Chapter 1

Introduction to Laboratory Tools
CHAPTER 1. INTRODUCTION TO LABORATORY TOOLS

1.1 Overview

The purpose of this lab is to learn some of the fundamentals about your oscilloscope that can be used throughout your engineering career. Here is a list of the different tasks to be completed during the lab.

- View a single waveform and take measurements using an Oscilloscope.
- View multiple waveforms and take measurements.
- Explore the concept of triggering.
- Use the oscilloscope's measurement functions.

1.2 Pre-lab

In future labs, you will be required to complete a pre-lab for each lab. This pre-lab will cover concepts that will make the lab much easier to complete. The pre-labs are graded and are due at the beginning of your lab section each time a new lab is started. If you do not have your pre-lab finished at the beginning of the lab period, you will receive no credit for the pre-lab.

1.3 Preparation

A lab notebook is suggested so you can keep all your information together in one place. This information will be useful for writing the lab reports and is an industry practice often required at various companies.

Suggestions of what to bring to lab each week:

- Your ENGR 202 kit of parts. (Obtained from TekBots store)
- Mini Grabber to banana plug. (See Figure 1.1a, Obtained from TekBots store).
- BNC cables with Mini Grabber. (See Figure 1.1b, Obtained at TekBots store).
- Oscilloscope Probe (See Figure 1.1c, Obtained at TekBots store).
- Your lab notebook.

Figure 1.1: Tools for Lab
1.4 Procedure

1.4.1 Formation of Teams

Each team will have two group members. If the lab has an odd number of people, there will be one group of three for that lab. You will be spending your entire term in these groups, so be sure that you trade information about how to contact each other. As a member of a team, everyone needs to contribute towards the team’s success. Please review the questions below and decide how you will divide your responsibilities.

- How will you divide the task of writing the final turned in report?
- How will you divide the task of conducting the lab?
- How will you decide when something is “finished well enough” to turn in?
- How will you handle problems when they show up?

1.5 Using the Oscilloscope

This objective outlines the following: how to set-up the oscilloscope (oscope), information about the MEASURE function of the oscilloscope, and finally, how to trigger the oscilloscope.

Oscilloscope Basics

Although the oscilloscope may appear to be overwhelming because of its many buttons and knobs, there are only a few things needing to be understood in order to successfully view a waveform. An oscilloscope displays voltage signals versus time. The time is the X-axis and the voltage is the Y axis. The oscilloscope can be adjusted to different ranges of each axis as well as the ‘zoom’ on each. This is very similar to what might be done on a graphing calculator in a mathematics course. This can be seen in Figure 1.2.

![Figure 1.2: Voltage vs. Time Graph for Oscilloscope](image)

To adjust the ‘zoom’ on the oscilloscope so the signals displayed can be better observed, the VOLTS/DIV and SEC/DIV knobs are used. Some scopes will have multiple VOLTS/DIV knobs. These scopes have multiple channels that can be used to take measurements. When each channel is turned on, it will be displayed on the screen. This is useful when comparing different voltage signals to each other. Each knob will adjust each channel independently. As the names suggest they determine the number of volts and seconds that will be displayed per division, i.e. it adjusts the

![Figure 1.3: Example Waveform as Seen on Oscilloscope](image)
CHAPTER 1. INTRODUCTION TO LABORATORY TOOLS

scale of the Y axis. Even if an oscilloscope have multiple voltage channels, the time axis will always be the same for every channel, hence there is only one knob for SEC/DIV.

In figure 1.3 a series of grid lines in both the horizontal and vertical directions can be seen. Each one of the grid lines makes a 'division' on the screen. For example this waveform has 6 vertical 'divisions' from its top peak to its bottom peak. Comparing either the two of the lower peaks or two of the upper peaks, it can be seen that there are approximately 5 horizontal 'divisions' between them.

If the 'volts per division' (VOLTS/DIV) was set at 1 VOLTS/DIV then this waveform would be 6V peak-to-peak. If the 'seconds per division' (SEC/DIV) was set at 1\(\mu\)s on the scope then this waveform has a period of about 5\(\mu\)S or a frequency of 200 kHz.

1. If figure 1.3 has a VOLTS/DIV of .5V, what would the Peak to Peak voltage be?

\[V\] (1)

2. If the SEC/DIV was set to 250\(\mu\)s, what would the Frequency be? Assume the period is 5 divisions. A useful relationship is \(T = 1/f\), where \(T\) is the period in seconds and \(f\) is the frequency in Hz.

\[Hz\] (2)

Setting up the Oscilloscope

In order to set-up and use the oscilloscope, follow these steps:

1. Find the oscilloscope on your lab bench. Using your oscilloscope probe, connect the BNC end to 'Channel 1'. See Figure 1.4.

