Agilent Technologies

EducatorsCorner.com Experiments

An Introduction to Using the Agilent 54622D Digital Oscilloscope, E3631A DC Power Supply, 34401A Digital Multimeter, and 33220A Arbitrary Waveform Generator

By:

Walter Banzhaf
University of Hartford
Ward College of Technology
USA

Equipment Required

- Agilent 54622D Mixed-Signal Oscilloscope with two 10X attenuating probes and digital probe kit
- Agilent E3631A DC Power Supply
- Agilent 34401A Digital Multimeter with two test leads (one red, one black)
- Agilent 33220A Function/Arbitrary Waveform Generator

Introduction:

This is a basic introduction to four pieces of equipment listed above; be sure to consult the Agilent manuals for each piece of equipment for information on safety, operational details, specifications and other information.

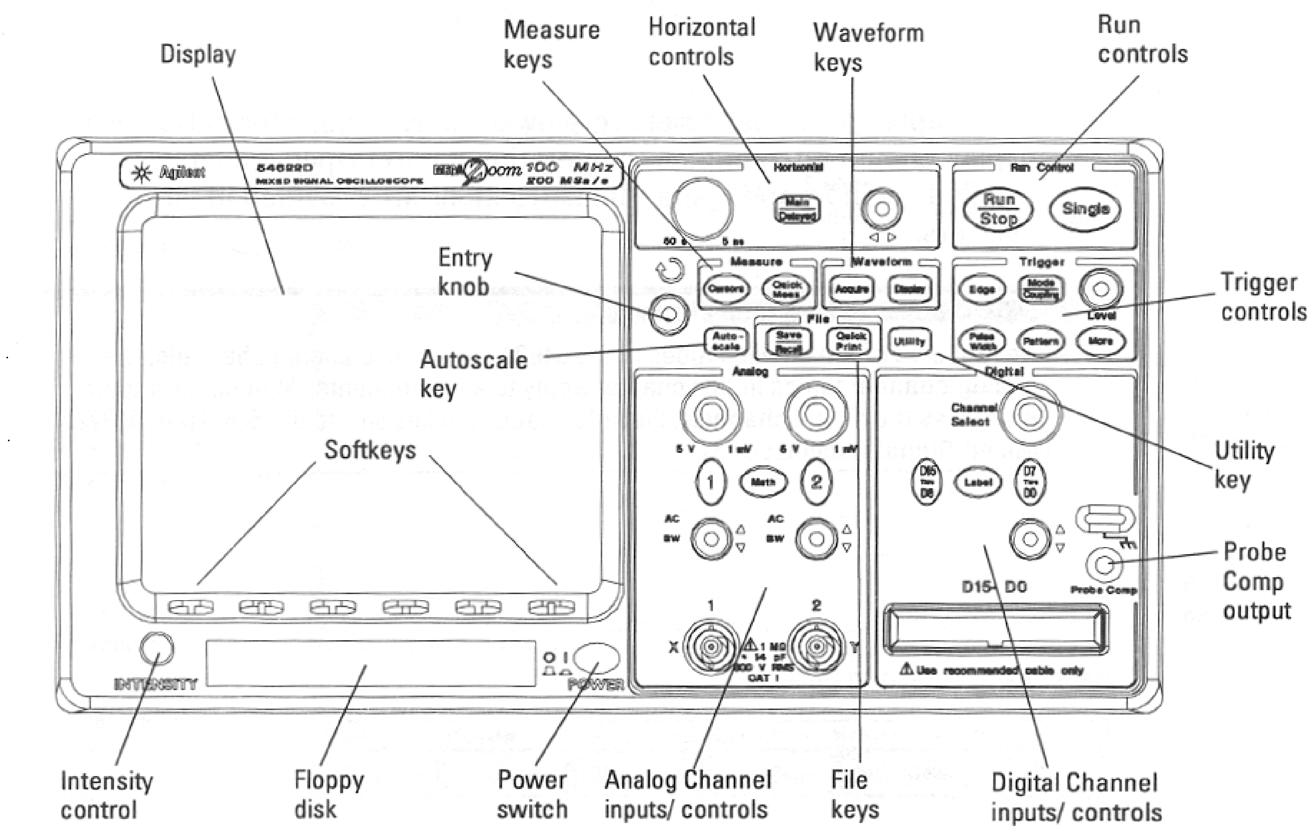
Basic electronic laboratory equipment allows us to supply DC power to a circuit (using the DC power supply, if the equipment doesn't have its own power supply), measure voltages, currents and resistances in the circuit (with the digital multimeter), apply time-varying signals (e.g. sine waves and square waves) to the circuit (with the function/arbitrary waveform generator), and observe displays of voltage versus time (using the oscilloscope). The mixed-signal oscilloscope allows us to look at analog signals and up to 16 digital signals simultaneously.

This experiment is based on the 90/10 philosophy: 90% of the time, one uses 10% of the features of a product, be it a computer, software (such as a word processor), a car, or electronic test and measurement equipment. After completing this experiment you will have a solid understanding of how to make use of these four basic laboratory instruments.

Page	Procedure

Part One – The Agilent 54622D Mixed-Signal Oscilloscope- Analog Inputs

The oscilloscope is the most versatile measurement tool you have available in the laboratory. What follows will help you to learn about making initial adjustments to your oscilloscope (and its probes), to get it ready for viewing signals. You will also learn to use some of the very useful features of this instrument. While you may think this is a very complex and sophisticated instrument (it is), learning how to use it is not that difficult. Most features are intuitive, every button provides and immediate "***help***" screen, and you'll soon be very comfortable using it.



As you can see above, the front panel is logically laid out, making it easy to find what you need: vertical input **Analog** controls, vertical input **Digital** controls, horizontal controls, trigger controls, measure keys, waveform keys, file key and ***softkeys*** (their functions change, depending on what you are trying to do).

For example, if you press the analog channel **1** button, the softkeys display functions that will change things about analog channel 1. But, if you press the **Edge** button (in the Trigger section), the softkeys offer things you can select about triggering.

Basic Start-Up Procedure: *(Refer to the picture of the oscilloscope front panel as you go through this procedure).*

1) Turn the oscilloscope ON by pressing the white button at the lower right corner of the CRT screen (the ***graticule*** of the cathode-ray tube). You should see the "*Startup Menu*", which includes ***Softkeys*** **(buttons directly under the CRT screen**) labeled Getting Started, Using Quick Help, About Oscilloscope, and Language. Try them all.

2) You probably weren't the last person to use the oscilloscope, and it's probably NOT set up to do the measurements you want to make. A good way to start is to return the oscilloscope to its "Default" condition by pressing the **Save/Recall** ***Hardkey*** in the **File** area of the front panel, then pressing the **Default Setup** softkey. Do this now (see Agilent's information below).

   You will now see a horizontal line in the middle of the CRT graticule, and at the top right is a blinking "**Level**". This area of the display is telling you that the oscilloscope is configured to trigger its sweep on **Channel 1**, using **positive edge** triggering in **Auto Level** mode, and it's NOT finding a signal at the trigger **Level** of **0.00 V**. This makes sense, as there is no input signal at this time.

To apply the default factory configuration

To set the instrument to the factory-default configuration, press the **Save/Recall** key, then press the **Default Setup** softkey.

The default configuration returns the oscilloscope to its default settings. This places the oscilloscope in a known operating condition. The major default settings are:

Horizontal main mode, 100 us/div scale, 0 s delay, center time reference

Vertical (Analog) Channel 1 on, 5 V/div scale, dc coupling, 0 V position, probe factor to 1.0 if an AutoProbe probe is not connected to the channel

Trigger Edge trigger, Auto level sweep mode, 0 V level, channel 1 source, dc coupling, rising edge slope, 60 ns holdoff time

Display Vectors on, 20% grid intensity, infinite persistence off

Other Acquire mode normal, Run/Stop to Run, cursor measurements off

IMPORTANT: The Default Setup does **NOT** change the how waveforms are saved to a floppy disk. You should be sure to check that your oscilloscope file format for saving waveforms is **TIF** (tagged image file). This can be selected by using the **Utility** hardkey, then the **Print Confg** softkey followed by the **Format** softkey where **TIF** is selected. A saved waveform (the first one will be called **PRINT_00.TIF**, then **PRINT_01.TIF**, etc.) can easily be imported into a Word document as a picture.

Another graphic file **Format** choice is **BMP** (bitmap). While this is also easily imported into Word, it takes twice as long to write the file (e.g. **PRINT_01.BMP**) to the diskette, and the file is much bigger (about 6 times bigger!). The last choice for file format is **CSV**; this is *Comma-Separated Value* format. It is not a graphic file, but is suitable for importing into a spreadsheet program.

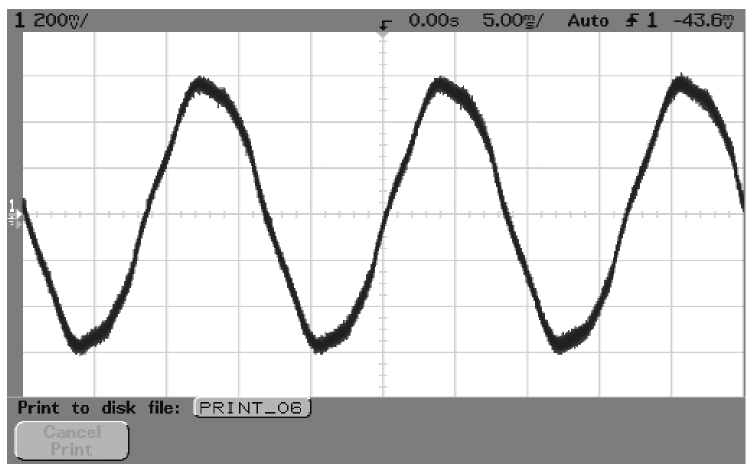
How to get help: Pressing **and holding** ANY key (hardkey or softkey) will bring up a help screen on the display. What a handy way to learn about its features!

