

Tektronix 2440

Digital Storage Oscilloscope

Operation

This section is composed of two subsections. The first contains basic operating information and techniques that should be understood before you attempt to make measurements with your instrument. The second consists of procedures designed to quickly acquaint a first-time user with all the operating controls and most menu selections.

Operating Considerations

Graticule

The graticule is internally marked on the faceplate of the CRT to eliminate parallax-viewing error and to enable accurate measurements. The graticule is marked with eight vertical and ten horizontal major divisions. Major divisions are further divided into five subdivisions of 0.2 division each, marked along the center vertical and horizontal graticule lines (see Figure 2-1). Percentage marks for rise-time and fall-time measurements are located on the left side of the graticule. The vertical deflection factors and horizontal timing are calibrated to the graticule so that accurate measurements may be made directly from the CRT.

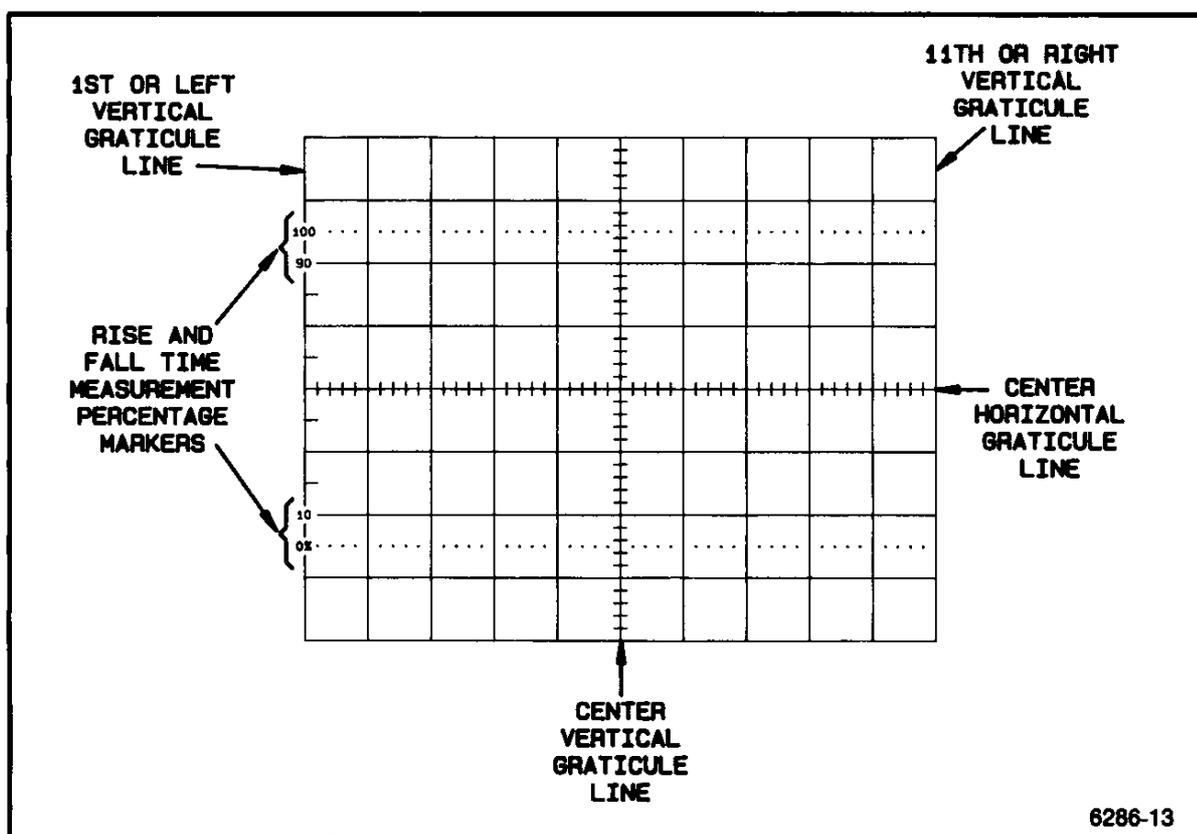


Figure 2-1. Graticule measurement markings.

Time and Voltage Measurements

This instrument provides several methods for making both time and voltage measurements. The various methods produce different degrees of accuracy and require different amounts of time and care.

Using the graticule markings to measure voltage or time produces the quickest, but least accurate, results. To improve the accuracy of this method takes a lot of time, because you must precisely position the waveform on the graticule and carefully count graticule divisions. Direct graticule measurements should be used only where precision is of less importance than speed.

Using the Cursor functions to measure voltage or time on displayed waveforms produces highly accurate, precise results. The setup time to make a particular measurement is fast, and the measurement result is displayed as readout on the ORT. Using the cursors avoids errors due to display gain and ORT trace non-linearity and eliminates the inconvenience of counting and interpolating graticule markings.

The cursors also offer flexibility in that they can be set up to measure either the difference between cursor positions or the difference between the cursor position and an absolute reference point, namely ground for voltage measurements and the Record Trigger Point for time measurements. Two coupled-cursor modes tie each VOLT cursor to a corresponding TIME cursor allowing slew rate to be measured (SLOPE mode) and voltage measurements to be made at chosen time points on the waveform (V@T mode). The 1/TIME-cursor mode is available for making frequency measurements. You can also select the units of measurement to serve a wide range of applications.

The Delay by Time and Δ Time features for the B HORIZONTAL mode are used to make time-difference measurements that require the highest degree of accuracy and/or involve delays beyond the on-screen limits of the TIME cursors. Time measurements using the B Delayed trace take the most time to set up, but they avoid introducing errors due to cursor misalignment at the measurement point on the waveforms.

Details of all the available cursor functions and units are provided in Section 5 “Controls, Connectors, and Indicators.” Use of the various cursor and B Delay modes for making measurements is described in Section 3, “Basic Applications.”

Acquiring Data

Both CH 1 and CH 2 input signals are acquired simultaneously at all times, whether the signal is being displayed or not; therefore, the user can call up the undisplayed channel signal after entering Save mode. Either channel signal may be independently inverted during acquisition.

Waveforms may be acquired in NORMAL, ENVELOPE, or AVG (average) Storage mode. NORMAL produces a “live” trace similar to that seen on a conventional oscilloscope. ENVELOPE mode depicts any variations of the waveshape, since minimum and maximum data-point values for each sample interval are displayed. AVG Storage mode gives a very clean display by averaging out uncorrelated noise. The user can select the number of acquired waveforms to be incorporated into the ENVELOPE or AVG display.

REPET mode may be used with any of the three acquisition modes to extend the Useful Storage Bandwidth to 300 MHz when viewing repetitive signals. REPET is also available for SINGLE SEQ operation, where the actual number of acquisitions occurring for a single sequence is the number required to meet the USB (Useful Storage Bandwidth) for the SEC/DIV setting used and adequately “till in” the waveform for good display. When REPET is not used, the Useful Storage Bandwidth is 200 MHz, and digital interpolation is used to provide the waveform data points between samples taken at the maximum sampling rate (for SEC/DIV settings of 50 ns and faster).

A SAVE-ON- Δ feature enables the scope to compare the incoming signal against a reference envelope waveform. If the signal falls outside the reference envelope, the scope switches to the SAVE Mode to preserve the out-of-limit waveform for analysis. There are two applications in “Storage Applications” (Section 3) that illustrate methods of using this feature, both from the front panel and the GPIB.

Acquired waveforms are also saved when the user presses the front-panel button labeled SAVE. The scope immediately preserves the current CH 1 and CH 2 waveforms and ADD/MULT functions (whether they are displayed or not). All cursor functions can be used to make measurements on the saved waveform displays, and the SAVE waveforms may be horizontally and vertically positioned and expanded. Expansion of SAVED waveforms is nondestructive; it is done as a display function only, so the original waveform may be returned by merely deexpanding the display. Once saved, the waveforms may be transferred to a SAVEREF memory where they may be preserved for extended periods of time. Saved waveforms can be called up at any time for analysis or for use as reference waveforms to compare with the live waveforms being acquired.

Grounding

The most reliable signal measurements are made when the scope and the unit under test are connected by a common reference (ground lead) in addition to the single lead or probe. The ground lead of the probe provides the best grounding method for signal interconnection and ensures the maximum amount of signal-lead shielding in the probe cable. A separate ground lead can also be connected from the unit under test to the oscilloscope ground jack on the front panel using a banana-tip connector.

Signal Connections

Probes

Generally, probes offer the most convenient means of connecting an input signal to the instrument. Furthermore, they are shielded to prevent pickup of electromagnetic interference. The standard 10X probes supplied with this instrument offer a high input impedance that minimizes circuit loading. This allows the circuit under test to operate with a minimum of change from the normal, unloaded condition. Also, the subminiature body of these probes has been designed for ease of use when probing circuitry containing close lead spacing.

The probe and its accessories should be handled carefully at all times to prevent damage. Dropping the probe, or striking it against a hard surface, can cause damage to both the probe body and the tip. Use care to prevent the cable from being crushed or kinked, and do not place excessive strain on the cable by pulling it.

The standard-accessory probe is a compensated 10X voltage divider. It is a resistive voltage divider for low frequencies and a capacitive voltage divider for high-frequency signal components. Inductance introduced by long signal or ground leads may form a series-resonant circuit. This resonant circuit affects system bandwidth and can oscillate (ring) if driven by a signal containing significant frequency components at or near its resonant frequency. Ringing can then appear on the scope display and distort the true signal waveform. Always keep both the ground lead and the probe signal-input connections as short as possible to maintain the best waveform fidelity.

Misadjustment of probe compensation is a common source of measurement error. Because of variations in oscilloscope input characteristics, probe compensation should be checked and adjusted, if necessary, whenever a probe is moved from one oscilloscope to another or between channels of a multichannel oscilloscope. The probe compensation adjustment procedure is found in "Checks and Adjustments" in Section 4 of this manual, and in the instructions supplied with the probe.

AUTOMATIC SCALE FACTOR SWITCHING. The VOLTS/DIV scale factors, displayed on the CRT, can be automatically switched by either GPIB-initiated control changes or by any change in the probe attenuation factor. Table B-1 in Appendix B of this manual shows the range of the VOLTS/DIV switch for all available Tektronix coded probes. The Readout Volt/Div settings given in the section of the table labeled "Expanded" are obtained using firmware data expansion routines to process waveforms acquired at the basic minimum Volts/Div setting of 2 mV. These expanded settings are only available for SAVE and/or AVERAGE mode waveforms.

Coaxial Cables

Cables used to connect signals to the input connectors may have considerable effect on the accuracy of a displayed waveform. To maintain the original frequency characteristics of an applied signal, only high-quality, low-loss coaxial cables should be used. Coaxial cables must be terminated at both ends in their characteristic impedance to prevent signal reflections within the cable. The built-in 50- Ω termination for the input of the scope should be used for interconnection of 50- Ω system signals to the scope. If this is not possible, then use suitable impedance-matching devices.

Input Precharging

When the input coupling is set to GND, the input signal is connected to ground through a precharging network consisting of the input-coupling capacitor in series with a 1-M Ω resistor. Since this allows the input-coupling capacitor to charge to the average DC voltage level of the input signal, it prevents large voltage transients, which can be generated when the input coupling is switched from GND to AC, from reaching the amplifier input. This precharging network also protects external circuits to the extent that it reduces the current levels drawn from the external circuitry during capacitor charging.

