Experiment 2
Complex Impedance, Steady State Analysis, and Filters

Purpose: The objective of this experiment is to learn about steady state analysis and basic filters.

Background: Before doing this experiment, students should be able to
bullet Determine the transfer function of a two resistor voltage divider.
bullet 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 (real and imaginary axes).
bullet Determine the complex impedance of capacitors and inductors from the values of the components and the operating frequency of whatever power supply is being used.
bullet Determine the values of capacitors and inductors from the information printed on them. (Review the Quiz 1 formula sheet.)
bullet Review the background for the previous experiment.

Learning Outcomes: Students will be able to
bullet Do a transient (time dependent) simulation of RC, RL and RLC circuits using Capture/PSpice.
bullet Do an AC sweep (frequency dependent) simulation of RC, RL and RLC circuits using Capture/Pspice, determining both the magnitude and the phase of input and output voltages.
bullet Determine the general complex transfer function for RC, RL and RLC circuits and simplify for high and low frequencies.
bullet Be able to define what is meant by high and low frequencies in the context of RC, RL and RLC circuits.
bullet Identify whether an RC, RL or RLC circuit is a low-pass filter, a high-pass filter, a band-pass filter or a band-reject filter.
bullet Find the corner frequency for RC and RL circuits and the resonant frequency for RLC circuits.
bullet Find a practical model for a real inductor and determine the range of frequencies in which the real inductor behaves nearly like an ideal inductor.
bullet Measure inductance and capacitance using a commercial impedance bridge.

Equipment Required
bullet DMM (Digital Multimeter)
bullet Analog Discovery (with Wave Forms)
bullet Oscilloscope (Analog Discovery)
bullet Function Generator (Analog Discovery)
bullet DC Power Supply (Analog Discovery)
bullet Impedance Bridge (located on center table)
bullet Components: 1kΩ resistor, 1µF capacitor, 100mH inductor, 0.068µF (683) capacitor
bullet Protoboard

Helpful links for this experiment can be found on the Links by Experiment page.

Pre-Lab

Required Reading: Before beginning the lab, at least one team member must read over and be generally acquainted with this document and the other required reading materials listed under Experiment 2 on the EILinks page.

Hand-Drawn Circuit Diagrams: Before beginning the lab, hand-drawn circuit diagrams must be prepared for all circuits either to be analyzed using PSpice or physically built and characterized using your Analog Discovery board.
Part A – RC circuits, RL circuits, and AC Sweeps

Background

*Complex polar coordinates:* Complex numbers allow you to express a single number in terms of its real and imaginary parts: \( z = x + jy \). \( j \) (the symbol \( i \) is used in mathematics) is used to represent the square root of \(-1\). You can also represent a number in the complex plane in terms of polar quantities, as shown in Figure A-1. The length of the line segment between a point and the origin and the angle between this segment and the positive \( x \) axis form a new complex number: \( z = A \cos \theta + jA \sin \theta \) where \( A = \sqrt{x^2 + y^2} \) and \( \theta = \tan^{-1}\left(\frac{y}{x}\right) \).

![Figure A-1.](image-url)

*Impedance and basic circuit components:* Each basic circuit component has an effect on a circuit. We call this effect impedance. You should remember that impedance causes a circuit to change in two ways. It changes the magnitude (amplitude) and the phase (starting position along the time axis) of the voltages and currents. When a resistor is placed in a circuit, it affects only the amplitude of the voltages. When capacitors and inductors are placed in a circuit, they influence both. We can use complex polar coordinates to represent this influence. A capacitor will change the amplitude by \( \frac{1}{\omega C} \) and shift the phase by \(-90^\circ\). An inductor will change the amplitude by \( \omega L \) and shift the phase by \(+90^\circ\). (Recall that \( \omega \) is called angular frequency and is equal to \( 2\pi f \).) We can represent these changes easily in the complex polar plane. We represent impedance by the letter \( Z \). Therefore,

\[
Z_R = R \\
Z_C = -\frac{j}{\omega C} = \frac{1}{j\omega C} \\
Z_L = j\omega L
\]

Experiment

*The Influence of a Capacitor*

In this section, we will examine how a capacitor influences a circuit using *Capture/PSpice.*

- Create the simple RC circuit shown in Figure A-2 in *Capture*
  - Locate the VSIN source in the SOURCE library. Set the amplitude to 200mV, frequency to 1kHz and offset to 0V.
  - Locate the resistor (R) and capacitor (C) in the ANALOG library. Leave the resistor at 1k, but change the value of the capacitor to 1uF. [In *PSpice*, \( \mu \) represents micro, \( \mu (10^{-6}) \).]
  - Choose the 0 ground from the ground SOURCE library and put the wires into the circuit.
Set up a simulation for this circuit. Choose a Transient Analysis. Run to time 4ms. Choose a step size of 4us.

