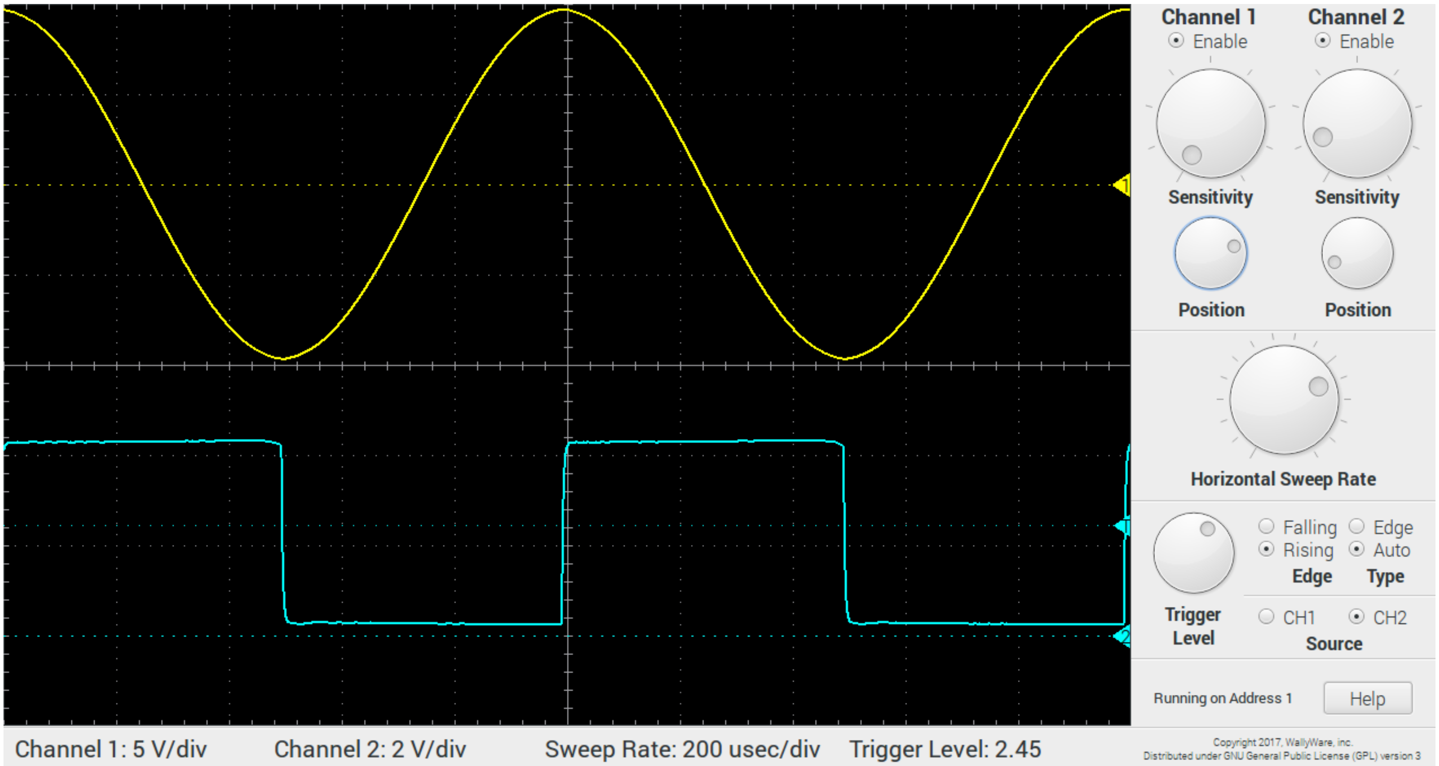
DAQC2plate Oscilloscope Manual



Revision 1.0

Introduction **3**

Before You Get Started **4**

Specifications **4**

Oscilloscope Operation **5**

- Layout 5
- Connections 6
- Vertical Controls 7
    - Channel Enable 7
    - Sensitivity 8
    - Position 10
- Horizontal Sweep 12
    - Control 12
    - Aliasing 14
- Trigger Functions 15
    - Source 15
    - Edge 17
    - Type 17
    - Level 17

Revision History **20**

Introduction

The oscilloscope is probably the most powerful tool on an Electrical Engineer's workbench. With it, a number of signal characteristics can be examined and measured. These include amplitude, frequency, duty cycle, waveform shape, and the presence of noise.

The first oscilloscopes were built around cathode ray tubes (CRTs) like older televisions. In those devices, a beam of electrons was focused onto a small screen to draw a rapidly moving dot. While the vertical position of this dot was being controlled by the instantaneous voltage on the input channel, the horizontal position was controlled by a time base internal to the scope. The resultant image was a picture of the signal on the CRT:

Image courtesy of [Richard Sears Vintage Electronics]

Today, oscilloscopes use high speed A/D converters to capture the waveform and small LCDs to display it. But for the the most part, the controls on the front panel and terminology are the same.

The oscilloscope application described in this manual is a basic two channel instrument that will allow a beginner to learn and a more experienced user to make quick measurements.

Before You Get Started

The following applies to all of the DAQC2plate applications:

1. You might be tempted to run the Oscilloscope App and the Function Generator App at the same time. However, due to how the Raspberry Pi handles certain hardware resources, this could cause contention issues which would result in both apps misbehaving. Even if you try this with two different DAQC2plates on a single RPi, you could still have problems. So, do not run more than one application at a time and follow this rule: One stack = One app.
2. When launched, an application will run on the first addressed DAQC2plate on the stack. For example, if you have two DAQC2s stacked together at addresses 0 and 5, the application will run on the board at address 0.
3. Do not expect these applications to have the same performance of $1000 instruments from Fluke and Tektronix. They are for the most part, capable of operating in the audio band of 10 to 20Khz.
4. All knobs can be rotated with a mouse pointer for coarse adjustments. For fine adjustments, the left and right arrows on your keyboard can be used.

Specifications

The oscilloscope mode of the DAQC2plate has the following specifications:

Input Range	+/-10 volts
Input Bandwidth	100,000 hertz
Offset Voltage	+/- 50mV
Gain Error	+/- 1.5%
Sample Rate - Single Channel Operation	1M samples/second maximum
Sample Rate - Two Channel Operation	500,000 samples/second maximum
Resolution	12 bits
Horizontal Sweep Rates	1000, 500, 200, 100, 50, 20, 10, 5, 2, 1 msec/div, and 500, 200, 100, 50, 20, 10 μsec/div
Vertical Sensitivity Ranges	5, 2, 1 Volt/div, and 500, 200, 100, 50, 10 mV/div
Trigger Delay + Jitter	4+2 microseconds

Oscilloscope Operation

Layout

The various regions of the oscilloscope display are highlighted below. Note that there are five primary functional areas:

1. The Screen
2. Enable and the Vertical controls
3. Horizontal Controls
4. Trigger Controls
5. Information Area below the Screen.

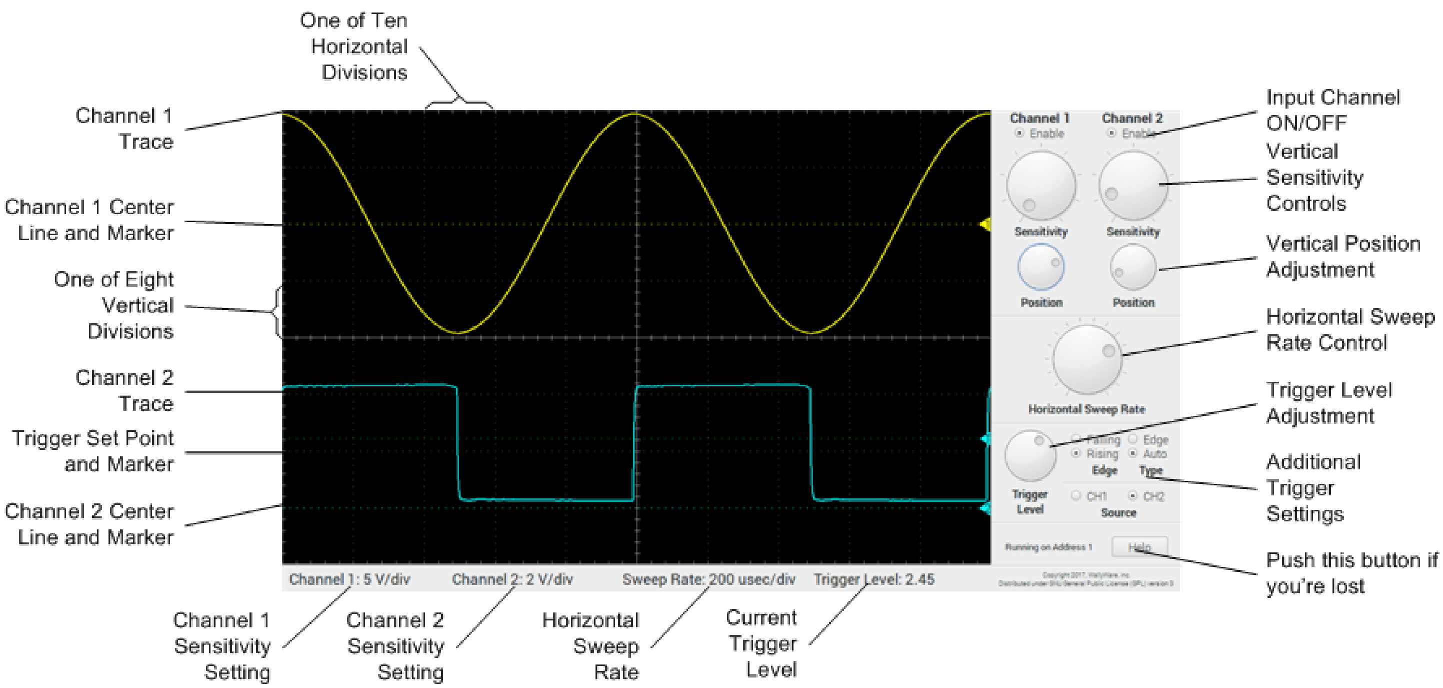


Figure 1

Connections

Refer to the figure below on where to connect your signals to the DAQC2plate. Channel 1 is on Analog Input 0 (AIN0) and Channel 2 is on Analog Input 4 (AIN4). These are pins 1 and 5 on the terminal block. Connect the ground signal from the circuit you're probing to pin 9 of the terminal block.

