Xi'an Jiaotong-Liverpool University
西交利物浦大学

Department of Electrical and Electronic Engineering

EEE103 Electrical Circuit 1

Lab Manual & Report[1]

Name : 初云

ID Number : 1613453

Name(s) of Lab Partner(s) : 褚一杰

ID(s) of Lab Partner(s) : 1612312

Contents

1 Introduction to EEE103 Lab Experiment

1.1 Before Lab Experiment

Students should at least read through this lab manual for once to be able to complete the online lab-preparation quiz and carry out the experiment smoothly. You will be allocated with a fixed lab partner and a designated experimental table. Relevant information will be provided on ICE from Week 1. You should consult your module instructor or teaching assistant (TA) if you are unsure about the details.

1.2 During Lab Experiment

Before the lab, you will be informed about the time and venue. Please be punctual and take note of the following:

i. Record the measurement results in the tables provided.

ii. Follow the experiment procedure carefully and in step-by-step order.

iii. Feel free to communicate with your peers concerning relevant procedures and experimental results. In cases where necessary, please consult your TA for support.

iv. Please inform your TA after you complete the respective experiment for on-site assessment of the lab report.

1.3 After Lab Experiment

After the lab, please check your results and answer all the questions provided in the "Results and Discussions" part. Scan this lab report and generate a single PDF file for final submission. Alternatively, produce a full electronic lab report in PDF format on your own. The submission deadline is available in the Module Teaching Plan on ICE.

2 Experiment I: Voltage and Current Measurements, and Thévenin Equivalent Circuits

2.1 Objectives

In this experiment, you will be introduced to some important apparatus of DC circuit and signal measurement. You should achieve the following outcomes after completing Experiment I:

i. Know how to use a DC power supply.
ii. Acquire the necessary skills to construct a circuit on a breadboard.
iii. Be able to use a multimeter for measurement.
iv. Understand the fundamental idea of the Thévenin equivalent circuit.

2.2 Apparatus

Here is the list of items for your experiment. Please have the following items ready before you start:

i. DC power supply: see Section A.1 for more details;
ii. Breadboard: see Section A.2 for more details,
iii. Multimeter: see Section A.3 for more details,
iv. Banana cables for DC power supply, as shown in Figure 6,
v. A pair of probes for multimeter, as shown in Figure 9,
vi. Resistors (180 Ω, 100 Ω, 120 Ω, 680 Ω, 1.5 kΩ, and 1.2 kΩ): see Section A.4 for details on the colour code, and
vii. Jumper wires (5−10).

2.3 Task A: Measuring DC Voltage and Current

2.3.1 Experiment Procedure

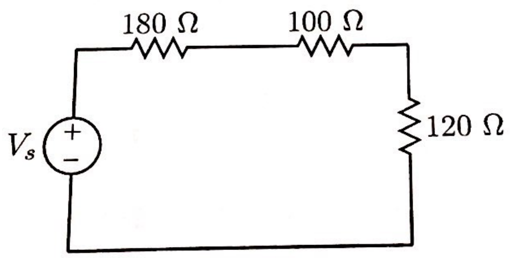


Figure 1: A simple circuit for DC voltage and current measurements

With the circuit setup in Figure 1, follow the procedures as below to complete this task.

i. Set up a circuit as shown in Figure 1 on the breadboard provided. Ensure that you have the correct resistors according to the colour reading. A guidance of the colour code is shown in Appendix A.4.

Use a multimeter to measure their values and fill in Table 1. Note that each resistor is distinguished by subscript. For instance, $R_{180\Omega}$ is to represent the resistor with a labelled resistance of 180Ω. After you have measured the resistance for all three resistors, the total measured resistance R_T^m can be obtained using the following equation:

$$R_T^m = R_{180\Omega} + R_{100\Omega} + R_{120\Omega} \tag{1}$$

Table 1: Measured Resistance

Resistor	$R_{180\Omega}$	$R_{100\Omega}$	$R_{120\Omega}$	R_T^m
Measured Resistance (Ω)	180.7	99.8	117.7	398.2

ii. Adjust the voltage output from the DC power supply to the required level, i.e. 10 V. You may want to use the multimeter to verify the voltage output. This is the common variable for both steps, i.e. $V_s = V_T = V_T' = 10$ V.

iii. Measure the voltage across each resistor: $V_{180\Omega}$, $V_{100\Omega}$ and $V_{120\Omega}$ in Table 2 and current through the circuit I in Table 2 using the multimeter.

R_T is then calculated by Eq. (2). This is the total resistance calculated based on Ohm's Law as shown below:

$$R_T = V_T/I \tag{2}$$

Record your results in Table 2.

Table 2: Measuring the current and voltage for each resistor

V_s (V)	$V_{180\Omega}$ (V)	$V_{100\Omega}$ (V)	$V_{120\Omega}$ (V)	V_T (V)	I (A)	$R_T = V_T/I$ (Ω)
10	4.56	2.53	2.92	10	0.249	401.61

Note that in this step, we measure the current I and the total voltage V_T. Use Ohm's law to calculate the total resistance.

iv. We now use the measured resistance value and the known terminal voltage, i.e $V_T' = V_s$ to derive the currents and voltages for each resistor.

Using the resistance obtained in Table 1, we can directly obtain the total resistance R_T'. $R_T' = R_T^m$. Given that the total voltage $V_T = V_T' = V_s$ is known, we can apply the voltage division and Ohm's Law to find the individual voltage across each resistor and the current I'.

For example, the voltage across 180 Ω resistor can be calculated using the following equation:

$$V_{180\ \Omega}' = \frac{R_{180\ \Omega}}{R_{180\ \Omega} + R_{100\ \Omega} + R_{120\ \Omega}} \times V_T' \tag{3}$$

Similarly, we can calculate the voltages across $R_{100\ \Omega}$ and $R_{120\ \Omega}$ and fill in Table 3.

Calculate the current through the circuit using:

$$I' = \frac{V_T'}{R_T'} \tag{4}$$

Write down your answers in Table 3.

Table 3: Calculate the voltage and current for each resistor

V_s (V)	$V_{180\ \Omega}'$ (V)	$V_{100\ \Omega}'$ (V)	$V_{120\ \Omega}'$ (V)	V_T' (V)	I'(A)	$R_T' = R_T^m$ (Ω)
10	4.5	2.5	3	10	0.025	400

2.3.2 Results and Discussions

After obtaining the results from the experiment, answer the following questions:

i. Is there any difference between I and I'? Is there any difference between R_T and R'_T? What is the reason for your observation?

