PC INSTRUMENTS

BenchTop for Windows

User s Guide

About this document - This document contains documentation for BenchTop and BenchTop Lite for Windows. Features which apply only to BenchTop for Windows are indicated within the text. For complete specifications and installation instructions for hardware products, please refer to the BenchCom Operator's Manual.

BenchTop User's Guide. Document #98-06-24

Printed in the USA.

Table of Contents

Chapter 1
Installation

Introduction

This manual describes the installation and operation of the BenchTop Lite for Windows software package and the BenchTop Plus for Windows software package. BenchTop Lite is a subset of BenchTop. Both packages contain instrument controls, cursors, waveform storage and recall, instrument set-up storage and recall, and features that take advantage of the Windows environment. The advanced Features contained only in the BenchTop Plus package are described in Chapters 5 and 6. These features include analytical functions and automated test functions.

Throughout the manual, both packages will be referred to as “BenchTop” or “BenchTop for Windows”. Features not included in the BenchTop Lite for Windows package (except those in Chapters 5 and 6) will be properly denoted. If you are using the BenchTop Lite for Windows package, the icons on the computer screen for the advanced features will be grayed.

To run BenchTop for Windows you must have installed some combination of PC Instruments cards in your PC, unless you are using the demonstration version of BenchTop (see the next paragraph). Complete instructions for hardware installation are covered in the BenchCom User's Manual supplied with each instrument. You can install BenchTop for Windows before or after you have installed the hardware, but the hardware should be installed prior to running the hardware configuration program (described later in this chapter).

Instrumentation cards are not required if you are using the demonstration version of BenchTop for Windows. All of the instrument controls will be usable, including those controls that create arbitrary waveforms.   The demonstration software will simulate waveforms for the oscilloscope display.

Software Installation

To install BenchTop for Windows from the Windows 3.1 Program Manager, insert Disk 1 and choose File|Run. Type the name of the 3.5" disk drive followed by "setup". For example:.

```
a:setup
```

Press Enter or click on "OK" and follow the instructions on the screen.

If you are installing under Windows 95 or Windows NT, choose “Run” from the Start menu and enter “a:setup” as above.

The setup program will copy files for BenchTop into three different directories. The hardware configuration program and release notes will be placed in the default \PCI directory. Program files for BenchTop will be placed in the \BTOP subdirectory beneath the \PCI directory. The installation program will also copy some files to the Windows System directory. Table 2-1 below lists all of the files installed to the computer and the default directory of each file.

Directory:	File:	Description
c:\pci	pcicnf16.exe	Hardware configuration program
c:\pci	pciwview.cnf	SCPI configuration file
c:\pci\btop	btop.exe	BenchTop for Windows executable
c:\pci\btop	btop.hlp	BenchTop for Windows help file
c:\pci\btop	pciscpi.hlp	PC Instruments SCPI help file
c:\pci\btop	pcisas.exe	DDE Server for Signal Analysis
c:\pci\btop	pcidx.exe	Data Exchange utility program
c:\windows\system	pciwin.dll	PC Instruments Dynamic Link Library
c:\windows\system	bivbx11.dll	Support Dynamic Link Library
c:\windows\system	bwcc.dll	Support Dynamic Link Library
c:\windows\system	bc453rtl.dll	Support Dynamic Link Library
c:\windows\system	owl501.dll	Support Dynamic Link Library
c:\windows\system	pcinst.vbx	PC Instruments VBX Library
c:\windows\system	bids501.dll	Support Dynamic Link Library

There may be additional system files installed that are not listed in Table 2-1.

Configuring Hardware

After installing the BenchTop for Windows software, you should run the PC Instruments hardware configuration program (PCICNFxx.EXE). A program icon for the hardware configuration program will be placed in the PC Instruments group during installation. If you have already run the hardware configuration program while installing BenchCom, you do not need to re-run it.

Removing BenchTop From Your Computer

Windows 3.1
An uninstall program icon will be created for you in the PC Instruments group when you install BenchTop for Windows. To uninstall the program, double-click on this icon and follow the directions on the screen.

Windows 95/NT
To uninstall BenchTop from Windows 95 or Windows NT, double click "Add/Remove Programs" from the Control Panel (Click the Start button, select "Settings", then select "Control Panel"). In the list of installed software, select BenchTop for Windows and click the "Add/Remove" button. Follow the instructions to complete the uninstallation of the software.

After either (Windows 3.1 or Windows95/NT) uninstall program is complete, there may be some files or directories left on your computer from BenchTop. This is because the uninstallation program will only remove the files that were installed with BenchTop for Windows. Any files created in the \PCI or BenchTop directories after installation are not deleted. For example, data files or hardware configuration files. If you wish to delete these files, you will have to use Explorer or another file management utility to delete them.

Chapter 2
Getting Started

Introduction

BenchTop for Windows provides comprehensive control over all PC Instruments products from a true Multiple Document Interface (MDI) under Microsoft Windows version 3.1 or Windows 95. The program will display multiple instruments within moveable, resizeable frames.

Overview of Features

The following section describes the basic features provided by BenchTop for Windows.

Integrated Windows Environment

BenchTop has been designed to take advantage of the Windows environment wherever possible. For example, instrument data can be copied directly from BenchTop to another Windows program through the Windows clipboard. Also, BenchTop is able to print to any Windows configured printer or fax manager which makes exporting or printing results or data much easier than under DOS.

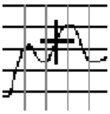
Moveable, resizeable waveform displays

Each instrument displayed in the BenchTop workspace may be moved and resized as desired. This means you can customize the look and feel of your instrument(s) to best suit your needs. In addition, the layout and positioning of windows can be stored to *workspace files* on your hard drive and recalled later to save you time when setting up for a particular measurement or function. The same functionality applies to instruments: you can store instrument settings to files on your hard drive and recall them later to quickly setup for a particular measurement.

Point-and-Click Instrument Controls

All of the instruments in BenchTop are controlled via easy to use point-and-click controls like knobs, up/down arrows, and buttons. So controlling your instruments doesn't require that you learn a complicated command language or memorize bizarre keystroke combinations.

On-screen Cursors



Both the oscilloscope and arbitrary waveform generator instrument displays have on-screen cursors that allow you to investigate a signal simply by pointing at it with the mouse and reading the time and voltage information directly on the screen. You can also measure the time and voltage difference between two points on a waveform. And for the arbitrary waveform generator, you can use the mouse to edit the contents of the Arbs memory **directly** by "drawing" on the Arb display with the mouse.

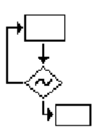
Signal Analysis*

Over twenty different waveform parameters can be measured using either the Signal Meter or Pulse Analysis displays in BenchTop. You can configure these displays independently to show whatever measurements you are interested in, as well as have multiple analysis windows open for investigating different channels of oscilloscope data.

Boundary Testing*

Boundary Testing refers to using an oscilloscope to measure a signal and compare it to a previously stored "gold" signal. Using BenchTop for Windows you can create boundary files from waveform files, store the boundaries to your hard drive, and then recall them later to perform tests on signals. This ability is ideal for production environments where new units are expected to behave like a "golden board".

Automated Test Language*



enchTop for Windows is the PC Instruments Automated
uage (ATL) Interpreter. ATL is a programming language
u create test sequences that can be used in production
ironments to guide an operator through a series of tests.
The ATL interpreter makes performing complicated tests like boundary tests simple and allows you to create a custom interface including prompts, dialog boxes, and message boxes. You can also log test results and information from the operator to files on the hard drive for review and archiving purposes.

Online Help

BenchTop for Windows has a comprehensive context sensitive help file which can be invoked directly from the program to help you find answers to the most common operating questions.

Constantly Evolving

We have worked hard to make BenchTop for Windows as useful as possible. At the same time we recognize that to remain useful a tool must evolve. That's why we are continuously enhancing and upgrading BenchTop for Windows with new and improved features. As major versions are released, you'll be contacted about upgrades. We also value feedback from our customers and we take your suggestions seriously. So feel free to call us at (330)-762-8500 and let us know what you think.

* This feature is not available in BenchTop Lite for Windows.

Starting BenchTop for Windows

From the Program Manager in Windows open the PC Instruments group and double click on the BenchTop for Windows icon. This will start BenchTop for Windows and the main BenchTop window will appear. The BenchTop window will contain the panels that were last left on when BenchTop was previously exited. If this is the first time that BenchTop has been started the BenchTop window will be empty.

Errors Starting BenchTop

The following errors may be encountered when starting BenchTop.

File Not Found

This error is caused if BenchTop can not locate a configuration file during startup. The following configuration files are required:

File:	Description:	Default Location:
pci.cnf	Hardware configuration file	c:\PCI
pciwview.cnf	SCPI configuration file	c:\PCI

To fix the problem, check that these files are in the appropriate directories. Note that the first file (pci.cnf) is created by the hardware configuration program, PCICNF.EXE.

If you have moved the PC Instruments files from the original locations that you installed them to, you'll need to modify either your AUTOEXEC.BAT file or the Windows INI file, WIN.INI. Within AUTOEXEC.BAT, change the line which reads

```
set PCICNF=c:\PCI
```

to specify the appropriate directory. In WIN.INI, change the line under the PC Instruments section as shown below to the appropriate directory:

```
[PCInstruments]
PCICNF=c:\pci
```

Checksum Mismatch

This error is caused by an error in the configuration or installation of an instrument. The most likely problem is that the base address of a board is set incorrectly. To fix the problem, recheck the base address selections made in the hardware configuration program and the DIP switch settings on the instruments. If all base addresses are set correctly, the next most likely source of the problem is a hardware conflict between a PC Instruments card and another peripheral in your PC. Try changing the base address of an instrument to another available address. If you still have trouble, please call PC Instruments technical support.

Chapter 3
Using BenchTop for Windows

The Basics

In BenchTop for Windows, each instrument exists as a "child" window within the application's workspace. A new window (instrument) is created by selecting a menu option or pressing a button on the Toolbar. The program will automatically support any number and configuration of PC Instruments cards.

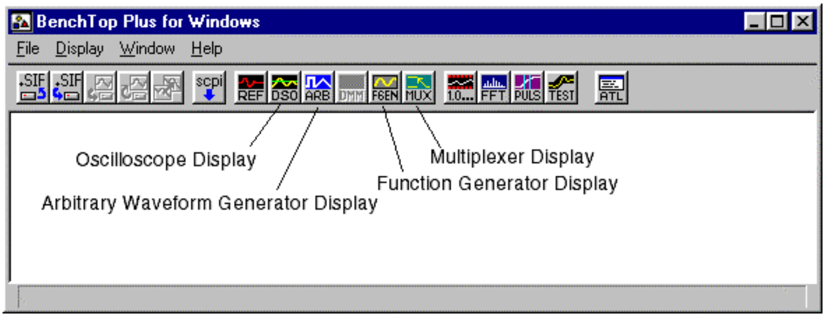
Prior to running BenchTop for Windows for the first time, you need to run the hardware configuration program that comes with both BenchTop and BenchCom. This program (described in the installation section of the BenchCom User's Manual) will create the hardware configuration file that is used by BenchTop to determine what type of instruments you have installed.

Getting Help

To get help while using BenchTop for Windows, press F1 or select Help|Contents from the menu bar. The BenchTop for Windows help file (btop.hlp) will be invoked using the Windows help engine. Most of the information contained in this manual can also be found by utilizing the online help.

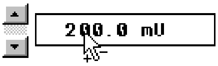
Creating Displays

As shown in the figure below, buttons on the Toolbar (just below the menu bar) can be used to create displays for whatever type of instruments you have installed in your PC. If a button is grayed out or missing, this indicates that the software does not recognize that the instrument linked to that button is installed.



Note: If an instrument is installed in the computer but the button(s) are grayed out (or the program won't start), exit Windows and refer to the Installation section in Chapter 1.

Modifying Settings



Many of the control panels used in BenchTop use controls similar to the one shown to the left to set instrument parameters like trigger level or clock rate. These controls can be modified either by clicking the up/down arrows to the left of the readout or by clicking directly on the digits shown. When the mouse is moved over a digit, the cursor changes to the "plus-minus" arrow as shown. When this cursor is shown, left click to increment the digit that the mouse points at, or right click to decrement the digit. You can also left/right click on the units (e.g. left click on kHz will change the value to MHz).

The Toolbar

The Toolbar allows many of the program's features to be activated quickly and without navigating through menus. The Toolbar also supports “fly-over-help”. As the cursor is moved from one button to another, a descriptive message will be displayed in the message bar at the bottom of the program’s main window. The functions available from the Toolbar are listed below:

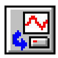
Restore instrument settings from a file

Use this button to load a Script Input File (SIF) from the hard drive. SIF files contain setup (e.g. volts/division, trigger source, filter settings, etc) information for the instrument(s) installed. SIF files do not normally contain workspace information (location of windows, open controls, etc.). If you are using one or more arbitrary waveform generators, note that recalling a SIF file may take up to 20 seconds if you have a large amount of waveform data stored in the file.

Save instrument settings to a file

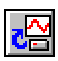
Use this button to save the current instrument settings to a file on the hard drive. This will store the settings (e.g. timebase, vertical range, etc.) of all instruments to a file on your hard drive. The waveforms from the arbitrary waveform generators will also be stored. BenchTop will prompt you for the filename. The file created will be a Script Input File (SIF) format. SIF files can be viewed and edited with an ASCII text editor (e.g. Notepad).

Store waveform data to a file



Use this button to save the currently selected instrument's data to a file on the hard drive. Note that before you press this button you have to select an instrument display by clicking on it with the mouse. BenchTop will prompt you for both the filename and file format. You can save the waveform data to either a SCPI-DIF file, two-column ASCII file, or single column "raw" ASCII file. The SCPI-DIF file has the benefit of saving the instrument settings (like trigger level and input coupling) and time/date stamp along with the data. The two-column ASCII format is useful for transporting the data to other programs (like Excel or MathCad).

Recall waveform data from a file



This button will load a waveform file into an arbitrary waveform generator. The input file can be either a SCPI-DIF formatted file, two-column ASCII, or one-column ASCII.

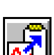
Copy DSO waveform to Arbitrary Waveform Generator

If you have both an arbitrary waveform generator and an oscilloscope installed, this button will allow you to transfer the current data from the scope directly to the Arb. This will destroy the current contents of the Arb, so you should save the Arb setup (by pressing the "Save Instrument Settings to a File" button) if you don't want to loose the information.

Print the waveform display

his button to print the currently selected instrument's data to the er. The display will be printed to the printer with whatever annotations (tick marks, readouts, etc.) are currently selected. You can edit the printer setup by selecting Print Setup from the File menu. Note that the display to be printed must be selected (clicked on with the left mouse button) prior to pressing the print button.

Copy the selected instrument display to the clipboard



Use this button to copy the graphical instrument display to the Windows clipboard. To paste the display into another application, simply start (or switch to) the receiving Windows program and choose Edit|Paste or the equivalent command for the particular program. From the clipboard, the image can be pasted directly into any Windows program that supports BMP data.

Copy the selected instrument data to the clipboard

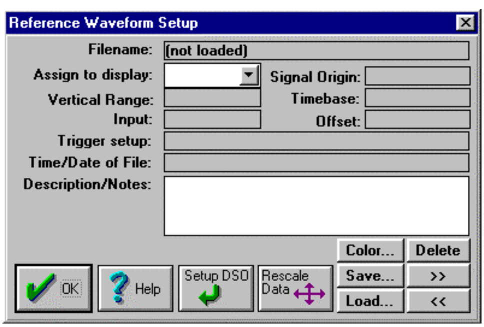

Use this button to copy the instrument data to the Windows clipboard. You can configure how the data is copied by choosing "Clipboard Options..." from the File menu. To paste the data into another application, simply start (or switch to) the receiving Windows program and choose Edit|Paste or the equivalent command for the particular program.

Enter SCPI command directly

Press this button if you want to enter a SCPI command directly. SCPI stands for Standard Commands for Programmable Instruments. It is an industry standard for the syntax of a command between an instrument and a computer. SCPI commands are documented in the BenchCom User's Guide and information concerning the SCPI commands is also available from the Help menu.

Activate Reference Waveform Dialog box

Pressing this button will activate the reference waveform dialog box, as shown below. Reference waveforms are data files that were saved previously with the Store Waveform Data command and reloaded to display within an oscilloscope display along with "live" data.



To close the reference waveform dialog box, click the OK button. To load a reference waveform from the hard drive, click the "Load..." button. A file-dialog box will appear listing the available waveform files.

Once you have loaded a reference waveform, you can assign it to a particular oscilloscope display (if multiple displays exist) by selecting a display from the "Assign to Display" drop-down list box. If multiple references are loaded, you can navigate through them using the Forward (">>") and Backward ("<<") buttons. The filename, settings, time/date stamp, and notes are displayed for each particular reference waveform that has been loaded.

You can change the color used to draw the reference waveform by selecting the "Color" button. If you make changes to a reference waveform (e.g. changing the color or adding notes) you can save the reference by pressing "Save".

With a reference waveform loaded, you can press the "Setup DSO" button to automatically configure the scope (i.e. timebase, offset, trigger level, etc.) to the settings contained in the reference waveform file. Alternatively, you can press "Rescale Data" to rescale the reference waveforms voltage data to the currently selected scope's vertical range and offset.

The Delete button will clear the waveform from the screen and delete it from the computer's memory. The Delete button will not remove the waveform from the hard drive.

Create a New Oscilloscope Display (PCI-41x/42x/43x)

Press this button to create a new oscilloscope display. The scope display is used to control the scope and view scope data. Generally, you will have one oscilloscope display for each scope installed in your computer. The oscilloscope display and its controls are described later.

Create a New Arbitrary Waveform Generator Display

Press this button to create a new arbitrary waveform generator display. The arb display is used to view the contents of the arb's memory and manipulate its settings. Usually you will have one Arb display for each arb installed in your computer. The arbitrary waveform generator display and its controls are described later.

