

Lab Equipment

EECS 311

Fall 2009

Contents

<i>Lab Equipment Overview</i>	<i>pg. 1</i>
<i>Lab Components</i>	<i>pg. 4</i>
<i>Probe Compensation</i>	<i>pg. 8</i>
<i>Finite Instrumentation Impedance</i>	<i>pg.10</i>
<i>Simulation Tools</i>	<i>pg. 10</i>

1 - Laboratory Equipment

1.1 Proto-boards

A proto-board provides a fast way of hooking up circuits with no soldering. The proto-board consists of an array of interconnected holes. Connections are made by pushing leads into the holes. When using a proto-board, only straight wires of small diameter (a resistor lead or smaller) should be inserted into the holes. In general, when building a proto-board circuit, a good rule is "the fewer the wires, the better." A messy circuit is likely to perform poorly and be a pain to debug.

Some of the labs (the design labs in particular) may require more than one testing session to complete. For these labs, it is advantageous to be able to keep your circuits constructed between testing sessions. For this purpose, you may check out a proto-board from your lab GSI. Note that this proto-board must be returned prior to the final. If it is not returned before the final, you will receive a grade of "incomplete" for the course.

1.2 Oscilloscopes

General

An oscilloscope is the most useful piece of test equipment in the laboratory. Basically, it is a device which graphically displays time-varying voltage signals. In particular, it is most often used to display voltages as a function of time.

Each of the stations in the EECS 311 laboratory is equipped with an oscilloscope. You should already be familiar with the use of oscilloscopes from previous required courses (e.g., EECS 215). To best familiarize yourself with the oscilloscope used in the EECS 311 laboratory, browse through the "User's Guide" located in the lab.

Some Helpful Pointers.

The following usage pointers are helpful in insuring the most accurate circuit measurements in the EECS 311 laboratory:

- (i) Use an oscilloscope probe whenever possible. The probe may or may not have a built in attenuator, and the oscilloscope has a setting that will scale the measured voltage accordingly. This feature is under "Utilities".
- (ii) Make sure each probe is grounded.
- (iii) Compensate the oscilloscope probe frequently (see page 8).
- (iv) Always know the location of the ground reference located on the display.
- (v) Use an external trigger whenever possible.

1.3 Digital Multimeters (DMM's)

Voltage Measurements.

A DMM can be used to make measurements of both AC and DC voltages. The measurement procedure involves setting the DMM to the appropriate scale, selecting AC or DC volts, and connecting the probe between the two points to be measured. If set to the wrong scale, the display will usually flash or give some other error indication.

Some points to keep in mind:

- (i) The EECS 311 DMM's will usually not load the circuit excessively, because they have an input impedance of several $M\Omega$'s.
- (ii) Voltage measurements are made in parallel.
- (iii) When reading AC volts, most meters will read true RMS only if the AC voltage is a sine wave. Some DMM's, usually marked "True RMS", will read the RMS voltage of arbitrary waveforms.
- (iv) When reading AC volts, the DMM has a fairly low upper frequency limit, usually a couple of kHz or less. Consult the instruction manual for further details.

Resistance Measurements

To measure resistance, set the DMM to ohms and select the appropriate scale. Connect the probes across the resistance to be measured. Some fine points:

- (i) When taking data using the high resistance scale, make sure not to handle the probe tips. Handling will add your skin resistance in parallel with the resistance to be measured, which will cause the DMM to read a value lower than the actual resistance.
- (ii) Only certain scales will supply enough current to forward bias a diode. These are marked with a diode symbol.
- (iii) Resistance measurements should be made *outside* of the circuit whenever possible. Any in-circuit measurements made should be considered suspect.


Current Measurements

When measuring current, it is necessary to set the DMM to mA and to select the appropriate scale. Note that the probes are connected to a different set of jacks for current measurements. One must always remember to make current measurements in **SERIES**, not **parallel**. Connecting the meter across a voltage is likely to blow a fuse in the meter. Both AC and DC current can be measured. The same limitations as those on AC voltage measurements apply here.

1.4 *Function Generators*

Function generators are so named because they produce various voltage waveforms. The most common are square, sine, and triangle. The function generators in the EECS 311 lab have various triggering and gating functions. To adjust the peak-to-peak voltage of the output signal, **AMPLITUDE** controls are provided. The DC level of the signal can also be adjusted with the **OFFSET** control. To adjust the frequency, use the **FREQUENCY** button.