External Triggering

The A and the B trigger signals can be independently obtained from a variety of sources. Samples of the CH 1, CH 2, and ADD waveforms are available as trigger sources. Sometimes, however, you may need a trigger source different from the one that corresponds to the input-channel for the signal of interest. In this case, you can apply a trigger-source signal to an unused vertical channel or to either of two external trigger input channels. While the vertical channels can condition a wide variety of signals to produce triggers ranging from millivolts to thousands of volts in amplitude, the external trigger input channels (without the use of external attenuation) can only divide their inputs by either a factor of 1 or a factor of 5.

Operators Familiarization Procedures

Introduction

The Tektronix 2440 is an easy-to-use Digital Oscilloscope that provides you with an accurate and flexible waveform measurement and analysis tool. A combination of front-panel controls and menu-driven selections provides fast and convenient setup of the instrument operating modes. Menu selections allow access to the many waveform acquisition and processing functions while maintaining an uncluttered front panel.

Selected menu functions, front-panel control settings, and measurement results are displayed in the CRT readout. In the menu displays, an underscore marks the active operating mode or processing function. A mode or function is off if there is no underscore beneath its menu entry.

Readout Display

The ORT readout display tells you how the instrument controls are set up; therefore, the rotating switches and control knobs have no physical markings to indicate the control setting. A key to the type of readout information displayed and its location on the screen is illustrated in Figure 2-2.

Front-Panel Controls

The front-panel controls are divided into two types:

- Those that directly affect System Operation (that is, VOLTS/DIV, SEC/DIV, HORIZONTAL and VERTICAL POSITION, and the specific menu selection buttons)
- Those that call up a menu from which you must make a selection to change an instrument function (that is, VERTICAL MODE, CURSOR selections, and most of the TRIGGER selections)

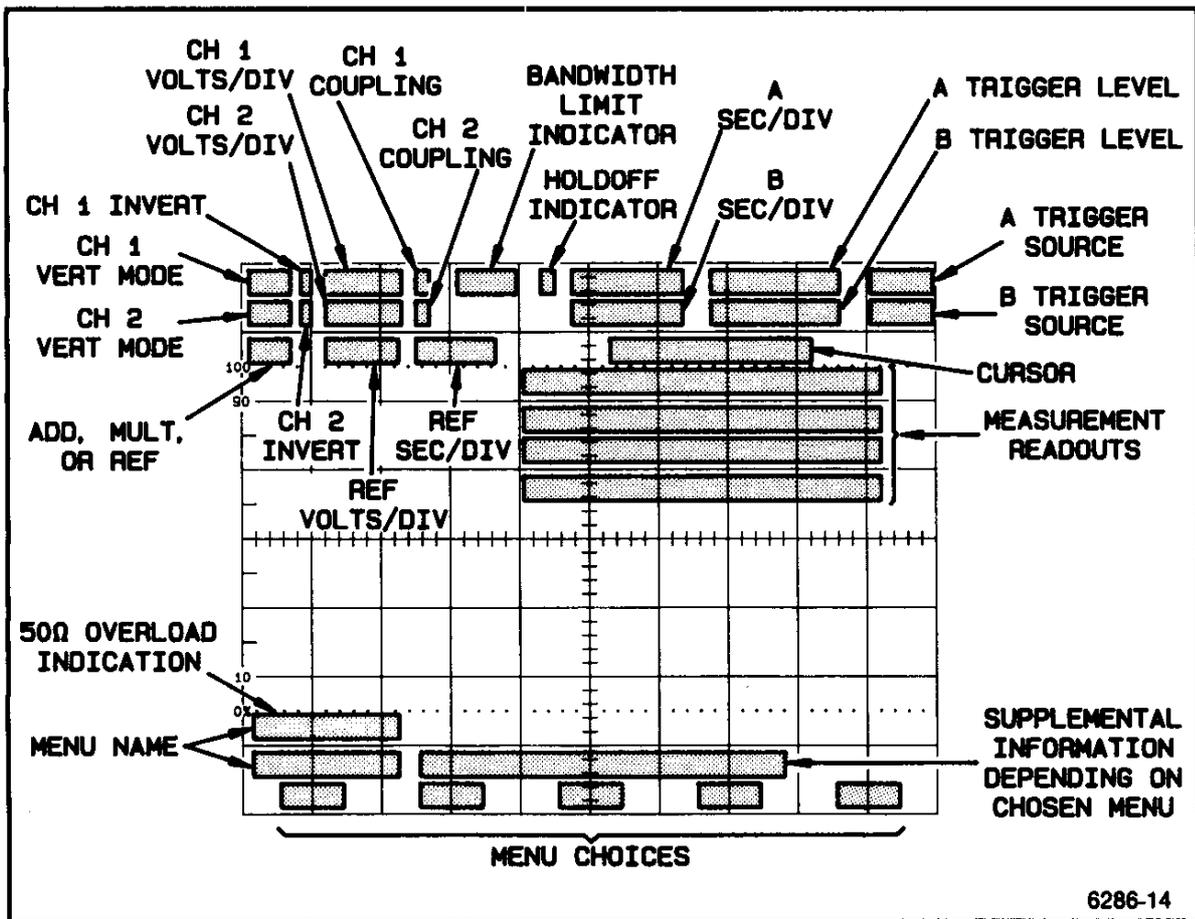


Figure 2-2. Readout display locations.

Menu control buttons work in several different ways as dictated by the type of function they are controlling. In certain instances they merely toggle a function on and off, in others, they are used to make further selections once the main function chosen by that menu button has been made. For some functions, the menu selections are self-canceling, such as when two menu choices are mutually exclusive.

In some menus it is possible, for convenience, to cycle through the displayed choices by repeatedly pressing the front-panel button that called up the menu. For example, while the COUPLING control menu is displayed, you can select among AO, DO, and GND simply by repeatedly pressing the appropriate COUPLING/INVERT button. The following familiarization procedures point out how the various menu control buttons work.

Familiarization Procedures

These procedures will acquaint you with the System menus and front-panel function buttons. By following the step-by-step instructions and performing the simple exercises, you will see how the various controls affect the instrument. Once you understand how the menus control the operating system, and you see how quickly initial set-ups can be made, it should be easy to develop efficient techniques for making specific measurements.

The detailed Operation of each control and connector is described in "Controls, Connectors, and Indicators," in Section 5 of this manual. A complete list of the control menus is included at the end of that section.

Getting a Display

1. With the scope connected to an appropriate power source, push the POWER button IN (green indicator is seen in the button). The scope does a power-on self test each time it is turned on. After a few seconds, the self test will be completed and the instrument will be ready for Operation.

NOTE

If the Instrument fails its power-up self test, see "Start Up" under "Preparation For Use" in Section 1. If the scope powers up in SAVE mode (not acquiring) a message 15 displayed instructing you to push STORAGE ACQUIRE.

After the self test has finished, there may or may not be a visible display. This depends on the front-panel settings in effect when the instrument was powered off, because the same settings are reestablished when the instrument is turned back on. If the READOUT intensity has not been completely turned down, you should see some sort of readout display. If a display source (VERTICAL MODE) is turned on, and the DISPLAY intensity is not turned down, some waveform displays or traces should be on screen. Whether or not there are visible displays on screen, perform the following two steps:

2. Push STATUS/SELECT to set the READOUT INTENSITY to 65%. This level yields viewable readout and menu displays.
3. Push PRGM to display that menu, and push the menu button labeled INIT PANEL. This sets the front-panel to known default settings and yields a display of the CH 1 source.

Remember to use this two-step procedure (Steps 2 and 3) whenever you want to get a visible display.

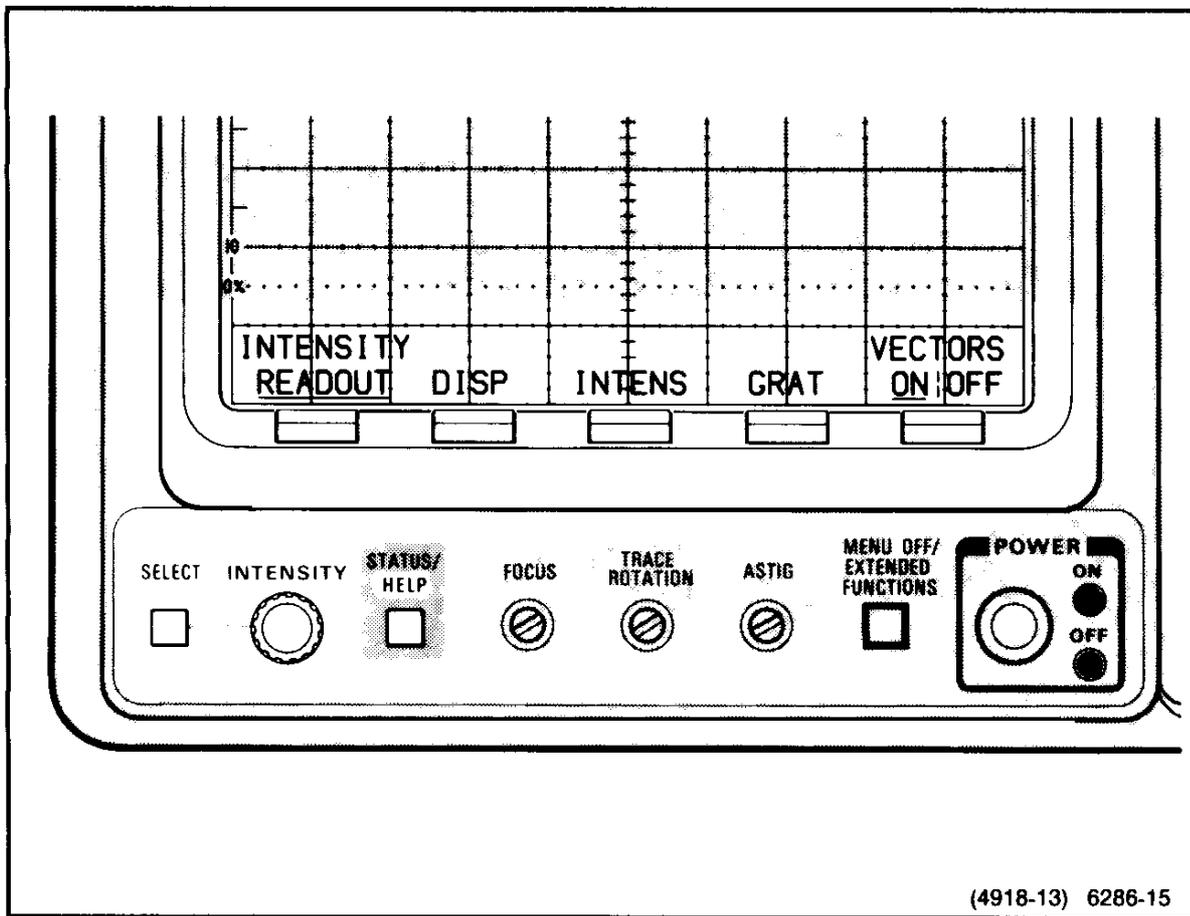
NOTE

While INIT PANEL changes most front-panel controls to predefined states, certain controls (such as those accessed via the EXTENDED FUNCTIONS menus) are not allowed to be changed. See Table B- 15 in Appendix B for a list of the states set up by INIT PANEL.