Place two voltage markers on the circuit: one between the source and the resistor and one between the resistor and the capacitor. The leftmost marker in the circuit will display the input to the circuit. The rightmost marker will display how the circuit is influencing this input, i.e., it displays the output.

Run the simulation. You should get an output with two sinusoids on it. You should see that the circuit has influenced both the amplitude and the phase of the input. How close is the phase shift to $-90^\circ$? Note: Use one of the later cycles to determine this. Copy this plot and include it in your report.

**Hint/Suggestion:** The impedance expressions above for $R$, $L$, and $C$ work in 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 (its steady-state). Thus, when we are using transient analysis to help understand steady-state, we ignore at least the first cycle or two of the sinusoids and focus on the latter part of the signal. You will find at times that the signal does not reach its steady-state in the number of cycles you are displaying. To be sure you are in steady-state, you can set up your simulation profile so that it begins at 10ms and runs to 14ms or even start at 100ms and run to 104ms. Because the circuits we analyze in this course usually have only a small number of components, the extra time necessary to run to 104ms is usually not large. For now, try starting at 10ms and see how things look. For reference, the full simulation of the processor in your computer can take several computer years to complete. This is usually done with many computers and still can take weeks.

**PSpice** has another type of analysis that lets you look at the behavior of a circuit over a whole range of frequencies. It is called an AC sweep. We know that the influence of the capacitor depends on $\omega$ and this is related to 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 pressing on the edit simulation button.
- Choose AC Sweep/Noise from the drop down list box.
- Choose a logarithmic sweep type with a start frequency of 1 and an end frequency of 1Meg. Set the points per decade to 100. (This will give you plenty of points and the plot will be nice and smooth.)
- You need to do one more important thing before you run the simulation. You cannot do an AC sweep without setting a parameter for the VSIN source called AC. Set the AC parameter to your amplitude, 200mV (400mVp-p). Note that each component in *PSpice* has many more parameters than those that appear on the screen. If you double click on your source, you will open the spreadsheet for the VSIN source. Parameter values can be changed in the spreadsheet in addition to clicking on the parameter value next to the component symbol in the circuit diagram.
- Run the simulation.

You should see two traces. These traces are showing you the amplitude of your input and output at all frequencies between 1 and 1MegHz. Note that the horizontal scale of the plot is now in Hertz. The input trace is a straight line at 200mV. This makes sense because the amplitude of the input is 200mV 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 1kHz? Does this match the amplitude of the transient you plotted at 1kHz? Copy the AC sweep plot of your RC circuit and include it in your report.

- We now know that a capacitor will influence the amplitude of a circuit and this amplitude influence changes at different frequencies. What about the phase? Capture/PSpice provides special markers for displaying the phase. First, remove both voltage markers from your circuit. From the Capture main menu choose PSpice→Markers→Advanced→Phase of Voltage. Place two phase markers (marked with VP) on your circuit in the same locations as the voltage markers were before. Rerun the AC sweep simulation. Note: You can also change your display from your plot using the Add Trace option.

- Now you should be looking at two traces that represent the phase of the input and the output of the circuit at all frequencies between 1 and 1MegHz. Note that the y axis is now in degrees. The input phase does not change. It is always zero because the sine wave in PSpice is drawn starting at zero by default. However, the phase for the output over 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°? What is the phase shift at 1kHz? Does this correspond with the phase shift of the later cycles that you got in your transient at 1kHz? Copy the AC sweep plot of the phase of your RC circuit and include it in your report.

The Influence of an Inductor

In this section, we will repeat the procedure above for the simple circuit with an inductor shown in Figure A-3 below.

