D: Equivalent Impedance

Include the following plot:

1. LTspice AC Sweep plot from the circuit in Figure D.1 with the 5% point marked with the cursor.

Answer the following questions:

1. When the capacitor is added, for what range of frequencies does the capacitor change the voltage across R2 by less than 5%?
2. What is the amplitude of the output of the parallel  $RC$  circuit? What is the value you calculated for the equivalent impedance ( $Z_{eq}$ ) of the parallel combination at 1 MHz?
3. Find the transfer function for the parallel  $RC$  circuit. Determine the magnitude of the transfer function at 1 MHz. Calculate the amplitude of the output voltage for this circuit at 1 MHz? How well does it agree with the output amplitude you found using LTspice?

# Breadboard Template