INIT PANEL can be used to quickly return the instrument to known operating conditions without searching the STATUS display or the menu selections to determine each front-panel control state. (This is useful when the front-panel controls have been left in seldom-used settings.)

If you want to readjust the intensity levels which were established by pushing STATUS and INIT, perform Steps 4-7; otherwise, go to the next procedure, "Front-Panel Setup."

4. Press the SELECT button to display the INTENSITY control menu. Figure 2-3 illustrates the SELECT menu entries and the position of the Menu Control buttons.
5. Press the READOUT menu button and use the INTENSITY control knob to set the readout intensity to a viewable level without excessive brightness. (Clockwise rotation of knob increases intensity.) The INTENSITY control is a continuous-rotation pot with no end stops to designate physical maximum or minimum rotation. You decide by observation when you have reached maximum or minimum intensity.
6. Now select GRATicule and adjust the illumination to the minimum (off) level. GRAT controls the edge lighting of the scribed ORT graticule markings for dimly lighted work areas and oscilloscope photography. The INTENS menu selection is used to adjust the contrast between the normal trace and the intensified zone in A INTEN displays. There's no need to adjust INTENS at this time.
7. Press the DISP menu button and use the INTENSITY control to adjust the waveform trace to a viewable level.



(4918-13) 6286-15

Figure 2-3. SELECT menu and menu control buttons.

Front-Panel Setup

Assuming you performed Steps 1-3 of the previous procedure, you are now ready to set up the remaining front-panel controls for a basic operating mode. Proceed as follows:

1. Push TRIGGER MODE and select AUTO in the displayed Mode menu.
2. Now set up the front-panel shown in the following list. (Some controls may already be set correctly.) In general, the bold-face headings indicate the area on the front-panel where you will find the listed controls, the left column lists the control, and the right column gives its setting. Some controls call up a menu. Remember when using menus: to turn a function on, underline its menu label by pressing the button beneath it; to turn a function off, remove the underline in the same way. If necessary, review the procedure in Section 1 for more detailed instructions.

VERTICAL CONTROLS

MODE
 CH 1 and CH 2 ON (underlined)
 ADD and MULT OFF
 YT | XY YT (toggling choice)

VOLTS/DIV
 CH 1 and CH 2 20 mV

COUPLING/INVERT
 CH 1, CH 2 COUPLING AC
 CH 1, CH 2 INVERT OFF

CH 1 POSITION Set trace to 1.5 divisions above graticule center.
 CH 2 POSITION Set trace to 1.5 divisions below graticule center.
 BANDWIDTH 20 MHz
 SMOOTH ON|OFF OFF
 CH 1 and CH 2 VARIABLE CAL

HORIZONTAL CONTROLS

MODE A
 A SEC/DIV 500 μ s

B TRIGGER CONTROLS (Press A/B TRIG for B Trigger menus)

SLOPE + (plus, indicated by front-panel LED)

MODE

RUNS AFTER ON

EXT CLK OFF

SOURCEVERT (CH 1 is used)

CPLG AC

TRIG POSITION 1/2

EXT CLK OFF

A TRIGGER CONTROLS (Press A/B TRIG for A Trigger menus)

SLOPE + (plus)

MODE AUTO (already set in Step 1)

SOURCEVERT (CH 1 is used)

CPLG AC

TRIG POSITION 1/2

CURSOR CONTROLS

FUNCTION All OFF (none underlined)

DELAY CONTROLS

EVENTS OFF

Δ TIME OFF

STORAGE CONTROLS

ACQUIRE

NORMAL ON

REPET OFF

SAVE ON Δ OFF

DISPLAY REF All OFF (none underlined)

3. Perform the following steps to set up the SYSTEM menu.
 - a. Press the MENU OFF/EXTENDED FUNCTIONS button (located immediately left of the POWER switch) twice. You should get the EXT FUNCT menu.
 - b. Press the bezel button under SYSTEM to get the SYSTEM menu. In the new menu, set PREFLT ON:OFF to ON.
 - c. Press the button under MISC to get the MISC menu; then set BELL ON | OFF to ON and TRIG T ON | OFF to ON.
 - d. Push the MENU OFF/EXTENDED FUNOTIONS button to exit the EXTENDED FUNOTIONS menus.
4. Center the Trigger Position Indicator (a small “T” riding on the CH 1 and CH 2 baseline traces) horizontally on the graticule using the Horizontal POSITION control.
5. Connect the two standard accessory 10X probes to the CH 1 and CH 2 Vertical Input BNC connectors. The CH 1 and CH 2 VOLTS/DIV readouts should now be 200 mV.
6. Connect the probe tips of both probes to the CALIBRATOR output connectors and the ground lead to scope ground. A two-division peak-to-peak display of the CALIBRATOR output signal should now be seen in both channels. The display may or may not be stable depending on the setting of the A Trigger LEVEL control.
7. Use the following procedure to set the A Trigger LEVEL.
 - a. Press the TRIGGER MODE button.
 - b. Press the A/B TRIG button, if necessary, to obtain the A TRIGGER MODE menu.
 - c. Select AUTO LEVEL Trigger mode. Now the trigger level automatically follows trigger signal changes to maintain stable triggering. If you adjust the TRIGGER LEVEL control to set the level beyond the peak-to-peak limits of the trigger signal, the scope automatically changes the level to regain a stable trigger.

You now have a basic front-panel display setup for viewing signals applied to the CH 1 and CH 2 inputs. In the “Getting Acquainted” procedure in Section 1, you saw how Auto Setup was used to perform much the same function; that is, to get the front-panel set up for a usable display. With Auto Setup, the instrument automatically executes many of the steps you have just performed, illustrating the power and convenience of this feature. We'll examine Auto Setup more closely later in this procedure.

Storing Front-Panel Setups

The AutoStep Sequencer function can be used to store single front-panel settings under a label. Let's save the current setup for later use...

1. Press the PRGM front-panel button to display the AUTOSTEP SEQUENCER menu.
2. Press the SAVE menu button. This calls up a submenu for labeling your front-panel setup with a 1-6 character name so it can be recalled later.
3. Create the label FP1 (front-panel 1) for your front-panel setup by using the arrows under ROLL-CHARS as outlined in Steps a-c:
 - a. Select the first character for your label. Press the ↓ to step forward through the alphabet first and then through the digits 0-9. The ↑ steps from 9-0 and from z-a. (There is a "blank space" character between the digit 9 and letter a.)
 - b. When you have displayed the letter or digit for the first character of the label, push CURSOR ⇐ to move to the next character. Repeat Step a to select the letter or digit for the next character of your label.
 - c. Repeat Step b to include up to 6 characters in your label. You can return to any character by continually pushing the cursor button, since it reverses the selection order after the first or sixth character is selected.
4. Press the SAVE menu button to assign the completed label to the current front-panel settings.

NOTE

You can create labels with as few as one character and can leave any character position (1-6) blank. Simply push SAVE when the label has the number of characters you want, in the positions you want them.

5. When you push SAVE, the scope displays a message indicating your chosen label and telling you to set up the controls. You could now change the controls if you wish, but since you already set up the controls earlier in this procedure, just push the front-panel button PRGM to display the ACTIONS menu.
6. The ACTIONS menu allows you to specify different functions to be executed when the front panel is recalled. Since at this point we only want to store our front-panel settings, just push the menu button SAVE SEQ.

To recall any front panel stored, push PRGM and select RECALL from the AUTOSTEP SEQUENCER menu. Next, use the SELECT arrows in the RECALL menu to underline the label for the front-panel desired. Finally, press RECALL to implement the selected setup.

Performing SELF CALibration

The SELF CAL feature assures you that the most accurate measurements possible are being made. Self Calibration should be performed after instrument warm-up, whenever the ambient temperature changes by more than $\pm 5^{\circ}\text{C}$, and immediately prior to making a series of measurements when the highest level of accuracy is required.

NOTE

For about ten minutes after power-on (whether the instrument is warm or not), the message NOT WARMED UP is displayed in the CAL/DIAG menu. This message warns you that the temperature of the scope may not be stabilized. The message can be ignored and the SELF CAL procedure initiated at any time, but optimum calibration results are obtained after the temperature is stabilized and the message is removed.

Let's do a SELF CAL of the scope...

1. Push the MENU OFF/EXTENDED FUNCTIONS button twice (the first time to turn off the menus, the second time to turn on the EXT FUNCT menu).
2. Press the CAL/DIAG menu button to display the Calibration/Diagnostic menu.
3. Press the SELF CAL menu button to start the calibration; the message "RUNNING" should appear in the menu display. After a few seconds the self calibration is completed ("RUNNING" message leaves the display) and a PASS message should be above the SELF CAL label. The scope is then ready to return to its operating state.

NOTE

If the self calibration fails, the self-diagnostic mode is entered. In this event, push MENU~ OFF/EXTENDED FUNCTIONS twice and repeat Step 2 to rerun the self calibration. If errors persist, the scope should be referred to a qualified service person. Any fatal test errors should also have caused a failure of the power-on self test when the scope was first turned on. See Appendix A for more

information on the Self Test and Self Calibration features.

Depending on the test failed, the scope may function adequately for the measurements you need to make. Press the MENU OFF/EXTENDED FUNCTIONS button to exit the error display, and check the scope operation to determine if it will function for your purposes. In any event, the instrument should be referred to a qualified service person at the first opportunity.

4. Press MENU OFF/EXTENDED FUNCTIONS to turn off the Calibration/Diagnostics menu, then push ACQUIRE to start the waveform acquisitions again. Note the scope 5 returned to the setup you created prior to executing SELF CAL. This is a characteristic of SELF CAL Operation.

Since you won't need CH 2 displayed for the next part of the procedure, use the VERTICAL mode menu to turn it off. Then center the CH 1 display vertically on screen.

Using the SEC/DIV Control and a Horizontal Graticule Measurement

1. Turn the SEC/DIV control slowly clockwise through the settings to 500 ns, then counterclockwise back to 500 μ s. See how the A SEC/DIV readout changes and note the effects on the CALIBRATOR waveform.

Notice that the CALIBRATOR output frequency changes with the SEC/DIV switch setting every 3 settings between a maximum and a minimum output frequency. See Table B-2 in Appendix B for the CALIBRATOR output frequency for each SEC/DIV setting.

2. Now check the period of the CALIBRATOR signal by determining the time of one complete cycle using the following procedure.

a. Use the Horizontal POSITION control to align the beginning of a cycle (the negative-to-positive rising edge) with any convenient vertical graticule line, and determine the number of horizontal divisions needed for one complete cycle of the CALIBRATOR signal. The center horizontal graticule line is graduated in 0.2 division increments to help you interpolate between the large division markings.

b. Multiply the number of divisions (and/or decimal fraction parts of a division) by the SEC/DIV readout to calculate the CALIBRATOR signal period. Frequency is calculated by taking the reciprocal of the period. Since you set the SEC/DIV back to 500 μ s in Step 1, the period should equal approximately 2 ms and the frequency, 500 Hz.

