

Published on Instrumentation LAB (http://instrumentationlab.berkeley.edu)

Home > Lab Assignments > Lab 2-Linear Circuits II

Lab 2 - Linear Circuits II

University of California at Berkeley

Donald A. Glaser Physics 111A

Instrumentation Laboratory

Lab 2

Linear Circuits II

© 2015 by the Regents of the University of California. All rights reserved.

References:

Hayes & Horowitz Chapter 1, p. 32–60

Horowitz & Hill Chapters 1.06, 1.12-1.24, 5.01-5.02, 5.04-5.05, Appendix H

Appendix B (skim)

Web resources Wikipedia.org

Physics 111-Lab Library Reference Site

Reprints and other information can be found on the Physics 111 Library Site. [1]

Legends: See the Syllabus Web pages for details on introductory Advice next to Glossary.

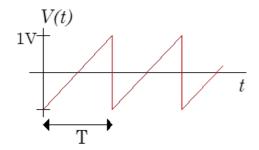
NOTE: You can check out and keep the portable breadboards, VB-106 or VB-108, from the 111-Lab for yourself (Only one each please)

In this week's lab you will continue investigating linear circuits. Fourier analysis, scope probe frequency properties, inductors, resonant circuits, and computer simulations will be studied.

Before coming to class complete this list of tasks:

- · Completely read the Lab Write-up
- · Answer the pre-lab questions utilizing the references and the write-up
- · Perform any circuit calculations or anything that can be done outside of lab.
- Plan out how to perform Lab tasks.

1. Calculate the Fourier coefficients for the ramp waveform shown below:



- 2. How does the scope probe work?
- 3. Approximately what is the Q of the KFOG-FM transmitter?
- 4. The Q in the RLC tank circuit drawn in Sec 2.6 is proportional to R. By shifting the location of the resistor, draw a circuit in which the Q is proportional to 1/R.

Time Dependent Circuits

Circuit analysis is straightforward if all the signals are time independent, i.e. DC. The response of circuit to time dependent (AC) signals like sine waves is more complicated because the response to the signal may not be in phase with the signal, and may depend on frequency.

For example, a circuit driven by a voltage source $V = V_1 \cos(\omega t)$

Since I and V are not necessarily in phase, the resistance can no longer be a pure real quantity. We use a new term for complex resistances: the *impedance* Z. The magnitude of the impedance has much the same function in Ohm's law (now V = ZI), as did the resistance R; it determines the relation between the magnitudes of I and V. The phase angle of Z determines the phase shift between I and V. Note that resistance is redefined to be the real part of the impedance, and the **reactance** is defined to be the imaginary part of the impedance.

Clearly a resistor has a pure real impedance $Z_R=R$ and induces no phase shifts. Capacitors have impedance $Z_C=rac{1}{j\omega C}$ and inductors have impedance $Z_L=j\omega L$.

Capacitor impedance decreases with frequency; inductor impedance increases with frequency. Both capacitors and inductors induce 90° phase shifts, but the phase shifts are in opposite directions.

Any linear circuit [2] can be analyzed using the impedance formulas. The familiar parallel and series resistor addition formulas carry over directly; just substitute the capacitative and inductive impedances for R. For example, the impedance of two capacitors in parallel is

$$Z=rac{Z_{C_1}Z_{C_2}}{Z_{C_1}+Z_{C_2}}=rac{1/j\omega C_11/j\omega C_2}{1/j\omega C_1+1/j\omega C_2}=rac{1}{j\omega (C_1+C_2)}$$

Analyze any circuit just as you would if all the components were resistors, but keep track of the complex parts, and you will get the right answer. Thévenin circuit reduction works as well, though the Thévenin resistance becomes a complex, frequency dependent impedance.

And that's all we need to know about complex impedances for this class. But as physicists we should understand the formal differential equations methods that underlie these simplifications, which can be found in most E&M texts.

