GaGe

GageScope®

User's Guide

For version 3.1

P/N: 0045145
Reorder #: MKT-SWM-GSW09
0405

GageScope®
User's Guide

Eighth Edition (May 2001)

How to reach Gage Applied Technologies, Inc. for Product Support

Toll-free phone:	1-800-567-4243	Toll-free fax:	1-800-780-8411
Outside North America: Tel:	(514) 633-7447	Fax:	(514) 633-0770
Web Form:	www.gage-applied.com/support.asp		

Table of Contents

Software and Documentation License Agreement

CAREFULLY READ THE FOLLOWING TERMS AND CONDITIONS BEFORE OPENING THE CD PACKAGE. OPENING THE CD PACKAGE INDICATES ACCEPTANCE OF THESE TERMS AND CONDITIONS. IF YOU DO NOT AGREE WITH THEM, PROMPTLY RETURN THE PACKAGE UNOPENED AND YOUR MONEY WILL BE REFUNDED.

Title to the media on which the program is recorded and to documentation in support thereof is transferred to you, but title to the program is retained by Gage Applied Technologies, Inc. You assume responsibility for the selection of the program to achieve your intended results, and for the installation, use, and results obtained from the program.

License

Under the terms and conditions of this License Agreement you may:

a) use the program on a single machine;

b) copy the program into any machine-readable or printed form for backup or modification purposes in support of your use of the program on the single machine. Copying of documentation and other printed material is prohibited;

c) modify the program and/or merge it into another program for your use on the single machine (any portion of this program merged into another program will continue to be subject to the terms and conditions of this Agreement); and

d) transfer the program and license to another party if the other party agrees to accept the terms and conditions of this Agreement. If you transfer the program, you must at the same time either transfer all copies whether in printed or machine-readable form to the same party or destroy any copies not transferred; this includes all modifications and portions of the program contained or merged into other programs.

You must reproduce and include the copyright notice on any copy, modification or portion merged into another program.

You may not use, copy, modify, or transfer the program, or any copy, modification or merged portion, in whole or in part, except as expressly provided for in this License Agreement.

If you transfer possession of any copy, modification or merged portion of the program to another party, your license is automatically terminated.

Terms

The license is effective until terminated. You may terminate it at any time by destroying the program together with all copies, modifications and merged portions in any form. The license will also terminate upon conditions set forth elsewhere in this Agreement or if you fail to comply with any term or condition of this Agreement. You agree upon such termination to destroy the program together with all copies, modifications and merged portions in any form.

Limited Warranty

Gage Applied Technologies, Inc. warrants the media on which the program is furnished to be free from defects in materials and workmanship under normal use for a period of one year from the date of delivery to you as evidenced by a copy of your receipt.

THE PROGRAM IS PROVIDED “AS IS” WITHOUT WARRANTY OF ANY KIND, EITHER EXPRESSED OR IMPLIED, INCLUDING BUT NOT LIMITED TO THE IMPLIED WARRANTIES OF MERCHANTABILITY AND FITNESS FOR A PARTICULAR PURPOSE. THE ENTIRE RISK TO THE QUALITY AND PERFORMANCE OF THE PROGRAMS LIES WITH YOU. SHOULD THE PROGRAM PROVE DEFECTIVE, YOU (AND NOT GAGE APPLIED TECHNOLOGIES, INC.) ASSUME THE ENTIRE COST OF ALL NECESSARY SERVICING, REPAIR OR CORRECTION.

SOME JURISDICTIONS DO NOT ALLOW THE EXCLUSION OF IMPLIED WARRANTIES, SO THE ABOVE EXCLUSION MAY NOT APPLY TO YOU. THIS WARRANTY GIVES YOU SPECIFIC LEGAL RIGHTS AND YOU MAY ALSO HAVE OTHER RIGHTS WHICH VARY FROM ONE JURISDICTION TO ANOTHER.

Gage Applied Technologies, Inc. does not warrant that the functions contained in the program will meet your requirements or that the operation of the program will be uninterrupted or error free. However, Gage Applied Technologies, Inc. warrants the media on which the program is furnished to be free from defects in materials and workmanship under normal use for a period of one year from the date of delivery to you as evidenced by a copy of your receipt.

Limitations of Remedies

Gage Applied Technologies, Inc.’s entire liability and your exclusive remedy shall be:

a) with respect to defective media during the warranty period, Gage Applied Technologies, Inc. will replace media not meeting Gage Applied Technologies, Inc.’s “Limited Warranty” if returned to Gage Applied Technologies, Inc. or its authorized representative with a copy of your receipt, or

b) if Gage Applied Technologies, Inc. or its representative is unable to deliver replacement media free of defects in materials and workmanship, you may terminate the Agreement by returning the program and your money will be refunded.

IN NO EVENT WILL GAGE APPLIED TECHNOLOGIES, INC. BE LIABLE FOR ANY DAMAGES, INCLUDING ANY LOST PROFITS, LOST SAVINGS OR OTHER INCIDENTAL OR CONSEQUENTIAL DAMAGES ARISING OUT OF THE USE OF OR INABILITY TO USE SUCH PROGRAM, EVEN IF GAGE APPLIED TECHNOLOGIES, INC. OR ITS AUTHORIZED REPRESENTATIVE HAS BEEN ADVISED OF THE POSSIBILITY OF SUCH DAMAGES, OR ANY CLAIM BY ANY OTHER PARTY.

SOME JURISDICTIONS DO NOT ALLOW THE LIMITATION OR EXCLUSION OF LIABILITY FOR INCIDENTAL OR CONSEQUENTIAL DAMAGES, SO THE ABOVE LIMITATION OR EXCLUSION MAY NOT APPLY TO YOU.

General

You may not sublicense, assign or transfer the license or the program except as expressly provided in this Agreement. Any attempt otherwise to sublicense, assign or transfer any of the rights, duties or obligations hereunder is void.

This Agreement is governed by the laws of the Province of Quebec, Canada.

The parties agree that this Agreement be written in English. Les partis consentent à ce que cette entente soit rédigée en anglais.

Should you have any questions concerning this Agreement, you may contact Gage Applied Technologies, Inc. in writing. Please check the Gage website for the most current address: http://www.gage-applied.com/aboutgag/contact.htm.

YOU ACKNOWLEDGE THAT YOU HAVE READ THIS AGREEMENT, UNDERSTAND IT AND AGREE TO BE BOUND BY ITS TERMS AND CONDITIONS. YOU FURTHER AGREE THAT IT IS THE COMPLETE AND EXCLUSIVE STATEMENT OF THE AGREEMENT BETWEEN US WHICH SUPERSEDES ANY PROPOSAL OR PRIOR AGREEMENT, ORAL OR WRITTEN, AND ANY OTHER COMMUNICATIONS BETWEEN US RELATING TO THE SUBJECT MATTER OF THIS AGREEMENT.

Introduction

GageScope® is a powerful software package which, finally, bridges the gap between oscilloscopes and computers, helping scientists and engineers get the most out of their measurements.

All this performance is available without writing a single line of code or drawing any diagrams. GageScope® starts being productive right out of the box.

GageScope® takes full advantage of the Windows operating system, making the user interface for CompuScope and CompuGen cards very easy to learn and use.

On-screen controls and toolbars give you complete control of all capture and display features, allowing you to change settings while data capture is in progress. As many as 60 channels can be viewed together on the screen, and signals can be saved to disk for future use.

Optional software advanced analysis tools allow the power of GageScope® to be further enhanced by allowing deep FFTs, unattended transient capture, averaging, waveform parameter calculations and correlation. Many more analysis tools will become available in the future, making GageScope® even more powerful.

GageScope® is now available in the ***Professional*** and the ***Standard*** **Editions**.

- ***The Professional Edition***

  This edition contains the following advanced analysis tools:

  - Autosave
  - Averaging
  - Waveform Parameters
  - FFT
  - Extended Math

- ***The Standard Edition***

  All advanced analysis tools, with the exception of the Extended Math tool, are included in the Standard Edition but with limited functionality. The limitations are listed as follows:

  - AutoSave can only save a maximum of 4 files
  - While Averaging, running average is not available and the maximum averaging depth is 8192
  - Waveform Parameters tool supports only the following parameters:
    - Period
    - Frequency
    - Mean
    - RMS
  - The maximum length of FFT is 4096 points
  - Extended Math tool is not available

Features

- **No Programming Required – Works Out of the Box.** GageScope® brings all the advantages of the Pentium processor, Windows operating system and the Internet to the CompuScope and CompuGen cards without requiring the user to write a single line of programming code or draw diagrams. It works straight out of the box!
- **Automatic Detection of CompuScope or CompuGen Cards.** GageScope® automatically detects the presence of CompuScope and CompuGen Cards in the system, bringing true Plug-n-Play functionality to the Test and Measurement user.
- **Simple Interface, Complete Control.** Change such settings as sample rate, trigger depth, timebase, input range, and so on, with a click of the mouse. All options are viewable onscreen instead of in menus and dialog boxes.
- **Dynamic Display.** Adjustments made to any GageScope® settings are automatically reflected in the Display Window. Change the sample rate, channel mode, timebase, vertical scale, and other settings on the fly.
- **Zoom In and Out to Fully Utilize Deep Acquisition Memory.** Deep memory acquisition really comes to life with zooming capability of GageScope®. You can view the entire record, click on the Zoom tool, select the portion of the signal that is of interest to you and zoom in. Completely painless!
- **Scroll Through the Signal Using Familiar Windows Scroll Bars.** Now you don't have to scroll through large sets of data using a slow and ineffective scroll knob on a traditional DSO. You can use standard Windows horizontal and vertical scroll bars to view different portions of the signal.
- **Import Signals into Reports and Manuals.** Capturing signals in the Windows environment allows you to effortlessly bring the waveform plots into word processing documents such as project reports and product manuals.
- **Remote Control of your CompuScope or CompuGen Cards.** GageScope® can be run remotely using standard Windows utility programs. This allows you control over your CompuScope or CompuGen cards even when you are not physically in the laboratory.
- **Simultaneous Time and Frequency Domain Display.** Now you can view captured signals in both time and frequency domain using the powerful FFT capabilities of GageScope®.
- **Save Signals and Setups to Disk.** A virtually infinite number of signals can be saved using GageScope®. Saving signal files in binary or ASCII format allows you to post-process the data using analysis packages such as MATLAB, Excel, Mathcad etc. Setup files can also be saved, allowing you to easily go back to a previously set up experiment.

- **InfoView .** Not only does GageScope® allow you to capture signals and make measurements, it also provides results which can be displayed in the InfoView window on the screen.

- **Flexible Printing.** If you use GageScope®, you can print to any Windows compatible printer or fax, including network devices, to get a hard copy of your measurements.

- **Royalty Free File Viewer.** Signal files saved by GageScope® can be viewed by your colleagues on as many different stations as you desire, using the royalty-free signal viewer supplied with GageScope® and downloadable from www.GageScope.com.

- **Independent Timebase and Sample Rate.** Have you ever wanted to capture a signal at a very fast sample rate, but view it with a long timebase? CompuScopes with deep memory are capable of doing so, but many DSOs are not. GageScope® allows you to decouple sample rate from timebase for more flexible measurements.

- **Independent Vertical Scale and Full Scale Input Range.** Almost all oscilloscopes on the market assume that the signal they are measuring does not have enough dynamic range to require more than 8 divisions (256 rows of pixels across 8 divisions) on the oscilloscope screen.

  While this is true for an 8 bit DSO, it is not so for CompuScope cards which can have as much as 16 bit resolution which would require 65,536 pixel rows to do justice to the dynamic range.

  As such, GageScope® completely decouples Vertical Scale from Full Scale Input Range, allowing the user to scroll up and down on any vertical portion of the signal.

- **Unattended Transient Capture with Time and Date Stamp.** Optional AutoSave analysis tool for GageScope® allows completely unattended transient capture and storage to disk. A date and time stamp is also included with the saved data for record keeping purposes.

- **Advanced Analysis Tools.** GageScope® advanced analysis tools currently include Waveform Parameters, AutoSave, Averaging, FFT Analysis, and the Extended Math. These analysis tools dramatically enhance the functionality of GageScope®.

How to Use this Manual

To get the most out of GageScope®, it is recommended that you read through this manual.

- Part A, *Getting Started,* provides detailed instructions on installing GageScope®, running the program, and configuring your CompuScope and CompuGen cards with the Instrument Manager.
- Part B, *Tutorials*, contains seven tutorials designed to introduce how GageScope® works. If you are new to GageScope®, then *Tutorial 1: Fundamentals* is the best place to start.
- Part C, *Reference*, lists every feature of GageScope® along with detailed descriptions on how they work.
- Part D, *Appendices,* describes the GageScope® SIG binary signal file format, default GageScope® settings, help on setting the I/O address on CompuScope cards, and other information related to using CompuScope cards. It also contains information on how to configure CompuScope and CompuGen cards using the Instrument Manager utility.
- Part E, *Glossary,* lists the terms used in this manual, with definitions for each.

Important Notes

- In order to produce most of the examples throughout the manual, CompuScope 2125, CompuScope 8500, and CompuGen 1100 cards were used. In instances where examples refer to settings that are specific to the equipment being used, it may be necessary for you to alter the suggested settings to suit your particular instrument.
- CompuScope is sometimes abbreviated “CS” for brevity. For example, CompuScope 12100 could be written as CS12100. CompuGen is abbreviated as “CG”.
- “CompuScope x012” and “CSx012” are used to refer to the 12 bit ISA bus CompuScope cards: CS8012A, CS8012, CS6012, CS1012, and CS512.
- “CompuScope x012/PCI” and “CSx012/PCI” are used to refer to the 12 bit PCI bus CompuScope cards: CS8012A/PCI, CS8012/PCI, CS6012/PCI, CS1012/PCI, and CS512/PCI.

Specifications

System Requirements

- PC with a Pentium 200 MHz processor; 266 MHz Pentium II or higher strongly recommended
- Microsoft Windows 95, Windows 98, Windows NT 4.0 or Windows 2000 operating system
- Arial font is required for the user interface. If it is not present on the Operating System, it can be installed from the original OS install disk(s)
- 32 MB RAM with Windows 95/98, 48 MB RAM with Windows NT/Windows 2000; 128 MB RAM or more recommended on all systems
- Hard disk free space: 20 MB to install the GageScope® application; 60 MB for full install (includes Online Help facility and PDF version of User Manual). 128 MB minimum free space necessary for operation of the GageScope® program
- Virtual memory under Windows NT and 2000: it is recommended that you increase the Swap File Size by 128 MB to operate GageScope®
- CD-ROM drive
- 15-inch monitor 800 x 600 (Super VGA); 17-inch 1024 x 768 Ultra VGA recommended. Note that your system must be set to 256 color display in order for the FFT capability to display results properly.
- PC keyboard and 2-button mouse (3rd button, if present, is not used)
- Internet Explorer 4.1 or higher; version 5.0 required for full functionality
- The drivers for the PC-based cards must be installed. The PC-based cards are the CompuScope and CompuGen card(s). You must run the Instrument Manager utility to configure the PC-based cards.

Supported Instruments

Currently supported instruments include:

CompuScope Cards

CS85G, CS82G, CS8500, CS14100, CS1450, CS12100, CS1250, CS1602, CS512/PCI,

CS512, CS2125

CompuGen 1100

Note

For an updated list of supported instruments, visit the GageScope® web site at http://www.gagescope.com

System Control Settings

CompuScope Cards

Sample Rate...................... Instrument dependent (5 GS/s to 1 kS/s)
Capture Modes .... Dual or single channel, if supported by your hardware
Trigger Source .... Ch 1, Ch 2, Ext or Pattern Triggering, if available
Trigger Slope .... Positive or Negative
Trigger Level .... User selectable
Timeout............................ 1 ms to unlimited
Post-trigger Depth .... 0 to 100% of CompuScope memory
Pre-trigger Depth .... 0 to 100% of CompuScope memory
Depth Units...................... Samples
Multiple Record............... Available, if supported by your CompuScope

CompuGen Cards

Sample Rate...................... Instrument dependent
Output Channels .... Instrument dependent
Trigger Source .... Internal or External
Mode................................ Triggered, Continuous, or Burst
Source .... AWG Channel loaded or created within GageScope
Gain .... Instrument dependent
Offset .... Instrument dependent
Filter .... Instrument dependent

Display Control

Timebase.......................... Instrument dependent; no less than 2 samples per division and no more than the entire signal in full range can be displayed.
Display Features .... All channels plus the Grid, X axis, Y axis, Zero Lines, and Trigger Line can be hidden. Colors for these elements as well as the background color are customizable.

Channel Control

Max # of Channels 60
Full Scale Input .... ±20 mV to ±205 V, if supported by your CompuScope
Input Coupling................. AC or DC, if supported by your CompuScope
Impedance .... 50 Ohm and 1 Mega Ohm, if supported by your CompuScope
Vertical Scale .... 5 mV/div to 1000 V/div, if supported by your CompuScope.
Polarization....................... Normal or inverted
Display Methods.............. Mean, MinMax or Skip Samples
Other options .... Connect Dots on/off, color, pen style, pen width

Advanced Analysis Tools

AutoSave................................Allows unattended transient capture to disk with time and date stamp for record keeping

Averaging...............................Allows averaging or co-adding of multiple acquisitions. Averaged or co-added data can be displayed and saved to disk

Waveform Parameters ....Mean, RMS, Amplitude, Peak-to-peak, Period, Frequency, Fall Time, Rise Time, Positive Width, Negative Width, Positive Duty, Negative Duty, Positive Overshoot, Negative Overshoot, Peak, Trough, Top Value, Bottom Value, TAA Pos, TAA Neg, TAA, Pk-Pk Time, Pk-Tr Time, Tr-Pk Time, Tr-Tr Time, Ext-Ext Time, Base Line, Delta Base Line

FFT Analysis..........................Up to 4 million point FFT

Extended Math ....Allows creation of sub-channels, auto- and cross-correlation, differentiation and integration.

Math

Addition, subtraction, multiplication or division of up to two channels, physical or disk-based

Info View

Window.............................Docking type (resizable)

Display ....Any number of parameters from any channel

Other Features

Saving ....GageScope® .SIG files, AWG .EGG files, ASCII .TXT files, GageScope® Setup .INI files

Loading ....GageScope® .SIG files, AWG .EGG files, GageScope® Setup .INI files

Printing.............................Any Windows 95/98, Windows NT or Windows 2000 compatible printer; color printing supported

Materials Supplied

GageScope® CD
GageScope® User's Guide

Warranty

One year warranty on the media which contain the GageScope® program.

All Specifications are subject to change without notice

Getting Started

Follow the simple, 4-step procedure outlined below to properly install GageScope® and start running within minutes of opening the GageScope® software package:

- Install and verify the Windows Drivers for your CompuScope or CompuGen card.
- Install GageScope®.
- Configure GageScope® to work with your CompuScope and CompuGen card(s) using the Instrument Manager utility.
- Launch GageScope®.

Note :

The procedure outlined in this section assumes that you are installing the software using a CD.

If you have downloaded GageScope® software off the Gage web site, please extract the SETUP and data files to a folder on your hard disk and run the SETUP program from that folder.

Installing the Windows Device Drivers for your CompuScope or CompuGen Card

In order to operate your CompuScope or CompuGen card under GageScope®, you must install the appropriate Windows driver for your CompuScope or CompuGen card before launching GageScope®.

For instance, if you have a CS8500 card and want to operate GageScope® under Windows NT, you must install the CompuScope Win NT Drivers in addition to installing GageScope® under the same operating system.

Failure to install the appropriate driver will result in GageScope® not being able to communicate with the cards installed in your system.

For complete instructions on installing the Windows Driver for your CompuScope, consult the Driver Installation Guide that came with the CompuScope or CompuGen hardware.

Installing GageScope®

Step-by-step Installation

1 Insert the GageScope® CD-ROM into your CD-ROM drive.
The installation program will start automatically after a few seconds.

2 If you are running a Windows 95 system that does not have the appropriate version of DCOM95 installed, you will be asked if you want to install DCOM95. You must select the installation of DCOM95.

After DCOM95 has been installed, you will then have to reboot your PC and run the GageScope® installation program again.

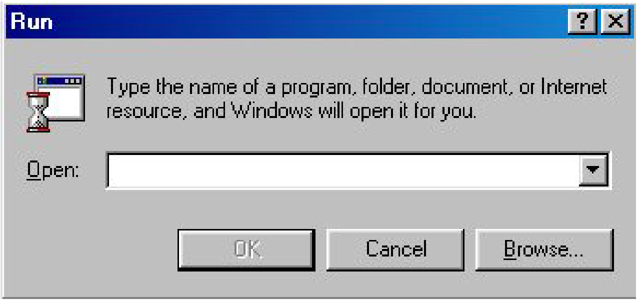
If you are installing GageScope® under Windows 98, Windows NT, or Windows 2000, the **DCOM95** will not be installed as these operating systems come with all DCOM components required by GageScope®.

3 If your system does not have Internet Explorer 5.0 or equivalent installed, you will be asked to install Internet Explorer 5.0 from Microsoft's web site.

After Internet Explorer 5 has been installed, you will then have to reboot your PC and run the GageScope® installation program again.

4 ***In case the installation program does not start automatically, follow these steps to start the setup process:***

Click **Start** from your desktop and select **Run**. You will see the **Run** dialog box as follows:



5 Type ***CDROM drive letter*:\GageScope\Disk1\Setup.exe** in the **Open** text box The CDROM drive letter depends on the settings in your PC.

Click **OK**.

6 The installation program will begin its setup procedure. You should then see the Welcome screen.

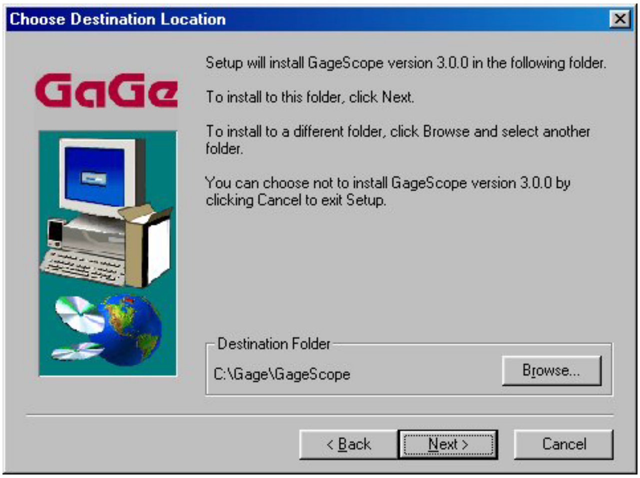
Click **Next**.

7 In the User Information screen, enter your **Name**, **Company** name and **Serial** number, found on the back of your CD-ROM jewel case.

If you do not enter a proper Serial Number, some of the functionality of GageScope® may not be available to you.