Using CH 1 Controls and a Vertical Graticule Measurement

1. Set the CH 1 VOLTS/DIV control clockwise to 50 mV, then switch slowly through settings counter clockwise to 1 V. Note the effect on the VOLTS/DIV readout and the waveform amplitude.

NOTE

Between 500 mV and 1 V per division, the attenuator switch activates with a clicking sound.

2. Set the CH 1 VOLTS/DIV control to 100 mV for a four-division peak-to-peak display.

3. Use the graticule division markings and the VOLTS/DIV setting to determine the peak-to-peak voltage of the CALIBRATOR signal in the following manner:

a. Align a peak of the display with any convenient horizontal graticule marking to determine the peak-to-peak amplitude in divisions. The center vertical graticule line is graduated in 0.2 division increments to help you determine fractional parts of the major divisions.

b. Calculate the peak-to-peak amplitude of the CALIBRATOR signal by multiplying the number of divisions (and/or decimal fraction part of a division) by the VOLTS/DIV readout. (You should get 400 mV.)

4. Press CH 1 COUPLING/INVERT button to display the CH 1 COUPLING menu. Additional pushes of the button will rotate the input COUPLING selections first to DC, then GND, then back to AC. Watch the vertical position of CH 1 change as you switch between AC and DC. Also, note that the symbol displayed with the VOLTS/DIV readout changes with each COUPLING selection.

You can also use the menu buttons to select any of the functions in the COUPLING menu, including the 50- Ω termination and INVERT features.

NOTE

AC COUPLING and 50- Ω input termination are mutually exclusive; selecting one will deselect the other.

5. Select 50- Ω input termination for Channel 1. Observe that the COUPLING switches from AC to DC and the Ω symbol is displayed following the CH 1 VOLTS/DIV readout. (The signal display amplitude will drop to zero in 50- Ω termination as the CALIBRATOR signal is dropped across the 10X high impedance probe.)

6. Again select AC and observe that the 50- Ω termination is turned off.
7. Press the CH 1 VARIABLE button to display the VARIABLE menu. Press and hold the \downarrow menu button until the displayed peak-to-peak amplitude decreases to about 1 division. Also note the symbol preceding the VOLTS/DIV readout indicates when the CH 1 display is uncalibrated.
8. Press and hold the \uparrow menu button to increase the display amplitude back to about 1.5 divisions peak-to-peak.
9. Return to the calibrated VOLTS/DIV settings by pushing the CAL menu button.

Using SAVE and DISPLAY REF Storage Modes

SAVE mode is normally entered in one of three ways: pushing the SAVE Storage mode button, as a result of a SAVE ON Δ or at the end of a SINGLE SEQ acquisition.

SAVE Mode freezes any waveform acquisition in process and holds the waveform displayed for saving as a reference, making any type of measurements needed, or outputting via the GPIB. SAVE mode may be entered using a command via the GPIB.

Displayed along with the SAVED waveform are a count of the number of acquisitions made in the current process before SAVE was entered and a real-time clock display. The time in HRS (hours) is the scope run-time since the last COLD START. (The two least-significant digits, right of the colon, indicate minutes.)

Upon entering the SAVE Storage mode, the SAVEREF SOURCE control menu is displayed. The menu permits the selection of any displayed VERTICAL MODE signal as the source of the reference signal to be stored. Once a source is selected, the SAVEREF DESTINATION menu is displayed so the user can choose which of the four reference memories is to store the selected source.

Besides storing selected VERTICAL MODE signals, the SAVEREF SOURCE menu can be used to copy a stored REFERENCE waveform to another memory location. If REF is selected for the source, a menu is displayed to allow one of the four available reference waveforms to be selected. Once the REF is selected as a source, the SAVEREF DESTINATION menu appears as before for selection of the REF destination into which the REF source is to be copied.

The SAVEREF SOURCE menu provides a third Option for selecting REF sources and destinations. The STACK REF selection treats the reference memories as a push-up stack. REF1 is the bottom stack location and REF4 is the top. The first push of STACK REF stores a single-channel display first into REF1, then repeated pushes moves it to REF2, then REF3, then REF4, and finally off the stack. Previously stored waveforms are pushed ahead toward the top of the stack with each push. When more than one waveform is being displayed, a predefined storage plan is used to place selected waveforms in certain reference memory locations. Basically, if CH 1, CH 2, and either ADD or MULT are displayed, pushing STACK REF will store CH 1 in REF1, CH 2 in REF2, and the ADD or MULT function in REF3 or REF4. See Table B-1 3 in Appendix B for the detailed STACK REF storage arrangement.

1. You should have a display of CH 1 centered on screen from the previous procedure.
2. Press the SAVE Storage mode button. This freezes the waveform acquisition and displays the SAVEREF SOURCE menu.
3. In the SAVEREF SOURCE menu, push CH 1. The Channel 1 signal is now selected as the source of the reference waveform to be stored, and the SAVEREF DESTINATION menu is displayed to select the SAVEREF memory location to store it.
4. Press the REF1 menu button. The Channel 1 signal is now stored in reference memory 1, and the SAVEREF SOURCE menu returns for further source selections.
5. Now select REF as the source choice. The four reference locations are now displayed so you can choose which reference memory you want as the source.
6. Select REF1 as the source. In the SAVEREF DESTINATION menu that then appears, push REF3 as the storage location. You have now copied the REF1 waveform into the REF3 reference memory.
7. To display the stored references, push the DISPLAY REF button. This calls up a menu so you can choose the reference waveform for display. A REFERENCE memory with no waveform stored in it will be labeled EMPTY. If the empty memory is turned on for display, an invalid waveform is displayed. An invalid waveform has vertical "fill" areas alternated with short trace segments.
8. Press REF1 and REF3 to display those reference waveforms superimposed. Use the VERTICAL MODE menu to remove the CH 1 SAVED waveform from the display, then push DISPLAY REF again to return that menu to the display.

NOTE

The fact that both waveforms are displayed can be deduced by noticing that both REF1 and REF3 must be turned off before the waveform disappears from the display. Also, although REF waveforms cannot be positioned vertically, they can be horizontally positioned independently and in unison, as we will see.

9. Press the HORIZ POS REF menu button and set REF HPOS INDLOOK to LOOK (if not already on). Rotate the horizontal position control to simultaneously move all REF waveforms (whether displayed or not) with SAVED or "live" VERT MODE waveforms.

10. Set REF HPOS to IND to unlock the positioning. Press REF3P, then rotate the Horizontal POSITION control. Note that it now positions the REF3 waveform independently of other waveforms.

NOTE

REF waveforms can be horizontally positioned only when the HORIZ POS REF menu is displayed. One and only one REF is always selected, as indicated by the underline. (If INDLOCK is set to LOCK, it doesn't matter which REF is selected since the REF waveforms are positioned in unison with each other and the VERT mode waveforms, whether displayed or not.) The HORIZONTAL POSITION menu must be displayed for the Horizontal POSITION control to position a reference waveform either independently or in unison.

11. Select REF1 P to be positioned horizontally, and position that reference waveform using the Horizontal POSITION control.

Although we could use the DISPLAY REF menu and the ACQUIRE front-panel button to turn off the REF waveforms and return to “live” acquisition mode, let’s use the stored front panel we saved earlier in this section...

12. Recall the stored dual-channel front panel by doing the following:

a. Push PRGM and select RECALL from the SEQUENCE menu.

b. Use the SELECT arrows in the RECALL menu to underline the label (FP1) of the sequence you created for the dual-channel setup.

Press RECALL to implement the selected setup.

Using Dual-Channel Displays

1. Press VERTICAL MODE and select ADD. Three displays should now be present: CH 1, CH 2, and ADD. Observe that the CH 1 and CH 2 SAVED signals are digitally added together; it is not necessary to acquire a signal in ADD Vertical mode to obtain the ADD display.

2. Turn off the CH 1 and CH 2 displays. Check that the ADD display is four divisions in amplitude.

3. Use the CH 1 and CH 2 Vertical POSITION controls to observe that both controls position the ADD trace.

4. Press CH 1 COUPLING/INVERT panel button and turn CH 1 INVERT ON. Observe that the ADD display, the CH 2 signal minus the CH 1 signal, is reduced to approximately a baseline trace.

5. Press the CH 2 COUPLING/INVERT panel button and turn CH 2 INVERT ON. Observe that the ADD trace, the inverted CH 1 signal plus the inverted CH 2 signal, returns to four divisions in amplitude.

6. Turn on the CH 1 and CH 2 displays.

7. Press the SAVE button, then push the STACK REF button in the displayed menu. STACK REF treats the SAVEREF memories as a stack and automatically saves the CH 1 signal in REF 1, the CH 2 signal in REF 2, and the ADD waveform in REF 3.

8. Press the Vertical MODE button and turn off ADD.

9. Switch the display to XY mode and observe the display of CH 1 versus CH 2. In an XY display, the CH 1 signal is supplying the X-axis (horizontal) deflection and CH 2 is supplying the Y-axis (vertical) deflection. Therefore, the CH 1 Vertical POSITION control moves the display horizontally and the CH 2 Vertical POSITION control moves the display vertically. The Horizontal POSITION control does not position the XY display, but it does control which 512 data points of the 1024 data point record are displayed.

10. Return the display mode to YT and turn off CH 2.

Review of ENVELOPE and AVG (Average) ACQUIRE Modes

If you performed the introductory procedure in Section 1, this next exercise is a review and you can skip ahead to “Using SINGLE SEQ Trigger Mode” without missing information.

1. Press the ACQUIRE panel button. Next, press the ENVELOPE menu button several times, changing the number above the label each time. (This is how you vary the number of acquisitions that contribute to the ENVELOPE display before the sequence resets.)

2. Continue to press the ENVELOPE menu button until CONT (continuous) appears above the label. Now the instrument saves the cumulative maximum and minimum values for each sample point without resetting.

3. Use the CH 1 Vertical POSITION control knob to move the waveform up and down. Observe how the display grows vertically to approximate the effect of amplitude changes and dc level shifts in the incoming signal.

4. Now push the ACQUIRE panel button again to erase the continuous ENVELOPE display and start the acquisitions for a

new ENVELOPE sequence.

5. Select BANDWIDTH and change the menu setting to FULL. Press ACQUIRE to return to that menu.
6. Select AVG (Average) acquisition mode and vertically reposition the trace to center screen. Press and release the AVG menu button until the number 2 appears above the label.
7. Repeatedly press the AVG menu button, stepping through the range from 2 to 256. Notice that the displayed waveform becomes cleaner as the number of averaged waveforms increases. This shows how Average acquisition mode improves the signal-to-noise ratio of the displayed waveform. (Table B-3 in Appendix B gives the expected improvement ratio versus the number of averages.) Return the AVG setting to 16.
8. Rotate the CH 1 Vertical POSITION control a small amount and observe that averaging for the display starts over (as seen by the increased noise).

NOTE

Any change in a front-panel control that affects the waveform data being acquired causes the AVG acquisition sequence to start over. Pressing the ACQUIRE button also starts a new acquisition sequence.

