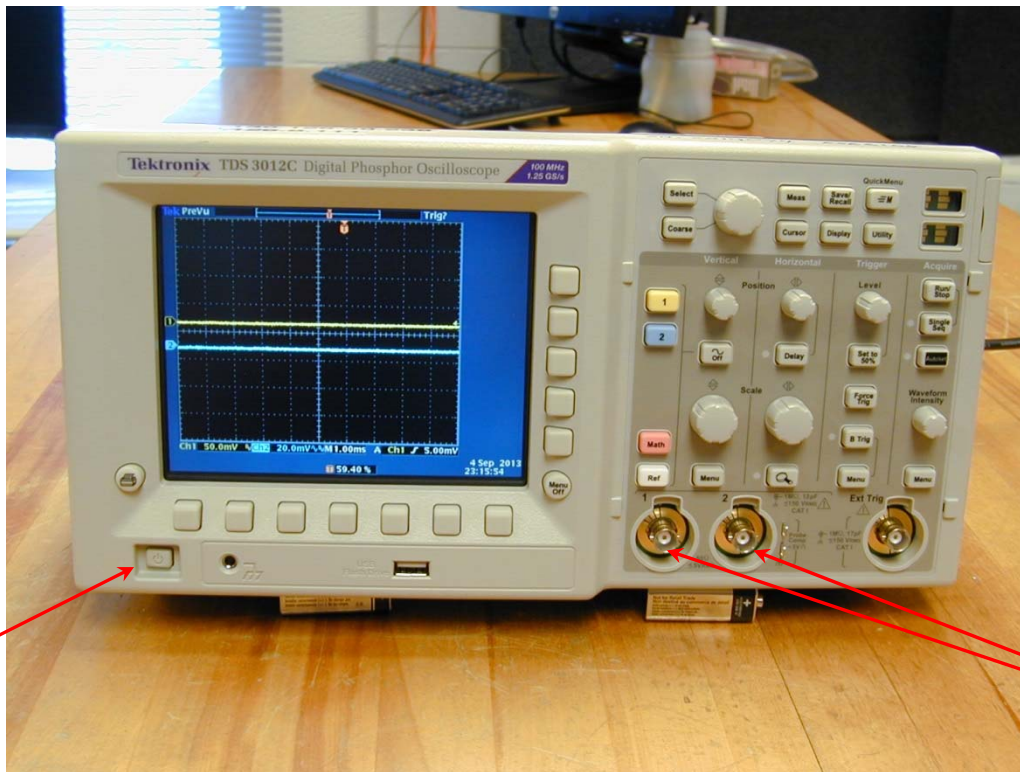


Basics of Using the Tektronix 3012 Digital Oscilloscope

This document {briefly} describes the basic features of, and how to use the Tektronix 3012 Oscilloscope. Detailed information is contained in the Tektronix 3012 User's Manual.



AC Pwr
On/Off
Switch

Ch1 & Ch2
Inputs

Oscilloscopes are devices used to display time-varying voltage signals. The Tek 3012 'Scope can display two such signals simultaneously (the yellow and blue traces on the above screen).

Plug in the Tek 3012 'Scope to the 120Vac wall power, then turn it on by pressing the AC power On/Off Switch. It takes a few seconds for the 'Scope to boot up {it has a specialized processor in it}. The power-on settings of the Tek 3012 'Scope default to those selected/operative at the time when it was last used.

On the right-hand side of the Tek 'Scope the controls are organized into four vertical columns. The yellow (blue) buttons on the left-hand side of this section of the 'Scope select/activate Ch1 (Ch2) respectively.

The knobs/buttons in the **Vertical** column enables the user to set/adjust various of the Ch1/Ch2 input parameters/settings of the 'Scope. Rotating the large round knob in the **Vertical** column changes the voltage sensitivity setting of the 'Scope (for the selected Ch1/Ch2) from 1 mV to 100 V per vertical division on the above waveform display screen. The **Off** button in the middle of the **Vertical** column deletes/deactivates the display of the currently selected Ch1/Ch2 (yellow/blue button, respectively). The **Menu** button at the bottom of the **Vertical** column displays additional parameters that can be selected from the buttons immediately below and to the right of the waveform display screen, such as DC or AC coupling of the input signal, limiting the bandwidth (BW) of the 'Scope, ... The **Position** knob at the top of the **Vertical** column simply sets the location of the 0 Volts reference point on the display screen of the 'Scope. Explicitly try using each of these knobs/controls/buttons to see/learn what each does, how it functions and/or what it controls.

The knobs/buttons in the **Horizontal** column enables the user to set/adjust the time base of the ‘Scope – *i.e.* the time units per horizontal division on the display screen of the scope. By rotating the large round knob in the **Horizontal** column, the ‘Scope’s time base can be changed in discrete steps ranging from 1 ns to 10 s per horizontal division of the ‘Scope. The **Position** knob at the top of the **Horizontal** column simply sets the location of the $t = 0$ horizontal reference point on the display screen of the ‘Scope. Pressing the **Delay** button in the **Horizontal** column calls up a menu that enables the user to delay the selected Ch1/Ch2’s signal, relative to the time of the ‘Scope’s **Trigger** (discussed below). Pressing the button with the **Magnifying Glass Icon** calls up yet another menu that enables the user to {locally} expand the time base about the trigger time.

