

INTRODUCTION TO OSCILLOSCOPE

BEFORE YOU BEGIN

PREREQUISITE LABS

- ▶ Introduction to MATLAB

EXPECTED KNOWLEDGE

- ▶ Ohm's Law: $v = iR$
- ▶ MATLAB Command Environment

EQUIPMENT

- ▶ TDS3034B Digital Phosphor Oscilloscope
- ▶ AFG3000 Series Arbitrary/Function Generator

MATERIALS

- ▶ BNC to BNC cable
- ▶ BNC T connector
- ▶ 50 Ω terminal resistor
- ▶ Resistor Kit
- ▶ Oscilloscope probe
- ▶ Formatted 1.44 MB 3-1/2" floppy diskette

OBJECTIVES

After completing this lab you should know how to:

- ▶ Compensate oscilloscope probes.
- ▶ Use the TDS3034B Oscilloscope to measure the amplitude, frequency and period of signals.
- ▶ Save the data from the TDS3034B Oscilloscope as graphic and data files.
- ▶ Plot the data from the TDS3034B Oscilloscope in MATLAB.

INTRODUCTION

Oscilloscopes are tools that allow engineers to view signals graphically. The oscilloscope, often just called the scope, displays the signals as a plot of magnitude versus time.

There are two types of oscilloscopes: analog and digital. Analog oscilloscopes use a cathode ray tube and display the signal much like a television set displays an image. Digital oscilloscopes sample the signals digitally and are more flexible in how they display, manipulate, and store the signals.

PRELAB

CIRCUITS

Build the circuit shown in Figure 1.

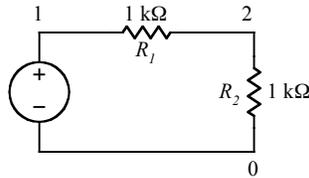


Figure 1. Circuit used in Lab Exercise 7.

MATLAB

Matrix and Vector Subscripts

Given a matrix A with m rows and n columns, the element in row i and column j of matrix A is referenced as $A(i,j)$. Subscript expression with the colon ($:$) operator refers to portions of the matrix. For example, given the matrix

$$A = \begin{bmatrix} 8 & 1 & 6 \\ 3 & 5 & 7 \\ 4 & 9 & 2 \end{bmatrix}$$

$a = A(:,1)$ would result in the column vector $\begin{bmatrix} 8 \\ 3 \\ 4 \end{bmatrix}$,

$b = A(1,:)$ would result in the row vector $[8 \ 1 \ 6]$,

and

$C = A(2:3,1:2)$ would result in the matrix $\begin{bmatrix} 3 & 5 \\ 4 & 9 \end{bmatrix}$.

Answer Questions 1 – 5.

Creating a Range

Answer Questions 6 – 7.

Recall that the colon operator can also be used to create a range of evenly spaced variables. The syntax for the colon operator is

variable = start:incr:stop

where

start is the starting value

incr is the increment (or step) value

stop is the final value

Answer Questions 8 – 10.

TURNING ON THE OSCILLOSCOPE

To turn on the oscilloscope, press the power on button on the lower left-hand side of the oscilloscope. Shortly afterwards, you will see a boot screen, and then a start up screen. Press the MENU OFF button to proceed.