- Create the RL circuit in Capture
  - Delete the capacitor in your circuit.
  - Locate the inductor (L) in the ANALOG library. Change the value of the inductor to 100mH.

Note: It is better to create a new project for each circuit you analyze rather than just modifying the same one over and over. The circuit project models are very small so they will not take up much space on your computer and it will be easier to compare notes with your partners and re-run things, if necessary, when you are writing your report.

- Return to your transient analysis by choosing Transient in the drop down box. Your simulation values (run to 4ms and step size 4us) should still be there.
- Remove the phase markers and replace the voltage markers on the circuit.
- 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°? Note: Use one of the later cycles to determine this. Copy this plot and include it in your report.
• Now return to your AC sweep analysis by changing the value in the drop down list box. Again the parameters you set before (from 1 to 1Meg with 100 points per decade) should not have changed. 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 1kHz? Does this match the amplitude of the transient you plotted at 1kHz? 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. Remove both voltage markers from your circuit. From the Capture main menu choose PSpice→Markers→Advanced→Phase of Voltage. Place two phase markers (marked with VP) on your circuit in the same locations as the voltage markers were before. Rerun the AC sweep simulation.

• Again, you will see two traces. 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°? What is the phase shift at 1kHz? Does this correspond with the phase shift you got in your transient at 1kHz? 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 learned 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 PSpice.

Part B – 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 1k resistors divides a voltage in half at 10Hz as well as it does at 100kHz. 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 $H(j\omega)$ for any circuit such that

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

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. How and why we can do that is discussed in detail in the course notes. Now, we can combine impedances in the complex polar domain in just the same way that we combine resistances in the time domain. Now we can also easily define our transfer function using the voltage divider rule, as illustrated in Figure B-1.
Filters: Most electrical signals are made up of many frequencies. In the first experiment, 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.

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

\[ Z_{R_1} = R_1. \]
Applying the usual voltage divider relation for this series combination of two impedances gives us

\[ V_B = \frac{Z_{C1}}{Z_{C1} + Z_{R1}} V_A = \frac{1}{j\omega C1} V_A \]

Note that the relationship between the input voltage \( V_A \) and the output voltage \( V_B \) is now 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 PSpice 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

\[ V_{out} = V_B \approx \frac{Z_{C1}}{Z_{C1}} V_A = V_A = V_{in} \]

or, more simply, that

\[ \frac{V_{out}}{V_{in}} = 1 \]

Note that the input and output voltages are essentially identical. 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,

\[ V_{out} = V_B \approx \frac{1}{j\omega R1C1} V_A = \frac{1}{j\omega R1C1} V_{in} \]

or that

\[ \frac{V_{out}}{V_{in}} = \frac{1}{j\omega R1C1} \]

This ratio clearly goes to zero as the frequency goes to infinity. It is also negative imaginary and, thus, has a phase of −90°.

**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.

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

**Experiment**

There are four standard markers available when we want to display the results of our simulations. We have used the first one for voltage at a point. Next we will use the differential marker (the second choice with the two probes) to indicate the voltage difference between two points.
In this section we will explore the impedance of the capacitor circuit. If we want to apply the voltage divider equation to form the transfer function, then the voltage drop over the two components must add up to the input voltage at all frequencies.

- In Capture/PSpice, load the capacitor circuit (Figure A-2) or recreate it by replacing the inductor (Figure A-3) with a 1uF capacitor. Remove the markers from the circuit. 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 $V_C$, use we will use voltage differential markers.
  - Chose a differential marker. Place the positive voltage marker (marked with a plus) at the end of the capacitor closest to the source. This follows the convention that voltage is positive at the end where the current enters. Place the negative marker on the end of the capacitor closest to ground.
  - Place a second differential marker on either side of the resistor.
  - Place a regular voltage marker at the input (next to the source).

- Rerun the transient analysis from part A (4ms with a step size of 4us).