3) **QUICK PERFORMANCE CHECK:**
Connect a 10X attenuating probe to Channel 1. Change three (3) settings as follows:

a) **Vertical Section:** Channel 1, change **Probe Factor** (a softkey that says **Probe**) to **10:1**. *You can do this by pressing the oval button labeled "1", and then turning the control called the "**Entry Knob**", located just to the right of the CRT, below the horizontal section, with an illuminated curved arrow above it.*

b) **Vertical Section:** Channel 1**,** change to **500 mV/div** scale. *You can do this by pressing the oval button labeled "1", and then turning the control right above the oval button and observing the vertical scale (at the top left side of the CRT screen).*

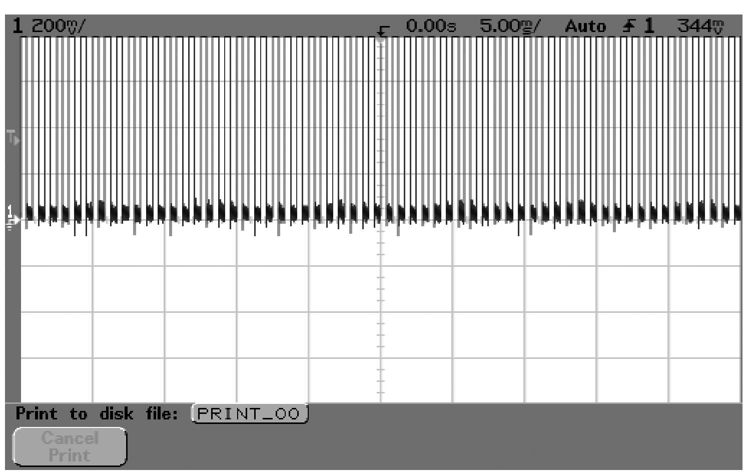
c) **Horizontal Section:** change time/division to **5 ms/div**, using the larger control.



Now, touch the probe tip with your finger. If you see a few cycles of a sine wave with several cm of vertical deflection (like the display to the right), both your oscilloscope and your probe are functioning. *Note: you may need to use **200 mV/div** or **1V/div**, depending on where you are.*

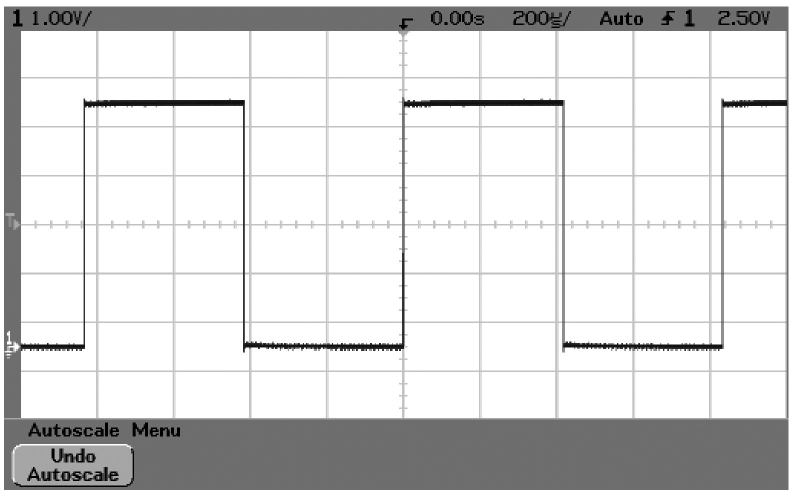
It turns out the "signal" you are observing is 60 Hz. power line noise, and your finger (which is most likely connected to your body, a crude antenna) is providing a very convenient "test" signal picked up from the power wiring in the building.

4) Connect the Channel 1probe tip to the **Probe Comp** output (near the right edge of the front panel). The display should look like the display below: not very satisfactory, but you can see there's some kind of waveform.



5) Three controls need to be adjusted to give a satisfactory display: the Vertical Section **volts/div** and **position**, and the Horizontal Section **time/div**. These controls can be adjusted two ways: by **YOU**, or by the oscilloscope.

For now, let's have the oscilloscope do the adjustment. Press the **Autoscale** hardkey, a white button just above the Vertical Section. You should see a display very similar to the one below with about 2_ cycles of the squarewave (*although the corners may not look quite so square as they do below)*.



Note that a softkey called **Undo Autoscale** has appeared at the lower left of the display. This can be a ***very*** valuable and time-saving button to use, under certain circumstances. Try "undoing autoscale" by pressing this softkey button. Then, press the **Autoscale** hardkey again, and you should see about 2_ cycles of the squarewave.

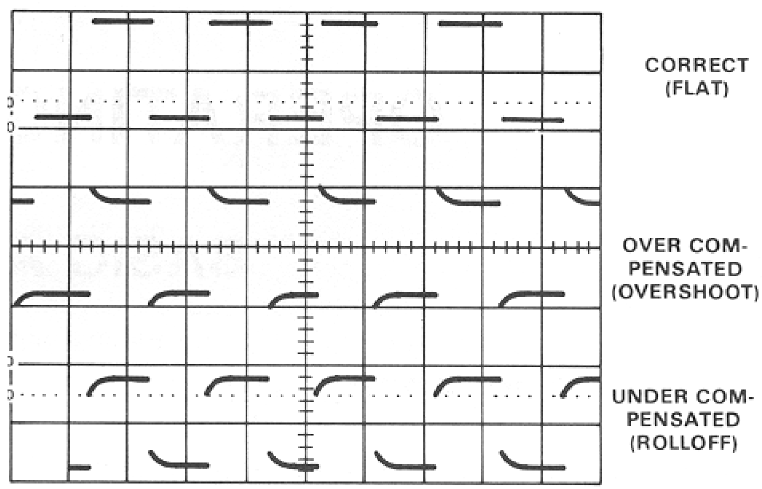
The next step (number 6, Probe Compensation) is so important it has an entire page devoted to it.

6) **PROBE COMPENSATION:** This step is a "***MUST DO***" procedure, ***each and every time you turn on an oscilloscope***. In fact, you must do it if you move a probe from one channel to another, and especially from one 'scope to another 'scope (for example, from the analog to the digital 'scope).

If you don't make sure that each probe is compensated properly, you can wind up with incorrect waveforms and major measurement errors. The error can be seen immediately with pulse waveforms (like the squarewave shown below), but does NOT show up at all for sinusoidal waveforms (AC signals). **All measurements of AC signal amplitude and phase can have big errors if compensation is not done!**

In fact, performing probe compensation is so important that oscilloscope manufacturers put a "Probe Adjust" or "Probe Comp(ensation)" output voltage on the front panel of most oscilloscopes.

PROBE COMPENSATION PROCEDURE:



a. Turn on the oscilloscope.

b. Connect a 10X probe to the Vertical Channel 1 BNC input connector, and connect the probe tip to the **Probe Comp** output on the lower right side of the front panel.

c. Press the **Save/Recall** key, then press the **Default Setup** softkey under the display.

d. Press the **Autoscale** key on the front panel. You should now see a square wave, with 5 divisions peak-to-peak vertically.

e. Check the waveform presentation for overshoot and rolloff (see Figure above). If necessary, adjust the **probe compensation screw** on the probe assembly for flat tops on the waveforms.

f. Disconnect the probe connected to Channel 1, and set it aside. Connect the other 10X probe to the Vertical Channel 2 BNC input connector.

g. Press the **Autoscale** key on the front panel. You should now see a square wave, with 5 divisions peak-to-peak vertically.

h. Check the waveform presentation for overshoot and rolloff (see Figure above). If necessary, adjust the **probe compensation screw** for flat tops on the waveforms.

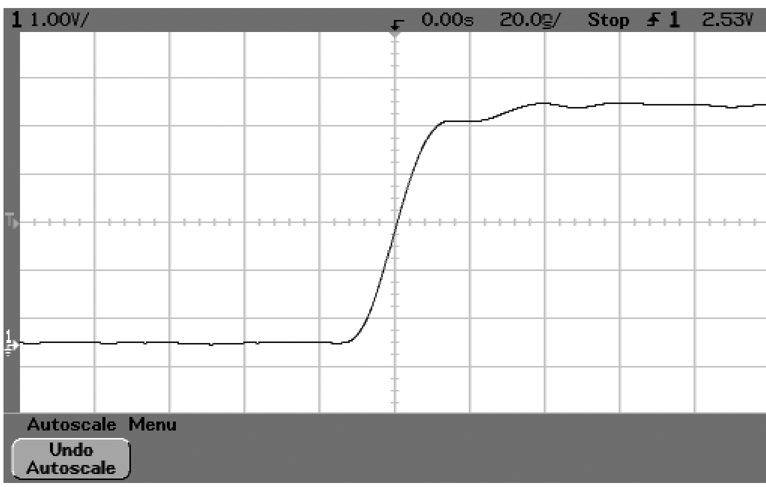
Make sure that both probes, for Channel 1 and Channel 2, are properly compensated before proceeding.

Features to Try - Vectors:
The 54622D oscilloscope is a digital instrument that makes a graph of **voltage versus time** by putting individual dots on the screen. By contrast, an analog 'scope creates a graph which is continuous (no dots are used).

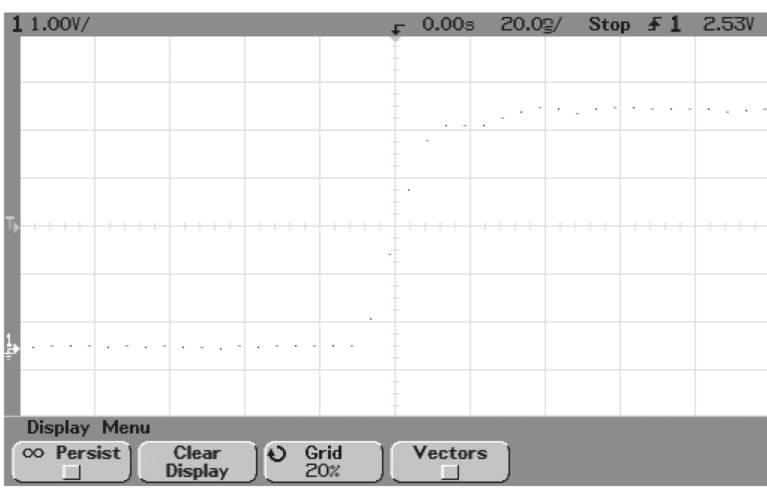
Each dot represents an (x,y) pair (time, voltage) in a Cartesian coordinate system. If the dots are close together, your eyes are fooled and you see a continuous line; if the dots are too far apart, you'll see individual dots. To overcome the problem of seeing dots, a **Vectors** mode can be used which "connects the dots". This can be seen very clearly as follows.

YOU DO IT:

1) Press the **Run/Stop** button, so that it changes from **green** (Run) to **red** (stop). The waveform displayed, and its dots, are now frozen in memory.
2) Change the Time/Div from 200 us/div to 20 ns/div. You should see the display below (Vectors turned ON).
3) Now press the **Display** hardkey (in the **Waveform** section), and press the **Vectors** softkey. Now the display (Vectors turned OFF) should appear. The individual dots are very apparent, and this is not as "pleasing" a display. Normally, **Vectors** can be left **ON**.



