



## Class #7: Experiment L & C Circuits: Filters and Energy Revisited

In this experiment you will revisit the voltage oscillations of a simple  $LC$  circuit. Then you will address circuits made by combining resistors with inductors and capacitors. This should be done as a team of two students.

*Be sure that one of your team reads the entire write-up before beginning your experiments.*

**Purpose:** The objective of this experiment is to learn more about energy storage, steady state analysis and basic filters.

Background: Before doing this experiment, students should be able to

- Use Ohm's Law to determine the voltage across, current through and/or resistance of a resistor when given information on the other two parameters.
- Determine the total resistance of a small number of resistors in series or parallel
- Determine the transfer function of a two resistor voltage divider.
- Determine the power generated by a DC voltage source or dissipated by a current carrying a DC current.
- Do a transient (time dependent) simulation of RC, RL and RLC circuits using LTspice IV.
- Determine the resonant frequency of an LC circuit.
- Determine the values of capacitors and inductors from the information printed on them.
- Review the background for previous experiments.

Learning Outcomes: Students will be able to

- Do an AC sweep (frequency dependent) simulation of RC, RL and RLC circuits using LTspice IV, determining both the magnitude and the phase of input and output voltages.
- Experimentally determine how RC, RL and RLC circuits behave at high and low frequencies.
- Be able to define what is meant by high and low frequencies in the context of RC, RL and RLC circuits.
- 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
- Find the corner frequency for RC and RL circuits.
- 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.

Equipment:

1 inductor, inductance  $L = 1$  mH

1 capacitor  $C = 1$   $\mu$ F

1 resistor 1 k $\Omega$

Analog Discovery function generator and oscilloscope

1 diode (1N914) **from parts kit**

A protoboard

## Pre-Lab

*Required Reading:* Before beginning the lab, read over and be generally acquainted with this document and the other **reading and viewing** materials listed under Class #7 on the ECSE 1010 website.

*Hand-Drawn Circuit Diagrams:* Before beginning the lab, hand-drawn circuit diagrams must be prepared for all circuits either to be analyzed using SPICE or physically built and characterized using your Analog Discovery board.

### A. Revisiting Energy Storage

In the experiment done during Class #5 (Electromagnetic Oscillations LC Circuits), you were asked to export the data in your plot, saving it as a comma-separated-value (csv) file called DiodeLC.csv. These instructions are found right after question 5. Following the directions

- Open the file in Excel (see videos). You should have three columns: one for time, one for the source voltage and one for the voltage across the capacitor and inductor. If your data is good, the voltage across the capacitor/inductor should be a decaying sinusoid (aka damped sine wave – see Wikipedia [https://en.wikipedia.org/wiki/Damped\\_sine\\_wave](https://en.wikipedia.org/wiki/Damped_sine_wave)). Note that both cosine and sine waves are generally called sinusoids, as you can see in the Wikipedia entry. Plot the two voltages for the range of time starting at the beginning of the damped sine wave and ending when the oscillations of the wave are barely visible. The plot should look like the damped sine wave at the upper right on the Wikipedia page. Save and fully annotate the plot in your report.
1. You found the oscillation frequency in the previous experiment. Write the value down since you will be using it again in this experiment. This information should be included in your annotated plot.

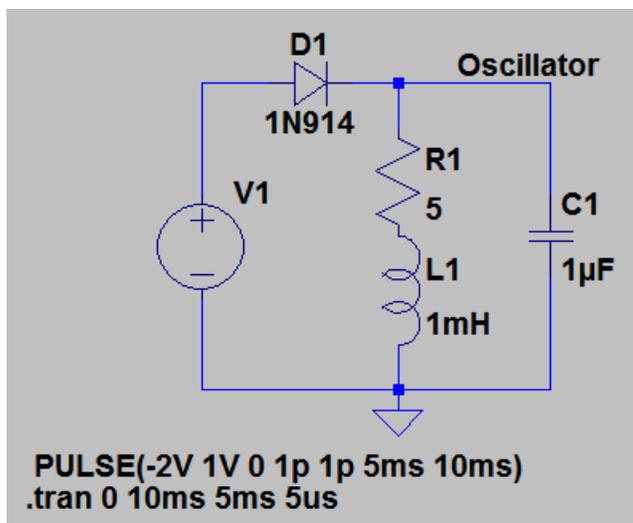


Figure A-1

- Next we want to simulate the circuit using LTspice. The circuit to be simulated is shown in Figure A-1. Note that a realistic model of an inductor is used that includes resistance.