Create a Digital Multi-Meter Display (PCI-501)

Pressing this button will activate a new Digital Multimeter control. The DMM control lets you view measurements made by the PCI-501 and manipulate its settings. Each DMM installed in the computer will have one DMM control. The Digital Multimeter control is described in more detail later.

Create a Function Generator Display (PCI-303/305)

Pressing this button will activate a new function generator control. The function generator control lets you view and manipulate the settings for the PCI-303 generator. Each generator installed in the computer will have one function generator control. The function generator control is described in more detail later.

Create a Multiplexer Display (PCI-70x)

Pressing this button or selecting “New Multiplexer Display” from the

Display menu will create a graphical dialog box for controlling a multiplexer card. The multiplexer display is described in detail in Chapter 3.

Create a Signal Meter display

Pressing this button will create a new window for measuring oscilloscope signal parameters such as minimum and maximum voltage, RMS, etc. This window is described in detail in Chapter 4.

Create a Pulse Analysis display

Pressing this button will create a new pulse analysis window. The pulse analysis window can be used to measure and display pulse parameters from an oscilloscope channel. Pulse parameters include risetime, falltime, dutycycle, etc. This window is described in detail in Chapter 4.

The Menu Bar

The menu bar provides access to many of the same program features as are available via the buttons on the toolbar. Each menu is described below.

The File Menu

Items under the File menu affect global program options. Some of the items under File are Window-specific, but are kept under the File menu to be consistent with other Windows programs.

Restore Instrument Setup
Save Instrument Setup
Instrument settings (e.g. timebase, vertical range, etc.) can be saved and loaded from files on the hard drive. These files, called "SIF" files (Script Input Files), can be created at any time by choosing "Save Instrument Setup" and loaded later using "Restore Instrument Setup". A SIF file can be edited with any ASCII text editor (e.g. Notepad).

Store Waveform Data
Waveform data from either a scope display or arbitrary waveform generator display can be stored to a file on the hard drive using the "Store Waveform Data" selection. The resulting file can be either a SCPI-DIF formatted file, or a two-column ASCII file.

Print
Print will allow you to print the currently selected instrument display. Before selecting Print, you must choose a display within the workspace by clicking once on its title bar.

Print Setup
Print Setup allows you to configure your Windows compatible printer.

Clipboard Options
This option lets you customize the manner by which data is copied onto the Windows clipboard (using "Copy Data to Clipboard", see below).

Execute an ATL File
This selection will launch an Automated Test Language file from the harddrive. Refer to Chapter 6 for information on the Automated Test Language.

Exit
Exit will shut down BenchTop for Windows and store the workspace in "btop.wsp".

The Display Menu
Items under the Display menu affect properties of individual instrument displays, as well as the creation of new displays.

Load Workspace
Save Workspace As
A workspace file contains the setup information for the screen. This includes the position and settings of all instrument displays and the size and position of the program itself. To create a workspace file, setup the program the way you want it and choose "Save Workspace As". The filename you enter can then be recalled later using "Load Workspace".

Reference Waveforms
This selection will activate the Reference Waveform dialog box. Reference waveforms are data files that were saved previously with the Store Waveform Data command and reloaded to display within an oscilloscope display along with "live" data. See the description of the reference waveform dialog box above (under "Toolbar").

New Scope Display
New Arb Display
New DMM Display
New Function Generator Display
Each of the "New ... Display" options creates a new window within the BenchTop workspace to represent the instrument type indicated. If

you have not installed or configured a type of instrument, the option(s) will be grayed out or missing from the menu bar. The function of these selections is identical to the corresponding Toolbar buttons.

New Signal Meter
This selection will create a new Signal Meter display. The Signal Meter is described in detail later in Chapter 4.

New Pulse Analysis Display
This selection will create a new Pulse Analysis display. The pulse analysis display is described in detail later in Chapter 4.

SCPI Command Box
This selection will activate the SCPI Command dialog box, which allows you to type SCPI commands directly (without using the built-in point-and-click controls of BenchTop).

Disable Updates when not Active
This selection is toggled on and off each time it is selected. When on (checked), BenchTop will automatically stop updating the oscilloscope display(s) if the program is backgrounded. When off (unchecked), the program will continue to update all scope displays even when BenchTop is not the "top" application. If you frequently task-switch between BenchTop and other Windows programs, system performance can be greatly enhanced by enabling this option.

The Window Menu

The items under the Window menu control the layout of multiple instrument displays (if active) and special Window related functions.

Cascade
Cascade will arrange the instrument displays in a layered, ascending pattern.

Tile
Tile Horizontally
Tile and Tile Horizontally will arrange the instrument displays so they do not overlap.

Arrange Icons
Arrange Icons will sort any minimized instrument displays so they appear in a row at the bottom of the BenchTop workspace.

Close All
Close All will destroy all instrument displays currently open.

Copy Display to Clipboard

This selection will create a bitmap image (snapshot) of the currently selected instrument display and copy it onto the Windows Clipboard. From the clipboard, the image can be pasted directly into any Windows program that supports BMP data.

Copy Data to Clipboard

This selection will copy the data from the currently selected instrument display onto the Windows Clipboard. The exact format of the data placed on the clipboard is configured by selecting "Clipboard Options" from the File menu (see above).

The Help Menu

The help menu allows access to on-line help files using the Windows Help Engine.

Contents

This will open the Table of Contents for the main BenchTop help file - btop.hlp.

SCPI Reference

This will open the SCPI Reference help file - pciscpi.hlp.

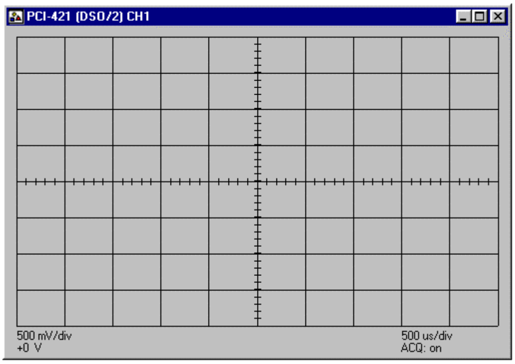
About

The "About" dialog box contains revision information about BenchTop.

Chapter 4
Instrument Controls

The Oscilloscope Display

To activate an oscilloscope display, either select the oscilloscope icon from the Toolbar or select New Scope Display from the Display menu. After the oscilloscope is activated, an annotated oscilloscope waveform display will appear similar to shown below. By default, the scope display shows the instrument's vertical range (Volts/division), offset voltage, timebase (seconds/division), and acquisition status (on/off/armed).



Waveform Display

As shown above, the waveform display area has a 8x10 grid for the Y/T display mode and a 8x8 grid for the X/Y display mode. Characteristics of the display such as annotation, colors, cursors, and tick marks are controlled in the pop-up menu described later in this section.

Title Bar

The Title Bar has three pieces of information. The first item is the model number of the oscilloscope (e.g. PCI-421). The second item is the instrument identifier (e.g. DSO/1). The three letters indicate that this waveform display is for a Digital Storage Oscilloscope and its instrument identifier is 1. The instrument identifier is the number of the configured instrument. The third item identifies the channel that is being displayed.

Volts/Division and Offset Readouts

The current volts/division setting and offset level for Channel 1 (if it is on) and Channel 2 (if it is on) are indicated in the lower left-hand corner of the waveform display. The offset level is adjustable and it is the voltage at the middle of the waveform display. The readouts can be turned off using the pop-up control menu for the display (see below).

Ground Level Indicator

The Ground Level Indicator is a horizontal dashed line. If the offset voltage level is beyond the display range, the Ground Level Indicator will disappear. The Oscilloscope Control Menu (described later) has an on/off control for the Ground Level Indicator.

Trigger Level Indicator

The Trigger Level Indicator is the triangle on the left-hand side of the waveform display area. As the trigger level is adjusted the triangle will move up and down indicating the approximate trigger level. If the trigger level is beyond the display range the triangle will point either up or down. The color of the Trigger Level Indicator will match the color of the trigger source waveform. If the trigger source is External, the Trigger Level Indicator will be white. The trigger slope is indicated by a “+” or “-” within the Trigger Level Indicator. The Oscilloscope Control Menu (described later) has an on/off control for the Trigger Level Indicator.

Cursor Volts and Time

Underneath the waveform display area and in the center of the oscilloscope display panel is the voltage readout and time readout of the crosshair cursor. As the mouse cursor is moved over the oscilloscope display panel it changes to a crosshair ("+") and the cursor voltage and time readouts indicate the voltage and time at the crosshair. The crosshair can be made to snap to the waveform or be allowed to be free floating in the voltage-time plane (or in the voltage-voltage plane if the display mode is XY). The particular waveform that the cursor snaps to is selected in the Window Setup dialog described later in this chapter.

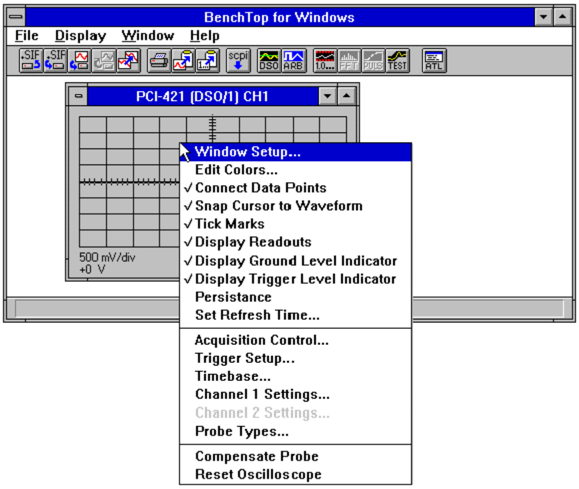
The crosshair cursor can also be used to make difference measurements (also known as "delta T and delta V" measurements). Position the crosshair over a reference point on the waveform and hold down the left button of the mouse. An "*" will appear to the left of the cursor voltage and time readouts indicating that a reference measurement is being made and a second crosshair cursor will appear. Move the mouse (while holding down the left button) to a different point on the waveform and the readouts will indicate the difference in time and voltage between the two crosshairs.

Time/Division Readout
The present timebase setting is indicated in the lower right-hand corner.

Acquisition Readout
This indicator is located in the lower right-hand corner and it can have one of three states: 1) ACQ: OFF, 2) ACQ: ON, or 3) ACQ: ARMED. If the Acquisition Readout indicates "ACQ: OFF", then the oscilloscope's acquisition circuits will not recognize any trigger events and will not collect a sweep of data. If the Acquisition Readout indicates "ACQ: ON", then the oscilloscope's acquisition circuits are actively recognizing trigger events and acquiring data. If the Acquisition Readout indicates "ACQ: ARMED", then the oscilloscope's acquisition circuits are waiting for a trigger event in order to begin acquiring data.

Moving and Resizing the Oscilloscope Display Panel
A consistent feature of Windows software is the ability to move and resize application windows. To move the oscilloscope display window move the mouse cursor to the window title bar and drag the title bar to the desired location. Resizing the window is accomplished by positioning the cursor in the lower right-hand corner of the window frame. The cursor will change to a two-headed arrow and by dragging the mouse the window will change sizes. The full screen box in the upper right-hand corner of the window frame can be used to expand the oscilloscope display window.

Activating the Oscilloscope Control Menu
The controls associated with the oscilloscope are accessed by positioning the mouse cursor over the oscilloscope display panel and clicking the right button of the mouse. A pop-up menu will appear listing the available controls for the display (see figure below). To select a particular control, drag the mouse (holding the left button down) so that the control is highlighted and then let go of the left button , or move the cursor to the control and click the left button of the mouse. The oscilloscope controls are listed below.



Window Setup...

This control selects whether Channel 1, Channel 2, or Both channels are displayed. It also provides the capability to create an XY display. If the computer contains two or more oscilloscope boards, this control provides the capability to determine which oscilloscope board is being controlled by the oscilloscope display.

Edit Colors...

Select this option to edit the colors used in the waveform display. You can change the waveform color(s), grid color, and background color.

Connect Data Points

This option is toggled on or off each time it is selected. When on, the data points will be connected with line segments. When off, the data points will be drawn as individual dots on the waveform display.

If the waveform display has gaps where there does not appear to be any data, try turning this option on or resizing the waveform display. This affect is caused by multiple data points overwriting each other on the screen. To minimize this affect, setup Windows to use a video mode of at least 800x600 pixels.

Snap Cursor to Waveform

When checked, the crosshair cursor ("+") will snap (i.e. lock) to one of the displayed waveforms. When unchecked, the crosshair cursor is able to be positioned anywhere within the Volts-time plane.

Tick Marks
This option is toggled on or off each time it is selected. When on, the waveform grid is enhanced with smaller lines along the center horizontal and vertical grid lines.
If the Tick Marks are on, they will be automatically copied to the clipboard along with the waveform image using the Toolbar button (see below).

Display Readouts
This option is toggled on or off each time it is selected. When on, several readouts are displayed directly on the waveform display below the grid and the ground level indictor and trigger level indicator are displayed. When off, the waveform display is shown without any annotations.

Display Ground Level Indicator
This option is toggled on or off each time it is selected. When on, a horizontal dashed line is displayed on the oscilloscope display indicating the position of 0.0 Volts. If the 0.0 volt level is beyond the range of the oscilloscope display, the horizontal dashed line will disappear.

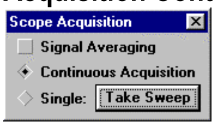
Display Trigger Level Indicator
This option is toggled on or off each time it is selected. When on (checked), a triangle is displayed on the left side of the oscilloscope display area pointing to the approximate trigger level. If the trigger level is beyond the range of the oscilloscope display, the right pointing arrow on the left of the display will change to an up (or down) pointing arrow indicating that the trigger level is above (or below) the presently displayed range of voltages. The color of the Trigger Level Indicator matches the color of the trigger source waveform. If the trigger source is External, the Trigger Level Indicator will be white. A "+" or "-" will be displayed within the Trigger Level Indicator depending on the trigger slope.

Persistence
When this option is checked, BenchTop will not erase the pixels on the computer screen when a new waveform is displayed. This mode is useful when monitoring the change in the shape of a waveform or when viewing modulated waveforms.

Set Refresh Time..
This option sets the time (in ms) interval between waveform display updates. A longer time interval will result in fewer screen updates per second but will free more system resources for other programs. Typical values of Refresh Rate are 100ms to 250ms.

The minimum time that can be achieved depends on the computer's speed (CPU and video adapter) and on the resources needed by any other programs running concurrently with BenchTop for Windows. If other applications are not operating properly, or if BenchTop for Windows is slow to respond to the mouse or keyboard, increase the time interval between screen updates. See also the “Disable Updates when not Active” option described under “The Menu Bar”.

Acquisition Control...



The oscilloscope can either be set to continuously acquire new waveforms or it can be set to take a single waveform at a time. If “Continuous Acquisition” is selected, the oscilloscope board will continually acquire sweeps of data into its on-board buffer memory. If Signal Averaging is enabled, the oscilloscope board will use its on-board DSP processor to perform a sweep average on each new sweep of data. Signal averaging reduces the amount of noise that is asynchronous to the trigger event. Signals that are asynchronous or have unstable triggers may appear to “glide” or shift between screen updates. In this case, signal averaging should be turned off. The PC will transfer waveforms from the oscilloscope to the display at a rate determined by the “Set Refresh Time...” control described in the previous section.

In the Single mode, the oscilloscope board will acquire one sweep of data after it has been armed (by pushing the Take Sweep button) and after it has been triggered. It will not acquire another sweep until it has been re-armed by pushing the Take Sweep button again. This mode is useful for capturing transient signals when the timebase is set to the real-time sampling mode (2µs/div and slower).

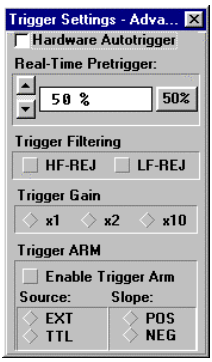
Trigger Setup...

The oscilloscope can trigger on three different sources with a variable level and a variable slope. Also provided is an Automatic Trigger Level button, a user input entry for the trigger level, and a selector that provides for three different horizontal trigger positions in the real-time acquisition mode.

The trigger options include the Source, Level (voltage), and Slope. The oscilloscope "remembers" the trigger level and slope for each source, therefore you can toggle between alternate sources without resetting the trigger level or slope.

Pushing the Auto button will cause the oscilloscope to automatically find the optimal trigger level for the selected source. The User button provides the capability to type in a particular trigger level.

The Trigger Position control selects the pre-trigger percentage for Real-Time sampling timebases. PCI-42x and 43x series oscilloscopes have two modes of acquisition, Real-Time sampling and Equivalent-Time sampling. Which mode is used for these instruments is determined by the timebase setting. If you set the oscilloscope to 2us/div or slower it will use Real-Time sampling. In this mode of operation, you can choose how much data should be collected before the trigger (10%, 50%, or 90%). For the Equivalent-Time sampling timebases (1.8us/div or faster), the pretrigger percentage is fixed and can not be changed.

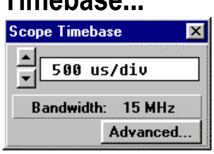


For PCI-44x series oscilloscopes, advanced triggering features are available by pressing the Advanced button on the Trigger control. This will activate the dialog box shown to the left.

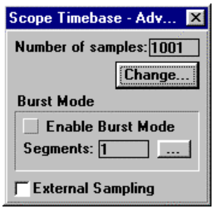
From the Advanced Trigger settings control, you may select a variety of specialized trigger functions:

- Hardware Autotrigger – When enabled, the scope will trigger itself if normal triggers are not received within a preset time period.
- Real-Time Pretrigger – Provides more accurate pretrigger control than is available with the standard trigger controls.
- Trigger Filtering – Provides a High Frequency or Low frequency reject applied to the trigger signal (does not affect the quality of the signal for measurements).
- Trigger Gain – Produces a fixed gain of x1, x2, or x10 to the trigger signal.
- Trigger ARM – When enabled, this will prevent the scope from triggering until the Arm condition has been met. The scope can be Armed on either an external TTL signal or via the External BNC input.