Vectors turned ON



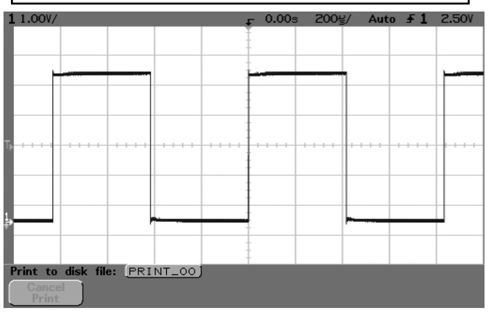
Vectors turned OFF

Features to Try - Averaging:
Our building, and most laboratories, are "dirty" in the electromagnetic sense. This is due to the power line noise, computer systems, lighting, and RF (radio-frequency) signals that create unwanted voltages on top of the voltages we do want to measure.

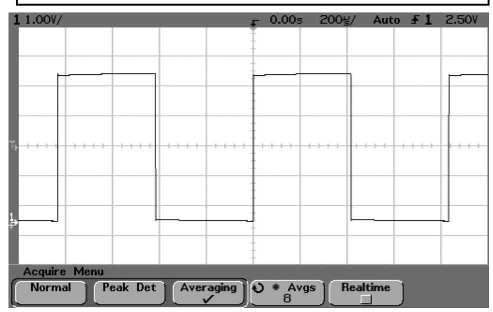
One way to minimize the problem (clean up the "dirt") is to use the **Averaging** feature of a digital 'scope. Look closely at the waveform below, on the left. It is the Probe Comp squarewave, and shows a "fat" line for the upper and lower parts of the square wave. This is caused by the noise voltage superimposed on top of the signal voltage.

In the waveform below on the right, **Averaging** has been turned on, using 8 averages. Note that the tops and bottoms of the square wave are much "thinner" (i.e. less noise is present), because the noise has been averaged out of the display.

Probe Comp Signal, No Averaging



Probe Comp Signal, With Averaging



YOU DO IT: Now try it yourself, using the probe compensation signal on the front panel.

1) Get back to the basic factory default settings, use **Autoscale** and look at the Probe Comp waveform. You should see the display shown above on the left (averaging is OFF now).

2) Press the **Acquire** hardkey in the **Waveform** section of the front panel, and turn on averaging by pressing the **Averaging** softkey. You should now see the display shown above on the right (averaging is ON).

Note that the number of averages (# of Avgs) can be changed, as needed, over a wide range of values (the default number is 8). This feature ***may*** create problems. **To see the effect of a large number of averages, do the following:**

3) Turn averaging OFF (by pressing the NORMAL softkey). Remove the probe tip from the Probe Comp output, then reconnect it. Note the nearly instantaneous change in the display (from squarewave to flat line).

4) Now turn averaging ON and set the # of Avgs to 8. Remove the probe tip from the Probe Comp output, then reconnect it. Note that the change in the display (from squarewave to flat line) is rapid, but gradual (i.e. it is not instantaneous). Leave the probe disconnected.

5) Leave averaging ON and set the # of Avgs to 4,096. Connect the probe tip to the Probe Comp output. Note that the change in the display (from flat line to squarewave) is very slow indeed. **QUESTION TO PONDER:** If time base is set to 200 µs/div, and there are 10 divisions on the graticule, how long does it take before 4,096 complete "sweeps" have occurred? This is the time needed for the display, with squarewave removed, to become a flat line, ***if*** one sweep begins right after the prior one ends. **Turn averaging OFF (by pressing the NORMAL softkey) when done with this step.**

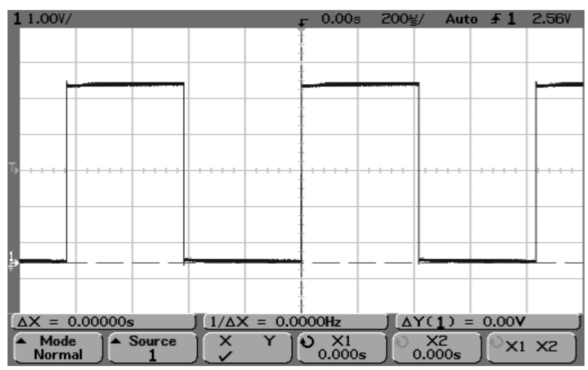
Features to Try - Using Cursors to Measure Time & Voltage:

In the "good old days", the user of an analog 'scope spent a lot of time measuring key parameters of a waveform by counting divisions. The number of ***horizontal divisions*** gave information about ***time*** (the period of a waveform, or pulse width, or time difference between two waveforms). The number of ***vertical divisions*** told the user how big the voltage was (peak-peak, maximum, minimum, average value, etc.).

Let's do some measurements by manually positioning the ***cursors*** on our Probe Comp waveform.

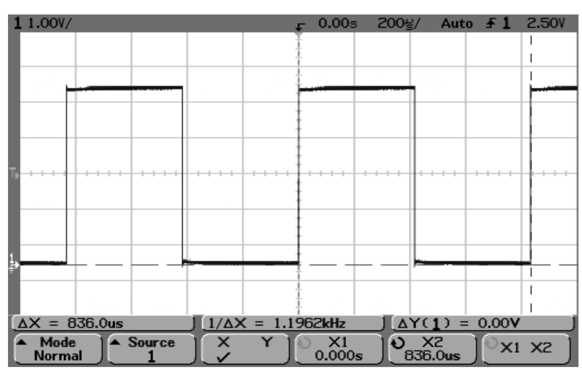
1) Connect Channel 1 probe to the Probe Comp output terminal, and press the **Autoscale** hardkey.
2) Press the **Cursors** hardkey, and you will see the waveform and softkey labels as shown below in the left picture:

In this picture note that the (X Y) softkey has **X** checked, and both the X1 and X2 cursors (dashed vertical lines) are at 0.000s (this is the horizontal center of the display).



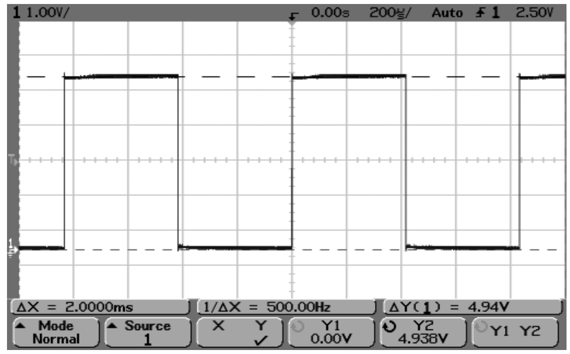
3) Press the X2 softkey, and turn the **Entry Knob** to move the X2 cursor to the rising edge nearest the right side of the display.

Now the X1 cursor is at 0.000s, the X2 cursor is at 836.0us, and ΔX = 836 – 0.00 = 836 us. Frequency = 1/ΔX = 1.1962 kHz.



4) Now press the (X Y) softkey, which will make the **Y** checked, and both the Y1 and Y2 cursors (dashed horizontal lines) are at 0.00V (this is the bottom of the waveform).

5) Press the Y2 softkey, and move the Y2 cursor line to the top of the waveform by using the **Entry Knob**. Now Y2 is about 4.94 V, and the ΔY = 4.94V. Your display should look like the one on the right.



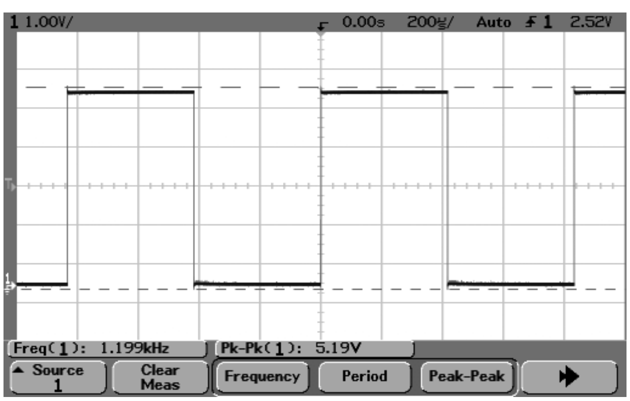
Features to Try - Using "Quick Meas" to Measure Time & Voltage:
As you just found, manually positioning the ***cursors*** on a waveform can give you important information about a waveform. Since these kinds of measurements need to be done all the time, your oscilloscope is designed to make many common measurements automatically. These include:

- Frequency, period and peak-peak voltage
- Maximum, minimum, rise time, fall time and duty cycle
- RMS (root-mean-square or effective value), +width, -width, average and amplitude
- Top, base, overshoot, preshoot, and X at Max
- Phase shift (Ch. 1 compared to Ch. 2), time delay (Ch. 1 compared to Ch. 2)

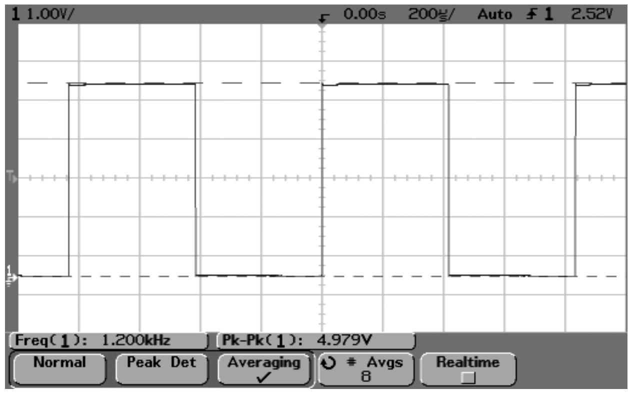
Let's do some measurements now using the **"Quick Meas"** feature.

1) Connect Channel 1 probe to the Probe Comp output terminal, and press the **Autoscale** hardkey.

2) Press the Press the **Quick Meas** hardkey. You will see the display below. The 'scope has determined that the frequency is 1.199 kHz, and the peak-peak amplitude is 5.19 Vpp.



What is wrong with the amplitude value of 5.19 Vpp is that it is a bit wrong (too big), due to the noise on the Probe Comp signal. You can see that the dashed horizontal cursor lines are positioned above and below where they should be (on the top and bottom of the waveform).



Get rid of the noise by pressing **Acquire** hardkey, and then the **Averaging** softkey (and average 8 waveforms). We can see that with averaging on, the amplitude is now 4.979 Vpp, 4 % lower than 5.19Vpp. The dashed cursor lines now lie right on the top and bottom of the waveform.

Features to Try – Saving Waveforms to a Floppy Disk:

The Default Setup does **NOT** change the how waveforms are saved to a floppy disk. You should be sure to check that your oscilloscope file format for saving waveforms is **TIF**. This can be selected by using the **Utility** hardkey, then the **Print Confg** softkey followed by the **Format** softkey where **TIF** is selected.