OSCILLOSCOPE SCREEN

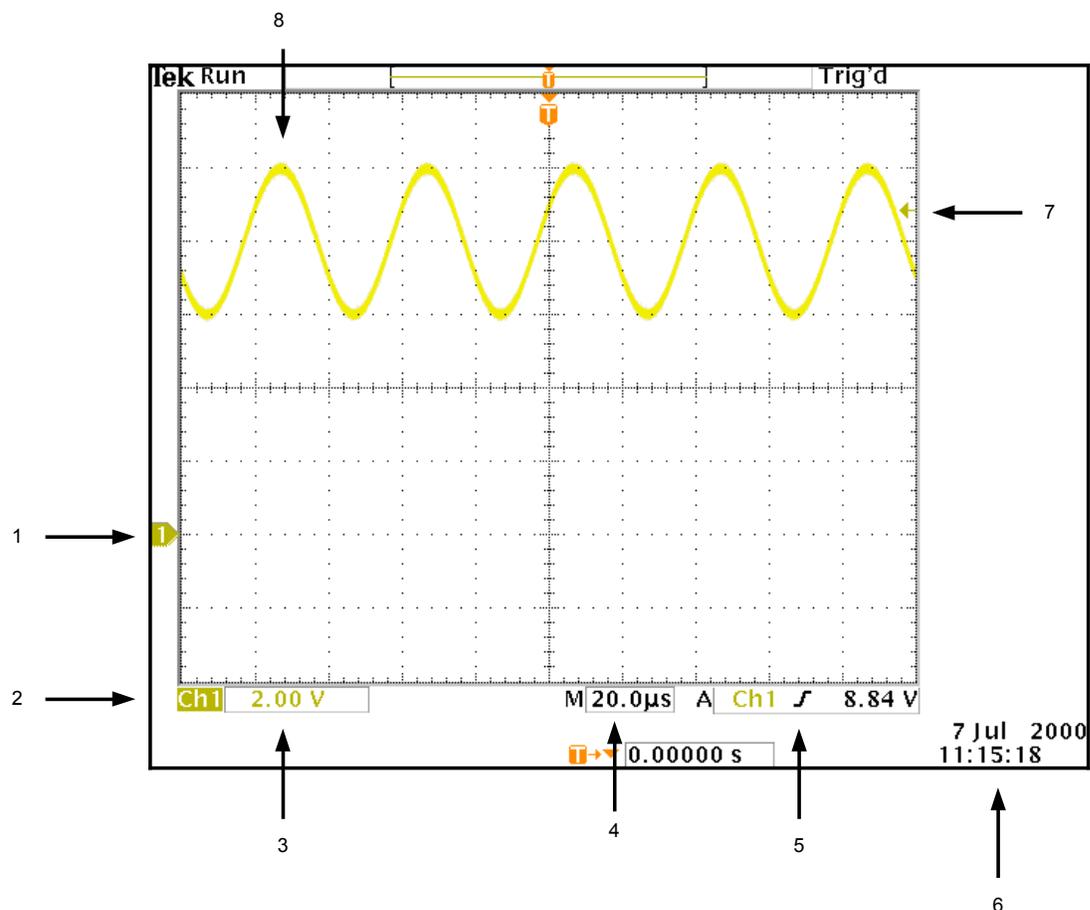


Figure 2. Oscilloscope Screen

Figure 2 depicts the typical oscilloscope screen. The elements of the screen are as follows:

1. Waveform baseline icon. Shows the zero volt level of the waveform.
2. Channel indicator.
3. Channel scale factor.
4. Time per division.
5. Trigger slope.
6. Date and time.
7. Trigger level icon.
8. Visual representation of waveform.

OSCILLOSCOPE SETTINGS

As for the arbitrary/function generator, the default settings of the oscilloscope can be manually changed, and are retained when the oscilloscope is turned off.

Fortunately, the oscilloscope settings can be restored to the factory settings. It is a good practice to reset the oscilloscope to factory settings after it is turned on and before any measurements are taken. This way you can always begin working with the scope from the same initial setting.

To restore the oscilloscope to factory settings, press the SAVE/RECALL button at the top of the oscilloscope. Two menus will be displayed: one on the bottom of the screen and one on the right hand side of the screen (see Figure 3). From the bottom menu, press the button bellow “Recall Factory Setup.” Then press the button on the right hand side menu to confirm the change. The oscilloscope is now set to factory settings. Press the MENU OFF button, which is located next to the lower left side of the screen, to remove the menu.