1. In this experiment, I is measured by the multimeter, and I' is calulated by the formula $I' = \frac{V_a}{R_T}$. So the reason for the observation may be that the multimeter's measurement. exist the errors. and the experiment is not constant which means the experiment may effect the result.

2. The R_T is calculated by $R_T = \frac{V_T}{I}$ and R'_T are obtained by $R'_T = R_T$. From the result. we've got R'_T is larger than R_T. The reason maybe the resistance will change as the temperature of resistors are increasing as the current flowing.

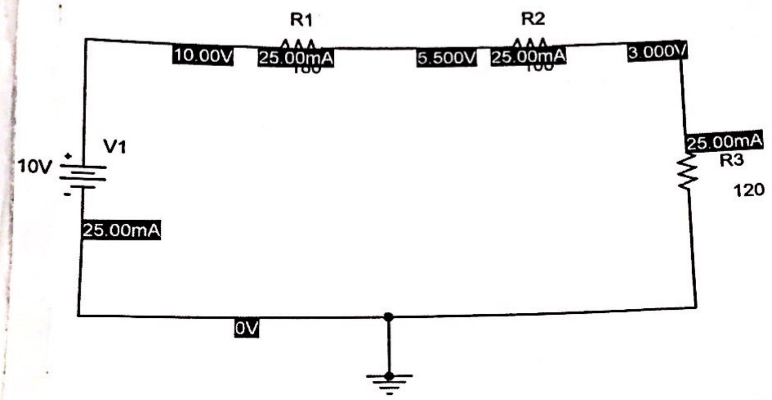
ii. In Table 2, if you sum up the voltage across each resistor and compare the sum with V_s, can you see the difference? Please give your explanation.

~~[illegible]~~ $4.56+2.53+2.92 = 10.01 > 10 = V_s$.

The sum of the voltage is larger, The power supply maybe not constant. and the measurement always exist the error. what's more, the change of the experiment may also affect the results

2.3.3 PSpice Simulation

You need to use PSpice to simulate and verify the results you have obtained. For instructions of using PSpice, refer to Page 103 in the textbook. Please print, cut and glue the output from PSpice simulation results onto the box below.



Include a short description of the PSpice's output below:

From the Pspice's simulation. figure, we can see that the data are very close compared with our experiment result. which means our experiment is successful. In addition, it also shows that our result exists some errors, which we should take it seriously next time.

2.4 Task B: Finding the Thévenin Equivalent Circuit

2.4.1 Thévenin Theorem

Thévenin theorem is an important concept for electrical circuit analysis. The theorem states that any two-terminal network of resistors and voltage sources can be represented by just one resistor R_{Th} and a voltage source V_{Th}. Figure 2 illustrates the scenario of the Thévenin equivalent circuit.

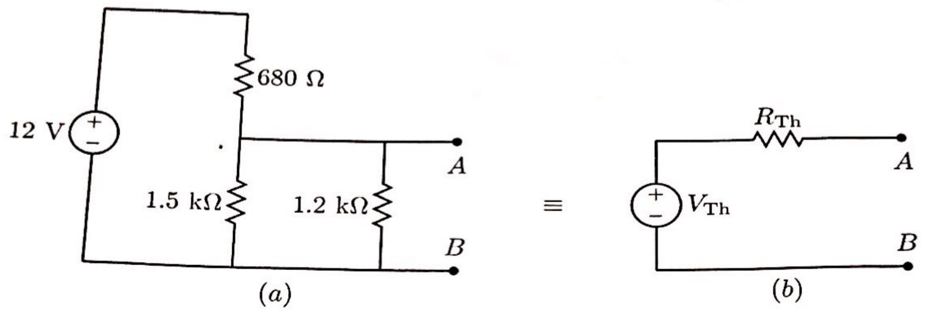


Figure 2: (a) Original circuit, (b) Thévenin Equivalent circuit

We can see that the circuit diagram shown in Figure 2(a) can be transformed into a simple circuit shown in Figure 2(b). According to Thévenin theorem, these two circuits have an equivalent effect at terminals A and B. For instance, the terminal voltage across A and B will be the same for both circuits.

2.4.2 Experiment Procedure

In this task, you will learn how to obtain these two important elements R_{Th} and V_{Th} in the Thévenin theorem and observe how these two circuits are equivalent as seen from terminals A and B.

i. **Circuit setup for Figure 2(a).** Set up the circuit according to the circuit diagram in Figure 2(a) on the breadboard.

ii. **Measurement of V_{Th}.** Use a multimeter to measure the open circuit voltage between terminals A and B. **Note that this voltage is the measured V_{Th}.** Record the value in Table 4.

iii. **Calculation of V_{Th}.** Apply voltage and current laws to calculate the open voltage between terminal A and B. **Note that this voltage is the calculated V_{Th}.** Record the value in Table 4.

iv. **Measurement of R_{Th}.** Short-circuit the DC voltage source in Figure 2(a) and measure the resistance between terminals A and B. **The measured resistance is R_{Th}.** Record the value in Table 4. According to Thévenin theorem, the Thévenin equivalent resistor can be obtained by measuring the terminal resistance of a network while short-circuiting the voltage source or open-circuiting the current source inside.

v. **Calculation of R_{Th}.** Short-circuit the DC voltage source in Figure 2(a). **Calculate the resistance between terminals A and B, which is the calculated R_{Th}.** Record the value in Table 4.

vi. **Obtain the $I_{1k\Omega}$ in original circuit.** Connect a 1 kΩ resistor to terminals A and B in Figure 2(a). Use a multimeter to measure the current $I_{1k\Omega}$ through this 1 kΩ resistor. Record the value in Table 5.

vii. **Obtain the $I_{1k\Omega}$ in Thévenin equivalent circuit.** Set up the circuit according to the circuit diagram in Figure 2(b) on the breadboard. Use the calculated V_{Th} and R_{Th} for the setup. Connect a 1 kΩ resistor to terminals A and B. Use a multimeter to measure the current $I_{1k\Omega}$. Record the value in Table 5.

Table 4: Values of V_{Th} and R_{Th}

	Measured	Calculated
Thevenin voltage, V_{Th}	5.90	5.94
Thevenin resistor, R_{Th}	334.1	336.7

Table 5: Current through 1 kΩ resistor

	Original Circuit	Thévenin Equivalent Circuit
$I_{1k\Omega}$	4.46mA	4.48mA

2.4.3 Results and Discussions

Answer the following questions:

i. Write a short description of your findings based on this experiment.