Another graphic file **Format** choice is **BMP** (bitmap). While this is also easily imported into Word, it takes twice as long to write the file (e.g. **PRINT_01.BMP**) to the diskette, and the file is much bigger (about 6 times bigger!). The last choice for file format is **CSV**; this is *Comma-Separated Value* format. It is not a graphic file, but is suitable for importing into a spreadsheet program.

1) Get a waveform on the display, perhaps by connecting Channel 1 probe to the Probe Comp output terminal, and press the **Autoscale** hardkey.

2) Use the **Quick Meas** hardkey to measure frequency, peak-peak voltage, and period. The first two happen automatically; you must press the **Period** softkey to measure period.

3) With a floppy disk in the 3.5-inch disk drive under the display, press the **Quick Print** hardkey. The displayed waveform, including the frequency, peak-peak voltage, and period, will be written to a file on the floppy disk. If it's the first waveform saved, the file will be called **PRINT_00.TIF**.

4) You can see that the file was saved successfully by pressing the **Utility** hardkey, then the **Floppy** softkey, and then the **File:** softkey. A list of the file(s) saved will appear on the display, with the date and time they were written. *This is a good opportunity to see if your 'scope's clock is set correctly; if it isn't, press the* **Utility** *hardkey, then the* **Options** *softkey and the* **Clock** *softkey and make changes to the year, month, day, hour and minute as needed.*

Importing Saved Waveform Displays into a Word document:

This can be done either in lab (if a PC is available) or at home.

1) Insert the floppy disk containing the stored waveform(s) into a PC.

2) Open a Word document (new or existing).

3) Click on **Insert** in the toolbar at the top of the screen, and move the cursor arrow down to **Picture**.

4) Choose the **From File** option, and Look In the **3_ Floppy (A:)** directory for **PRINT_00.TIF**

5) Click on **PRINT_00.TIF**, and then click on the **Insert** icon. Within a few seconds the saved display (waveform and data measured with **Quick Meas**) should appear in your document. You can change the size of the image a number of ways: clicking on the image and dragging a corner inward will make the image smaller.

HANDY HINTS:

Pressing **and holding** ANY key (hardkey or softkey) will bring up a help screen on the display. This is a great way to learn about oscilloscope features.

Softkeys are at the bottom of the display. They will appear when a **Hardkey** is pressed.

DON'T use the oscilloscope to delete files on your floppy disk, except files that IT created.

Autoscale may not give a desirable result; use **Undo Autoscale** if this happens.

When you use **Autoscale**, the 'scope looks for repetitive waveforms 10 mVpp or larger, a duty cycle greater than 0.5%, at no less than 50 Hz, and will turn off any channel not meeting those criteria. It will choose as its trigger source whichever of these has a valid signal on it, in the order listed: External Trigger, Channel 2, then Channel 1.

If you have signals on channels 1 and 2, when you use **Autoscale** the 'scope automatically chooses channel 2 as the trigger source. You may need to manually change the trigger source to channel 1.

Measurements made with **Quick Meas** may give incorrect results, particularly on noisy signals. Look *carefully* at the cursor lines to see if ***you*** agree that the cursor lines are showing what you want to measure. If your displayed signal is noisy for any reason, try using **Averaging** to clean it up. If the noise is too big to eliminate with **Averaging**, using manual cursors to make your measurements.

The **Entry Knob** (at the right side of the display) is used to change many settings, depending on which **Softkeys** are showing.

Be sure the **Probe** value (attenuation factor) for both channels is set correctly for your probes.

Every time you return the oscilloscope to **Default Setup**, the probe attenuation factor is set to 1.0:1. You must change it back to 10:1 (assuming you are using 10X probes).

You can move the trigger point, which is the center of the graticule by default, using the **Main/Delayed** hardkey and the **Time Ref** softkey.

The intensity of the grid on the graticule may be changed by pressing the **Display** hardkey and using the **Grid** softkey.

You can save a "setup", which stores the waveforms, cursors, math measurements, horizontal, vertical and trigger settings to a floppy disk or to an internal oscilloscope memory. When you recall that information, you can elect to restore the setup, or the waveform trace, or both.

Part Two – The Agilent 54622D Mixed-Signal Oscilloscope- Digital Inputs

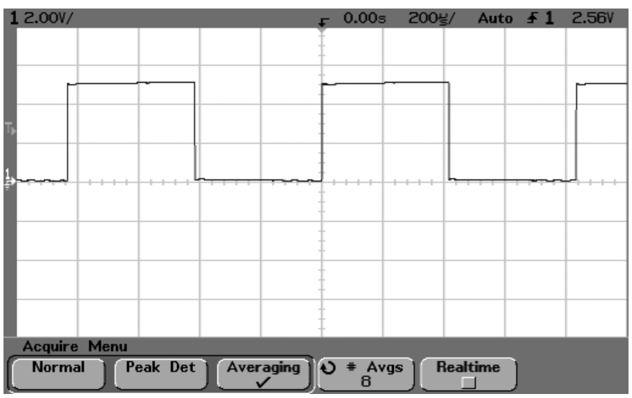
Features to Try - Turning On the Digital Channels

1) OK, now that we are familiar with the mixed-signal oscilloscope using its analog inputs, now let's get acquainted with the digital inputs and their controls.

2) First, return the oscilloscope to its "Default" condition by pressing the **Save/Recall *Hardkey*** in the **File** area of the front panel, then pressing the **Default Setup** softkey.

3) Next, connect a (compensated) 10X analog probe from channel 1 input to the Probe Compensator output (probe tip to "Probe Comp", probe ground the the ground connection immediately above.

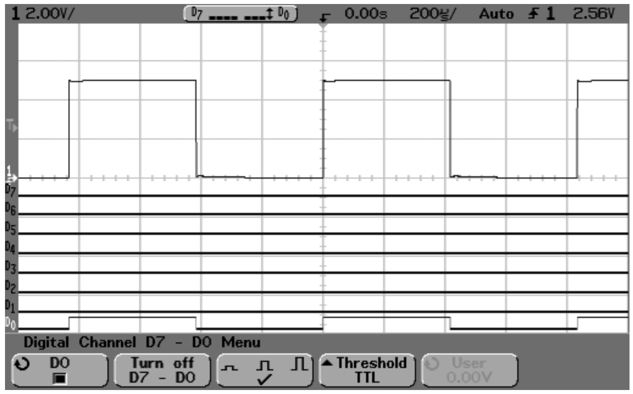
4) Press the **Autoscale** hardkey, and adjust the display so that it looks like the one below (note that **Averaging** is being used to remove noise from the waveform):



What we see above is a unipolar squarewave, going from 0 V to 5 V. This sounds like a digital signal, and we can use it to try out the digital input capability of the mixed-signal oscilloscope (using just one of the 16 digital input channels).

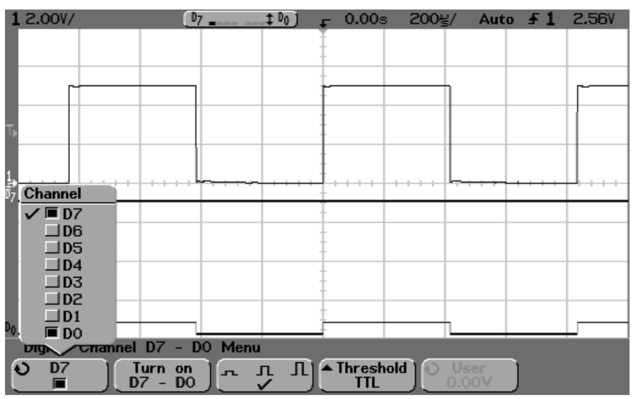
5) Connect the 16-channel cable (part of the digital probe kit) to the **D15-D0** input connector on the front of the oscilloscope.

6) Connect the **D0** lead to the "Probe Comp" output (where the 10X probe tip is still connected). Now press the **D7 Thru D0** hardkey in the digital section of the front panel, and you should see analog channel 1 on the top of the display, and eight digital channels (D0 - D7) on the bottom. See the display below:

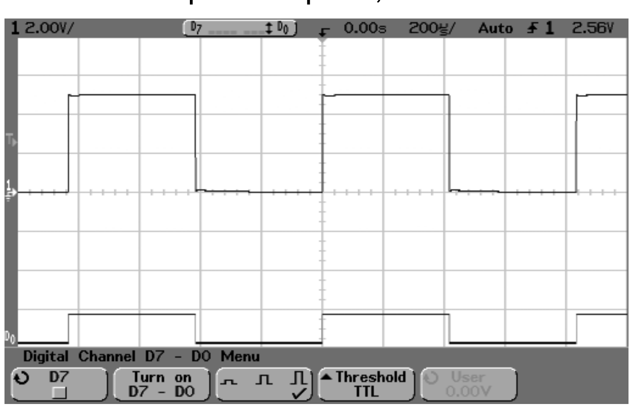


Notice that the **D0** line, at the bottom, is a squarewave, going between logic level 0 and 1,and **D1 - D7** are straight lines (logic level 0).

7) Let's turn off the unneeded digital channels: turn the **Entry Knob** until the arrow is next to **D1**, and press the left-most **softkey**. This will toggle **D1** on and off. Leave it off. Now, repeat this process, turning OFF **D2 - D7**. The display below shows **D0** and **D7** ON, and **D7** is about to be turned OFF.



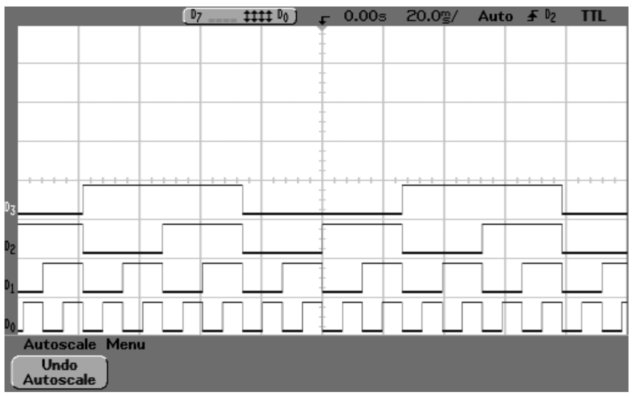
8) The last control we may want to change is the size of the digital channel(s), which is the middle softkey in the Digital Channel D7 - D0 Menu. Press it once, and digital channel 1 (**D1**) now is about one division peak-to-peak, as shown below.



Note that using the **Digital Channel Position** control (directly under the **D7 Thru D0** hardkey**)** we can move the **D0** display up and down on the screen, as needed. Also look at the top of the graticule in the display above; the **D7** _ _ _ _ _ _ _ _ ↕ **D0** box shows that **D7 - D1** are turned OFF, and that **D0** is going between logic 1 and 0.