- When you run your simulation, you will obtain three voltages traces. To see the sum of $V_R$ and $V_C$, we must add a trace in PSpice.
  - Find the symbolic name for the $V_R$ trace by looking at the bottom of your plot. Each trace on the plot has a name that corresponds to its color. It should look something like $V(R1:1,R1:2)$. If your resistor has a different name or a different polarity relative to ground, the expression will not be exactly the same.
  - Go to the Trace menu in the PSpice program and choose Add Trace.
  - Type in the name of the $V_R$ trace from the bottom of the plot.
  - Type a + (plus) or click on the + symbol in the functions menu.
  - Find the signal corresponding to $V_C$ on the plot. Type in the name of the $V_C$ trace.
  - The expression in the bottom box should look something like: $V(R1:1,R1:2)+V(C1:2,0)$. (Since the names and polarities of your components may vary, your expression may not be identical.)
  - Click on OK.

- When have added the trace 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 10Hz (run time = 400ms and step size=400us) and 10kHz (run time = 0.4ms and step size=0.4us). 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, $H=V_{out}/V_{in}$, becomes independent of current, and simplifies to an expression in terms of the component values only.

\[
H = \frac{V_{out}}{V_{in}} = \frac{IZ_C}{IZ_R + IZ_C} = \frac{Z_C}{Z_R + Z_C}
\]

*Transfer Function of the Capacitor 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.

- Remove the differential markers from the capacitor circuit (Figure A-2) and replace the two original voltage markers. Run the AC sweep (from 1 to 1MegHz at 100 points per decade).

- We know that $H$ 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 PSpice program and choose Add Trace.
o 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.

o Type in / (divided by) or click on the / symbol in the functions menu.

o Find the signal corresponding to your input and click on it.

o The expression in the bottom box should look something like: V(R1:2) / V(V1:+)

o 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 corner frequency of the circuit is the frequency at which the transfer function is 0.707. Use the cursors to mark this location on your plot.

  o Click on the cursor button on the PSpice toolbar. or
  o Click on the colored symbol for the trace you added (H) at the bottom of your plot. This places the cursor on this trace.
  o Drag the cursor along the trace until the y coordinate is as close as possible to 0.707
  o Mark this point with the button on the toolbar. or
  o What is the corner frequency? Copy this plot and include it in your report.

• In the video lectures or lecture slides you were given an expression for the corner frequency of an RC circuit: \( \omega_c = 1/RC \). Calculate the corner frequency. Don’t forget that f (in Hertz) is \( \omega/2\pi \). How does this compare to the frequency you found with PSpice?

• The equation for the transfer function of a series circuit is \( H = \frac{Z_2}{Z_1 + Z_2} \). 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 PSpice behave the same at low and high frequencies?

Transfer Function of the Inductor 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 Capture/PSpice, recreate the inductor circuit (Figure A-3) by replacing the capacitor with a 100mH 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 corner frequency on your plot. What is the corner frequency? Copy this plot and include it in your report.

• Can you derive the equation for the corner frequency of this circuit? Calculate the corner 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 PSpice?

• Find the transfer function of the inductor circuit using the equation for H. Take the limit of this function at high and low frequencies. Does the plot you made with PSpice behave the same at low and high frequencies?

Build the circuit
Now you can build one of these circuits on your protoboard, hook it to the function generator and compare its behavior to the simulation. You may find the cursors and the Measurements view will be useful.

- Build the capacitor filter circuit (Figure A-2) on your protoboard using a 1uF capacitor and a 1k ohm resistor.

- Set the function generator to a 1kHz signal with an amplitude of 200mV, (400mVp-p).

- Display the input signal on channel 1 and the output signal on channel 2. 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?

- At the corner frequency, the transfer function is equal to 0.707. The transfer function is defined as \(\frac{V_{out}}{V_{in}}\). If \(V_{in}\) is 200mV, what should the amplitude of the output be at the corner 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 corner frequency for an RC circuit is given by \(f_c = \frac{1}{2\pi RC}\). How close is the corner frequency of the circuit you built to the calculated one?