There are three ways to adjust the output of the function generator. First, press the button of the parameter you would like to change. To change the value (ie frequency, amplitude, or offset) either:

- (ii) Turn the knob  clockwise to increase the value or counterclockwise to decrease the value.
- (ii) Use the arrow buttons on the function generator:
 - () up arrow = increase value
 - () down arrow = decrease value
 - () left and right arrows, select digit to change

- (iii) Enter the desired value directly using the **Enter Number** button. To do this, press **Enter Number**. Then enter the value you want using the numeric keypad. When you are done, press **Enter**.

Of particular interest is the external triggering output. Whenever possible, this signal should be directly connected to the external triggering input of the oscilloscope.

1.5 **Capacitance Meters**

In EECS 311, we will only be concerned with simple capacitance meters. To use a capacitance meter:

- (i) Depress the measurement button without anything connected and use the **OFFSET** or **ZERO** control to zero the display.
- (ii) Insert the device to be tested into the slots on the front of the capacitance meter. Keep any leads as short as possible.
- (iii) Depress the measurement button.

1.6 **Electronic Components**

Resistors

The resistors used in the EECS 311 lab are all 5%, 1/4 Watt carbon film resistors. "5%" refers to the resistance tolerance at room temperature (i.e., how far off from the stated value it may be), "1/4 Watt" refers to the maximum safe power dissipation, and "carbon film" refers to the resistor construction.

The resistance value is indicated by color coded bands. Learning to read the color code will save you both grief and time in the lab (since you will not have to use the DMM to measure every time you need to know an approximate resistor value). The colors represent digits as indicated in Table I.

TABLE I. Resistor Color Coding

COLOR	Value
Black	0
Brown	1
Red	2
Orange	3
Yellow	4
Green	5
Blue	6
Violet	7
Gray	8
White	9
	Tolerance
Silver	10%
Gold	5%

The actual value of the bands depends on their relative position. The right most band indicates tolerance. To read the color code, hold the resistor so the gold or silver tolerance band is on the right. (20% resistors do not have a gold or silver tolerance color band, however, they have a space where the band would be). Now, reading from left to right, the first two color bands are read directly as numerals. The third band (sometimes called the multiplier band) indicates the number of zeros to be concatenated on the first two digits.

For example, if you read BROWN-GREEN-ORANGE-GOLD, the resistor would be of 5% tolerance and would have the value 15000, or 15 k Ω . In other words, "1", "5", and 3 zeros (for orange) following. BROWN-BLACK-YELLOW-SILVER would be a 10% 100000, or 100 k Ω resistor. In other words, "1", "0", and 4 zeros (for yellow) following.

Capacitors

The marking for capacitors is not nearly as standardized as that for resistors. The most common method is to mark the capacitor in picoFarads ($\text{pF} = 10^{-12}$ Farads) using the same format as resistors, but with numerals instead of colors. For example, 101 indicates 100 pF ("1", "0", and 1 zero following), 100 would be 10 pF. Ask your GSI for help in identifying capacitors.

We will use two basic types of capacitors in the EECS 311 lab: ceramic and electrolytic. Ceramic caps are relatively temperature stable and have the advantage of being non-polarized; that is, they can be charged to either polarity. Electrolytic capacitors, on the other hand, can only be subjected to a specific DC voltage polarity. The polarity is marked on the

outside of the capacitor. DO NOT hook up an electrolytic capacitor backwards, as this will almost certainly damage the capacitor. (It could possibly explode.) Electrolytic capacitors have the advantage of possessing a high capacitance per unit volume; i.e., the largest valued capacitors will often be electrolytic.

Semiconductor Diodes and Transistors.

The diodes used in the EECS 311 lab are general purpose signal diodes and are marked as shown in Fig. 1.

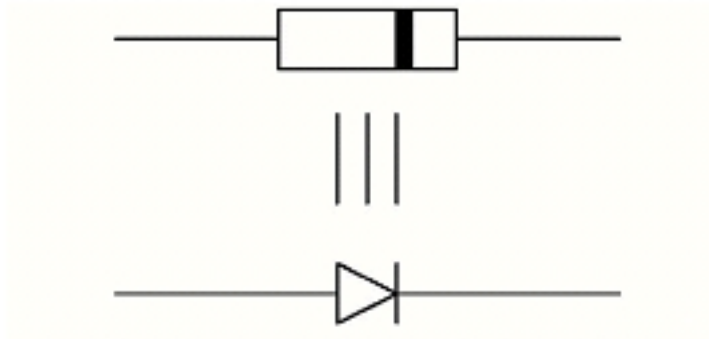


Figure 1: Correspondence between component markings and diode polarity.

Transistors do not have standard markings or pinouts. Ask your GSI for help in identifying transistor pinouts.