Timebase...



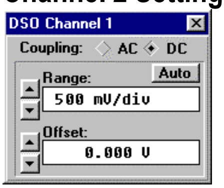
The timebase knob selects the sweep time (in seconds/division) that the oscilloscope uses while acquiring a sweep of data. The measurement bandwidth is indicated below the knob. For the Real-Time sampling mode (2µs/div and slower) the bandwidth is 1 MHZ. For the Equivalent-Time sampling mode (1.8µs/div and faster) the bandwidth will increase to the rated bandwidth of the installed oscilloscope (50MHz, 100MHz, 200MHz, or 300MHz).



For PCI-44x series oscilloscopes, advanced timebase features can be accessed by selecting the "Advanced" button from the timebase control. This will activate the control shown to the left. From the Advanced Timebase control, the number of samples to be used for each sweep acquired by the instrument can be set.

Burst mode is a specialized acquisition mode where the oscilloscope will "divide" the number of samples in the sweep into a specific number of data "segments". It will then treat each of these data segments as an independent sweep of data, with 0% pretrigger for each segment. After each segment is filled, the oscilloscope will re-arm itself for

Channel 1 Settings...
Channel 2 Settings...



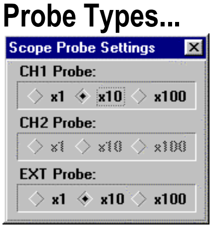
These controls set the vertical range (in Volts/div) and offset (in Volts) for the respective oscilloscope channel. The range and offset controls take into account the probe type that is being used (i.e. the voltage range and offset are relative to the connection point at the tip of the probe).

As the offset level is changed, the ground level indicator (i.e. the horizontal dashed line) will move up and down on the oscilloscope display panel. If the offset voltage is greater than (or less than) the range of voltages displayed on the oscilloscope display, the ground level indicator will disappear.

The input coupling can be set to be either AC or DC. DC coupling permits signals from DC to the full bandwidth of the oscilloscope to be measured. AC coupling will block the DC portion of the signal being measured. AC coupling is used to eliminate the DC offset from the signal being measured.

The Auto button will automatically set the Range and Offset controls to the appropriate settings to view the incoming signal. There will be instances where the Auto button will not find an appropriate setting.

Probe Types...

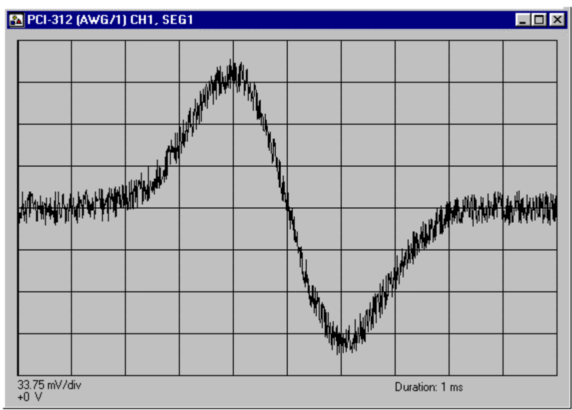


This control selects what type of probe (X1, X10, or X100) is attached to each BNC on the instrument. Note that BenchTop adjusts all vertical range and offset values according to the values set for the probe settings.

This means that if your scope has vertical ranges from 10mV/div to 1V/div and you set the probe type to x10 (times ten), then the vertical range control for the scope will then go from 100mV/div to 10V/div. The vertical range is always referenced to the *probe tip*.

The Arbitrary Waveform Generator Display

Support for PC Instruments Arbitrary Waveform Generators (PCI-311/312) is provided through the Arbitrary Waveform Generator Display in BenchTop for Windows. An Arbitrary Waveform Generator is referred to as an “Arb” or as an “AWG”. To create an Arb display, press the Arb button on the Toolbar or select New Arb Display from the Display menu. By default, the Arb display shows the vertical range of the Arb (Volts/division), the offset voltage, and the duration (in seconds) of the first memory segment (see below).



Waveform Display

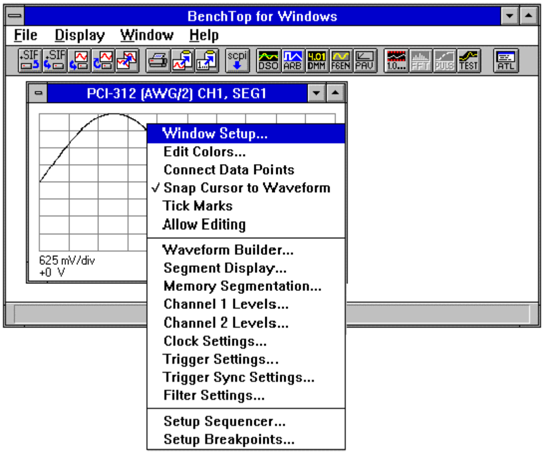
As shown above, the Arb display shows the contents of the Arb’s memory on an 8x10 grid similar to an oscilloscope display. If the Arb’s memory is segmented (see below), only one segment of the memory can be displayed at a time. In order to view each of the segments of the Arb's memory, create multiple Arb control panels and use each one to view one of the segments. You can change which memory segment is displayed by using the “Segment Display” control as descibed later.

Title Bar

The Title Bar has four pieces of information. The first item is the model number of the arbitrary waveform generator (e.g. PCI-312). The second item is the instrument identifier (e.g. AWG/2). The three letters indicate that this waveform display is for a Arbitrary Waveform Generator and its instrument identifier is 2. The instrument identifier is used by the software to link the display panel to a particular instrument card. The third item identifies the channel that is being displayed. The fourth item indicates the segment being displayed.

Activating the Arbitrary Waveform Generator Control Menu

To activate the control menu for the arbitrary waveform generator, position the mouse cursor somewhere over the waveform display and press the **right** mouse button. As shown in the following figure, a pop-up menu will appear. The figure shows the pop-up menu when the Arbitrary Waveform Generator user interface is selected.   If the Function Generator user interface or the Pulse Generator user interface is selected the pop-up menu will have fewer options. See the “Window Setup...” description for further information.



To select an item from the menu, select it with the left mouse button, or drag the highlight bar to the desired item and release the left mouse button.

Window Setup

The Window Setup dialog box lets you select which arbitrary waveform generator and channel is displayed in the instrument display. There is also a control that determines whether the cursor will be attached to Channel 1 or Channel 2 (this control only has meaning if a Dual Channel Arbitrary Waveform Generator is installed and both channels are being displayed).

There is also a selection that permits the arbitrary waveform generator card to be controlled as if it was a function generator, pulse generator, or as an arbitrary waveform generator. This selection is very powerful. While the hardware is an arbitrary waveform generator, BenchTop takes advantage of the “Virtual Instrument” concept and provides the user three different user interfaces to control the same instrument card.

Arbitrary Waveform Generator User Interface

This user interface permits the user to change and control every aspect of the arbitrary waveform generator card. It is the most powerful user interface and the main concepts used to create waveforms are the number of samples in the waveform and the sampling clock of the generator. The controls for the Arbitrary Waveform Generator user interface are described in detail in the section titled Arbitrary Waveform Generator Controls.

Pulse Generator User Interface

This user interface provides controls such as pulse period, duty cycle, and edge transisition times. The main concept used to create waveforms is the pulse period. The user interface software will set the parameters of the arbitrary waveform generator card to the optimal settings to create the desired pulse waveform. The controls for the Pulse Generator User Interface are described in the section titled Pulse Generator Controls.

Function Generator User Interface

This user interface is the easiest to use and the main concept used to create waveforms is the waveform shape and its frequency. The user interface software will set the parameters of the arbitrary waveform generator card to the optimal settings to create the desired waveform. The controls for the Function Generator User Interface are described in the section titled Function Generator Controls.

Edit Colors...

Select this option to edit the colors used in the waveform display. You can change the waveform color(s), grid color, and background color.

Connect Data Points

When Connect Data Points is checked, samples in the Arb’s memory are connected on the screen with straight lines. Uncheck this option to show just the data samples as points on the instrument display.

Snap Cursor to Waveform

When the mouse cursor is moved over the Arb instrument display, a cursor appears and the position (in time and Volts) is displayed at the bottom of the waveform grid. To make the cursor snap to the data in the Arb’s memory, select Snap Cursor to Waveform from the Arb’s control menu. When this option is off, the cursor and readouts will be free-floating with the mouse cursor.

Tick Marks

When Ticks Marks is checked, sub-graticule lines are displayed on the T and Y axis of the waveform grid.

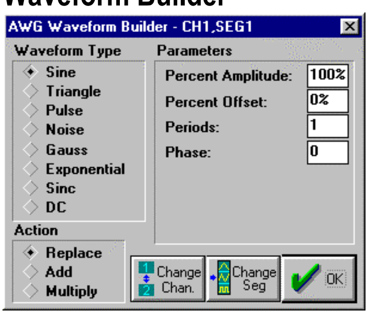
Allow Editing

This option enables waveform editing with the mouse. When it is on (checked) you can edit the displayed waveform by "drawing" directly on the instrument display. The drawing action of the mouse will depend on whether the Snap Cursor to Waveform option (see previous paragraph) is on or off.

If the Snap Cursor to Waveform option is on, the mouse will be in a "rubber-band" editing mode. Position the cursor at the point on the waveform where you want the new vector to start (i.e. the anchor point). Push and hold the left mouse button. Move the mouse cursor away form the anchor point and a new vector will be drawn from the anchor point to the mouse cursor. When you have created the vector you want, release the left mouse button and the new vector will be drawn.

If the Snap Cursor to Waveform option is off, the anchor point will not be attached to the waveform. Position the mouse cursor at the point that you want the new vector to begin. Note that as you move the mouse cursor, the voltage and time of the cursor's location is indicated in the lower center of the display area. When you have the mouse cursor at the start of the new vector, push and hold the left button. Move the mouse cursor to the end of the new vector and release the left button and the new vector will be drawn.

Waveform Builder

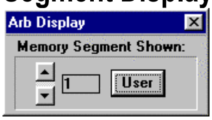


The Waveform Builder is a comprehensive control that allows you to create simple or complex waveforms without entering formulas. To use the Waveform Builder, first select a channel and segment to place the waveform data into by using the "Change Chan" and "Change Seg" buttons. The currently selected channel and memory segment are displayed in the title of the Waveform Builder dialog box. Next choose a waveform type (e.g. Sine, Triangle, etc.), then edit the parameters for the waveform type chosen (or keep the defaults). Finally, select one of the three "action" options: Replace, Add, or Multiply. If Replace is selected, the new waveform data will overwrite the existing contents of the Arb's memory. If Add is chosen, the new waveform data will be added to the waveform already in the destination memory segment. And if Multiply is selected, the data already in the destination memory segment will be multiplied by the new waveform. Once all the selections are made, press OK to load the segment.

Using this technique, you can create complicated signals that are “composites” of simpler waveform types. For example, by creating a single-period sinewave first (using Replace), then Adding a three-period sinewave and then Adding a five-period sinewave, you can create a sine wave with 3rd and 5th harmonic distortion. Similarly, by multiplying successive waveforms you can create modulated signals, damped signals, and chirps with relatively little work.

To close the Waveform Builder, double click the icon in the upper lefthand corner of the dialog box.

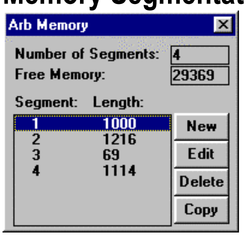
Segment Display



This control sets which memory segment is displayed. Press the Up/Down arrows to scroll through the defined memory segments, or press “User” to jump to a particular segment. To create and edit the lengths of memory segments, use the Memory Segmentation control described below.

The Segment Display control is very useful when using the Waveform Builder control. Use the Segment Display control to view the segment that is being built by the Waveform Builder control.

Memory Segmentation

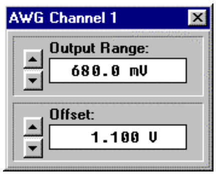


The Memory Segmentation control is used to configure the Arb’s memory. Segmentation refers to subdividing the available memory into separate pieces, so that different waveforms can be placed in each segment. This feature is usually used along with the Sequencer control to create sequenced waveforms.

The New button is used to create new segments and the Delete button is used to Delete segments. To Edit the length of a memory segment, highlight the segment number in the list box and then press the Edit button. Note that when increasing the length of an existing memory segment, the new samples are initialized to zero. When decreasing the length of a memory segment, the waveform contents are truncated.

The number of segments defined and the memory that is available to be used for new segments are indicated in the status boxes at the top of the Memory Segmentation control panel.

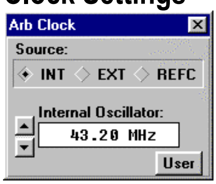
Channel 1 Levels
Channel 2 Levels



These controls are used to set the vertical range and offset for each channel of the Arb. The "output range" refers to the maximum Peak-to-Peak amplitude which the Arb will generate into a **50 Ohm load**. If you are using the Arb to drive a high-impedance (e.g. 1MΩ) load, the output of the Arb will be twice the 50Ω vertical range. Similarly, the Offset assumes a 50Ω output load. The sum of half of the output range (i.e. the peak output voltage) and the offset can not exceed ±6 Volts into a 50Ω load (±12 Volts into a high impedance load). For example, if the output range is set to 10V (Peak-to-Peak), the offset must be between ±1V (10Vpp = ±5V). Again, these values assume a 50Ω load.

The Output Range value is the peak-to-peak amplitude of the output waveform (into a 50Ω load) for a full scale DAC waveform. As an example, if the Output Range is set to 3 Vpp, and the Waveform Builder is used to load a sine wave with a 75% amplitude, the peak-to-peak voltage of the output will be 2.25 Volts.

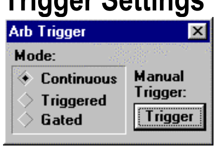
Clock Settings



This control sets the sample clock source and the internal sample rate on the Arb. If the Arb clock source is set to External, clocking is done by applying TTL pulses (up to 35MHz) to the external clock input. If it is set to REFC, clocking is done by the on-board reference crystal at 10MHz. When the source is set to INT, the samples are clocked by the internal, programmable clock oscillator.

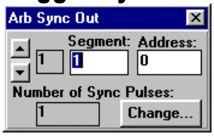
When the Clock Source is set to INT, the frequency readout adjusts the sampling clock for the output DACs on the Arbitrary Waveform Generator. For more information on using the frequency readout, see "Modifying Settings" in Chapter 3. By clicking on the User button a clock rate can be typed in with up to 4 digits of resolution.

Trigger Settings



The Trigger Settings control determines how and when the Arb will output its waveform data. When set to Continuous, the Arb will continuously output its waveform data (including sequenced waveforms if the sequencer is on). When the mode is set to Triggered, the Arb will output one waveform (or one sequenced waveform) each time a positive edge (TTL) is received on the BNC labeled TRIG/REF IN, or when the Trigger button is pressed. In Triggered mode, the Arb will ignore triggers that are applied while the waveform is being generated (i.e. it is not retriggerable). Finally, in Gated mode the Arb will continuously output its waveform data as long as there is a gate signal (TTL High) present on the BNC labeled TRIG/REF IN. Once a waveform starts in the Gated mode, the Arb will completely finish the waveform even if the Gate signal is deactivated during the waveform.

Trigger Sync Settings



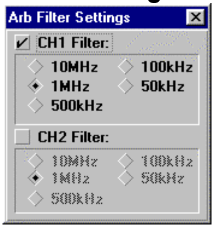
This control is used to determine the position(s) in time of the Sync Output pulse in the Arb's memory. The Sync Output is a TTL signal that can be used as a triggering signal for an oscilloscope or other measuring instrument. It is located on the BNC labeled "SYNC OUT". You can define as many Sync pulses as you want, up to the total number of samples on the Arb.

Each Sync pulse is assigned a segment number and address. These numbers specify which memory segment (see above) and relative address within the memory segment will correspond to the generation of the output pulse. To change the segment number or address of a sync output pulse, simply click on the value to be changed, retype it, and press Enter. Use the Up/Down arrows to view each Sync Out pulse that is defined.

The trigger sync output will remain active for one sample clock period.

The Sync Out pulse will be active every time the chosen segment and relative address are accessed. As an example, if the waveform sequencer is used to repeat segment 1 five times, and a Sync Out pulse is programmed to be in segment 1, the Sync Out pulse will occur five times. Keep this in mind when you are using the waveform sequencer. You could set-up the Arb to output one Sync Out pulse at the beginning of the sequence or you could have the Sync Out pulse happen many times during the sequenced waveform.

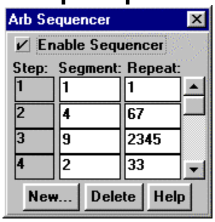
Filter Settings



Each output of the arbitrary waveform generator can have one of several filters switched on to "smooth" the output signal. These output filters are used to reduce the high frequency noise generated by the high speed DACs used on the Arb. When the filter is enabled, you can choose from one of the five available filters: 10 MHZ, 1 MHZ, 500 kHz, 100 kHz, or 50 kHz. All filters are single poll passive low-pass filters, except the 10 MHZ filter, which is a two pole active filter.

Use the lowest frequency filter that will not adversely affect the output waveform. As an example, if the output waveform is a sine wave at 50 kHz, you should use either the 100 kHz filter or the 500 kHz filter. The 100 kHz filter will eliminate more noise but it will also attenuate the signal by 11%. If the output waveform is a 50 kHz square wave, you'll need to use a higher frequency filter in order to prevent the harmonics of the square wave from being attenuated.