2. Connect the probe onto the PROBE COMP, next to CH 1 on the oscilloscope. The black grabber (Ground) should be connected to the lower of the two connections. The probe tip should be clipped to the upper connection. See Figure 1.5.

   It is very important to note that all of the ground cables on all of the oscilloscope channels are connected together inside of the oscilloscope. Use caution to always connect all of the ground cables to the same circuit node.

3. Press the CH 1 MENU button. This changes the display so that options for adjusting the properties of Channel 1 are displayed. If you press the button again, Channel 1 will be removed from the display. Press one more time to have it reappear. The options for Channel 1 are displayed along the right hand side of the screen. These are adjusted by using the unmarked buttons next to each option. See Figure 1.6.

   - Set the coupling to AC. This center's your waveform on the reference line. Remember when the channel is AC coupled, it will not display DC voltages, since the DC component of a signal is not displayed.
   - Set the probe to 10\(x\). Most general-use oscilloscope probes are 10\(x\), while very sensitive probes maybe 100\(x\) or even 1000\(x\).

4. As previously mentioned, an oscilloscope measures voltage with respect to time. The X-axis is therefore the time axis and the Y-axis is the voltage axis. However, just like graph paper, the oscilloscope can be set to whatever scale you want. In the lower left corner of the display, CH1 X.XX V is displayed. This is the VOLTS/DIV of Channel 1. Similarly, values for CH2 and the time axis are displayed if that channel is turned on.

5. To view the 'PROBE COMP' signal, the oscilloscope must be set to the correct scales. The PROBE COMP wave outputs a 5V peak-to-peak square wave. Set the CH1 voltage scale using the VOLTS/DIV knob for CH1 so that you can see the entire waveform.

6. The frequency of the PROBE COMP waveform is 1000 Hz. Using the SEC/DIV knob, adjust the time scale so that about five periods of the wave are displayed on the screen. Remember that the period is the inverse of the frequency.
1.5. USING THE OSCILLOSCOPE

1.5.1 Oscilloscope Built-in Tools

The CURSOR Function

The cursor can measure voltage and time differences. By using the buttons on the right hand side of the screen either voltage or time measurements can be selected. Once the type and source of the measurement have been selected use the position knobs to place the cursors where desired. Using the probe comp square wave as seen in figure 1.6 calculate the following using the cursor function. To make the task easier, press the 'Run/Stop' button to freeze the image on the screen first.

1. Find the Amplitude of the Waveform ______________ V (3)
2. Find the Period ______________ s (4)
3. Find the Frequency ______________ Hz (5)

 Cursors can be used for a variety of functions. One that will be done in this course is to measure phase shift between two waveforms.
CHAPTER 1. INTRODUCTION TO LABORATORY TOOLS

The MEASURE Function

The oscilloscope can also display measurements of waveforms such as the period, frequency, or peak-to-peak voltage. Pressing the MEASURE button on the oscilloscope will display the measurements on the right side of the screen. The button next to each box changes the channel as well as which type of measurement the box displays. For each measurement two things can be adjusted, Source and Measurement Type. One of the buttons will change whether Source or Type is highlighted. The other 4 buttons modify the selected item for each of the 4 measurements. Try displaying Frequency and Peak-to-Peak voltage for Channel 1.

The TRIGGER Function

The trigger level on the oscilloscope sets the voltage that triggers the oscilloscope. The trigger prevents horizontal drift of the trace. Find the TRIGGER LEVEL knob on the oscilloscope. Notice it moves an arrow on the right of the screen. Observe what happens if the arrow is moved outside the range of the waveform.

1.5.2 Using the Function Generation

Find the function generator on the top shelf of your lab bench. See figure 1.7. To connect to the function generator output, a BNC cable will be required.

![Tektronix CFG253 function generator](figure1.7)

Figure 1.7: Tektronix CFG253 function generator

![Connecting Oscope to Function Generator](figure1.8)

Figure 1.8: Connecting Oscope to Function Generator

1. Connect the BNC cable into the function generator’s MAIN output and then to the oscilloscope probe. The red mini-grabber should be connected to the tip of the probe while the black mini-grabber connects to the ground lead of the oscilloscope probe, as shown in figure 1.8.

2. Using the oscilloscope and the function generator, the next step will be to create a sine wave with the same characteristics (V_{peak} and frequency) as the PROBE COMP wave. The only difference is that the new wave will be a ‘sine’ wave not a ‘square’ wave. During this lab only sine wave will be used, but it is good to understand that there are other wave shapes used in engineering.

   Point of Interest: Sine waves are considered ‘pure’ wave and only have a frequency at a single point. All other repeating waveforms are composed of collections of sine waves at different frequencies.

3. Since the desired sine wave is known, the oscilloscope can be properly configured ahead of time to display it. Please note at this point the frequency generator should be turned off.