Using SINGLE SEQ Trigger Mode

With the SINGLE SEQUENCE mode, you can select an acquisition process that, when completed, does not start again until you direct it to. As the SINGLE SEQ acquisition completes, the scope switches to the SAVE Storage mode to freeze the waveform display. That waveform may then be saved for reference, transmitted to a data collection device, or analyzed as required before you start another acquisition. Let's use AVERAGE mode (ENVELOPE could also be used) to explore this feature...

1. Set the ACQUIRE Mode to AVERAGE 64.
2. Set the A TRIGGER MODE to SINGLE SEQ.
3. Watch the Trigger Status Indicators (TRIGED, READY, and ARM). When they stop flashing (or the TRIG'D indicator light goes out, depending on the SEC/DIV setting) and the SAVE mode is entered, the single-sequence acquisition of 64 averages is complete.
4. Press ACQUIRE to restart the SINGLE SEQ, and again watch as the Trigger Status indicators flash (or TRIG'D light remains on solidly) until the single sequence has completed.

You could STORE this saved waveform, if needed as a reference, in SAVEREF memory using the process you learned previously in the demonstration about SAVE and DISPLAY REF modes.

Using the Cursors

Volts Cursors

Leave the display as set up from the SINGLE SEQ demonstration to start this part of the procedure; you will make the measurements on the SAVED CH 1 waveform.

Remember, in Section 1, how we measured the amplitude of the calibrator using the CURSOR FUNCTIONS? Let's review that procedure and also look at some other uses and modes for the CURSORS...

1. Press the CURSOR FUNCTION button, then select VOLTS to make voltage measurements.
2. Press the CURSOR UNITS button to call up the UNITS menu and select VOLTS for units. Set the Δ | ABS menu choice to A.

Delta mode (Δ) provides two cursors for voltage difference measurements. ABS cursor mode provides a single cursor, referenced to the ground-position indicator.

3. Move CURSOR/DELAY knob clockwise and counterclockwise. Observe how the "active" (dashed line) cursor is positioned. Set the active cursor to the top of the CALIBRATOR signal waveform.
4. Press the SELECT button and see that the second cursor becomes the active one. Position it to the bottom of the waveform display. The cursor readout now indicates the CALIBRATOR amplitude.

Now let's look at some ABS (absolute) measurements...

5. Switch Δ | ABS to the ABS cursor mode. Only one cursor will be displayed.
6. Position the VOLTS cursor to the ground indicator (a small \oplus at the left edge of the screen) and observe that the readout is 0 volts when exactly aligned with ground. Cursor measurements in ABS mode are taken relative to ground level.

7. Set the VOLTS cursor to the positive peak of the waveform to see that the readout is positive (above ground) and equal to about 1/2 the total amplitude of the square wave. Position the cursor to the negative peak. Notice the readout is now negative (below ground) and has about the same absolute value (as you would expect with an ac-coupled, square-wave CALIBRATOR signal).
8. Switch back to Delta mode cursors and align the cursors with the top and bottom of the waveform as you did in Steps 3 and 4.

This instrument lets you store the difference between the cursors for use as a reference. You can then make ratiometric comparisons between the stored value and other voltages measured with the cursors. Let's see how it's done...

9. Press the NEW REF menu button. You have just saved the present VOLTS cursor difference (approximately 400 mV) as the reference level for making percentage and dB measurements.

As long as you leave VOLTS selected as the units, the readout indicates only the present voltage difference between the cursors. When % (percent) or dB (decibels) is selected, the measurement function becomes ratiometric.

10. Select % units. The VOLTS cursor readout should now read 100%.
11. Move the active cursor to the vertical center of the waveform.

Notice the readout changes to about 50%, indicating the cursors are measuring about 1/2 of the total amplitude of the CALIBRATOR signal which was stored as a NEW REF. If you position the cursor so it's separated by more than 4 divisions from the other cursor, the readout exceeds 100% since the amplitude is greater than that stored for NEW REF. See "Cursor Measurements" in Section 3 for more information on making ratiometric measurements.

V@T Cursors

These coupled cursors provide voltage readout for VOLTS cursors that are confined within the amplitude extremes of the waveform. Each VOLT cursor (one for ABS, two for Δ) reads the amplitude of the waveform at the point where its coupled TIME cursor is positioned. Since you can't move the TIME cursors horizontally off the waveform, measurements using these coupled cursors are limited to the waveform amplitude. As with the VOLTS FUNCTION, alternative units of measurement are % and dB.

1. Set the CURSOR UNITS to VOLTS in the ~ cursor mode. Press CURSOR FUNCTION and select V@T.
2. Rotate the CURSOR/DELAY control to move the active cursor pair over several cycles of the waveform. Did you notice that the active (segmented) vertical cursor moves to the amplitude level corresponding to the placement of the active horizontal cursor? Also, notice that the active vertical cursor does not leave the waveform.
3. Position the active cursor pair to read the peak amplitude of a positive peak of the CALIBRATOR signal.
4. Press SELECT and position the second TIME cursor to read the peak amplitude of the negative peak.
5. Read the peak-to-peak voltage of the waveform.
6. Press UNITS to display that menu. Push the NEW REF button, set the CURSOR UNITS to ABS mode, and select % units for the measurement.
7. Read the peak amplitude as a percent of the reference you set in Step 6.
8. Position the cursor pair to the opposite peak of the signal and read the percentage. With AC input coupling, the percentage difference is a measure of the nonsymmetry of the CALIBRATOR square wave. (The difference is typically small.)

SLOPE Cursors

SLOPE cursors behave just like V@T cursors. The readout is given in VOLTS/SEC to indicate slew rate or slope (rate of voltage change with time). Another unit of measurement for SLOPE cursors is percentage, used when comparing against a reference slope. SLOPE measurements require that delta cursors are on at all times; therefore, the Δ | ABS label is omitted from the SLOPE UNITS menu.

The CALIBRATOR signal is a poor signal source for displaying SLOPE measurement unless the SAVE horizontal expansion is used. That is because the CALIBRATOR signal frequency changes for different settings of the SEC/DIV switch. The following procedure simulates an increased rise-time signal for demonstrating the SLOPE cursors.

1. Select SLOPE in the Cursor FUNCTION menu; then select SLOPE in the Cursor UNITS menu.
2. Use the Horizontal POSITION control to place the Trigger Point Indicator T, and the rising edge of the CALIBRATOR signal, on the center vertical graticule line.
3. Use the SELECT button and the Cursor Position knob to place both time markers of the SLOPE cursors on the trigger point.

4. The scope should still be in SAVE from the previous procedure. Advance the SEC/DIV switch to 5 gS. This expands the display by a factor of 100 times and produces a display with noticeable rise time.
5. Position the SLOPE cursors to bracket a linear portion of the leading edge and check the SLOPE readout.
6. Change the position of the SLOPE cursors to bracket a portion of the waveform with a different slope and observe how the readout varies.
7. Switch the SEC/DIV setting back to 500 μ s.

NOTE

The user must determine the sign of the slope from observing the waveform.

TIME Cursors

TIME cursors enable rapid measurement of period, pulse width, or time difference in seconds. Alternative units, used for making ratio-metric measurements, are % (percent) and DEGREES. Choose % from the UNITS menu to express your measurement as a percentage of the reference value; choose DEGREES to find phase differences between your measurement and the reference.

1. Press the CURSOR FUNCTION button and select TIME cursors.
2. Press the CURSOR UNITS button and select SEC for the time readout units.
3. Position the active cursor (the cursor with the most dots) to one of the rising edges of the displayed CALIBRATOR square-wave signal.
4. Press SELECT to activate the other cursor and position it to the next rising edge (either left or right, as convenient).

NOTE

The waveform record is 1024 data points long; the display is 500 data points long. Since cursors may be positioned anywhere within the record length, the TIME cursors may be used to scroll through the complete record merely by positioning the active cursor from one end to the other.

5. Read the time of one period of the CALIBRATOR signal.
6. Press NEW REF to set the reference value to the period of the CALIBRATOR signal.
7. Now select the % units for the TIME cursor readout. Observe that the readout is 100%.
8. Measure the percentage of the first half cycle of the CALIBRATOR period compared to the whole period. Then measure the second half cycle. Are both half cycles of the CALIBRATOR square wave equal? (There is usually a small difference because the CALIBRATOR signal is not perfectly symmetrical.)
9. Position the TIME cursors for 100% at the original measurement points and select DEGREES units for the TIME cursors readout. Observe that the readout switches to 360 (one complete period = 360) when the cursors are correctly aligned.
10. Position the active cursor to the falling edge at the center of the CALIBRATOR signal period. Observe that the readout is approximately 180°.
11. Select SEC units for the TIME cursors and switch to ABS cursor mode.
12. Position the displayed cursor (only one in absolute mode) to the Trigger Point Indicator. Use the TIME cursor to scroll the display if the Trigger Point Indicator is not presently displayed near center screen horizontally.
13. Position the cursor to the left and to the right of the Trigger Point Indicator. Time is measured relative to that trigger point, and the time readout is negative when the TIME cursor is positioned before (to the left of), and positive when positioned after (to the right of), the Trigger Point.

1/TIME Cursors

When you select 1/TIME as the Cursor function, Hz replaces SEC in the UNITS menu. You can now conveniently measure a signal's frequency because the scope automatically performs the calculation that converts seconds to hertz. If you want to express your measurement's frequency as a percentage of the reference frequency, choose % from the UNITS menu. For phase measurements, choose DEGREES; they work exactly the same as with TIME cursors.

1. Use PRGM to recall the front-panel setup FP1. (See Step 7 of "Storing Front-Panel Setups.")
2. Select 1/TIME cursors from the CURSOR FUNCTION menu.

Notice that when you selected the Cursor function, you also called up a target menu for CH 1 and CH 2. Whenever more than one

signal is displayed, the target menu lets you specify the source to be used for cursor measurements. The readout for cursor measurements can then be scaled to take into account the settings (VOLTS/DIV, etc.) of the specified source.

Since the measurement is 1/TIME, and both CH 1 and CH 2 are acquired and displayed at the same SEC/DIV setting, the target selected in this case doesn't matter.

3. Push CURSOR UNITS and set to Hz.
4. Position the 1/TIME cursors to bracket one full period of the CALIBRATOR square wave. Verify that the frequency readout is very near 500 Hz.
5. Press NEW REF to replace the reference with the newly-measured frequency. Set the UNITS to % and check that the readout is 100%.
6. Set the SEC/DIV switch to 5 ms and, if necessary, reposition the CURSORS to define one full period of the displayed CALIBRATOR square wave.
7. Observe that the percentage readout is 10%, indicating that the CALIBRATOR signal frequency is now 10% of the reference frequency.
8. Switch UNITS to Hz and read the frequency ($500 \text{ Hz} \times 10\% = 50\text{Hz}$).

DELAY Features

DELAY by TIME

The Delay-by-Time function is used with the A INTEN and B Horizontal modes. A INTEN mode is used to locate areas of interest within the A waveform record for closer examination using B Horizontal mode. Using z~ TIME delay mode, precise timing measurements can be made between two points on a single channel (for pulse width and rise-time measurements) or between single points on different channels (for propagation-delay measurements).