**Summary**

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

**Part C – Transfer Functions, Filters and RLC Circuits**

**Background**

**Phasors:** Circuits, like most engineering systems, can be divided up 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 can write a 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 what is called phasor notation. Euler's identity tells us that exponentials with imaginary arguments can be related to sines and cosines by \(e^{j(\omega t + \phi)} = \cos(\omega t + \phi) + jsin(\omega t + \phi)\) that permits us to write the output voltage, for example, in the equivalent form \(V_{out} = \text{Re}\{V_1 e^{j(\omega t + \phi)}\} = \text{Re}\{V_{out} e^{j\phi}\}\) 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 PSpice simulations, we will be able to add voltage markers that will do this for us. One can access these markers by starting at the PSpice menu, then going to the Markers menu and, finally, the Advanced menu or use Add Trace on plots.
If we apply the phasor form to the equations that characterize voltage and current for capacitors and inductors, we obtain the following:

\[ V_L = \text{Re}\{V_o \, e^{j\omega t}\} = L \frac{dI_L}{dt} = L \left(\frac{d}{dt}\text{Re}\{I_o \, e^{j\omega t}\}\right) = j\omega L \text{Re}\{I_o \, e^{j\omega t}\} = j\omega L I_L \]

or more simply,

\[ V_L = j\omega L I_L \]

and, following the same kind of 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, 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 \( Z = R + jX \).

**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 and series combinations of a resistor, an inductor and a capacitor. This figure shows series RLC circuits with 4 possible input-output choices.

Oscillating systems usually have such a natural resonance. This frequency, where the inductive and capacitive impedances cancel, is called the resonant frequency. The expression for the resonant frequency is \( \omega_0 = \frac{1}{\sqrt{LC}} \).

**More complex filters:** Simple RLC circuits can be used to create more types of filters than simple RC or RL circuits. You can create any of the four generic circuit types shown in Figure C-2.

Each of the above circuits 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 \( H(j\omega) \) for frequencies near zero and for very large frequencies, we can determine which type of filters they are. For example, note that at both high and low frequencies, the transfer function for case C-1(d) is equal to one, and thus the input voltage appears unchanged at the output. [If you want to verify this, redraw the circuit at low and high frequencies replacing the capacitor and the inductor by the appropriate approximation (short or open). You should be able to see that in each case, the output point, C, is at the same voltage as the source. This means that the input signal is appearing 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 across the 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-reject filter.

**Modeling electrical components:** When we build circuits, we use a function generator to produce the input voltage, a resistor, an inductor and a capacitor. However, we cannot use just a voltage source and three other components when we model them in PSpice, since real devices usually cannot be modeled by a single parameter. Let us go through each of the four components of circuits and see what is necessary for realistic analysis or simulation.

- **Function Generator** – Most commercial function generators have an internal resistance of 50Ω. Thus, in a realistic circuit, we must use an ideal sinusoidal voltage source and a resistor to represent the function generator. In the case of the FG on the Analog Discovery this resistance can be neglected.
- **Resistor** – Except at very high frequencies, resistors behave in an essentially ideal manner. Thus it is almost always sufficient to represent the resistor in a circuit as a single resistor.
- **Inductor** – Since inductors are made with a long piece of wire, they usually have a significant resistance, in addition to their inductance. Thus, we must include an additional resistor to model the resistance of 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 to represent a capacitor.

No components we can make are really ideal. Resistors also have inductance and, sometimes, capacitance. Inductors have capacitance. Capacitors have resistance and inductance. Fortunately, we have figured out 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 R1, L1 and C1.

**Experiment**

**Simulation of an RLC Circuit**

In this section, we will use PSpice 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.

![Figure C-3](image)

**Figure C-3.**

Assume that the amplitude of V1 is 200mV, (400mVp-p). Note that there is nothing to represent the impedance of the function generator (assumed zero) and R1 represents the resistance of the 100mH inductor.

- Add a voltage marker at the input of L1 (top of V1). This is the input voltage. Add a second marker at the upper right hand corner between R1 and C1. This is the output voltage. Perform an AC sweep from 10Hz to 100kHz. Since we have now done several AC sweeps, can you explain why we do Decade sweeps rather than Linear sweeps?

- Create a double plot (two plots in the same window) of the magnitude and phase of the transfer function.
- 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 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.
  - You should also calculate the value for the theoretical resonant frequency and mark it on the plot.

\[
 f_0 = \frac{1}{2\pi\sqrt{LC}}
\]

- You will also mark the actual resonant frequency of the circuit you build in the next section on this plot.
  - Note that the calculated value is close to, but not exactly at, the resonant frequency. The equation gives you a simple mathematical way to get close to the resonant frequency of a circuit. Sometimes the expression will give the exact value but it depends upon the configuration of the circuit. The exact value of the resonant frequency can be found by examining when denominator of the transfer function goes to zero. We do not go into these details in this class. For us the approximation is sufficient.