Setup Sequencer

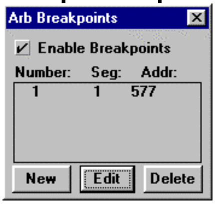


Sequencing refers to linking individual waveforms (segments) together to create a complex signal. The PCI-311/312 provides extensive sequencer control. When the sequencer is disabled, the Arb will output the segment specified as "Step 1" in the sequencer setup.

To create a waveform sequence check the "Enable Sequencer" box and enter the appropriate information into the list box. To add steps to the sequence, press the New button and enter the number of new steps you want to create. To edit a step in the sequencer, simply click on the item (e.g. segment number or repeat number) that you want to change, type in the new value, and press Enter. You can have up to 100 Steps, and each step can be repeated up to 32767 times.

The Delete button will activate two successive dialog boxes. The first dialog box will confirm your choice to delete the selected Step and the second dialog box will give you the choice to delete all Steps after the highlighted step.

Setup Breakpoints



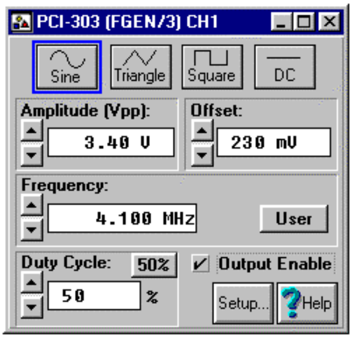
A breakpoint is a position in the Arb's waveform where the output will "freeze" until a pulse is received on the TRIG/REF IN input BNC connector. You can have up to 99 breakpoints, but they must reside in the same memory segment. If the breakpoints are not all in the same memory segment, only the breakpoints in the last memory segment specified will be accepted.

To enable the Breakpoints, check the "Enable Breakpoint" box. Use the New, Edit, and Delete controls to modify the selections in the list box.

The breakpoint control can be used in automatic test equipment to provide a series of different waveforms that are activated with a hardware signal. For example, the Arb could be programmed with the sequencer off and several different waveshapes concatenated together in one segment. Each of the waveshapes has a corresponding breakpoint. Then, every time a pulse is received by the TRIG/REF IN input, a new waveform will be generated.

The Function Generator Control

To activate the PCI-303 Function Generator from the main window select the function generator icon on the Toolbar, or select New Function Generator Display from the Display pull-down menu. The following control panel will appear after the function generator is activated.



Waveform Selection Buttons

The waveform shape is selected by clicking on one of the four Waveform Selection buttons. Four waveforms are available: 1) Sine wave, 2) Square wave, 3) Triangle wave, and 4) DC. Ramp waveforms (also known as sawtooth waveforms), pulses, and distorted sinewaves are also possible by applying a duty cycle that is not equal to 50% (see Duty Cycle Readout and Reset later in this section).

Amplitude Readout

This control varies the peak-to-peak voltage of the selected waveform. The indicated voltage assumes that the output cable is terminated in a 50Ω load. For open loads, the output voltage is 2X greater than the indicated voltage on the Amplitude readout.

If the DC waveform is selected, the Amplitude readout will disappear.

Offset Readout

The Offset readout provides the capability to add a DC level to the sine wave, square wave, and triangle wave. When the DC waveform is selected, the Offset readout controls the amount of DC voltage at the output of the function generator. The maximum output voltage of the generator (peak AC + DC) is listed in the specifications for the generator.

Frequency Adjustment Readout

This readout displays the currently selected output frequency of the function generator.

Duty Cycle Readout and Reset

The duty cycle of the output waveform can be adjusted from 20% to 80%. Ramp waveforms can be created by applying a non-50% duty cycle to a triangle wave, and pulse waveforms can be created by applying a non-50% duty cycle to a square wave. The duty cycle adjustment will not affect the output frequency. The 50% button will return the duty cycle to 50%.

Output Enable

This check box control turns the output waveform on and off. When the Output Enable box is checked, the function generator will output the selected waveform at the selected frequency and amplitude. When the Output Enable box is not checked, the output will be 0 volts. In either case, the output impedance will remain 50Ω.

Setup...

Clicking on this control will bring up a dialog box that describes the function generator that is being controlled by the function generator control panel. Use this control to select different function generator cards to control when there is more than one function generator card installed in the computer.

Help...

The Help function provides an annotated picture of the function generator control panel and also provides a context sensitive help menu for using the Knob controls.

Closing the Function Generator Control Panel

The output waveform is not affected when the function generator control panel is closed (closing the control panel is accomplished by double-clicking on the close box in the upper-left-hand corner). To turn off the output waveform before closing the function generator control panel, uncheck the Output Enable box.

The Digital Multimeter Control

To activate the PCI-501 Digital Multimeter from the main window select the DMM icon on the Toolbar, or select New DMM Display from the Display pull-down menu.
Since the control panel can be resized, a compact panel can be created that only contains the display area and hides the controls.

Display Area

This area displays the result(s) of the measurement. The left side of the display is used to indicate the DC voltage, AC voltage (RMS), DC current, AC current (RMS), or Resistance. The right side of the display is used to display the frequency/period of the AC input (if the installed DMM has a counter option), or the reference value in the Relative measurement mode.

Function Buttons

The measurement functions of the DMM are selected by choosing one of the six Function Buttons. The DMM supports 2-Wire resistance measurements and 4-Wire Resistance measurements. Consult the specifications for the installed DMM to determine when to use the 4-Wire measurement.

Range Buttons

The measurement ranges, including Auto, are selected with these buttons. The most versatile operation is provided by using the Auto range, but faster operation is possible by using a fixed range.

Counter Control

This toggle button has three indications. The counter circuits can be Off, or enabled to measure the input Frequency, or enabled to measure the input Period. The counter is only operational when AC Volts or AC Current is selected, and when the installed DMM has the counter option. The result of the counter measurement will be displayed on the right-hand side of the display.

Note: The response time of the computer will degrade when the counter is enabled on the PCI-501.

Relative Button

This button enables the DMM to make Relative measurements. When the Relative button is clicked, the DMM will take the next reading (the Reference reading) and subtract it from all subsequent readings. The Reference reading will be displayed on the right-hand of the display if the counter is disabled.

Setup...

Clicking on this control will bring up a dialog box that describes the digital multimeter that is being controlled by the digital multimeter control panel and the Reading Rate selector. Use this control to select different digital multimeter cards to control when there is more than one digital multimeter card installed in the computer.

Reading Rate Selector

This control is available in the Setup dialog box and provides the capability to set the Reading Rate of the DMM. The DMM will integrate out the noise generated by the power line when the reading rate is set to an integer number of power line cycles.

Help...

The Help function provides an annotated picture of the digital multimeter control panel.

Closing the DMM Control Panel

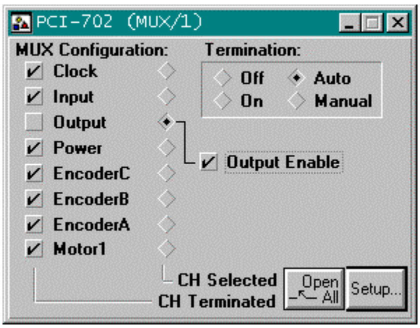
The DMM card is not affected when the digital multimeter control panel is closed (closing the control panel is accomplished by double-clicking on the close box in the upper-left-hand corner). The DMM card will continue to make measurements, but it will not use computer resources to update the display.

Reducing the DMM Control Panel to an Icon

By clicking on the box in the upper-right-hand corner of the function generator control panel the control panel will be reduced to an icon. The DMM will continue to make measurements but the computer will not access the DMM for new measurements to be displayed.

The Multiplexer Control

To activate the PCI-70x Multiplexer Control, select the MUX icon on the toolbar, or select “New Multiplexer Display” from the Display menu. The multiplexer control consists of a graphical representation of the state of the multiplexer as shown below.



MUX Configuration

The multiplexer configuration consists of the channel selected and the termination settings for each channel.

To change which channel is selected, simply click on the appropriate channel identifier. For example, in the dialog box above, click on “EncoderA” to switch to that channel.

You may open all of the channel routing relays as well as the output enable relay at once by pressing the “Open All” button. This button will not affect the termination of each channel, unless the termination mode is set to Auto.

Termination Mode

The termination mode (available on the model PCI-702) determines how channel termination is performed. The termination modes are:

Off None of the channels are terminated by the multiplexer. This is the default termination mode, and the only termination mode that is supported by the PCI-703.

On All channels are terminated by the multiplexer.

Manual Each channel’s termination can be independently controlled.

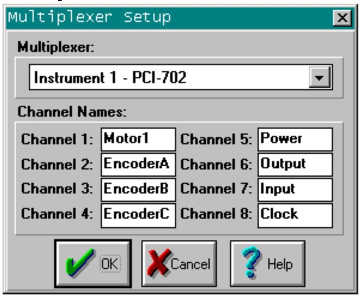
Auto All channels are terminated by the multiplexer except for the selected channel.

When the termination mode is set to Manual, you may change the termination of each channel individually by clicking on the termination check box to the left of the channel identifier.

Output Enable

The output relay of the multiplexer is controlled independent of the channel routing and termination relays. To enable (close) the output relay, check the output enable check box. To disable (open) the output relay, uncheck the box.

Setup



Pressing the Setup button will activate the dialog box shown to the left. From the Multiplexer Setup dialog box you can select which of the installed multiplexers to control. You can also assign identifiers to each channel of the multiplexer. A channel identifier can be any string that describes the function of the specified channel.

Chapter 5
Advanced Features

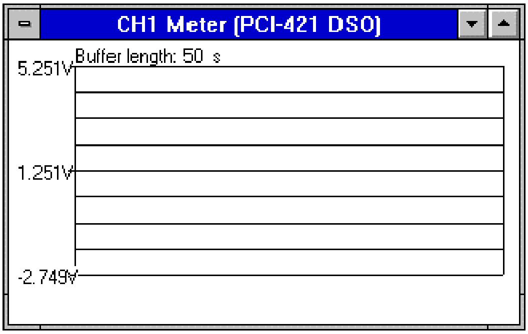
Introduction

The items discussed in this chapter are considered “advanced” features because they extend the basic functionality of your PC Instruments cards through software. These features are not available in BenchTop Lite for Windows.

The Signal Meter

The Signal Meter performs calculations on waveforms acquired by the oscilloscope and plots the results of the calculations on a strip chart display. The Signal Meter can calculate the Maximum Voltage, Minimum Voltage, Peak-to-Peak Voltage, Mean Voltage, Standard Deviation, RMS Voltage, and the estimated frequency of the acquired waveform. By displaying the results of these calculations on a strip chart display, the changes in waveform characteristics can be monitored over a long period of time. The strip chart display can be disabled if only a numeric read-out indication of the calculation result is desired.

To activate the signal meter press the signal meter button on the Toolbar or select “Display|New Signal Meter” from the menu. The signal meter will appear initially as shown below.



To access the control menu for the signal meter display, place the mouse cursor somewhere over the display and press the **right** mouse button. A popup menu will appear listing the available setup options for the signal meter.

Window Setup

The window setup dialog box lets you set which scope and channel are being measured by the signal meter.

Edit Colors

You can edit the colors used to draw the signal meter display by choosing this option.

Choose Font

Choose this option if you want to edit the font that is used to display the signal measurements being performed. This is useful if you want to draw attention to a particular measurement on the screen.

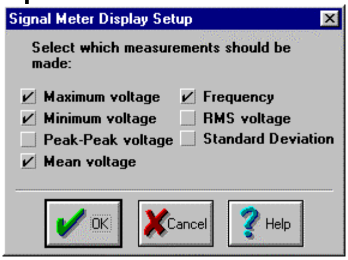
Display Readouts

This option toggles on and off each time it is selected. When on (checked) the signal parameters that have been selected (see Options below) will be displayed below the strip chart.

Display Chart

This option will toggle on and off each time it is chosen. When selected, the strip chart will be displayed. When off, the chart and chart annotations will be removed and the selected measurements (see Options below) will be shown using a numeric readout.

Options



To determine which measurement options are made by the signal meter, select "Options" from the control menu for the display. Any combination of the available measurements can be made by the signal meter.

Set Capture Rate

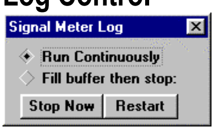
The capture rate is the time between signal measurements. For example, if you set the capture rate to 200ms, the signal meter will make 5 waveform measurements per second - *assuming the scope is being triggered fast enough and the scope timebase is less than 20ms/div*. The shortest time between measurements is dependent on the speed of your computer, the rate at which the scope is triggered, and the timebase of the scope. Typical values for the capture rate are from 100ms to 1s, depending on how often you want to make the selected measurements.

Note that if the scope is not receiving triggers or the capture rate is set faster than the scope can acquire data, the signal meter will “fall back” to the fastest rate at which it can operate. This ensures that each waveform from the scope is only analyzed by the signal meter once.

Set Buffer Size

The buffer size refers to the number of measurements that the signal meter will store in its buffer. Each time the buffer “wraps around”, you will see a vertical line on the chart which indicates the start of the buffer memory. If you are using the signal meter to continuously measure some parameter(s), this value is unimportant.

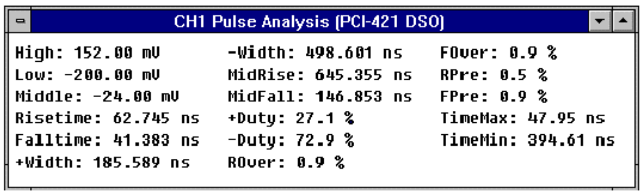
Log Control



This dialog box lets you control the state of the signal meter. When the signal meter is in the “Run Continuously” mode, it will overwrite the signal meter buffer (see Set Buffer Size above) when it is full. If you choose “Fill Buffer then Stop”, the signal meter will acquire however many waveforms are necessary to fill the current buffer size, then stop. This is useful if you want to make a particular number of measurements and then store them to the hard drive.

The Pulse Analysis Display

A wide variety of pulse analysis parameters can be measured using the Pulse Analysis Display. Examples of the pulse analysis measurements include risetime, falltime, overshoot, and dutycycle. All of the pulse analysis measurements are shown in the sample pulse analysis display below, and described in more detail below.



To access the control menu for the pulse analysis display, place the mouse cursor somewhere over the display and press the **right** mouse button. A popup menu will appear listing the available setup options for the window.

Window Setup

The window setup dialog box lets you set which scope and channel are being analyzed by the pulse analysis display.

Edit Colors

You can edit the colors used to draw the pulse analysis display by choosing this option.

Choose Font

Choose this option if you want to edit the font that is used to display the signal measurements being performed. This is useful if you want to draw attention to a particular measurement on the screen.

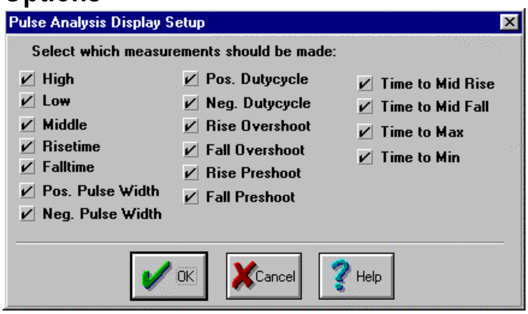
Use Histogram

By default, the method used to calculate the pulse parameters for the Pulse Analysis display uses histograms to generate a Probability Density Function (PDF) for the signal data. This technique is used to estimate the signal High and Low values, and many of the other pulse parameters follow from this data. Optionally, you can tell BenchTop to ignore the histogram data and instead to use the signal peaks (maximum and minumum) instead of the estimated High and Low values. For some signals (e.g. triangle waves) this will result in better pulse parameter results.

Use Histogram Averaging

When this option is on (checked), the method used to calculate the pulse parameters will average histogram data as new sweeps of data are recieved from the oscilloscope. If the input signal is noisy, this may improve the quality of the parameter estimates. When this option is off, pulse parameters are calculated from each sweep of data without regard to past waveshapes.

Options



To determine which measurement options are made by the pulse analysis display, select “Options” from the control menu for the display. Any combination of the available measurements can be made by the pulse analysis display. The Pulse parameter definitions used by BenchTop are derived from the IEEE Standard Pulse Terms and Definitions (IEEE Std. 194-1977). The techniques used by BenchTop are derived from the IEEE Standard on Pulse Measurement and Analysis by Objective Techniques (ANSI/IEEE Std 181-1977).

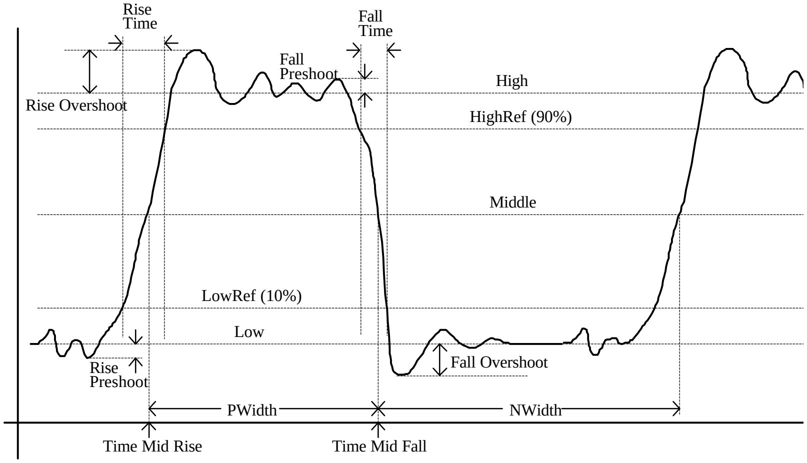
The following table lists each of the Pulse Parameters that can be calculated by the pulse analysis display.