Run the simulation, plot the input and output voltages (using LTspice) and save and fully annotate the plot in your report document. For the equivalent series resistance of the inductor, you will have to look up what value to use in the inductor spec sheet. You can find the spec sheet by going to Digilent's list of components in your parts kit.

We are simulating the experiment because the simulation will make it possible to determine more parameters than we can get experimentally. (e.g. the currents in each component). Once we verify that the simulation gives essentially the same result as the experiment (for the parameters that can be measured experimentally), we are generally free to use the additional details available.

- In addition to the input and output voltages, also plot the inductor current. You do not need to save this plot for your report. We will do the plotting in Excel. Export your data to a new .txt file (remember the name and location). Then import it into Excel (instructions are in the first Excel video). You should have four columns of data: time, input voltage, capacitor (output) voltage and inductor current.

Now we will use Excel to determine and plot the energy stored in the inductor and capacitor as a function of time.

- Create two new columns in Excel. The first will be the energy in the inductor  $\frac{1}{2}LI^2$ . The second will be the energy in the capacitor  $\frac{1}{2}CV^2$ . Plot the two energy columns as a function of time. Save and fully annotated this plot in your report.
2. Discuss the two energy values, how they change with time, their relation to one another ... For example when is the energy stored mostly or entirely in the inductor? When is it stored mostly or entirely in the capacitor? What trends with time do you see? Be as quantitative as you can, but mostly address the big picture.

We will return to determining the energy after Part B.

## B. RL, RC, RLC Filters

In Part B, we will investigate how filters work using our knowledge of voltage dividers. As a quick reminder, a version of a generic voltage divider circuit is shown in Figure B-1. For this circuit

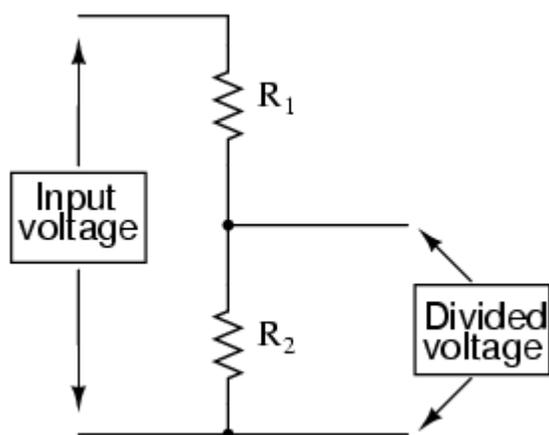


Figure B-1

$V_{out} = V_{in} \frac{R_2}{R_1 + R_2}$ . In this experiment, we will

replace  $R_2$  with a capacitor and then observe how the capacitor works at high and low frequencies. This will allow us to understand the general operation of filters.

- Build the circuit at the left using a  $1k\Omega$  resistor for  $R_1$  and a  $1\mu F$  capacitor in place of  $R_2$ . For the voltage supply, use a 10Hz sinusoid with an amplitude of 1V (2V peak-to-peak). Plot both the input and output voltages (output across C), save the plot in your report and fully annotate it (e.g. amplitude, frequency).
- Repeat the measurement with a source frequency of 10kHz. For both plots, you should show 3 to 5 cycles of the sinusoidal voltages.

- From your knowledge of how voltage dividers work, identify which of your two cases shows the capacitor acting approximately like a short circuit (zero resistance) and which case shows the capacitor acting approximately like an open circuit (near infinite resistance). Show your working experiment to a TA or instructor, along with your annotated plots and your answer to this question. Explain why your answer makes sense. Obtain a signature that you have completed this task.

For one of the cases you identified, the output voltage is essentially identical to the input voltage. For this frequency, we say that the RC circuit passed the input to the output. For the other case, the output voltage was near zero. For this latter case, we say that the RC circuit blocks the input. Circuits that pass some signals and block others are called filters. Circuits that pass low frequencies and block high frequencies are called low-pass filters. Circuits that pass high-frequencies and block low frequencies are called high-pass filters.

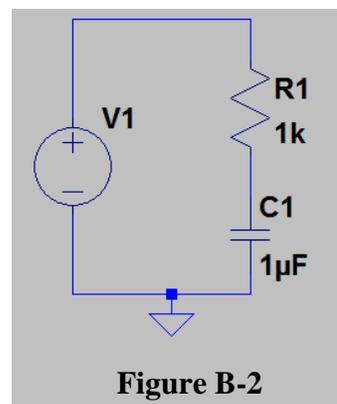
- What kind of filter is the RC circuit you built? What kind of filter would the circuit be if you swapped the positions of the R and C?