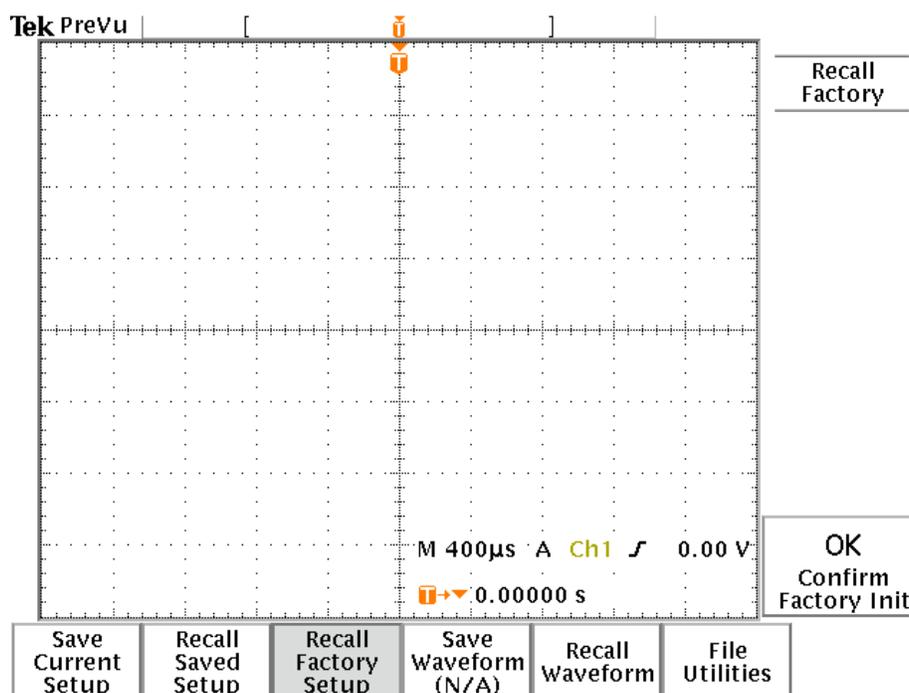


Figure 3. Recalling Factory Setup

OSCILLOSCOPE PROBES

The oscilloscope probes that came with your Laboratory Kit are switchable between 1X and 10X. When set to 1X, the oscilloscope will display the actual signal to scale. When set to 10X, the oscilloscope reading will be one tenth of the actual signal.

PROBE COMPENSATION

Before using the oscilloscope probe, the oscilloscope must be calibrated, or compensated, to account for the non-ideal effects that the probe has on voltage measurements. Oscilloscope probes should be compensated every time they are used, especially if the probes are being used with a different oscilloscope than last time the probes were compensated. The following steps describe how to compensate the oscilloscope probes. Compensate both of the oscilloscope probes that came with your Laboratory Kit.

1. Connect the scope probe to channel 1.
2. Attach probe tip and reference lead (ground) to the PROBE COMP connectors located between the channel 2 and channel 3 terminals.
3. Switch the oscilloscope probe to 10X.
4. Press the AUTOSSET button. You should see a square wave similar to the one in Figure 4.
5. Adjust the oscilloscope probe until you see a square wave with a flat top.