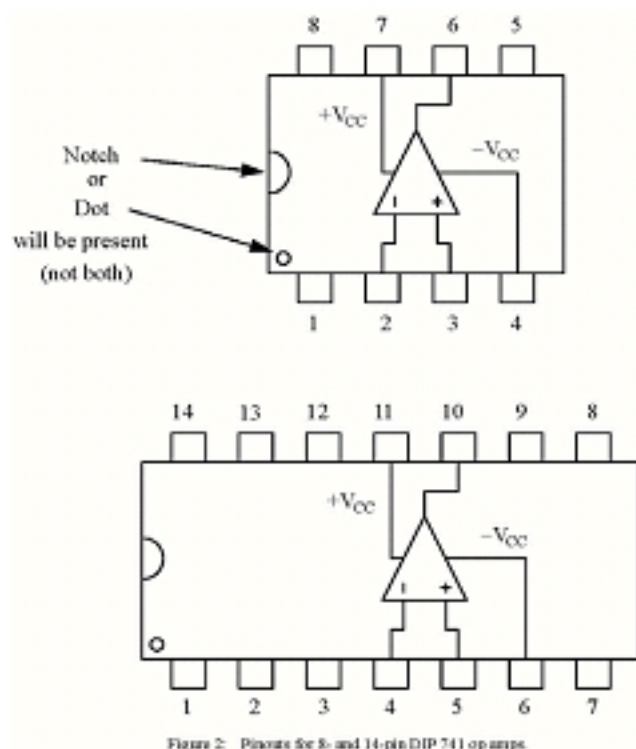


Figure 2: Pinouts for 8- and 14-pin DIP 741 op amps.

Integrated Circuits.

The only integrated circuit used in the EECS 311 lab is the 741 op amp. Different manufacturers make 741's with different prefixes. To name a few, uA741 indicates that the chip was manufactured by Fairchild Semiconductor, NE741 indicates Signetics, and LM741 indicates National Semiconductor. Despite different manufacturers, all of these chips are functionally identical.

The 741 comes in an 8- or 14-pin DIP (Dual In-line Package). The pinout for a 741 is given in Fig. 2. The pin numbers in any DIP package can be determined by first looking down on the IC such that the notch or dot is facing left. Pin 1 is now in the lower left hand corner. The pin numbers increase as you count counterclockwise around the IC starting at pin 1. This is illustrated in the example sketches of the 741 in Fig. 2.

More information on the actual 741 op amps to be used in the lab, as well as all of the other electronic components used, can be found in the data sheets presented in Appendix A. Make sure you consult the data sheet whenever you are unsure of the pinout for a particular device.

2 – Probe Compensation

Whenever measurements of a physical system are made, the act of measurement alters the system of interest, thus, altering the value of the desired parameter. For the case of the laboratory measurements to be made in EECS 311, all of the measurement equipment exhibits finite input/output impedance, which can load the circuits being measured, changing their properties and altering the actual values being measured. Resistive loading of amplifiers is a topic for EECS 311 lecture. As will be seen in this lab, capacitors can also significantly load circuits (as can inductors).

As an example of loading by measurement, consider the use of an oscilloscope to measure the output of a low-pass filter consisting of R_1 and C_1 . The required experimental setup for this measurement is depicted in Fig. PL1.2(a) below.