Now click **Next**.

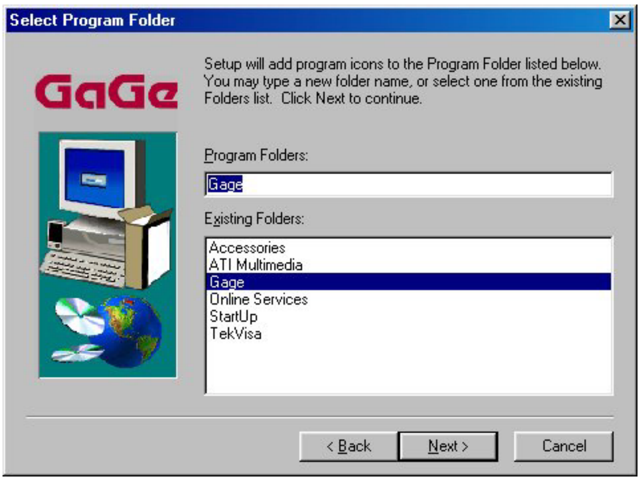
8 Choose a destination location for GageScope®.



It is recommended that you accept the default location of **C:\Gage\GageScope**, but if you wish to use a different location, click on the **Browse** button and select the destination of your choice.

Click **Next**.

9 Now you must select a Program Folder. It is recommended to accept the **Gage** program folder, click **Next**; otherwise, type in another name and click **Next**.



10 The **Start Copying Files** screen gives you a final notice of what will be installed, and in which directory. The components being installed depend on the Edition of GageScope® you have purchased. The current installation shows that you are installing the Professional Edition of GageScope®. The settings will change if you install the Standard Edition of GageScope®.

If you have made a mistake, click **Back**. Otherwise, click **Next** to continue.

11 The installation process will commence. You will see the installation progress indicators during the installation process. When the installation is successfully completed, you will be prompted to register online:

Click **Yes** to register online. Then follow the onscreen instructions to complete the registration process.

Click **No** to register later. For the current installation, we have opted to register later.

12 You will see the **Setup Complete** screen.

The default option shown above is to restart your computer at this stage. You can restart your computer later by selecting the other option.

Please note that GageScope® will start working only after a restart, as some components must be registered with the Windows operating system.

13 Installation is now complete.

IMPORTANT:

- You must now proceed to Install the Windows device drivers for your CompuScope and CompuGen card(s), if you have not done so prior to installing GageScope® program.
- After installing the drivers, use the Instrument Manager utility to configure these PC-based cards.

Configuring CompuScope and CompuGen Cards with Instrument Manager

The Instrument Manager configures one or more CompuScope and CompuGen cards in your system.

It is very important to install the appropriate Windows drivers for your CompuScope or CompuGen cards before attempting to use Instrument Manager.

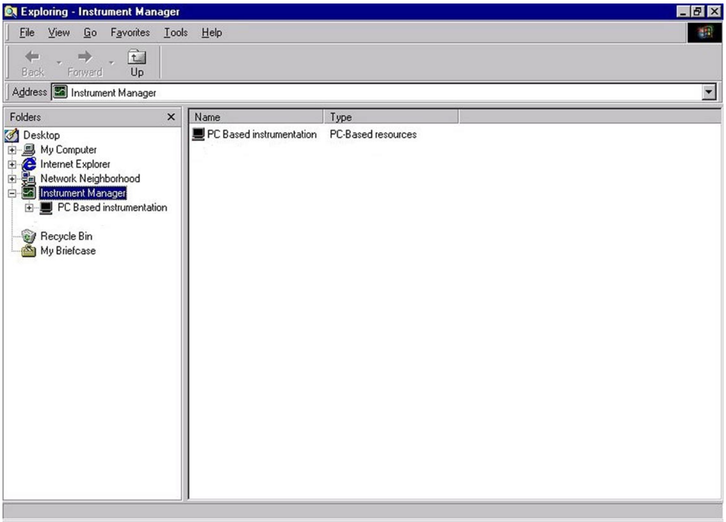
For example, if you have a CompuScope 85G present in your PC, you must install the PCI drivers first before using the Instrument Manager utility.

Refer to the CompuScope driver installation guide for step-by-step PCI driver installation procedure for the PCI-based CompuScope cards.

Step-by-step configuration procedure

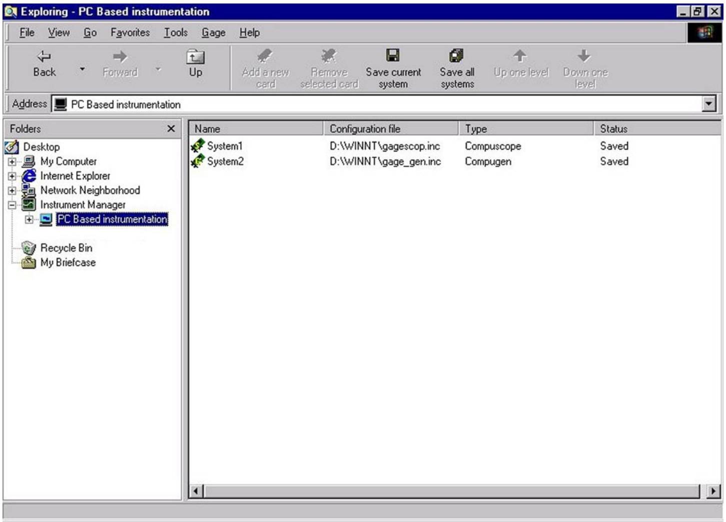
1 Launch Instrument Manager from your desktop using the following sequence of left mouse clicks: **Start --> Explore --> Instrument Manager.**

Click **Instrument Manager** Icon to launch the Instrument Manager Explorer. You can also get directly to this step by double-clicking on the Instrument Manager Icon on your computer's desktop.



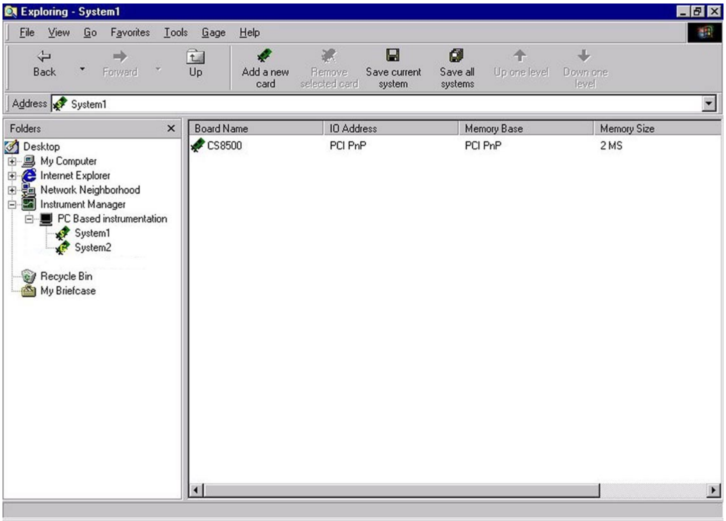
In the right pane, you will find the PC-based Instruments under the **Name** and **Type** columns.

2 Click **PC-based Instrumentation** Icon, . The PC-based Explorer opens:



Note that the right pane displays the PC-based resources currently present in the system. **System1** comprises CompuScope card(s) while **System2** consists of CompuGen card(s).

3 Double click on **System1** to view the System1 configuration as follows:

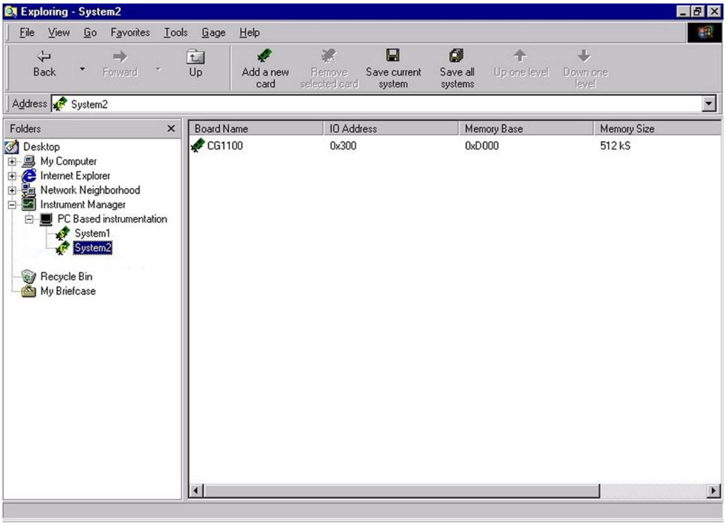


You can see that the CompuScope card, CS8500, present in the system, has 2MegaSamples of onboard memory.

The CS8500 configuration settings are saved in the Status field. The Status field is the last field in the right pane. You may have to use the horizontal scroll bar to view this field. In case the Status field does not show Saved, click on **Save configuration** icon from the toolbar or the drop down **Gage** menu.

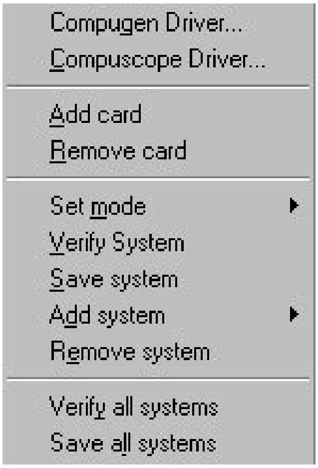
Follow the same procedure for the CompuGen card to view and save the configuration settings for that card.

4 Double click on **System2** to view the System2 configuration as follows:



You can view the configurations settings for a single CompuGen card, CG1100, present in your PC. Click on **Save configuration** icon from the toolbar or the drop down **Gage** menu.

Gage Menu in the Menu bar within Instrument Manager Explorer allows users to configure newly installed Gage cards, set various options like adding, removing, saving file configurations for Gage CompuScope and CompuGen cards. Click on the **Gage** in the menu bar.The Gage menu drop down list appears:



The details of each individual item in the menu are explained in Appendix A

Once the CompuScope and CompuGen cards are configured properly, these need to be verified for proper operation. Click **Verify System** and **Verify all Systems** to make sure that the newly installed card(s) are verified.

The description given above is a quick verification procedure in case the driver installation encountered no problems. You must consult Appendix A for detailed description of the items in the Instrument Manager utility.

Launching GageScope® for the First Time

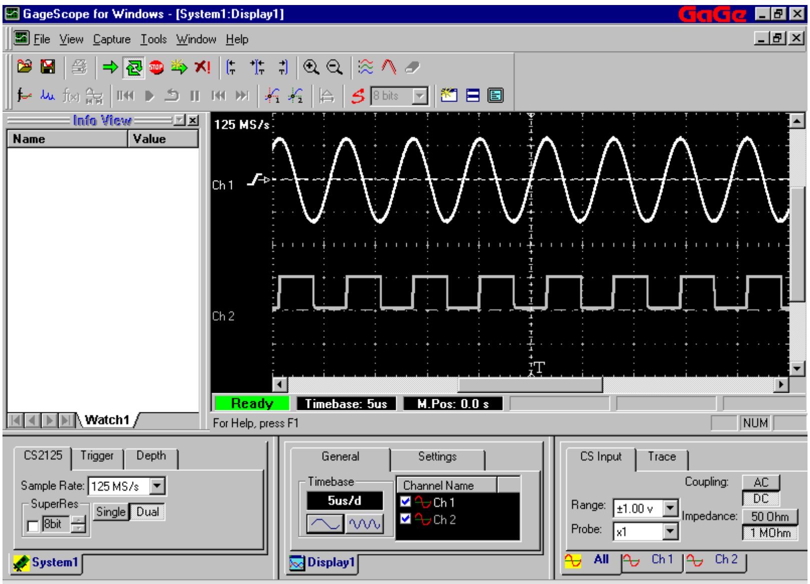
Before you run the program, connect a signal (from a signal generator or other source) to the Channel A input of your CompuScope card.

Launching GageScope®

1 A Windows shortcut icon was placed on the desktop during installation. Double-click this icon to run GageScope®.

You can also access GageScope® through the Start menu. Click on **Start, Programs, Gage, GageScope, GageScope**.

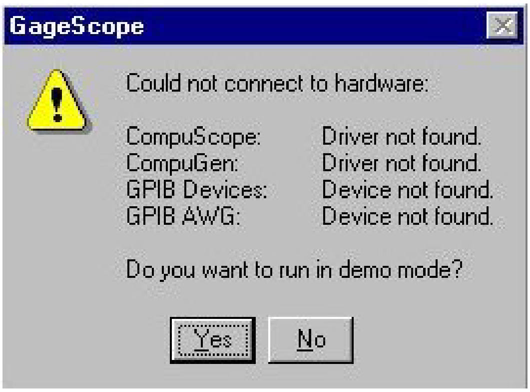
2 If your CompuScope card is configured properly, GageScope® will start.



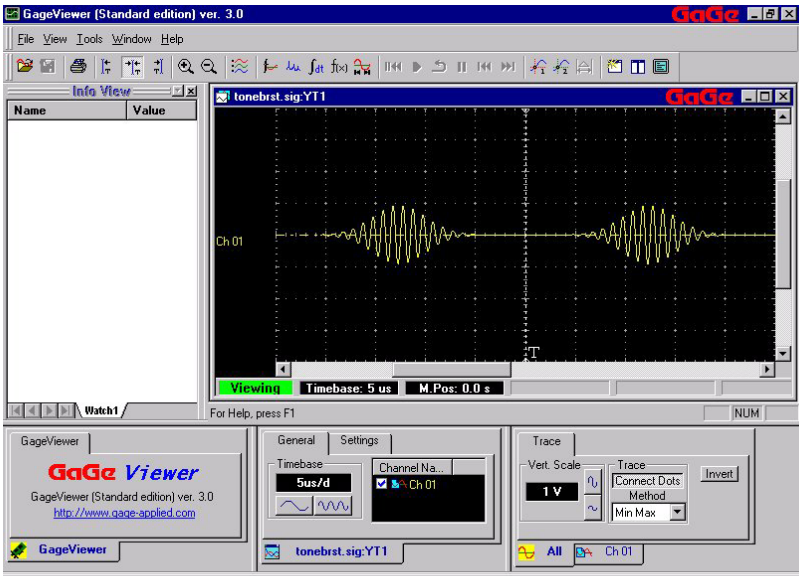
In the above example, two signals are being acquired using a CompuScope 2125. Your screen may differ. If you are not feeding any signals into the input channel(s), you will see a flat line.

No Instruments Detected?

- If you do not have any instrument installed in your system, you will see the following message at launch:



- You will be asked if you want to run GageScope® in Demo Mode.



As you are not connected to a live instrument, some features are disabled. You can, however, load and save disk-based GageScope® files (extension .SIG, .EGG, etc). To load a file, click on **File > Load Channel**. Sample .SIG files are located in the **Signal Files** folder inside the **C:\Gage\GageScope** folder. Sample .EGG files are located in the **AWG Equations** folder inside the **C:\Gage\GageScope** folder.

Default Settings

In order to display a signal immediately upon startup, GageScope® assumes certain defaults when it is run for the first time. Subsequent sessions will set the CompuScope and CompuGen hardware to the same settings that were in use when the previous session was closed.

The defaults for the first session are:

CompuScope Card Settings

Capture Mode:	Continuous
Channel Mode:	Dual (if two channels are available)
Sample Rate:	Maximum allowed by your CompuScope
SuperRes Mode:	Off

Trigger Settings

Trigger:	Set to channel 1
Trigger Level:	0 mV
Timeout:	10 ms
Slope:	Positive
Pre-trigger depth:	4096 Samples (if available)
Post-trigger depth:	4096 Samples (if available)

Channel Settings

Identification:	Channel A of the CompuScope is channel 1 Channel B of the CompuScope is channel 2
Colors:	Channel 1 set to yellow, channel 2 set to green
Input Range:	All channels set to ±1V (±2V for CS1016)
Position:	Depends on the number of channels. If there are two channels, channel 1 is displayed in the top half of the Display Window, and channel 2 is displayed in the bottom half. If there is only one channel, it is displayed in the middle of the Display Window
Vertical Scale:	All channels set to 1 Volt per division
Method:	Min Max
Trace:	Connect Dots
Impedance:	1 MOhm (if the CompuScope allows it)
Probe:	X1

Display Settings

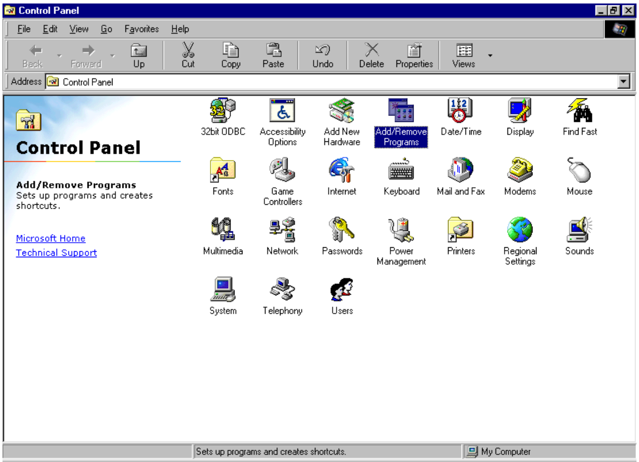
Timebase:	5 us/d
Channels Shown:	All channels visible. A CompuScope channel that is not connected to a signal will display as a flat line
Persistence:	Off
Grid:	On
X/Y Axis:	On
Zero Lines:	On

Uninstalling GageScope®

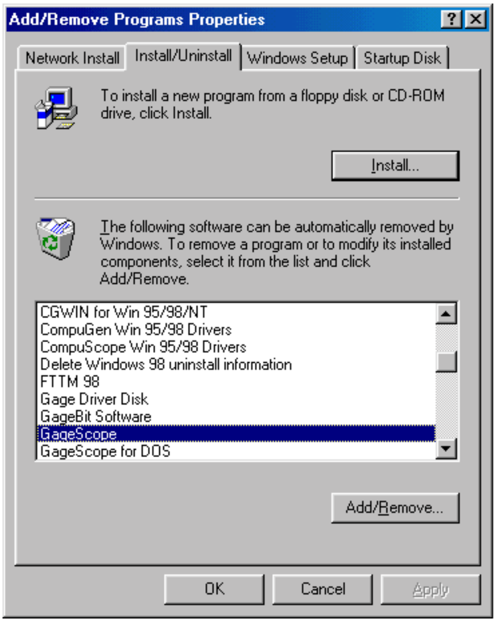
GageScope® is equipped with a special uninstall program, allowing you to uninstall all components at once. To uninstall, simply follow these steps:

1 In your Windows Start menu, select **Settings → Control Panel**.

2 Double-click on **Add/Remove Programs**.



3 The Add/Remove Program Properties dialog box will appear:



In the list of programs, select GageScope®. Click **Add/Remove**.

4 The **Select Components** dialog will appear.

You have the option to either reinstall the GageScope® program or completely remove it from your system.

5 To completely uninstall all installed GageScope® components, select **RemoveAll** from the **Select Components** screen:

Click **Next**.

6 All components will be uninstalled as soon as you confirm by clicking **OK**.

7 Once your system is no longer busy, close the Control Panel, re-open it and select Add/Remove Programs to verify that GageScope® and all installed components have been removed from the list of installed programs.

Tutorials

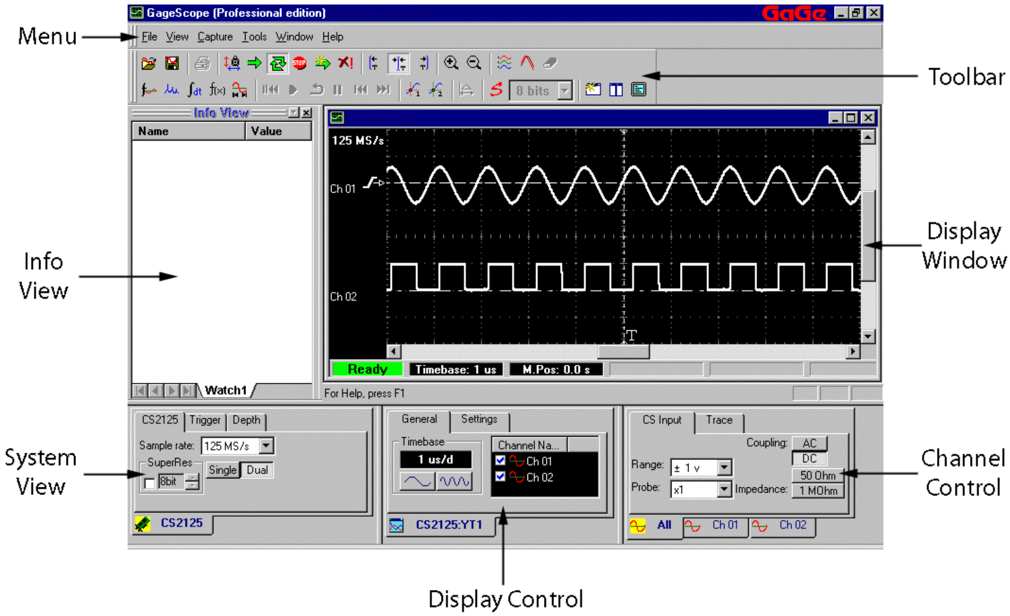
About the Tutorials

This chapter consists of seven tutorials:

Each section is a self-contained tutorial. If you are new to GageScope®, it is recommended that you do the tutorials in the order they are presented, as each section assumes you have gone through the previous material.

Tutorial 1: Fundamentals

The GageScope® screen contains menus, a toolbar, a display area, and controls for changing most settings.

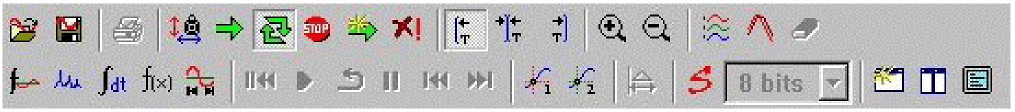


Menu Bar

The menu bar gives you mouse access to many GageScope® features, such as opening and saving files. As you will soon see, most features are accessed through easy-to-use on-screen controls.

Toolbar

The Toolbar gives you one-click access to commonly used features such as opening and saving files, cursors, capture controls, SuperRes, trigger alignment, zoom tools, full screen display etc.



If you wish to find out what each button does, position the mouse pointer over a button – a "tool tip" will appear. For more information, see the *Toolbar* in the Reference section of this manual.