4. Using the CH1 VOLTS/DIV knob, set the voltage per division so that the wave will fit on the screen but not be too small. Then, using the SEC/DIV knob, set the seconds per division so that at least one period will fit on the screen, but not more than about ten periods.
1.6. MEASURING REAL SIGNALS

5. Turn the function generator on, and push the sine wave button under FUNCTION.

6. The amplitude knob closest to the power switch on the function generator allows you to adjust the voltage output, and you adjust the frequency using the multiplier buttons and the fine adjustment knob. However, the oscilloscope must be used to measure the frequency and amplitude; the knobs on the function generator cannot be trusted to be accurate.

7. Get your waveform as close as possible using the grid markers on the oscilloscope and the cursor function to find more precise numbers.

It may help to move the horizontal position of your wave so that the zero crossing of the wave goes through a grid line. Use the HORIZONTAL POSITION knob above the SEC/DIV knob to adjust the horizontal position.

8. Double-check with your TA that everything has been set up correctly. While this portion doesn’t need to be included in a lab report, it’s important that you understand how to use the function generator and oscilloscope together.

1.6 Measuring Real Signals

Now that the oscilloscope basics have been covered, inspection of a real signal is next. Using the lab supplied audio cables and the .wav files on the lab webpage, generate and display a waveform of a stereo signal. Choose two random files from the folder. However, make them as truly random as you can. Don’t just choose the first two, last two, middle two, etc....

Each group must answer the following question for the two random waveforms only!

File name: ______________________________ (6)

Right channel: Frequency: ______________________________ (7)

Right channel: Amplitude: ______________________________ (8)

Left channel: Frequency: ______________________________ (9)

Left channel: Amplitude: ______________________________ (10)

Time Shift of Right Channel compared to Left Channel: ______________________________ (11)

File name: ______________________________ (12)

Right channel: Frequency: ______________________________ (13)

Right channel: Amplitude: ______________________________ (14)

Left channel: Frequency: ______________________________ (15)

Left channel: Amplitude: ______________________________ (16)

Time Shift of Right Channel compared to Left Channel: ______________________________ (17)
1.7 LTspice Basic Tutorial

1.7.1 Getting Started

LTspice will be a program used at OSU for simulating and designing analog circuits. It is highly suggested that students learn how to use LTspice. There are many online tutorials that can make a great addition to this basic introduction to LTspice. Please note that the Windows version of LTSpice is used here. Mac/Unix-based versions might differ greatly in the UI.

1. Open LTspice and start a new schematic.
2. Look at Figure 1.9 to get a basic idea of the different menu icons.
3. Start by adding TWO resistor's to the schematic by clicking the ‘add resistor’ button in the top menu. The placement of the resistor does not matter. The circuit to be designed is a voltage divider. Right-clicking the mouse will return the cursor to a normal cursor.
4. Then add a voltage source by clicking on the ‘add component’ button in the top menu. Search through the list of parts and select the one labeled ‘voltage’.
5. To set the voltage and resistance of the components right click on the element and enter the value. One of the resistor’s should be $10\,\Omega$ and the other $25\,\Omega$. The voltage source should be set to 5V.
6. Once all the component values have been set they can be wired together. Use the ‘draw wire’ to connect them. See the schematic Figure 1.10.
7. It is very useful to give certain important nodes (often called Nets) distinct meaningful names. Name the net between the two resistors.
8. After connecting the components **MAKE SURE TO ADD A GROUND REFERENCE.** Use the ‘GND’ button in the top menu. Where you place the ground in the circuit will dramatically effect the outputs. Remember that ‘ground’ is only a concept as it acts as a reference point in the circuit to compare other things to. It is recommended that ground is connected to the negative terminal of the Voltage source.
9. Before simulating the circuit selecting the correct ‘type’ of simulation is important. Based on the type of information the user would like about the circuit, LTspice can simulate in different ways. Some common simulations are DC operating point, AC Analysis, and Transient Analysis. A the DC operation point analysis will be performed first. The output of this simulation is a list of all the DC node voltages and currents in the circuit. Select ‘simulate’ from the tool bar. Then select ‘edit simulation cmd’. See figure 2.10 Select the tab that says ‘DC op pnt’. Click ‘OK’ and place the ‘.op’ on the schematic.
10. The final step is to run the analysis. Select the the ‘Run’ button in the top menu. A file will then open with the node voltages and current through the elements.
1.8 Study Questions

For this laboratory turn in a copy of the lab with values filled in as well as answers to the following questions. Each person should turn in their own work (though you are free to discuss these questions with others).

1. In your own words, describe DC and AC coupling mode on the Oscope. Identify a reason to use each when inspecting a circuit.

2. In your own words, describe and draw the differences between a Magnitude voltage and a Peak-to-peak Voltage. Remember axis labels.

3. What is the frequency range of the function generators in the lab? What does the DC offset knob do?