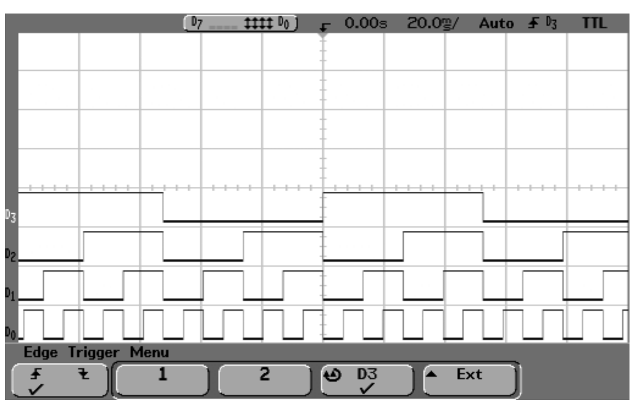
9) If you have a 4-bit counter available (e.g. a TTI 7493 IC or CMOS CD40161 or CD4024- use 4 of the 7 bits), connect the input to a suitable oscillator. Then, connect digital channels **D0 - D3** to the counter output, with **D3** going to the MSB (most significant bit).

10) Press the **AutoScale** button, and you should see a display somewhat like the one below:

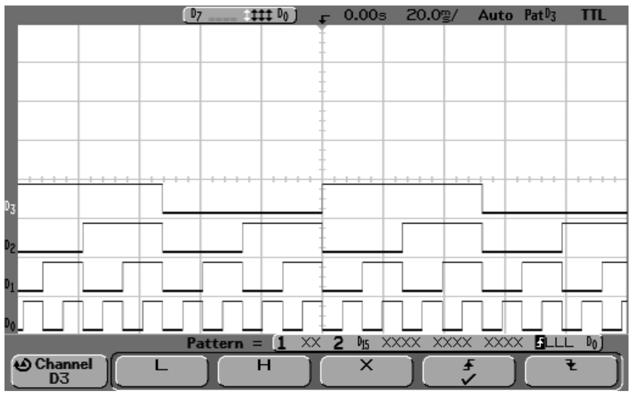


11) Triggering can be a bit tricky at first, and **AutoScale** doesn't always make a triggering choice that will give a "good" display. For example, in the display above of **D0 - D3**, the trigger source chosen is **D2** (with positive edge triggering), and the **D3** waveform is jumping around.

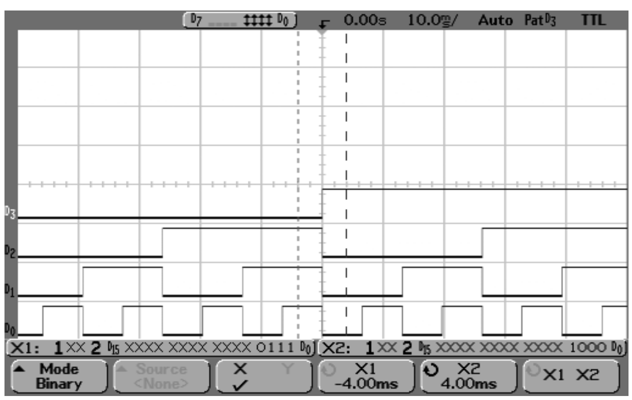
12) Press the **Edge** hardkey in the **Trigger** section, and rotate the **Entry Knob** to select **D3** as the trigger source (as shown below). Since **D3**, the MSB, is the lowest frequency of the four digital channels, a stable display will result.



13) Another triggering method that can be used is **Pattern** triggering. Press the **Pattern** hardkey in the **Trigger Section**, and use the **Entry Knob** to select **D0**. Press the **L** softkey (L = logic Low), and the rotate the **Entry Knob** to select **D1**. Press the **L** softkey again, and select **D2**; again press the **L** softkey. Lastly, Select **D3**, and press the up arrow (↑) softkey, to choose positive-edge triggering on channel **D3**. This process will now make the trigger occur when **D2 - D0** are **Low**, and **D3** has a positive edge. The display should look like the one below.

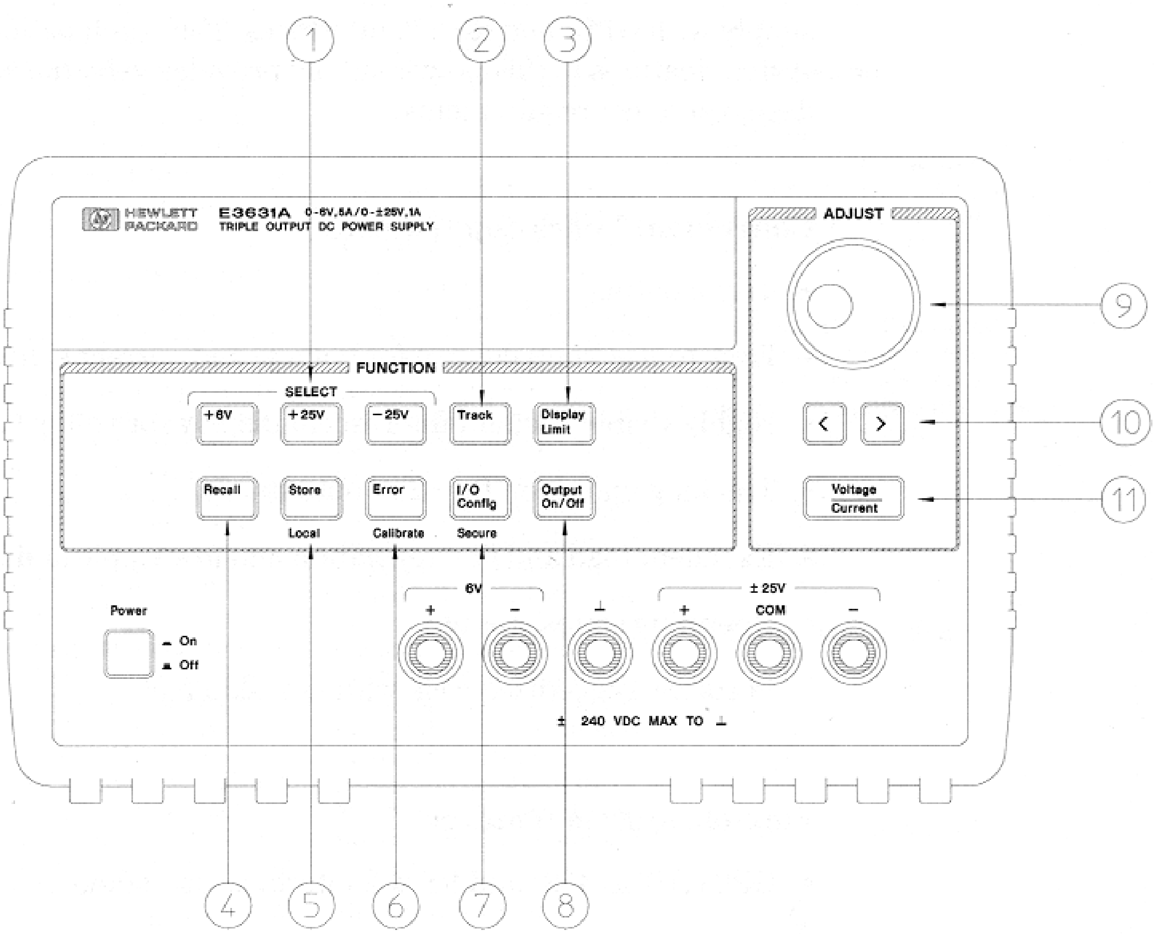


14) Cursors can be used with a digital display to show the logic levels in either **Binary** mode or **Hex** mode. Press the **Cursors** hardkey (in the **Measure** section), then press the **Mode** softkey to select **Binary**. The X1 and X2 cursors are positioned in the display below to show the logic values of 0111 just before the trigger point, and 1000 just after the trigger (the time base was changed to 10 ms/div for clarity). Try **Hex** mode as well.



Part Three – The Agilent E3631A Power Supply

A power supply is used to provide DC voltage(s) needed by a circuit that doesn't supply its own power. The Agilent E3631A is actually three power supplies:0 to 6 V, 0 to 25 V, and 0 to -25 V. In this section you will see how to set the voltage of each supply and how to use "current limiting" on each supply to protect your circuit.

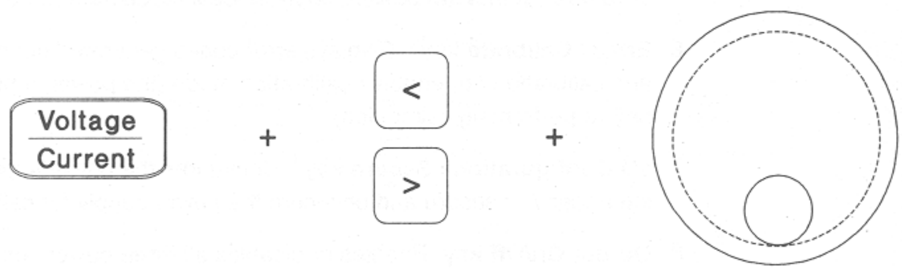


1 Meter and adjust selection keys
2 Tracking enable/disable key
3 Display limit key
4 Recall operating state key
5 Store operating state/Local key
6 Error/Calibrate key
7 I/O Configuration / Secure key
8 Output On/Off key
9 Control knob
10 Resolution selection keys
11 Voltage/current adjust selection key

Setting the voltage limit is quite simple to do, and to understand. Refer to the front panel picture on the previous page, and the instructions below.

You can set the voltage and current limit values from the front panel using the following method.

> **Use the voltage/current adjust selection key, the resolution selection keys, and the control knob to change the monitoring or limiting value of voltage or current.**



1 Press the `Display Limit` key after turning on the power supply.

2 Set the knob to the voltage control mode or current control mode using the voltage/current adjust selection key.

3 Move the blinking digit to the appropriate position using the resolution selection keys.

4 Change the blinking digit to the desired value using the control knob.

5 Press the `Output On/Off` key to enable the output. After about 5 seconds, the display will go to the output monitor mode automatically to display the voltage and current at the output.

Setting the Output Voltage:
Let's say you have built a circuit that needs +5 V. So, you follow the process above to set the +6 V supply to +5.000 V. Do this now, and verify that the output voltage is 5.000 V.

Or, your circuit needs +/- 15 V, in which case you follow the process above to set the +25 V supply to +15.00 V, and the -25 V supply to -15.00 V. Do this now, and verify that the voltages are +15 V and -15 V (both measured with respect to the black COM terminal).

However, taking the time to also set the current limit can be a really smart precaution, in case a wiring error, defective component, or accidental connection (e.g. bridging two pins on an IC) during testing would cause excessive current to flow and damage your circuit. The +6 V supply can source 5 amperes, and the +/- 25 V supplies can source 1 amp; this is a substantial amount of current, and can create a lot of smoke and damage to components in your circuit.