Delay time may be set to many times the B SEC/DIV setting (5242.8 times for most B SEC/DIV settings--see Section 6). This means that B Delay acquisitions are not confined to within the time set by the A time base, and that the intensified zone will not be present on the A waveform record if the delay is set to more time than the total A record length. We start the procedure with setup conditions that set the delay time for a visible intensified zone on the A waveform record.

1. Recall the front-panel setup FP1.
2. Press the DELAY by TIME button. Then turn and hold the Cursor/Delay control fully counterclockwise to set the delay time to its minimum setting, as shown in the DELAY TIME readout.
3. Select A INTEN Horizontal mode and turn the A and B SEC/DIV knob clockwise until the B acquisition rate is 50 μ s.
4. Press the intensity SELECT button and switch between DISP and INTENS, using the INTENSITY knob to adjust the levels for a visible intensified zone on the trace.
5. Press the DELAY TIME button again to return the delay time readout to the CRT.
6. Hold the Horizontal POSITION control counterclockwise to position the end of the waveform record to center screen.
7. Turn and hold the Cursor/Delay control clockwise to increase the delay - time setting. Observe that the intensified zone moves off the A trace when the delay time exceeds the time between the A trigger point and the end of the A record.

NOTE

If the intensified zone is not present on the A trace in A INTEN Horizontal mode, it might be that the delay time exceeds the time from the A trigger point to the end of the A record.

8. Recenter the A trigger point marker on the center vertical graticule line.
9. Reduce the delay time to minimum to bring the intensified zone on the display.
10. Select B Horizontal MODE and observe the B waveform record acquired at 50 μ s/div.
11. Rotate the Cursor/Delay control to observe the effect on the position of the delayed waveform.
12. The delay time readout is the amount of elapsed delay from the A Record Trigger to the B Record Trigger. Decrease the delay time to minimum.
13. Turn on Δ TIME delay mode and select A INTEN Horizontal mode again.
14. If necessary, press the TIME button to underline the Δ DELAY TIME menu label and set the corresponding delay time to

minimum.

15. Set the B SEC/DIV to 10 μ s.
16. Now increase the Δ DELAY TIME to about 2 ms, positioning the second intensified zone on the rising edge of the next cycle of CALIBRATOR signal to the right of the A Record Trigger point.
17. Switch to B Horizontal mode, and use the Cursor/Delay control to precisely align the two leading edges. (This is a time adjustment, and you may need to increase the intensity to make the leading edges visible. Also, there will be a little trigger jitter.) You have now precisely measured the period of the CALIBRATOR square wave as indicated by the Δ DELAY TIME readout.
18. Turn Δ TIME off. The DELAY TIME readout is the time elapsed from the A Record Trigger to the B Record Trigger.
19. Hold the Cursor/Delay control clockwise until you reach 52.429 ms, which is the maximum delay time at 10 μ s per division. (If Δ TIME were on, it would be the maximum total delay of DELAY TIME plus Δ DELAY TIME.)
20. Switch the B SEC/DIV setting to 5 μ s. Observe that the DELAY TIME readout is reduced to 26.214 ms, the maximum delay time possible for 5 μ s per division.

NOTE

When dealing with long delays at a particular B SEC/DIV setting, switching to the next faster B SEC/DIV setting will cause the DELAY TIME setting to be limited to the maximum for that SEC/DIV setting. The delayed waveform will be relocated in time, and you must reset the DELAY TIME to the desired value when switching back to the slower SEC/DIV setting. Also, if ~ 1 DELAY is on and the sum of the DELAY TIME plus the Δ DELAY TIME reaches the maximum limit, any further increase in the DELAY TIME setting causes the ~ 1 DELAY TIME setting to decrease (down to zero if the DELAY TIME is increased to its maximum limit).

DELAY by EVENTS

With this delay feature, you can delay the A Record Trigger by a selected number of B-trigger events. Since the B-trigger circuitry is the source of the events, proper B-triggering conditions must be set (LEVEL, SOURCE, CPLG, etc.).

1. Recall front-panel setup FP1. The initial setup conditions saved for the B trigger are: MODE—RUNS AFTER; SOURCE—CH 1; OPLG—AC. Verify the trigger conditions by pushing the TRIG STATUS front-panel button.
2. Press A/B TRIG to display the B Trigger Level readout and set the LEVEL for 0 V. A level of 0 V with AC trigger coupling sets the level in the middle of the trigger signal and assures that triggering occurs.
3. Set the A SEC/DIV switch to 50 μ s. This yields a high enough CALIBRATOR signal frequency so that you won't have to wait very long for all the events to occur when the EVENTS COUNT is high.
4. Press the DELAY EVENTS button and turn EVENTS ON.
5. Use the Cursor/Delay control to set the EVENTS COUNT to 1, if not already there. The count will wrap around the end counts from minimum to maximum or maximum to minimum if the control knob is held in the rate region for a moment after the end count has been reached.

Notice the rate at which the display is continuously updating. In the next three steps, watch the effect on the update rate as you increase the EVENTS COUNT.

6. Increase the EVENTS COUNT number up about 1000 counts. Notice that the update rate is slowed slightly.
7. Increase the count to about 10,000 counts. Notice that now the display takes several seconds to update. Notice also that the Trigger READY indicator can be seen slowly flashing; its rate indicates the length of time between A acquisitions. A new waveform is acquired only after the set number of B-trigger events has been counted.
8. Increase the EVENTS COUNT to maximum (65536), and then hold the control hard clockwise for a few seconds to wrap the count back around to 1.
9. Adjust the B-Trigger LEVEL to a point where the waveform stops updating (outside the ± 200 mV range).
10. Press STATUS/HELP and observe that the trigger message reads TRIG WAIT: EVENTS and that the EVENTS COUNT = 1. These messages tell you that no events are occurring. Probable causes are: wrong trigger level, wrong source, or wrong coupling.
11. Reset the B-trigger level to start acquiring again (set to 0 V) and push STATUS/HELP twice to rewrite the status display.
12. Note that the trigger message has changed to COMPLETING ACQUISITION.

Extended Features

The following features allow you to operate the instrument in modes not usually available with conventional oscilloscopes. The Auto

Setup feature is demonstrated first, followed by two automatic measurement features. Finally, the AutoStep Sequencer is demonstrated.

Remember how Auto Setup was used in “Get Acquainted“ to quickly get you a usable display of the CALIBRATOR output? It turns out that the calibrator isn’t the best signal to use to explore Auto Setup and the other Extended Features because the calibrator period and amplitude vary with the SEC/DIV setting. To explore the Extended Features, you need to obtain the following equipment:

**Table 2-1
Equipment Required**

Item	Requirements	Recommended
1. Calibration Generator	Capable of outputting a square wave signal with an amplitude between 20 mV and 20 V. It should also have a period of 1 ms and a rise time longer than 50 ns, but less than 70 μ s.	TEKTRONIX PG506 Calibration Generator ^a
2. Coaxial Cable Connectors: BNC	impedance: 50 Ω . Length: About 40 inches.	Tektronix Part No. 01 2-0057-00
3. Dual-Input Coupler	Connects: BNC female to dual-BNC male.	Tektronix Part No. 067-0525-01

^a Requires a TM 500-Series Power-Module Mainframe.

Using Auto Setup

1. Recall the front-panel setting FP1. Select VERTICAL MODE and turn off CH 2.
2. Connect the Standard Output of the Calibration Generator to CH 1 and CH 2 inputs through a 5042 cable and a dual-input coupler.
3. Set the generator’s output to 0.5 V (the generator’s frequency should be 1 kHz).
4. Push the front-panel button labeled AUTO to do an Auto Setup on the input waveform for CH 1.

The scope displays the message AUTOS SETUP WORKING: PLEASE WAIT as it acquires information about the CH 1 waveform. Once it has characterized the waveform sufficiently to allow vertical and horizontal scaling, the waveform is sized to yield a usable display on-screen.

When Auto Setup is executed, it uses the mode selected in the Auto Setup menu. When you did an Auto Setup in Step 4, the mode selected was VIEW, which is designed to yield a display of 3 to 5 cycles over 10 divisions. This gives a good overall display. (In case you’re wondering, you selected VIEW mode when you reestablished the front-panel setup in Step 1.) Let’s try some other modes...

5. Push the bezel button labeled PERIOD. Note that the menu entry RES HI | LO appears. Push the AUTO button.

In this mode, the scope automatically sets the acquisition rate so that about 1 cycle of the waveform, triggered on the positive TRIGGER SLOPE, is displayed on screen. (PERIOD and VIEW always trigger on the positive edge of the waveform; you can change the TRIGGER SLOPE manually if you wish.) The waveform is vertically scaled so that the waveform fits in about the center 5 divisions on screen.

The RES HI | LO entry you noted determines how the scope sizes the waveforms in the parameter-oriented modes (i.e., all modes except VIEW). This instrument has a 20-division horizontal record length. When resolution is set to low, the parameter associated with the mode (in this case, PERIOD) is sized for best display over the 10 divisions on screen.

6. Switch RES HI | LO to HI and do another Auto Setup.

Notice that the period of the waveform is now spread over more divisions; in fact, it may not be completely contained within the 10 divisions on screen. For RES HI settings, the parameter associated with the mode is contained within the 20-division record length. You may have to position the trace horizontally to view the entire parameter. By spreading the parameter over more divisions, more sample points are obtained for the parameter. This yields better RESolution of the waveform.

The other modes are PULSE and EDGE. The vertical and horizontal scaling is similar for these modes. Briefly, in PULSE mode the scope does an Auto Setup which displays a positive or negative pulse. (The pulse is defined as that part of the rectangular waveform’s cycle having the *least* time duration). EDGE mode yields a display of the rising or falling edge of the waveform depending on the EDGE mode setting. The horizontal resolution for both PULSE and EDGE mode is determined by the RES HI | LO setting the same as for PULSE. These modes are covered in detail in Sections 3 and 5. Let’s look at PULSE...

7. Set PULSE mode on and execute another Auto Setup.

Notice that the positive section of the waveform is treated as a positive pulse and horizontally scaled to fit in about 20 horizontal divisions for the HI RES setting.

8. Switch RES to LO and reexecute Auto Setup. Notice the reduced horizontal scale (i.e. slower SEC/DIV setting).
9. Set the Auto Setup mode to PERIOD.

10. Push the VERTICAL MODE front-panel button. Turn CH 2 on.
11. Execute an Auto Setup.

Auto Setup can also make adjustments for various VERTICAL MODES. In this case, the waveforms in both CH 2 and CH 1 were sized to about 3 divisions and displayed overlapped at the vertical center of the screen. (Adjust vertical positioning slightly to see both waveforms.)

12. Set the Auto Setup mode to VIEW and do another Auto Setup.

Notice that this time the waveforms are scaled for about 2 divisions with the CH 1 waveform centered vertically around the graticule 2 divisions above graticule center and the CH 2 waveform centered around the graticule line 2 divisions below graticule center. Since in View mode it's assumed you want to see and compare waveform amplitudes, the waveforms are offset vertically on screen. If PULSE, PERIOD, or EDGE is selected, it's assumed that you are more interested in comparing time differences between waveforms. Therefore, the channels are displayed overlapped.