Display Window

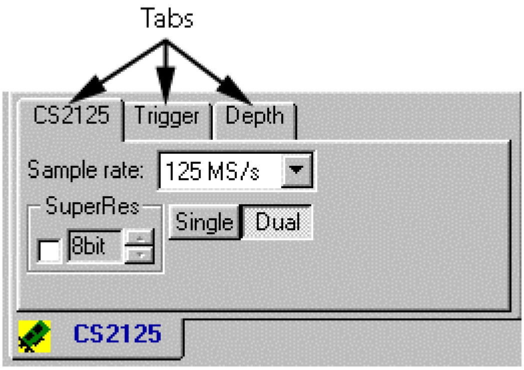
The Display Window displays live signals being acquired from your CompuScope card as well as signals you load from disk. You can also move signals and change certain settings directly in this window.

Status Bar

The Status Bar is located at the bottom left of the Display Window. It shows the status of the CompuScope card (triggered, waiting for a trigger, etc.), the current timebase, and the relative position of the trigger (M.Pos).

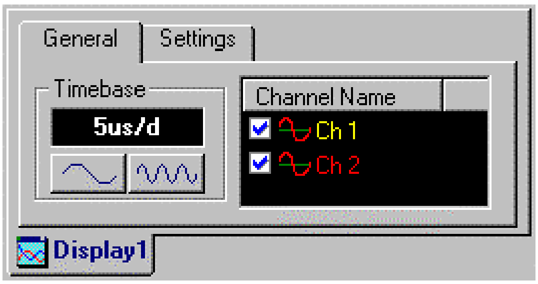
System Control

The System Control contains options for setting the sample rate, the channel mode (single or dual), type and level of trigger, and trigger depth. Related settings are grouped into tabs. For example, settings related to the amount of data captured (pre-trigger depth, post-trigger depth, multiple record, etc.) are in the Depth tab.



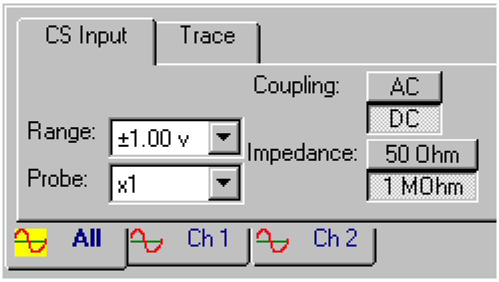
Display Control

The Display Control contains options for changing the look of the current display, such as the timebase and channel visibility, along with display element properties such as grid color. These settings do not affect the captured data.



Channel Control

The Channel Control contains options for changing the settings of individual channels, such as position, vertical scale, input range, and coupling.

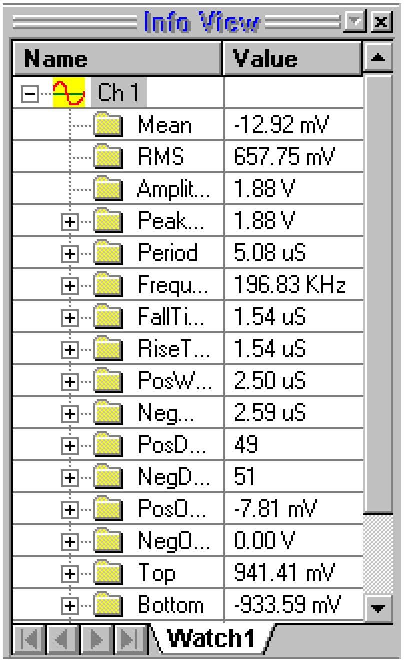


InfoView Palette

The InfoView palette is used to display various measurements being performed on the signal. These capabilities are normally performed through optional advanced analysis tools, which add extra functionality to GageScope®.

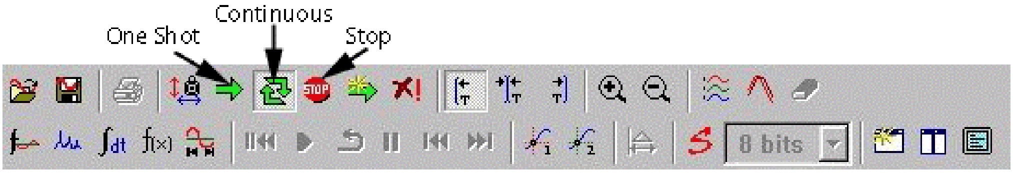
For example, InfoView is the window used by the Waveform Parameters analysis tool, which performs measurements such as Mean, Amplitude, Peak-to-peak, and many others on the specified channel of the active display.

Below is a sample of what the Waveform Parameters analysis tool displays.



About Capture Modes

GageScope® has two capture modes: Continuous and One Shot. These modes are represented in the capture portion of the toolbar along with a “Stop” button, an “Abort” button and a “Force Trigger” button.



The “pressed-in” button indicates the current mode. By default, GageScope® sets the capture mode to Continuous upon startup.

Continuous Mode In Continuous Mode, GageScope® repeatedly waits for a trigger event, acquires and displays the data, then waits for another trigger event.

One Shot Mode In One Shot Mode, GageScope® waits for a trigger event, acquires and displays the data, then stops.

When you click on the **One Shot** button, GageScope® automatically disables its timeout feature. In other words, the program will wait forever for a trigger event to occur. This allows for unattended capture of both pre- and post-trigger data.

Stop Button If you are in Continuous or One Shot Capture Mode and click on the **Stop** button, GageScope® finishes the current capture, then stops. If a trigger event has yet to happen, GageScope® acquires and displays the data that existed the moment you pressed the **Stop** button. If the acquisition were to take more than 2 seconds to finish after you press **Stop,** a dialog box will appear with a completion bar which shows how much longer you have to go before the acquisition is to finish.

Abort Button If you are in Continuous or One Shot capture mode and click on the **Abort** button, GageScope® aborts the current acquisition immediately. This is helpful for very long acquisitions. The data you see is that which existed when you clicked on **Abort**.

Force Trigger Button If you want to force a trigger to occur upon manual command, you can click on this button.

To see changes on screen as you make adjustments to your signal, leave GageScope® in Continuous Capture Mode. If you are sure of your settings and are setting up a trigger in order to do a one-time capture of data, use One Shot Mode.

Triggering is discussed in more detail in Tutorial 3.

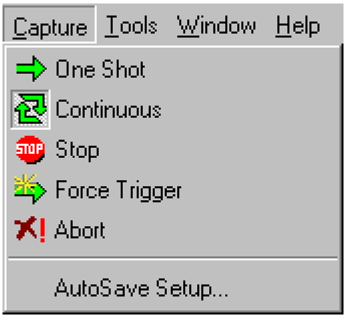
Capture Mode Exercise

The following exercise will demonstrate the different capture modes and their characteristics. The exercises will only be useful if you have a signal connected to your CompuScope card, and if triggering is left to its default settings. To avoid any problems during this tutorial, do not change any settings other than those demonstrated in the examples.

Set the Capture Mode to Continuous

1 Check if GageScope® is set to continuous capture. (The **Continuous** button  in the toolbar should be "pressed in.")

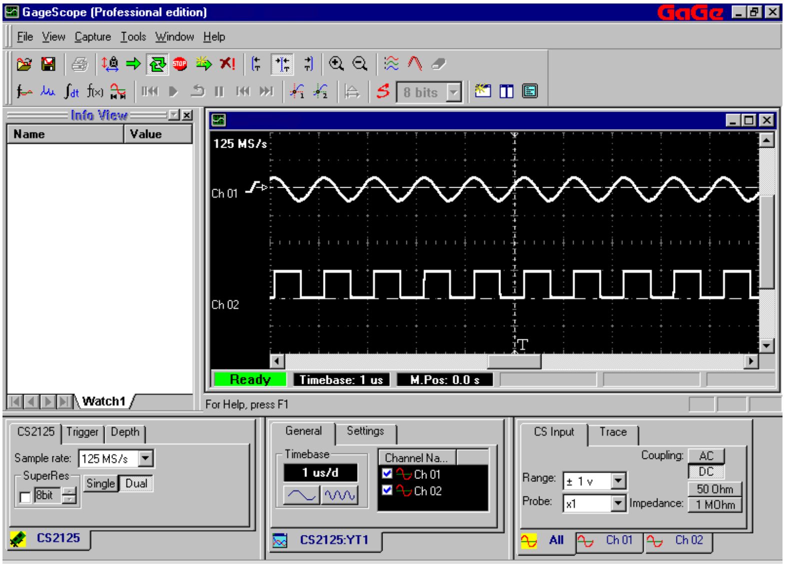
You can also click on the Capture menu—the item that is "pressed in" indicates the current mode.



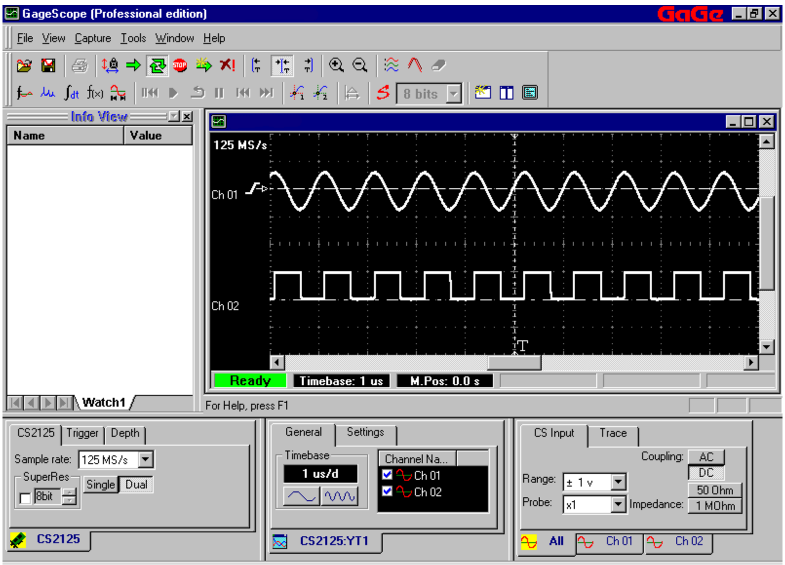
2 As GageScope® is set to capture continuously, the signal you are seeing is "live."

As demonstrated in the next few pages, any adjustments you make to your signal generator are immediately reflected in the Display Window.

3 Reduce the amplitude of the signal on your generator. The signal in the Display Window should change accordingly. Note that channel A of the CompuScope card corresponds to channel 1 in GageScope®.

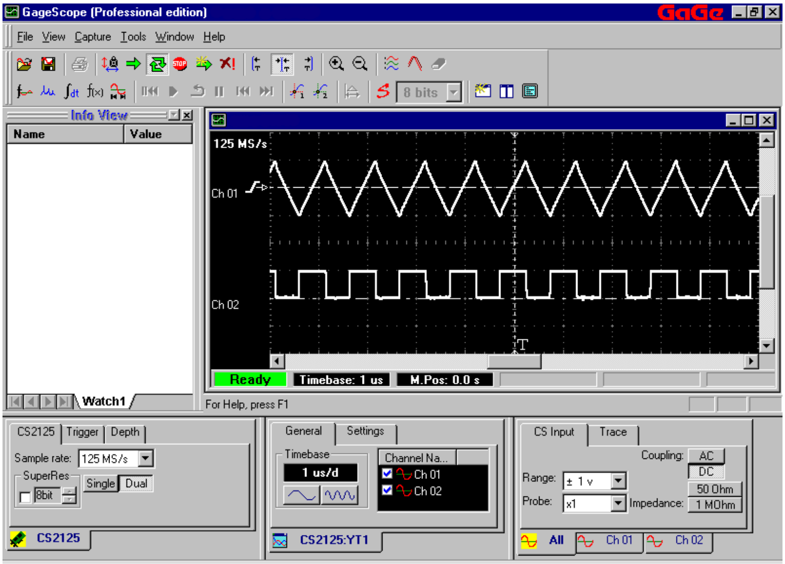


4 Increase the amplitude of your signal back to its original setting. The signal in the Display Window should increase accordingly.



5 Change the signal to a triangle wave.

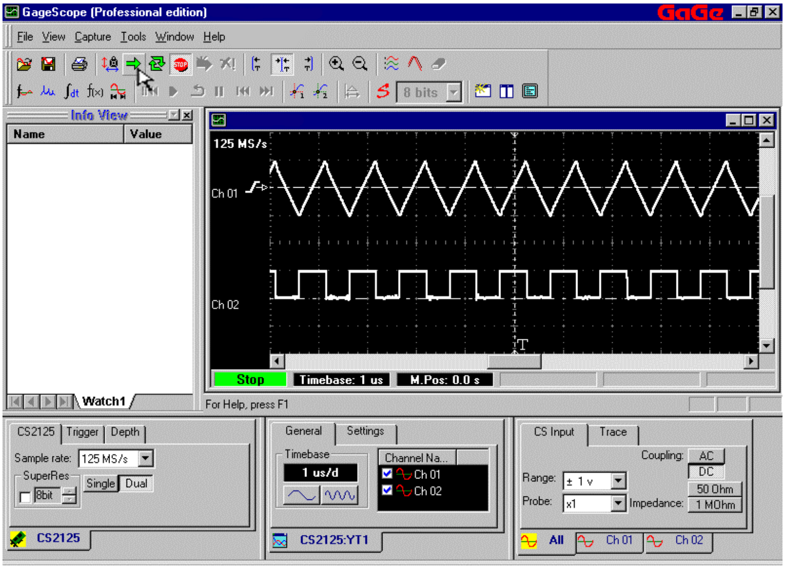
Again, the signal in the Display Window should change immediately, as GageScope® is set to capture continuously.



Set the Capture Mode to One Shot

1 Click on the **One Shot** button in the Toolbar. You can also choose **One Shot** from the Capture menu.

2 Once you click on **One Shot**, the **Stop** button becomes active.



This is normal. When you clicked on **One Shot**, GageScope® waited for a trigger event, acquired and displayed the data. If the trigger settings were not changed since launching GageScope®, the trigger event occurred immediately.

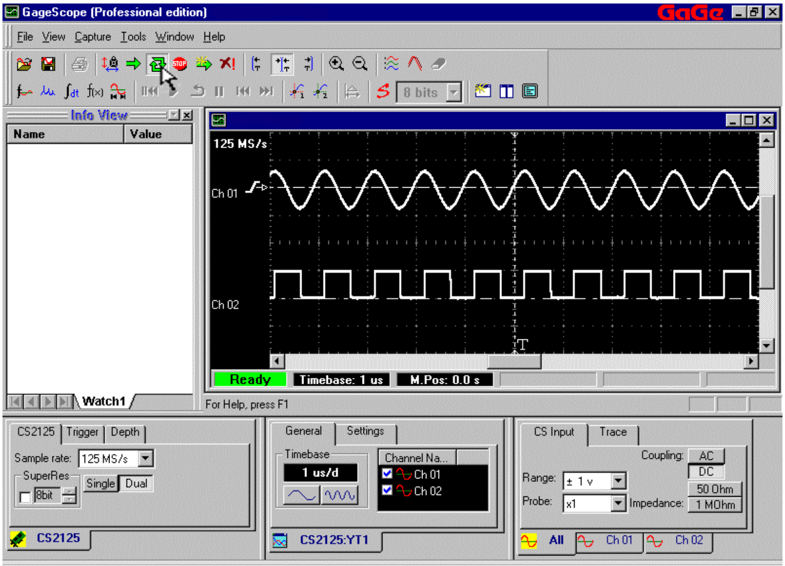
Once the capture was done, instead of looking for a new trigger event, GageScope® stopped all trigger-and-capture operations. This is the reason the **Stop** button became enabled.

3 Change the signal on your generator to a sine wave.

As GageScope® is in Stop Mode, the signal in the Display Window does not change—GageScope® is not looking for new trigger events, and no new captures are being performed.

Keep in mind that, although the signal in the Display Window is a triangle wave, the signal supplied by your generator is now a sine wave.

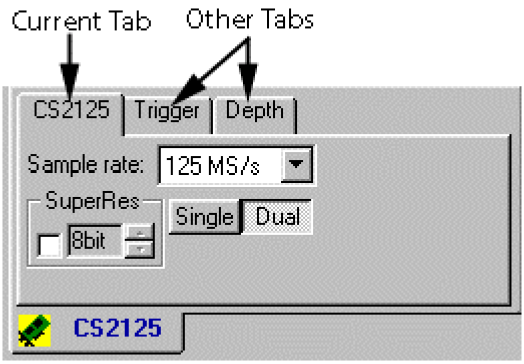
4 Click on the **Continuous** button  in the toolbar. The Display should change to a sine wave as soon as you press the **Continuous** button, as GageScope® immediately looks for, then acts on, a trigger event.



Again, as the trigger is set to the same channel that is being acquired, the trigger event and the subsequent change in the display occur immediately.

About Controls and Tabs

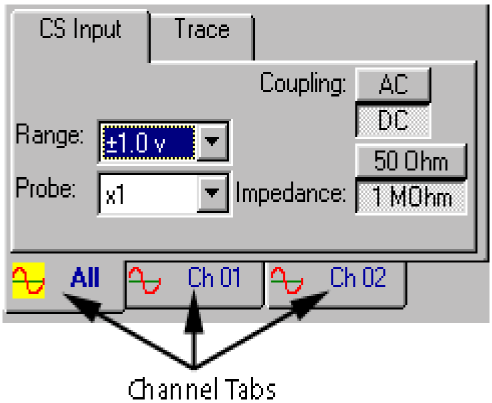
Tabs allow you to quickly switch between the many different parameters found in each control. The current tab is indicated by it being “in front,” as shown below. To switch to a different tab, click on that tab with the left mouse button.



In the System Control, you should have at least one tab with the name of your CompuScope card. As a CompuScope 2125 was used for this tutorial, **CS2125** shows up as a tab in the System Control.

Control-level tabs

Some controls have more than one tab at the bottom of the control. For example, the Channel Control (at the lower right of the screen) has tabs underneath the control, one for each channel.



When making changes to channel settings, always verify that the proper channel tab is in front.

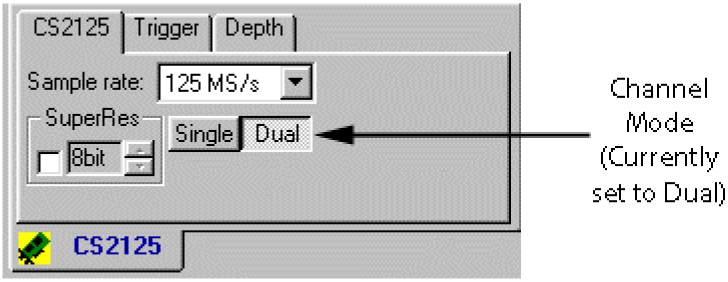
Channel Mode

The Channel Mode setting defines whether the CompuScope card should operate in single or dual channel mode.

- In dual channel mode, both channels A and B on your CompuScope card are available for input.
- In single channel mode, only channel A is available for input.

Note that not all CompuScope cards support this feature; the CompuScope 8500 and CompuScope 1016 are single channel only, for example, while the CompuScope 512 and CompuScope 512/PCI are dual channel only.

Click on the tab that is farthest to the left in the System Control to see this setting. The name of this tab will depend on your CompuScope model. In the case of this tutorial, the tab is titled CS2125.



Effects on Sample Rate

The advantage of single channel over dual channel is that the maximum sample rate in single channel is twice that of dual channel. For example, the maximum sample rate of a CompuScope 2125 in dual channel mode is 125 MS/s, while in single channel mode, it is 250 MS/s.

You can also use the entire memory buffer of your CompuScope for one channel when you are in single-channel mode. In dual-channel mode, the memory buffer is divided equally between the two channels.

Changing the Channel Mode

By default, channel mode is set to Dual. If you do not see the Channel Mode setting, your card does not have the ability to switch channel modes.

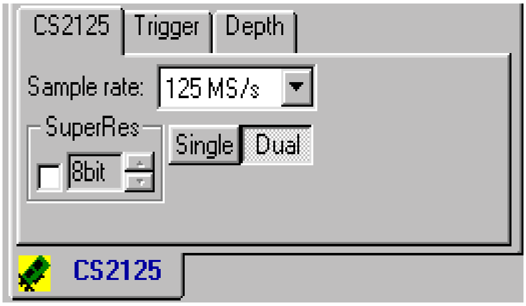
To change to single channel mode, click on the Single button.

Change Back to Dual

Click on the Dual button to change the channel mode back to dual. For the rest of this tutorial, leave this setting in Dual Channel mode. (As the CompuScope 8500 and CompuScope 1016 support single channel mode only, you will have to stay in single channel mode.)

Sample Rate

The sample rate defines how many sample points per second should be acquired by the CompuScope card. The available sample rates will change depending on the CompuScope model and the channel mode.



It is best to set the sample rate to at least twice that of the input frequency. For example, if you are sampling a 1 MHz signal, you should set the sample rate to 2 MS/s or higher.

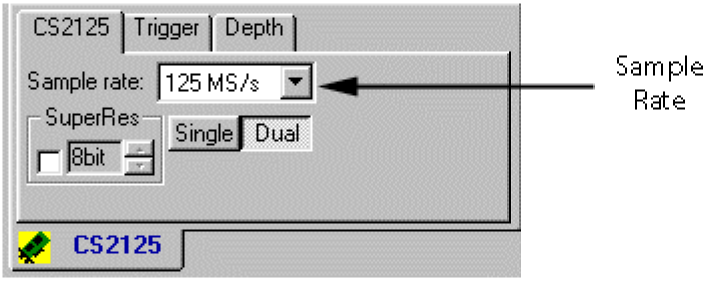
Obviously it is preferable to have as many samples per cycle of the input frequency as possible. The industry standard is to have five to ten samples per cycle. In other words, sample rate should be five to eight times the input frequency to get a good representation of the signal on the screen.

Effect of Channel Mode on Sample Rate

The maximum sample rate in single channel mode is twice that of dual channel mode. For example, the maximum sample rate of a CompuScope 2125 in dual channel mode is 125 MS/s, while in single channel mode, it is 250 MS/s.

Initial Settings

Upon startup, GageScope® sets the channel mode to dual and the sample rate to the maximum available. As a CompuScope 2125 was used for this tutorial, the sample rate was set to 125 MS/s, the maximum available in dual channel mode.


CS2125
Trigger
Depth
Sample rate:
125 MS/s
SuperRes
8bit
Single
Dual
CS2125
Sample Rate

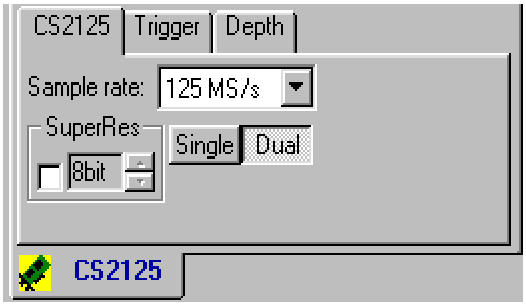
Sample Rate Exercise

In this short exercise you will change the sample rate to a very low value and then back to a high value.