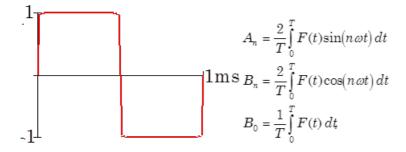
Background

Fourier Analysis and Repetitive Waveforms

In last week's lab, we showed how the concept of complex impedances extends our ability to analyze circuits to those driven by pure sinusoidal waveforms. However, our circuits will be frequently driven by waveforms more complicated than pure sinusoids. Fortunately complex impedance analysis is easily extended to include circuits driven by any *repetitive* waveform by using the fact that any repetitive waveform can be decomposed into a (possibly) infinite series of harmonic sinusoids. The mathematical technique, which performs the decomposition, is called Fourier Analysis – you should have already studied this technique in your mathematics classes. In summary, any waveform F(t) which is repetitive [has a period T defined such that F(t+T) = F(t) for all t] can be synthesized from the infinite sum

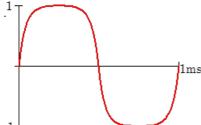
$$F(t) = \sum_{n=0}^{\infty} A_n \sin(n\omega t) + B_n \cos(n\omega t)$$

where the Fourier coefficients are defined as



where $\omega = 2\pi/T$. The lowest order sinusoid (n=1) in the infinite sum is called the fundamental; all the other waveforms are referred to as the nth harmonic.

For example, the square wave shown to the right has Fourier coefficients $A_n = 4/n\pi$ for n odd, with all other coefficients equal to zero.



The beauty of using Fourier Analysis in circuit analysis is that the response of any linear circuit to a repetitive waveform is the sum of the responses of the circuit to each individual harmonic. Thus a circuit with a transfer function H(w), driven by the repetitive waveform F(t), has an output

$$\sum_{n=0}^{\infty} \left[A_n \sin(n\omega_o t) + B_n \cos(n\omega_o t) \right] H(n\omega_o)$$

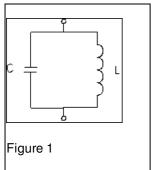
For example, when a 1kHz square wave is passed through a $au=4 imes10^{-5}s$ s low pass filter [transfer function

 $H(\omega) = 1/(1+j\omega\tau)$], the calculation is summarized in the table below and the output waveform is shown in the graph to the right.

Harmonic	Frequency (Hz)	Square Wave Coefficient	Low Pass Filter Transfer Function	Output Coefficients	
n	$\omega_n/2\pi$	A_n	$H(\omega_n)$	$A_n H(\omega_n)$	$ A_n H(\omega_n) $
1	1000	1.273	0.941 – 0.236j	1.198 – 0.301j	1.235
3	3000	0.424	0.638 – 0.481j	0.271 – 0.204j	0.339
5	5000	0.255	0.388 – 0.487j	0.099 – 0.124j	0.159
7	7000	0.182	0.244 – 0.43j	0.044 – 0.078j	0.090
9	9000	0.141	0.163 – 0.37j	0.023 – 0.052j	0.057
11	11000	0.116	0.116 – 0.32j	0.013 – 0.037j	0.039
13	13000	0.098	0.086 – 0.28j	0.008 – 0.027j	0.029
15	15000	0.085	0.066 – 0.248j	0.006 – 0.021j	0.022

Back To Top

Resonant Circuits



A circuit combining an inductor and capacitor in parallel, sometimes called a tank circuit, will oscillate at the frequency $\,\omega=\frac{1}{\sqrt{LC}}$.

Such resonant circuits are frequently used in physics and electronics. They have two primary applications: detecting signals oscillating at a known frequency, and generating signals at a specific frequency. The first application is exemplified by radio tuners which pick out a specific radio station (say at 1MHz) from the enormous selection of available radio stations, and the second is exemplified by the radio station transmitter itself, which must broadcast at a precisely controlled frequency. [3]

Resonant circuits are endlessly analyzed in the course textbooks. Here we will review only a

few points:

- Understand the energy flow: the energy sloshes back and forth (at the resonant frequency) between the capacitor (maximum capacitor voltage, no current) and the inductor (maximum current, no voltage across the capacitor.)
- The oscillation amplitude of a resonant circuit will be much higher when driven with a frequency near resonance than when driven off resonance. Thus, resonators will preferentially respond to near resonant signals, and can be used to detect such signals even when these signals are masked by signals at other frequencies.
- Remember that the impedance of an inductor is proportional to j while the impedance of a capacitor is proportional to 1/j = -j. Both inductors and capacitors induce 90° phase shifts, but in *opposite directions*. Consequently, their phase

shifts are 180° apart from each other. At resonance the magnitudes of the impedances are equal. Add them together and you get zero! Thus the impedance of a series combination of inductors and capacitors is zero, and the impedance of a parallel combination, like that drawn in Fig 1, is infinite.