- What type of filter is this?

**Build the Circuit**

Now you can build the circuit in Figure C-3 on your protoboard, hook it to the function generator and compare its behavior to the simulation.

- Before you build the circuit, you should measure the exact values of the components.
  - Measure the DC resistance of the inductor’s resistor using the **DMM**. Do not use the impedance bridge for this measurement.
  - Measure the inductance of the inductor and the capacitance of the capacitor using the impedance bridge on the center table. The connectors may be a bit loose. Be sure they are making good contact.

- Change your **PSpice** simulation to correspond to the actual parameters of your circuit. 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 protoboard. Remember that you need only two components: the 100mH inductor (with an internal resistance of about 90Ω) and the 1µF capacitor. The 1µF capacitor will have 105 written on it if it doesn’t have 1 µF written directly. Remember that R1 represents the resistance of the 100mH 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 it is close to the resonant frequency you found using PSpice and the one you calculated using the theoretical equation. It will 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, the same circuit gave you three similar, but not exact, values.

- Determine 5 representative frequencies that cover the range of your simulation: a low frequency, a high frequency, the resonant frequency, some point between the resonant frequency and the high frequency, and some point between the resonant frequency and the low frequency. Do not choose a point where the output disappears completely.
• 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 scope and manually determining the ratio of the amplitudes and the phase shift. Cursors will be useful.

• Put the values that you measured on the PSpice AC sweep plot you made using the actual 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 covered 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 the parallel components until we have a series circuit and then we can apply the voltage divider. 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:

\[
\begin{align*}
\text{series} & : Z_T = Z_1 + Z_2 + \cdots + Z_n \\
\text{parallel} & : \frac{1}{Z_n} = \frac{1}{Z_1} + \frac{1}{Z_2} + \cdots + \frac{1}{Z_n}
\end{align*}
\]

Experiment

*Combining Impedances*

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

- Set up a voltage divider in Capture with two 50Ω resistors and a VSIN source with 200mV amplitude, 1kHz 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 1Hz and an end frequency of 15MEG Hz. You should use about 100 points per decade. The AC Sweep Type has been chosen as *Logarithmic - Decade* since frequency effects usually only become obvious when we change orders of magnitude. This generates a log...
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 function generator.

- For what range of frequencies does the capacitor change the voltage across R2 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 figure out at least the magnitude of the equivalent impedance of the combination of R2 and C1 at the frequency of 1MegHz.
  - Go back to your schematic and change the value of the source (V1) frequency to 1MEG.
  - Set up a transient analysis. Click on the Edit Simulation Settings button and set up for transient analysis using smaller times. Since 1MHz is 1000 times larger than 1kHz, 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, R2 and C1 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 R2 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 R2 and the capacitor seems to not 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 node label at the bottom of the PROBE window that corresponds to the side of resistor R1 connected to the source.
  - Hit the Delete button on your computer keyboard.

- Using the cursors, determine the amplitude of the sine wave oscillation across R2 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 R2/C1 combination, that we are calling Z. [Hint: Figure D-2 shows the voltage divider. You can use the voltage divider equation: \( V_z = V_1 \times \frac{Z}{R_1 + Z} \)]

- Check your answer by replacing the capacitor/resistor combination with a single resistor with the value of Z you calculated. Rerun the simulation. Does the output have the same amplitude as the combination?

- Now we can check if the transfer function gives us the same amplitude. Use the parallel rules for impedance to combine \( \frac{1}{j\omega C} \) and R. Then set up the transfer function for the circuit. Determine its magnitude at 1MegHz. (Don’t forget that \( \omega = 2\pi f \)). Multiply the value of \( H \) at this frequency by the input voltage amplitude (200mV). Is the output amplitude of your plot comparable to the one you calculated?
Summary

In this section you explored what happens when you combine two impedances in parallel. You have learned that it is possible to find transfer functions for and analyze circuits with components in parallel. You simply need to combine the impedance, $Z$, of the parallel components into an equivalent impedance using rules similar to those used to combine resistors.
Checklist and Conclusions

Provide the following packet. Include the cover/signature page attached to the end of this handout. The signatures are required for all statements with a signature line next to it. They must be signed by TAs or Professor(s) after seeing the results on the computer screen. Give all required results and answer the questions concisely. It is intended that these be quick to write.