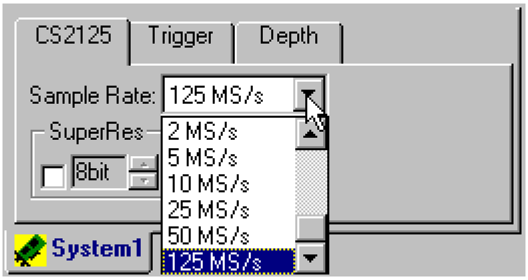
Change the Sample Rate to a Low Value (5 MS/s for example)

1 Verify that you are in dual channel mode and that the Continuous Mode button is active.

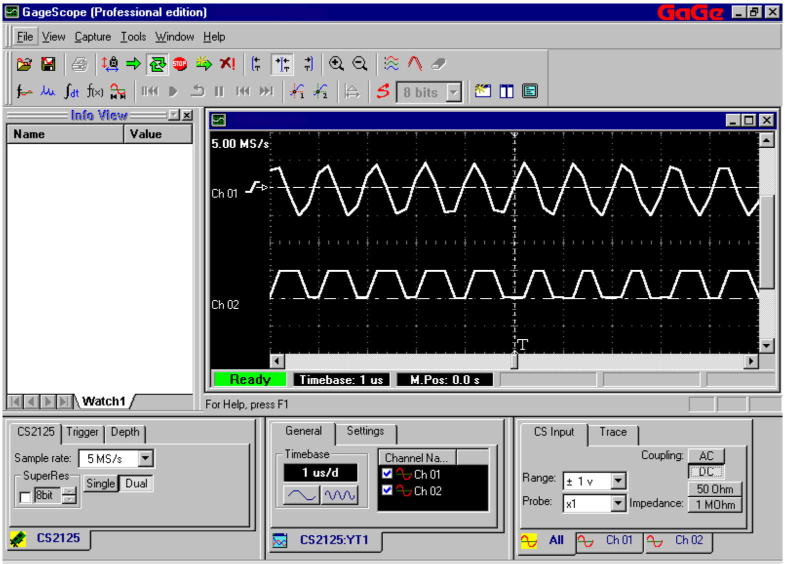
2 Click on the CS Tab in the System Control. In the case of this tutorial, the tab is titled **CS2125**.



3 Click on the Sample Rate drop-down menu to see a list of sample rates.



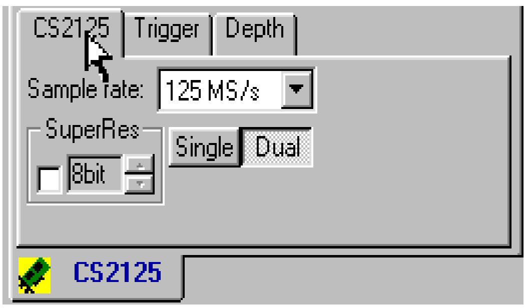
4 Scroll up the list and click on **5 MS/s** or similar small number. The low sample rate results in a signal that is not as smooth as before.



When the sample rate is low, the CompuScope card is acquiring fewer sample points per second. In other words, the time resolution of the signal decreases. With fewer points available for building a representation of the signal, the resulting waveform looks jagged.

Change the Sample Rate Back to Highest Available

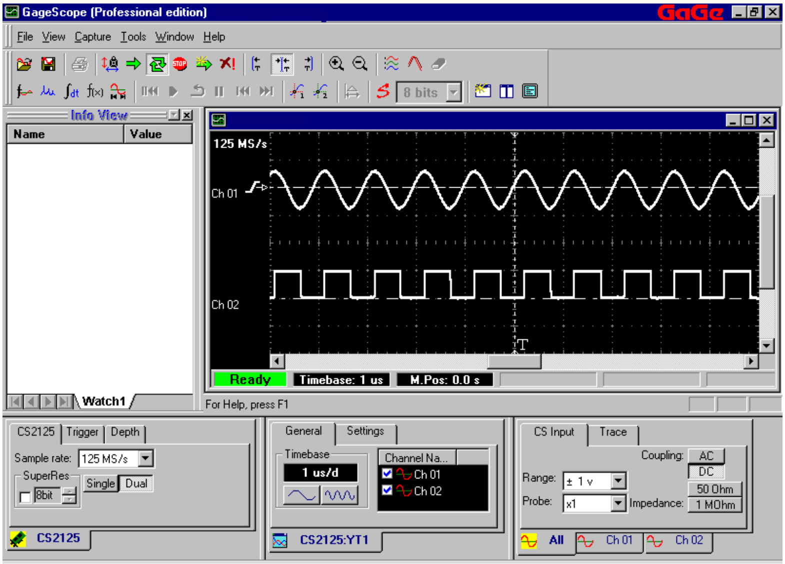
1 Click on the CS Tab in the System Control. In the case of this tutorial, the tab is titled **CS2125**.



2 Click on the Sample Rate drop-down menu to see a list of sample rates.

3 Scroll down to the bottom of the list.

4 Click on **125 MS/s** or similar high number. The higher sample rate results in many more acquisitions per second and a much smoother signal.



Note that sample rate is not linked to the timebase setting.

This has been done to allow you to take advantage of the very deep buffer memory on your CompuScopes. You can capture at a very fast sample rate and still view it with a very small timebase.

All digital oscilloscopes on the market control sample rate using the timebase you select, because they have limited memory buffers.

Hiding and Showing Channels

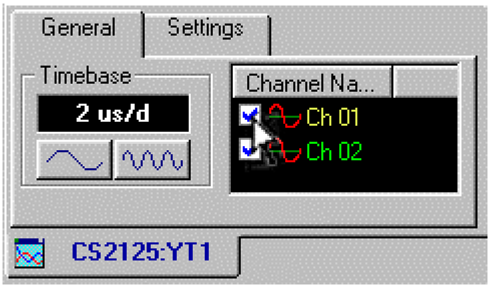
Hiding a channel temporarily removes the channel from the Display Window.

Note that you can also delete channels, but only math and file channels—you cannot delete live signals originating from a CompuScope card. In this exercise, we will merely hide the channel. The **Delete Channel** command is found in the Channel Context menu. Simply click with the right mouse button on a channel's zero line, and then click with the left mouse button on **Delete**.

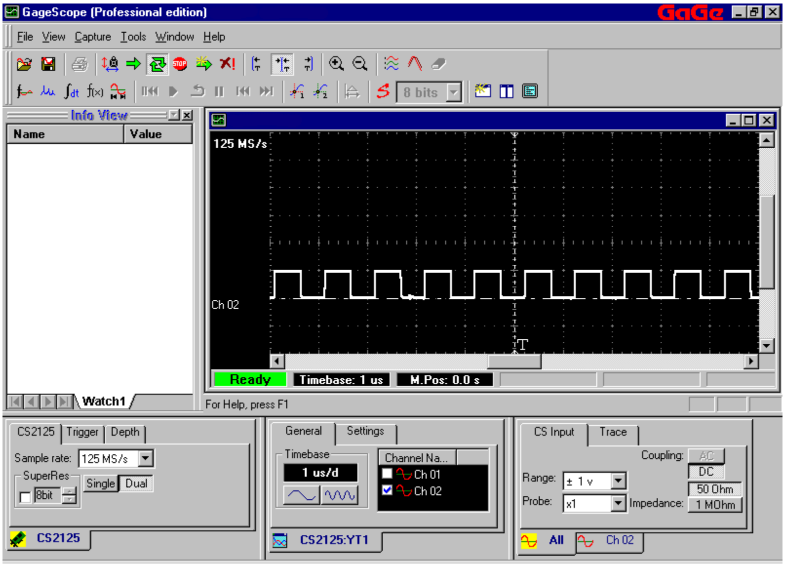
Hide Channel 1

1 In the Display Control (lower middle of the screen), click on the **General** tab.

2 Position the mouse pointer on the checkmark next to **Ch 1** in the list.



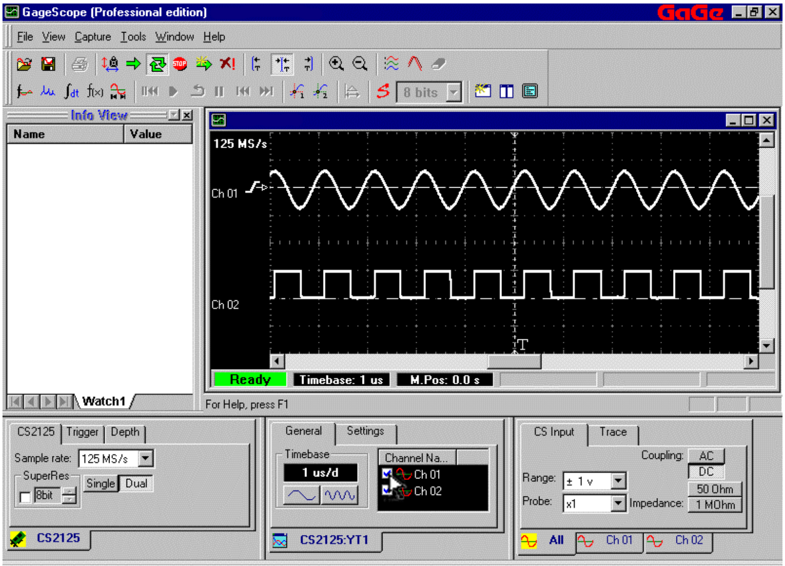
3 Click to “uncheck” it. Channel 1 should disappear from view.



Note that you are still in dual channel mode, as hiding a channel does not change the CompuScope card’s channel mode. The channel is still available, just hidden from view.

Unhide Channel 1

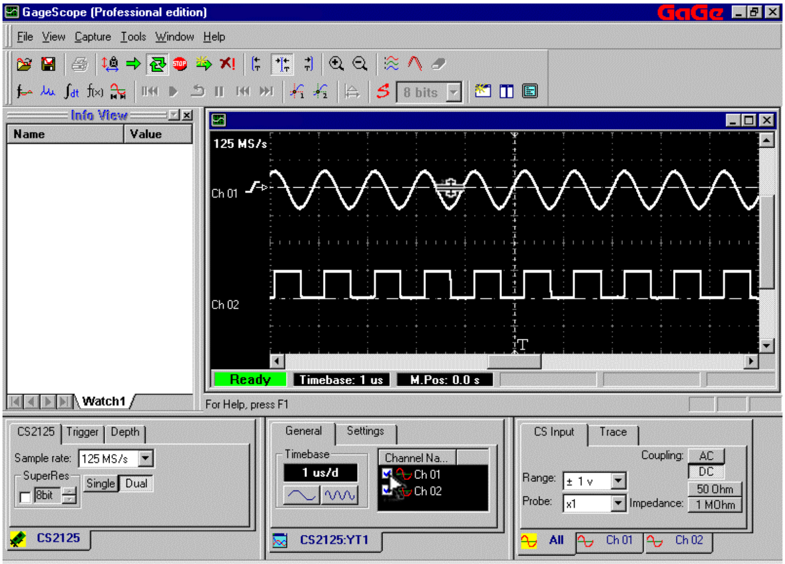
1 Click on the empty checkbox next to **Ch 1**. Channel 1 reappears.



Using the Context Menu to Hide and Show Channel 1

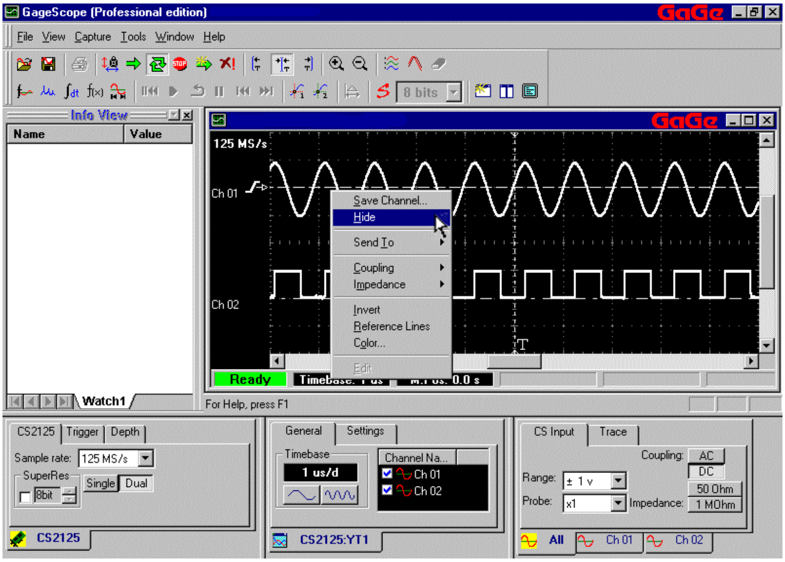
You can also hide channels directly through the Display Window.

1 Move the mouse pointer on top of channel 1. In response, channel 1 turns white to indicate it has the current focus. The cursor also changes to a “move” pointer.



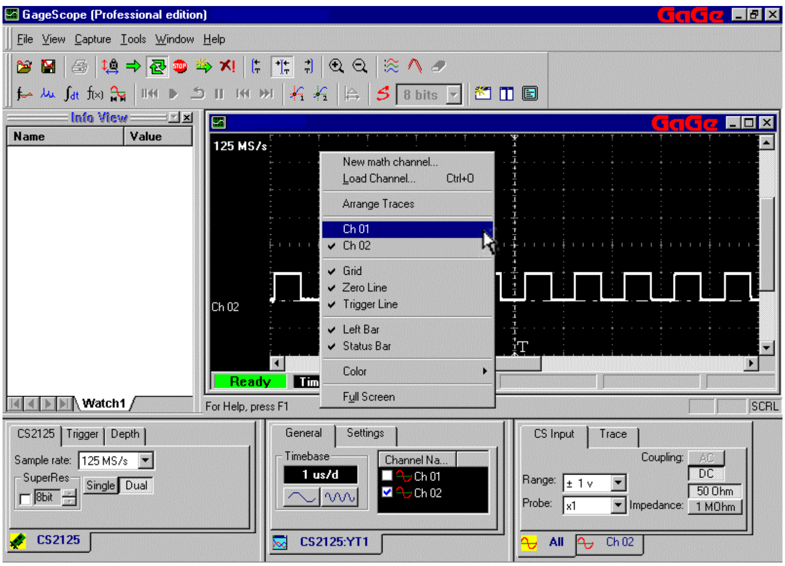
2 Click with the **right mouse button** to bring up the channel context menu.

3 Click with the **left mouse button** on **Hide**. Channel 1 becomes hidden.



4 To bring back channel 1, right-click on an empty area of the Display Window (and not on a channel) to bring up the Display Window context menu.

Notice that **Ch 1** has no checkmark, indicating that it is hidden.



5 Left-click on **Ch 1** in the context menu to bring it back.

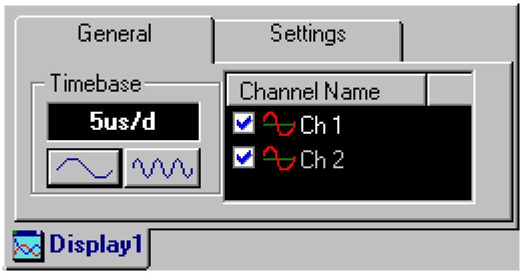
Timebase

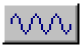
The timebase is a display function only and does affect the data being acquired. Changes to the timebase affect the display of all channels in the Display Window.

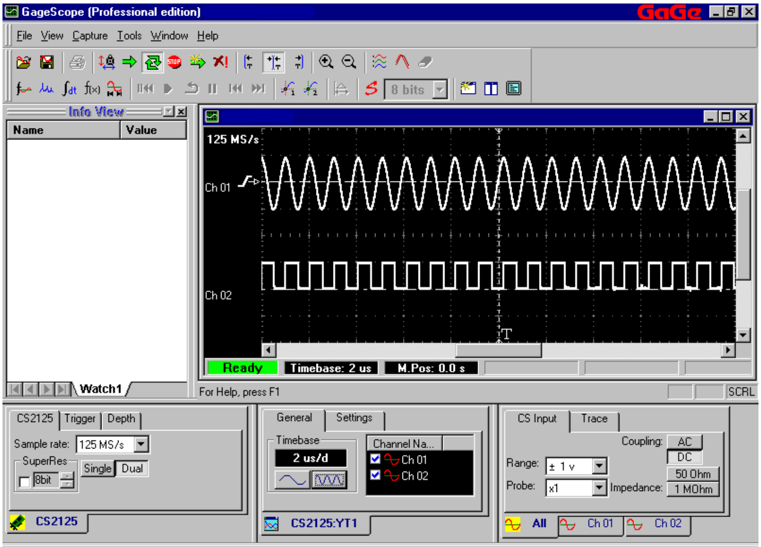
Increase the Timebase

1 If not already visible, click on the **General** tab of the Display Control, located in the lower middle of the screen.

Look for the word "Timebase." By default the timebase is set to **5us/d** (5 microseconds per division).



2 Click on  once to increase the timebase to **10us/d**.



Increasing the timebase increases the segment of time shown per division in the Display Window. Thus, when you increase the timebase, the signal peaks get closer together.

Note that your signal may look a bit different from the examples presented here.

Decrease the Timebase to 1us/d

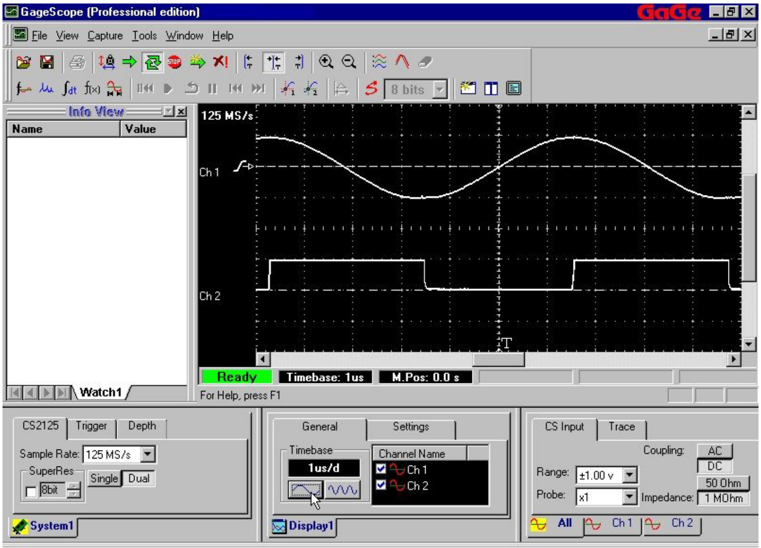
Use  to decrease the timebase.

1 If not already visible, click on the General Tab of the Display Control. The timebase is currently set to **10us/d**.

2 Click on  once to decrease the timebase to **5us/d**.

3 Keep clicking until the timebase reaches **1us/d**.

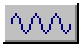
Your display should look something like this:

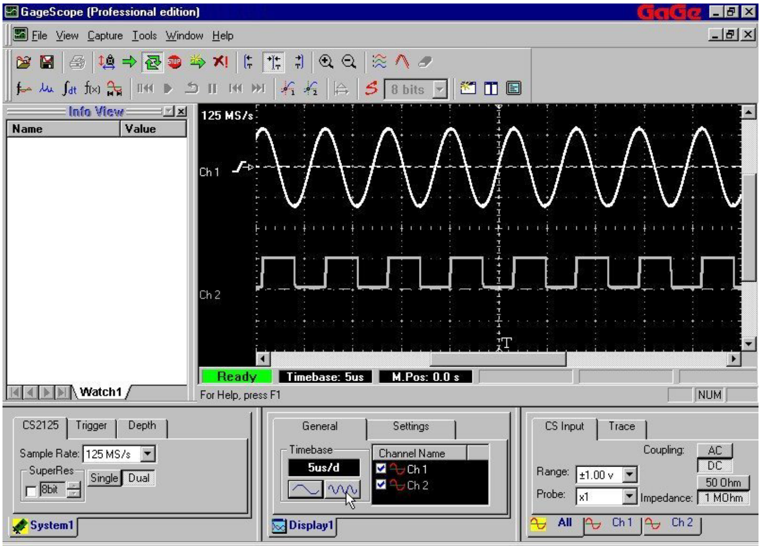


Decreasing the timebase decreases the segment of time shown per division in the Display Window. Thus, when you decrease the timebase, the signal peaks get farther apart.

Change the Timebase back to 5us/d

1 If not already visible, click on the **General** tab of the **Display Control**. The timebase should currently be 1us/d.

2 Click on  repeatedly until the timebase comes back to **5us/d**.



Note that sample rate is not linked to the timebase setting.

This has been done to allow you to take advantage of the very deep buffer memory on your CompuScopes. You can capture at a very fast sample rate and still view it with a very large timebase.

All digital oscilloscopes on the market control sample rate using the timebase you select, because they have limited memory buffers.

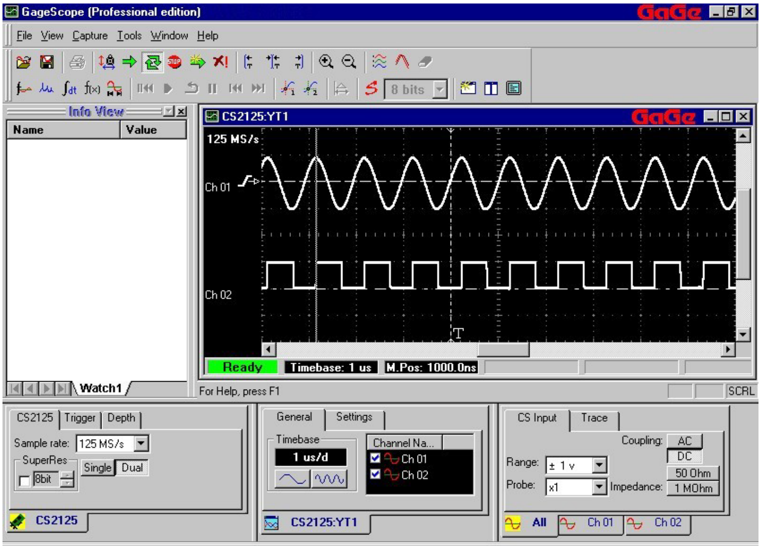
The Zoom Tools

The **Zoom In** tool magnifies the area you select, allowing for closer inspection of a particular part of a signal. The **Zoom Out** tool reduces the magnification. In the following example, you will zoom in on the second and third positive peaks of the sine wave signal.