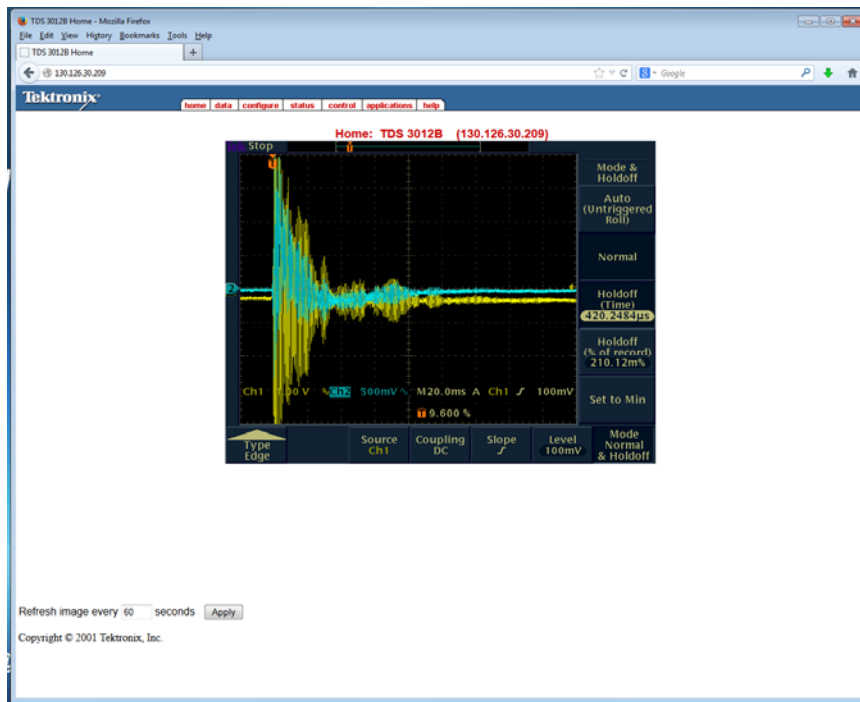
The knobs/buttons in the **Trigger** column enables the user to trigger the display of the Ch1/Ch2 signals either from *e.g.* the rising or falling slope of Ch1 (or Ch2), or *e.g.* from an external reference signal, or, to have the ‘Scope automatically trigger on Ch1, Ch2 or the ext. ref. signal. The **Menu** button at the bottom of the **Trigger** column additionally displays these parameters, which can be selected from the buttons immediately below and to the right of the waveform display screen.

The last/furthest to the right **Acquire** column has buttons that **Run/Stop** the display, allow only a **Single** Trigger; the **Menu** button in the **Acquire** column enables the user to access/select other related features. If the waveform(s) are too dim/too bright, rotate the **Waveform Intensity** knob in the **Acquire** column to adjust. Pressing the **AutoSet** button time-irreversibly resets the ‘Scope (essentially) to its factory default settings, modulo a bit of smarts for figuring out a suitable choice for volts/vertical division and time units/horizontal division on the ‘Scope’s display screen... this is a very useful button to press if the totality of the settings on the ‘Scope are so hopelessly screwed up to be able to figure them out in any reasonable amount of time...

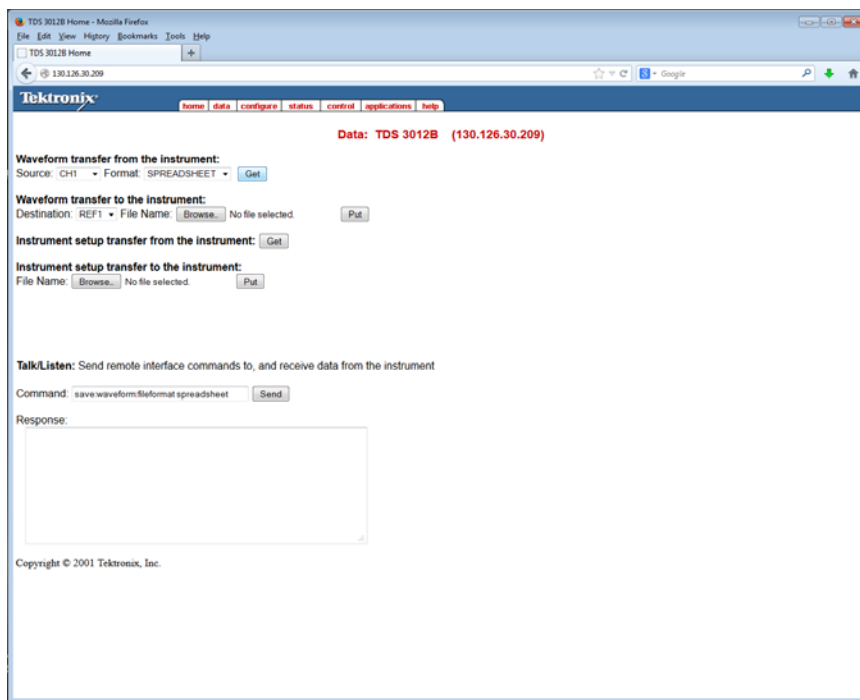
There are many more powerful features associated with the Tek 3012 ‘Scope – the above discussed are just the basics. The ‘Scope can measure frequencies/periods, voltage amplitudes, as well as more sophisticated things such as the harmonic content of the waveform(s), via Fast Fourier Transform (FFT) techniques... It is also possible to readout/waveform capture the ‘Scope’s screen display contents via the internet (see below)... Consult the Tek 3012 ‘Scope User Manual or ask one of the POM Lab TA’s for more information on using these more advanced measurement capabilities of the Tek 3012 ‘Scope.