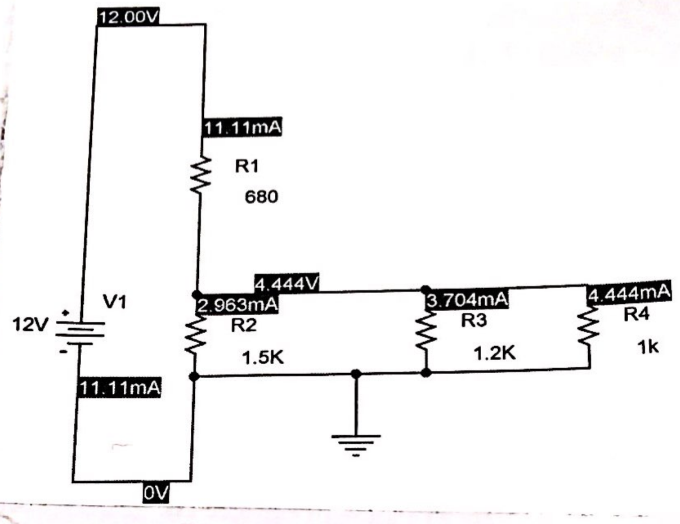
① The measured and calculated value of V_{TH} and R_{TH} is unequal, so there are some differences between ideal value and actual value due to the errors. ② The current through the original circuit is equal to the Thevein equivent circuit when apply 1kΩ resistor.

ii. Based on the comparison of the current through the 1 kΩ resistor in both configurations, is it sensible to use the circuit in Figure 2(b) to replace the one in Figure 2(a) if we would like to study any linear network connected between terminals A and B? State your reason(s).

Yes, Here is the proof ① from (b) we know: $U_{AB} = -I_{1k\Omega}R_{TH} + V_{TH}$ ② If we apply a independent current source between AB in (a), like . Using superposition theorem, when current is not working, $u' = V_{TH}$, when V_{oc} is short, we could get $u'' = -I_{1k\Omega}R_{TH}$. ③ Since $U = U' + u'' = -I_{1k\Omega}R_{TH} + V_{TH}' = U_{AB}$. Then (b) is able to place (a).

2.4.4 PSpice Simulation

You need to use PSpice to simulate both circuit setup in Figure 2 and verify your results. Once again, print and glue the output from PSpice onto the box below. Please print, cut and glue the output from PSpice simulation results onto the box below.



Include a short description of the PSpice's output below:

The current, through R_4 is: $I_4 = 11.11mA$, through R_5 is: $I_5 = 2.963mA$.
through R_7 is: $I_7 = 3.704mA$, In AB, the $I_{1k\Omega} = 4.444mA$.
$V_{TH} = 4.444V$.
Based on the simulation, there are some error compared with the figure, However the data is near, The experiment is successful.

3 Experiment II: Using the Oscilloscope

3.1 Objectives

A digital oscilloscope uses an analog-to-digital converter (ADC) to convert the measured voltage into digital information. After this experiment, you should be able to use the Oscilloscope to conduct basic signal calibrations and measurements.

3.2 List of Apparatus

Here is the list of items for this experiment. Please get the following items ready before you begin the experiment.

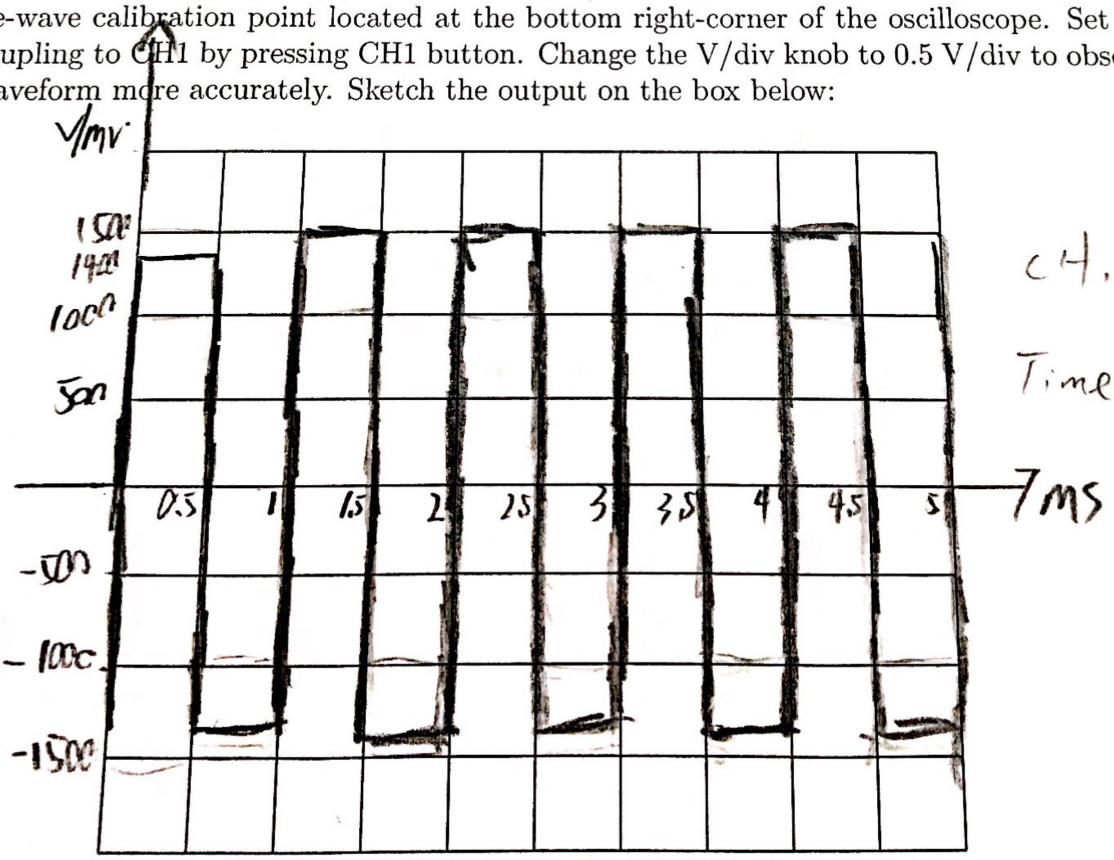
i. Oscilloscope,
ii. Signal generator,
iii. DC power supply,
iv. One oscilloscope probe,
v. One coaxial lead probe,
vi. Probe adjustment tool (very limited number, available upon request),
vii. Digital multimeter,
viii. Resistors (2×15 kΩ, 1×2.2 kΩ), Capacitors (2×1 μF, 1×0.1 μF), and
ix. Breadboard.

You are required to draw the oscilloscope outputs on the lab manual using the boxes provided. The recorded output are used for on-site assessment.

3.3 Task A: Calibration

Before using an oscilloscope, it has to be calibrated to ensure its normal operation. Follow the procedure below for this purpose.