Parameter:	Description:
High	The highest portion of a pulse waveform which represents a nominal value for the pulse.
Low	The lowest portion of a pulse waveform which represents a nominal value for the pulse.
Middle	The level halfway between the Low and High levels.
Risetime*	Time taken to make a positive transition from a level 10% higher than the Low level to 10% lower than the High level.
Falltime*	Time taken to make a negative transition from a level 10% higher than the Low level to 10% lower than the High level.
Positive Pulse Width	The time taken for a pulse to make a negative transition after a positive transition, measured as the pulse crosses the Middle level. If the pulse is periodic, the sum of Positive pulse width and negative pulse width is

Parameter:	Description:
	the period of the pulse train.
Negative Pulse Width	The time taken for a pulse to make a positive transition after a negative transition, measured as the pulse crosses the Middle level. If the pulse is periodic, the sum of Positive pulse width and negative pulse width is the period of the pulse train.
Positive Dutycycle	An estimate of the positive dutycycle expressed as the Positive Pulse Width divided by the period.
Negative Dutycycle	An estimate of the negative dutycycle expressed as the Negative Pulse Width divided by the period.
Rise Overshoot (ROver)	An estimate of the difference between the High level and the peak amplitude of the first rising edge for the signal. Expressed as a percentage of the waveform amplitude.
Fall Overshoot (FOver)	An estimate of the difference between the Low level and the negative peak value of the first falling edge for the signal. Expressed as a percentage of the waveform amplitude.
Rise Preshoot (RPre)	An estimate of the difference between the Low level and the negative peak value of the first rising edge for the signal. Expressed as a percentage of the waveform amplitude.
Fall Preshoot (FPre)	An estimate of the difference between the High level and the peak value of the first falling edge for the signal. Expressed as a percentage of the waveform amplitude.
Time-to-Mid Rise (MidRise)	The time from the trigger point to the middle of the first rising edge of the signal.
Time-to-Mid Fall (MidFall)	The time from the trigger point to the middle of the first falling edge of the signal.
Time-to-Max	The time (in seconds) from the trigger point to the highest sample acquired in the signal.
Time-to-Min	The time (in seconds) from the trigger point to the lowest sample in the signal.

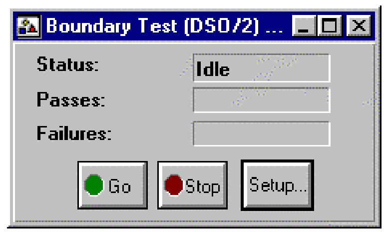
*Risetime and Falltime estimates require at least four samples present on each edge.

Below is a graphical picture of the pulse parameters:

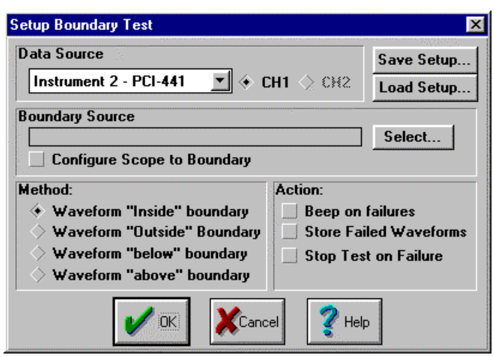

Rise
Time
Fall
Time
Fall
Preshoot
High
Rise Overshoot
HighRef (90%)
Middle
LowRef (10%)
Low
Rise
Preshoot
Fall Overshoot
PWidth
NWidth
Time Mid Rise
Time Mid Fall

Boundary Testing

In BenchTop, boundary testing refers to comparing "live" data (from an oscilloscope card) to a pair of stored waveforms (stored in a single reference file). Boundary testing can be done in BenchTop for Windows using the Boundary Test dialog box. To activate the boundary test dialog box, press the boundary test button on the Toolbar or select “Display|New Boundary Test” from the menu. The Boundary test dialog will appear as shown below.



Before the boundary tester can be used it must be configured. To setup the test to be done, press the Setup button. The following Boundary setup dialog box will appear.



The following items are set from the boundary test setup dialog box:

Data Source

The data source is the oscilloscope channel that is used to acquire the signal to be tested. If you have more than one PC Instruments oscilloscopes installed in your PC, you can select a particular scope from the drop-down list box (see figure above). For two channel scopes, you can select either channel 1 or channel 2 using the radio buttons to the right of the list box.

Boundary Source

The boundary source is the file which is used to compare to the “live” data. The boundary source can be either a two-sided boundary or a single sided “reference”. To create boundary files, use BenchTop to acquire the “gold” signal and store it to the hard drive. To convert the reference waveform to a two-sided boundary, use the PC Instruments Data Exchange program (PCIDX.EXE) included with BenchTop.

If you check “Configure scope to boundary” the oscilloscope will automatically be setup to match the timebase, vertical range, etc. in the boundary file. If the scope is not setup properly prior to starting the test BenchTop will display a message warning you that the scope does not match the boundary setup.

Method of Test

The comparison technique used is set in the Method group box. Each selection is described in the table below.

Method:	Signal passes if...
Inside	The signal is between the two limiting waveforms.
Outside	The signal is either completely above the upper limiting waveform or completely below the bottom limiting waveform.
Below	The signal never exceeds any limiting waveform(s) in the boundary source.
Above	The signal never goes “beneath” any limiting waveform in the boundary source.

Actions

The selected action(s) determine how BenchTop should respond when the boundary test fails. If “Beep on failures” is selected, the system will beep (or play the System Exclamation sound if a sound card is installed) each time a failure is detected. If “Store failed waveforms” is selected, the “bad” data will be stored to a series of files with incrementing file names (starting at FAIL000.DIF). Finally, if “Stop test on failure” is checked BenchTop will test until a failure is detected, then stop testing.

Saving and Loading Test Setups

You can store the entire setup for a boundary test to a file on your hard drive by pressing the “Save Setup...” button. To reload the test setup later, simply press “Load Setup...” and select the filename that you used to save the test under.

Starting the Boundary Test

To start the boundary test once it has been configured, simply press the “Go” button on the boundary test dialog box.

Stopping the Boundary Test

To stop the boundary test while it is running, press the red “Stop” button on the boundary test dialog box.

Chapter 6
Automated Test Language

Introduction

BenchTop Automated Test Language (ATL) is an interpreted programming language built into BenchTop for Windows. The purpose of BenchTop ATL is to facilitate the generation of automated test sequences. ATL provides decision making, operator messages, and the ability to send SCPI commands to the instruments. A standard automated test may include:

- Interaction with the operator
- Configuration of the instrument(s) in the PC
- Comparison of signals to existing boundary files
- Summarizing or storing results for the operator

BenchTop ATL provides a flexible format and standard set of functions and language constructs which simplify the automated test generation. Using BenchTop ATL, the programmer is not required to write low-level code to control the oscilloscope, handle the boundary comparisons, or interface to the operator through the Windows API. All of these details are handled by ATL and accessed through predefined functions that are embedded into BenchTop ATL. The Automated Test Language interpreter is not available in BenchTop Lite for Windows.

Invoking the ATL Interpreter

An ATL file can be executed in one of two ways. The first method is to click on the ATL button on the Toolbar or choose `Execute ATL File` from the `File` menu. This will show a standard file dialog box from which you can choose the ATL file to run. Alternatively, you can specify an ATL file on the command line when starting BenchTop. This method is useful if you want BenchTop to always run an automated test when started by an operator. To specify the ATL file this way, use the program manager to edit the BenchTop icon's properties and add the name (including path) of the ATL file to the "Command Line" string for the program.

Language Structure

Programming BenchTop ATL is similar to programming in Basic, with some similarities to C programming. Input files are in standard ASCII format and may be created and edited by any ASCII editor (e.g. Notepad for Windows). In general, the BenchTop ATL format is not case sensitive. That is, upper and lower case letters are not significant. Exceptions to this are the naming of pre-processor macros and the comparison of String type variables. Lines in an ATL file may be up to 255 characters long (after macro expansion).

ATL Constructs

A "construct" refers to the basic syntax used to perform fundamental operations. The following is a list of BenchTop ATL constructs:

Declare

The Declare construct is used to define user variables. See the "Variables" section below.

If .. else .. endif
If .. [command]

The If construct can be used to perform conditional operations within the ATL file. For example,

```
Declare I as Integer
I = 4
if I < 5
/* this section will be executed */
else
/* this section will be skipped */
endif
```

Optionally, the If construct can be used as shown below,

```
Declare I as Integer
I = 4
if I < 5 Message "I is less than 5"
// program continues...
```

In the second case, the If construct can not have an "else" portion, and there is no "endif" given afterward.

The variable(s) used in the If construct can be of any supported type (see "Variables", below) and the comparison logic used may be any one of the symbols from the table below.

Symbol	Meaning
<	Less than
>	Greater than
=	Equal to
<>	Not equal to
<=	Less than or equal to
>=	Greater than or equal to

Goto

The Goto construct can be used to jump to another location in the file. For example,

```
/*  ... */
begintest:         // Define a label
/* ... */
  // repeattest is a user variable
If repeattest Goto begintest
/* ... */
```

The label specified after Goto may be defined anywhere in the file, before or after the Goto statement.

GoSub <label> .. Return

The GoSub / Return construct re-directs the interpreter to another location in the file and then returns to where the GoSub appeared. For example,

```
Message "Program start"
GoSub Sub1
Message "Program complete."
End
Sub1:
       Message "Performing Subroutine 1"
Return
```

GoSub statements may be nested up to 25 levels deep.

`End`

The End directive will terminate the ATL interpreter and close the ATL file. The End statement is optional and does not have to appear in the input file.

Pre-processor Directives

A “Pre-processor Directive” as used in BenchTop ATL is a statement in the ATL file which is used to perform macro substitutions, include other files, or provide for comments within the file. The term “Pre-processor” refers to the method whereby the items are scanned and interpreted as the file is read, making the reader of the file (the ATL interpreter) unaware that some substitution or change is being made to the file. The following pre-processor directives are supported by BenchTop ATL:

`#include<filename>`

This directive can be used to re-direct input to another file. When the ATL interpreter encounters the #include statement, it automatically opens the file specified and begins executing lines from the included file.

`#define`

This directive can be used to declare a symbol which can be used later within the ATL file. For example, you could specify a string at the beginning of an ATL file like:

```
#define TESTNAME "Automated Test for Amplifier B"
```

Then you could use this "macro" later simply by referencing TESTNAME,

```
Message "Please connect to test point 3" TESTNAME
```

In this case, the resulting message box would have "Automated Test for Amplifier B" as the title bar. Symbols defined in this way can also be used with the #ifdef/#ifndef directives described below.

#ifdef MACRONAME ... #else ... #endif
#ifndef MACRONAME ... #else ... #endif

These directives ("if defined" and "if not defined") can be used to conditionally execute a series of lines within an ATL file depending on the existence of a predefined macro. For example, if you define a symbol at the beginning of an ATL file like:

```
#define DOTEST1
```

Then later in the file you could execute a series of lines depending on the existence of the DOTEST1 macro,

```
#ifdef DOTEST1
        Message "Doing test #1"
#else
        Message "Not doing test #1"
#endif
```

Using pre-processor macros in this way is equivalent to hard-coding constants that can not be changed during run-time.

Comments

BenchTop ATL supports two forms of comments. The first comment format is begun by a forward slash - asterisk combination (/*) and ended with an asterisk - forward slash (*/). This style of comment can be anywhere within a line and can span across many lines. The second comment format is begun with a double forward slash (//) and extends to the end of the line.

An ATL input file may contain comments at any point in the file. Comments are ignored by BenchTop.

Variables

Within a BenchTop ATL file you can define variables of several types to store information as the ATL file is executed. A variable must be defined before it can be used, but variable declarations may be placed anywhere within the file. To define a variable, use the Declare statement:

```
Declare <varname> AS <vartype>
```

Where <varname> is the name of the new variable and <vartype> is one of the supported variable types listed in the table below.

Variable Type	Description	Data Range
Integer	Signed 16 bit integer	-32768 to +32767
Float	Signed 32 bit floating point	-3.4e38 to 3.4e38
String	Character data array	Dynamic, maximum 32768
Data	Waveform data	Dynamic, as needed
Dialog	User-defined dialog box	Dynamic, as needed

A variable name must begin with a letter and may be up to 30 characters long. Reserved words (e.g. Declare, If, Else, Endif, Goto, End, Integer, etc.) may not be used as variable names.

After a variable has been declared, it may be used in assignment and comparison statements wherever supported. The following lines illustrate some use of variables:

```
Declare str As String            // Declare a string var
Declare signumber as Integer     // Declare an integer
signumber = 10                   // Assign value
str = "Testing signal number "   // Assign string
str += signumber                 // Append "10" tostring
Message str                      // Display the string
If signumber > 5                 // Do a comparison
 Scpi "trigger:level 1V"         // Issue a SCPI command
Else
 Scpi "trigger:level -1V"        // Issue a SCPI command
Endif                            // End the IF construct
```

Assignment, comparison, or mathematical operators must always be preceeded by and followed by whitespace (i.e. space or tab character). Note that conversion from Integer or Float variable types to Strings is automatic when assignment or concatenation is being done. For example, in the statement

```
str += signumber
```

from the example above, the value stored in the integer variable `signumber` is automatically converted to a string and concatenated to the string variable `str`.

Numeric Variables

The Integer and Float variable types may be used in mathematical statements or comparisons. The typical syntax for such a statement is:

```
<variable> <operator> <variable or literal value>
```

The following table lists the types of operations that can be performed on these types of variables. Note that conversion to/from inteter or floating point types is done automatically. For example, it is permissible to assign a float variable to an integer, or vice versa.

Manipulator:	**Meaning:**
=	Equals (assignment)
+=	Add to
-=	Subtract from
*=	Multiply by
/=	Divide by

In addition to the operators above, integer types can be modified in a bit-wise fasion using the operators below

Manipulator:	**Meaning:**
|=	OR with
&=	AND with
^=	XOR with

For example, you could manipulate an integer variable as shown below:

```
Declare i as Integer
Declare j as Integer
i = 0
i |= 256  // i now contains the bit pattern 0000000100000000
j = 1
j |= i    // j now contains the value 257
j &= 2    // j now contains the value 0
```

String Variables

String variables can be used to hold any kind of information. If the string needs to contain quotes, use a backslash-quote pair. To place a backslash in the string, use a double backslash. For example:

```
Declare filename as String
Declare instruction as String
filename = "c:\\test\\rev_0\\boundry.dif"
instruction = "Please type \"AMP ON\" on the terminal."
```

Data Variables

The Data variable type is of special interest because it is used to store waveform and boundary information. The sample code below defines two Data variables:

```
Declare waveform as Data
Declare boundary1 as Data
```

Once a Data variable has been declared it can be assigned by referencing "live" data from a PC Instruments scope, or by referencing data that has been stored in a file:

```
waveform = "sense1:data1"  // Chan 1 data from Scope #1
boundary1 = "bound001.dif"  // SCPI-DIF formatted file
```

The stored waveform file format may be either a reference waveform or a boundary waveform. The difference between a reference and a boundary is that a boundary waveform has two data sets (an upper reference and a lower reference) whereas a reference waveform is a single waveform. The exact type of Data being assigned to the variable is determined by the ATL interpreter automatically.

Once Data variables have been assigned, they may be manipulated much the same way as simple variable types. That is, they can be used in If statements like:

```
If waveform = boundary
      Message "The signal passed the boundary test!"
Else
      Message "The signal failed the boundary test."
Endif
```

In the example above, the variable boundary would be a boundary waveform. The "=" operator would return TRUE if the data in the waveform variable is "within" the boundary. That is, if each data point of the waveform is beneath the corresponding upper-waveform data point and above the corresponding lower-waveform data point in the boundary. If any data points in the waveform are outside of this boundary, the If test will execute the Else condition.

In addition to comparing waveform data to boundary files, waveforms can be compared to stored waveforms ("single sided" boundaries). For example, if a signal should never exceed some previously measured signal. This could be done like:

```
Declare waveform as Data
Declare limitwave as Data
waveform = "sense1:data1"
limitwave = "c:\\limits\\ref004.dif"
if waveform > limitwave
  MessageBeep 3
  Message "The signal exceeded the reference waveform"
Endif
```

Note that anytime two Data variables are compared, the comparison is done on a point-by-point basis. That is, sample #1 of the waveform is compared to sample #1 of the reference, sample #2 is compared to sample #2, etc.

Dialog Variables

The Dialog datatype allows you to create completely custom dialog boxes to be used in the ATL file. To create custom dialog boxes, you will need a compiler capable of creating Dynamic Link Libraries (DLL's). For example, Borland C++ or Microsoft Visual C++.

Once you have created your dialog box resources and compiled them into a DLL, you can access them from BenchTop ATL simply by referencing the name of the DLL, the resource id of the dialog box, and the resource IDs of the controls on the dialog box that you want to interact with. For example,

```
Declare testoptions as Dialog
testoptions = "c:\\bin\\atldata.dll" 1 101 104 105
```

The code above assigns the dialog box (testoptions) to the resource in the file ATLDATA.DLL with the resource id "1". The assignment also specifies that we want to communicate to three items within the dialog box, who's id numbers are 101 104 and 105. These items may be check boxes or radio buttons, for example.

To activate a dialog box (display it to the operator modally), use the function DlgExecute. For example,

```
Declare ok as Integer
ok = DlgExecute testoptions
```

As the example shows, the DlgExecute function returns an integer. This value indicates how the operator dismissed the dialog box. For example, by pressing OK or CANCEL. The return value is equal to the resource id of the button used to dismiss the dialog box.

The table below lists the functions that are used to interact with custom dialog boxes.

Function	Parameters	Description
DlgExecute	<dialogname>	Execute the dialog box.
SetDlgItemState	<dialogname>,<item>,<0|1>	Set the item on or off.
GetDlgItemState	<dialogname>,<item>	Get the state of the item.
SetDlgItemText	<dialogname>,<item>,<text>	Set the item’s text.
GetDlgItemText	<dialogname>,<item>	Get the item’s text
AddDlgList	<dialogname>,<item>,<text>	Add a string to a listbox.