The following should be included in your experimental checklist. 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 THE SIGNAL AT WHICH POINT.

Hand-Drawn Circuit Diagrams for all circuits that are to be analyzed using PSpice or physically built and characterized using your Analog Discovery board.

Part A –RC circuit, RL circuits, and AC Sweeps (22 points)

Include the following plots:
1. PSpice transient plot of RC circuit (in Figure A-2). (2 pt)
2. PSpice AC sweep plot of the RC circuit voltage. (2 pt)
3. PSpice AC sweep plot of the RC circuit phase. (2 pt)
4. PSpice transient plot of RL circuit (in Figure A-3). (2 pt)
5. PSpice AC sweep plot of the RL circuit. (2 pt)
6. PSpice AC sweep plot of RL circuit phase. (2 pt)

Answer the following questions:
1. What is the amplitude and phase of the output of the RC circuit at 1kHz? (2 pt)
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? (2 pt)
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°? (2 pt)
4. What is the amplitude and phase of the output of the RL circuit at 1kHz? (2 pt)
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? (1 pt)
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°? (1 pt)

Part B - Transfer Functions and Filters (26 points)

Include the following plots:
1. PSpice plot of capacitor and resistor sum adding to the input voltage at 1kHz. (2 pt)
2. PSpice plot of capacitor and resistor sum adding to the input voltage at 10Hz. (1 pt)
3. PSpice plot of capacitor and resistor sum adding to the input voltage at 10kHz. (2 pt)
4. PSpice plot of transfer function of RC circuit (Figure A-2) with corner frequency marked. (2 pt)
5. PSpice plot of transfer function of RL circuit (Figure A-3) with corner frequency marked. (2 pt)
6. Wave Forms picture of RC circuit at 1kHz. (1 pt)
7. Wave Forms picture of RC circuit at corner frequency. (1 pt)

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). (1 pt)
2. What kind of filter is the RC circuit? (1 pt)
3. At what frequency did you find the corner on the PSpice 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? (3 pt)
4. Find the transfer function of the RC circuit in terms of R, C and jω. Take the limit of this function at very low and very high frequencies. Show that these results are consistent with the PSpice plot of the transfer function. (3 pt)

5. What kind of filter is the RL circuit (Figure A-3)? (1 pt)

6. Derive the equation for the corner frequency of the RL circuit. (2 pt)

7. At what frequency did you find the corner on the PSpice 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? (3 pt)

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? (1 pt)

Part C (16 points)

Include the following plots:
1. PSpice plot of the transfer function of the RLC circuit in Figure C-3 (magnitude and phase), with three resonant frequency values marked [PSpice value, calculated value, experimental value] (original component values). (3 pt)
2. PSpice 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. (3 pt)

Answer the following questions:
1. Why is it necessary to plot the phase and the magnitude of the transfer function separately, rather than on the same plot? (1 pt)
2. For the three RLC circuits labeled (a), (b) and (c) 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 (d) 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 at high frequencies. Also recall that an inductor can be modeled as a short 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 PSpice if you want. (3 pt)
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? (2 pt)
4. Determine the resonant frequency of the RLC circuit you analyzed with PSpice. (This occurs at the extreme point.) Calculate the resonant frequency with the equation f = 1/[2π√(LC)]. How similar are they? What factors do you think account for the discrepancy? (3 pt)
5. Why do you suppose it is that, in practice, we generally use filters designed with capacitors and not inductors? (1 pt)

Part D (12 points)

Include the following plots:
1. PSpice AC Sweep plot of circuit in Figure D-1 with 5% point marked with cursor. (2 pt)

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%? (1 pt)
2. What is the amplitude of the output of the parallel RC circuit? What is the value you calculated for the equivalent impedance (Z) of the parallel combination at 1MegHz? (3 pt)
3. Find the transfer function for the parallel RC circuit. Determine the magnitude of the transfer function at 1MegHz. Calculate the amplitude of the output voltage for this circuit at 1MegHz? How well does it agree with the output amplitude you found using PSpice? (6 pt)

List member responsibilities (4 pt) Note that this is a list of responsibilities, not a list of what each partner did. It is very important that you divide the responsibility for each aspect of the experiment so that it is clear who will make sure that it is completed. Responsibilities include, but are not limited to, reading the full write up before the first
class; collecting all information and writing the report; building circuits and collecting data (i.e. doing the experiment); setting up and running the simulations; comparing the theory, experiment and simulation to develop the practical model of whatever system is being addressed, etc.