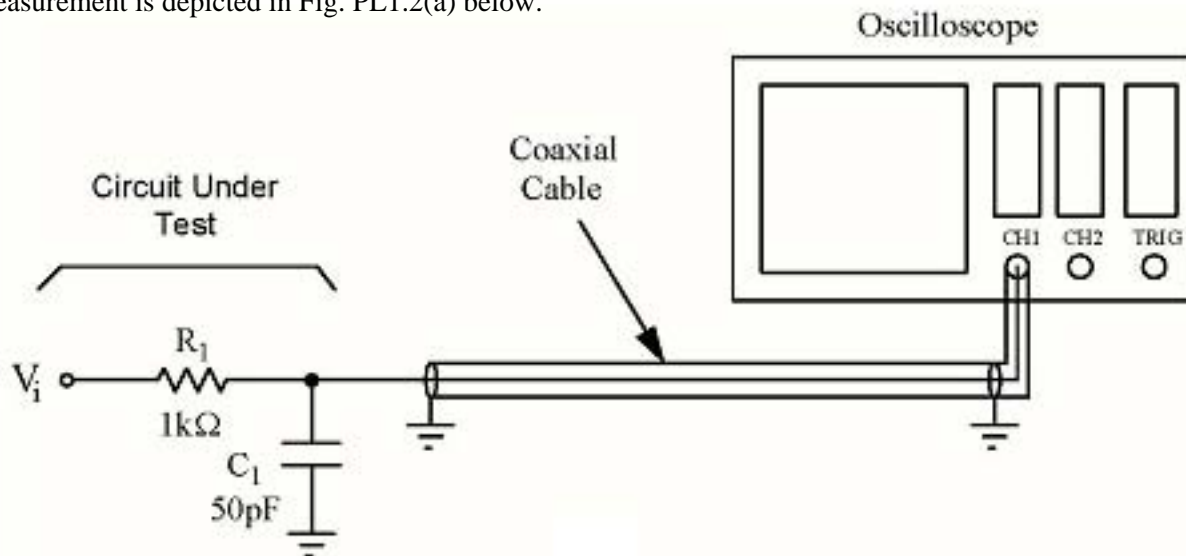


Figure PL1.1

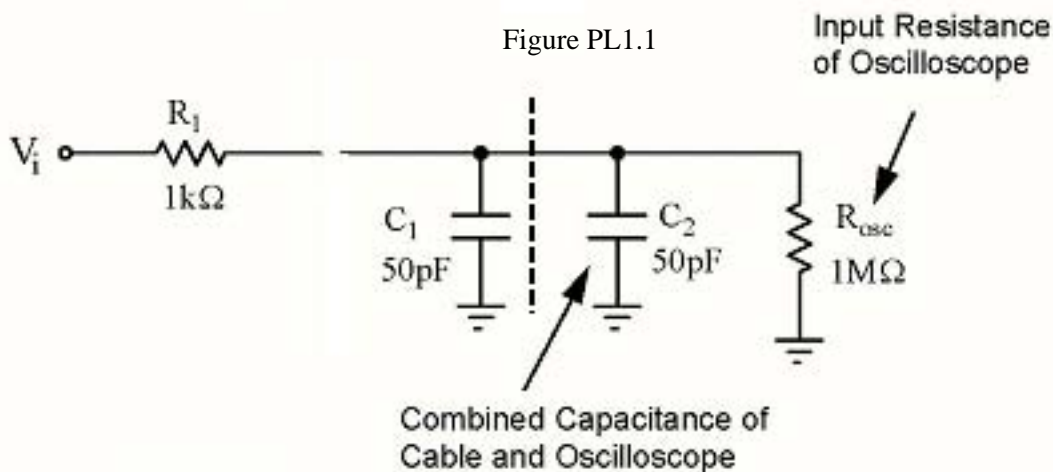
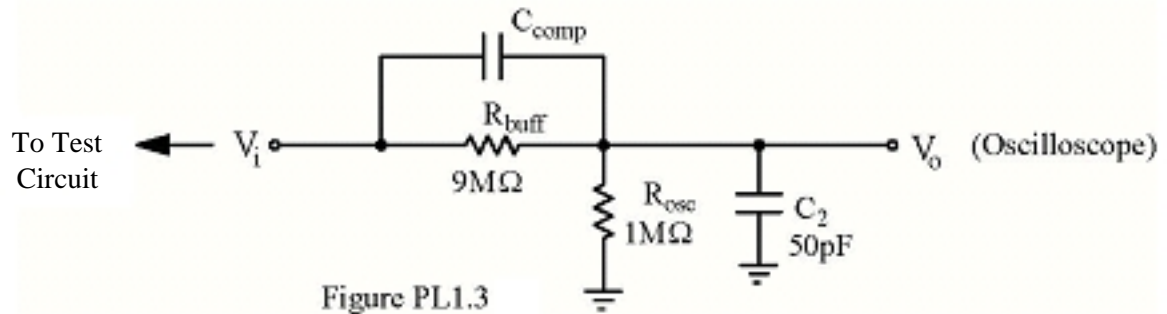


Figure PL1.2

Figure PL1.2(b) is an equivalent circuit schematic for the experimental set-up of Fig. PL1.2(a). The combined capacitance of the oscilloscope and the coaxial cable is represented by $C_2 = 50$ pF, and the input resistance of the oscilloscope is modeled by $R_{osc} = 1$ M Ω .

To minimize the effects of loading, oscilloscope probes are often built as *compensated attenuators*. The equivalent circuit schematic for a typical compensated attenuator is shown in Fig. PL1.3. Note that V_i in Fig. PL1.3 is not the V_i in Fig. PL1.1 and Fig. PL1.2. The voltage V_i in Fig. PL1.3 is the voltage across C_1 in Fig. PL1.1 and Fig. PL1.2.