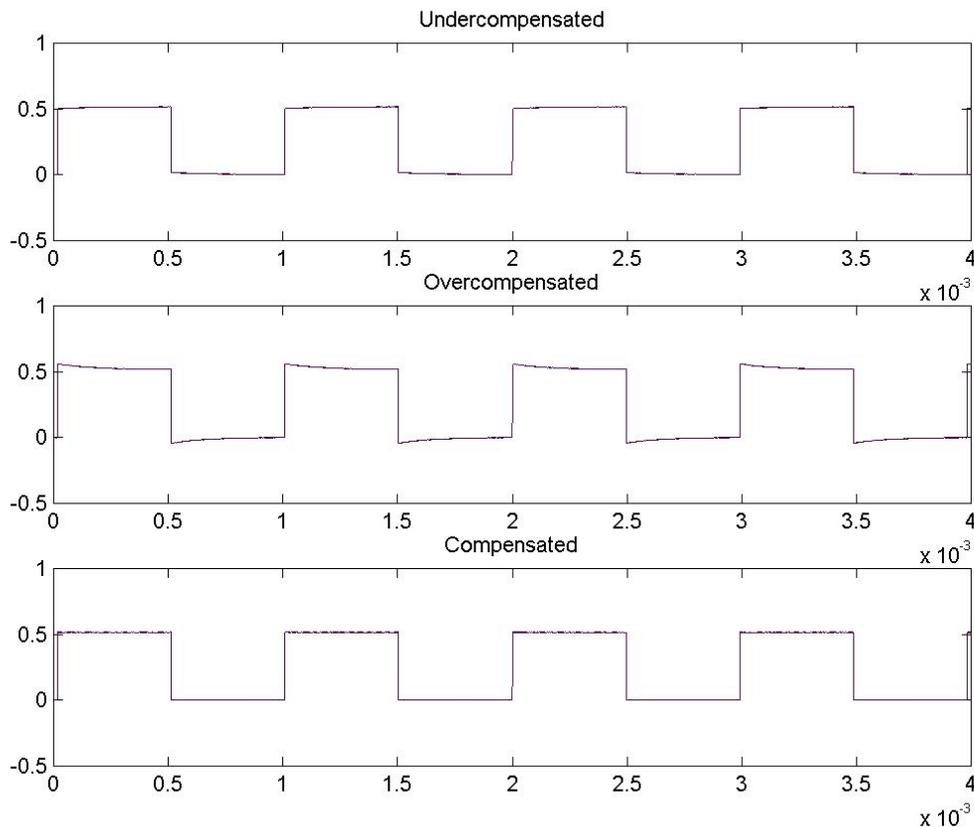


Figure 4. Under, Over and Correct Probe Compensation

SCREEN MEASUREMENTS

The oscilloscope can be used to measure the different properties of signals. In this section you will learn how to use the oscilloscope to measure the amplitude, frequency and period of a signal. The signal that you measure will be the default signal supplied by the arbitrary/function generator.

Turn on the arbitrary/function generator. Use a BNC to BNC cable to connect output 1 of the function generator to channel 1 of the oscilloscope. Turn On the Output of Channel 1 of the function generator. On the screen of the oscilloscope you should see a yellow “fuzz.” Press the AUTOSET button on the oscilloscope. After a second or two you should see a sine wave.

MEASURING AMPLITUDE

Answer Question 11.

The arbitrary/function generator expects a $50\ \Omega$ terminal load. With a load significantly larger than $50\ \Omega$, the function generator will supply double the voltage it is set to. That is why the oscilloscope has a sine wave with amplitude of 1 V, or a peak to peak voltage of 2 V. There are two ways to fix this:

1. With a BNC T connector, connect a $50\ \Omega$ terminal load to the input of the oscilloscope.
2. Set the oscilloscope to internal impedance of $50\ \Omega$.

On the oscilloscope, between the channel 1 and 2 terminals, and slightly above, press the MENU button. This menu button brings up the menu for channel settings. On the bottom of the right hand menu you will see the omega symbol (Ω). Pressing this button toggles the input impedance between $1\ \text{M}\Omega$ and $50\ \Omega$.

Answer Question 12.

MEASURING FREQUENCY AND PERIOD

Answer Questions 13 – 14.

CURSOR MEASUREMENTS

For more accurate readings, the oscilloscope has cursors that you can activate and move on screen to measure the amplitude, frequency and period of a signal. To activate the cursors, press the CURSOR button at the top of the oscilloscope panel. The menu shown in Figure 5 will appear on the oscilloscope.

MEASURING AMPLITUDE

To measure the amplitude of the signal, select “H Bars” from the menu. You will see two horizontal lines: one dashed line and one solid line. The solid line is the selected cursor. On the top right corner of the screen you will see a delta symbol (Δ) and an “at” symbol (@). The Δ

symbol display shows the difference between the two cursors and the @ symbol is the amplitude of the selected cursor.

Using the knob at the top left of the control panel, move the selected cursor to the bottom of the signal so that it sets on the lower peak of the signal. To toggle between the two cursors, press SELECT (top left of control panel). Move the second cursor so that it sets on the upper peak of the signal.

Answer Question 15.

MEASURING FREQUENCY AND PERIOD

To measure the frequency and period of the signal, select “V Bars” from the menu. Now in the upper right of the screen you will see two sets of measurements. The measurements on the right are the values of where the selected cursor intersects the waveform. The measurements on the left give the difference between the cursors and the time position of the selected cursor.