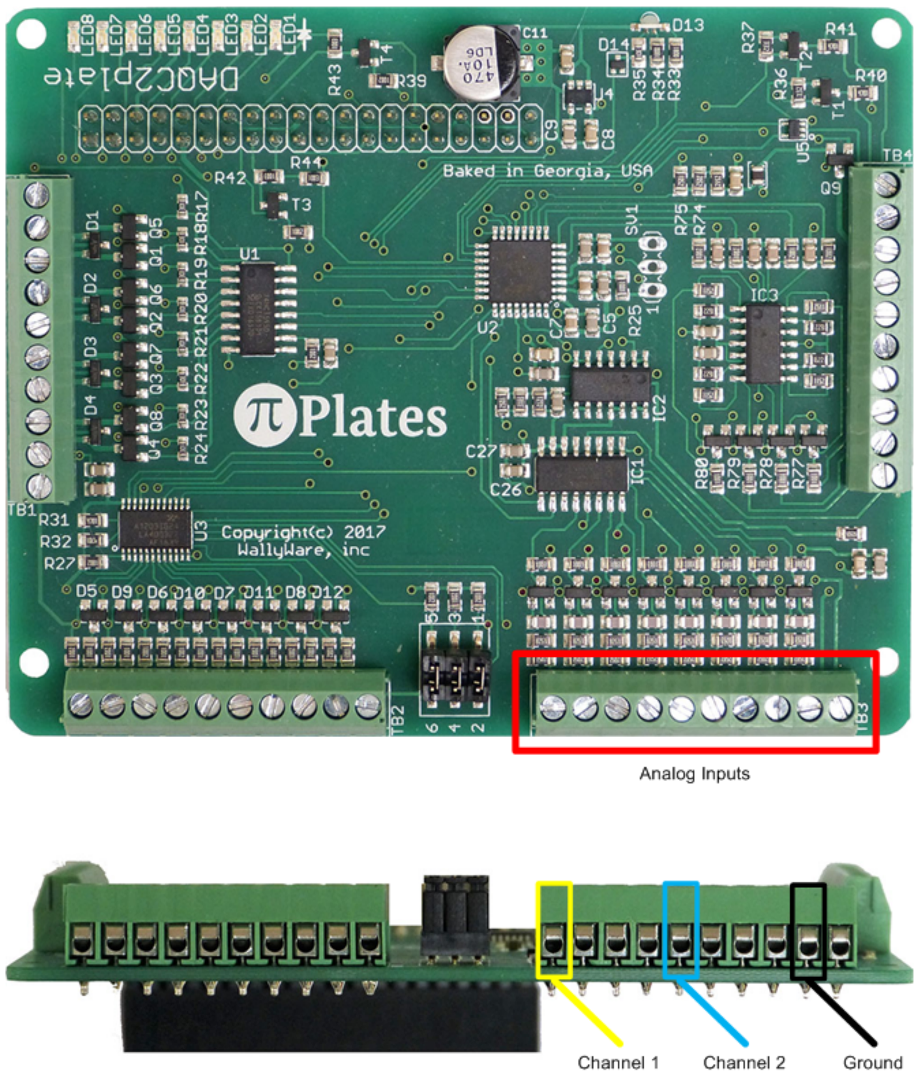


Figure 2

1. The input impedance of an analog input is 1 Megohm. As long as as you are measuring signals from sources such as power supplies and/or amplifier outputs, this should not be a problem. However, if you are probing passive networks that include resistor values greater than 100 Kohms, the input impedance may affect your circuit and your results.
2. The input bandwidth of 100 Khz. For sinusoidal waveforms this will likely not be a problem but for signals with higher frequency components like square waves, you may see some slight rounding of the edges.
3. If an input is enabled and left unconnected, The display will show a DC signal of 1.33VDC. So, ground that input if you want to see zero volts, or disable it.

Vertical Controls

The areas of the application that are relevant to the Vertical Controls are shown below:

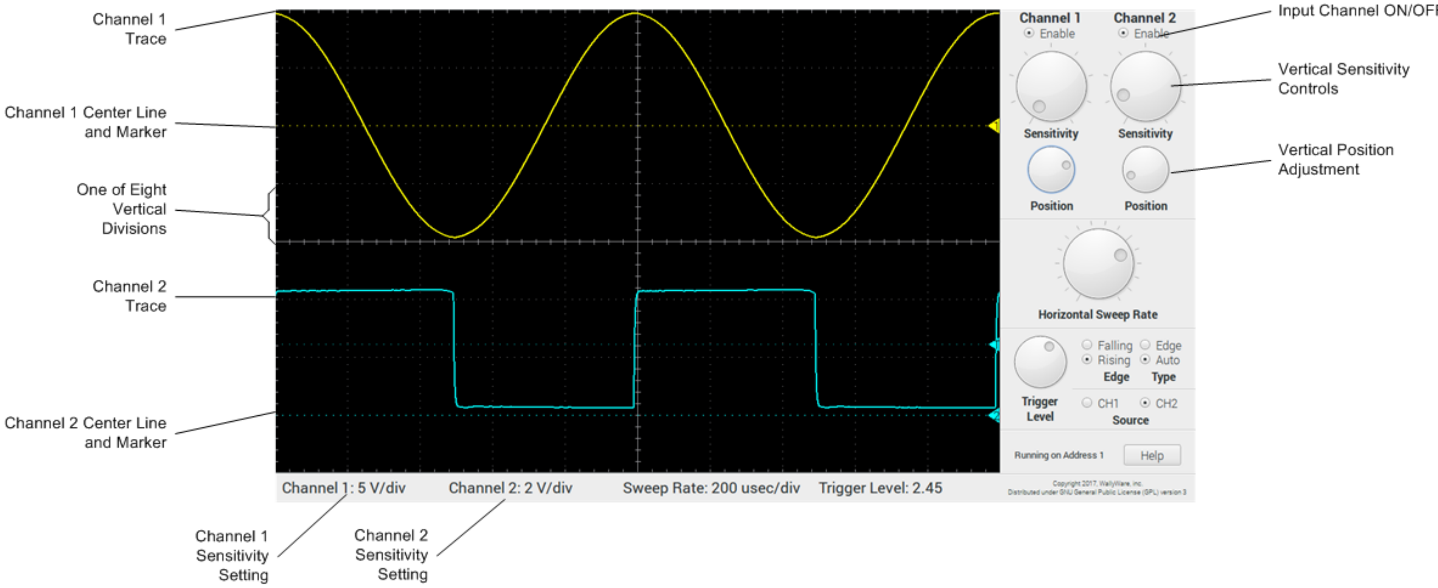


Figure 3

Channel Enable

Before appearing on the screen, an input channel has to be Enabled. The two buttons on the top of the control area tell the oscilloscope to which channel to display. You can have any combination of the two channels enabled.

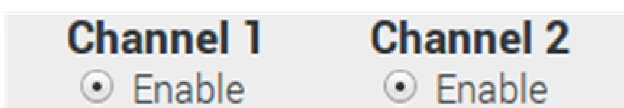


Figure 4

When first launched, the Oscilloscope application has Channel 1 enabled and Channel 2 disabled. Once enabled, the data from the selected channel will appear on the screen in a unique color. Channel 1 data is yellow and channel 2 data is blue.

In addition, there is a small, numbered chevron on the right side of the screen that indicates the channel number and zero volt position of the signal:

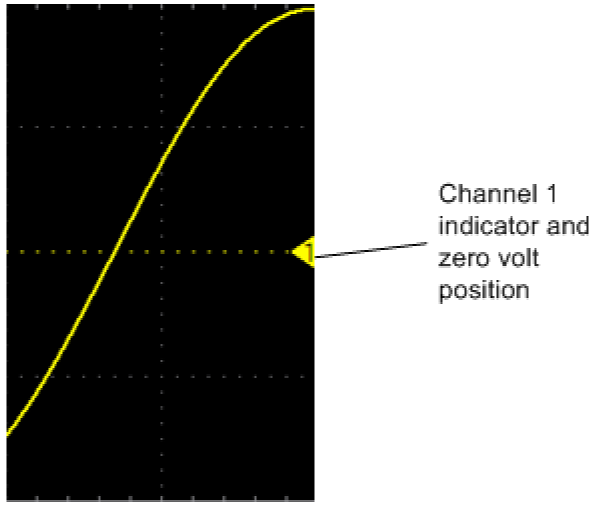


Figure 5

Sensitivity

All oscilloscopes have a grid on their screens. The space between any two adjacent horizontal or vertical lines on the grid is called a division. On an oscilloscope, the vertical axis is measuring volts so the units of vertical sensitivity are volts/division or volts/div. As you rotate the sensitivity knob clockwise, the volts/div value will decrease (because the sensitivity is decreasing) and the vertical size of your signal will grow.

Figure 6

The current vertical sensitivity for both channels is displayed in the information area at the bottom of the screen.