Function	Parameters	Description
GetDlgListSel	<dialogname>,<item>	Get the index of the selected item from the list.
SetDlgListSel	<dialogname>,<item>,<num>	Set the selected item for the listbox to <num>.

In each of the functions listed above, <dialogname> refers to the name of the Dialog variable, and <item> refers to the resource id of the item within the dialog box being accessed.

Functions

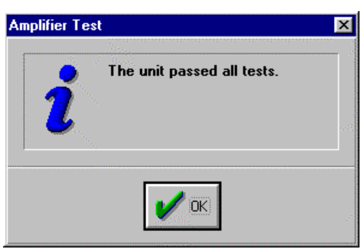
There are several functions built in to BenchTop ATL. Each of these functions are described below.

Message <message string> <title string>

The Message function can be used to display a simple message or instruction to the operator.   For example, the line:

```
Message "The unit passed all tests." "Amplifier Test"
```

would produce the following message box:



Messages produced using the Message function are displayed modally. That is, program execution is suspended until the message has been dismissed by the operator.

MessageBeep <beepcode>

The MessageBeep function can be used to cause Windows to beep or play a system sound if the PC is equipped with a sound card.   The <beepcode> parameter specifies which of the system sounds is played:

Beep Code:	System sound (in win.ini):
0 (default)	Beep the PC Speaker
1	System Default
2	System Asterisk
3	System Exclamation
4	System Hand
5	System Question

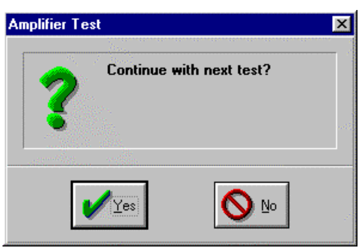
The MessageBeep function is useful for getting the operator's attention or alerting the operator to some condition (particularly in process control applications). If the PC does not have a sound card, or the beep code is not recognized, the MessageBeep will cause the PC speaker to beep.

Confirm <message string> <title string>

The Confirm function allows the ATL program to prompt the operator for a Yes/No response to some question. The return value from the Confirm function may be assigned directly to an Integer variable. The following is an example:

```
Declare go as Integer
go = Confirm "Continue with next test?" "Amplifier Test"
if go = 1 // The operator pressed "yes"
....
endif
```

The example would generate a confirmation message as shown below:



Confirmation Messages are displayed modally. That is, program execution is suspended until the message has been dismissed by the operator.

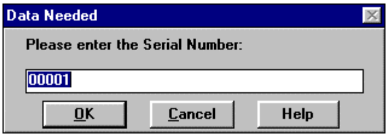
Prompt <prompt string> <default answer string> <title string>

The Prompt function can be used to get a string from the operator. The return value from the Prompt function can be assigned directly to a String variable. Below is an example:

```
Declare sn as String
sn = Prompt "Please enter the Serial Number:" "00001"
```

```
/* ... */
Declare msg as String
msg = "Unit #"
msg += sn
msg += " failed at least one test."
Message msg "Test Results"
```

The example would generate the Prompt message shown below:



If the Cancel button is pressed by the operator, the returning string will be cleared ("").

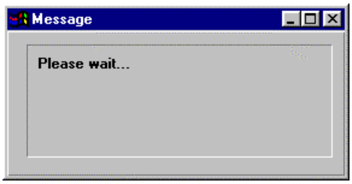
Poster <message> <title> [xposition] [yposition]

The Poster function is a special message function which can be used to display test progress or other information to the operator when no response is needed (for example, while boundary testing is being done or during a timed delay). There may only be one Poster message displayed, and the Poster message can be removed from the screen by specifying empty quotes as the message string.

For example, the Poster message could be used as shown below:

```
Poster "Please wait..."
Delay 1000 // pause for a second
Poster ""
```

In the example, the following message would be shown for one second then removed from the screen:



The Poster message is not shown modally. That is, if it is left on the screen, it will not interfere with other messages or prompts to the operator. The Poster window may be placed at a specific location on the screen by supplying the optional xposition and yposition coordinates (in pixels).

SCPI <scpi string>

This function can be used to send a SCPI command to any of the PC Instruments configured in the PC. Complete SCPI documentation is provided in the BenchCom User's Manual (Appendix A and B). Also see Appendix B of this manual for information about the SCPI commands which can be used to control the BenchTop user interface.

SCPIResponse

This function retrieves a response from the SCPI command parser and places the result in a variable. The return value may be assigned to any type of variable that has been defined. Before calling SCPIResponse, you must call SCPI with the appropriate Query. For example, the following code queries the current trigger level, retrieves the value, increases it by a Volt, then sets the trigger level to the resulting value:

```
Declare triglevel As Float
Declare msg as String
Scpi "Trigger:level?"            // Query
triglevel = ScpiResponse         // Retrieve
triglevel += 1.0                 // Increment
msg = "trigger:level "           // Build string
msg += triglevel
Scpi msg                         // Set trigger level
```

Delay <millisecond delay>

This function can be used to suspend the ATL interpreter for the specified number of milliseconds (up to 32.7 seconds).

Max <data variable type>
Min <data variable type>
Mean <data variable type>

These functions will analyze the waveform data within a Data variable and return the appropriate value, in Volts. The result can be assigned directly to a Float type variable. For example:

```
Declare maxvoltage as Float
Declare signal as Data
Declare str as String
signal = "sense1:data1"      // Get ch1 data
maxvoltage = Max signal      // Calculate max value
str = "The maximum voltage: " // Build a string...
str += maxvoltage
Message str                  // Display message
If maxvoltage > 1.5         // Compare max to 1.5V
      MessageBeep 3          // Alarm sound
```

```
        Message "Signal is too large!"
Endif
```

WriteToFile <filename> <string> <APPEND|OVERWRITE>

This function will write a string to the specified file. The file may be appended (the default, if no third parameter is given) or overwritten with the string data. The following is an example:

```
Declare id as String
Declare results as String
id = Prompt "Please enter the serial number:" "00001"
// ...later when tests are done...
results = id
results += " passed all tests"
WriteToFile "amptest.out" results
```

WinExec <program name>

This function will start another windows program. Any command line parameters to the program to be started may be placed in the <program name> string. For example:

```
WinExec "notepad.exe amptest.out"
```

GetIniString <filespec> <section> <parameter> <default>

This function will retrieve data from a Windows INI file. For example, you may have a file called TEST.INI which contains the section:

```
[TestParms]
MaxVolts=3.5
```

Then you could use GetIniString within an ATL file to retrieve this information:

```
Declare max as float
max = GetIniString "test.ini" "TestParms" "MaxVolts" "0"
```

ClearGosub

This function clears all pending "Returns". That is, if your test procedure is nested using the Gosub/Return construct, you may use ClearGosub at any point to clear any nested Return statements. This can be useful in situations where your test needs to abort or reset.

LogConsole <message> <title>

The LogConsole function is used to create a scrolling "console" display which shows the operator up to 15 lines of text. Each time the LogConsole function is issued, the new message is appended to the bottom of the console and the display is scrolled up if needed. Each message on the log console can be up to 80 characters long. If the message is "" (empty quotes), the log console is removed from the screen.

Error Codes

The following table enumerates the run-time errors which may be reported by the ATL interpreter while executing an ATL file.

Error Value:	Description:	Possible cause:
-1	Syntax Error	Misspelling of function or variable name.
-2	Declaration Error	Misspelling of variable type.
-3	Goto Error	Misspelled label.
-4	If construct Error	Bad syntax, unknown symbol, or misspelled variable.
-5	If construct Error	
-6	If construct Error	Else encountered without corresponding If.
-7	If construct Error	Endif encountered without corresponding If.
-8	Stack Error	Stack overflow*
-9	Stack Error	Stack underflow*
-10	Can't Open File	Misspelled filename or drive.
-11	Assignment Error	Misspelled variable name.
-12	Assignment Error	Uninitialized variable or type mismatch.
-13	UI Error	Attempted to destroy absent UI device*.

*Internal Error, please report to PC Instruments.

The most common cause of errors in an ATL script are:

- Failure to preceed and follow assignment symbols with whitespace. For example, the line
  `i+=2`
  Is an error and should be corrected to appear like:
  `i += 2`

- If statement not matched with an EndIf statement. Unless you are using a single-line IF statement such as:

  ```
  IF a > 2 goto jumpa       // OK, no endif is needed
  ```

  You must “close” the IF block with an EndIf statement.

- Function or variable name spelled incorrectly. Double check the spelling of functions and variables.

Debugging

To assist you in debugging your ATL scripts, there are two predefined macros that affect the execution of the ATL interpreter. To use either of these predefined macros, simply place them at the top of your ATL file. When you are finished debugging the script, simply remove or comment-out the #define statements.

The first of the predefined macros is the ATLTRACEWINDOW macro. When this symbol is defined as shown below, the ATL interpreter will display information about the ATL script as it is executed.

```
#define ATLTRACEWINDOW 1
```

In addition to displaying runtime information in a window, the ATLTRACEWINDOW macro will also create a trace file, called “atl.out” in the current directory. This file will contain more information about the runtime behavior of your ATL script than is displayed in the trace window.

The second predefined macro affects the speed at which your ATL script is executed. To use it, define the macro as shown below

```
#define ATLLINERATE x
```

Where x is replaced by a value representing the number of milliseconds to pause between lines in the ATL script. The default linerate is 50ms. Larger values will cause the ATL script to execute slower, giving you time to watch the trace window or other output from your program.

Example Listing

The following listing is an example ATL program that illustrates how many of the BenchTop ATL commands can be used. In this example, the operator is prompted for a serial number for the board to be tested. Then the oscilloscope is setup and the boundary file (which would have been created earlier) is loaded. After this, the operator is prompted to "connect up" to the unit to be tested and a boundary test is performed. The results of the boundary test are then presented to the operator as well as logged to a file on the hard drive ("test.out"). After the test is complete, the program asks the operator if the test should be run again.

```
// ****************************************************
// SAMPLE1.ATL - BenchTop ATL Sample File
// ****************************************************

scpi "gui:window:destroy:all" // Erase all windows

Declare waveform as Data      // These are the
Declare boundary1 as Data        // program's variables
Declare isready as String
Declare serialnum as String
Declare msg as String
Declare again as Integer

StartTest:    // used in the Goto statement at end of
test

serialnum = Prompt "Please enter the unit's serial
number:" "00001" "Example Test Routine"

Poster "Setting up instruments..." "Please Wait"

scpi "mmemory:recall 1,boundary.dif" // Setup the scope
boundary1 = "boundary.dif"           // Load the boundary

Poster "" // Remove the "setup" message

Message "Connect the scope Channel 1 to the test point."

scpi "init:immediate" // Tell the scope to take a sweep

poster "Waiting for data..."

WaitForData: // wait for the scope to finish sweep.

     Scpi "sense:data:ready?"
     isready = ScpiResponse
if isready = "no" Goto WaitForData

waveform = "sense1:data1" // Assign the data source

poster "Got data."
delay 1000 // pause for a second for the user
poster ""
```

```
msg = "Unit #"
msg += serialnum

if waveform = boundary1 // Do the test
     Message  "The signal passed the boundary test."
     msg += " Passed the test"
     WriteToFile "test.out" msg APPEND
else
     MessageBeep 3
     Message  "The signal failed the boundary test."
      msg += " Failed the test"
     WriteToFile "test.out" msg APPEND
endif

again = Confirm "Do another test?" "Example Test"
if again = 1 goto StartTest

Message "End of ATL"

end
```

Appendix A
Product Support

Contacting PC Instruments

You may contact PC Instruments either by calling our customer support number below, or via the internet at our homepage described below.

Customer Support:	(330) 762-8500
Fax:	(330) 762-8855

When calling or emailing for technical support, please supply the model number and serial number of your instrument, as well as any other pertinent informaiton (i.e. Operating system, type of computer, other instruments installed in the PC).

Returning Your Instrument for Service

Should your instrument card require service, the applications engineer will take your instrument's serial number (see the title page of the Operation and Programming Manual or the sticker on the instrument itself) and issue a Return Material Authorization (RMA) number.

Return your instrument along with any other information that may assist PC Instruments' technicians in repairing your instrument. Put the card in an antistatic bag and repack it in the original shipping package. Ship it to the address below:

PC Instruments Incorporated
526 South Main Street #311
Akron, Ohio 44311

After your instrument has been repaired, it will be returned to you using the same shipment method as was used to send the defective card to PC Instruments.

You will be billed for repairs not covered by PC Instruments warranty.

Internet Access to PC Instruments

You can contact PC Instruments and get technical support and software updates via the Internet by email or by accessing our World Wide Web page. Using any web browser, our URL is:

```
http://www.pcinstruments.com
```

for technical assistance, email us at:

```
help@pcinstruments.com
```

for any other questions or comments, please send email to:

```
info@pcinstruments.com
```

Appendix B
SCPI-GUI Commands

Introduction

This appendix describes the SCPI commands which may be issued through BenchTop to control the user interface. These commands can be placed either in workspace (.wsp) files, or used in ATL programs via the SCPI function (see Chapter 5). A workspace files containing GUI commands or other SCPI commands (or both) may be executed by issuing the SYSTEM:SCRIPT command (see the BenchCom User's Manual, Appendix A).

Although these commands are formatted to be similar to the Standard Commands for Programmable Instruments (SCPI) standard, they are unique to BenchTop. For more information about SCPI commands in general, see the BenchCom User's Manul.

Overview

Controlling the BenchTop user interface is relatively simple and requires only a few SCPI commands. The SCPI tree for the Graphical User Interface (GUI) commands is given below.

```
GUI
      :REPLay <filename>
      :MOVE <<NORM|MIN|MAX>|<X,Y,W,H>>
      :SET <SETTING>,<VALUE>
      :WINDow
          :CREATe <WTYPE>,[SCPI ID]
          :DESTroy <WINDOW>
              :ALL
          :MOVE
<WINDOW>,<<NORM|MIN|MAX|UPDATE>|<X,Y,W,H>>
          :SET <WINDOW>,<SETTING>,<VALUE...>
```

For examples of all of the SCPI-GUI commands, see any workspace (.wsp) file created by BenchTop. These files are ASCII text formatted and contain a complete list of SCPI-GUI commands necessary to setup the BenchTop workspace to any configuration.

Description of Commands

GUI:REPLAY <filename>

Description: This command will envoke the ATL interpreter and execute the ATL file specified.

Parameters: The parameter specifies the ATL file to be run. If the filename contains path information, it should be enclosed in quotes.

Example: GUI:REPLAY "c:\pci\atl\test01.atl"

GUI:MOVE <<NORM|MIN|MAX>|<X,Y,W,H>>

Description: This command will move and resize the BenchTop program window.

Parameters: The parameter list can be either a set of screen coordinates (x,y,w,h) or one of the following:

MAX	to maximize the BenchTop program
MIN	to minimize the BenchTop program
NORM	to return BenchTop from a maximized or minimized state back to it's original size and position.

If screen coordinates are provided, all values are in pixels relative to the upper lefthand corner of the screen.

Example: GUI:MOVE 40,20,560,440

GUI:SET <SETTING>,<VALUE>

Description: This command will set a BenchTop parameter.

Parameters: At this time, the only BenchTop setting which can be controlled by this command is the background tasking option (Display|Disable Updates When Not Active from the menu bar). When this setting is on, BenchTop will automatically stop updating the oscilloscope display(s) if the program is backgrounded. When off, the program will continue to update all scope displays even when BenchTop is not the "top" application.

Example: GUI:SET tasking,on

GUI:WINDOW:CREATE <WINDOWTYPE>,[SCPI ID]

Desciption: This command will create an instrument display of the specified type.

Parameters: The first parameter specifies the type of window to be created. It may be one of the following:

SCOPE, oscilloscope display
ARB, arbitrary waveform generator display
FGEN, function generator display
DMM, digital multimeter display
BNDTEST, boundary test dialog
PAV, phase angle voltmeter display
PANALYSIS, pulse analysis display

The second parameter is optional and specifies the SCPI id of the instrument to be assigned to the display. If not given, BenchTop will automatically assign the new window the SCPI id of the first instrument of the appropriate type in the hardware configuration file.

Example: gui:wind:creat scope

GUI:WINDOW:DESTROY <WINDOW>

Description: This command will destroy a single instrument display. This will not affect the operation of any instruments. Any dialog boxes associated with the instrument display (e.g. timebase control) will be destroyed also.

Parameters: The parameter specifies the window to be destroyed. Every instrument display (see GUI:WINDOW:CREATE) is assigned a *window number* by BenchTop when it is created. The window numbers start at 0 for the first window created. In order to remove a particular display from the screen, you must know its number.

Example: GUI:WINDOW:DESTROY 0

GUI:WINDOW:DESTROY:ALL

Description: This command will destroy all of the instrument displays and dialog boxes currently showing. This will not affect the settings or operation of any instruments.

Parameters: None.

Example: GUI:WIND:DESTROY:ALL

GUI:WINDOW:MOVE <WINDOW>,<<MIN|MAX|NORM|UPDATE>|<X,Y,W,H>>

Description: This command will move and resize an instrument display within the BenchTop workspace. The instrument display must have already been created either by the user or by issuing the GUI:WINDOW:CREATE command.

Parameters: The first parameter specifies the window to be moved. Every instrument display (see GUI:WINDOW:CREATE) is assigned a *window number* by BenchTop when it is created. The window numbers start at 0 for the first window created. In order to move a particular display, you must know its number.

The remaining parameters may be either a set of screen coordinates, or one of the keywords:

Parameter Value:	Means:
MIN	to minimize the instrument display
MAX	to maximize the instrument display
NORM	to convert the instrument display from minimized or maximized back to it's "normal" state.
UPDATE	to force the window to redraw (useful from within an ATL script when you want a display to update).

If screen coordinates are given, they are:

X the horizontal location (in pixels) of the upper lefthand corner of the instrument display. This is relative to BenchTop, where 0 is the leftmost position.