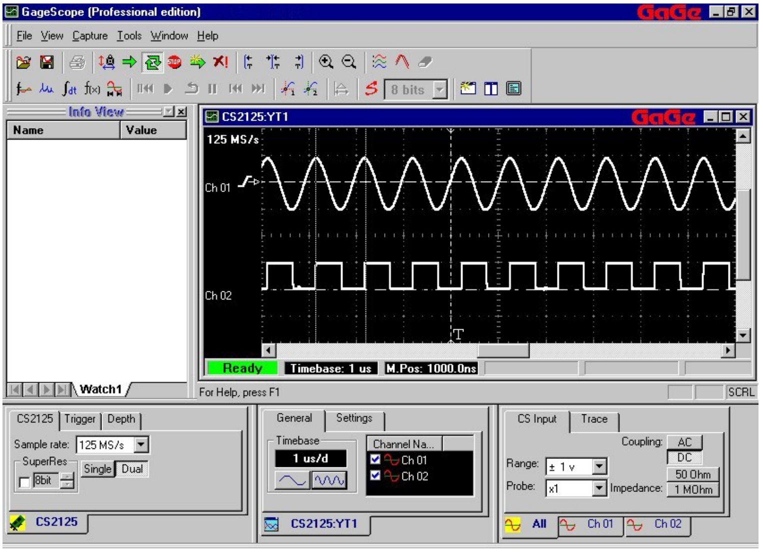
Zoom In the Second Negative Peak of the Sine Wave

1 Click on the “zoom in” icon in the toolbar.

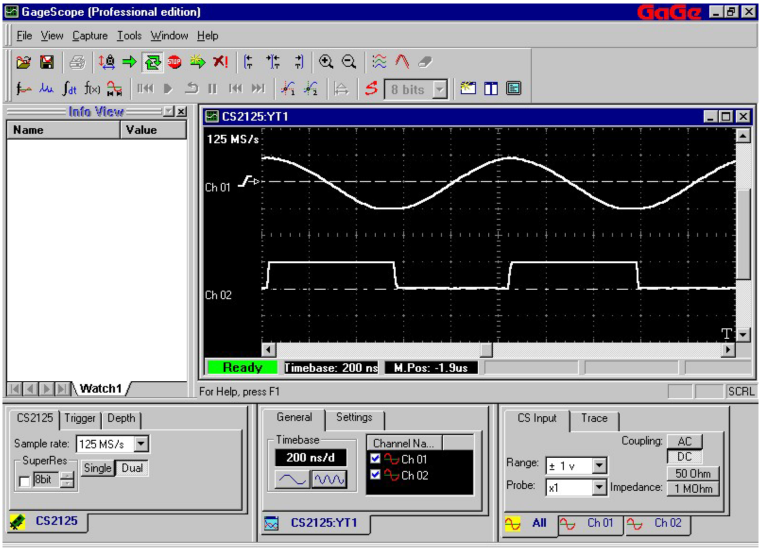
2 Press and hold down the mouse button on the second peak of the signal in the Display Window. A vertical line will appear where you first click.



3 Drag the mouse to the third peak in the Display Window. A second line will follow the movement of the mouse.



4 Let go of the mouse button. The area you selected is magnified.



When magnifying a particular area, the zoom tool actually changes the timebase based on the width of your selection, then centers the area in the Display Window.

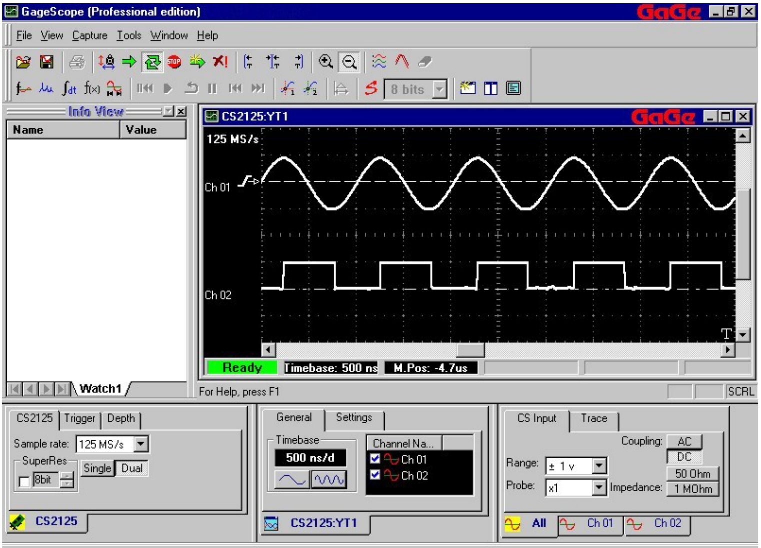
In this example, GageScope® adjusted the timebase to **200 ns/d** in order to magnify the Display Window the amount you specified.

Note that zooming changes the horizontal scale only—it does not affect the vertical scale of the channels.

Zoom Out to 5us/d

1 Click on the **Zoom Out** icon.

2 Click the mouse somewhere in the Display Window to zoom out (increase the timebase) one increment. The timebase changes to 500 ns/d.



3 Once you click with the **Zoom Out** tool, the pointer changes to an arrow again. To return to 5us/d, use the **Zoom Out** tool once more. You can also use the button.

Customizing the Display

Many Display Window elements (Grid, X Axis, Y Axis, Trigger Line, and Zero Lines) can be turned off using the Settings tab of the Display Control.

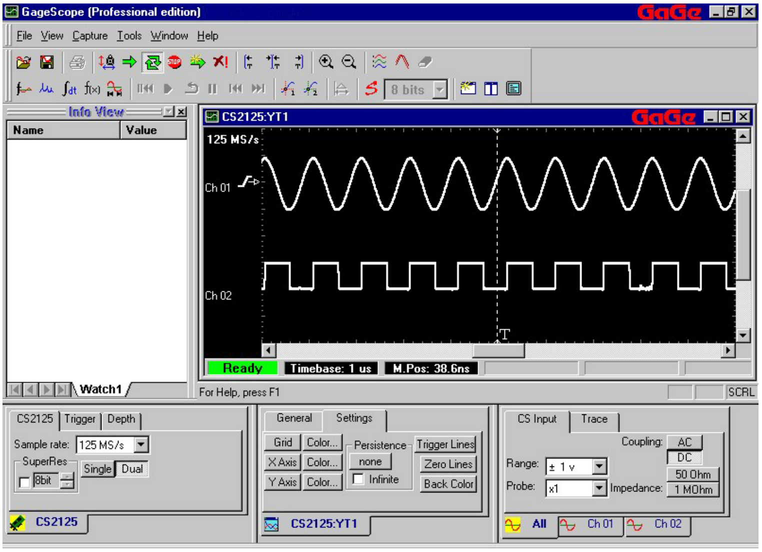
Hide All Window Elements

In the following example, you will hide all Display Window elements.

1 Click on the **Settings** tab of the **Display Control** to bring it to the front. A Display element whose button is "pressed in" is visible. All Display elements are on by default.

2 Click on the buttons **Grid**, **X Axis**, **Y Axis**, **Trigger Level**, and **Zero Lines** to turn each of these Display Window elements off.

As you click each button, notice how the Display Window changes. In the example below, all elements have been turned off.



Show All Window Elements

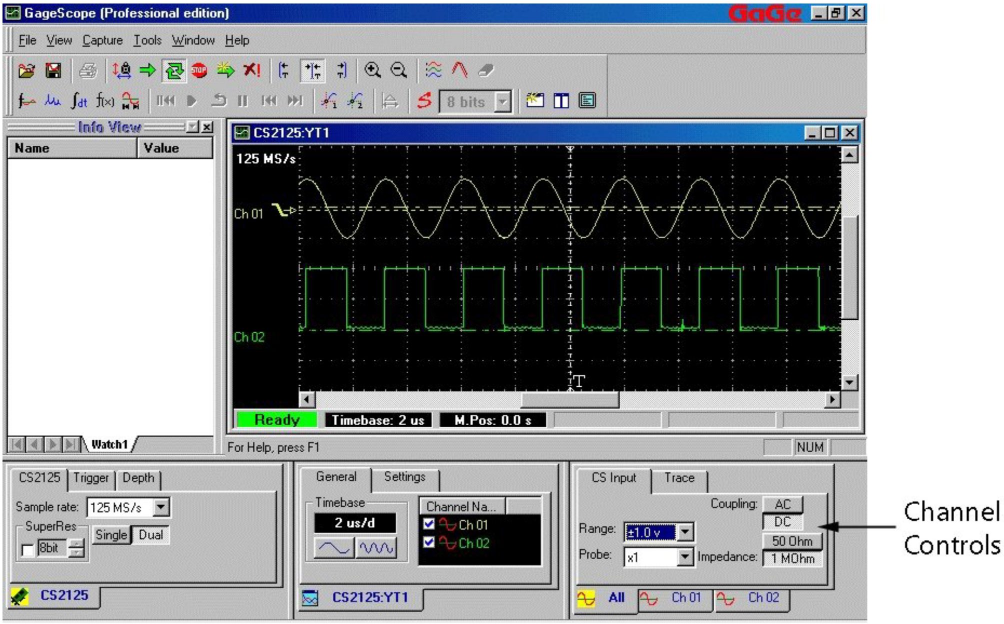
3 Click on the buttons **Grid**, **X Axis**, **Y Axis**, **Trigger Level**, and **Zero Lines**. As you click each button, the corresponding Display element reappears.

About the Other Buttons in the Settings Tab

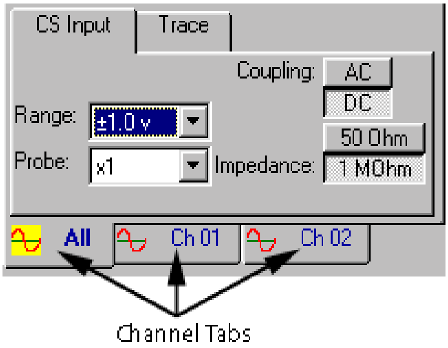
The **Color** buttons next to **Grid**, **X Axis** and **Y Axis** allow you to change the color of these Display Window elements. The **Back Color** button allows you to change the background color of the Display Window.

Tutorial 2: Working with Channels

A channel can originate from a live signal through a CompuScope card, or it can be loaded from disk. Most channel settings are changed through the Channel Control at the lower right of the screen.

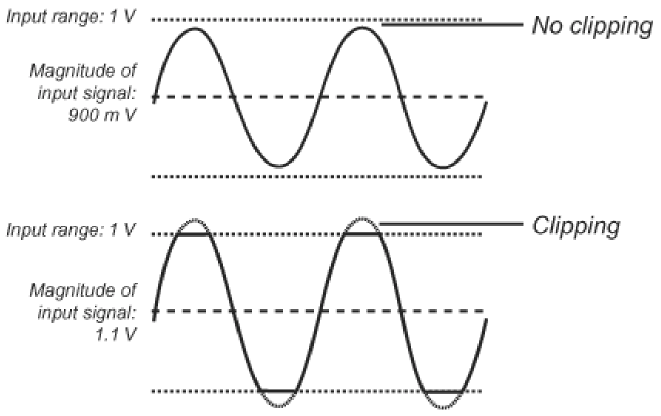


Be sure to verify that you are making changes to the proper channel by first clicking on the Channel tabs at the bottom of the Channel Control. The channel tab that is “in front” is the channel that will be affected by your changes. If the **All** tab is in front, you will be making changes to all current channels.



Input Range

The input range is the range in Volts that GageScope® will measure when acquiring data. If the input range is less than the maximum magnitude of an input signal, the peaks of the signal will be cut off.

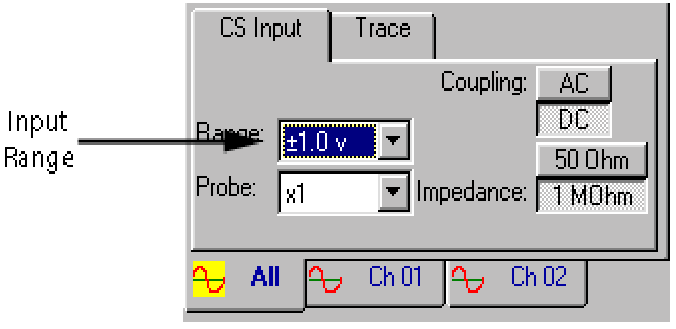


Note: You will not be able to change this setting if you are using a CompuScope 1016, as the input range for this card is fixed at ±2V.

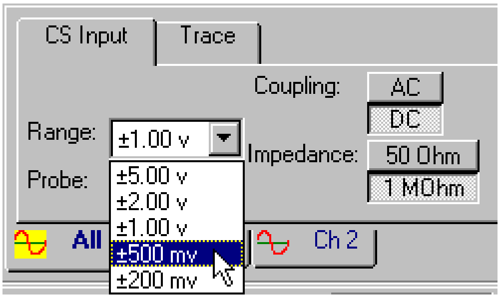
Change the Input Range of Channel 1 to ±500mV

1 Click on the **Ch 1** tab in the **Channel Control** (at the bottom of the control).

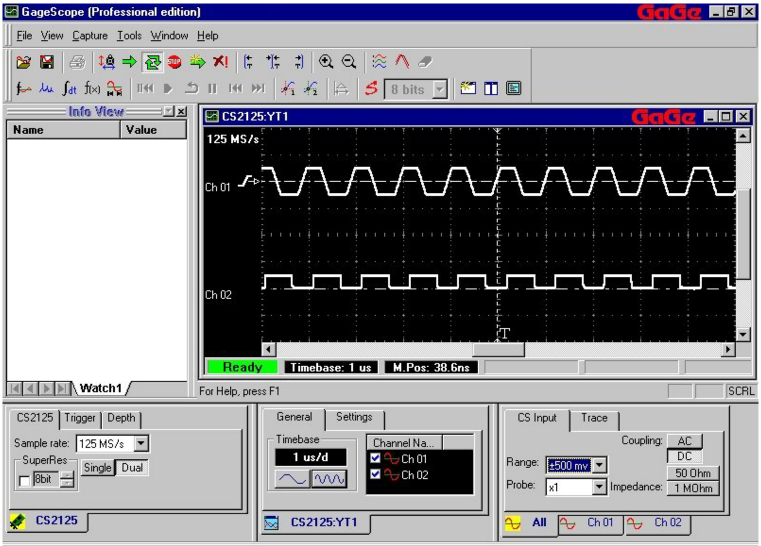
2 Click on the **CS Input** tab. By default the Input Range is set to **±1V**.



3 Click on the drop-down menu and choose **±500mV** (or a similar small number) from the list.

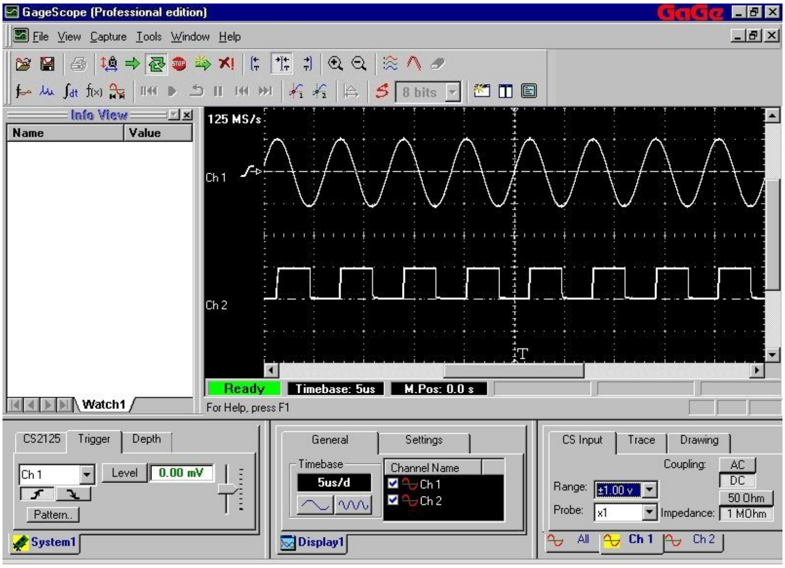


4 As ±500mV is smaller than the peaks of the signal created by the generator (which for these examples is set to about 1.8 V), a significant amount of clipping is seen—the positive and negative peaks are cut off.



Change the Input Range of Channel 1 back to ±1V

1 Click on the Input range drop-down menu and choose **±1V** from the list.



Coupling

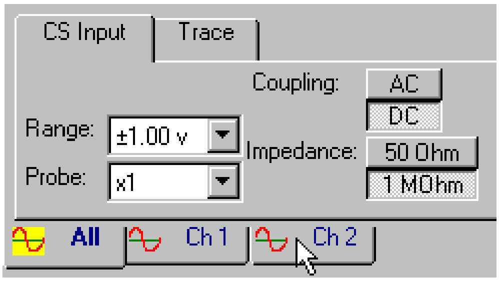
You may have noticed that, in the examples shown so far, channel 2's signal was only visible above its zero line. This is because GageScope® has automatically set all channels to DC coupling.

To see the signal centered around zero, you can change to AC coupling.

Note: You will not be able to change this setting if you are using a CompuScope 1016, as this card is DC only. Also note that only DC coupling is available when Impedance (appearing just below Coupling in the CS Input Tab of the Channel Control) is set to 50 .

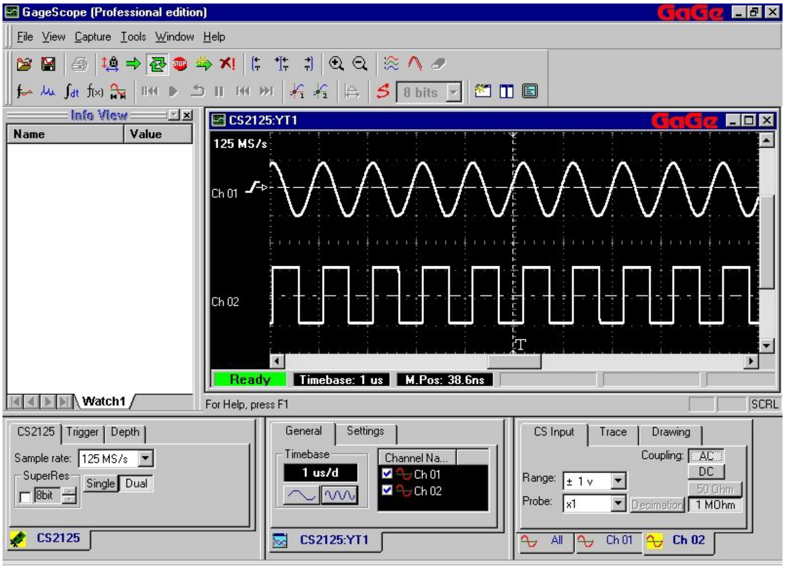
Change the Coupling of Channel 2 to AC

1 Click on the **Ch 2** tab in the Channel control.



2 Click on the **CS Input** tab. By default the Coupling is set to **DC**.

3 Click the **AC** button.



Note that Input Range is not linked to the Vertical Scale setting.

This has been done to allow you to take advantage of the high resolution offered by CompuScopes. You can capture a signal with ±1V range and still view it with a vertical scale of 50 mV/div.

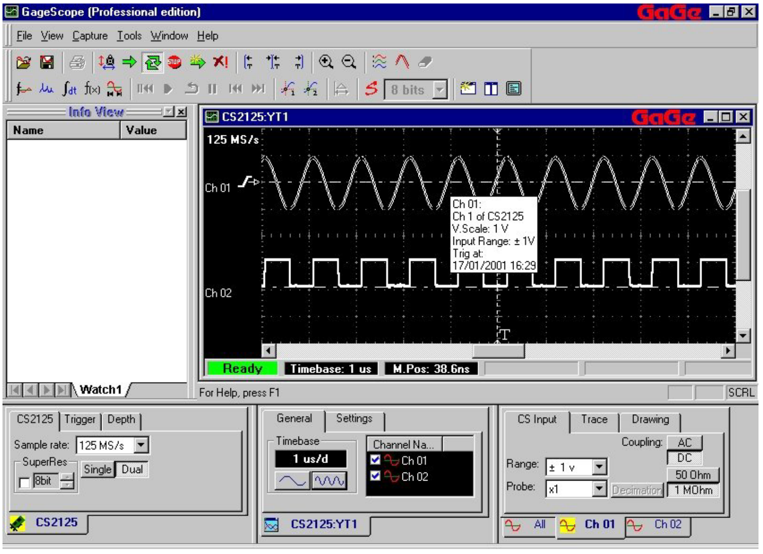
All digital oscilloscopes on the market are limited to 8 bit resolution and can digitize the signal with 256 levels only. This way, they can fit all 256 levels on screen. CompuScopes, which offer up to 16 bit resolution, digitize the signal with up to 65,536 levels. There is no screen in the world which can display that many pixels. Hence, the ability to display only a portion of the capture signal's dynamic range is necessary.

Channel Position

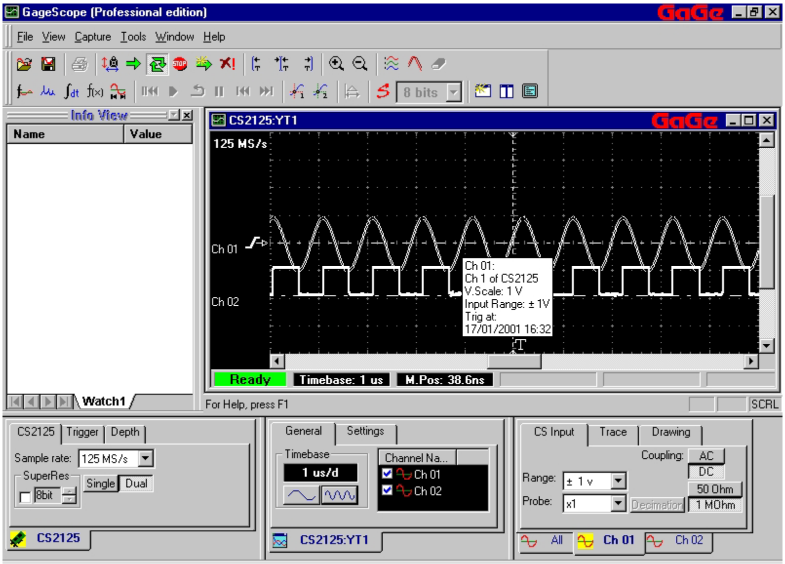
You can change the position of channels by moving them directly in the Display Window or through the position buttons in the Channel Control.

Move Channel 1 Down via the Display Window

1 Position the mouse pointer on the zero line of channel 1. The signal changes to white, and the arrow pointer changes to a drag pointer.



2 Hold down the mouse button, then drag the mouse up and down.



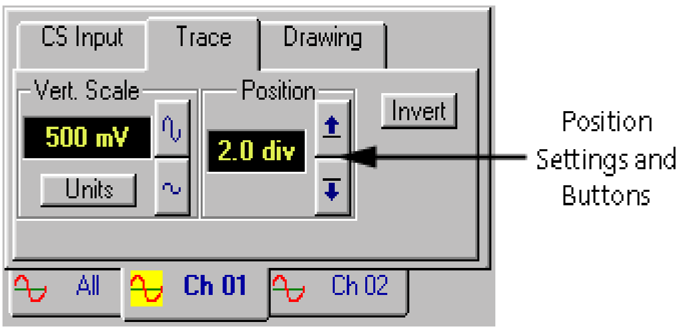
3 Before continuing to the next example, move Channel 1 to the Display Window's X Axis (in the middle of the window).

