

## Experiment 2 Complex Impedance, Steady State Analysis, and Filters

**Purpose:** The objective of this exercise is to learn about steady state analysis. You will also be introduced to filters.

Equipment Required

- **DMM** (Digit Multimeter)
- **Rensselaer IOBoard Rev D** (with Mobile Studio Desktop)
- **Oscilloscope** (Rensselaer IOBoard)
- **Function Generator** (Rensselaer IOBoard)
- **DC Power Supply** (Rensselaer IOBoard)
- **Impedance Bridge** (located on center table)
- **Components:** 1k resistor, 1 $\mu$ F capacitor, 100mH inductor, 0.068 $\mu$ F (683) capacitor
- **Protoboard**

Helpful links for this experiment can be found on the links page for this course:

<http://hibp.ecse.rpi.edu/~connor/education/EILinks.html#Exp2>

### 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:  $z = x + jy$ .  $j$  (i 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  $\theta = \tan^{-1}\left(\frac{y}{x}\right)$ .

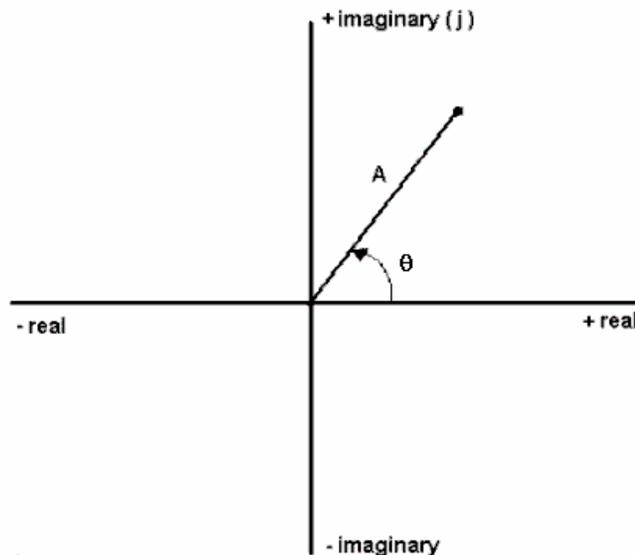


Figure A-1.

*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). 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  $1/\omega C$  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,

$$Z_R = R \quad Z_C = -\frac{j}{\omega C} = \frac{1}{j\omega C} \quad Z_L = j\omega L$$

## 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*, u represents micro,  $\mu$  ( $10^{-6}$ ).
  - Choose the 0 ground from the ground SOURCE library and put the wires into the circuit.

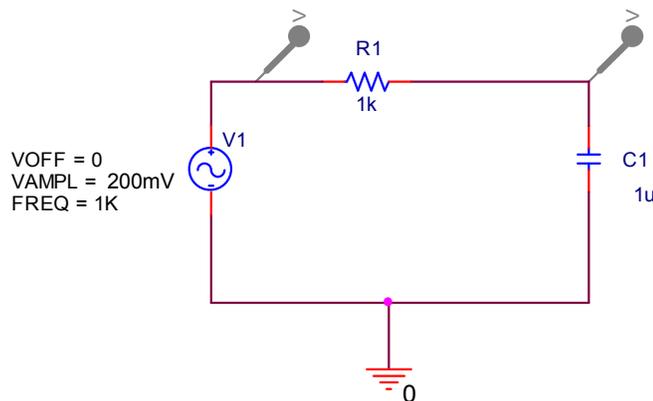


Figure A-2.

- 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 at the input and one at the corner. The leftmost marker in the circuit shows you the input to the circuit. The rightmost marker shows you how the circuit is influencing this input.
- 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. Print this plot and include it in your report.
- *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.)
  - You need to do one more *important* thing before you run the simulation. For some reason known only to *PSpice*, you cannot do an AC sweep without setting a parameter for the VSIN source called AC. 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 spread sheet for the VSIN source. Set the AC parameter to your amplitude, 200mV, ( 400mVp-p.)

- Run the simulation.
- You should see two traces. These traces are showing you the amplitude of your output at all frequencies between 1 and 1MHz. 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. 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? Print 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 looking at 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.
- 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 1MHz. 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  $-90^\circ$ ? 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? Print 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.