To be able to really characterize how RC filters work, it is useful to have a measure of what we mean by high frequencies and low frequencies. The ratio of the output voltage to the input voltage is about 1 for frequencies passed by the filter and about zero for frequencies blocked. The boundary between high and low is given by what is called the corner frequency, which we can determine from the product of R and C, just like the charging and discharging time constant.  $f_c = \frac{1}{2\pi RC} \text{ Hz}$ . Note that, at this point, you may find it useful to look up high pass and low pass filters and corner frequency. Focus on RC circuits. LR circuits work in a similar manner.

- Calculate the corner frequency for your RC circuit.
  - Repeat your measurement, setting the source voltage frequency to the corner frequency. Plot and save it to your report, with full annotation. If you have done this correctly, the ratio of the output to the input voltage should be a little more than 0.7.

Practically speaking, a frequency is low if it is much less than the corner frequency and high if it is much more. Exactly how much less or more depends on the application. You are free to define the conditions to be met. To see more clearly how this works for the RC circuit, we will use LTspice to simulate its operation at a large range of frequencies.

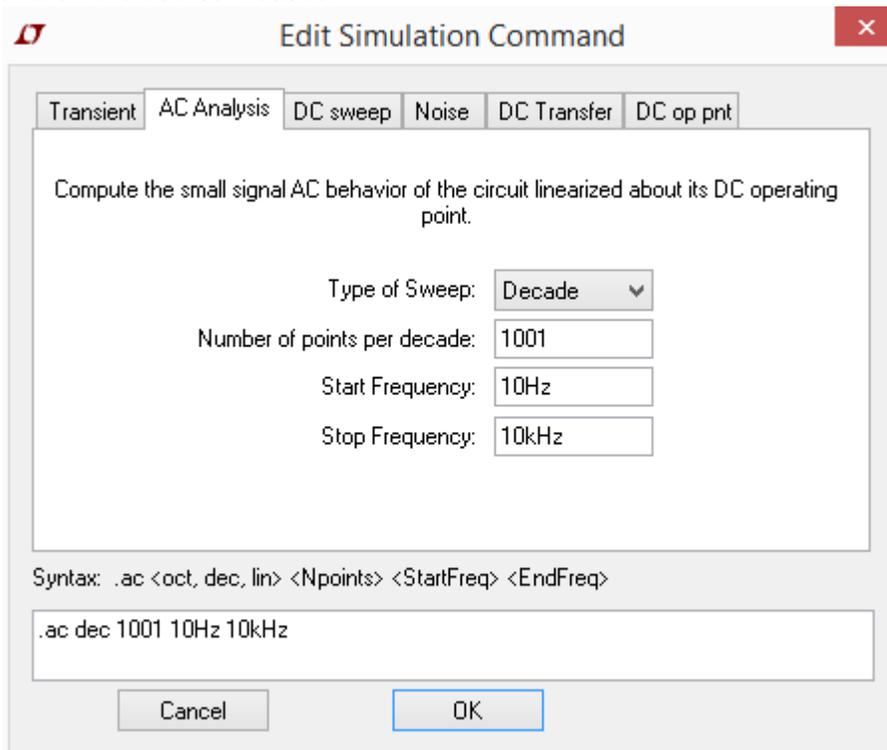
- Set up the circuit in LTspice as shown in Figure B-2. To see a range of frequencies, choose AC Analysis rather than Transient when you set up your simulation. The simulation window will look like Figure B-3. Use the parameters shown to obtain the circuit response from 10Hz to 10kHz. A large number of points per decade is chosen to produce a really smooth plot. As usual, plot both the input (source) voltage and the output voltage across the capacitor. Unfortunately, the plot will not be in a useful form for us without some modifications. First, hover your mouse pointer over the scale on the left and then click the mouse. A new window will appear that will let you change the type of plot. Find a drop-



