
Lab 0: Introduction to Lab Equipment and Components

Introduction

The first part of this lab introduces you to the lab equipment and components you will use for labs through some simple exercises. Proper use of the equipment and components is essential for successful completion of the labs. The following is a list of the instruments you will use in the labs.

Signal generator Generation of various voltage signals of the specified frequency and amplitude

DC power supply Generation of a constant voltage to power your circuit

Multimeter DC voltage, DC current, and resistance measurement

Oscilloscope Measurement of voltage versus time

You will use the following components to build various circuits.

MOSFET Two kinds of NMOS and PMOS transistors

Resistor Fundamental component

Potentiometer Used as a variable resistor and voltage divider

Capacitor Used to block DC voltage and also to filter out high frequency signals and noises

Among the components above, the potentiometer may be new to many of you. The potentiometer is a three-terminal device with two terminals connected to each side of a fixed resistor and the last terminal mechanically swept along the resistive material of the fixed resistor by the knob as shown in Figure 1. A potentiometer can be used to realize a variable resistor as well as a variable voltage divider as shown in Figure 2. Both configurations are used in this course.

The second part of this lab introduces you to LTSpice, a freeware computer software implementing a SPICE simulator of electronic circuits, produced by semiconductor manufacturer Linear Technology. Circuit simulation is an essential part of analog design and you will use LTSpice to verify your designs before implementing them during the lab sessions.



Figure 1: Potentiometer symbol.

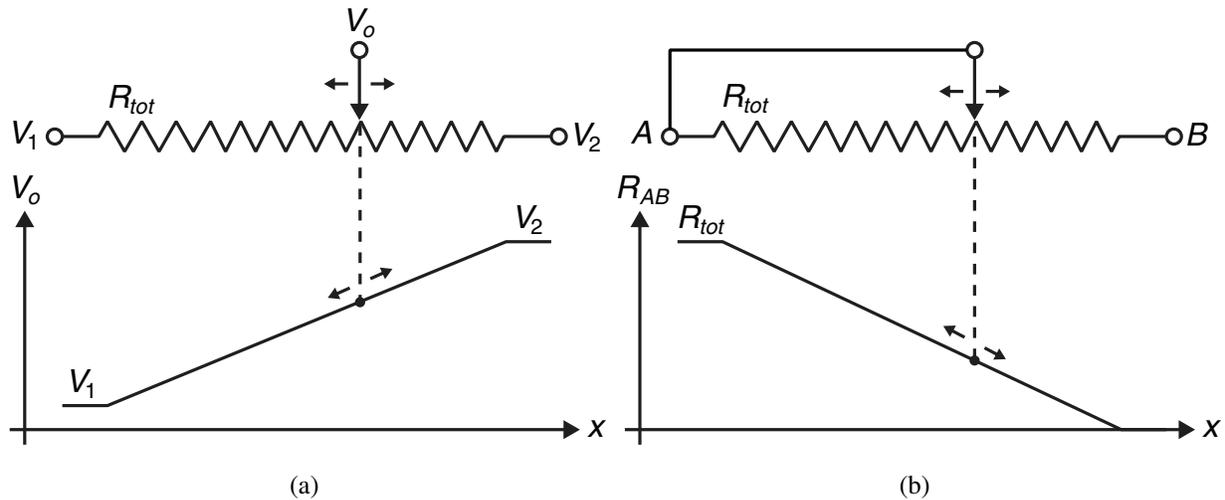


Figure 2: A potentiometer as a (a) variable voltage divider and (b) variable resistor.

Preparation

1. Grouping and breadboard

Form a group of two. A breadboard and components for the labs will be provided in the lab. You only need one breadboard per group.

2. Transistor datasheet

Obtain the datasheets for ALD1101 (NMOS chip), AL1102 (PMOS chip), IRF510 (NMOS power transistor), and IRF9510 (PMOS power transistor). Print the pin assignment diagram of each device, and paste it on your lab book as you will need it in each lab (except this lab).

3. Reference Book

Download "*CMOS Integrated Circuit Simulation with LTspice IV*" by "*Erik Bruun*" from <https://bookboon.com/en/cmos-integrated-circuit-simulation-with-ltspice-ebook>. It is useful material to get familiar with LTSpice containing lots of examples. For lab preparations you may be referred to certain sections of the book. The textbook will be referred to as the "LTSpice textbook".