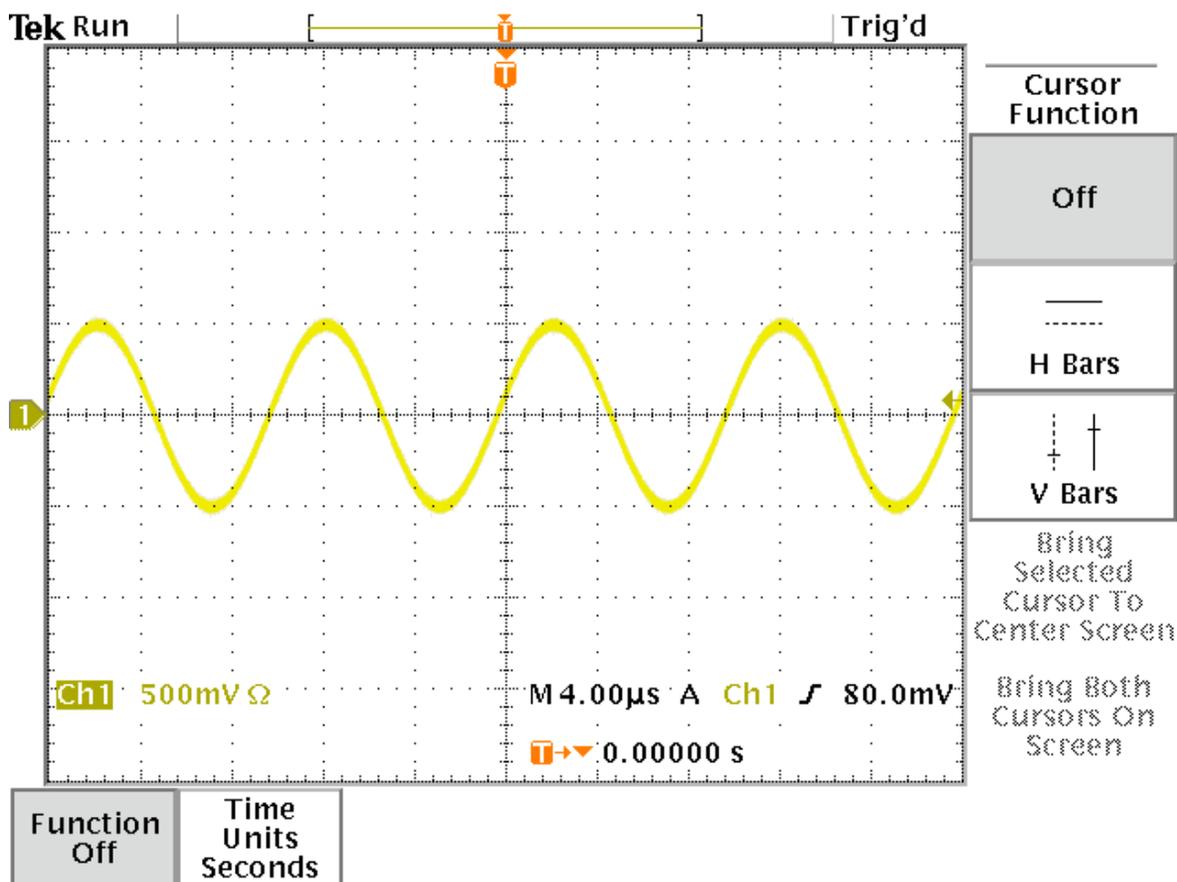


Figure 5. Cursor Menu

Move one of the cursors to the center of a peak. Move the other cursor to the center of an adjoining peak.

Answer Question 16.

To remove the cursors, press “Off” from the menu.

AUTOMATIC MEASUREMENTS

The oscilloscope also has built in functions to measure most display signals. To access the automatic measurements press the MEASURE button located towards the top of the oscilloscope. Pressing the MEASURE button brings up the menu shown in Figure 6. The “-more-” menu option cycles through the available measurements. The first item on the bottom menu cycles through the available channels to make measurements for.

Answer Question 17.