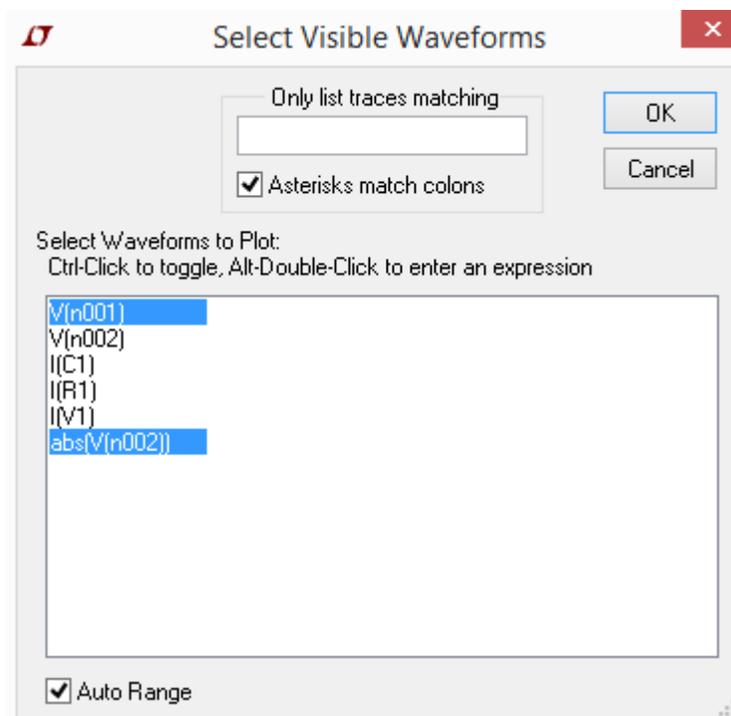
**Figure B-2**

down menu that says Bode and change the plot to Cartesian. This will make the vertical scales linear like we see with Analog Discovery.

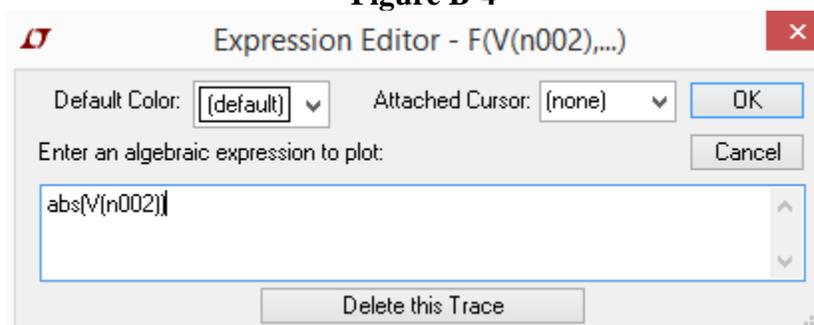
Next hover your mouse over the scale on the right and click your mouse. In the window that appears, choose 'don't plot the imaginary component.' Finally, click on the menu button for Pick Visible Traces. A window like Figure B-4 will appear showing all possible voltage and current traces that you can display. We want to create a new type of display that uses a mathematical expression. It says in the window that you should hold down the Alt key and double click to enter an expression. Specifically, hold down the Alt key and double click on the output voltage {here it is V(n002)}. We want to display the absolute value of the voltage so the expression you should create is  $\text{abs}(V(n002))$ . Type that in the new window that appears (Figure B-5). After you select OK and go back to the Select Visible Waveforms window, make sure that both this new expression and the input voltage are highlighted so both will be displayed. If the input is missing, hold down the Ctrl key and click on the symbol for the input voltage. Then select OK and you should see the input and output voltages for all frequencies from 10Hz to 10kHz. Note that information on the wide range of mathematical expressions available for creating interesting LTspice plots can be found on the course website.



**Figure B-3**



**Figure B-4**



**Figure B-5**

- Save the plot you just created to your report and annotate it. Include the following information: Input Voltage, Output Voltage, the three frequencies used for Analog Discovery measurements (10Hz, the corner frequency, 10kHz). Note that your horizontal (frequency) axis will be logarithmic to make the features of the circuit response easier to identify. If you have trouble understanding how it works, ask a TA or instructor.
6. For what range of frequencies would you consider the output voltage to be approximately equal to the input? To be quantitative, the output should be at least 95% or more of the input. For what range of frequencies would you consider the output voltage to be approximately zero? Again, to be quantitative, the output should be less than 5% of the input.
  7. From your knowledge of how voltage dividers work, identify which of your two ranges shows the capacitor acting approximately like a short circuit (zero resistance) and which case shows the capacitor acting approximately like an open circuit (near infinite resistance). Show your working simulation to a TA or instructor, along with your annotated plots and

your answer to this question. Explain why your answer makes sense. Obtain a signature that you have completed this task. \_\_\_\_\_