Y the vertical location (in pixels) of the upper lefthand corner of the instrument display. 0 is the topmost location.

W the width of the instrument display, in pixels.

H the height of the instrument display, in pixels.

Example: `GUI:WINDOW:MOVE 0,10,10,320,200`

GUI:WINDOW:SET <WINDOW>,<SETTING>,<VALUE>

Description: This command is used to set options for a particular instrument display. These options include screen colors, activating dialog boxes, etc.

Parameters: The first parameter specifies the window to be affected. Every instrument display (see GUI:WINDOW:CREATE) is assigned a *window number* by BenchTop when it is created. The window numbers start at 0 for the first window created.

The second parameter indicates the item for the instrument display that is to be changed. For a complete list of instrument display settings for each instrument display type, see the tables below.

Example: `GUI:WINDOW:SET 0,CONNECT,ON`

Oscilloscope Display Settings		
Setting:	**Additional Parameters:**	**Description:**
TICKS	ON | OFF	Turns tick marks on or off.
READOUTS	ON | OFF	Turns "display readouts" on or off.
CURSORSNAP	ON | OFF	Turns "Snap cursor to Waveform" on or off.
CONNECT	ON | OFF	Turns "Connect data points" on or off.
PERSIST	ON | OFF	Turns "Persistance" on or off.
CH1COLOR CH2COLOR BACKCOLOR	"RED GREEN BLUE"	Sets the indicated color to the new value. (8 bit values)

Oscilloscope Display Settings		
GRIDCOLOR		
CH1DLG CH2DLG TRGDLG INPDLG ACQDLG TIMEDLG SETUPDLG	ON | OFF, "X Y"	Turns the indicated dialog box on or off, and sets the position if on (X and Y pixels from the upper lefthand corner of the BenchTop workspace).
REFRESH	RATE	Sets the refresh rate for the display to RATE milliseconds.
GNDIND	ON | OFF	Turns the "Ground level indicator" on or off.
TRGIND	ON | OFF	Turns the "Trigger level indicator" on or off.
SWTRIG	ON | OFF	Turns "Enable Software Autotrigger" for the display on or off.
DISPLAY	CH1 | CH2 | BOTH	Sets the channel to be displayed in the instrument display.

Arbitrary Waveform Generator Display Settings		
Setting:	**Additional Parameters:**	**Description:**
TICKS	ON | OFF	Turns tick marks on or off.
READOUTS	ON | OFF	Turns "display readouts" on or off.
CURSORSNAP	ON | OFF	Turns "Snap cursor to Waveform" on or off.
CONNECT	ON | OFF	Turns "Connect data points" on or off.
EDIT	ON | OFF	Turns "Allow Editing" on or off.
CH1COLOR CH2COLOR BACKCOLOR GRIDCOLOR	"RED GREEN BLUE"	Sets the indicated color to the new value. (8 bit values)
CH1DLG CH2DLG	ON | OFF, "X Y"	Turns the indicated dialog box on or off, and sets the position if on. The X

Arbitrary Waveform Generator Display Settings		
MEMDLG TRIGDLG SEQDLG VIEWDLG CLKDLG SYNCDLG BRKDLG SETUPDLG WGENDLG PGENDLG FGENDLG FILTDLG		and Y coordinates are in pixels, relative to the upper lefthand corner of the BenchTop workspace.
DISPLAY	CH1 | CH2 | BOTH	Sets the channel to be displayed in the instrument display.

Pulse Analysis Display Settings		
Setting:	**Parameters:**	**Description:**
DISPHIGH	ON | OFF	Turns High Level display on or off.
DISPLOW	ON | OFF	Turns Low Level display on or off.
DISPMID	ON | OFF	Turns Middle Level display on or off.
DISPRTIME	ON | OFF	Turns Risetime display on or off.
DISPFTIME	ON | OFF	Turns Falltime display on or off.
DISPPWIDTH	ON | OFF	Turns Positive Pulse Width display on or off.
DISPNWIDTH	ON | OFF	Turns Negative Pulse Width display on or off.
DISPTMRISE	ON | OFF	Turns Time-to-Mid Rise display on or off.
DISPTMFALL	ON | OFF	Turns Time-to-Mid Fall display on or off.
DISPPDUTY	ON | OFF	Turns Positive Dutycycle display on or off.
DISPNDUTY	ON | OFF	Turns Negative Dutycycle display on or off.
DISPROVER	ON | OFF	Turns Rise Overshoot display on or off.
DISPFOVER	ON | OFF	Turns Fall Overshoot display on or off.
DISPRPRE	ON | OFF	Turns Rise Preshoot display on or off.
DISPFPRE	ON | OFF	Turns Fall Preshoot display on or off.
DISPTMAX	ON | OFF	Turns Time-to-Max display on or off.
DISPTMIN	ON | OFF	Turns Time-to-Min display on or off.

Pulse Analysis Display Settings		
REFRESH	RATE	Sets the refresh rate to RATE milliseconds.

Signal Meter Display Settings		
Setting:	**Additional Parameters:**	**Description:**
CHART	ON | OFF	Turns strip-chart display on or off.
DISPMAX	ON | OFF	Turns Max Level display on or off.
DISPMIN	ON | OFF	Turns Min Level display on or off.
DISPMEAN	ON | OFF	Turns Mean display on or off.
DISPFREQ	ON | OFF	Turns Frequency display on or off.
DISPSTDDEV	ON | OFF	Turns Standard Deviation display on or off.
DISPRMS	ON | OFF	Turns Root-Mean-Square display on or off.
DISPPP	ON | OFF	Turns Peak-to-Peak display on or off.
REFRESH	RATE	Sets the refresh rate to RATE milliseconds.

Appendix C
Signal Analysis Commands

Introduction

The SCPI commands described in this appendix are for use with BenchTop© Plus or the BenchTop Plus Signal Analysis Library which is bundled with BenchTop Plus. The functionality of these commands is built into BenchTop Plus via the point-and-click graphical user interface (i.e. Pulse Analysis display, FFT Display). This appendix describes the commands as they would be used from environments such as:

- From within BenchTop Plus using the Automated Test Language (ATL) interpreter.
- From a user written Windows C or C++ program.
- From Visual Basic for Windows
- From LabVIEW for Windows.

In each case, the signal analysis commands are issued to the PC Instruments DLL (Dynamic Link Library) PCIWIN.DLL in the same manner as instrument commands (see the BenchCom User's Manual, Volume 2 and 3).

Pulse parameter definitions are derived from IEEE Standard Pulse Terms and Definitions (IEEE Std 194-1977) and pulse parameter measurement techniques are derived from IEEE Standard on Pulse Measurement and Analysis by Objective Techniques (ANSI/IEEE Std 181-1977). Fast Fourier Transform (FFT) calculations are based on a radix-2 algorithm and all data records will be zero-padded to the closest power-of-two length equal to or greater than 1024 (i.e. 1001 data point record has 23 zeros added to the end of the record).

Description of Pulse Measurements

The following pulse parameter measurements are supported by the Signal Analysis commands. Additional commands support basic frequency domain measurements.

LOW :	The lowest portion of a pulse waveform which represents a nominal value for the pulse.
HIGH :	The highest portion of a pulse waveform which represents a nominal value for the pulse.
MIDDLE :	Halfway between the LOW and HIGH levels.
Rise Time :	Time taken to make the first positive transition from the 10% level to the 90% level.
Rise Overshoot :	The difference between the HIGH level and the peak level to which the pulse initially rises, expressed as a percentage of the waveform amplitude.
Rise Preshoot:	The difference between the LOW level and the minimum level of the pulse before it begins transition from 10% to 90%, expressed as a percentage of the waveform amplitude.
Time Mid-Rise:	The time when the first positive transition from 10% to 90% crosses the MIDDLE level.
Fall Time :	Time taken to make the first negative transition from the 90% level to the 10% level.
Fall Overshoot:	The difference between the LOW level minimum level to which the pulse initially falls, expressed as a percentage of the waveform amplitude.
Fall Preshoot:	The difference between the HIGH level and the peak level to which the pulse initially rose before beginning its transition from 90% to 10%, expressed as a percentage of the waveform amplitude.
Time Mid-Fall :	The time when the first negative transition from 90% to 10% crosses the MIDDLE level.
PWidth :	The time taken for a pulse to make a negative transition after a positive transition, measured as the pulse crosses the MIDDLE level. If the pulse is periodic, the sum of PWidth and NWidth is the Period of the pulse train.
NWidth :	The time taken for a pulse to make a positive transition after a negative transition, measured as the pulse crosses the MIDDLE level. If the pulse is periodic, the sum of PWidth and NWidth is the Period of the pulse train.
PDuty :	The ratio of PWidth to Period expressed as a percentage of Period.

NDuty : The ratio of NWidth to Period expressed as a percentage of Period.

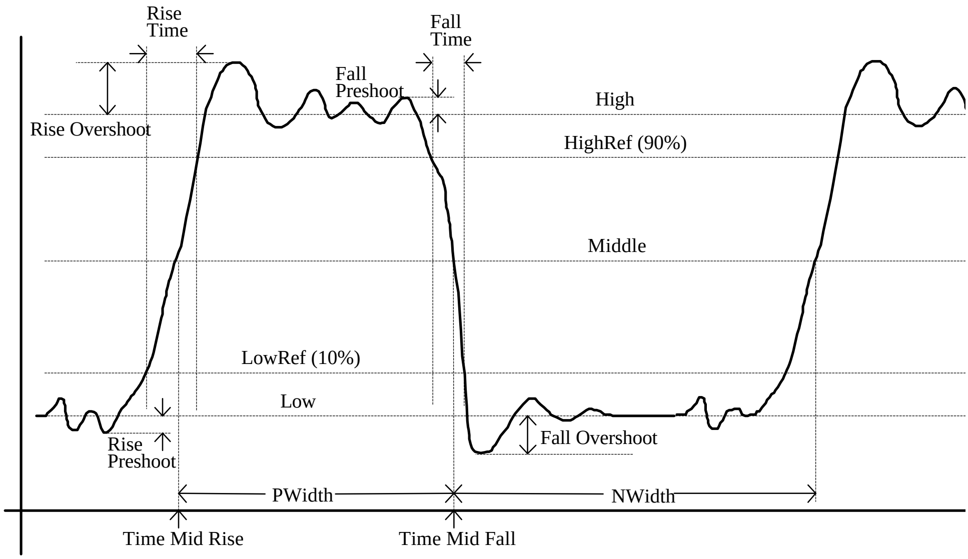


Figure C-1: Basic Pulse Time Features

Description of Signal Analysis Techniques

Location of Low and High Levels for Pulse Parameter Measurement

LOW and HIGH levels are needed to compute the following pulse parameters:
MIDDLE, amplitude, rise time, rise preshoot, rise overshoot, time mid-rise, fall time, fall preshoot, fall overshoot, time mid-fall, PWidth, NWidth, PDuty and NDuty.

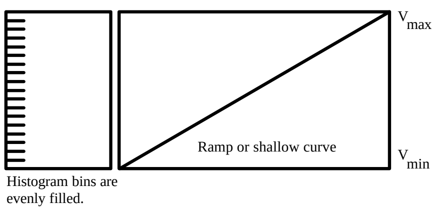
The procedure for the automatic location of LOW and HIGH requires the data be placed into a histogram to produce a probability density function (PDF) for the waveform. If the waveform is a pulse shape, its PDF will indicate the LOW and HIGH levels as two modes indicating where the pulse spends most of its time.

PC Instruments oscilloscopes return integer data justified to the MSB of a 16-bit signed integer. The ADC resolution varies between 8 to 12 so for histogram binning the data will be taken from the upper 10 bits of the 16-bit integer.

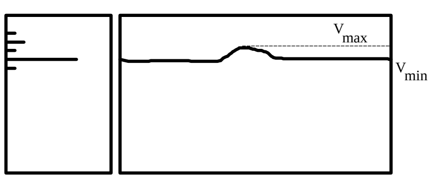
The ideal pulse data set will produce a distinctly bimodal (two modes or clusters of filled bins) histogram. One mode will represent the Low level, the other the High level of the pulse. At least three other types of PDF could occur. One is that no distinct modes are present, a second is that only one distinct mode is present and the third case is a histogram with three or more distinct modes.

If no distinct mode can be located the Maximum and Minimum levels of the data set may be substituted for the High and Low levels. There are two ways of making this substitution. The rise time and related measurement SCPI functions can be told to use alternate High and Low levels via a parameter list (see the Command Descriptions), or the histogram can be turned off so that rise time and related measurements are based on simple Maximum and Minimum of the data.

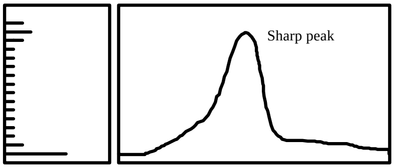
Ramp or shallow curve data sets can produce histograms with no distinct modes. Figure 3 illustrates the histogram produced by a ramp in the data set.



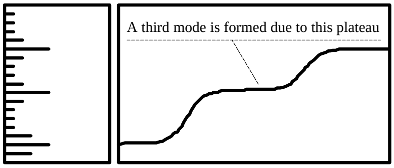
Insufficient vertical range of data can produce a histogram with only one distinct mode. It is recommended that the waveform occupy at least 1/3 of the vertical range of the instrument. Figure 4 illustrates a lack of vertical resolution in the data set and the resulting histogram.



A data set that is not invalid but could still cause problems for the automatic pulse parameter measurement software is an impulse with very few data points at its peak (valley). In this case, High (or Low) could be assigned V_{max} (V_{min}) as necessary. Figure 5 shows how a pulse could only produce a single distinct mode in the histogram.



Waveforms generated during a TDR measurement are prime candidates for a third mode in the histogram. Figure 6 illustrates a pulse that may be distorted by a reflection, causing an extra mode in the histogram.



SCPI Command Tree

Given below is the SCPI command tree for the signal analysis commands.

```
MEASure
  :DOWNload[:REFerence] <instrument> , <channel>
  [:TRANsform]
     :HISTogram?
       :AVERage <ON | OFF>
       :METHod <ON | OFF>
     :FFT?
       :VALue? <bin>
       :SIZE?
       :FORMat <MAGNitude|PHASe|COMPlex|DBM>
       :WINDow <RECT|BARTlett|HAMMing|HANN|BLACkman>
       :FBIN
             :MAXimums?
             :VALue? <bin>
  :LOW?
  :HIGH?
  :MIDDle?
  :RISE:TIMe? [low],[high]
     :OVERshoot? [low],[high]
     :PREShoot? [low],[high]
  :FALL:TIMe? [low],[high]
     :OVERshoot? [low],[high]
     :PREShoot? [low],[high]
  :NWIDth? [low],[high]
```

```
:PWIDth? [low],[high]
:PDUTycycle? [low],[high]
:NDUTycycle? [low],[high]
:TMIDdle:RISE? [low],[high]
:TMIDdle:FALL? [low],[high]
:TMAXimum?
:TMINimum?
```

Command Descriptions

MEASure:DOWNload[:REFerence] <instrument,channel>

Description: Transfers data from the specified instrument and channel to the Signal Analysis subsystem. This command must be sent each time a new waveform is to be analyzed.
Forms: Command and query.
Parameters: The instrument number (SCPI ID) followed by the channel number.
*RST Value: N/A
Example: `measure:download 1,1`

MEASure[:TRANsform]:HISTogram?

Description: Returns the histogram data. The number of elements returned equals the `MEASure:HISTogram:SIZE?` value.
Forms: Query only
Parameters: None.
*RST Value: N/A

MEASure[:TRANSform]:HISTogram:SIZE?

Description: Returns the number of bins used to create histogram data.
Forms: Query only.
Parameters: None.
*RST Value: N/A

MEASure[:TRANsform]:HISTogram:AVERage <ON | OFF>

Description: Toggles histogram averaging on or off. If histogram averaging is on, the histogram bin counts are divided by two before binning a new sweep of data. If histogram averaging is off, the histogram bin counts are re-set to zero before binning a new sweep of data. A query of the method will return the current setting without disturbing the setting.

Forms: Command and query.
Parameters: ON, OFF.
*RST Value: OFF.

MEASure[:TRANsform]:HISTogram:METHod <ON|OFF>

Description: Toggles the level determination method for the LOW and HIGH pulse parameters between the histogram method (ON) or using the Minimum and Maximum levels (OFF). A query of the method will return the current setting without changing it.
Forms: Command and query.
Parameters: ON, OFF.
*RST Value: ON.

MEASure[:TRANsform]:HISTogram:RESet

Description: Zero the histogram bin counts.
Forms: Command only.
Parameters: None.
*RST Value: N/A

MEASure[:TRANsform]:FFT?

Description: Returns an array of floating point numbers as either complex number pairs, magnitude only, or phase only depending on the current `MEASure[:TRANsform]:FFT:FORMat` setting. For the complex number pairs, the array is organized from the lowest frequency bin to the highest frequency bin with the real portion of the complex number given first, followed by the imaginary portion. For the magnitude, phase and dbm formats, the array is organized from the lowest frequency bin to the highest frequency bin and a single number is returned for each frequency bin. The number of data pairs or points in the array is given by the `MEASure[:TRANsform]:FFT:SIZE?` query and will be the next larger power of two from the number of data points in the time domain data most recently supplied by the `MEASure:DOWNload[:REFerence]` command.
Forms: Query only.
Parameters: None.
*RST Vaue: N/A

MEASure[:TRANsform]:FFT:VALue? <bin>

Description: Returns a single element from the FFT array. The format of the response is determined by the current value of the `MEASure:FFT:FORMat` setting.
Forms: Query only
Parameters: The index of a frequency bin, from 0 to `MEASure:FFT:SIZE?` - 1.

MEASure[:TRANsform]:FFT:SIZE?