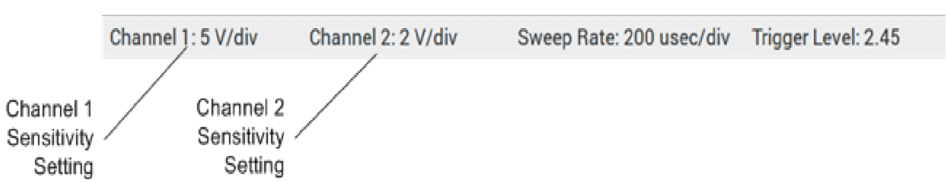


Figure 7

Let’s look at some examples to better understand the effects of the sensitivity adjustment. Here is a 1Khz sine wave on channel 1 with an amplitude of 10 volts peak to peak. With the sensitivity set to 5 V/div, the waveform occupies two vertical divisions:

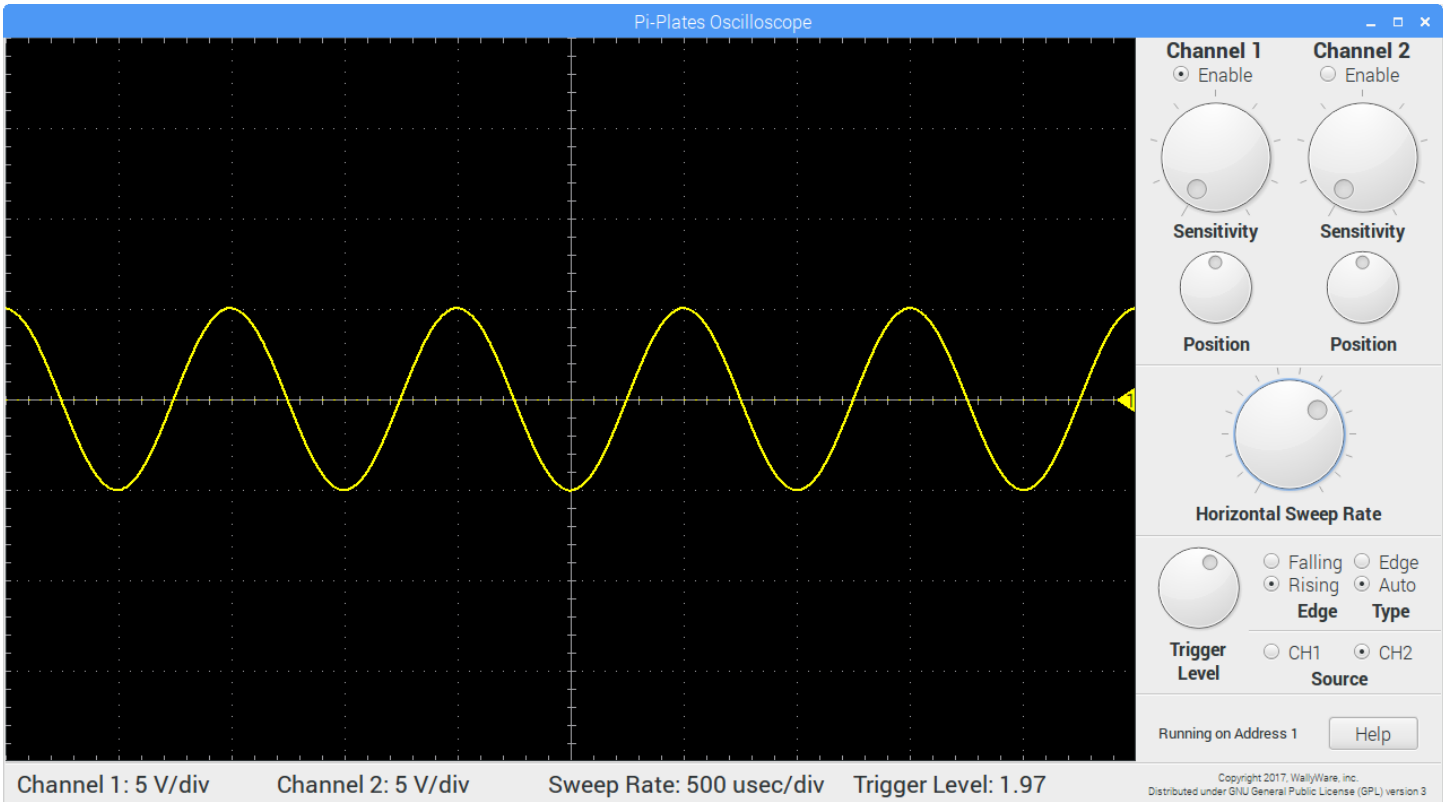


Figure 8

Note the Channel 1 sensitivity setting of 5 V/div in the information Area. Now lets see what setting the sensitivity to 2 V/div does to our waveform display:

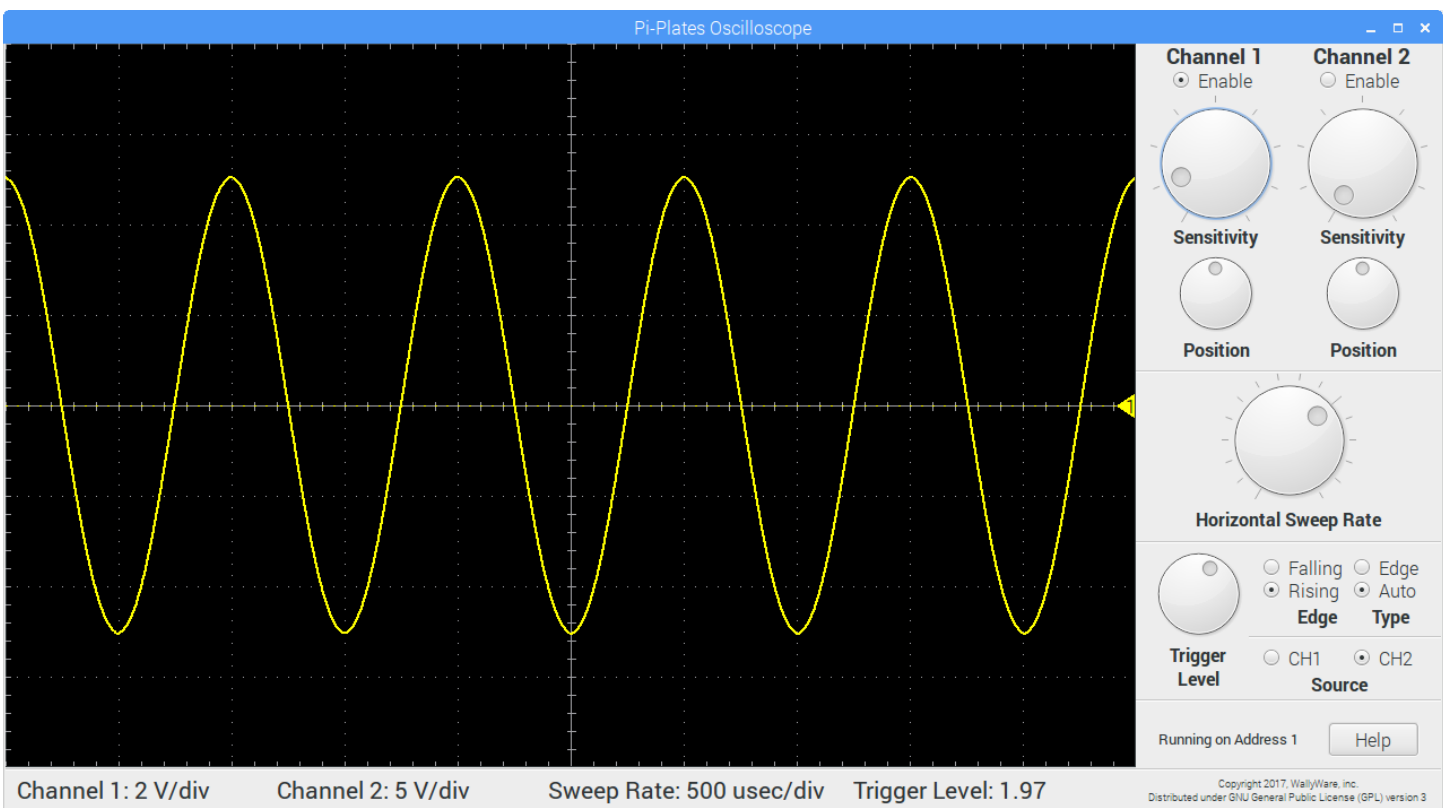


Figure 9

At 2 V/div, 10 volt peak-peak sine wave occupies five vertical divisions. Finally, we'll advance the sensitivity adjustment one more notch clockwise to 1 V/div:

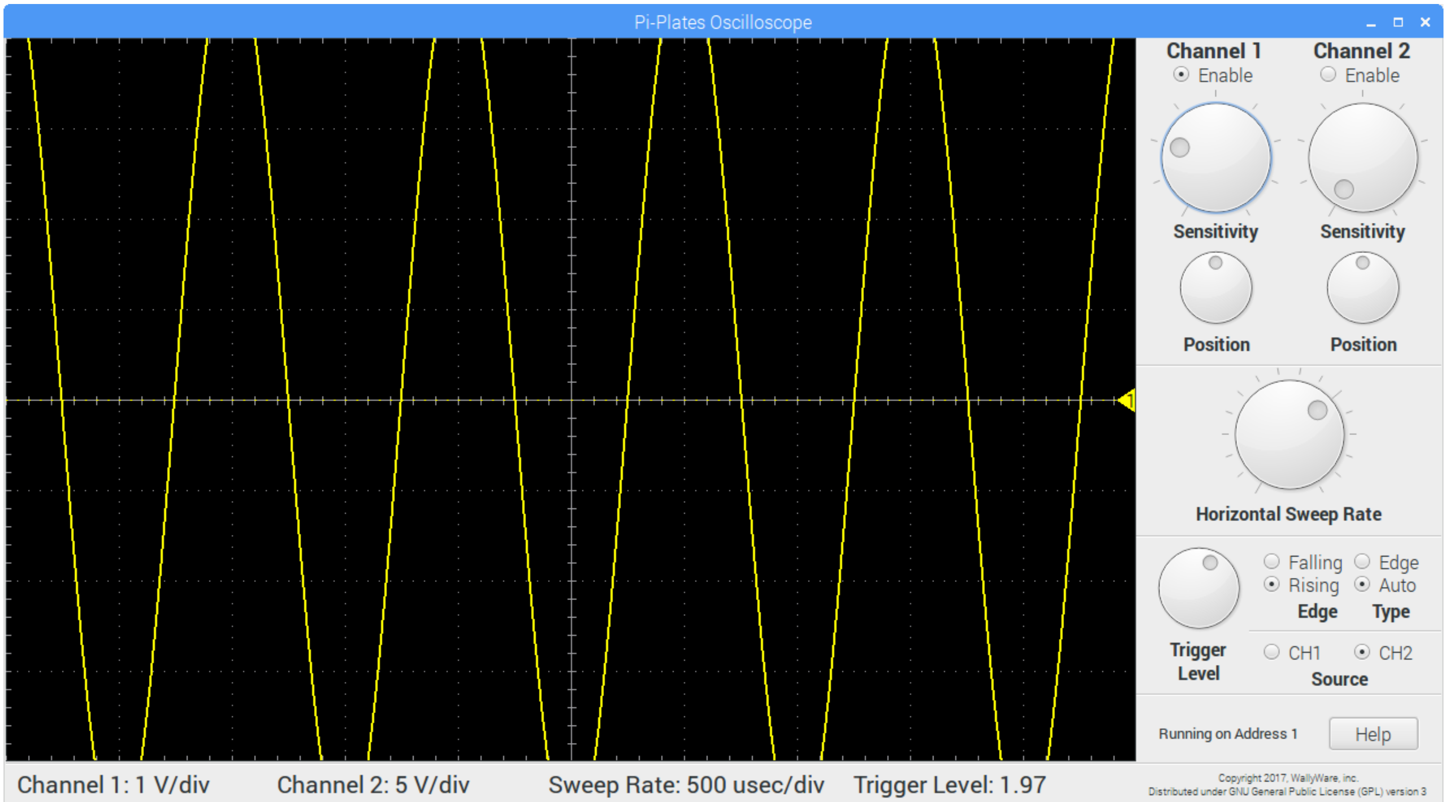


Figure 10

In the image above, our waveform has been clipped off at the top and bottom. Since the oscilloscope only has 8 vertical divisions, we cannot display a 10 volt peak to peak signal with a sensitivity of 1 V/div.