- (a) Derive an expression for the rise time of the compensated attenuator for the case when $C_{comp} = 0$.
- (b) Derive an expression for C_{comp} based on the values of R_{buff} , R_{osc} , and C_2 to yield the following transfer function:

$$\frac{V_o(s)}{V_i(s)} = \frac{R_{osc}}{R_{buff} + R_{osc}}$$

Note that due to the addition of C_{comp} , the rise time of the circuit can theoretically be made zero.

Compensating the Oscilloscope Probe

The first step in any lab is to organize the required equipment and to COMPENSATE THE OSCILLOSCOPE PROBE. The probe can be easily compensated in the following way:

- (i) Connect the probe to the test point located on the front of the HP oscilloscope. The test point provides a square wave of known properties. The scope must be DC coupled for this operation.
- (ii) Set the triggering to normal and adjust the trigger level for a stable display.
- (iii) Adjust the sweep time and vertical sensitivity to display about two cycles of the square wave.
- (iv) With an adjustment tool, gently adjust the compensation control on the probe while observing the image displayed on the screen. Continue to adjust the probe until the square wave has "as flat a top as possible", as indicated in the far right of Fig. L1.1.

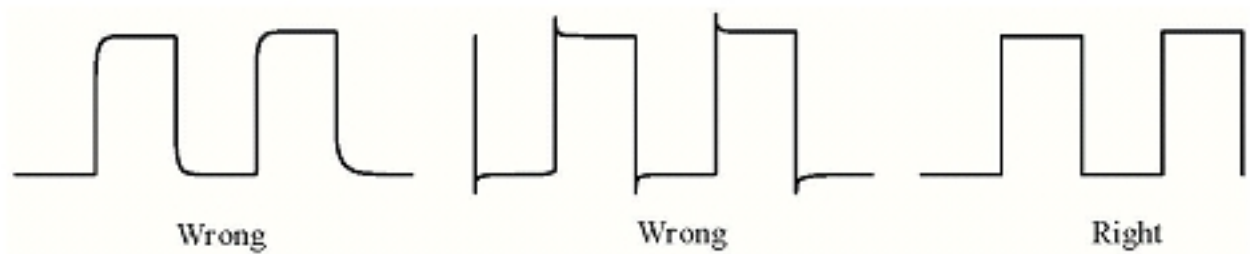


Figure L1.1

3 - Finite Instrumentation Impedance

Depending on the user defined load impedance, the function generator will scale the output voltage it generates to achieve the desired V_{out} . For instance, if the load is defined to be 50 ohms and the generator is set to output 1V, then it will in reality output 2V assuming that R_{gen} is also 50 ohms. However, if the load is defined to be in high-Z, then V_{gen} will only be 1V.

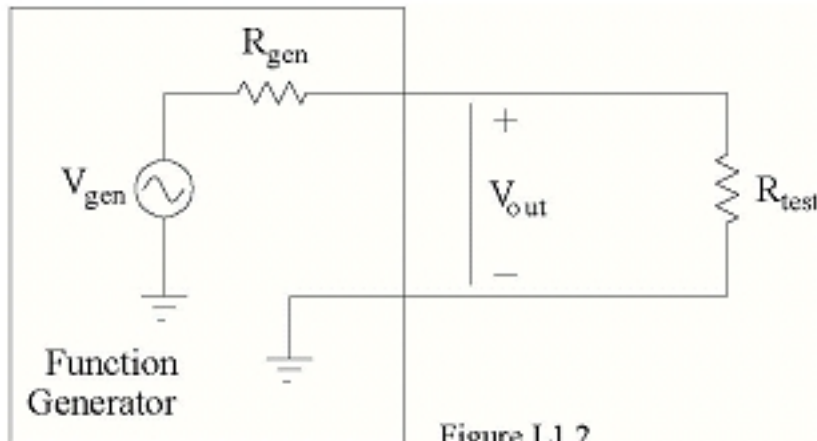


Figure L1.2

4 - Simulation Tools

As mentioned in lecture, EECS 311 will utilize the Cadence software package for schematic entry and simulation of circuits. Virtuoso is the package for circuit schematic entry, while Spectre is the package for circuit simulation. Feel free to ask your GSI for help in using the Cadence tools. Within this Laboratory Manual there are many references to the word SPICE. SPICE is the first extensive circuit simulator created in the 70's. Today, virtually all circuit simulator programs, including Spectre, use SPICE within their kernels. Thus, the device models used by Spectre and other circuit simulation tools are basically SPICE models, as they conform to the strict SPICE format. You will be provided with a tutorial covering basic use of the Cadence software.