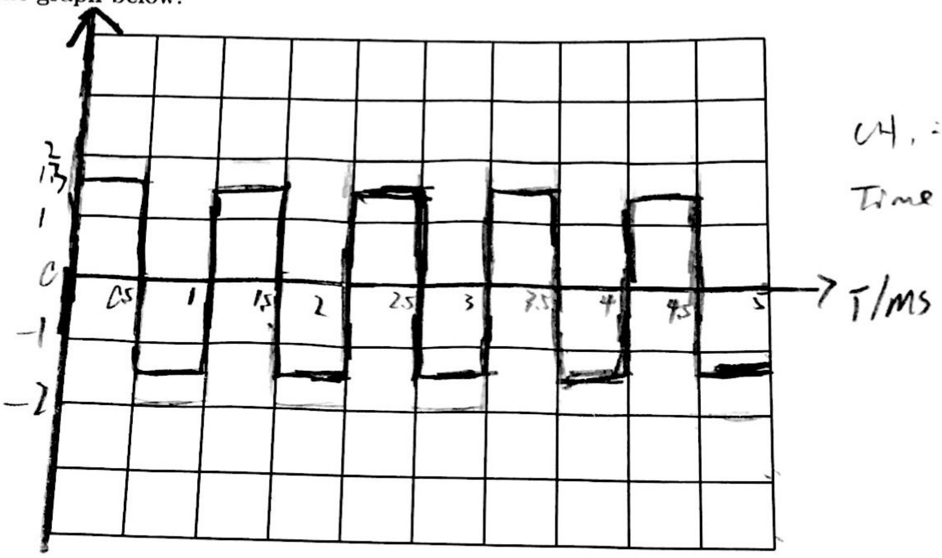
i. Connect an oscilloscope probe (switched to $\times 1$) to CH1 input and attach the probe to the 3V square-wave calibration point located at the bottom right-corner of the oscilloscope. Set the DC coupling to CH1 by pressing CH1 button. Change the V/div knob to 0.5 V/div to observe this waveform more accurately. Sketch the output on the box below:



CH1 = 500 mV.

Time = 500.0 us

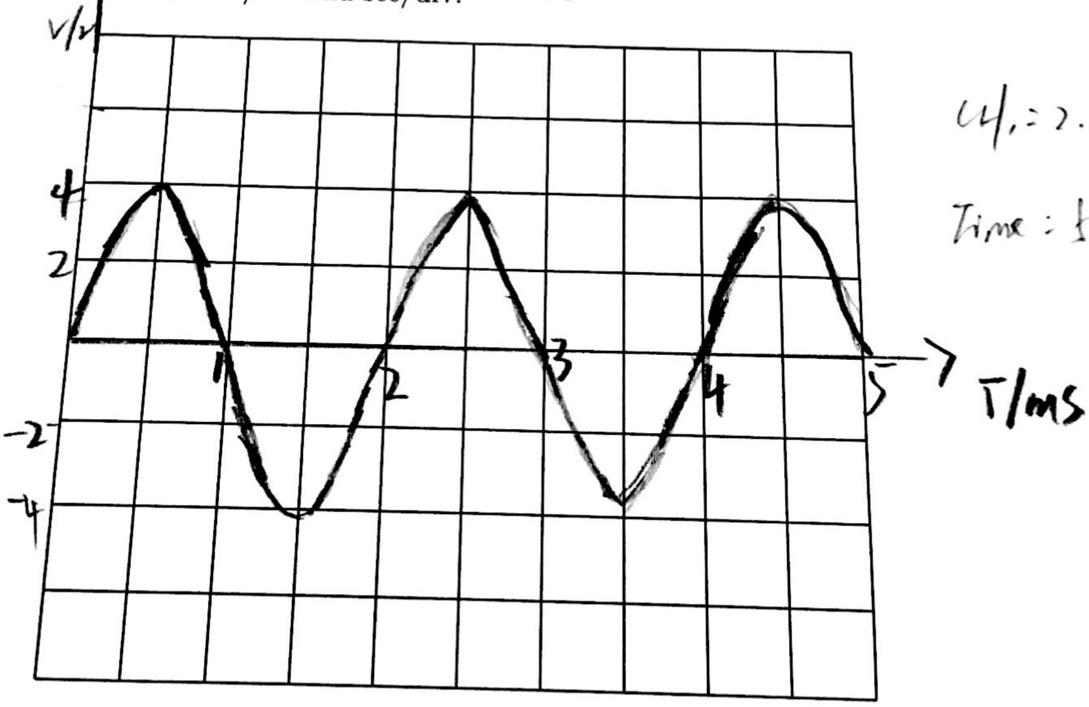
ii. Now switch your probe to ×10 mode[2]. Reconnect the 3V calibration point. Using the probe tool, gently adjust the probe head capacitance until the best square-ware is obtained by rotating the small screw built into the probe near the ×1 / ×10 switch. Once again, record the V/div value in the graph below:



CH1: 1.00V

Time = 500.0us

iii. Locate your **AC signal generator** and examine its controls. Adjust it to generate a 500 Hz sine wave. For the oscilloscope, set the V/div and sec/div to appropriate level[3]. Connect the signal from AC signal generator to oscilloscope using a probe (either ×10 probe or a coaxial lead). Increase the amplitude of the sine wave to 8 Vpp [4]. Record the waveform in the graph below with information of V/div and sec/div:



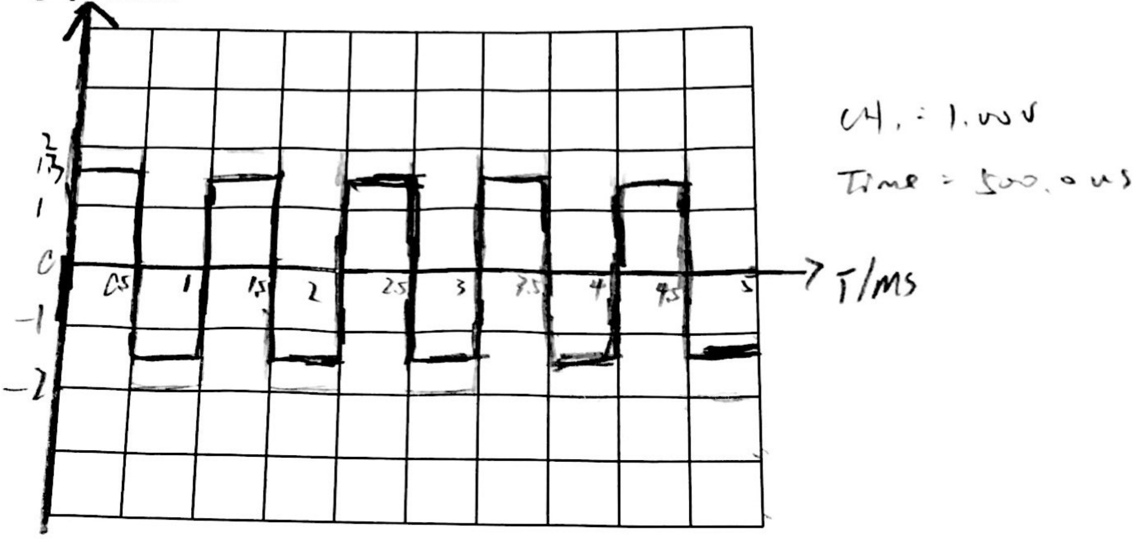
CH1: 2.00 V

Time : 500.0us

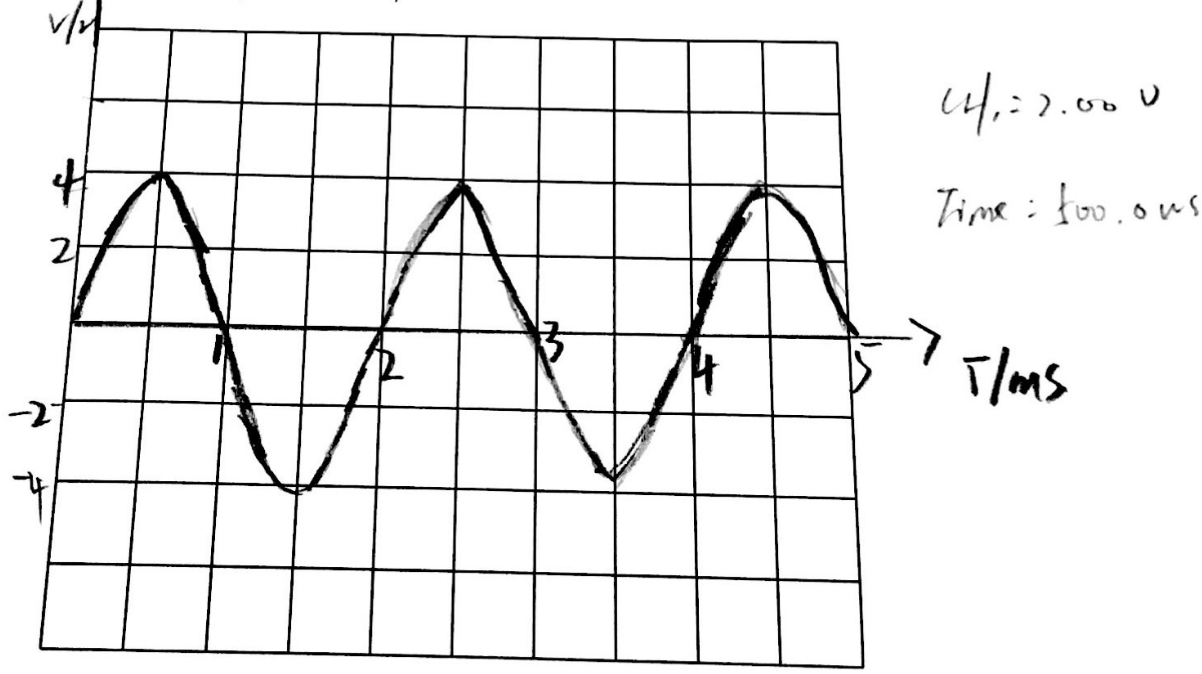
iv. Adjust the AC signal generator to 100 Hz, 8 Vpp . Check that the signal input of the oscilloscope is set to Ch1. If the signal is not static (drifting), press the 50% to set the trigger level to zero in the middle of the screen. Set the slope to trigger on the positive (rising) slope by pressing trigger menu button→slope→rising edge. Now set the slope to the negative (falling) slope and observe the difference. You should see that the signal starts from the left of the

[2] The signal amplitude is attenuated by a factor of 10 times. For instance, if the input signal amplitude is 10V, the captured signal amplitude will be 1V in oscilloscope.

ii. Now switch your probe to $\times 10$ mode[2]. Reconnect the 3V calibration point. Using the probe tool, gently adjust the probe head capacitance until the best square-ware is obtained by rotating the small screw built into the probe near the $\times 1$ / $\times 10$ switch. Once again, record the V/div value in the graph below:



iii. Locate your **AC signal generator** and examine its controls. Adjust it to generate a 500 Hz sine wave. For the oscilloscope, set the V/div and sec/div to appropriate level[3]. Connect the signal from AC signal generator to oscilloscope using a probe (either $\times 10$ probe or a coaxial lead). Increase the amplitude of the sine wave to 8 Vpp [4]. Record the waveform in the graph below with information of V/div and sec/div:



iv. Adjust the AC signal generator to 100 Hz, 8 Vpp . Check that the signal input of the oscilloscope is set to Ch1. If the signal is not static (drifting), press the 50% to set the trigger level to zero in the middle of the screen. Set the slope to trigger on the positive (rising) slope by pressing trigger menu button→slope→rising edge. Now set the slope to the negative (falling) slope and observe the difference. You should see that the signal starts from the left of the

[2] The signal amplitude is attenuated by a factor of 10 times. For instance, if the input signal amplitude is 10V, the captured signal amplitude will be 1V in oscilloscope.

[3] You can press the AUTO button for quick setting.

[4] Vpp means peak-to-peak value of the voltage

screen by "falling downward" if the trigger slope is set to negative and the signal starts by "rising upward" if the trigger slop is set to positive.

Now adjust the trigger level knob and record the trigger voltage whereby the signal starts to drift. Trigger voltage = 14.24 V.