Position

Each channel has a small Position adjustment just under the Sensitivity knob:

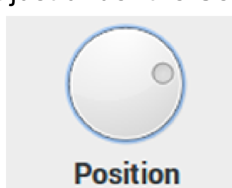


Figure 11

Rotating the Position knob will move the centerline of the selected channel up and down the screen. This is handy to use when you have both channels enabled and the waveforms are overlapping.

As an example, here is where our sine wave from the last example is located when the Position knob is in the center position:

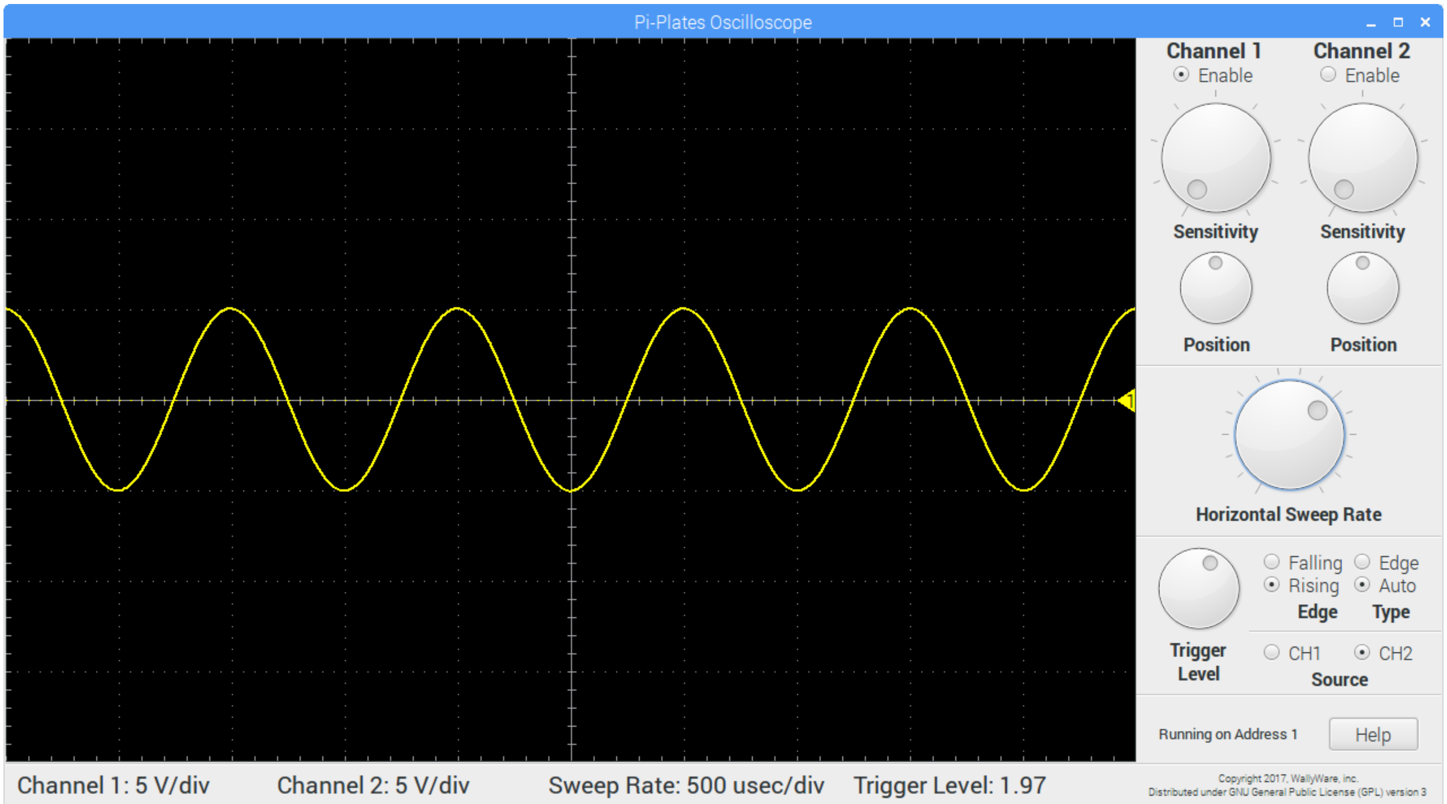


Figure 12

And, here is the location our sine wave when the knob is rotated 90 degrees clockwise:

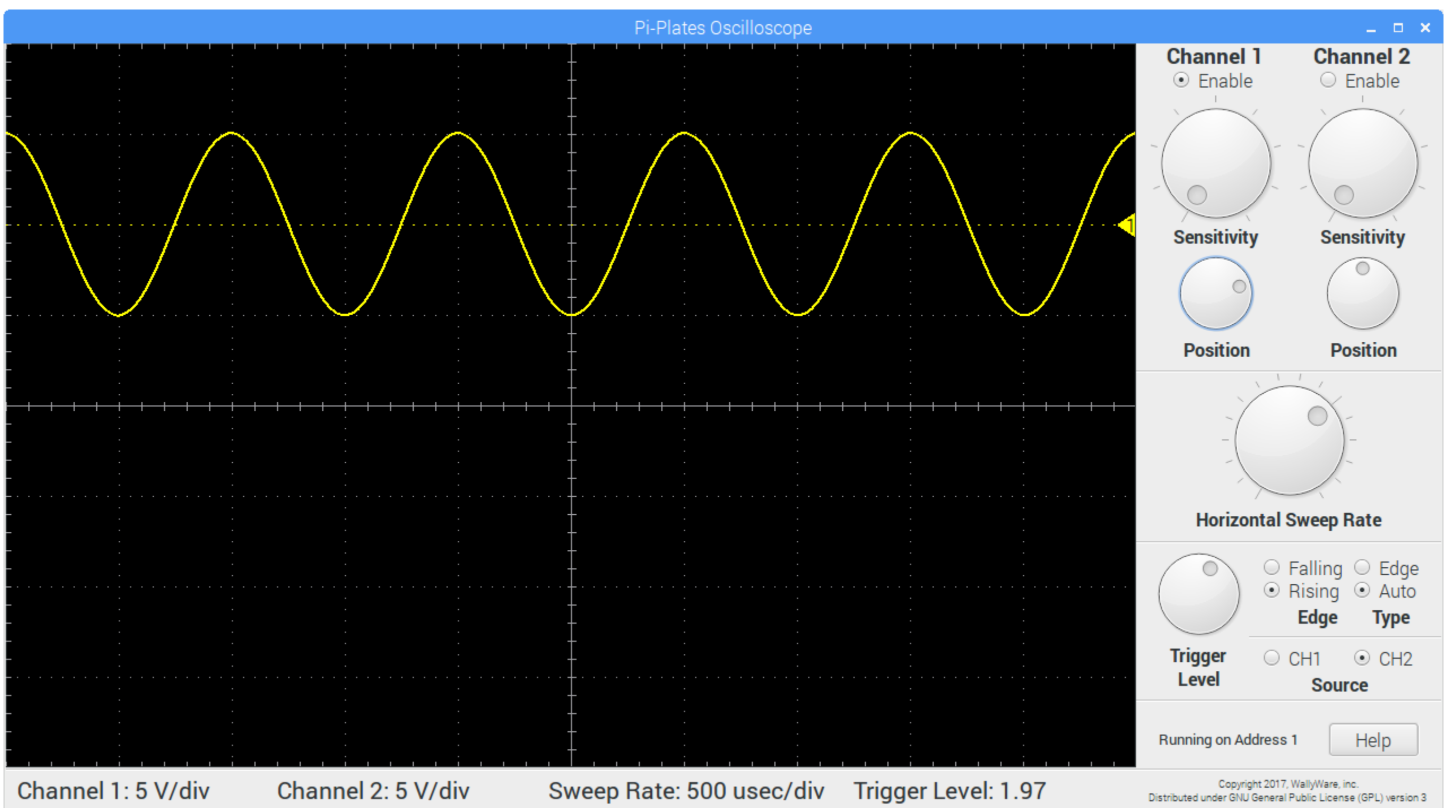


Figure 13

Remember, you can use the mouse for coarse adjustments and the direction keys for fine.

Horizontal Sweep

Recall from the introduction how oscilloscopes used to be based on CRTs and that the horizontal position of the dot being traced across the screen was controlled by an internal timebase. The rate at which the dot was swept across the screen was called “sweep rate.” We have adopted that terminology for our oscilloscope by labelling our horizontal controller “Horizontal Sweep Rate.”

Figure 14

Control

The horizontal axis of the oscilloscope is broken up into 10 divisions and has units of time. The terminology used for sweep rate is then seconds/division or sec/div. For our oscilloscope application, the current sweep rate is displayed in the information area below the display. Looking at the image below we see that the sweep rate is 1 msec/div or one millisecond / division. Since there are ten horizontal divisions then we can see that the displayed trace is 10 milliseconds long. And finally, we can see that there are ten cycles in our waveform. So we can calculate the frequency of our waveform as 10 cycles / 10 msec = 1000 cycles/second.