Read-Out of $V(t)$ vs. t Waveform Data from the Tek ‘Scopes via the Internet:

The Ch1/Ch2 $V(t)$ vs. t waveform data can be read out from the Tek ‘Scopes via the internet. To do this, first, turn of the AC power to the the ‘Scope. Then connect the end of one of the orange CAT-V Ethernet cables to the scope, and the other end of this cable to a nearby Ethernet **network box**. Then, power up the ‘Scope. During its boot-up process, you should see a message on the display screen of the ‘Scope telling you what its IP address is (*e.g.* 130.126.30.{208:212} for the 6 Tek ‘Scopes that we have/use in the POM lab); this IP address should indeed match to that given on the white label on the top of the particular ‘Scope that you are using. If you do **not** see this message, please contact/get the attention of one of POM TA’s in the lab. Next, bring up *e.g.* the Firefox web-browser on a PC in the lab, and type in the IP address (*e.g.* 130.126.30.{208:212} of the ‘Scope into the URL port of the web-browser in its upper LHS corner port. You should then get a web display similar to that shown in the figure below:



If you want to capture and then read out data associated with a **specific** waveform (or 2 such waveforms, if simultaneously using **both** channels of the ‘Scope (as in the figure above), we recommend setting the ‘Scope’s ACQUIRE setting to SINGLE SEQ mode (button on upper RHS of the ‘Scope). Once you have acquired the waveforms on the ‘Scope, then mouse-click on the red “**data**” tab on the Tek ‘Scope web-based readout application, which will look similar to that shown in the figure below:



Use the pull-down menus for **Waveform transfer from the instrument** to choose the **Source** (Ch1/Ch2) and the **Format** (choose SPREADSHEET). Then mouse-click on the **Get** button.

Save the resulting *getwfm.isf* file (*n.b.* comma-separated format text file) *e.g.* to your PC's desktop area. If you will be reading out more data, you will need to rename the *getwfm.isf* file in order to avoid overwriting of it by subsequent **Get** read-out operations...

The structure of the *getwfm.isf* file is shown in the example below – there are a total of 10K samples of Ch1/Ch2 $V(t)$ vs. t waveform data, time (voltage) info in the first (second) columns, respectively:

```
-1.920e-02,-0.22  
-1.918e-02,-0.20  
-1.916e-02,-0.20  
-1.914e-02,-0.26  
-1.912e-02,-0.20  
-1.910e-02,-0.24  
-1.908e-02,-0.24  
-1.906e-02,-0.24  
-1.904e-02,-0.26
```

We have written various MATLAB scripts to read in and plot the Tek 'Scope data contained in *getwfm.isf* files. In the main **Matlab_Analyses** folder in the P406 Backup Area { \\engr-file-03\phyinst\APL Courses\PHYCS498PM\Common\MATLAB_Analyses } is a sub-folder named **Tek_eScope** containing two Matlab scripts: **Tek_eScope.m** and **Tek_eScope_2x.m** If not already resident in the local MATLAB Work folder { C:\Program Files\MATLAB\R2012a\work } of your PC, copy the entire contents of the **MATLAB_Analyses\Tek_eScope** folder in the P406 Backup Area to the local MATLAB Work folder, then launch the MATLAB application – double-click on the MATLAB icon on your Desktop and then change/set the folder location of the MATLAB application to the **Tek_eScope** folder in the local MATLAB Work folder on the PC { C:\Program Files\MATLAB\R2012a\work\Tek_eScope }. Double-click on *e.g.* **Tek_eScope.m** (or if wanting to read in/plot both Ch1 .and. Ch2 data, double-click on **Tek_eScope_2x.m**), change the name of the one (or two) input files, save the new *.m file, then exit this file, then Left-mouse-click on **Tek_eScope.m** or **Tek_eScope_2x.m** to run the file. Plots of the data will then appear. If you want to save the pix, run **Tek_eScope_Save_Pix.m**, this will create a **Pix** sub-folder in the local **Tek_eScope** folder. If multiple 'Scope data files are to be analyzed/plotted, then rename the **Pix** folder to something physically meaningful to you so that it does not get overwritten by subsequent pix-saving...