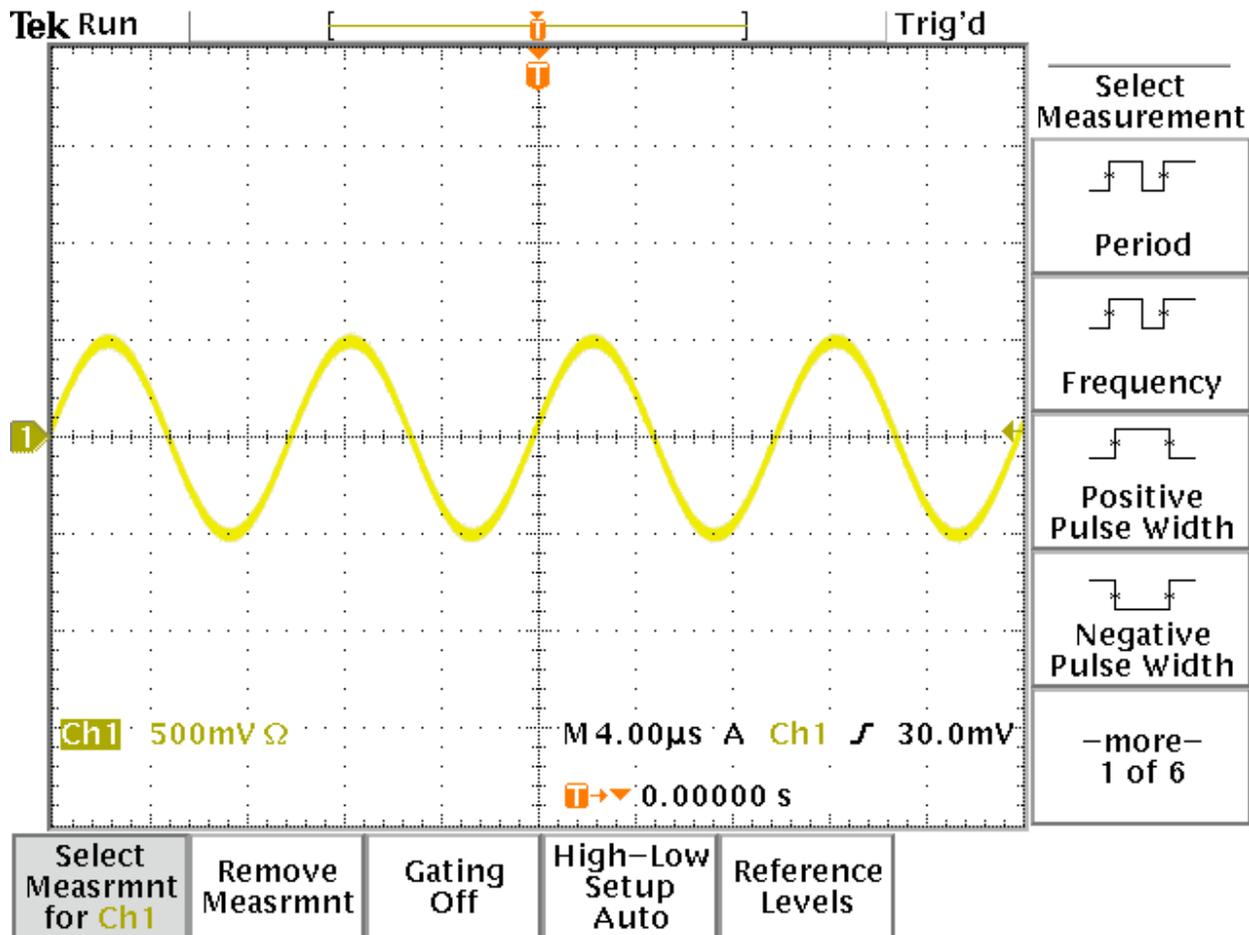


Figure 6. Automatic Measurements.

Remove the measurements from the screen by selecting “Remove Measrmt” and then “Remove All.”

TRIGGER

The oscilloscope trigger stabilizes repeated waveforms and captures single-shot waveforms. The trigger makes repeated waveforms appear static by repeatedly displaying the same portion of the signal. There are four types of triggers: edge, logic, pulse and video.

For this lab, and the labs to follow, you will mostly be concerned with edge triggering. The trigger level and slope controls provide the basic trigger point definition.

The slope control determines whether the trigger point is on the rising (positive) or falling (negative) edge of the signal. The level control determines where on the edge the trigger point occurs.

Answer Question 18 – 20.

INTENSITY: THE Z AXIS

The oscilloscope has a third z-axis, which is the intensity of the signal displayed. The more intense (brighter) the signal appears the greater the value z is. These are the parts of the signal that occur more often. For example, the natural response of a signal will appear to be dimmer than the forced response of the signal.

At maximum intensity, all waveform points are displayed at full brightness. As the intensity is decreased, the brightness of parts of the signal displayed also decrease. This is called intensity grading. The brightest parts are the points most frequently acquired; dimmer parts are less frequently acquired.

Answer Question 21.

CAPTURING DATA

There will be times when you will want to save the information from the oscilloscope. The signals displayed on the screen can be saved as a graphic images, which can be imported into word processors or presentation software, or data files, which can be read by spreadsheet programs or math processors.

DEFAULT FILE NAMES

By default, the oscilloscope saves the image or data file in the following format:

TEK?????.EXT

where

? represents a numeric place holder

EXT is the file extension for the specified file type.

For example, the first file will be named TEK00000.EXT, the second will be TEK00001.EXT and so on.