- Resonating circuits always have some dissipation, either from a deliberately added resistor or from imperfections in the circuit components. [4] The Q of a resonator is a measure of the "quality" of the resonator; the lower the dissipation, the higher the Q. There are several definitions of the Q:
- 2 pi times the number of times the resonator will ring or oscillate after it has been hit with an impulse excitation before the voltage drops to 1/e.
- 4 pi times the energy stored in the oscillator divided by the energy dissipated in one cycle.
- For a driven resonator, the oscillator resonant frequency w divided by the de-tuning frequency Dw. The detuning frequency is the driving frequency shift that results in the resonator power amplitude dropping by a factor of two.
- Approximately proportional to the amount that the oscillation amplitude peaks up when driven at resonance compared to the amplitude when driven far off resonance.
- Depending on the circuit configuration, various combinations of R, L, and C. Be warned...the Q is sometimes proportional to R, and sometimes to 1/R.

These definitions work best for high Q.

• A common misconception is that the highest possible Q is always best. Very high Q's are desirable for sources like transmitters, where very pure frequencies are needed. However, high Q's are not necessarily desirable for receivers like radio tuners, which must detect a frequency band^[5] Dw. In such applications the resonator is designed so that its Q is approximately equal to w/Dw.

Simulations

Numeric computer simulations have revolutionized many fields of physics and electronic design. Today's astonishingly powerful desktop computers have made simulations a common tool of almost every physicist. Simulations are used in every field: in condensed matter physics, astrophysics, plasma physics, even biophysics. And modern integrated circuits are almost entirely designed with the aid of computer tools and simulations. **BUT** beware of the allure of simulations. While simulations have their place for analytically intractable problems, an analytic solution is almost always worth a thousand computer simulations. Use them for problems like the band structure of a complicated crystal or to find the attractors in a chaotic system. Don't use them to solve for the behavior of a simple harmonic oscillator.

That said, in this course you will use computer simulations to study circuits that are, by and large, analytically tractable. Worse yet, all the circuits will be readily constructible on a breadboard. Always trust an experimental result over an analytic result over a simulation. So why are we having you run simulations? Our primary purpose is for you to get a flavor of what computer simulations can do, and how they can be run. In addition, some phenomena like phase shift scaling can be studied more easily with the simulations than in the lab.

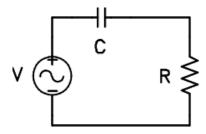


You will be required to simulate a small number of circuits as part of this course. If simulating stimulates you, you can play with many other circuits on your own. The circuits indicated by the computer icon are available in the lab and over the web. You can also design and simulate your own circuits.

The circuit simulator we use is called <u>MultiSim</u> [2], and is written by National Instruments. MultiSim is based on PSpice, a very powerful circuit simulation engine which was developed here at Berkeley. Spice by itself is almost impossible to use; it requires input in an obscure format reminiscent of the computer input cards that went out in the 70s. Many commercial companies have incorporated the Spice engine into commercial products. MultiSim is one such implementation. It has a decent MS Windows front end that mostly hides the native Spice input format, and an adequate back end graphing program that displays the results of your simulations. MultiSim program files can be downloaded from the My Computer **BSC Share** site in the 111-Lab.

MultiSim works by solving the system of differential equations, which corresponds to the entered circuit. It automatically selects the low level differential equations appropriate for each circuit component from a library of models, and then, according to the circuit topology, joins these low level differential equations into one master system of differential equations. Finally it solves this system numerically and graphs the results. For example, for the RC high-pass filter below, the capacitor is modeled by the differential equation $dV_C/dt = I/C$, and the resistor by the trivial differential equation $V_r = RI.^{[7]}$ The computer then automatically constructs and solves the system differential equation, dV/dt = RdI/dt + I/C.

The solution can be graphed as a function of time. Such solutions are called transient analyses. Perhaps more useful is MultiSim's ability to scan the master differential equation over a set of frequencies. The program can then extract some circuit characteristic like the transfer function, and graph it as a function of frequency.