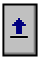
Raise Channel 1 to 1.0 div using the Position Buttons

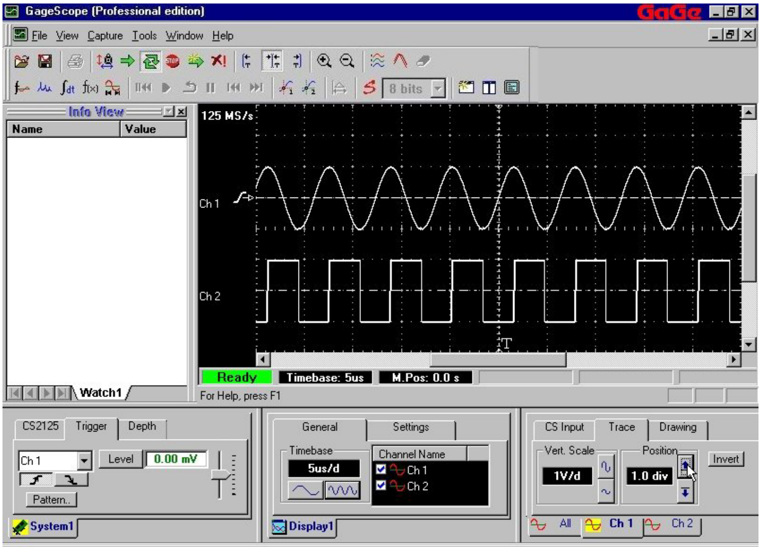
The Position Buttons are in the Trace tab of the Channel Control.

1 Click on the **Ch 1** tab in the Channel Control.

2 Click on the **Trace** tab.



3 Click on  repeatedly to raise the channel's position, until it reaches **1.0 div**.



Channel 1 in the Display Window moves as you click on the position buttons.

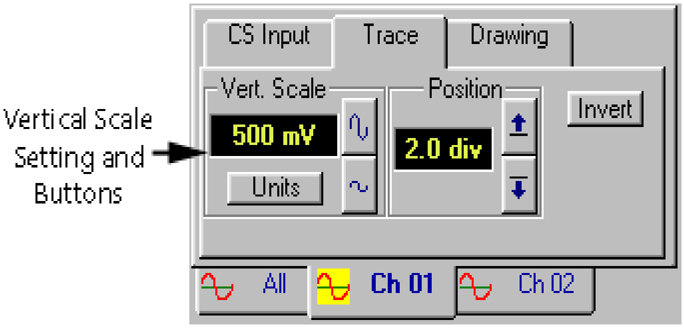
Vertical Scale

The Vertical Scale setting is located in the **Trace** tab of the Channel Control.

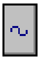
Change the Vertical Scale of Channel 1 to 5V/div

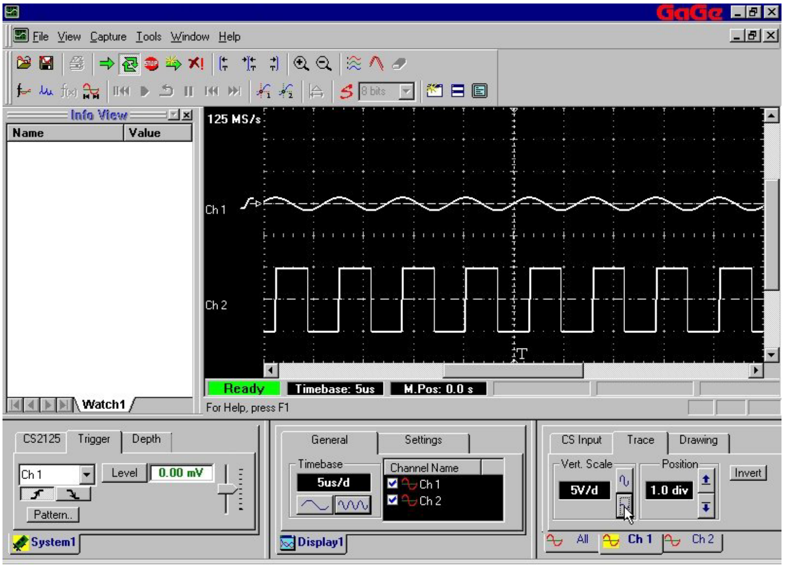
1 Click on the **Ch 1** tab in the Channel control.

2 Click on the **Trace** tab.

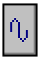


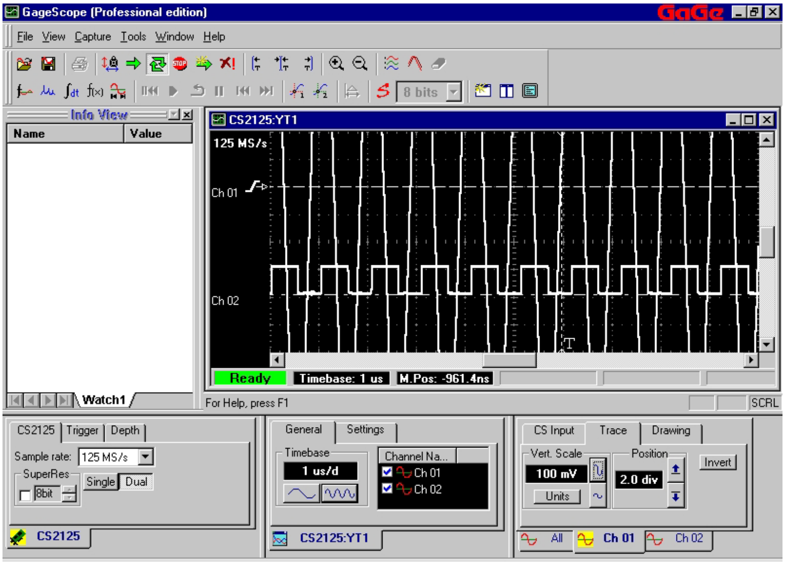
The Vertical Scale is currently set to **1V/d** (1 Volt per division). In other words, a sample point that measures 1V is displayed as 1 division high, a division being one square in the Display Window's grid.

3 Click repeatedly on  until the vertical scale is set to **5V/d**.



Reduce the Vertical Scale of Channel 1 to 100mV/div

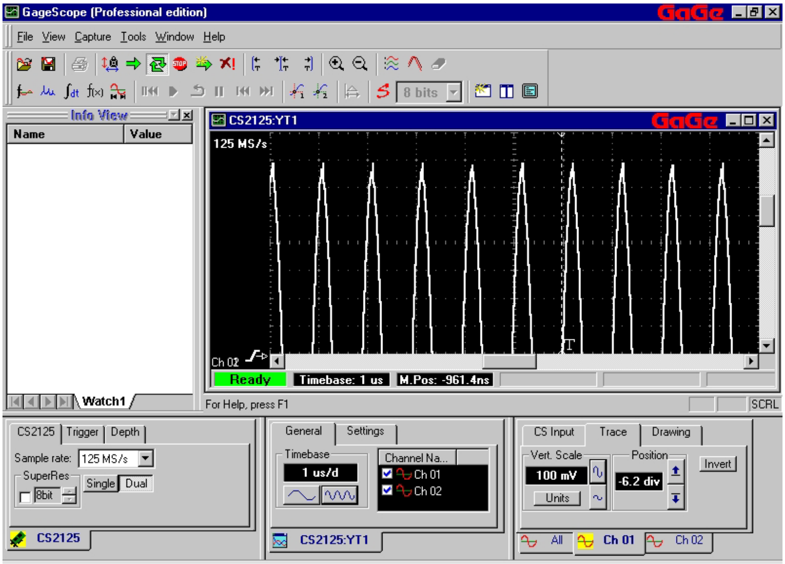
1 Click repeatedly on  until the vertical scale is set to **100mV/d**.



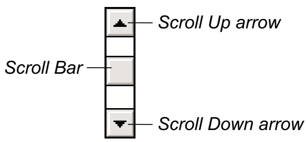
Scroll to See the Positive and Negative Peaks of Channel 1

When you increased the vertical scale to 100mV/div, the signal peaks became hidden from view. To see the peaks, you can use the vertical scroll bar.

1 Click repeatedly on **Scroll Up**  until you see the positive peaks of channel 1.



2 Alternatively, instead of clicking the up or down scroll arrow, you can drag the scroll bar itself to bring it to a new location. The Display Window is updated dynamically as you drag the scroll bar.



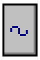
Drag the scroll bar down until you see the bottom of Channel 1.

This feature makes your GageScope®/CompuScope combo one of the most powerful oscilloscopes in the world. You are able to capture a signal with very high resolution and view a portion of it.

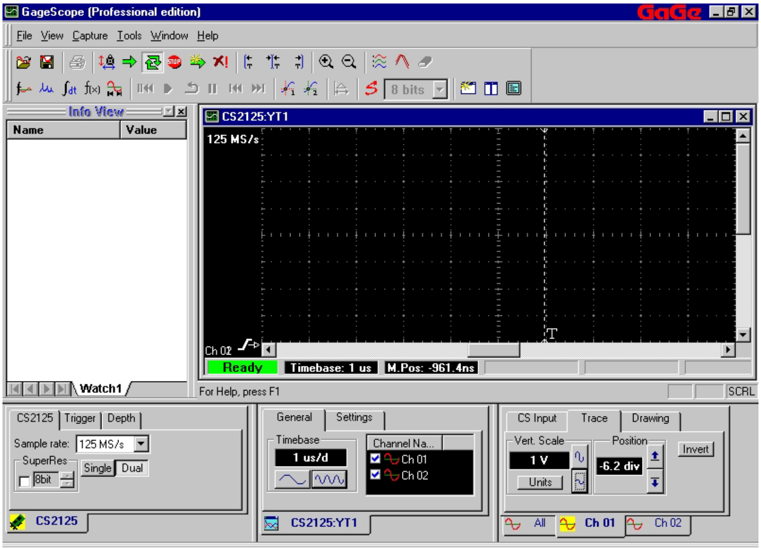
Change the Vertical Scale of Channel 1 back to 1V/div

1 Click on the **Ch 1** tab in the Channel control.

2 Click on the **Trace** tab.

3 Click repeatedly on  until the vertical scale is set to **1V/d**.

The signal may seem to disappear.

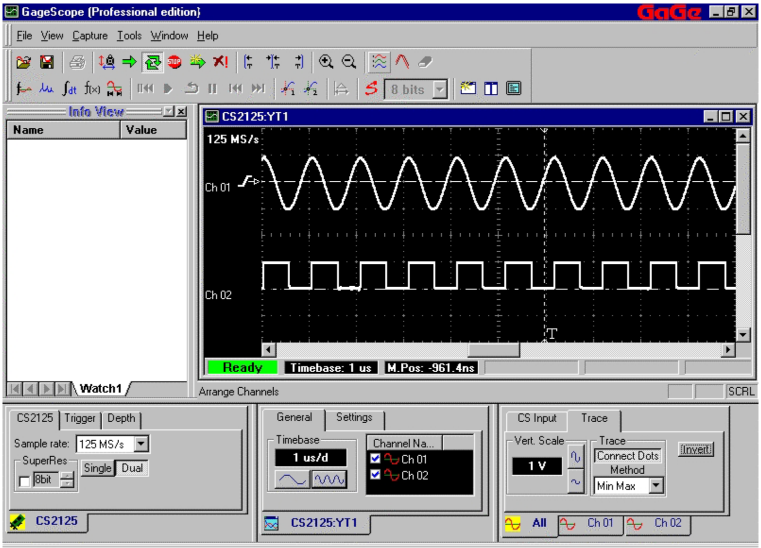


The signal is outside of the visible Display Window area. You can fix this easily with the Arrange Channels command, as demonstrated on the following page.

Re-arranging Channels Automatically

After returning the vertical scale back to 1V/div, the signals are not positioned nicely for viewing. An easy way to fix this is by using the **Arrange Channels** button, which automatically re-organizes visible channels in order of channel number.

1 Click on the **Arrange Channels** button  in the toolbar.



Note that Input Range is not linked to the Vertical Scale setting.

This has been done to allow you to take advantage of the high resolution offered by CompuScopes. You can capture a signal with ±1V range and still view it with a vertical scale of 50 mV/div.

All digital oscilloscopes on the market are limited to 8 bit resolution and can digitize the signal with 256 levels only. This way, they can fit all 256 levels on screen. CompuScopes, which offer up to 16 bit resolution, digitize the signal with up to 65,536 levels. There is no screen in the world which can display that many pixels. Hence, the ability to display only a portion of the capture signal's dynamic range is necessary.

Connect Dots

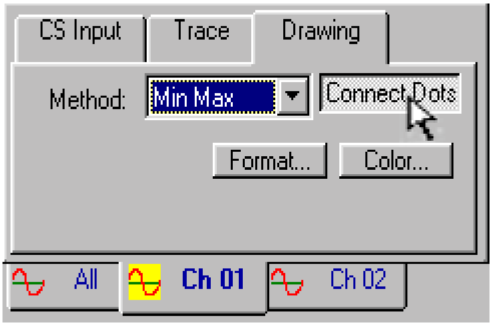
By default GageScope® draws a line between each sample point in order to display a continuous waveform. To have GageScope® display the acquired data as only points, you can toggle the **Connect Dots** button in the **Drawing** tab of the Channel Control.

Turn off the Connect Dots Setting of Channel 1

1 Click on the **Ch 1** tab in the Channel control.

2 Click on the **Drawing** tab. By default the **Connect Dots** button is on.

3 Click on **Connect Dots** to turn it off. Instead of a single line, channel 1 is displayed as points.



Turn Connect Dots Back On

1 Click on **Connect Dots** once again to turn this option back on.

Channel Color

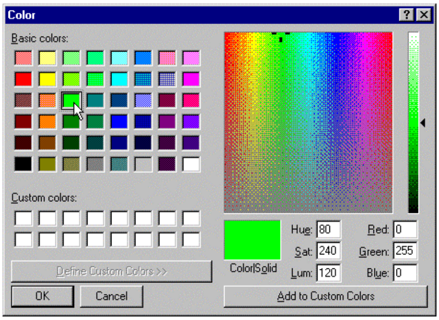
The **Color** button changes the channel color via the standard Windows color dialog.

Change the Color of Channel 1 to Green

1 Click on the **Ch 1** tab in the Channel Control (at the bottom of the control).

2 Click on the **Drawing** tab.

3 Click on the **Color** button. The standard Windows color dialog appears.



4 Click on the green color that is in the third row, third column.

5 Click the **OK** button.

Change the Color of Channel 1 back to Yellow

1 Click on the **Color** button in the Drawing tab of the Channel Control.

2 Choose the yellow that is in the second row, second column.

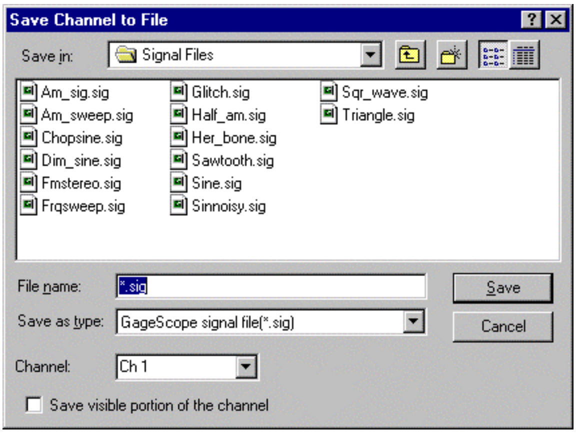
Saving and Loading Channels

GageScope® allows you to save waveforms to disk in the GageScope® SIG file format. In the following exercise, you will save channel 1 to the Signal Files folder of your GageScope® directory, then load the saved channel.

Save Channel 1 to Disk

1 Choose **Save Channel** from the File Menu, or click on **Save Channel** in the toolbar.

2 The “Save Channel to File” dialog appears.



3 At the bottom of the dialog, next to “Channel,” GageScope® gives you the option to choose which channel you want to save. In this example, **Ch 1** is listed.

If you wish, you can choose another channel to save by clicking on the Channel drop-down menu. Note that you can also save all visible channels at once by selecting **All** in the Channel list.

4 You can also specify that only the visible portion of the file be saved. To do this, click on the checkbox:

5 Enter the name **chan1.sig** in the “File name” section of the dialog box.

6 By default, GageScope® will save your files in the **Signal Files** folder. This folder contains sample signal files.

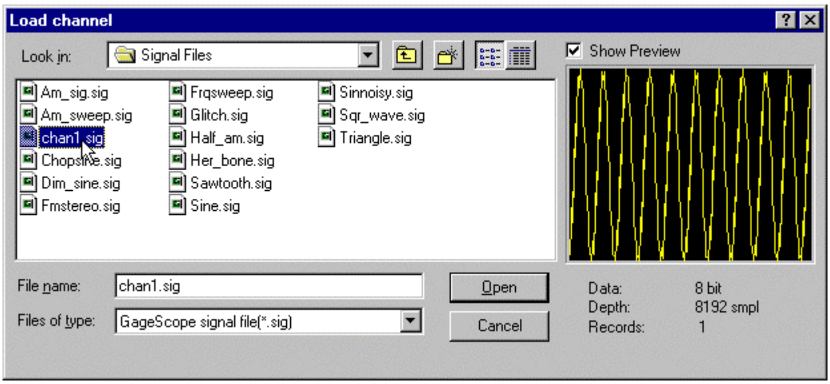
7 Click **Save**.

Load “chan1.sig” from Disk

1 Choose **Load Channel** from the File menu; or, click on **Load Channel**  in the toolbar. The Load Channel dialog appears.

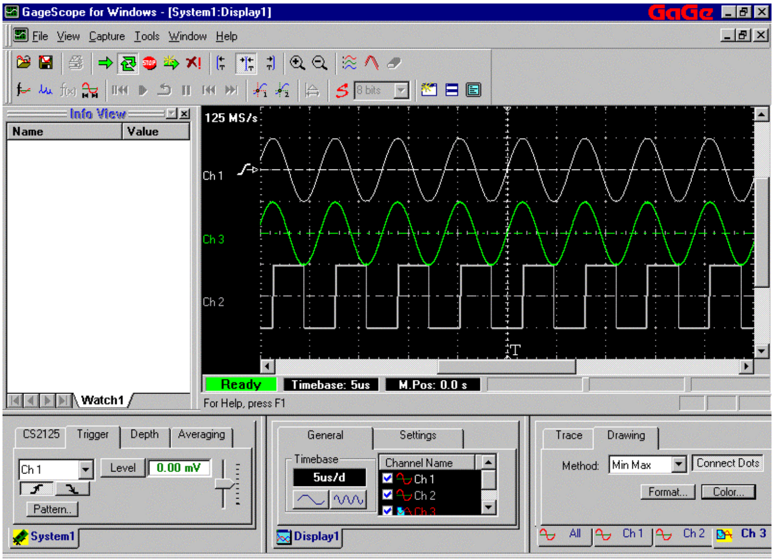
2 If you are not already in the Signal Files folder in C:\Gage\GageScope\Signal Files, move to the folder now.

3 Click on the file **chan1.sig**.



4 Click **Open**.

The channel you loaded should appear at the zero line of the Display Window, with a channel number of 3.



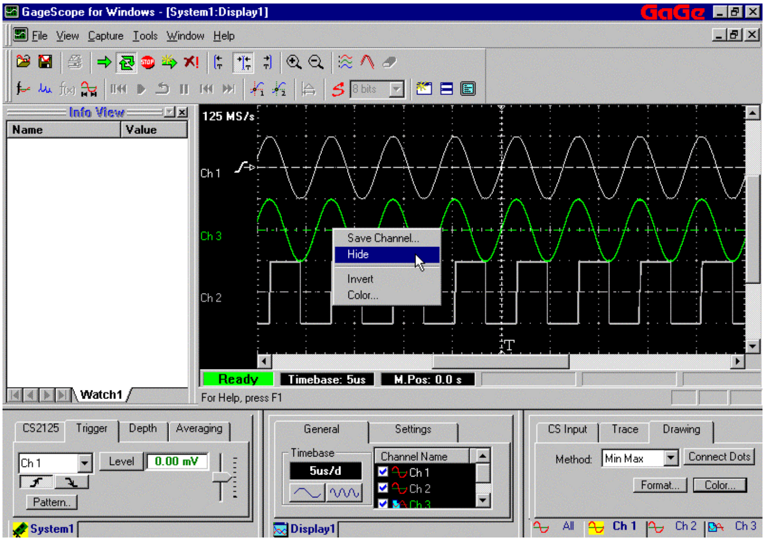
Once a channel is loaded, you can change many of its settings just like any other channel. Note, however, that this channel is not “live,” in that it is not linked to any live signal source via a CompuScope card.

Hide Channel 3

1 Position the mouse pointer on the zero line of channel 3.

2 Click with the **right mouse button** to bring up the channel context menu.

3 Choose **Hide** from the context menu.



4 Channel 3 disappears from the display window. While channel 3 remains in the list of channels in the Display Control (unchecked), it no longer appears as a tab in the Channel Control.

Note

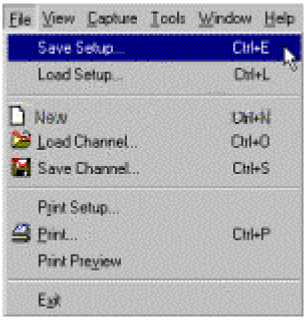
You can also delete channels, but only math and file channels—you cannot delete live signals originating from a CompuScope card. The **Delete Channel** command is found in the Channel Context menu. Simply click with the right mouse button on a channel's zero line, and then click with the left mouse button on **Delete**.

Saving and Loading Setups

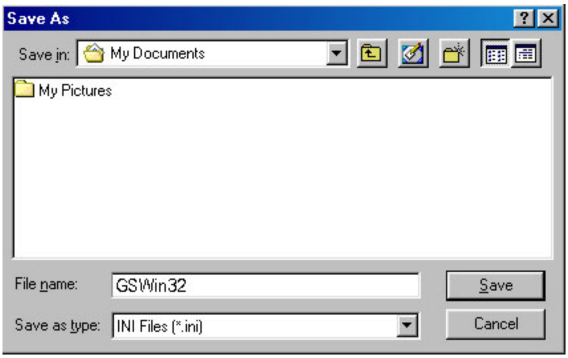
GageScope® allows you to save your current setup to so that you can load it again later. A Setup file contains all of the GageScope® settings which were current at time of saving. The Setup file has the extension .INI. In the following exercise, you will save the current GageScope® setup, make a change to the current setup, and then load the previously saved setup file.