New Summary/Overview Section There are two parts to this section, both of which require revisiting everything done on this experiment and addressing broad issues. In this experiment, you must complete this section and you will receive feedback on your work, but no formal grade. That will begin with Exp3.

1. Application: Identify at least one application of the content addressed in this experiment. That is, find an engineered system, device, process that is based, at least in part, on what you have learned. You must identify the fundamental system and then describe at least one practical application.

2. Engineering Design Process: Describe the fundamental math and science (ideal) picture of the system, device, and process you address in part 1 and the key information you obtained from experiment and simulation. Compare and contrast the results from each of the task areas (math and science, experiment, simulation) and then generate one or two conclusions for the practical application. That is, how does the practical system model differ from the original ideal?

Example from Experiment 1:

1. One of the key circuit configurations we will see is the voltage divider. You should be on the lookout for it in every experiment and project because almost all circuits involve some combination of components in series. Most applications of a voltage divider use this device to provide a reference voltage. For example, in a thermostat, a temperature sensor will output a voltage proportional to temperature. This voltage must be compared to a reference to see if a furnace or air conditioner should be turned on. If the sensor voltage is above the reference then the A/C is turned on, etc. This can also be used for a light sensor system or for the control of anything that can be measured by a sensor that outputs a voltage.

2. The ideal operation of a voltage divider can be easily derived from Ohm’s Law and simple circuit analysis. For the configuration shown at the right, the relationship between the input and output voltages is given by this equation. If you have a voltage source \( V_{in} \), then a reference voltage at any level from \( V_{in} \) to 0 can be produced by this circuit. In practical application, it is necessary to measure the voltage \( V_{out} \) in order to use it to control something like a lamp or furnace. In both simulation and experiment, connecting an oscilloscope or meter across \( Z_2 \) will add a resistor to the circuit in parallel with \( Z_2 \). Both the experiment and simulation showed that this additional resistor must be much larger than \( Z_2 \) or the reference voltage will be changed. Thus, from experiment and simulation, we conclude that a more complete model of the practical voltage divider (at least at this point in this course) should include the load and the restriction that the load resistance must be much larger than the divider resistors. The ideal divider info comes from Wikipedia and the loaded divider from a website at Tufts. The loaded divider is the practical system model that gives us confidence to reliably use the divider in a real world application.
Experiment 2

Section: ______
Report Grade: ______
____________________________________ Name
____________________________________ Name

Checklist w/ Signatures for Main Concepts

For all plots that require a signature below, you must explain to the TA or instructor:
• the purpose of the data (using your hand-drawn circuit diagram),
• what information is contained in the plot and
• why you believe that the plot is correct.
Any member of your group can be asked for the explanation.

PART A: RC circuit, RL circuits, and AC Sweeps

1. PSpice transient of RC circuit
2. PSpice AC sweep plot RC voltage _________________________
3. PSpice AC sweep plot RC phase
4. PSpice transient of RL circuit
5. PSpice AC sweep plot RL voltage
6. PSpice AC sweep plot RL phase _________________________
Questions 1-6

PART B: Transfer Functions and Filters

1. PSpice plot of capacitor and resistor at 1kHz
2. PSpice plot of capacitor and resistor at 10Hz
3. PSpice plot of capacitor and resistor at 10kHz
4. PSpice plot of transfer function of RC circuit marked corner frequency
5. PSpice plot of transfer function of RL marked corner frequency
6. Wave Forms picture of RC circuit at 1kHz ______________
7. Wave Forms picture of RC circuit at corner frequency_____
Questions 1-8

PART C: Transfer Functions, Filters, and RLC Circuits

1. PSpice plot of transfer function of RLC circuit (see above for markings)
2. PSpice plot of transfer function of RLC circuit (see above for markings)
Questions 1-5

PART D: Equivalent Impedance

1. PSpice AC sweep plot D-1 (5% point marked)______________
Questions 1-3

Member Responsibilities