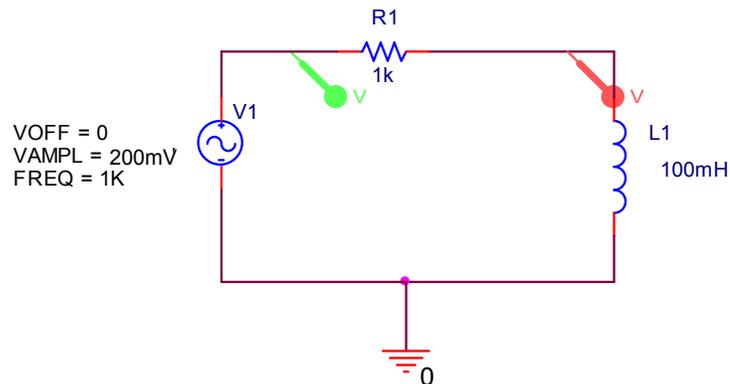


Figure A-3.

- 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.
- 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  $+90^\circ$ ? Note: Use one of the later cycles to determine this. Print 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? Print 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  $+90^\circ$ ? What is the phase shift at 1kHz? Does this correspond with the phase shift you got in your transient at 1kHz? Print 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, it 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.

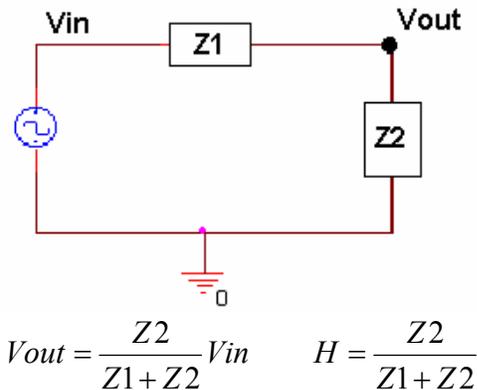


Figure B-1.

*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. In electronics a circuit that filters out certain frequencies while allowing others to remain unchanged is called a filter.

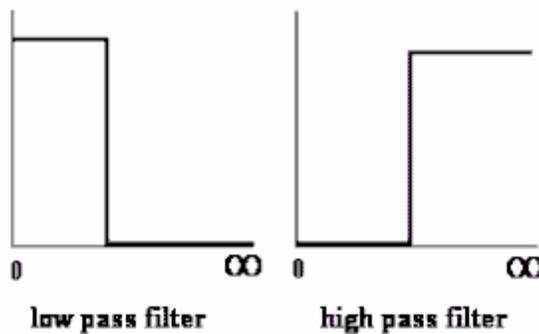


Figure B-2.

The two basic types of filters we will consider in this part are low pass filters (LPF) and high pass filters (HPF). 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 C1 is equal to  $Z_{C1} = \frac{1}{j\omega C1}$ , while the impedance of the resistor R1 is  $Z_{R1} = R1$ .

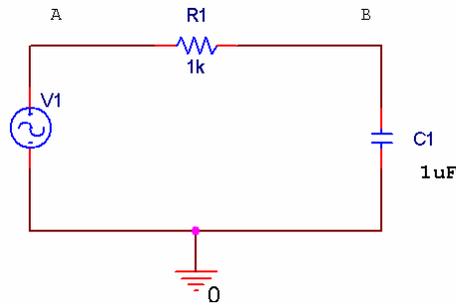


Figure B-3.

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

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

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

$$V_{out} = V_B \approx \frac{Z_{C1}}{Z_{C1}} V_A = V_A = V_{in} \text{ or, more simply, that } \frac{V_{out}}{V_{in}} = 1. \text{ Note that the input and output voltages are}$$

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,

$$V_{out} = V_B \approx \frac{1}{j\omega R1C1} V_A = \frac{1}{j\omega R1C1} V_{in} \text{ or that } \frac{V_{out}}{V_{in}} = \frac{1}{j\omega R1C1}. \text{ This ratio clearly goes to zero as the}$$

frequency goes to infinity. It is also negative imaginary and, thus, has a phase of  $-90^\circ$ .