Description: Returns an integer describing the number of data points the FFT will return. If the FFT is returning complex numbers, the SIZE represents the number of pairs of data points. This number will be the next larger power of two that will hold the most recently downloaded sweep, i.e. if a sweep contains 1001 data points and the FFT format is set to COMPlex, 1024 complex number pairs will be returned.
Forms: Query only.
Parameters: None.
*RST Value: N/A

MEASure[:TRANsform]:FFT:FORMat <MAGNitude | PHAse | COMPlex | DBM>

Description: Sets the format for the return of FFT data. COMPlex will return a series of real and imaginary data pairs, MAGNitude will return the magnitudes of the complex data, PHAse will return the phase angle of the complex data in radians and DBM will return the power delivered to a reference resistor of 50Ω referenced to 1mW (dBM=10$\log_{10}$ (data / 50 / 0.001W)). A query will return the current setting without affecting it.
Forms: Command and query.
Parameters: MAGNitude, PHAse, COMPlex, DBM.
*RST Value: MAGNitude.

MEASure[:TRANsform]:FFT:WINDow <RECTangular|BARTlett|HAMMing|HANN|BLACkman>

Description: Sets the window applied to time-domain data before performing the FFT. A query will return the current setting without affecting it. Consult a text on signal processing or spectral estimation for more information about the effect of windowed data on the results of an FFT. Digital Signal Processing, 2nd ed., Alan V. Oppenheim and Ronald W. Schafer or Modern Spectral Estimation, Theory and Application by Steven M. Kay are two good reference works.

Forms: Command and Query.

Parameters: RECTangular, BARTlett, HAMMing, HANN, or BLACkman
The window formulas are (assuming sampled time-domain data indexed from 0 to N-1):

Rectangular

$$w[n] = \begin{cases} 1 & 0 \le n \le (N-1) \\ 0 & otherwise \end{cases}$$

Bartlett

$$w[n] = \begin{cases} 1 - \frac{|n|}{N-1} & 0 \le n \le (N-1) \\ 0 & otherwise \end{cases}$$

Hann

$$w[n] = \begin{cases} \frac{1}{2} - \frac{1}{2}\cos\left(\frac{2pn}{(N-1)}\right) & 0 \le n \le (N-1) \\ 0 & otherwise \end{cases}$$

Hamming

$$w[n] = \begin{cases} 0.54 - 0.46\cos\left(\frac{2pn}{(N-1)}\right) & 0 \le n \le (N-1) \\ 0 & otherwise \end{cases}$$

Blackman

$$w[n] = \begin{cases} 0.42 - 0.5\cos\left(\frac{2pn}{(N-1)}\right) + 0.08\cos\left(\frac{4pn}{(N-1)}\right) & 0 \le n \le (N-1) \\ 0 & otherwise \end{cases}$$

*RST Value: RECTangular.

MEASure[:TRANsform]:FFT:FBIN:MAXimums?

Description: Returns the frequency bin indexes corresponding to the 10 largest magnitude frequency bins.
Forms: Query only
Parameters: None.
*RST value: N/A

MEASure[:TRANsform]:FFT:FBIN:VALue? <bin>

Description: Returns the center frequency and frequency width (both in Hertz) for the specified frequency bin (starting at 0).
Forms: Query only.
Parameters: The index of a frequency bin, from 0 to `MEASure:FFT:SIZE?` - 1.

MEASure:LOW?

Description: Returns a estimate of the LOW signal level of the pulse data most recently supplied by the MEASure:DOWNload[:REFerence] command. The method for determining the LOW level is selected by the MEASure:[:TRANsform]:HISTogram:METHod setting. If a LOW level cannot be located, the number -1.0e10 is returned to signify the error. If the MEASure[:TRANsform]:HISTogram:METHod is set to ON and a LOW level cannot be located, consider switching the METHod to OFF, thus using the minimum signal transition as the LOW level. Refer to Figure 1 to see the LOW signal level parameter in reference to a typical pulse shape.
Forms: Query only.
Parameters: None.
*RST Value: N/A.

MEASure:HIGH?

Description: Returns an estimate of the HIGH signal level of the pulse data most recently supplied by the MEASure:DOWNload[:REFerence] command. The method for determining the HIGH level is selected by the MEASure:[:TRANsform]:HISTogram:METHod setting. If a HIGH level cannot be located the number -1.0e10 is returned to signify the error. If the MEAsure[:TRANsorm]:HISTogram:METHod is set to ON and a HIGH level cannot be located, consider switching the METHod to OFF, thus using the maximum signal transition as the HIGH level. Refer to Figure 1 to see the HIGH signal level parameter in reference to a typical pulse shape.

Forms: Query only.

Parameters: None.

*RST Value: N/A.

MEASure:MIDDle?

Description: Returns an estimate of the MIDDLE signal level for the pulse data most recently supplied by the MEASure:DOWNload[:REFerence] command. If the MIDDLE level cannot be estimated the number -1.0e10 is returned to signify an error. If the MEAsure[:TRANsform]:HISTogram:METHod is set to ON and a MIDDLE level cannot be estimated, consider switching the METHod to OFF, thus using the maximum and minimum signal transitions to estimate the MIDDLE level. Refer to Figure 1 to see the MIDDLE signal level parameter in reference to a typical pulse shape.

Forms: Query only.

Parameters: None.

*RST Value: N/A.

MEASure:RISE:TIME? [low],[high]

Description: Returns an estimate of the rise time of the first positive transition from the LOWREF level to the HIGHREF level of the pulse data most recently supplied by the MEASure:DOWNload[:REFerence] command. The rise time will not be computed if there are fewer than four data points found on the first rising edge between the LOWREF and HIGHREF levels. If the rise time cannot be estimated, the number -1.0e10 is returned to signify an error. If the MEASure[:TRANsform]:HISTogram:METHod is set to ON and a rise time cannot be estimated even with sufficient samples on the pulse edge, consider switching the METHod to OFF, thus using the minimum and maximum signal transitions for LOW and HIGH. The user can specify voltage levels to use for LOW and HIGH that will override the current values without changing the LOW and HIGH computed from the data.. Refer to Figure 1 to see the rise time parameter in reference to a typical pulse shape.

Forms: Query only.

Parameters: [low] is a number to specify an overriding level for LOW. [high] is a number to specify an overriding level for HIGH.

*RST Value: N/A.

MEASure:RISE:OVERshoot? [low],[high]

Description: Returns an estimate of the difference between the HIGH and the peak amplitude to which the pulse data most recently supplied by the MEASure:DOWNload[:REFerence] command initally rises. The answer is expressed as a percentage of the waveform amplitude.

Forms: Query only.

Parameters: [low] is a number to specify an overriding level for LOW. [high] is a number to specify an overriding level for HIGH.

*RST Value: N/A.

MEASure:RISE:PREShoot? [low],[high]

Description: Returns an estimate of the difference between the LOW level and the negative peak value of the pulse data most recently supplied by the MEASure:DOWNload[:REFerence] command just prior to the first rising edge. The answer is expressed as a percentage of the waveform amplitude.

Forms: Query only.

Parameters: [low] is a number to specify an overriding level for LOW. [high] is a number to specify an overriding level for HIGH.
*RST Value: N/A.

MEASure:FALL:TIME? [low],[high]

Description: Returns an estimate of the fall time of the first negative transition from the HIGHREF level to the LOWREF level of the pulse data most recently supplied by the MEASure:DOWNload[:REFerence] command. The fall time will not be computed if there are fewer than four data points found on the first falling edge between the HIGHREF and LOWREF levels. If fall time cannot be estimated, the number -1.0e10 is returned to signify an error. If the MEASure[:TRANsform]:HISTogram:METHod is set to ON and a fall time cannot be estimated even with sufficient samples on the pulse edge, consider switching the METHod to OFF, thus using the minimum and maximum signal transitions for LOW and HIGH. The user can specify voltage levels to use for LOW and HIGH that will override the current values without changing the LOW and HIGH computed from the data.. Refer to Figure 1 to see the fall time parameter in reference to a typical pulse shape.
Forms: Query only.
Parameters: [low] is a number to specify an overriding level for LOW. [high] is a number to specify an overriding level for HIGH.
*RST Value: N/A.

MEASure:FALL:OVERshoot? [low],[high]

Description: Returns an estimate of the difference between the LOW and the minimum amplitude to which the pulse data most recently supplied by the MEASure:DOWNload[:REFerence] command initally falls. The answer is expressed as a percentage of the waveform amplitude.
Forms: Query only.
Parameters: [low] is a number to specify an overriding level for LOW. [high] is a number to specify an overriding level for HIGH.
*RST Value: N/A.

MEASure:FALL:PREShoot? [low],[high]

Description: Returns an estimate of the difference between the HIGH level and the peak value of the pulse data most recently supplied by the MEASure:DOWNload[:REFerence] command just prior to the first falling edge. The answer is expressed as a percentage of the waveform amplitude.

Forms: Query only.

Parameters: [low] is a number to specify an overriding level for LOW. [high] is a number to specify an overriding level for HIGH.

*RST Value: N/A.

MEASure:NWIDth? [low],[high]

Description: Returns an estimate of the time taken to make a positive transition after a negative transition, measured as the pulse crosses the MIDDLE level of the pulse data most recently supplied by the MEASure:DOWNload[:REFerence] command.

Forms: Query only.

Parameters: [low] is a number to specify an overriding level for LOW. [high] is a number to specify an overriding level for HIGH.

*RST Value: N/A.

MEASure:PWIDth? [low],[high]

Description: Returns an estimate of the time taken to make a negative transition after a positive transition, measured as the pulse crosses the MIDDLE level of the pulse data most recently supplied by the MEAsure:DOWNload[:REFerence] command.

Forms: Query only.

Parameters: [low] is a number to specify an overriding level for LOW. [high] is a number to specify an overriding level for HIGH.

*RST Value: N/A.

MEASure:PDUTycycle? [low],[high]

Description: Returns an estimate of the positive duty cycle of the pulse data most recently supplied by the MEASure:DOWNload[:REFerence] command. The positive duty cycle is computed as PWidth/(PWidth+NWidth) and returned as a percentage.

Forms: Query only.

Parameters: [low] is a number to specify an overriding level for LOW. [high] is a number to specify an overriding level for HIGH.

*RST Value: N/A.

MEASure:NDUTycycle? [low],[high]

Description: Returns an estimate of the negative duty cycle of the pulse data most recently supplied by the MEASure:DOWNload[:REFerence] command. The negative duty cycle is computed as NWidth/(PWidth+NWidth) and returned as a percentage.
Forms: Query only.
Parameters: [low] is a number to specify an overriding level for LOW. [high] is a number to specify an overriding level for HIGH.
*RST Value: N/A.

MEASure:TMIDdle:RISE? [low],[high]

Description: Returns an estimate of the time when the first rising edge of the pulse data most recently acquired from the MEASure:DOWNload[:REFerence] command crosses the MIDDLE level.
Forms: Query only.
Parameters: [low] is a number to specify an overriding level for LOW. [high] is a number to specify an overriding level for HIGH.
*RST Value: N/A.

MEASure:TMIDdle:FALL? [low],[high]

Description: Returns an estimate of the time when the first falling edge of the pulse data most recently supplied from the MEASure:DOWNload[:REFerence] command crosses the MIDDLE level.
Forms: Query only.
Parameters: [low] is a number to specify an overriding level for LOW. [high] is a number to specify an overriding level for HIGH.
*RST Value: N/A.

MEASure:TMAXimum?

Description: Returns a floating point indicating the time when the maximum value occurred in the pulse data most recently supplied by the `MEAsure:DOWNload[:REFerence]` command.
Forms: Query only.
Parameters: None.
*RST Value: N/A.

MEASure:TMINimum?

Description: Returns a floating point indicating the time when the minimum value occurred in the pulse data most recently supplied by the `MEASure:DOWNload[:REFerence]` command.

Forms: Query only.

Parameters: None.

*RST Value: N/A.

Appendix D
PCI Data Exchange

Introduction

This appendix describes the PCI Data Exchange application that comes with BenchTop for Windows. An icon for the Data Exchange program will be created in the PC Instruments group during installation. The PCI Data Exchange program will be referred to as Data Exchange or simply PCIDX throughout the rest of this appendix.

Overview

The purpose of the Data Exchange program is to provide a convenient method of viewing, editing, printing, and translating oscilloscope data files created with BenchTop. The program can also be used to create Boundary files that can be used by BenchTop in either ATL programs or by the Boundary Test dialog.

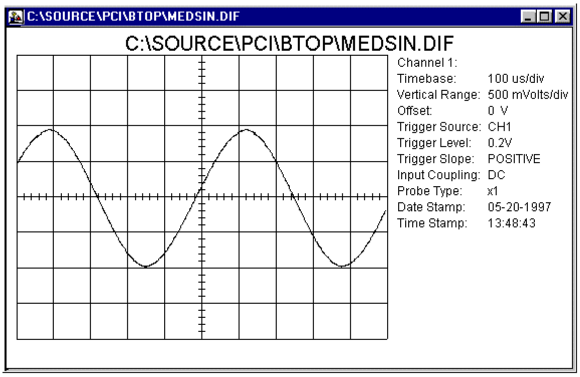
Files used within the Data Exchange program can be viewed within BenchTop by opening them with the Reference Waveforms dialog box and assigning the files to a scope display. The Data Exchange program is an "off-line" alternative to using BenchTop when there is no scope in the PC or you don't want to make measurements.

Program Operation

Opening a file

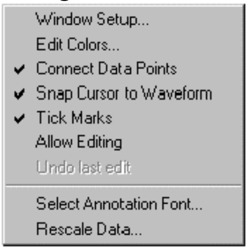
Assuming you have created a waveform file (either with BenchTop or by some other means) you can open and view the data simply by choosing Open from the File menu, or by pressing the "File Open" button on the toolbar.

Once the file has been opened, you will see a window with the waveform data and any scaling annotations available, as shown below.



The waveform display can be moved and resized as desired, and you can open multiple waveform displays at a time.

Using the Waveform Display



Similarly to BenchTop, each waveform display in the Data Exchange program as a control menu associated with it. To activate the control menu for a waveform display, move the mouse cursor over the display and press the right mouse button. A menu will appear, like the one shown here.

The Window Setup dialog box will let you pick how the display is annotated as well as defining a title for the waveform display. By choosing Edit Colors you can define what colors are used on the screen. Data points will be connected with straight line segments when Connect Data Points is checked.

When you move the mouse over the waveform display, a cursor will appear and a readout below the grid will display the time and voltage at the cursor. When Snap Cursor to Waveform is checked, the cursor will always "stick" to the waveform. When this option is off (unchecked), the cursor will be free to move whereever the mouse cursor goes. When multiple waveforms are displayed (as in creating a boundary file), you can select which waveform has the cursor by pressing the Tab key.

Waveform editing is enabled by selecting Allow Editing from the waveform display control menu. Once this option is on, you can edit the waveform data simply by "drawing" on the waveform display.

Recently Used Files

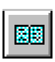
BenchTop automatically remembers the six most recently opened files. To reload one of the recent files, click on the file-cabinet icon on the toolbar and select the file you wish to open.

Merging Views

You can merge multiple waveform files into a single waveform display. To do this, start by opening a waveform file. Then, to add another waveform file to the display, select the "Add a Reference to the Current Display" button from the toolbar, or choose Open and Merge from the File menu. Note that all of the annotations and scaling will be done according to the information in the first data file that was opened. Additional data files merged with the view will be "traced" over the waveform display without rescaling.

Printing

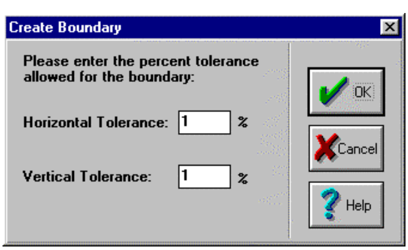
Waveform displays may be printed by selecting the Print button from the toolbar or by choosing Print from the File menu. The waveform display will be printed as it is shown (all annotations selected, and any title that has been set). If you have multiple waveform displays open at once, the currently selected waveform display will be printed.



You may also generate a print preview by pressing the button shown to the left or by choosing Print Preview from the File menu.

Creating Boundaries

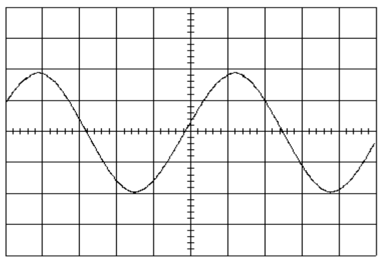
The Data Exchange program can automatically generate a boundary from a single waveform file. To do this, first open the source waveform file, then select the Create Boundary button from the toolbar or choose Create Boundary from the File menu. The dialog box shown below will then appear.



Creating the boundary involves *splitting* the single waveform into two separate data sets that represent the upper and lower waveform "limits". The amount of tolerance for which the original signal could vary and still lie within the boundary is expressed as a **percentage of the waveform display** in the horizontal and vertical directions. A vertical tolerance would indicate that the offset of the signal could change, whereas a horizontal tolerance would account for timebase jitter or variation in time between the signal and the trigger source.

Example

As an example, say you are testing the output of a sinewave generator that is supposed to output 1Volt Peak-to-Peak at a frequency of 10MHz. To generate the test data, you capture the output of a known good generator with the scope set to 200mV/div and a timebase of 20ns/div. This will yield a waveform display similar to shown below.



During the test, you want to verify that the output of the generator does not vary in amplitude by more than 5% and the frequency does not vary by more than 2%. To determine the tolerance values to use in the boundary creation dialog box, you do the following calculations:

The period of the 10Mhz signal is 100ns. 2% of 100ns is 2ns.
5% of 1VPP is 50mV.

At 20ns/div, or 200ns/sweep, 2ns is 1% of the horizontal display. Therefore, we use 1% horizontal tolerance in the boundary creation dialog box.

At 200mV/div, or 1.6VPP, 50mV is 3.125% of the vertical display. Therefore, we use 3.125% vertical tolerance in the boundary creation dialog box. The resulting boundary would appear as shown below.