4. Installation

Follow the section "Getting started" of the LTSpice textbook for LTSpice installation. Also some differences that may result in confusion for MAC users are provided there. In addition,

Table 1: Minimum parts list

Part	Description	Quantity
-	0.1- μ F ceramic capacitor	4
-	10- μ F electrolytic capacitor	5
-	10-k Ω multi-turn potentiometer	1
-	51- Ω resistor	1

<https://www.eecg.utoronto.ca/~johns/LTSpice/LTSpice.html>
has more info about LTSpice.

5. Start with Basic Simulations

Read Example 1.1 of textbook. In this section you will learn how to:

- Draw circuits using schematic editor
- Specify basic components
- Run simulations of operating point and dc sweep
- Run simulations with parameter sweeps
- Plot simulation results using waveform viewer

6. Time and Frequency Domain Simulations

Read Example 2.1 textbook. In this section you will learn how to:

- Perform a transient simulation in LTSpice
- Perform a small-signal (referred to as an AC) simulation in LTSpice

Lab - Part I: Lab Equipments and Components

A minimum parts list for this lab is shown in Table 1. This is the absolute minimum. You may bring more parts for your convenience and backup.

1. Breadboard preparation

Remove every component and wire from the breadboard. Insert power decoupling capacitors as shown in Figure 3 for all of the power rails. Although they may seem doing nothing, the decoupling capacitors are very important components. The output impedance of a power supply is nonzero, and the breadboard and wires add extra impedance on the power line. The decoupling capacitors are there to reduce the impedance. Note the polarity of the electrolytic capacitors.

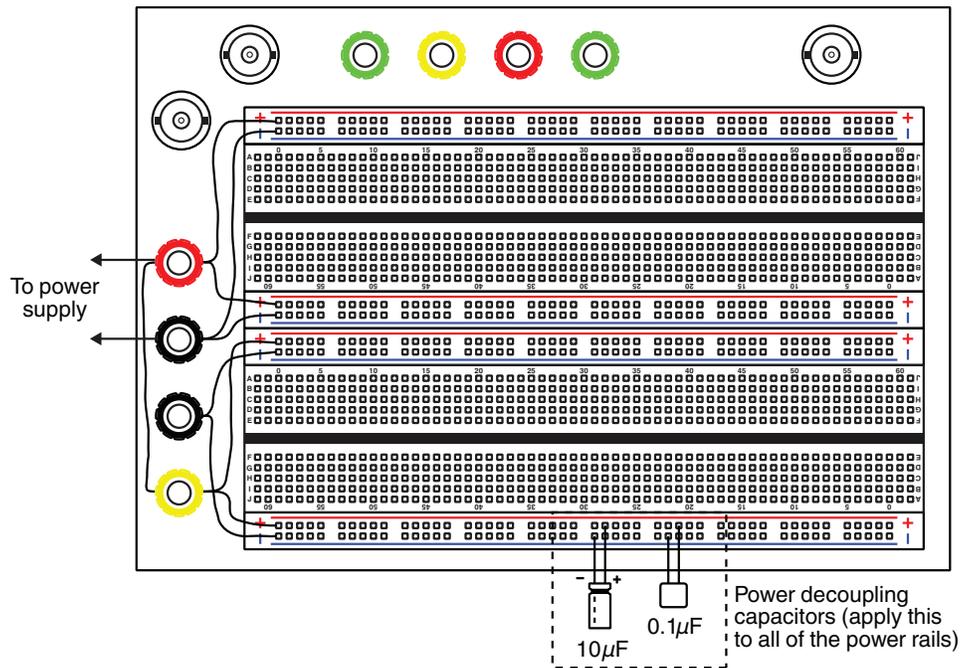


Figure 3: Recommended breadboard setup.

2. Power supply voltage measurement

Turn on the power supply without connecting it to the breadboard. Set the output voltage to 5 V. Turn off the power supply, connect it to the breadboard, and then turn on the power supply again. Measure the power supply voltage on the breadboard with a multimeter. Do this at several points on the breadboard to make sure that each entire power rail is powered.

3. Resistance measurement and current measurement

Configure the multimeter for resistance measurement, and probe the two terminals of a 10-k Ω resistor. Make sure you read resistance close to 10-k Ω .

Disconnect the multimeter, and configure it for current measurement. **Remember that the multimeter uses a different port for current measurement, and excessive current burns the protection diode inside multimeter. For example, probing the power supply directly in the current measurement mode would immediately burn the fuse.** Place the resistor and wire properly for measuring the current through the resistor with 5 V applied across it as shown in Figure 4(a). Then insert a potentiometer as a variable resistor as shown in Figure 4(b) and observe that the current changes as you turn the knob. Be sure to place the multimeter **series** with the component under measurement for current measurement.