*Note about “Low” and “High”:* While it may seem that terms like low and high, when applied to something like frequency, are fuzzy, ill-defined terms, 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  $1/\sqrt{2}$ .

## Experiment

### 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 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  $V_C$ , 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 in + (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. Print out 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. Print 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.
  - 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 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.
  - Click on the cursor button on the *PSpice* toolbar. 
  - 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. 
  - What is the corner frequency? Print this plot and include it in your report.
- In class you were given an expression for the corner frequency of an RC circuit:  $\omega_c = 1/RC$ . 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*?
- 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? Print 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.

- 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. Take a picture of this signal with the camera function in the software and include it in your report. 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  $V_{out}/V_{in}$ . 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  $f_c=1/(2\pi RC)$ . 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  $H = Z_{out}/Z_{in}$ .

## 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  $e^{j(\omega t + \phi)} = \cos(\omega t + \phi) + j\sin(\omega t + \phi)$  that permits us to write the output voltage, for example, in the equivalent form  $V_{out} = \text{Re}\{V_1 e^{j(\omega t + \phi)}\} = \text{Re}\{V_{out} e^{j\omega t}\}$  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 *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.

If we apply the phasor form to the equations that characterize voltage and current for capacitors and inductors, we obtain the following  $V_L = \text{Re}\{V_o e^{j\omega t}\} = L \, dI_L/dt = L \, (d/dt) \text{Re}\{I_o e^{j\omega t}\} = j\omega L \, \text{Re}\{I_o e^{j\omega t}\} = j\omega L \, I_L$  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, Ohms 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.

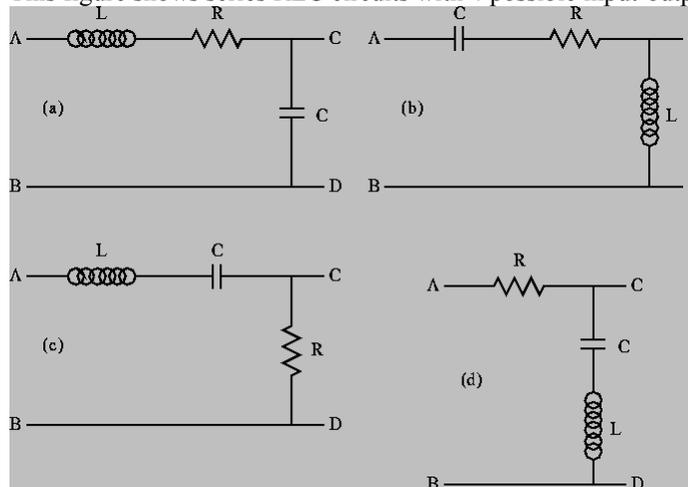


Figure C-1.

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 circuit types shown in Figure C-2.

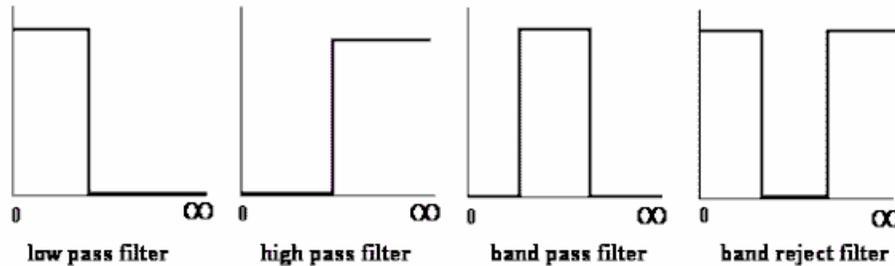


Figure C-2.

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** – We have already seen several times that many function generators have an internal resistance of  $50\Omega$ . 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 IOBoard 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.

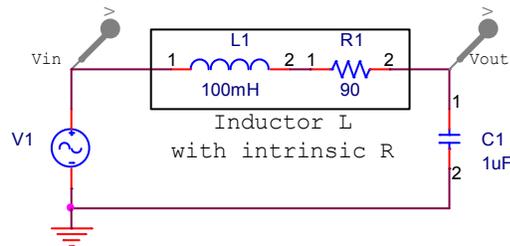


Figure C-3.