3.4 Task B: Phase Shifted Signal

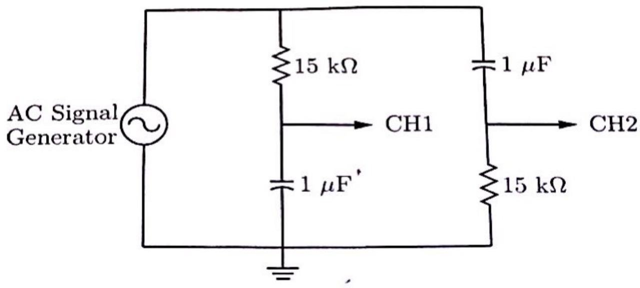
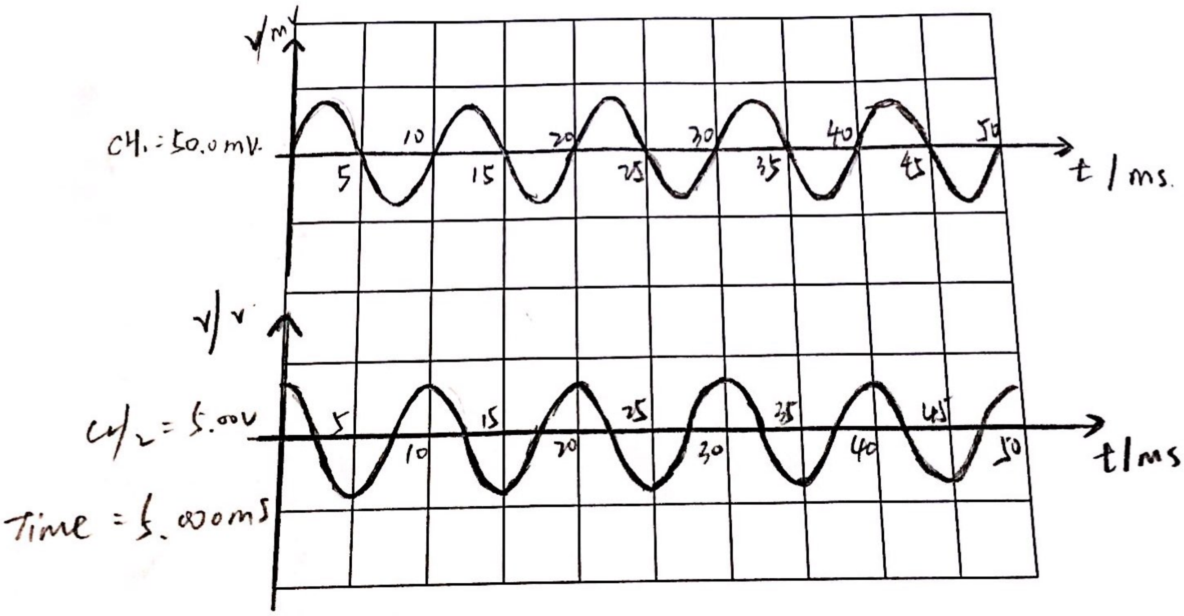


Figure 3: $\pi/2$ radian phase shifting circuit

Follow the procedure below and draw the two output graphs.

i. Connect the circuit in Figure 3. Adjust the AC signal generator to 100 Hz, 8Vpp. From Figure 3, connect output from 1 μF to CH1 and output from 15 kΩ to CH2 of the oscilloscope. Press the AUTO button for a quick setting of V/div and sec/div. To see both output signals, make sure that both CH1 and CH2 buttons are pressed. Draw these two signals in the graph below with the information of V/div and sec/div:



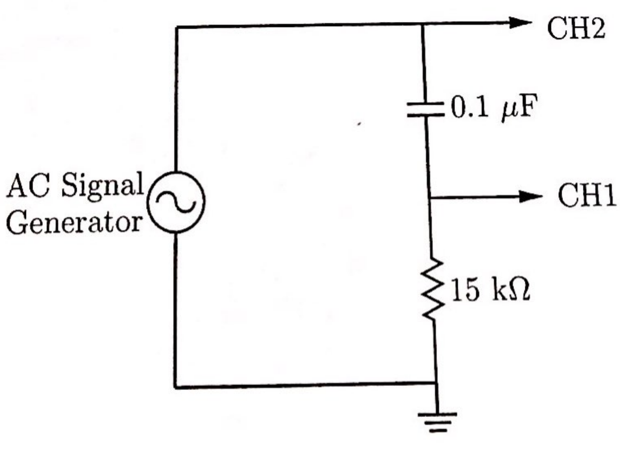
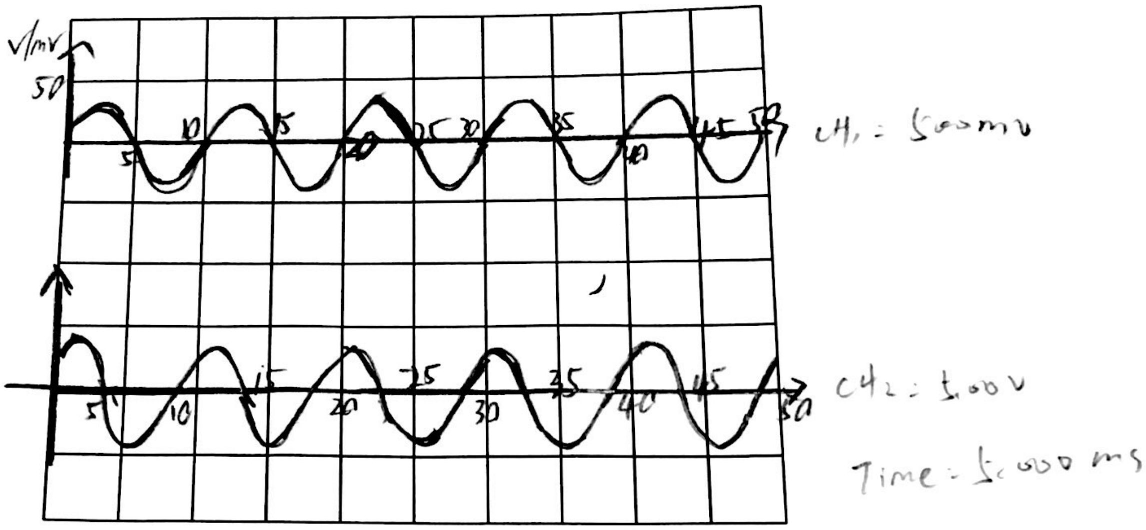


Figure 4: Phase shifting circuit

ii. Next, change the passive components to the one shown in Figure 4 and draw the output signal in the graph below and provide the information of V/div and sec/div:



3.5 Results and Discussions

Write a short conclusion for the output signals obtained for both circuits.

From the output signal of both circuits, we can see that resistance in figure 3 are different with the resistance in Figure 4. They have different phase in the figure, which shows that the different signals can be seen as inductance voltage.

4 Appendices

A Apparatus

A.1 DC Power Supply

The power source is an essential apparatus for the use in the lab. Figure 5 is an example of a typical DC power supply. The voltage output can be adjusted by twisting the voltage knob. You should note that there are three channels and the output from different channels is different. In this experiment, independent mode between channels is adopted so that each channel provides independent output power for external circuits.