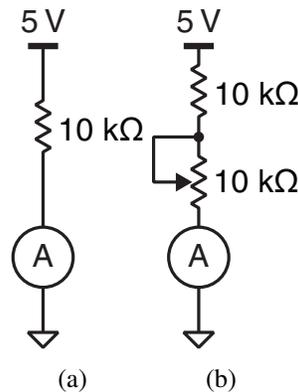


Figure 4: Setups for current measurement exercise.

4. Signal generator and oscilloscope

Setup the signal generator for 1-kHz 1-V_{pp} output with zero offset. Connect the signal generator to the breadboard with a BNC cable. Terminate the signal from the signal generator with a 50-Ω resistor by connecting one side of the resistor to the signal and the other to the ground.

Make sure that the oscilloscope probe is set to the 10X mode if it has a switch on it. Turn on the oscilloscope and open the vertical menu to enable the 60-MHz lowpass filter for both of the input channels and set 10X (not 1X) as the probe setting. The front panel of the oscilloscope is shown in Figure 5. This step is needed at the beginning of each lab. Connect the channel 1 probe to the 50-Ω resistor to measure the voltage across it. The alligator clip of the probe should be always connected to the metal body of the breadboard which is grounded. Open the trigger menu, and select channel 1 as the trigger source. Adjust the trigger level so that the level bar is within the swing of the signal under measurement as shown in Figure 6(a). Adjust the horizontal scale to display 4 periods of the sinusoid. Adjust the vertical scale to maximize the vertical size of the waveform on the screen. Try moving the trigger level away from the signal to get the oscilloscope untriggered as shown in Figure 6(b).

TRY TO AVOID USING THE AUTOSET! It will not work in many cases especially when the signal level is very low (like the next step), and it clears all of the oscilloscope settings (for example you need to enable the lowpass filter again). Plus, you would not learn how to use an oscilloscope properly if you keep relying it. There are many oscilloscopes that do not even have an autose.

5. Oscilloscope with external trigger and averaging

Reduce the output amplitude of the signal generator to 10-mV_{pp}, adjust the vertical scale to maximize the vertical size of the waveform, and observe that the oscilloscope has lost the trigger. With a small input signal, the oscilloscope cannot trigger properly. In such a case, external triggering

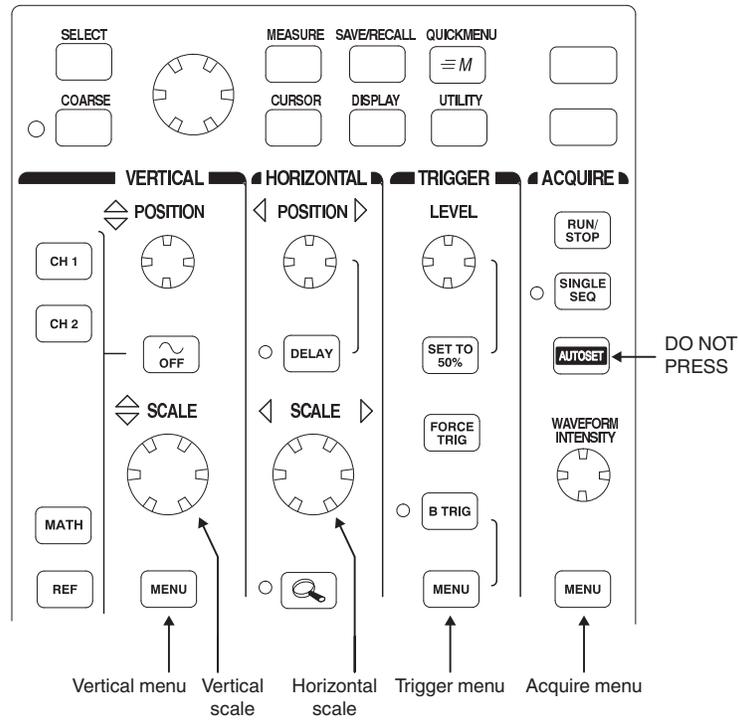


Figure 5: Oscilloscope front panel.