Setting the Current Limit:

1) Let's assume you have set the +6 V supply to 5.000V, and the display indicates you have the +6 V supply selected. To set the current limit, press the **Display Limit** button, then the **Voltage/Current** button. Use the control knob and the "Resolution Selection Keys" (the < and > keys underneath the control knob) to adjust the current limit to 0.020 A.

2) This setting of 0.020 A means that no matter what you do, the current from the +6 V supply (which is now set to 5.000 V) will never exceed 20 mA. Let's verify that two ways: with a short circuit, and with an LED.

3) First, note the voltage and current displays: the voltage is very close to 5.000 V, and the current is 0.000A (since nothing is connected to the + and - terminals of the 6 V supply). Also, at the right side of the display a "CV" is showing; this means the supply is in "Constant Voltage" mode, and will maintain 5.000 V *unless* the current reaches 20 mA, in which case the output voltage will drop to whatever value is needed to keep the current at 20 mA.

4) Connect a jumper wire (or alligator lead) between the + and - terminals of the 6 V supply. Note the voltage is 0.000 V, the current is close to 20 mA, and the "CV" has been replaced by "CC" ("Constant Current").

5) Now let's try this with an inexpensive LED. Connect an LED with the anode to the plus terminal, and the cathode to the minus terminal, of the 6 V supply (which is set to 5.000 V). Normally, without current limiting, an LED will be quickly destroyed by this procedure. What happens to your LED?

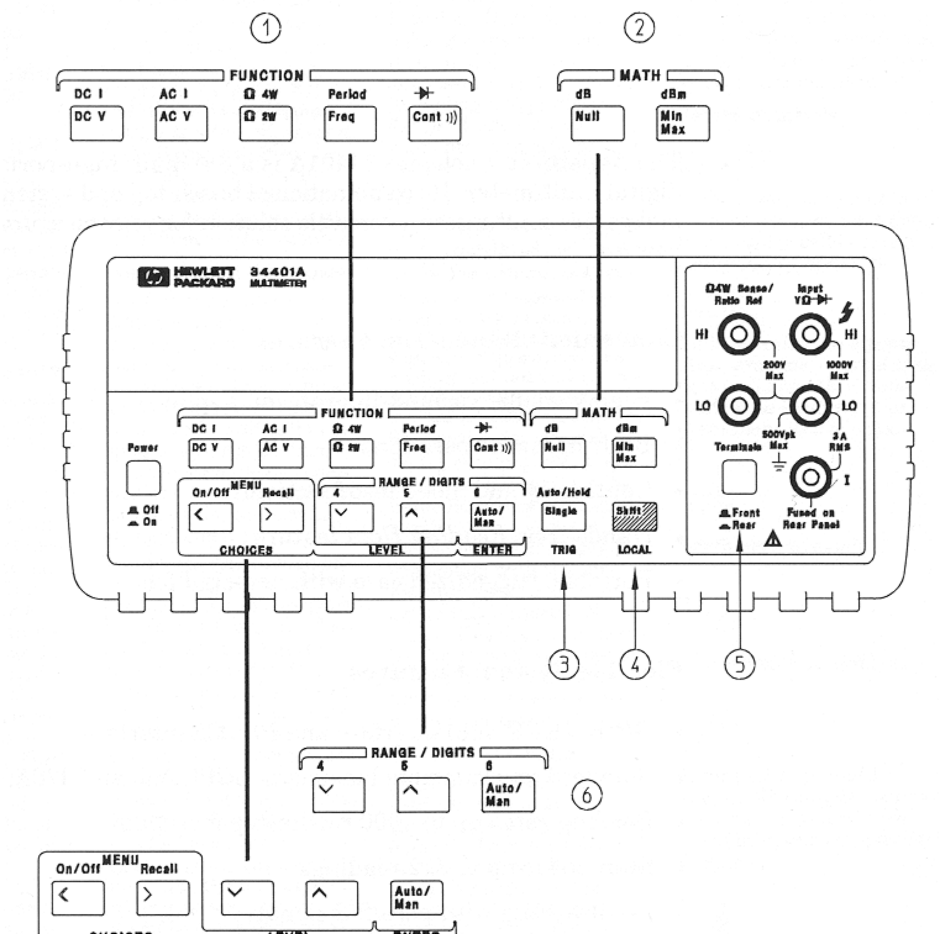
   What is the output voltage (it should be around 2.0 V, depending on the color of the LED)? What is the current (it should be near 20 mA). Your LED is emitting light, and will live to see another day, because the current limiting circuit protected it by lowering the voltage.

6) If you don't mind destroying an LED, you can see the effects of ***not using current limiting***. Turn the power supply off, then turn it on again. Set the +6 V supply to 5.000 V, and again connect the LED to the + and - terminals. What happened? LEDs will either burn out, or will get really hot (don't burn yourself on the leads of the LED).

This power supply has many useful features, including remote control and the ability to store and recall three setups from nonvolatile memory. Look in the Agilent manual for this instrument for complete information.

Part Four – The Agilent 34401A Digital Multimeter

The DMM (digital multimeter) is a very important laboratory instrument. This section will show you how to make three of the basic measurements: voltage, resistance and current. The 34401A has many capabilities beyond measuring V, R and I; consult the Agilent User's Guide for information on measuring frequency, period, continuity & diodes, and using the many features and functions of this DMM.



1 Measurement Function keys
2 Math Operation keys
3 Single Trigger / Autotrigger / Reading Hold key
4 Shift / Local key
5 Front / Rear Input Terminal Switch
6 Range / Number of Digits Displayed keys
7 Menu Operation keys

When you turn the DMM on, it will be in the "Power-On and Reset State":

Function	= DC volts
Input resistance	= 10M Ω (may be changed to 10G Ω ! for 100 mV, 1 V, & 10 V DC ranges)
Range	= Autorange
Resolution	= 5_ digits

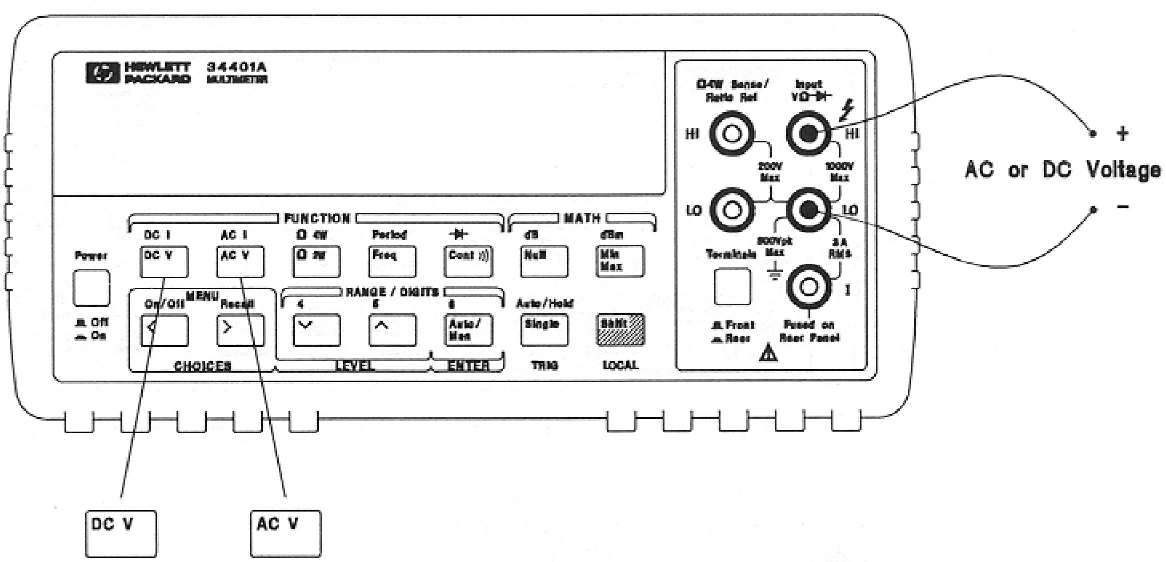
Measuring DC and AC Voltage

Key points:

- You ***must insert the voltmeter leads <u>across</u>*** the two points in a circuit for which you want to measure the voltage. Use the red and black input jacks as shown below.
- You *can* use the rear panel red and black input jacks also (be sure to select front or rear).
- If you use front and rear, you can easily and quickly select one of two voltages.
- The DMM will **Autorange**, unless you override it by selecting a range.
- You can choose the number of digits displayed, using the **DIGITS** buttons.
- Be sure to select **DC V** or **AC V**, as needed.
- AC voltage displayed is the true RMS value of the waveform.
- AC voltage is accurate to less than 1% up to 100 kHz.
- Input impedance for AC V is 1M Ω in parallel with 100 pF (not including test leads).
- Input resistance for DC V is 10M Ω, unless changed to 10G Ω ! for 100 mV, 1 V, & 10 V DC ranges

To Measure Voltage

Ranges: 100 mV, 1 V, 10 V, 100 V, 1000 V (750 Vac)
Maximum resolution: 100 nV (*on 100 mV range*)
AC technique: true RMS, ac-coupled

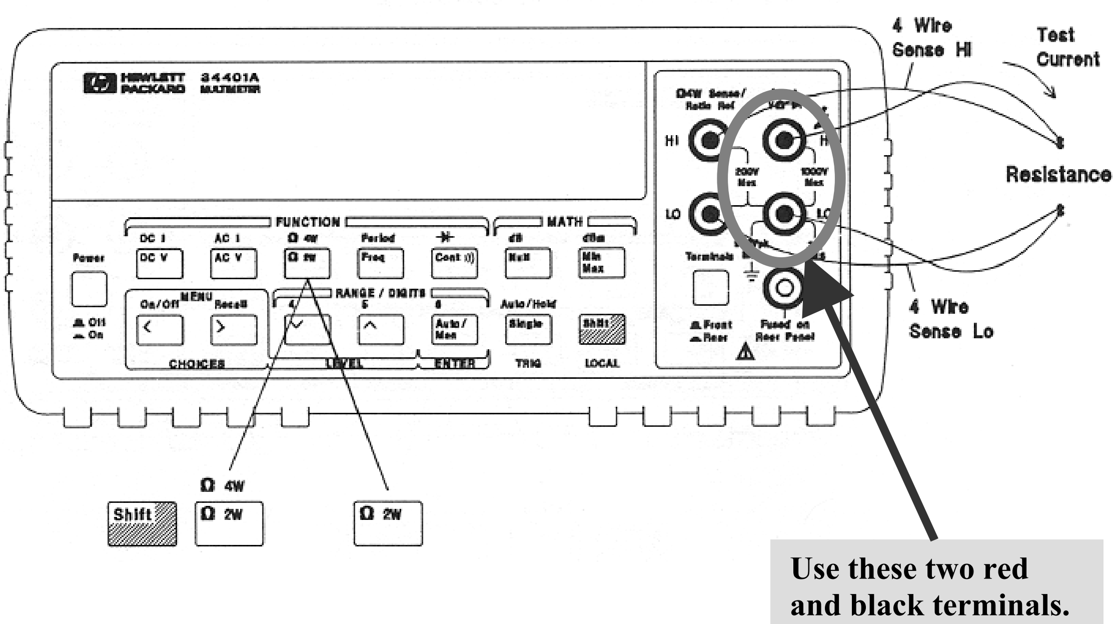


Measuring Resistance
Key points:

- **NEVER** measure resistance in a "live" circuit. Turn off all power to the circuit.
- If an ohmmeter is used in a "live" circuit, at best you will get incorrect readings; at worst you can seriously damage the DMM.
- You ***must insert the ohmmeter leads <u>across</u>*** the two points in a circuit for which you want to measure the resistance. Use the red and black input jacks as shown below.
- Use the red and black input jacks (on the right) as shown below.
- You *can* use the rear panel red and black input jacks also (be sure to select front or rear).
- If you use front and rear, you can easily and quickly select one of two voltages.
- The DMM will **Autorange**, unless you override it by selecting a range.
- You can choose the number of digits displayed, using the **DIGITS** buttons.
- Be sure to select **Ω 2W** (ohms, two-wire).
- **Ω 4W** (ohms, four-wire) is a more complex measurement that gives greater accuracy by eliminating the contact resistance (of the leads to the device under test).
- Make sure your hands don't contact both test leads; if they do, then you're measuring the device under test in parallel with ***you***.
- See User's Guide page 216 for the test current for each resistance range.

To Measure Resistance

Ranges: 100 Ω, 1 kΩ, 10 kΩ, 100 kΩ, 1 MΩ, 10 MΩ, 100 MΩ
Maximum resolution: 100 μΩ (*on 100 ohm range*)



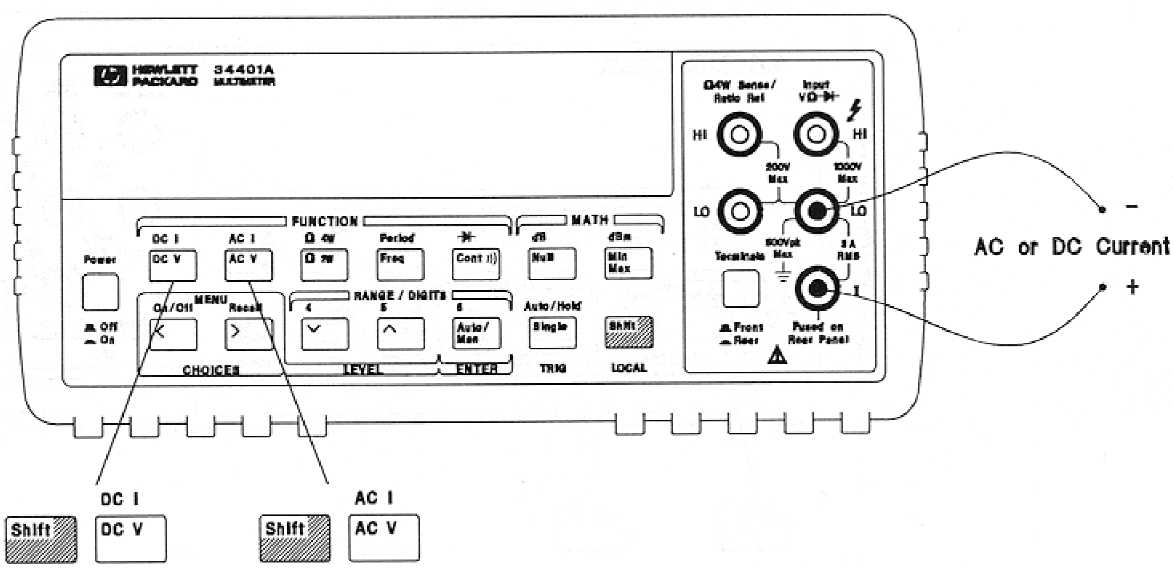
Measuring Current

Key points:

- You ***must insert the ammeter leads <u>in series</u>*** to measure current in a circuit.
- Use the red and black input jacks (on the right) as shown below.
- You *can* use the rear panel red and black input jacks also (be sure to select front or rear).
- If you use front and rear, you can easily and quickly select one of two currents.
- The DMM will **Autorange**, unless you override it by selecting a range.
- You can choose the number of digits displayed, using the **DIGITS** buttons.
- Be sure to select **DC I** or **AC I** to measure DC and AC current.
- See User's Guide page 216 for the "Burden Voltage" for each current range.

To Measure Current

Ranges: 10 mA (dc only), 100 mA (dc only), 1 A , 3 A
Maximum resolution: 10 nA (*on 10 mA range*)
AC technique: true RMS, ac-coupled



Part Five – The Agilent 33220A 20 MHz Function/Arbitrary Waveform Generator (AWG)

Introduction

The Function Generator is the instrument that creates input signals used to test circuits and systems in the laboratory. This section will show you how to create some basic waveforms commonly used in the lab: sine, square, triangle and pulse waveforms, including DC offset.

The 33220A has a large number of capabilities beyond measuring these basic waveforms; consult the Agilent User's Guide for information on capturing and using "arbitrary" waveforms, noise, signals with AM, FM and FSK modulation, sweeping signals over a range of frequencies. Operation of this "AWG" is easily learned, and fairly intuitive, as each front panel "hardkey" and "softkey" (the buttons right under the display window in the picture below) when pressed for 2 seconds will provide context-senstive "help" on the display.

Output Voltage and the Load Resistance

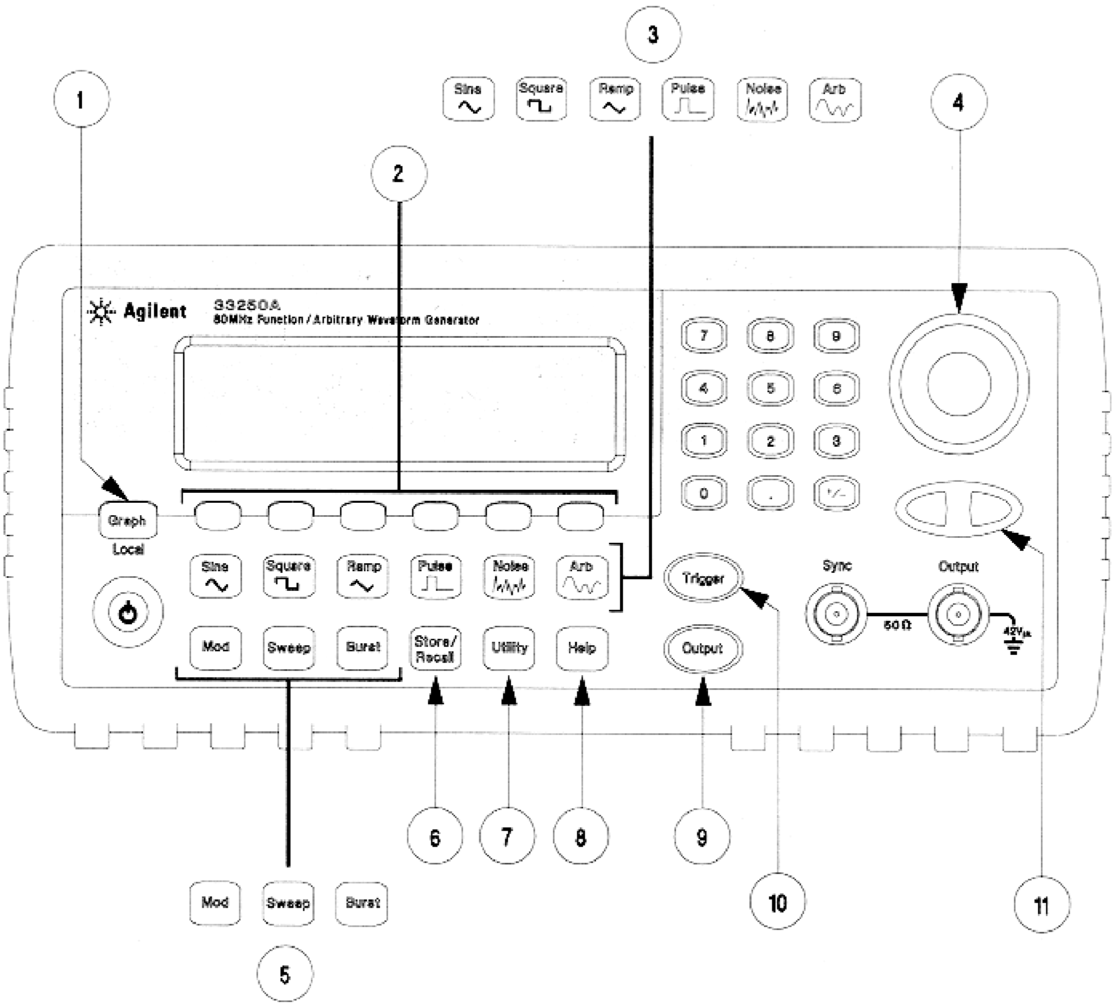
Since this is a very important consideration, let's address it right now. Basic concepts:

- You can easily set the amplitude and DC offset of the output waveform.
- The output resistance of the AWG is always 50 Ω - this can't be changed.
- ***If*** the AWG output connector is connected to a load (device under test) with a 50 Ω resistance, then the amplitude and DC offset voltage values you set ***will*** be the values across the load.
- ***If*** the AWG output connector is connected to a load (device under test) with a high resistance, then the amplitude and DC offset voltage values you set ***will NOT*** be the values across the load. The output voltage will be ***twice*** what you set. This may damage certain circuits (i.e. logic gate inputs).
- You can tell the AWG that the load resistance is not 50 Ω. This will make the displayed voltage (the one you set) and the open-circuit voltage the same.

We will practice using 50 Ω and "high Z" load resistances very soon.

The Front Panel Controls

Refer to the diagram below as you perform the procedures that follow.



1 Graph Mode/Local Key
2 Menu Operation Softkeys
3 Waveform Selection Keys
4 Knob
5 Modulation/Sweep/Burst Menus
6 State Storage Menu
7 Utility Menu
8 Instrument Help Topic Menu
9 Output Enable/Disable Key
10 Manual Trigger Key (*used for Sweep and Burst only*)
11 Navigation Arrow Keys

Step One - Creating a Sinewave With No DC Offset Voltage, High Z Load Resistance

1. A basic waveform you will now produce is a 1 kHz, 100 mVpp sine wave with no DC offset. Connect the **Output** (a BNC connector on the front panel) to the vertical input of an oscilloscope.