Save the Current Setup

1 Choose **Save Setup** from the File Menu.

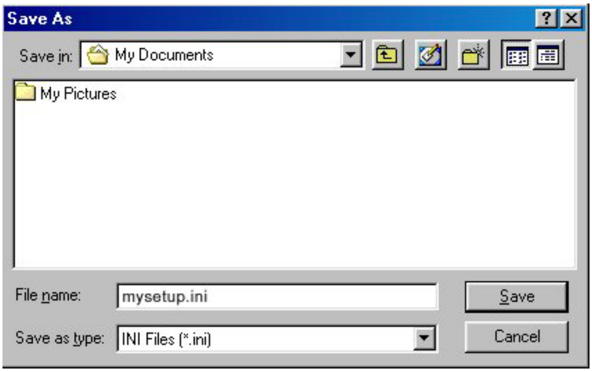


2 The Save Setup dialog appears.



The default file name is GSWin32.

3 In the “File name” field, type in **mysetup.ini**.

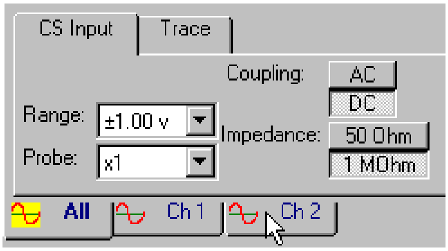


4 Go to the folder where you would like to save your Setup file.

5 Click **Save** to save the Setup file.

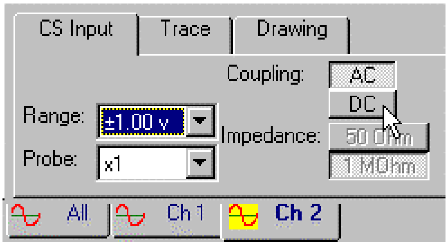
Make a Change to the Current Setup

1 Click on the **Ch 2** tab in the Channel control.

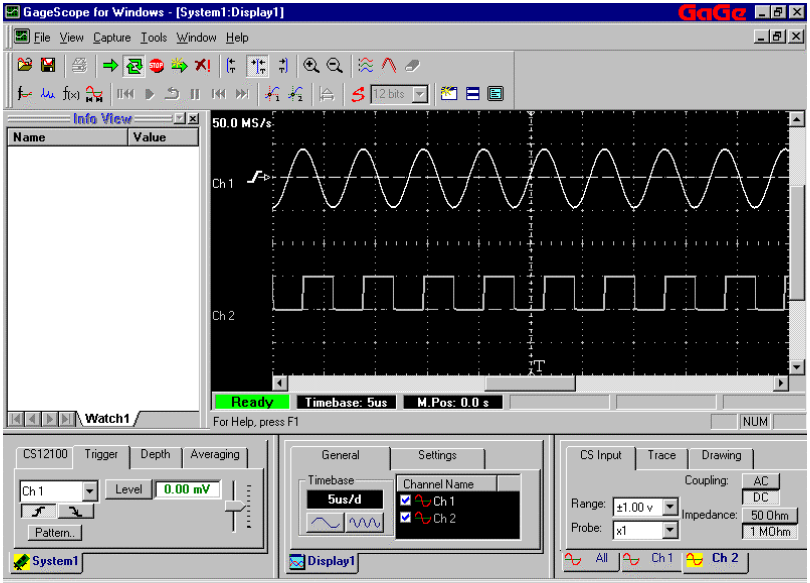


2 Click on the **CS Input** tab. The Coupling should be set to **AC** from one of the previous exercises.

3 Click the **DC** button.



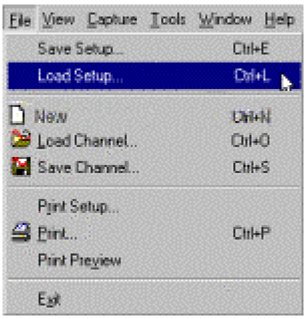
4 The coupling on Ch 2 is now DC.



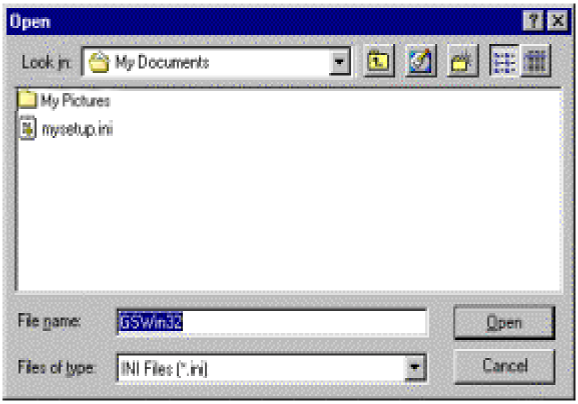
5 Click **Stop** in the Toolbar or in the Capture Menu.

Load the Previously Saved Setup File

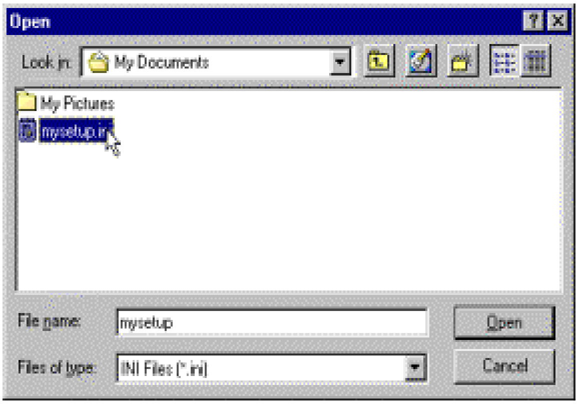
1 Choose **Load Setup** from the File menu.



2 The Load Setup dialog appears.

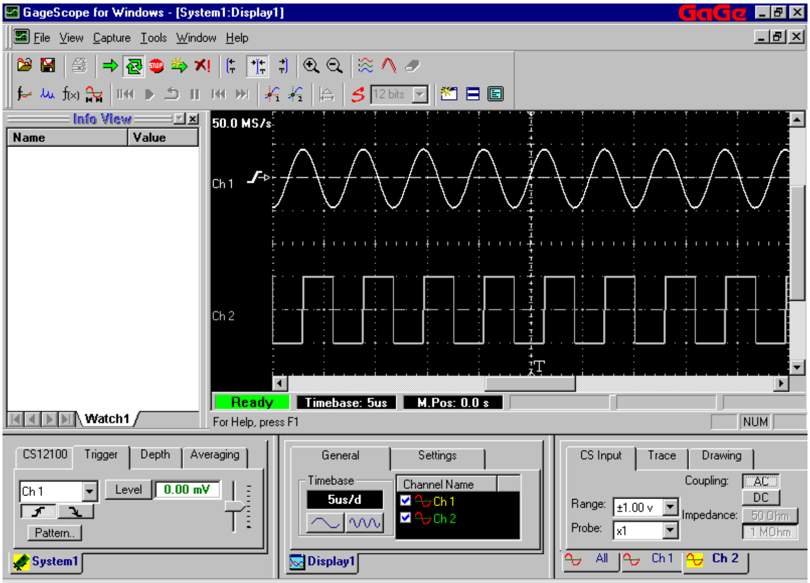


3 Go to the folder where you saved your Setup file and choose **mysetup.ini** from the list.



4 Click **Open** to load the Setup file.

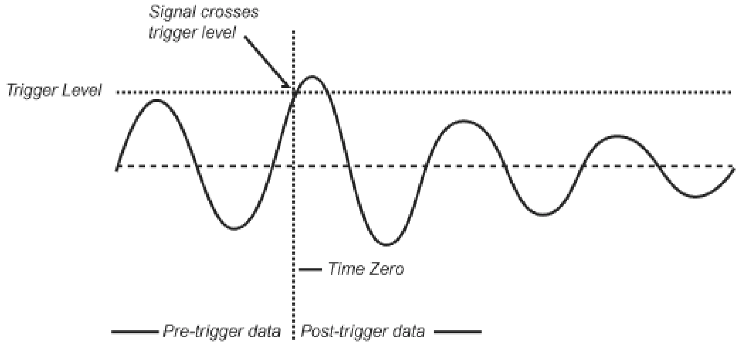
5 Click the Continuous Capture button in the Toolbar. Your previous settings are back—Channel 2’s coupling is again set to AC.



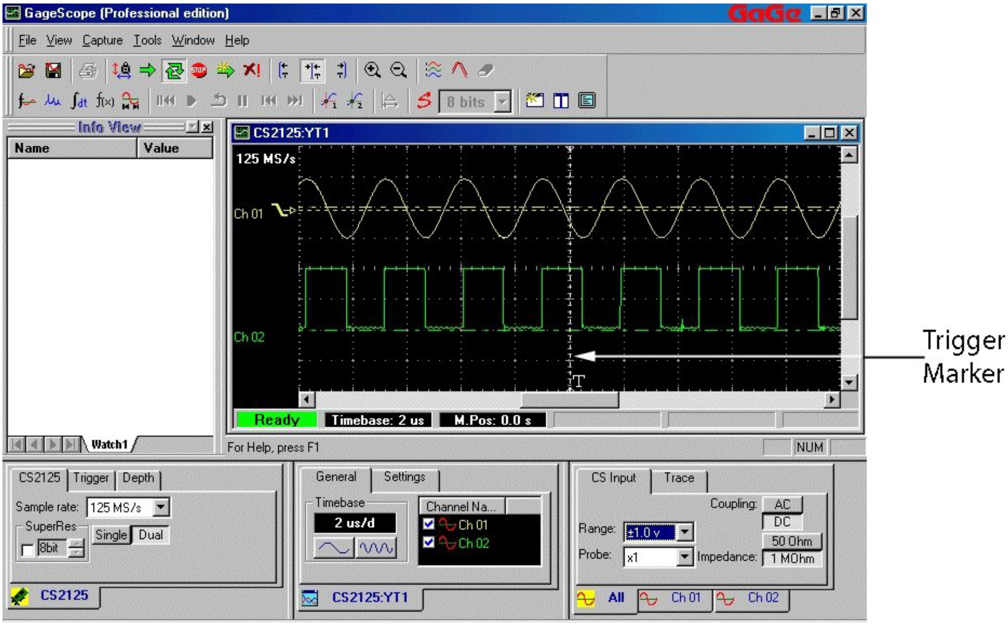
Tutorial 3: Triggering

Basic Concepts

Once GageScope® is set to Continuous or One Shot capture mode, it waits for a **trigger event**. One way for a trigger event to occur is if the voltage of an input signal crosses the **trigger level**. For example, if the trigger is set to +1V, and the signal's peaks is only at +900 mV, a trigger event does not occur. But if the signal rises to +1.1V, the trigger level will have been crossed, invoking a trigger event and subsequent data acquisition.



Once the trigger event occurs, **time zero** is established for the purposes of pre- and post-trigger data. Time zero is identified in the Display Window of GageScope® by the **trigger marker,** a vertical line with a small **T** at the bottom.



Pre- and Post-trigger Data

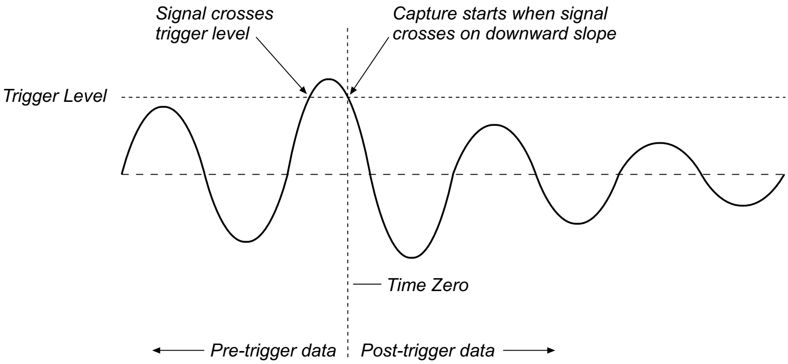
Pre-trigger data is the part of the signal acquired before the trigger event occurred, and is located to the left of the trigger marker. **Post-trigger data** is the part of the signal acquired after the trigger event occurred, and is located to the right of the trigger marker.

Once a trigger event occurs, GageScope® will ignore any new trigger events until the desired amount of signal is captured. Once the acquisition is complete, GageScope® will look for a new trigger event or stop, depending on whether the capture mode is set to Continuous or One Shot.

Trigger Events

The detection of a trigger event is affected by a variety of settings in GageScope®. For example, it is possible to change the **trigger slope**, so that the trigger occurs on the falling edge of a signal, instead of the rising edge, when the trigger level is crossed.

In the diagram below, the trigger event is occurring on the signal's falling edge.



The trigger source can also originate through the external trigger connector on the CompuScope card, if the card is so equipped, instead of an input signal.

The trigger event may not occur before a **timeout** is reached, in which case, in Continuous capture mode, GageScope® issues a **software trigger**, i.e., it triggers on its own. Or GageScope® will wait forever for a trigger to occur if timeout is disabled altogether. This useful feature allows for unattended one-shot capture.

Trigger Defaults

By default GageScope® sets up the trigger as follows:

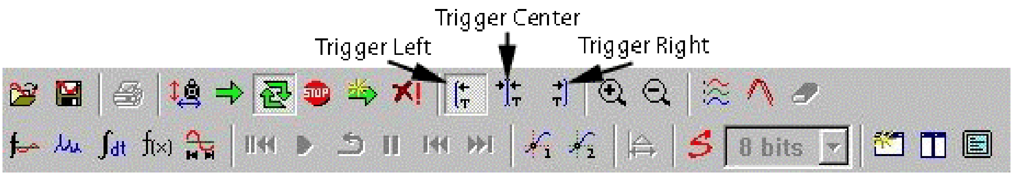
- Trigger is set to Channel 1.
- Slope is positive.
- Timeout is 10 ms.
- Trigger Level is 0 mV (zero line of channel 1).
- Pre-trigger depth is on and set to 4096 Samples.
- Post-trigger depth is set to 4096 Samples.

Setting up for the Examples

In order to continue this tutorial, make sure you have at least one input signal originating from a signal generator connected to channel A of your CompuScope card. For the examples used here, a 200 kHz sine wave signal with an amplitude of approximately 1.8V was connected to channel A of a CompuScope 2125.

Trigger Marker Align

The Trigger Align buttons allow you to quickly position the trigger marker to the left, center, or right of the display window.



The trigger marker indicates where the trigger occurred in the data acquisition. The data to the left of the “T” is pre-trigger data, and data to the right of the “T” is post-trigger data.

By default GageScope® centers the trigger marker in the Display Window.

Using the Trigger Align Buttons

1. Click the **Trigger Left Align** button  in the toolbar to position the trigger to the left of the display.

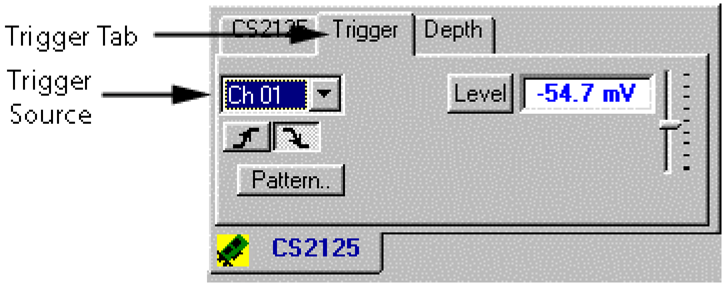
2. Click the **Trigger Right Align** button  in the toolbar to position the trigger to the right of the display.

3. Click the **Trigger Center Align** button  in the toolbar to position the trigger in the center of the display.

   These buttons have the same effect as scrolling toward the trigger using the horizontal scroll bar.

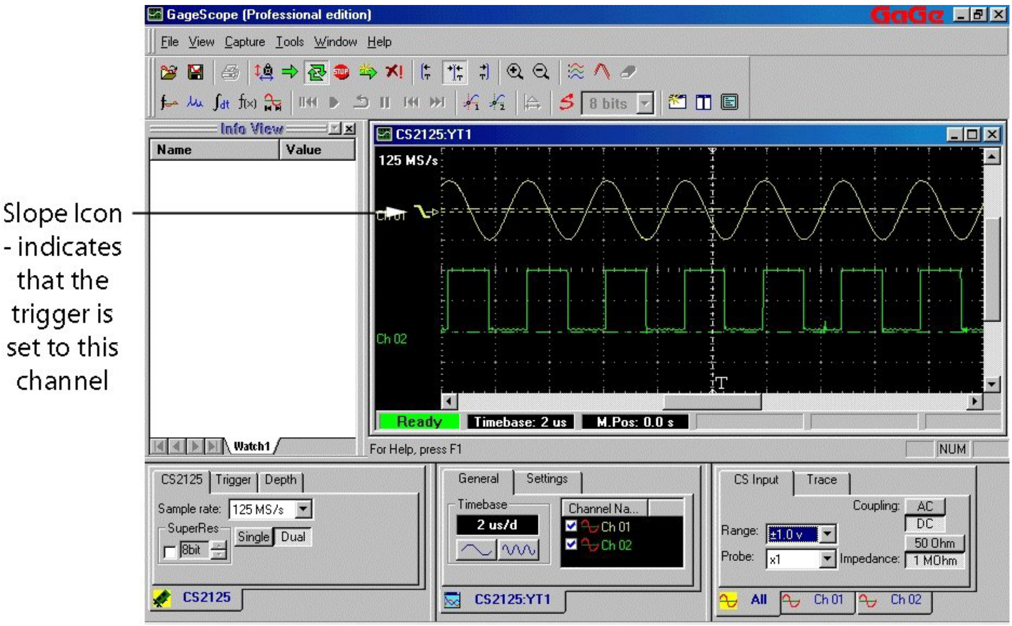
Trigger Source

The trigger source is located in the **Trigger** tab of the System Control.



The trigger source options are input signals (“Ch 1” and “Ch 2,” etc.), “External,” and “Disabled.” By default, the trigger source is set to **Ch 1** (input channel 1, which corresponds to channel A of the CompuScope card). Note that on the CompuScope 1016, the trigger source options are External or Disabled only.

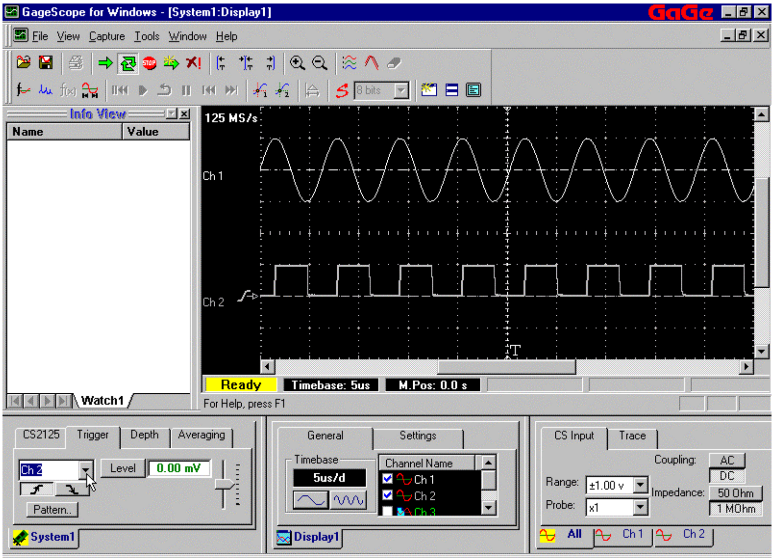
This is also indicated by the slope icon to the left of channel 1, as shown below.



Changing the Trigger Source

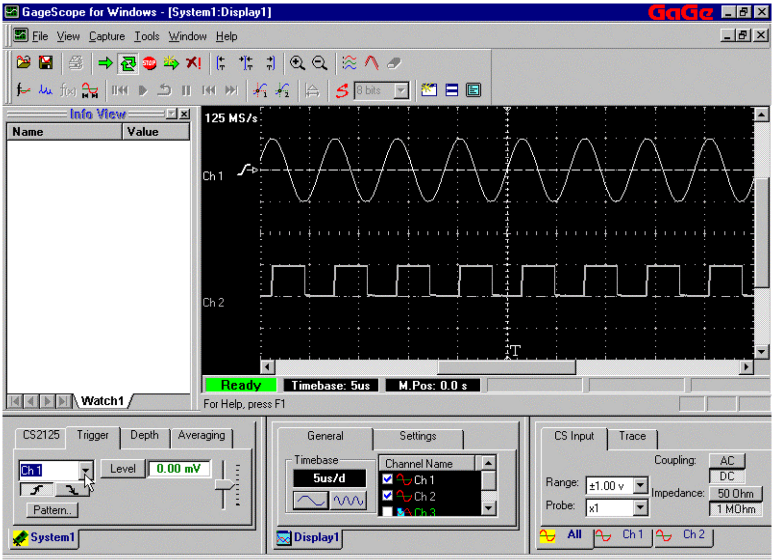
1 Click on the **Trigger** tab.

2 Choose **Ch 2** from the Trigger Source drop-down list. Notice that the slope icon in the Display Window has moved next to channel 2.



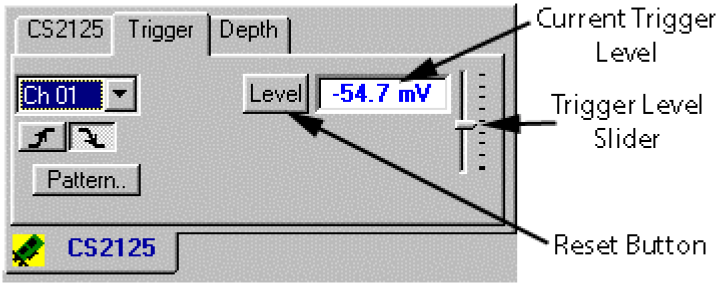
3 Click on the trigger list once more.

4 Click on **Ch 1**. The slope icon has moved back to channel 1.



Trigger Level

By default the trigger level is set to 0mV (the zero line of channel 1).