Figure 5: A typical DC power supply for lab usage

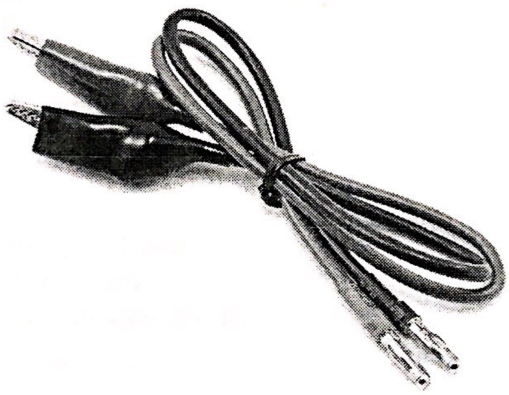


Figure 6: Banana cables for the DC power supply

A.2 Breadboard

A breadboard is used as the base for prototyping of electronic circuits. It does not require soldering and reusable and thus is popular to be used in prototyping and experimental design. Figure 7 is a typical breadboard used for a small project. It contains a series of holes electrically connected underneath. Figure 8 illustrates the connections underneath. If you are not sure whether any two holes are connected underneath, you can use a multimeter to check its resistance.

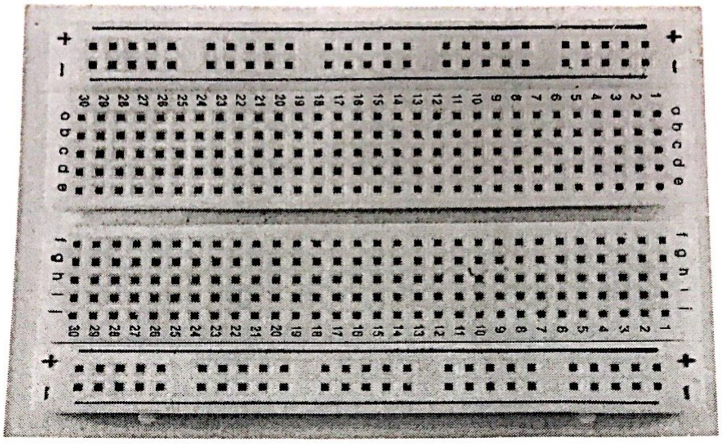


Figure 7: A picture of the breadboard used in the experiment.

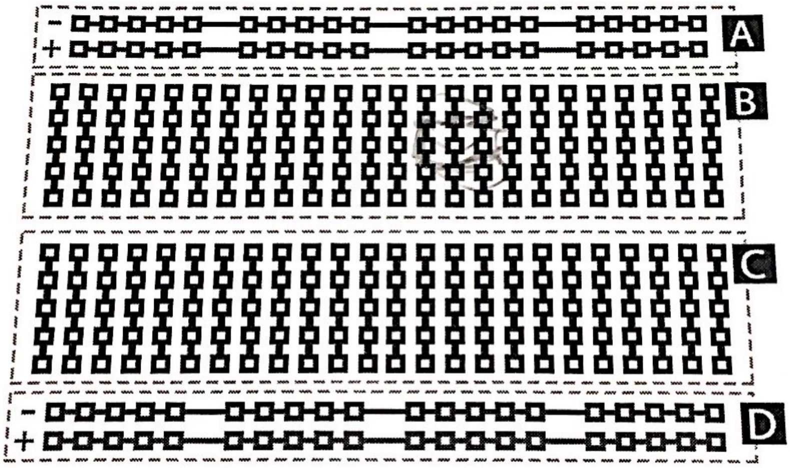


Figure 8: Connection layout diagram of breadboard.

A.3 Multimeter

A picture of a multimeter is shown in Figure 9. You need two long test probes for measurement. **Always remember to use the larger range (10 A) for current measurement before switching to a smaller one as shown in Figure 9.** If you are not sure about this, please consult your TA or a lab technician.

A.4 Resistor Color Code

The resistor colour code is a way of showing the value of a resistor. Instead of writing the resistance on its body, which would often be too small to read, a colour code is used. Ten different colours represent the numbers 0 to 9. Take note that the resistors provided in the lab have either four or five coloured bands. Please refer to the Figure 10 below for the details on the relevant colour codes.

Figure 9: A multimeter connected with a pair of test probes

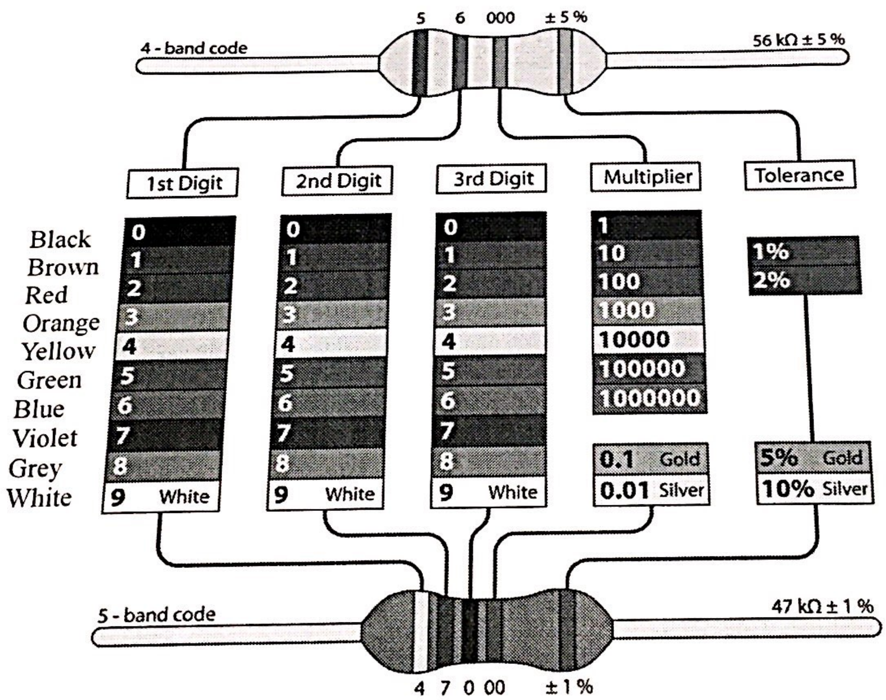


Figure 10: Color Codes of 4-band and 5-band resistors.

B Oscilloscope

B.1 Background

This lab presents the basic controls and the usage of an oscilloscope. It acquires the waveform as a series of samples and stores these samples until it accumulates enough samples to describe a waveform. The oscilloscope then re-assembles the waveform and displays it onto the screen. Figure 11 shows some of the important controls and settings of an oscilloscope.