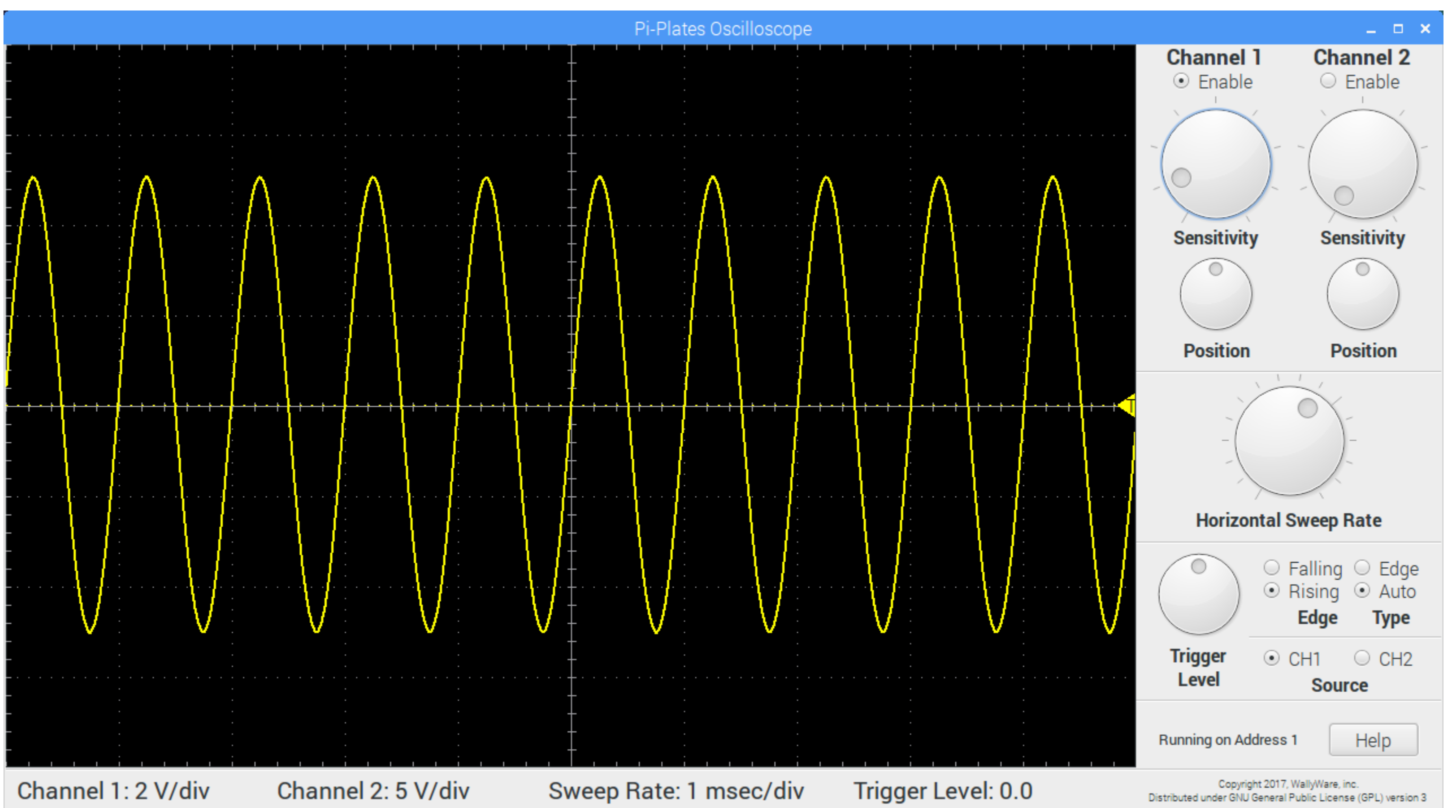


Figure 15

Rotating the Horizontal Sweep Rate knob one click clockwise increases the sweep rate to 500 usec/div (microseconds per division). Examination of the sine wave below shows that a full cycle covers two horizontal divisions. We can now calculate the frequency of our waveform a slightly different way:

one cycle / (2 X 500μsec) = 1000 cycles per second.

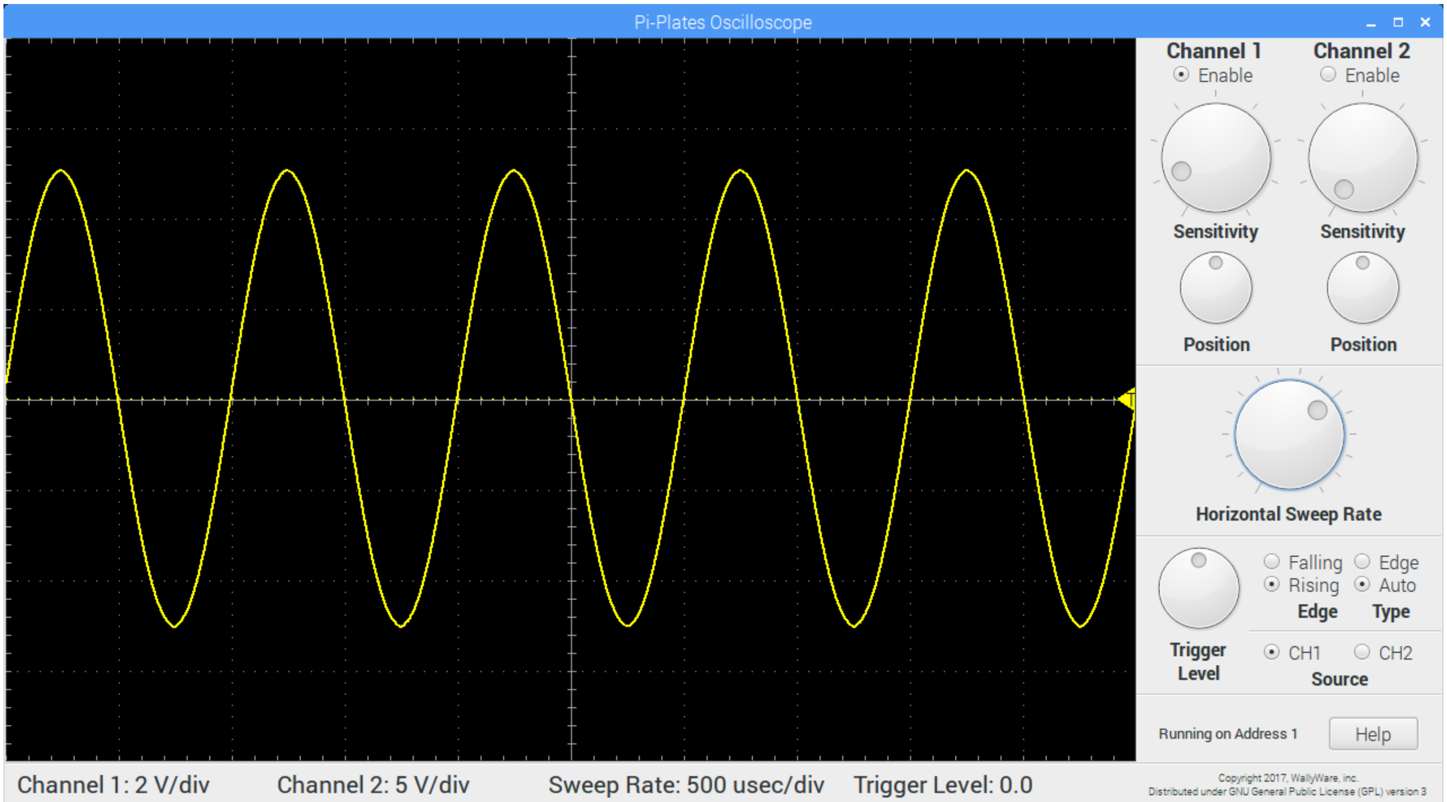


Figure 16

Once again rotating the Horizontal Sweep Rate knob one click clockwise increases the sweep rate to 200usec/div. Now we see that a single cycle of the sine wave covers five horizontal divisions. With a sweep rate of 200μsec/div, five divisions is equal to 1 millisecond. So we can now see that our waveform has a period of 1 millisecond. And since frequency is the inverse of period we can calculate the frequency of our waveform as:

Frequency = 1/period = 1/0.001 = 1000 cycles per second.

By controlling the sweep rate, it is possible to expand the waveform on the screen and examine it in more detail. It is also possible to better calculate the frequency and period with the increased resolution.