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 of the magnitude and phase of the transfer function.
  - Use the Add Plot 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?

- Print out this plot. Label the resonant frequency (extreme point) with the cursors.
- You should also calculate the value for the theoretical resonant frequency using the equation 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, 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**. The impedance bridge may not work as well for this measurement.
  - Measure the inductance of the inductor and the capacitance of the capacitor using the impedance bridge on the center table.
- 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. Print out 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.
- 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:

$$\begin{aligned} \text{series} \quad Z_T &= Z_1 + Z_2 + \dots + Z_n \\ \text{parallel} \quad \frac{1}{Z_n} &= \frac{1}{Z_1} + \frac{1}{Z_2} + \dots + \frac{1}{Z_n} \end{aligned}$$

### Experiment

#### Combining Impedances

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

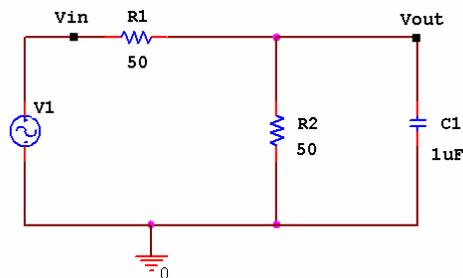


Figure D-1.

- Set up a voltage divider in *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 open the spreadsheet for the VSIN source and set the AC parameter to the amplitude of your signal, 200mV.*
- Set up an AC sweep of this circuit. Do a logarithmic sweep with a start frequency of 1Hz and an end frequency of 15MEG. 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 the HP 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. Print out 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:  $V_z = V_1 * (Z)/(R_1 + Z)$ ]

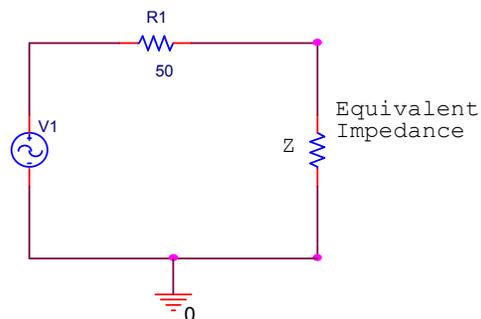


Figure D-2.

- 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  $1/j\omega C$  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.

## Report and Conclusions

The following should be included in your report. Everything should be labeled and easy to find. Partial credit will be deducted for poor labeling or unclear presentation.

### Part A (16 points)

Include the following plots:

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

Answer the following questions:

1. What is the amplitude and phase of the output of the RC circuit at 1kHz? (1 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  $-90^\circ$ ? (2 pt)
4. What is the amplitude and phase of the output of the RL circuit at 1kHz? (1 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? (2 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  $+90^\circ$ ? (2 pt)

### Part B (24 points)

Include the following plots:

1. *PSpice* plot of capacitor and resistor sum adding to the input voltage at 1kHz. (1 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. (1 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. Mobile Studio picture of RC circuit at 1kHz. (1 pt)
7. Mobile Studio 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  $j\omega$ . 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)

## Electronic Instrumentation

ENGR-4300

Experiment 2

- 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:

- 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)
- 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:

- Why is it necessary to plot the phase and the magnitude of the transfer function separately, rather than on the same plot? (1 pt)
- 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)
- 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)
- 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  $f = 1/[2\pi\sqrt{LC}]$ . How similar are they? What factors do you think account for the discrepancy? (3 pt)
- 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:

- PSpice* AC Sweep plot of circuit in Figure D-1 with 5% point marked with cursor. (2 pt)

Answer the following questions:

- When the capacitor is added, for what range of frequencies does the capacitor change the voltage across R2 by less than 5%? (1 pt)
- 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)
- 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)

### Summary (12 points)

- Summarize key points (8 pt)
- Discuss mistakes and problems (2 pt)
- List member responsibilities (2 pt)

**Total: 80 points for write up +20 for attendance = 100 points**

**Attendance: 3 classes (20 points), 2 classes (10 points), 1 class (0 points), out of 20 possible points**

**Minus 5 points for each late.**

**No attendance at all = No grade for experiment.**