An RC circuit driven by an exponential source turns out to be analytically tractable. By solving the appropriate differential equation, I will prove that the response is exponential, and find the correct multiplicative constant relating the circuit's output to its input. The first step in the set up is to write out what we know from ohm's law and impedance.

$$V_C = V_S - V_R$$
, $V_R = I(t)R$, $V_S = V_0 e^{t/T_S}$, $I(t) = C \frac{d}{dt} (V_S - V_R)$

This yields a Differential Equation of the Form: $rac{dI}{dt}+rac{I(t)}{RC}=rac{CV_0}{T_SRC}\,e^{t/T_S}$

And has the solution
$$I(t)=rac{V_0C}{T_s+RC}\,e^{t/T_s}$$
. Now $V_R=I(t)R=rac{V_0RC}{T_S+RC}\,e^{t/T_s}$ and $V_R=V_S\left[rac{RC}{T_S+RC}
ight]=V_S\left[rac{1}{rac{T_S}{RC}+1}
ight]$.

The point of this discussion is to give a mathematical background and proof for this common statement in electronics: Linear circuits have linear effects. We know this is true because all signals can be represented with exponentials and we just proved that the exponential is still an exponential on the output just off by a linear constant. We can also see that the constant is determined by the period or frequency of the signal and the RC (time constant) of the circuit.

In the Lab



Problem 2.1 - Fourier Analysis



Explore Fourier analysis with the computerized <u>Fourier Synthesizer</u> [3]. [8] Generate square, triangular, and pulse waveforms, and find the effect of small changes to the harmonic amplitudes. Type in and check the coefficients you derived (in the pre-lab questions) for a ramp waveform



Rebuild the high pass filter that you used in last weeks lab (LAB 1 Part 2 problem 1.2.6). Examine the output from the filter when driven by a square wave. Look at frequencies ranging from 20Hz to 20kHz. Explain your results in terms of Fourier decomposition.

Problem 2.3

Note how the real-world RC high-pass filter alters the shape of a square wave. Compare this to how high-pass filtering (by turning off the low frequency components) in the Fourier synthesizer alters the shape of the waveform. Why are these different? Include a discussion of phase shift and causality

Problem 2.4

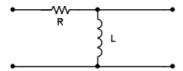
Rebuild your circuit as a low pass filter, and repeat question 2.2.

Inductors

Inductors are complementary to capacitors; almost any circuit that can be built with a capacitor can, at least in principle, be built with an inductor. In practice, inductors are rare because practical inductors are far less ideal than practical capacitors. For example, practical inductors tend to have high internal resistances. The increased use of integrated circuits has furthered the decline of inductors; while chip capacitors are easy to fabricate, inductors are almost impossible to build on a chip. However, inductors are still important in resonators and transformers.



Construct an inductor by winding 25 turns of 22 gauge insulated wire onto a toroid. Make sure that the ends of the wire are long enough to plug the inductor into the breadboard. Use the LCR meter to measure and record the inductance. Then setup the following circuit using a value of R that gives a circuit rolloff frequency of approximately 25kHz.



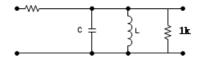
Use the signal generator to scan the input frequency from DC to 1MHz. Look at both sine and square waves. Does the circuit behave like a high-pass filter? What are the circuit's measured and calculated rolloff points? Draw the filter's response to representative square waves.

Create a Bode plot of Vout/Vin versus frequency.

Look at the circuit's response at even higher frequencies (say up to 10MHz). Notice that the circuit rolls off at very high frequencies. What causes this non-ideal behavior?

Problem 2.6 - RLC Circuit Resonator

Using the inductor that you made for Sec. 2.4, construct the following RLC



resonator circuit. Design the circuit to resonate around 7 kHz. What value capacitor should you use? Note that the 22k resistor isolates the function generator from the resonator circuit. Given the generally low impedance of this RLC circuit, the function generator along with 22k resistor acts as a current source. The $R_L = 1$ K sets the Q of the circuit to a value which can be measured reliably with equipment in this lab.

Set the function generator to produce the largest possible amplitude. Determine the output amplitude as a function of frequency from 100Hz to 1MHz. Make sure that you closely cover the resonance.