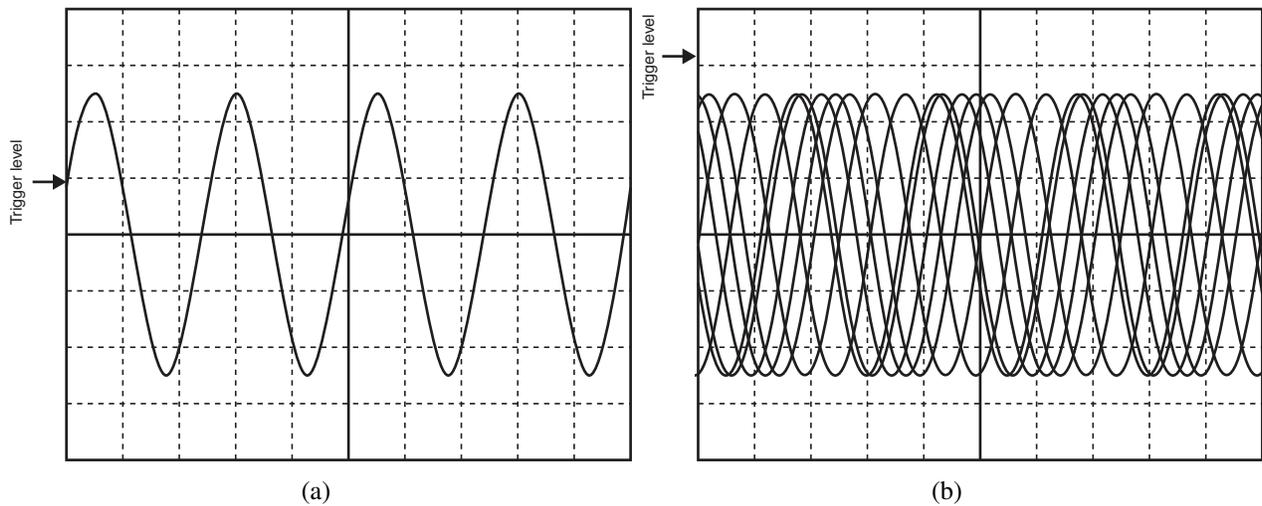


Figure 6: (a) Triggered and (b) untriggered oscilloscope screen.

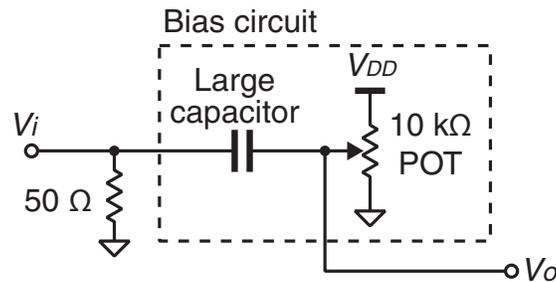


Figure 7: Bias circuit.

can be used to trigger to a large reliable signal in sync with the small signal under measurement. Such a signal is available from the sync output of the signal generator.

Connect the sync output of the signal generator to the external trigger input of the oscilloscope. Then, configure the oscilloscope to use the external trigger as the trigger source in the trigger menu. Adjust the trigger level. Once the oscilloscope is triggered, apply averaging to filter out the noise on the measured signal through the acquire menu. **Remember that averaging works only if the oscilloscope is well triggered.** Otherwise, you would get erroneous results.

6. Signal biasing with a potentiometer

Apply the signal bias circuit in Figure 7 to step 4., adjust the potentiometer while measuring the output DC voltage with a multimeter to set the bias voltage to 2 V. Then use the oscilloscope to confirm that you have a 1-V_{pp} 1-kHz sinusoid biased at 2 V.

Lab - Part II: LTSpice Simulation

1. Lowpass filter frequency and time domain responses

With LTSpice, build a first-order RC lowpass filter with a resistor size of 1kΩ and an f_{3dB} of 159kHz (recall $f_{3dB} = 1/(2\pi RC)$). Drive your filter with an ideal voltage source and look at the output voltage.

- Plot the frequency response.
- Plot the step response (let the step have a 1ns rise time).

2. Single transistor DC sweep simulations

With LTSpice, use the models given on the lab webpage, and build a single transistor circuit with an ALD1101 NMOS transistor with an ideal voltage source V_G driving the gate and an ideal voltage

source V_D driving the drain and the source is grounded (do not forget to bias the bulk with the correct voltage). Let both V_G and V_D voltage sources have a default DC voltage of 1V.

- a) Plot the drain current vs. drain voltage when sweeping V_D from 0 to 3V and leaving V_G fixed at 1V.
- b) Plot the drain current vs. gate voltage when sweeping V_G from 0 to 3V and leaving V_D fixed at 1V.