Before leaving Auto Setup, it's important to stress that various front-panel settings for VERTICAL MODE, TRIGGER SOURCE, and Auto Setup all influence how it operates. In general, VIEW mode will produce a usable display whenever a trigger source signal is available in the display source (CH 1 or CH 2) selected from the VERTICAL MODE menu, provided the signal can be triggered in AUTOLEVEL trigger mode. The sources the scope uses to trigger and size the displays vary with the front-panel conditions you set up. Read the description for control number 47, AUTO, in Section 5 and the applications for AUTO in Section 3. Once finished, you should be ready to use this convenient feature in all its modes.

Using MEASURE

1. Recall FP1, the procedure stored earlier in this section. Turn off CH 2.
2. Push AUTO to do an Auto Setup on the CH 1 waveform. Since Auto Setup executed in VIEW mode, you should have several cycles of the square wave displayed on screen.
3. Press MEASURE (next to PRGM) to display that menu, then press the SNAPSHOT menu button.

You have just executed the Snapshot mode for the Measure feature. It allows you to see at a glance many of the characteristics of the waveform for a single acquisition.

Realize that the accuracy of these measurements depends on the front-panel conditions set up. For instance, since we used Auto Setup in the VIEW mode, several cycles of the waveform are displayed on screen. This means that few sample points were obtained for the high-frequency components of the waveform. Those measurements relating to high-frequency components, such as RISE and FALL (rise- and fall-time) and OVRS and UNDS (over- and undershoot), should be discounted for this setup. However, since we obtained complete cycles of the waveform, we can trust those measurements related to amplitude and frequency, Such as P-P (peak-to-peak voltage), TOP and BASE (voltage at the Top and Bottom levels of the waveform, respectively), FREQ (frequency), etc.

In general, the screen tells you whether a particular parameter is valid or not. If you can't see the front-corner aberrations or you note that the rise time comprises little of the 20-division acquisition, you should set up the scope to display those parameters adequately. Let's do that now for the front edge of the waveform...

4. Press AUTO to display its menu. When Auto Setup finishes executing, set the mode to EDGE ($_ / _$) and RES to HI. Now do the Auto Setup for EDGE mode. Notice that in the resulting display the front corner is spread over several divisions.
5. Do another snapshot of the waveform. See Step 3 if you don't remember how.

Since the leading edge of the waveform is spread over several divisions, it now makes sense to use the measurements related to the front corner.

Note that some of the parameters have a string of "?" marks displayed instead of the parameter value. When the instrument cannot extract the parameter, it so indicates by displaying the string. In this case, since we used the EDGE mode to display the front corner of the waveform, the scope did not acquire an entire cycle of the square wave and could not extract those parameters pertaining to the period of the square wave.

6. Change the generator output to 1 V and push the menu button labeled AGAIN. The screen is updated with a new snapshot of the single acquisition of the waveform. (Note the new values for TOP and BASE.)
7. Select the VERTICAL MODE menu and turn on CH 2, leaving CH 1 turned on, also.
8. Press AUTO to display its menu. Change the mode to VIEW and reexecute Auto Setup.
9. Select the MEASURE menu. Set WINDOW OFF in the menu.
10. Push the menu button labeled MEAS TYPE.

The matrix you've displayed allows you to select up to four parameters that you wish to extract from the waveform. (When you select

a parameter, DISPLAY in the MEASURE menu is automatically set to ON.) Selected parameters will be displayed on screen and continually updated for subsequent acquisitions. Let's select some parameters...

11. One parameter in the matrix is underlined (DISTAL). Push the button labeled ON in the menu. Note that the matrix menu is replaced by a target menu. Use it to select the waveform for which you want to see the parameter displayed. The target menu always appears after you select a parameter or execute a snapshot measurement. If more than one display source is on screen.

12. Select CH 2 from the target menu. Since DISPLAY automatically turns ON in the main measure menu, the scope displays the parameter name and value on screen, along with the chosen display source. (It displays a default display source until you choose one.) Note that the parameter menu is also returned for further parameter selection.

Look at the parameter name you turned on in the parameter matrix. It's still underlined, but now there are two asterisks displayed, one on each side of the parameter name. The asterisks indicate which parameters are on so you can see that they are turned on regardless of whether DISPLAY is ON or OFF in the main MEASURE menu.

13. The arrow-labeled buttons allow you to select any parameter in the matrix. When you push an arrow button, the underline moves in the direction indicated.

14. Use the arrow-labeled buttons to move the underline to PERIOD in the matrix. Note that, as the underline moves away from the parameter you turned on, the parameter is no longer underlined; the underline selects the parameter, and the asterisks tell you when it's on.

15. When PERIOD is underlined, turn it on and select CH 1 as the target. The CH 1 period should be displayed.

16. Push ON to turn the PERIOD on again. Note that the target menu is displayed and a second CH 1 PERIOD readout appears on screen. This time select CH 2 as the target.

17. Turn DISTAL off in the matrix and turn PK-PK (peak-to-peak voltage) on. Select CH 1 as the target for PK-PK.

18. Increase the SEC/DIV setting 4 positions. Note that when the acquisition rate allows less than a complete period of a waveform to be acquired, the message NEED 3 EDGES replaces the parameter value in the readout. (Three transitions are needed to define a waveform cycle.) In general, the scope displays an error message if it can't extract the specified parameter.

19. Return the SEC/DIV back to its original setting. Now position the CH 1 waveform upward so that its top is several divisions off-screen. Notice that as the waveform exceeds the 10.24-division vertical-acquisition window, the message CLIP appears at left on screen. In general, the scope displays a warning message when it CAN extract a parameter, but detects a condition that makes the result questionable. A complete list of both error and warning messages is found in Appendix C of this manual.

20. Turn off CH 2 and center CH 1 vertically on screen.

21. Press MEASURE and set WINDOW on in that menu.

22. Press CURSOR FUNCTION and turn on the TIME cursors.

23. Use the CURSOR/DELAY control to move the active cursor forward the inactive one. Note that when the cursors no longer bracket at least one full cycle of the waveform, the message NEED 3 EDGES is displayed for the PERIOD measurements. Note also that the PK-PK voltage changes to approximately 0 Volts when the active cursor is superimposed on the inactive cursor.

When you switched WINDOW on in the MEASURE menu, you tied the measurements to the position of the TIME cursors. That's why, even though several cycles of the waveform are displayed on screen, you got the error message. There were not several cycles displayed *between the two cursors* when the message appeared.

24. Push MEASURE and select SETUP from the main MEASURE menu.

25. Set MARK ON | OFF to ON for the SETUP menu.

26. Move the active cursor so that the cursors bracket more than one waveform cycle.

Notice that two "X" 's appeared on screen. These are the markers you turned on in Step 25, and they indicate the points on the waveform where time-related measurements are being made.

27. Note the METHOD selections in the SETUP menu. These selections relate to the way the scope extracts waveforms.

28. Push LEVEL. This menu lets you define measurement reference levels on waveforms.

Let's learn more about the METHOD and LEVEL features by using HELP...

29. Push the STATUS/HELP button near the INTENSITY control. Select HELP from the menu displayed.

30. Push MEASURE.

The screen now displays a synopsis of MEASURE and its associated modes. The first two screens review some of the material you

have already covered. (Why not read It as a review?) Use the menu key labeled MORE to step through the screens of information; METHOD and SETUP information is on screens 3 and 4. Push EXIT to return to Scope mode when finished.

The MEASURE feature is discussed in Section 5, “Controls, Connectors, and indicators,“ and applications are found in Section 3. Also, in addition to other MEASURE-related information, Appendix C lists definitions of each parameter and the methods used for extracting those parameters.

Using PRGM (AutoStep Sequencer)

In this procedure, you have already used the AutoStep Sequencer to store front panels for later recall. While recalling front panels in this way is useful, AutoStep can be used for more powerful applications. Let’s see how...

1. Connect the STD AMPLITUDE OUTPUT of the generator to CH 1 through a 5042 coaxial cable. Set the generator output for a 5-V, 1 -kHz square wave.

We are going to assume that this waveform needs to be characterized for rise and fall times, top and base levels, and total amplitude peak-to-peak. What’s more, we’ll use our imagination and assume that this waveform is present on several instruments, so we need to make these measurements once for each instrument.

2. Recall the front panel you stored earlier. We will use it as a basis for our sequence steps.
3. Push the button PRGM and select SAVE from the AUTOSTEP SEQUENCER menu. Give the sequence a label (like TEST1) using the menu displayed.
4. When you have assigned the sequence a label, push the menu button SAVE to store the name and proceed with creating the sequence.

From the message on screen, you can see that this is where we set up the front panel for Step 1. Since we only want to make sure the instrument is operating properly and is set up to make the most accurate measurements possible, let’s not change any front-panel settings now, but just proceed to the ACTIONS menu. We will use Auto Setup later, in Step 2.

5. Push PRGM to move to the ACTIONS menu for Step 1.

You have now displayed the ACTIONS menu for the first step of your sequence. Notice that the display indicates the flow of events by labeling the beginning and end of the step and the events that occur between them. (The ACTIONS menu is a kind of time line for the substeps that occur for each step). Actions followed by <N> (oft) are not executed when the step is recalled; actions followed by <Y> (on) are executed. The LOAD STEP and MEASUREMENT events cannot be set; therefore, they are not marked either Y or N. They are in the menu to show when they occur relative to the actions that can be set.

Find the line under the Y or N following one of the actions. The arrow - labeled menu buttons are used to move that underline up and down in the ACTIONS menu to select the action desired.

6. Use the arrows in the menu to underline the SELF-CAL action and push Y N to toggle that action on (Y). Repeat for the SELF-TEST and PROTECT action. (PROTECT keeps the sequence from being accidentally deleted). All other actions should be turned oft.

7. Press NEXT STEP to exit the ACTIONS menu and store our first step.

The first step is now complete. When the sequence is recalled, the scope delivers the front panel you recalled prior to labeling this sequence (in procedure-Step 2), since you made no changes to the front panel when you created the step. Prior to recalling (loading) the front panel, It executes the SELF-CAL and SELF-TEST routines, since you selected those actions for Step 1.

8. The menu for setting up Step 2 is now displayed. Change the VERTICAL MODE setting to display CH 1 only, and change the COUPLING for CH 1 to DC. These menu changes will pause the scope to Auto Setup on the DC-coupled square wave in CH 1, when you later include Auto Setup as an action for this step.

9. Execute Auto Setup to verify that the Auto Setup menu is set to VIEW mode. (It not, set It to VIEW.) When Auto Setup finishes, push MEASURE to display that menu.

10. Select MEAS TYPE from the menu and turn on the parameter TOP, BASE, and PK-PK (Peak-to-Peak) using the method outlined under the MEASURE procedure in this section (Steps 11 to 17 under “Using MEASURE“).

11. This completes the front-panel setup for Step 2. Push PRGM to proceed to the ACTIONS menu.

Note that the ACTIONS menu is left as you set It up for Step 1. This feature allows you to just push NEXT STEP It you do not wish to change those actions. In this case, however, you do not need to SELF-CALibrate or SELFTEST the scope again. Also, now that you have set up the scope to measure the Top and Base levels of the waveform in Step 2, you probably need to PAUSE so the user of your sequence can record the measurements, take photographs of the display, etc. This is also where we turn on the Auto Setup action so It executes during the step in the mode verified in Step 9.