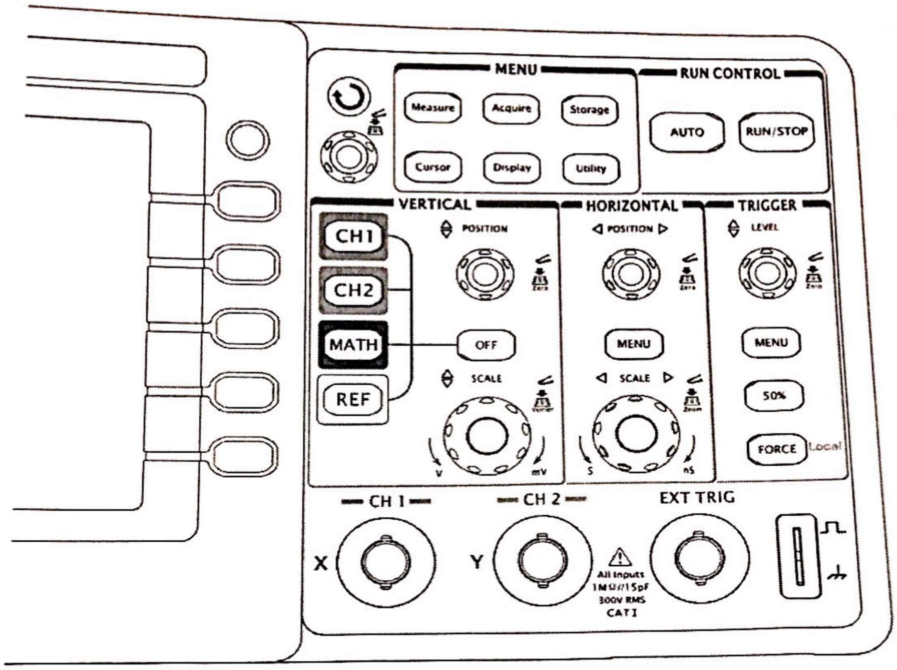


Figure 11: The control panel of an oscilloscope.

B.2 Oscilloscope Probes

Connecting an oscilloscope to measure or observe a voltage produced by an electronic circuit should ideally not affect the circuit. Therefore, the input resistance to each channel should be as high as possible, and the input capacitance should be as low as possible. Both would contribute to high input impedance. The input resistance of both channels is 1MΩ with input capacitance of approximately 15pF for the oscilloscopes used in this experiment. A probe can usually be switched to give either a direct reading on the oscilloscope (the $\times 1$ mode) or a signal reduced in amplitude by a factor of 10 (the $\times 10$ mode). If accurate measurements of the voltage at frequencies greater than 1 kHz are required, the probe should be switched to $\times 10$ mode. This increases the input resistance at the probe tip and reduces the input capacitance.

B.3 Time Mode

In the time (or sweep) mode, the oscilloscope samples the voltage levels of the input channels over time. The voltage on the input channels causes the display to deflect vertically, and the horizontal axis represents the time domain. You can adjust the seconds/division (sec/div) knob to obtain more or less of a waveform. Also, you can adjust the vertical sensitivity by turning the volts per division (V/div) knob.

B.4 Trigger Control

The oscilloscope can be used to observe repetitive signals. If the oscilloscope is sweeping the signal from a different location of its cycle each time, the oscilloscope screen will show a drifting wave shape. A trigger is used to catch the displayed signal in the same location every time the signal is captured, so that the repetitive signals appear static.

The trigger controls include source selection, level selection, slope selection, and time delay. The source can be set to CH1, CH2, line (e.g. 50 Hz power line), or external. The level control adjusts the voltage level at which the waveform will be captured. The slope of trigger determines if the trigger will on the signal's positive slope or negative slope. See Fig.12 for details. There is a trigger level indicator to show the voltage level to obtain static waveform.

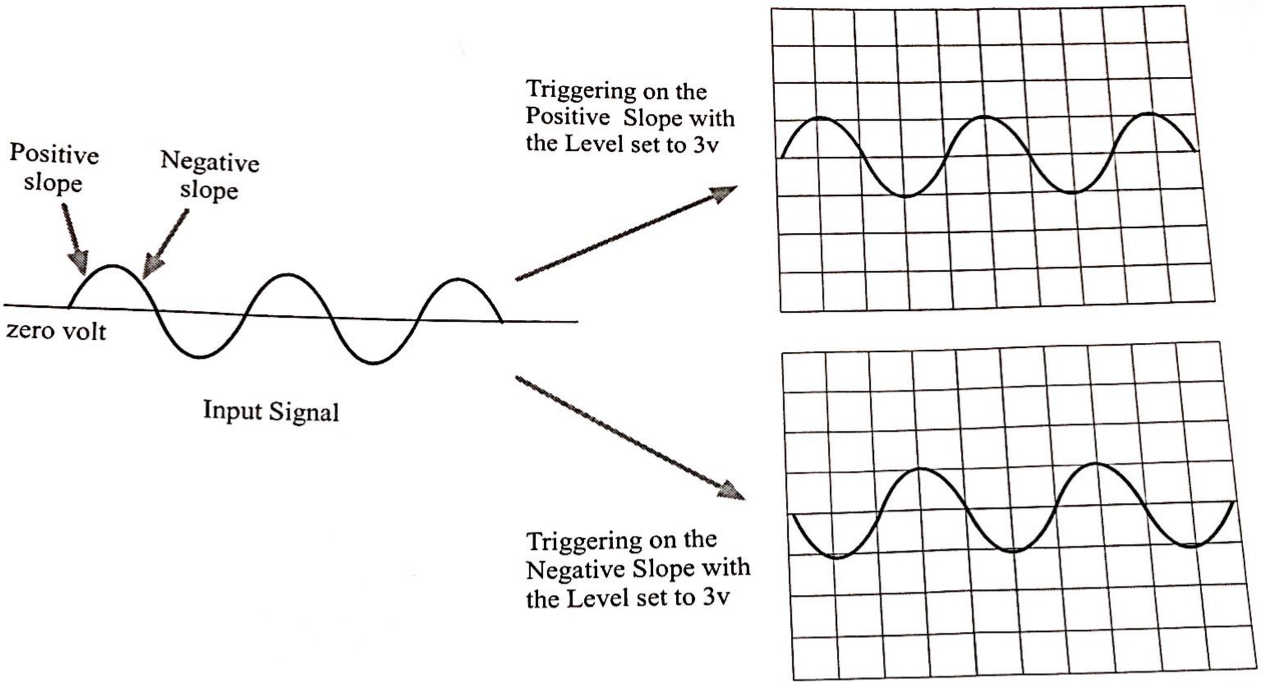


Figure 12: Illustration on the effect of different trigger slopes in an oscilloscope.