2. Turn the power **on** by pressing the white button at the lower left of the front panel. You are reminded that you can get **Help** for any key by holding it down.

3. The display says 1.000,000,0 kHz, with a picture of a sinewave on the right. It also says "Output Off", so the first thing we need to do is press the **Output** button. Press it now.

4. Press **AutoScale** on the mixed-signal oscilloscope (MSO) (or adjust its controls manually) to display the sinewave. Press **QuickMeas** on the MSO to measure the frequency and peak-to-peak voltage (should be very close to 1.000 kHz and 200 mVpp). You may want to turn on **Averaging** on the MSO to minimize noise and get more accurate readings.

5. Now, press the **Softkey** labeled **Ampl**; it should say 100.0 mVpp. **Wait a minute! What's going on here - we just measured 200 mVpp?** The reason for this discrepancy is that the oscilloscope input resistance is 1M Ω (High Z), ***not*** the 50 Ω the AWG *assumed* was the load resistance connected to the AWG output connector. How do we fix this discrepancy?

6. Since we are going to be using the MSO as our "Device Under Test" (i.e. the load connected to the AWG), and the MSO resistance is 1M Ω, **not** the 50 Ω the MSO "expected", we will tell the AWG what its load is.

7. Press the **Utility** hardkey, then the **Output Setup** softkey, and the **Load** softkey. This changes the AWG to "**High Z**" mode, and it will now display the correct output amplitude.

8. Press the **Sine** hardkey; the amplitude now displayed is 200.0 mVpp, which agrees with the amplitude as measured on the oscilloscope.

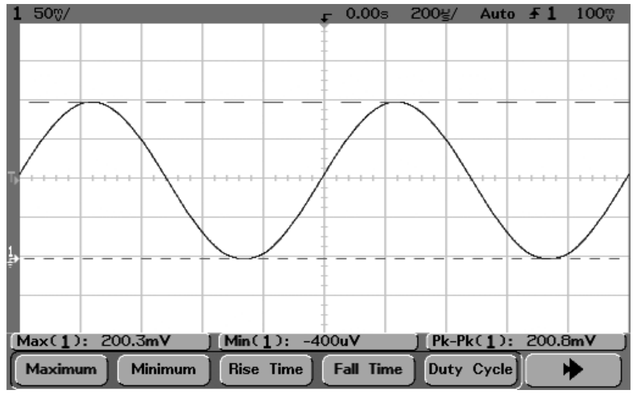
Be aware of the load resistance of the circuit or instrument you connect to the AWG. If you were connecting the AWG output to a logic circuit (which could be damaged by an input voltage that is too big) and did not change the AWG to "**High Z**" mode, the actual AWG output voltage can be twice what the AWG display indicates.

Step Two - Creating a Sinewave With DC Offset Voltage, High Z Load Resistance

1) Leave the AWG in "**High Z**" mode. Now we will add some DC offset to our 1 kHz sinewave, 200 mVpp. This can be done two ways.

2) Method 1: Press the **Offset** softkey. Use the **right "Navigation Arrow Key"** to move the cursor one place to the right on the amplitude display. Rotate the **Knob** one click **clockwise**; this will add +100 mVDC to the 200 mVpp sinewave. See the display below.

3) In the display to the right we can see that our 200 mVpp sinewave now has a minimum value near 0 V (-400 uV = -0.4 mV), and a maximum value of 200.3 mVpp. Note the ground symbol on the display.



4) Remove the 100 mV DC offset using the **Knob**. Our sinewave is still 200 mVpp.

5) Method 2: Press the **Offset** softkey, to select **LoLevel**. Raise this to 0.00 V. Now, press the "**HiLevel**" softkey, and use the **Knob** to raise the high level to 200 mV. This accomplishes the same result: a 200 mVpp sinewave, that goes between 0 V and 200 mV.

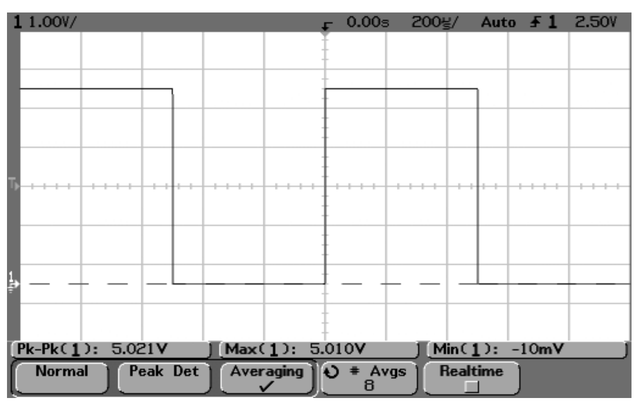
6) Press the **Graph** hardkey; you can see the waveform you have just created, and can modify the frequency, amplitude and DC offset using the yellow, purple and blue softkeys respectively. Try it. When you are done, press the **Graph** hardkey again.

Step Three - Creating a Squarewave With DC Offset Voltage, High Z Load Resistance

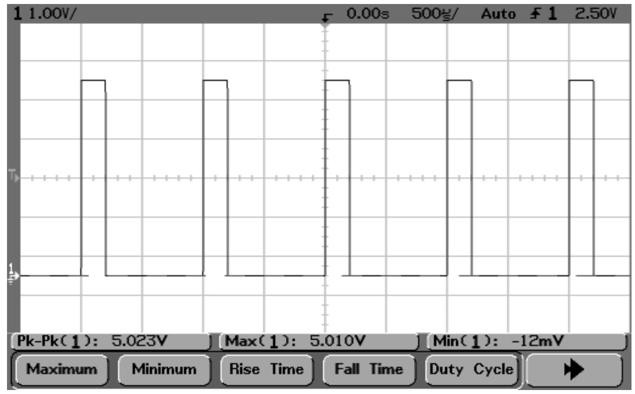
1) Create a squarewave compatible with TTL or 5 V CMOS logic by pressing the **Square** hardkey.

2) Press the **Ampl** softkey, which will turn it into **HiLevel** mode. Now use the Numerical Keypad to enter: 5.00, then press the **V** softkey. You now have a 5 Vpp squarewave, which goes between -2.5 V and + 2.5V. Verify this with the MSO.

3) Press the **LoLevel** softkey, and the display will say -100 mV. Change this with the **Knob** to be 000.0 mV. Your 5 Vpp squarewave, now goes between 0.0 V and + 5.0V. Verify this with the MSO.



4) You can turn the squarewave into a pulse train by pressing the **Duty Cycle** softkey. Change the duty cycle to 20% (the range is 20% to 80%), either using the **Knob** or the Numerical Keypad. Your 5 Vpp, 1 kHz pulse train, goes between 0 V and +5 V. Verify that yours looks like the one to the right.



5) If you need a pulse with a duty cycle less than 20% or more than 80%, use the **Pulse** hardkey

Step Four - Creating a Triangle or Ramp Waveform, High Z Load Resistance

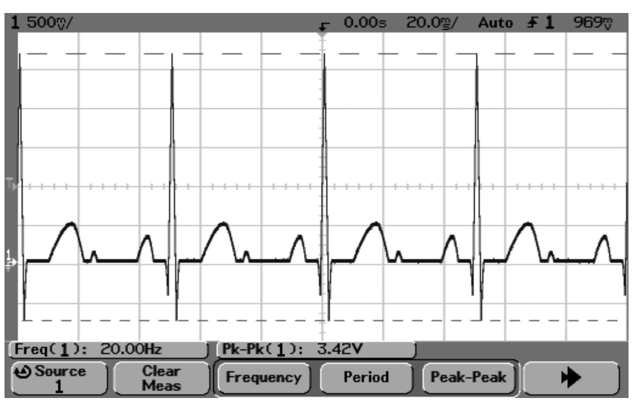
1) Press the **Ramp** hardkey. The resulting waveform is a classic sawtooth shape.

2) By varying the **Symmetry**, you can make a triangle waveform, and sawtooth waveforms with different shapes. Try it.

Other Waveforms and Features to Try

Noise (white, not pink) is available. Be sure that **Averaging** is turned off if you look at noise.

Arbitrary waveforms are non-standard voltages that either are stored in the AWG memory ("Built-In" waveforms) or that you capture and load into the AWG memory. Try pressing the **Arb** hardkey, then the **Select Wform** softkey, then the **Built-In** softkey, and finally the **Cardiac** softkey. To make it realistic, a human cardiac waveform should vary between approximately 0.33 Hz and 1 Hz (corresponding to 180 and 60 beats per minute). For now, to make it easy to observe on your MSO, set the frequency to 20 Hz (representing, perhaps, a tachycardic hummingbird after a double espresso); see the display below.



Other Useful Information and Review

As covered in the procedure, but of such importance that its repeated again: The output must be terminated by 50 Ω for voltages set on the AWG to be accurate. OR, for loads that are much higher resistance than 50 Ω, you can use the **Utility** key, then **Output Setup** softkey, and **Load** softkey to select **High Z**.

The **Help** function tells us that "Load Impedance / High Z / 50Ω Sets the value of the load attached to the [Output] connector (used for voltage settings). The Agilent 33220A has a fixed series output impedance of 50Ω, regardless of the value specified for the parameter. If the actual load is different than the value specified, the displayed voltage levels will not match voltage levels at the device under test."

To get context-sensitive help on any front panel key or softkey menu, press and hold down that key.

You can even get pure DC (like a small power supply) using **Utility** and **DC On**. This "power supply" will have a 50Ω output resistance, making its usefulness limited.

You can see a graph of your waveform with all its parts illustrated, in color, with the colors keyed to the controls that adjust those parameters (e.g. for a **Pulse** waveform, the pulse width, edge time, HiLevel, LoLevel, offset), by pressing the **Graph** hardkey.

Try using the **Mod** (modulation), **Sweep**, **Burst**, and **Help** hardkeys. Of course, for complete and detailed information consult the Agilent User's Guide.

This instrument has a frequency range of 1 µHz to 20 MHz for most waveforms. Think about the frequency of sunlight on the planet Earth: its period is one day. What is that, expressed as a frequency in hertz? Can you think of how to design a circuit to have this generator provide illumination to a plant, using an incandescent lamp, to simulate the period of earth's sunlight?

You *may* not have been the last person to use the AWG. When first turning the AWG on, be sure to return the AWG to its "factory default values" by pressing the **Store/Recall** hardkey, then the **Set to Defaults** softkey, and the **YES** softkey.