12. Set the SELF-CAL, SELF-TEST, and PROTECT actions oft (N); set PAUSE and Auto Setup on (Y). SET BELL on, also, to signal the user when the measurements are ready to be recorded.

13. Set the REPEAT action on; (we will see why later when we execute the procedure). Push NEXT STEP when finished.
14. Execute Auto Setup to display that menu. Change the mode to EDGE ($_ / _$) and RES HI | LO to HI.
15. Turn TOP and BASE parameters off and the RISE parameter on in the MEAS TYPE menu.
16. Push PGRM to proceed to the actions. Set REPEAT off, but leave the BELL, PAUSE, and Auto Setup on for this step. That way, the scope will pause and signal the user after it has set up on the waveform's rising edge and is displaying its rise time. This concludes Step 3 of the sequence.
17. Push NEXT STEP to proceed to Step 4. Execute Auto Setup to display that menu. Change the mode to EDGE ($_ _ / _$) and leave RES HI | LO set to HI.
18. Turn the RISE parameter off and set the FALL parameter on in the MEAS TYPE menu.

Notice that the rising edge is displayed on screen, and, since no falling edge is present, the FALL parameter value displayed is NO EDGE. When the sequence is stored and recalled, the falling edge of the waveform *will* be displayed along with the fall-time parameter.

When you proceed to the actions menu in the next procedure step, notice the order of events. The front panel is loaded and then the Auto Setup is executed. Since you executed Auto Setup in the rising-edge mode when you created Step 4, the front panel loaded for Step 4 displays the rising edge. You changed EDGE from rising mode to falling mode *after* you executed Auto Setup in Step 4, so when Auto Setup executes *as an action associated with Step 4*, the falling edge will be displayed.

19. Push PGRM to proceed to the actions. Again, leave the BELL, PAUSE, and Auto Setup as for Step 3.

This completes the sequence; now let's store it.

20. Push the menu button SAVE SEQ to save the sequence.

You have now labeled, created, and saved a four-step sequence. Let's review what you've done by running the sequence just saved...

21. Select RECALL from the main AUTOSTEP SEQUENCER menu. Use the arrow-labeled menu buttons to select your sequence and push RECALL to run it.

The sequence begins by executing Step 1. Notice that the message RUNNING SELF-CAL appears as the scope runs through that routine. Next, the display disappears for a time and the Trigger indicator LED's flash as the instrument runs its SELF-TEST. (The message RUNNING SELF-TEST appears part way through the routine). After a few minutes, the scope should finish Step 1 and immediately, since we didn't set the PAUSE action for Step 1, proceed to Step 2.

At Step 2, the BELL should ring to signal you that Step 2 is complete. Note that the square wave is sized properly for VIEW mode, and the TOP, BASE, and PK-PK (Peak-to-Peak) voltage values are displayed using the MEASURE feature. All these events correspond to our Step 2 setup.

22. Push PRGM to proceed to Step 3. Note that the rising edge is now displayed with the rise-time measurement chosen for Step 3. The PAUSE and BELL actions were also executed.

23. Push PRGM to execute the last step and display and measure the falling edge of the waveform.

Note that the menu still displays the message PUSH PRGM TO CONTINUE, even though we have already executed the last step of the sequence. Remember when we set the REPEAT action on for sequence-Step 2? This action causes the scope to loop back to Step 2 (or whichever step contains REPEAT) when the last step of the sequence is finished.

24. Change the generator output to the 10-Volt setting.
25. Push PGRM to branch back to the REPEAT (Step 2).

At the beginning of this procedure we said we had to make our measurements on several instruments. Here we just changed the amplitude setting of our generator, but you could change devices under test, test points to be measured, etc. as required. REPEAT 15 included in Step 2 instead of Step 1 so the scope can run an initialization routine (such as the SELF-CAL and SELF-TEST) once when the sequence is recalled and then skip the routine by looping through Steps 2-4 for successive measurements.

26. The scope is now at Step 2 of our sequence, with the 10-Volt square wave displayed with new Top, Base, and Peak-Peak levels shown.

We could go on to display the rise and fall times of the waveform, but let's just exit the sequence...

27. Push the menu button EXIT to get out of the sequence and display the RECALL menu. Push EXIT in the RECALL menu to return to the main AUTOSTEP SEQUENCER menu.

It you want to change the label of your sequence, you must copy the existing sequence, give it a new label, then delete the old sequence. Here's how it's done...

28. Select EDIT from the AUTOSTEP SEQUENCER menu, and use the familiar arrow-labeled buttons to pick out the sequence you wish to copy.

29. Press COPY to continue.

30. Now create the new label for your sequence as done in Step 3 of this procedure, and press SAVE when finished.

Note that the new label for your sequence appears in the list of current sequences. Now you can delete the old one:

31. Press EXIT to return to the main AUTOSTEP SEQUENCER menu.

32. Press DELETE to display that menu, and use the normal method to select the label of the sequence you want to delete.

33. Press DELETE to remove the sequence.

When you press DELETE you should hear the bell and see the message ERROR: PROTECTED. (Remember, you turned on the PROTECT action in procedure-Step 6.) The only way you can kill a protected sequence is to edit it. Here's a quick way to do that...

34. Push exit to return to the main menu and select EDIT.

35. Select the sequence you want to delete, using the usual method, and press EDIT to display the first step of that menu. Notice that the message displayed includes the sequence label and step number, along with information about how to change the front panel that was loaded for the step. Note also the menu label DELETE 10 BUFFER. Here, we don't want to change the step, we want to delete it:

36. Press DELETE 10 BUF.

To protect a sequence you must set the PROTECT action on for the sequence Step 1. (PROTECT in any other step is ignored). When you deleted the first step, you also deleted the actions, thereby removing protection for this sequence.

37. Push EXIT to return to the main menu and select DELETE from that menu.

38. Use the arrow-labeled buttons to select the sequence you have just edited. Press DELETE. Note that the sequence label is removed from the list.

There are ways to edit sequences besides merely deleting steps, but before we look at them, let's create a simple sequence that helps keep track of the steps we're going to edit...

39. Press EXIT to return to the main menu. Recall the front-panel setup we used at the beginning of this procedure. Disconnect the generator from CH 1.

40. Create the label "ED 1" and save it as you did previously.

41. Turn off CH 2 and position the CH 1 trace to the graticule line 3 divisions above graticule center. Push PRGM to advance to Step 1 actions.

42. Turn on only the PAUSE action. Press NEXT STEP.

43. Position the CH 1 trace to the graticule line 1 division above graticule center. Push PRGM to advance to Step 2 actions.

44. PAUSE should be the only action on. Press NEXT STEP.

45. Position the CH 1 trace to the graticule line 1 division below graticule center. Push PRGM to advance to Step 3 actions.

46. PAUSE should be the only action on. Press NEXT STEP.

47. Position the CH 1 trace to the graticule line 3 divisions below graticule center. Push PRGM to advance to Step 4 actions.

48. PAUSE should be the only action on. Save the sequence by pushing SAVE SEQ.

Let's look at the sequence...

49. Select RECALL from the main menu, then select and recall the sequence you just saved. Step 1 should now be displayed.

50. Use PRGM to step through the sequence, noting that the vertical position relates to the step number (most positive, Step 1; 2nd-most positive, Step 2, etc.). When you reach Step 4, the main menu is displayed. Press EDIT to change menus.

Let's use this new sequence to explore methods of editing sequences...

51. Select ED 1 from the list of sequences, then press EDIT.

As the message indicates, the front-panel for Step 1 of our sequence has been loaded. (Notice the vertical position of the trace.) You can now change this setup, if you wish.

52. However, since no changes are needed here, just push PRGM to call up the actions associated with Step 1.

53. Keep PAUSE and turn on BELL.

You have now edited Step 1 — both the front-panel setup (although you didn't change anything) and the associated actions. (You could also have continued without changing any at the Step 1 actions by pressing NEXT STEP.) If no other steps were to be edited, you would push SAVE SEQ to quit the edit session and save the revised sequence. In this case, however...

54. Press NEXT STEP to save the revised step and continue editing.

You are now at Step 2. Let's not edit this step now.

55. Push the up-arrow button in the menu.

Note the step number in the message changes back to 1. This is how you select steps to edit: when the front-panel is loaded and this menu is displayed, you can use the arrow-labeled buttons to select any step in the sequence you want to edit.

56. Select Step 4 for editing. Notice, as you proceed through Steps 1-4, that each step is loaded for editing in its proper order.

57. Instead of editing Step 4, you are going to add a step. Press ADD.

58. Center the trace vertically on screen. Select VERTICAL MODE and set YT | :XY to XY. Push PRGM, and then push NEXT STEP (without changing any actions).

The new step is now added as Step 5. Since Step 6 doesn't exist, the scope can't proceed; instead, it displays WARNING: NO MORE STEPS 10 EDIT.

If you had selected Step 2 for editing (rather than Step 4) and had then pressed ADD, the new step would have become Step 3 and the remaining steps in the original sequence would have been incremented by one. ADD puts the new step *after* the step selected and pushes the following steps down as a stack.

You can move a step from one position in the sequence to another by deleting it to a buffer and adding it back in wherever you want.

59. Press DELETE TO BUF to delete the step you just added. Use the arrow-labeled buttons to move to Step 1.

60. Press ADD. When the menu changes, press LOAD BUFFER. Press PRGM to get the actions menu and select NEXT STEP. Step 5 has now been moved to Step 2, and Steps 2-4 have become Steps 3-5. Using the arrow-buttons, go through all the steps at the sequence so you can verify the changes.

What if you want to add a step *before* Step 1? That also can be done. Move to Step 1 and add the new step. (You have already done this in procedure Steps 59 and 60.)

61. Select Step 1 and push DELETE TO BUF.

62. Press ADD and, when the menu changes, LOAD BUFFER. Then press PRGM followed by NEXT STEP to save what you have done.

When you deleted Step 1, Step 2 moved to Step 1. When you added the deleted Step 1, it was inserted after the new Step 1.

63. Return to Step 1 and press DELETE TO BUF. This should restore our sequence to its original 4 steps.

Copying steps is very similar to moving them. Let's put a copy of Step 3 after Step 4...

64. Select Step 3 and press DELETE TO BUF.

65. Select Step 2 and press ADD; when the menu changes, press LOAD BUFFER. Push PRGM, then NEXT STEP. You have just restored Step 3 to its original place in the sequence, but a copy of it remains in the buffer.

66. Select Step 4. To finish putting a copy at Step 3 after Step 4, press ADD, followed by LOAD BUFFER, PRGM, and NEXT STEP. You can use the arrow-buttons to step through the sequence and verify that you've copied Step 3 and put it after Step 4.

In general, to copy a step to another location: delete that step to the buffer, then use ADD twice, once to put it back in its original location and once to put it in its new location.

This completes the Operators Familiarization Procedures. Although most of the controls and functions have been exercised in these

procedures, not ALL functions have been covered. As mentioned earlier, Section 5 describes in depth the function of each control, connector, and indicator. Section 3 contains applications for many 2440 features, and the Programmers Reference Guide provides GPIB-related information. Also, in Section 7 of this manual is a short familiarization procedure for the Video Option, for those Instruments equipped with this option.