Create a Bode plot showing both measured and theoretical transfer functions. How well do the two curves agree? What is the experimental Q? Hint: Use the definition of db from Lab 1 (1.2.12) and $Q=f/\Delta f$; where the change in frequency is at the 3 db role-off point. The theoretical Q? Hint: use this equation $Q=\frac{\tau}{T}=R\sqrt{\frac{C}{L}}$

Back To Top

Transformers

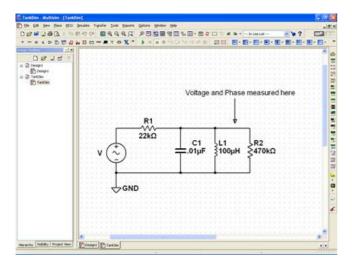
Pacific Gas and Electric provides us with power at ~120V, 60Hz AC. Most electronic circuitry works at far lower voltages: typically ±12V DC for analog circuits and +5V for digital circuits. Transformers are used to lower the wall voltages down to voltages useful for electronics. In the past transformers were also used to change the voltage- current characteristics of signals; however, most such uses are archaic.

Problem 2.7 (Special investigation needed here)

Construct a 5:1 transformer by winding five additional turns of a second wire around your toroid. Drive the 25-turn winding with a high amplitude, 10kHz signal. Isolate the transformer from the generator by inserting a 1k resistor between the generator and the transformer. Set the scope to look at the voltage across both the input (the 25-turn winding) and the output (the five-turn winding) of the transformer. What is the voltage ratio? Does it agree with the predicted value? Now reverse the transformer: drive the five-turn winding and look at the 25-turn winding. What is the voltage ratio between the input and output windings?

MultiSim

The first thing to do is copy all the <u>MultiSim</u> [2] files from the 111-Lab Network drive to your Desktop or My Documents folder. The first circuit that we will study with MultiSim is the resonator circuit that you constructed in Problem 2.6. Run MultiSim, double click on the TankSim.ms11 after MultiSim opens up, the monitor should display something like:



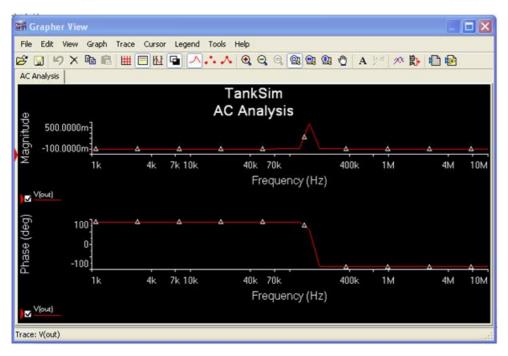
You will have to change the values of the circuit components to match the values you used for the circuit you constructed in

2.6. You can do this by double clicking on the value. The circle with a squiggle is the pictorial

representation of a sine wave source. The simulation is already set up to scan over a frequency range of 1kHz to 10MHz. [9]

Simulate the circuit by pressing Simulate → Analyzes → AC Analysis and then clicking on simulate on the window that appears.

The program shows the resulting graph in a new window that should look like the following window:



Both the transfer function (the voltage at the voltage probe vs. frequency) and the phase are displayed on the graph. Since the y-axis for these two curves are so different, the curves need to be plotted on two different graphs. This is easy to do manually, but has already been set up for you. The upper graph is the transfer function, and the lower is the phase curve. For future reference, understand how the curves are labeled:

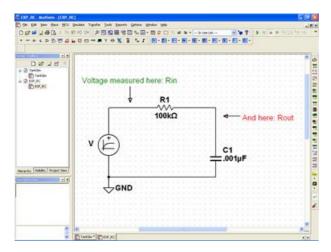
Problem 2.8 (Note: Never save up your signature requests, do them one at a time as they are needed. They are an opportunity for deeper exploration and reinforcement of foundations, and this process does not work well when compressed into one chunk.)



Go back to the schematic. Play with the component values and rerun the simulation. How do you increase the Q? Does the phase shift make sense?

The response of passive circuits to sine waves is easy to analyze using complex impedances. The response to non-sinusoidal signals is more difficult to determine. As was discussed earlier, repetitive signals can be analyzed by breaking down the signal into its frequency components (via Fourier Analysis), but the response to non-repetitive signals requires the direct solution of the system differential equations.

One such non-repetitive signal is an exponential voltage. Go back to MultiSim and use the \underline{O} pen tab item to select the Exp_RC schematic.



The circuit contains an exponential source and two voltage markers: the first records the signal output by the source itself, and the second records the response. Analyze the circuit using <u>Transient analysis</u>. Note that aside from a small transient in the beginning, the response to the exponential input appears to be an exponential with the same time constant.

Problem 2.9



Play with the circuit components. Is the response still exponential? An RC circuit driven by an exponential source turns out to be analytically tractable. By solving the appropriate differential equation, prove that the response is exponential, and find the correct multiplicative constant relating the circuit's output to its input.