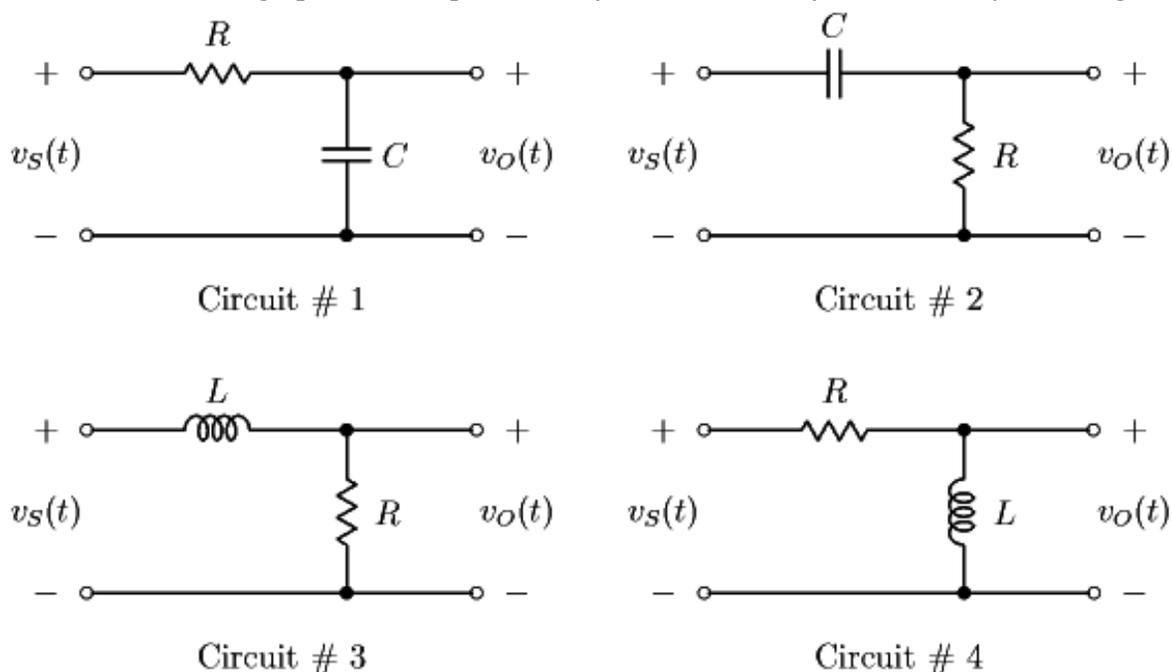
You should now understand how a capacitor behaves at low and high frequencies and have a general idea of how we define what we mean by low and high. In one frequency range (low or high) it will behave approximately like a short circuit and in the other it will behave like an open circuit.

8. In the table below, write the expressions ‘open circuit’ and ‘short circuit’ in the appropriate cells for the capacitor. We have not demonstrated it yet, but the frequency regions for inductors are exactly the opposite. Using this knowledge, fill in the cells for ideal inductors. Note that the cells for practical inductors are already filled in. Inductors are made with a fairly long piece of wire so they always have at least a small, but finite resistance. Thus, they can never behave like a short circuit. You may have already figured this out, but you can check your low frequency responses with an ohmmeter because DC ( $f = 0$ ) is clearly a low frequency.

Circuit Component	Low Frequencies	High Frequencies
Capacitor		
Inductor (Ideal)		
Inductor (Practical)	Small Resistor	Open Circuit

There are four possible two-component circuits that can be constructed with combinations of resistors, inductors and capacitors, as shown in Figure B-6. So far, we have only done experiments with Circuit #1. Each of these circuits is either a high pass or a low pass filter.

9. From what you know about Circuit #1 and the information in the table, label each of the circuits as high pass or low pass. Hint: you can confirm your answer by checking online.



**Figure B-6**



### C. Revisiting Energy Storage (Again)

In part A, you plotted the energy stored in the inductor and capacitor as a function of time. We would now like to use the information we have obtained on how inductors and capacitors behave at low and high frequencies to analyze the LC circuit just before it starts to oscillate.

- Redraw the circuit in Figure A-1 for DC or near DC frequencies. That is, use what you know about how practical L and C behave at low frequencies. (Here is where your ohmmeter measurements come in especially handy.) If a component behaves like a short circuit, replace it with a wire. If it behaves like an open circuit, replace it with a break in the wire. Label the oscillator voltage with the value you obtained from your experimental measurements for the time just before the oscillation starts.
10. Using your simplified circuit diagram and the measured oscillator voltage, determine the current through the resistor, which will also be the current through the inductor because they are in series.
  11. Using the current through the inductor you just determined and the voltage across the capacitor (the oscillator voltage), determine the energy stored in both components at the time just before the oscillation begins. Compare your results. Which component stores most of the energy at this time?