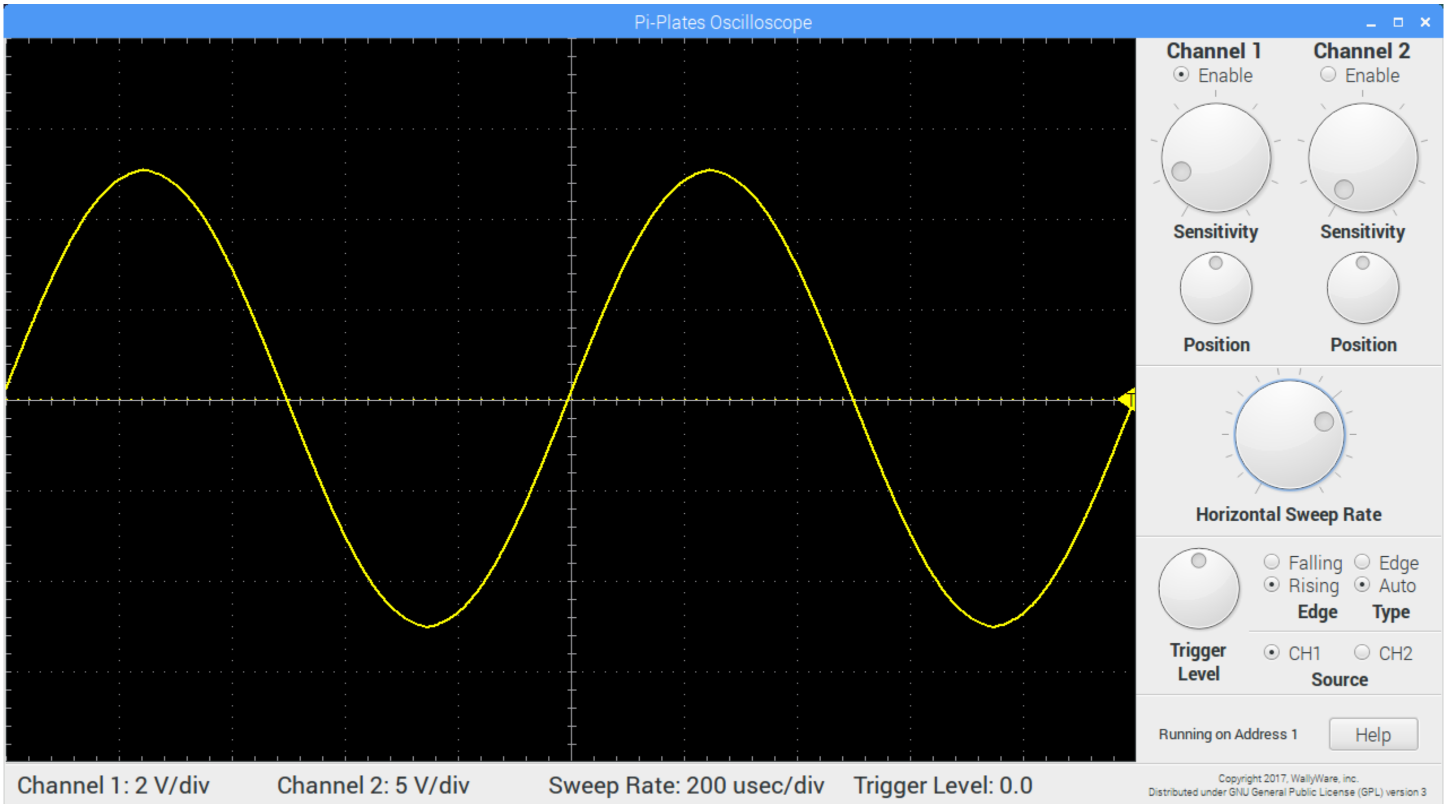


Figure 17

Aliasing

When using a digital oscilloscope, the user has to be aware of a potential phenomena called aliasing. Aliasing occurs when the sample rate of the oscilloscope is not fast enough to accurately capture the signal being measured. On the DAQC2plate, the sample rate is a function of the sweep rate. So, as you decrease the sweep rate, you are also decreasing the sample rate by a proportional amount.

An example of aliasing can be seen in Figure 18. With a 1 Khz signal on the input, we rotated the Horizontal Sweep Rate adjustment counterclockwise (CCW) to 200msec/div. Note that what appears to be a perfectly good signal is displayed. But then we can see that one cycle covers about 2.1 divisions. That means that the frequency of the displayed waveform is 1/(2.1*200msec)=2.38Hz and that doesn't make any sense.

So, to avoid aliasing, start with the highest sweep rate and rotate the Sweep Rate Adjustment knob CCW until you start to see one or two cycles of your input signal.

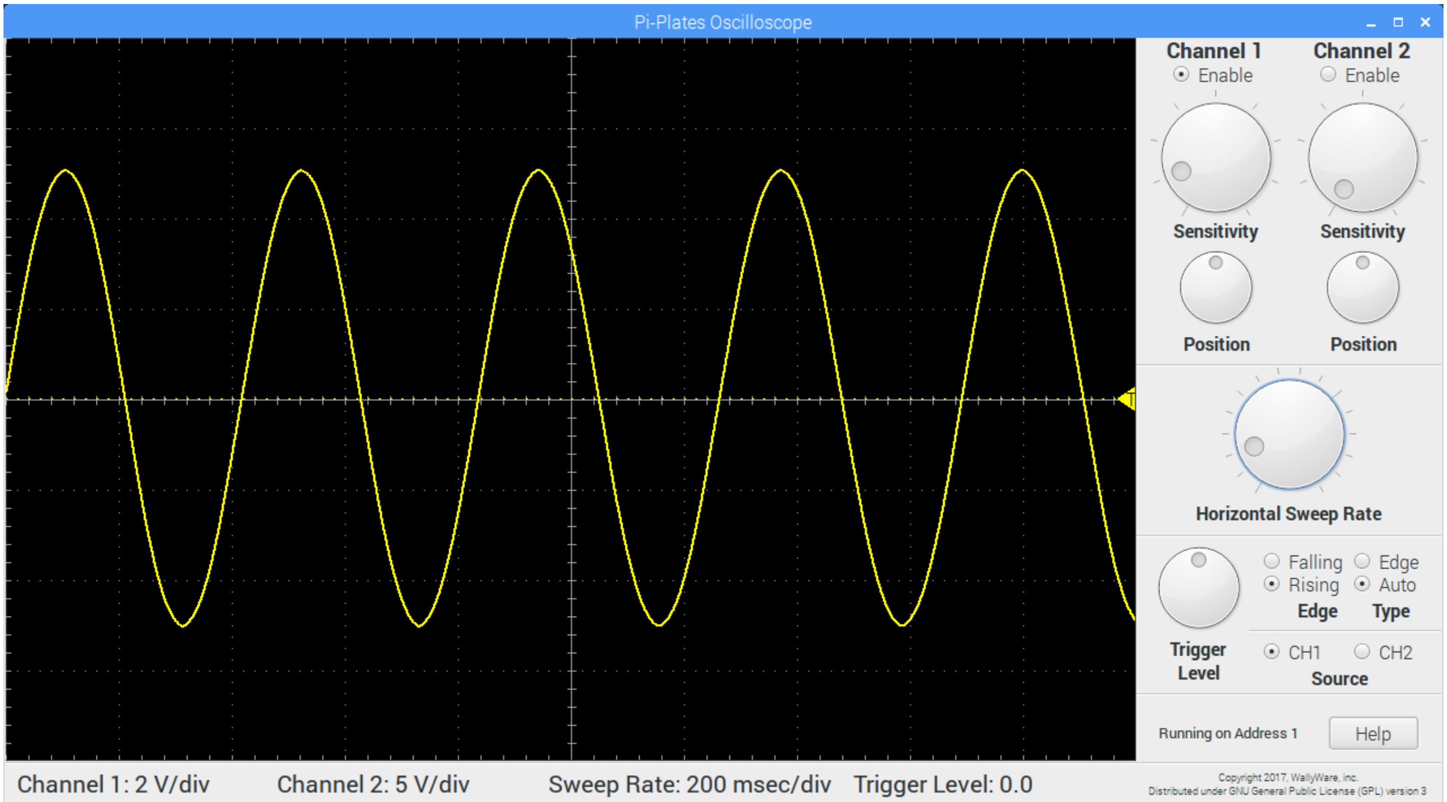


Figure 18

Trigger Functions

Trigger functions control when you start capturing data to display on the screen. This collection of controls is shown below:

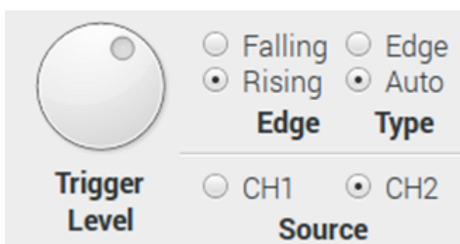


Figure 19

Source

Click on one of these buttons to select which of the input channels you want to use as your trigger source. Note that you can use a channel for trigger even if it is not enabled. In the screenshot below, we can see that Channel 2 is enabled and selected as the trigger source. Note the dashed line with the chevron labeled with a “T” on the right edge of the screen - this is the current trigger level.

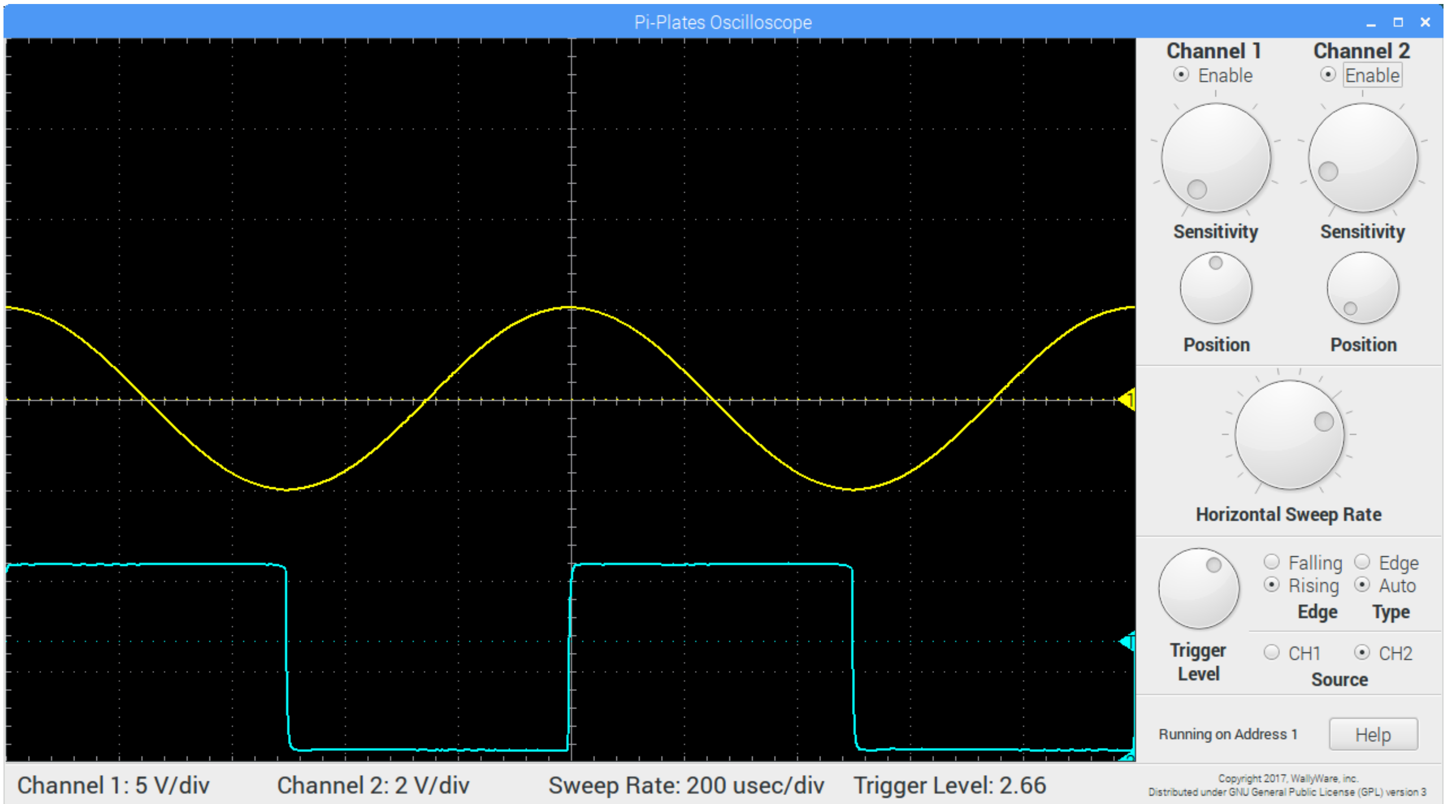


Figure 20

And this is what happens when channel 2 is disabled:

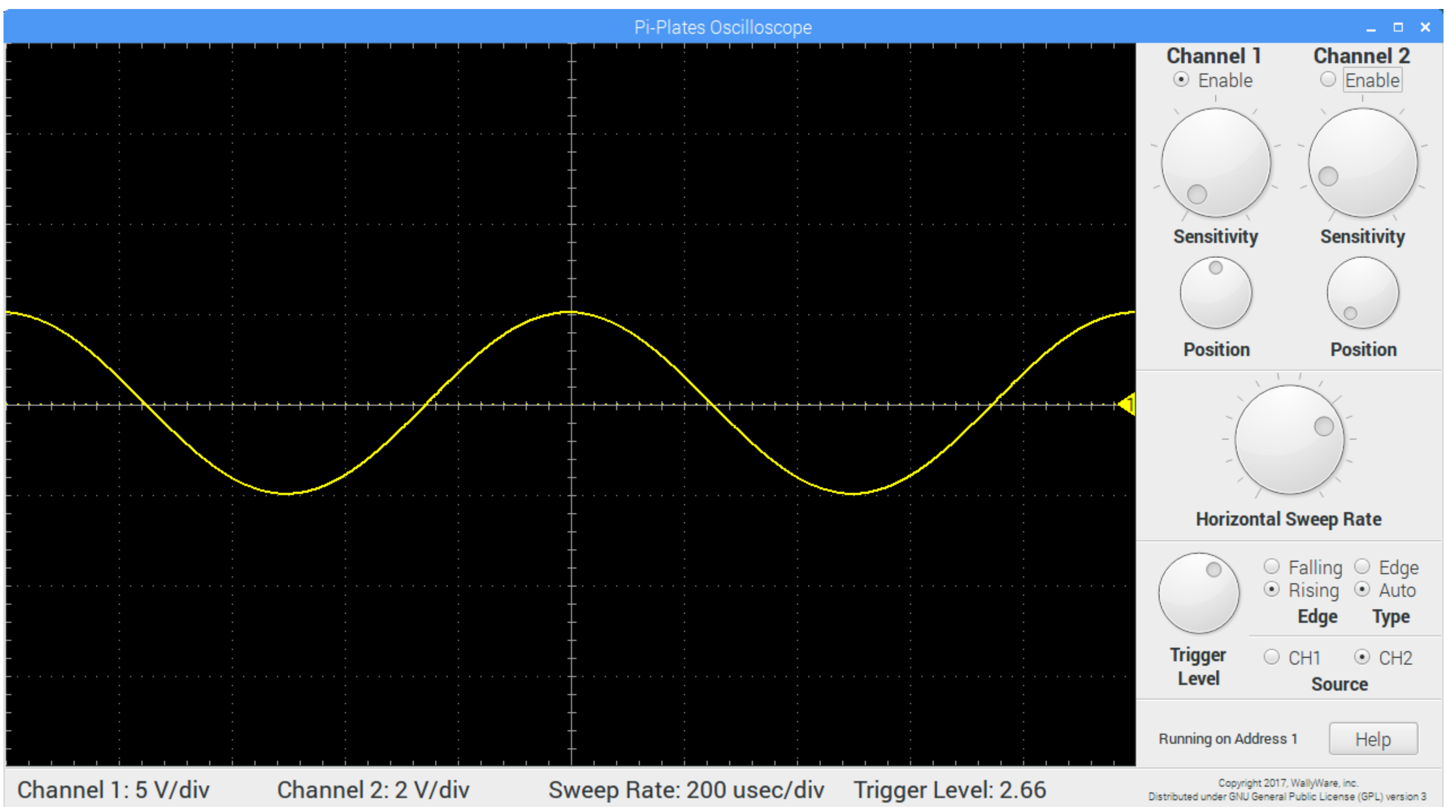


Figure 21

Channel 2 is no longer displayed but it is still driving the trigger.

Edge

Select whether you want to trigger on the rising edge or the falling edge of the trigger signal. In figure 20 we are triggering on the rising edge of channel 2. Here's what the traces look like when we select the falling edge:

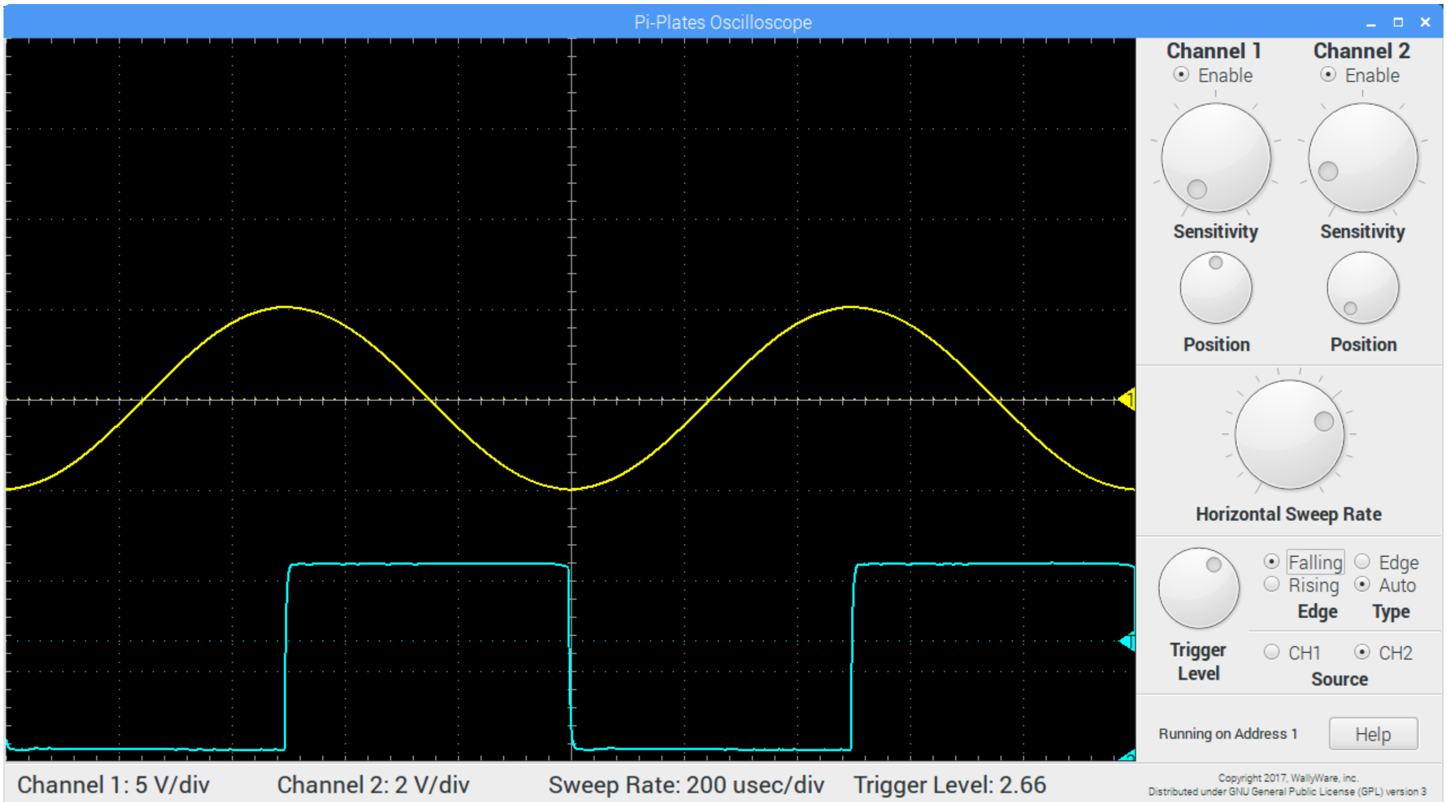


Figure 22

Note how the square wave on channel 2 starts on the falling edge and how the phase phase of both waveforms has shifted by 180 degrees.

Type

There are two types of trigger mode to choose:
Edge: this is the mode to use when you can see your trace and you want to fine tune the actual trigger point.
Auto: If you're not certain what your waveform is going to look like or if nothing is showing up on your screen, select Auto trigger and the scope will show all of the input waveforms after a brief delay. Note that if Auto is selected and the trigger level falls outside of the selected channel, you will likely see the waveform "roll" or show up in random horizontal locations.

Level

This adjustment controls the voltage level where the trigger occurs. It's best to explain with another example. Let's start with our 1Khz sine wave again set to a rising edge trigger on channel 1:

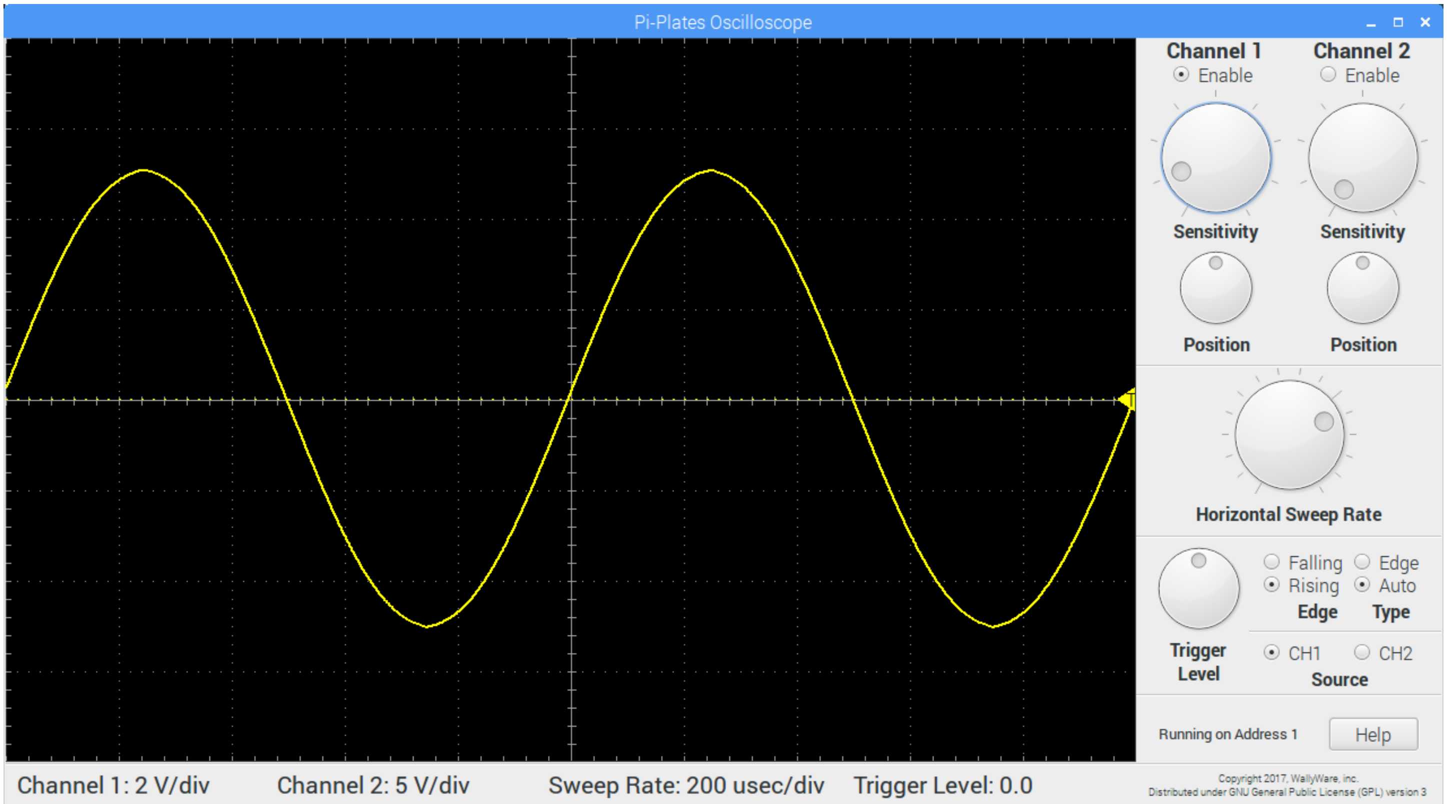


Figure 23

Note the yellow chevron with the letter "T" on the right of the screen. Also note the Trigger level is 0.0 in the information area at the bottom. Now we're going to raise the trigger level to 4.0 volts:

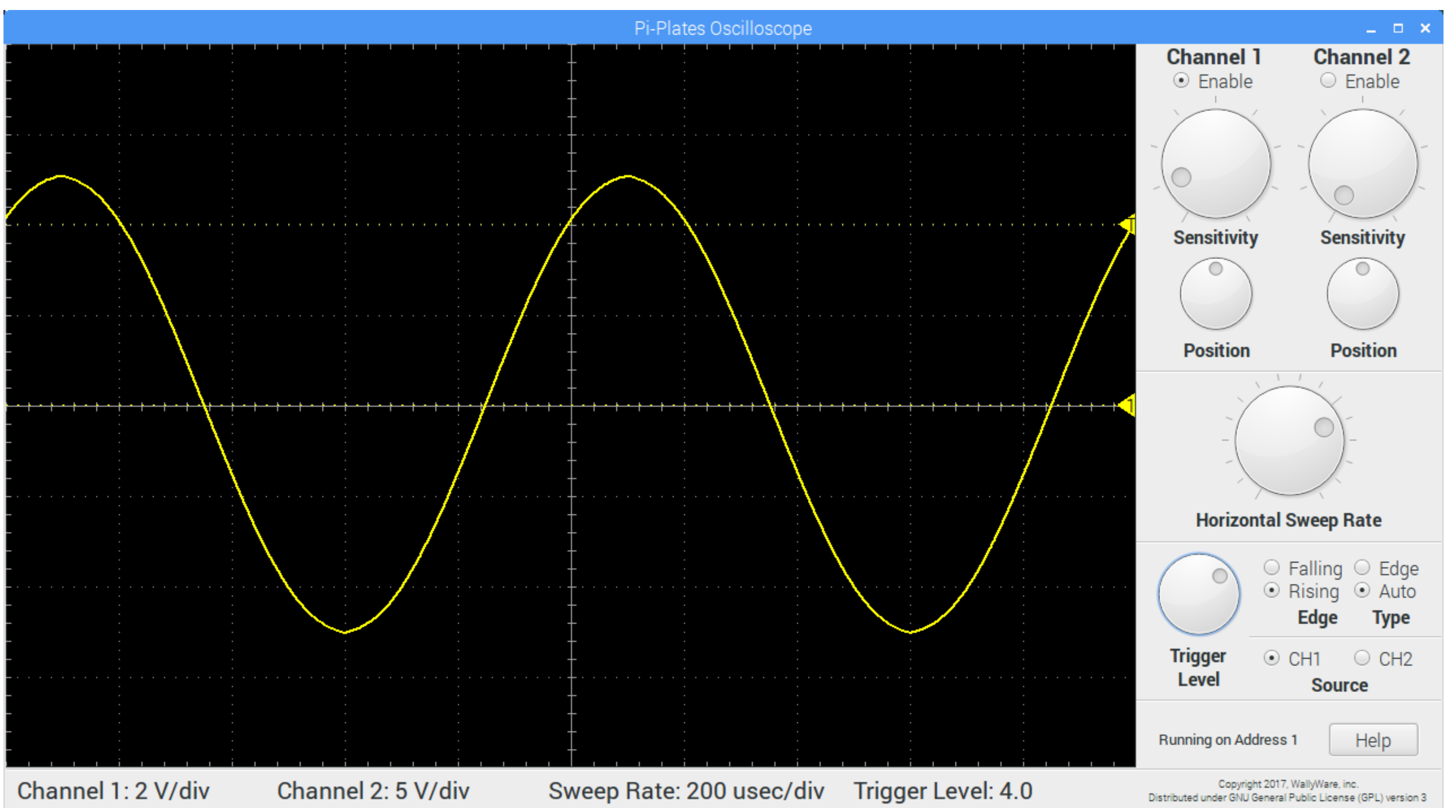


Figure 24

Note the chevron with the “T” is now two divisions above the reference line of the trace. This makes sense because our sensitivity is 2 volts/div. Also take note that the Trigger Level indicates 4.0 volts in the information are. Finally, look at the trace and observe that it begins as the riding edge crosses 4.0 volts.

Here’s a final example with the trigger set to -4.0 volts:

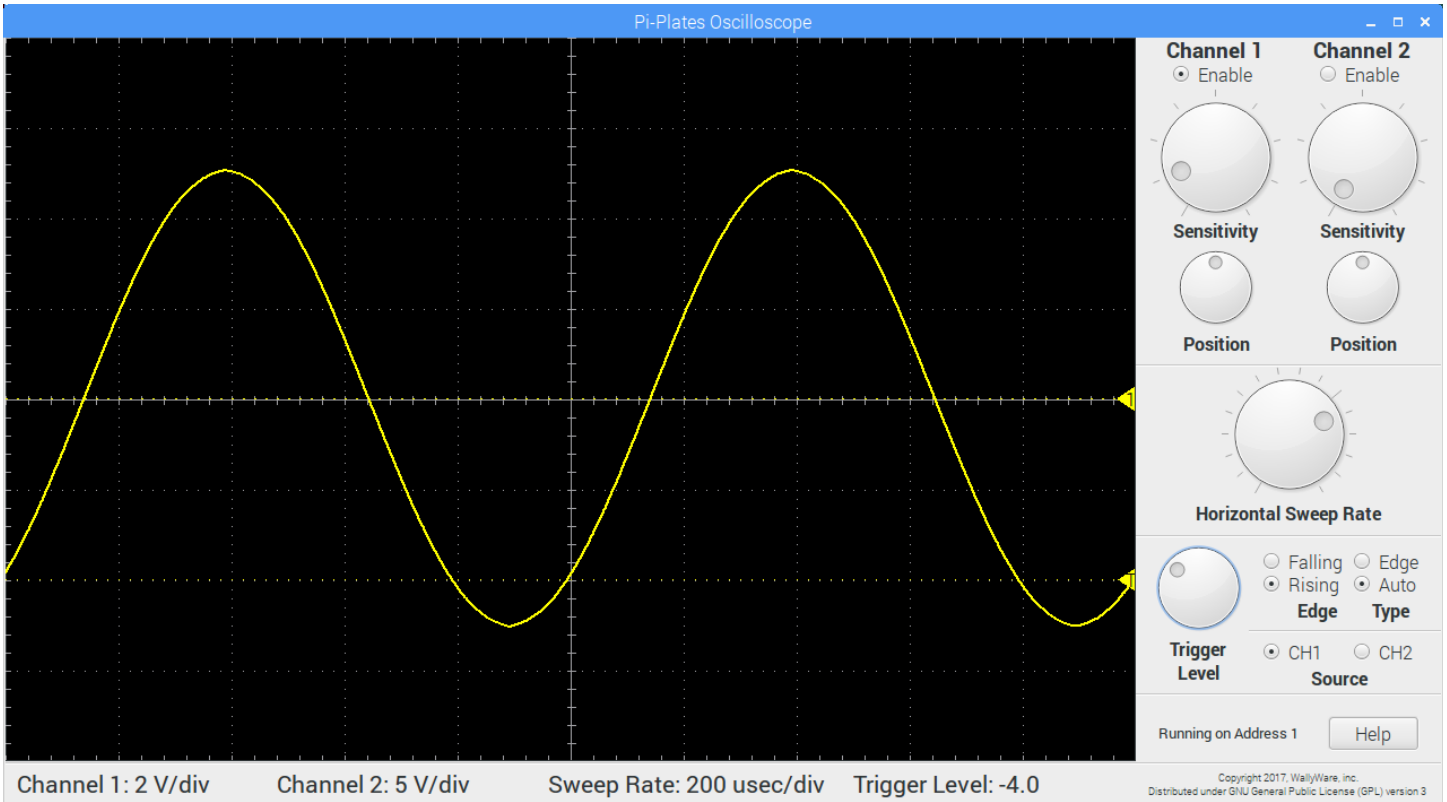


Figure 24

Revision History

Revision	Description
1.0	Initial Release