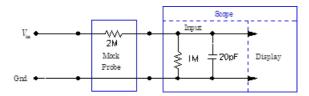
More on probes

As you studied in last week's lab, the scope probe increases the input impedance of the scope by a factor of ten – at the expense of attenuating the signal by a factor of ten. The input impedance of the scope by itself is 1MW in parallel with 20pF or 30pF. (The impedance is written on the face of the scope next to the BNC input jack.) The probe contains a resistor, which, in conjunction with the scope input resistance, forms a voltage divider.

Problem 2.10

Build the mock probe shown below and hook it up to the scope. (We do not stock 2M resistors; synthesize one out of the components we do stock. Obviously you do not have to build the internal scope circuit.

Start with the mock probe disconnected from the scope. Put 24V DC across the mock probe input. With the DMM, measure the voltage at the input and output of the mock probe. Why aren't the two measurements the same?



Connect the mock probe to the scope with a short coax cable and mini-grabbers. (Don't use the real scope probe.) Repeat the measurement of the voltage before and after the mock probe with the DMM. Is this set of measurements the same as the last set? With which DMM measurement does the scope display agree? Explain all these results.

Problem 2

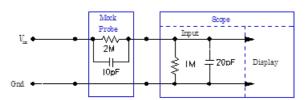
Connect the mock probe to the signal generator, and set it to output sine waves. Scan the signal generator frequency from 10Hz to 1MHz. Plot the response, using the scope to measure the size of the signal, as a function of frequency. What are the circuit's measured and calculated rolloff points? Does the circuit behave like a low-pass filter? Now draw some typical waveforms for square wave inputs.

Hint-The capacitance of the cable connecting the mock probe to the scope is important to the behavior of this circuit. You may wish to measure this capacitance. (Note that the 20pF capacitor is the value for the old generation of scopes. The scopes currently in the lab had an input capacitance of 11.5pF.)

Since a scope is supposed to display an undistorted representation of the input wave, this unintentional low pass filter is not tolerable. Real scope probes are "compensated" to remove the undesired probe low-pass filter.

Problem 2.12

Build the compensated mock probe below.



Scan the frequency range again. Does the compensated mock probe work better, especially for square waves? Now add a small capacitor (say 50pF) in parallel with the 10pF capacitor. Is the compensation better? Tweak the capacitor value until you get the best results. What value works best? Why do you need the extra capacitance? Is it possible to remove all the frequency dependence in the mock probe? How?

Please fill out the

After completing the lab write up but before turning the lab report in, please fill out the **Student Evaluation of the Lab Report** [4].

- [1] To avoid confusion with the symbol for current, we use j instead of i for the $\sqrt{-1}$.
- [2] More specifically, any circuit that consists only of resistors, capacitors, inductors, voltage sources, and current sources.
- [3] For instance, KFOG-FM's carrier frequency is allowed to drift only 2kHz (0.002%) from its assigned frequency of 104.5MHz.
- [4] Typical imperfections include resistance in the inductor windings and dissipation in the capacitor dielectric.
- [5] Radio stations need to transmit information, and information cannot be conveyed by a pure, single frequency wave. (The base, central frequency is called the carrier frequency.) Consequently stations broadcast radio waves in a band surrounding their carrier frequency. For example, the station KFOG-FM's carrier frequency is at 104.5MHz, but it actually broadcasts between about 104.450 and 104.550MHz.
- [6] To be fair to the theorists, remember that this statement was written by an experimentalist. Experimental results are sometimes wrong: witness Cold Fusion.
- [7] An active circuit element, such as a transistor, requires a much more complicated model. Fortunately, tens of thousands of such models are available in the libraries that come with the program.
- [8] The Synthesizer can be downloaded from this page: Fourier Synthesizer [3]
- [9] Use Analysis Setup AC Sweep to adjust the sweep frequency. If you change it, make sure you change it back for the next student.

Source URL: http://instrumentationlab.berkeley.edu/Lab2

Links

- [1] http://physics111.lib.berkeley.edu/Physics111/
- [2] http://instrumentationlab.berkeley.edu/multisim
- [3] http://instrumentationlab.berkeley.edu/labviewprograms
- $\hbox{[4] http://instrumentationlab.berkeley.edu/StudentEvaluation}\\$