SAVING AS A GRAPHIC FILE

When saving the signal as a graphic file, what is really being saved is the current screen of the oscilloscope, much the same way that screen capture utilities work in Windows. Whatever is displayed on the screen, including all signals and menus, is saved in the graphic file. The following steps outline how to capture the screen of the oscilloscope (they can also be used to send the screen image to a printer without saving):

1. Insert a 3-½ inch floppy diskette into the floppy drive of the oscilloscope.
2. Press the UTILITY button on the control panel.
3. Select "System Config" from the lower menu and cycle through the options until "Hardcopy" is highlighted.
4. Use the menu on the right side of the screen to select the hardcopy type. This will probably be a graphic format, but it may also be a printer type.
5. From the bottom menu, select "Optionst" to choose from "Portrait" or "Landscape". If you are saving the image as a file, you will want to select "Portrait". Choosing "Landscape" will cause the saved image to be rotated 90° counter clockwise.
6. From the bottom menu, select "Ink Saver" to toggle this option on or off. When the option is on, the background of the image will be white, when off, the background will be black.
7. When done making changes turn the menu off by pressing the MENU OFF button.

After the appropriate settings have been made, the screen may be saved or printed by pressing the "Hard Copy" button (the button with a picture of a printer on it) in the lower left-hand corner of the oscilloscope.

Answer Question 22.

SAVING AS A DATA FILE

When the oscilloscope saves the signal in a data file, it is saving the coordinates of the displayed signal. Each file contains 10,000 data points. There are two data file formats to choose from: a comma-delimited file (spreadsheet) and a data file (Mathcad).

The following steps outline how to save the waveform in a data file to be imported in to spreadsheets, such as Excel, and math processing programs, such as MATLAB:

1. Insert 3-½ inch floppy diskette into the floppy drive of the oscilloscope.
2. Press the SAVE/RECALL button on the control panel.
3. Select the waveform you want to save by pressing the colored channel buttons. For example, to save the waveform on channel 2, press the blue CH 2 button on the control panel.
4. Select "Save Waveform ChX" from the lower menu.
5. Select "To File" from the menu on the right side of the screen.
6. Select either "Spreadsheet File Format" to save the data in a comma delimited file, or "Mathcad File Format" to save the data in a data file.
7. Finally, select "Save ChX To Selected File."

Comma Delimited File

The spreadsheet file is actually an ASCII comma delimited file with the extension CSV. The signal is saved as XY coordinates separated by a comma. The following data values were taken from a sample file:

```
0.002,-1
-0.0019996,1.01
-0.0019992,0.98
-0.0019988,0.99
-0.0019984,0.99
```

The file will consist of 10,000 rows and two columns (a 10,000 X 2 matrix). The first column contains the X coordinate and the second column contains the Y coordinate. The MATLAB **DLMREAD** command can be used to read a comma delimited file.

Answer Question 23.

Data File

The data file is an ASCII text file. Each value is saved on a separate line within the file. The first four lines are parameters, followed by the values for the amplitude of the signal (the Y coordinate).

The four parameters are:

1. The number of data points.
2. Time per point or unit sample.
3. Trigger position.
4. Fractional trigger position.

We will not be concerned with the third and fourth parameters since they are not needed when importing and plotting the data in MATLAB. The following was taken from a sample data file:

```
10000
4e-07
50
0
0.1
0.08
0.1
0.08
```

In this example, the number of data values is 10,000 and the time interval between data samples is 400 ns. The trigger position and fractional trigger position are 50 and 0 respectively. The first data value is 0.1 (with a unit of volts) and starts on the fifth line. The MATLAB **LOAD** command can be used to read this file.

Answer Questions 24.

FLOATING OUTPUTS

Like the arbitrary/function generator, the oscilloscope is earth grounded. This means that both pieces of equipment share a common reference point. Close attention to detail needs to be used when taking measurements of waveforms from the arbitrary/function generator with the oscilloscope. As the following exercise demonstrates, improperly connecting the oscilloscope probe will result in false measurements.

Disconnect the arbitrary/function generator from the oscilloscope and connect it to the circuit you built for your Prelab Exercise. Connect one of the oscilloscope probes to channel 1 of the oscilloscope. Connect the oscilloscope probe tip at node 1 and the reference lead to node 2 of the circuit and measure the voltage across R_1 .

Answer Question 25.

Next, connect the oscilloscope probe with the tip at node 2 and the reference at node 0 and measure the voltage across R_2 .

Answer Question 26.

Switch the leads of the oscilloscope probe so that the tip is at node 0 and the reference is at node 2. Measure the voltage across R_2 .

Answer Question 27.