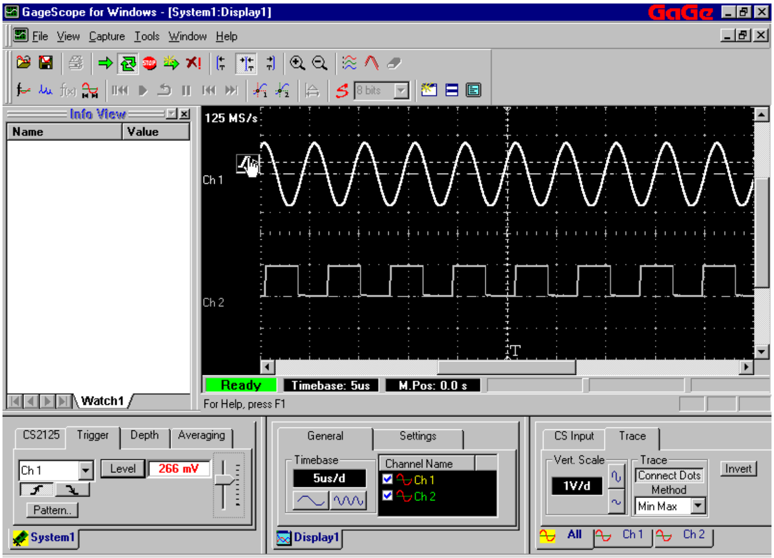


Change the Trigger Level by Dragging the Slope Icon

1 Position the mouse pointer over the slope icon at the far left of the signal.

2 Hold down the left mouse button.

3 Drag the mouse up or down to change the trigger level.



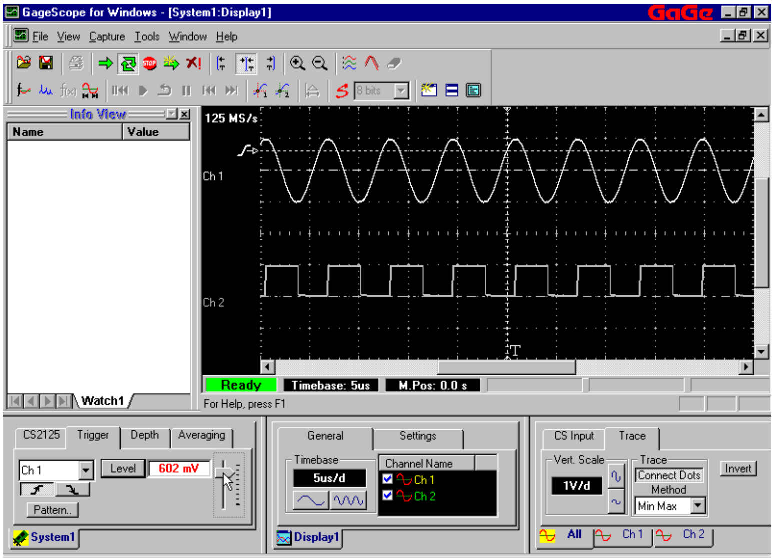
Note that if you raise or lower the trigger level so that it is higher or lower than the signal peaks, trigger events will stop occurring. This is because the signal voltage is not reaching the trigger level in order to start a capture.

Also note that the changes you make to the trigger level are reflected in the trigger level field in Trigger tab of the System Control (lower left of the screen).

Change the Trigger Level by Dragging the Slider

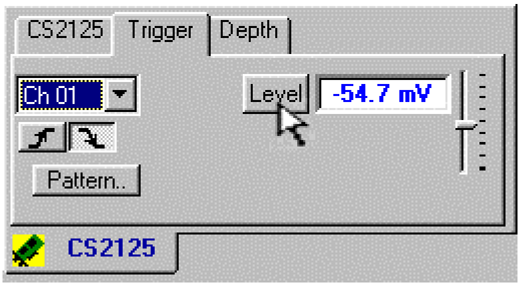
1 Click on the **Trigger** tab in the **System Control**.

2 Using the trigger level slider, raise the trigger level to approximately 600 mV.



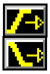
➧ Reset the Trigger Level to Zero

1 Click on the **Level** button to quickly reset the trigger level to 0.



Trigger Slope

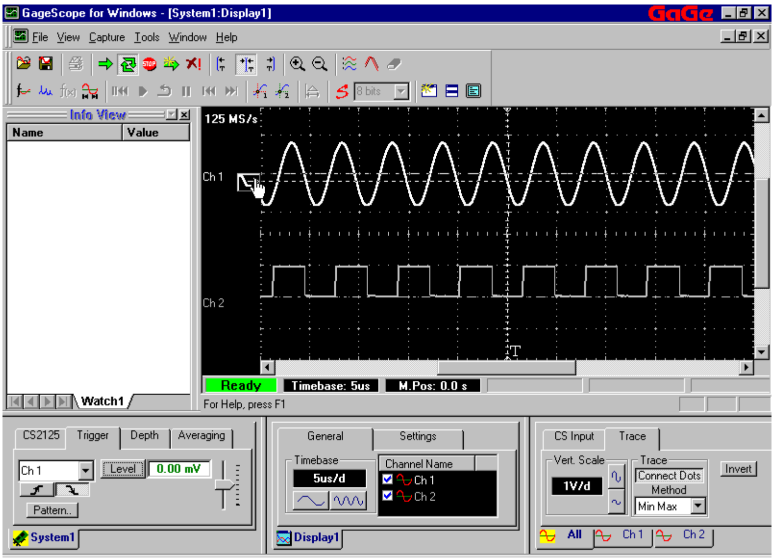
The slope icon in the Display Window reflects the current trigger slope.



indicates positive slope, i.e. the trigger occurs on the signal's rising edge.
indicates negative slope, i.e. the trigger occurs on the signal's falling edge.

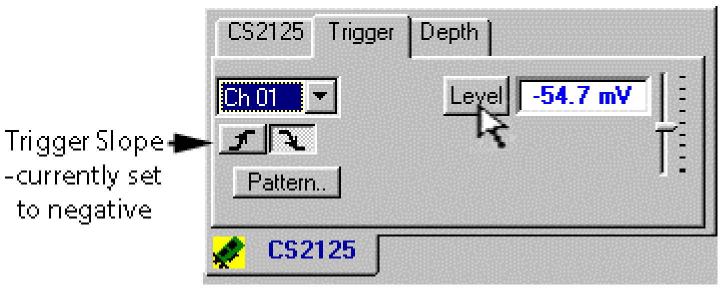
Change the Trigger Slope

1 Double-click on the slope icon in the Display Window to change the slope of the trigger. In response, the slope icon changes to a negative slope.



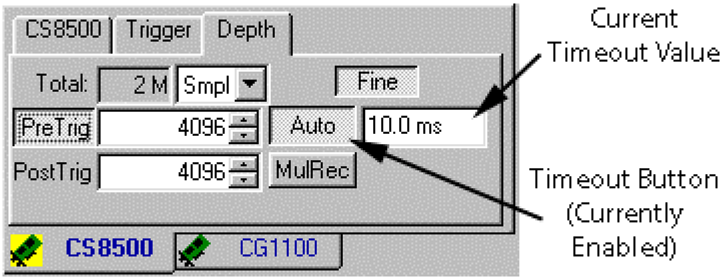
2 Double-click on the slope icon once again to change the slope back to positive.

The slope is also available through the Trigger tab.



Trigger Timeout

The timeout setting is the amount of time GageScope® waits for a trigger event. As there is one timeout setting for all available triggers, timeout is located in the Depth tab. The default is **10 ms**.



You can completely disable timeout by clicking on the **Auto** button. If you click on the **One Shot** capture button, timeout is automatically disabled.

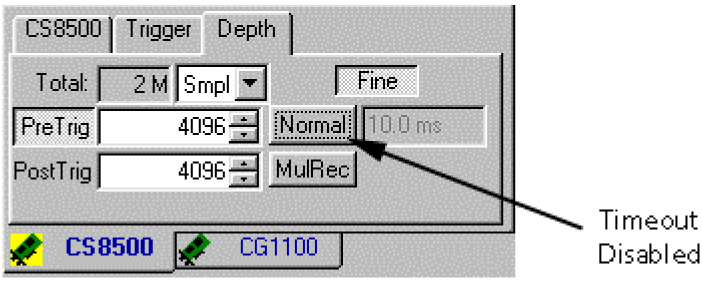
In the following exercise, you will disable the timeout, then prevent a trigger event from occurring.

Disabling the Timeout

1 Make sure you are in continuous mode (click on the **Continuous** mode button).

2 Click on the **Depth** tab in the System Control.

3 Click on the **Auto** button to disable it (it should now read **Normal** and not be "pressed in").



4 Click on the **Trigger** tab in the System Control.

5 Make sure the Trigger is set to **Ch 1**.

6 Click on the **Level** button to reset the Trigger level to 0.

7 The Status bar at the bottom left of the Display Window should state “Ready” on a green background. This indicates a trigger event is occurring properly.

Absence of a Trigger Event with Timeout Disabled

1 Raise the trigger level to higher than Channel 1’s peaks. Remember you can drag the trigger level (the slope icon) in the Display Window.

By moving the trigger level above the signal, you are preventing a trigger event from occurring.

2 The Status bar at the bottom left of the Display Window should now state “Waiting” on a magenta background. This indicates that GageScope® is waiting for a trigger event to occur.

Since you disabled the timeout, GageScope® will wait forever for a trigger event to occur.

3 Drag the trigger level back down to cause a trigger event and subsequent capture. The status bar should report “Ready” once more.

➧ Setting a Timeout of 5 Seconds

In the following exercise, you will set the timeout to five seconds, then prevent a trigger event from occurring. A value of five seconds is used so that you may have a bit of lead time before GageScope® issues a timeout.

1 Make sure you are in **Continuous** mode and that the trigger is set to channel 1.

2 Click on the **Depth** tab in the System Control.

3 Click on the **Timeout** button to enable it (it should be pressed in).

4 Click in the **Timeout** field.

5 Press **Backspace** or **Delete** to erase what’s in the field.

6 Type in **5 s** and press **Enter**. Be sure to use a lowercase ‘s.’

The letter **‘s’** stands for “seconds.” Other abbreviations, such as **ms** for milliseconds, and **us** for microseconds, are also valid units.

Five seconds is an extremely long time for data acquisition purposes, and in this case is used only for demonstration.

7 Click on the **Trigger** tab in the System Control.

8 Click on the **Level** button to reset the Trigger level.

9 The Status bar at the bottom left of the Display Window should state “Ready” on a green background. This means a trigger is occurring properly.

Absence of a Trigger Event with a Timeout of 5 Seconds

1 Raise the trigger level so that it is higher than Channel 1’s positive peaks.

2 The Status bar at the bottom left of the Display Window now states “Waiting” with a magenta background.

This indicates that GageScope® is waiting for a trigger event.

3 If you watch closely, a “Ready” message with a yellow background should appear in the status bar every 5 seconds.

4 Once a timeout is issued, GageScope® issues a software trigger (it triggers on its own) and performs a capture. It then re-arms the card for another trigger event—the status bar states “Waiting” again. Note that this is only true for continuous mode.

Change to One Shot Mode

1 Click on the **One Shot** icon. You may very briefly see “Stop” on the status bar with a blue background.

Thereafter, the status bar should report “Waiting” with a magenta background.

When you change to **One Shot** mode, GageScope® automatically disables the timeout. With timeout disabled, GageScope® will wait forever until a trigger event occurs (or until you press the **Stop** button). This allows for unattended one-shot capture.

A Trigger Event Exercise

In the following exercise, you will use your signal generator to create a trigger event. The data will be captured by GageScope® in One Shot mode.

If you have a signal generator that has a “one shot” button, you can use it to create the trigger event. If your generator does not have a “one shot” button, instructions are provided for simulating a trigger by removing and reconnecting the cables attached to your generator.

1 Click on the **Trigger Center Align** button.

2 Reset the trigger level to 0 (click the **Level** button in the **Trigger** tab).

3 Switch to **Continuous** mode.

4 • If your generator does not have a “one shot” button, unplug the cable from the signal generator that is connected to Channel A of the CompuScope card. Your sine wave should turn to a flat line.

5 Raise the trigger level of channel 1 to above channel 1’s zero line.

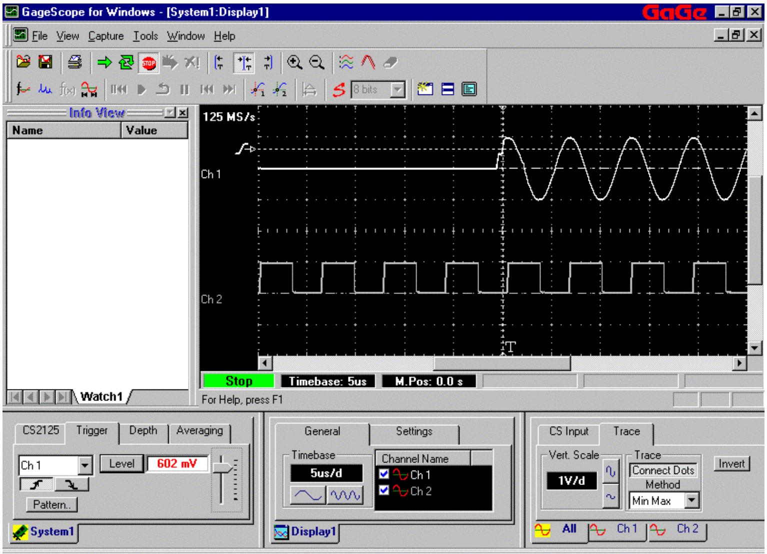
6 Set up a one shot capture by clicking on the **One Shot** button in the toolbar. Timeout will automatically be disabled, and the Status Bar should state “Waiting.”

7 • If your generator has a “one shot” button, press it now.

• Otherwise, plug the cable back into the signal generator as cleanly and quickly as possible.

8 The resulting pre-trigger signal should amount to a mostly flat line with some activity near the Trigger Marker.

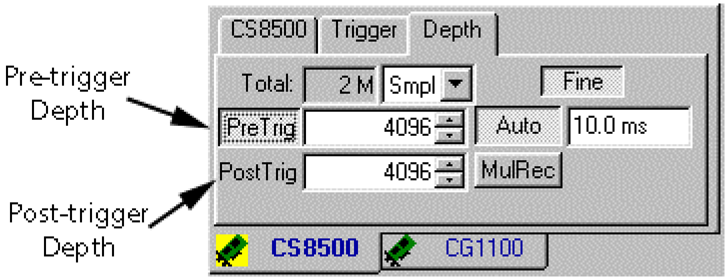
The post-trigger signal should show an uneven signal turning into a sine wave soon after the signal passed the trigger marker.



Notice how the post-trigger data did not begin until the signal measured at least as high as the trigger level. When the signal reached the point of the trigger level, the trigger event occurred and the capture began.

Introduction to Trigger Depths

Click on the **Depth** tab in the System Control to see the trigger depth settings.

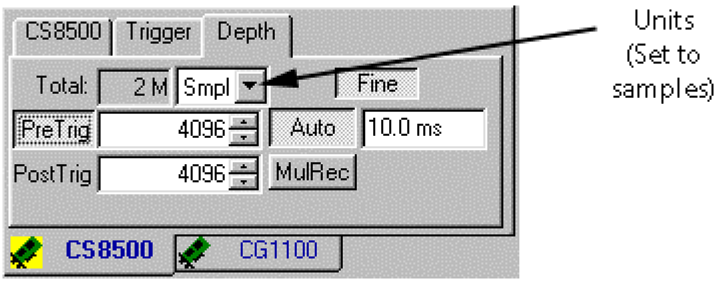


By default both pre- and post-trigger depth are set to 4096 samples. The trigger depths are limited by the amount of memory on-board your CompuScope card.

You can turn off pre-trigger acquisition by clicking on the **Pre Trig** button.

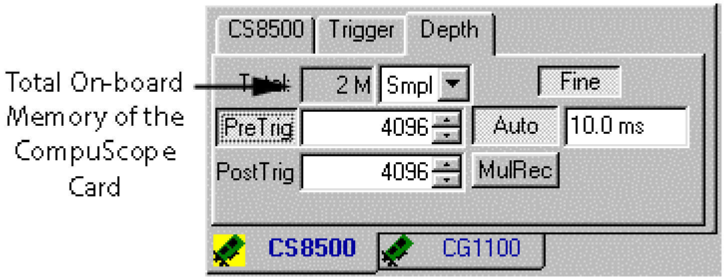
About Trigger Depth Units

Trigger depths are shown in number of **samples** (or points). This is displayed in the Units value in the **Depth** tab. This field cannot be changed.



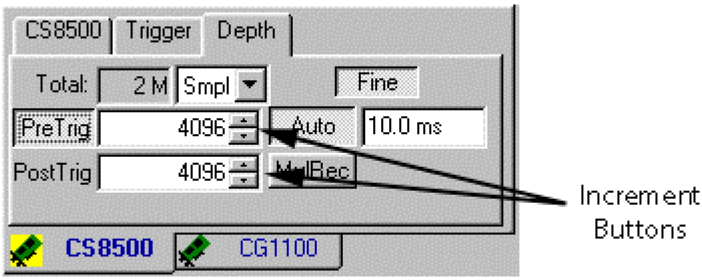
About the Trigger Total

The **Total** field reports the total amount of data that can be stored in the on-board memory of the CompuScope card. The total is shown in samples.



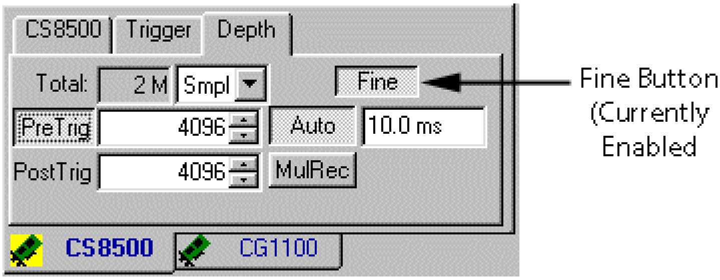
About Increment Buttons

The **Increment Up** and **Increment Down** buttons are located to the right of the pre- and post-trigger fields and are used to make changes to these settings.



About Fine Mode

Fine Mode is enabled by default.



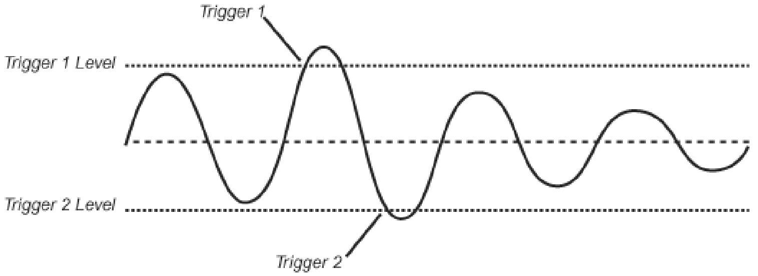
When **Fine Mode** is enabled, the increment buttons increase or decrease pre- and post-trigger data settings by 64 sample points. When **Fine Mode** is disabled, the changes are made in increments of 640 sample points.

Creating Multiple Independent Trigger Sources

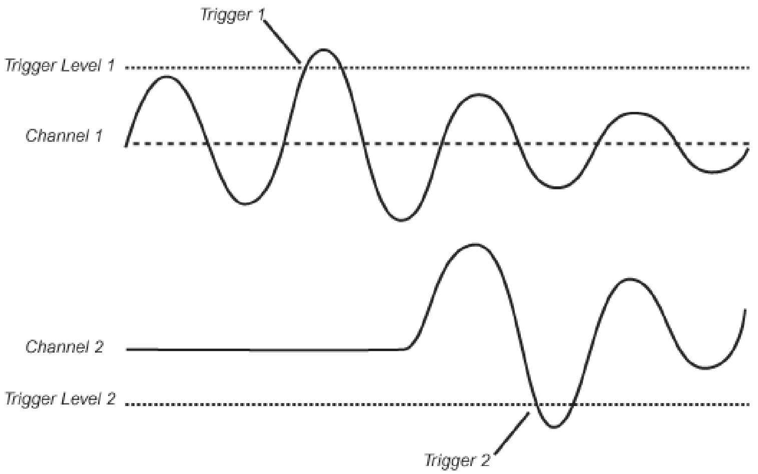
About Windowed Triggering

Certain CompuScope cards support “Windowed Triggering,” meaning that you can set two independent trigger conditions which are OR’ed together. Windowed triggering allows you to:

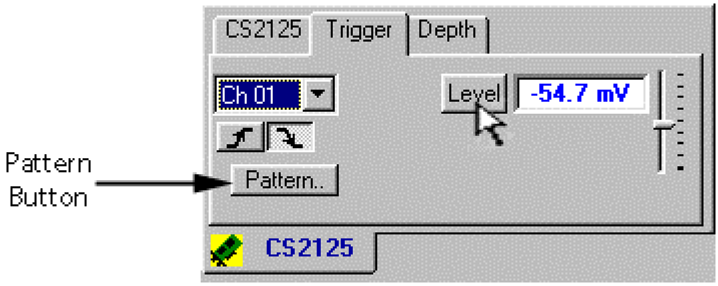
- Set two different triggers on the same channel



- Or set one trigger condition on each input of a CompuScope



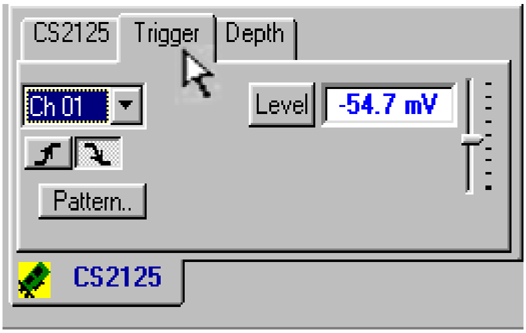
These triggers are set via the **Pattern** setting in the **Trigger** tab of the System Control.



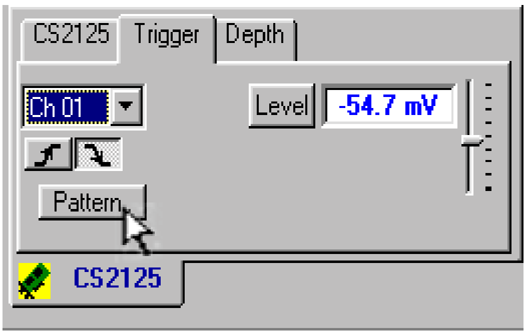
The Trigger Pattern dialog allows configuration of multiple independent triggers. Trigger source, slope and level can be modified, and trigger sources may be added and removed. Multiple triggers are not available for all CompuScope models.

Open the Advanced Trigger Configuration Dialog

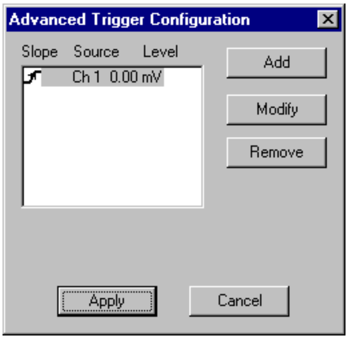
1 Click on the **Trigger** tab of the System Control if it is not already in front.



2 Click on the **Pattern** button.

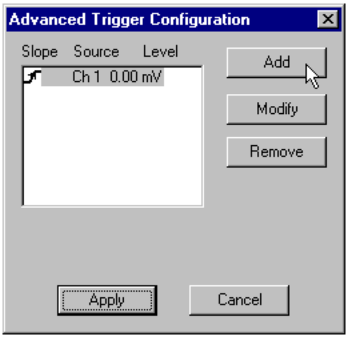


3 This brings up the Advanced Trigger Configuration dialog, listing all of the currently specified trigger sources. Currently only the default trigger source is listed.



Create Two Different Trigger Sources on Channel 1

1 In the Advanced Trigger Configuration dialog, click **Add**.



2 This brings up the **Trigger Source** dialog with a blank Source field.

3 Click on the **Source** drop-down list to select Ch 1.

- Note that on CompuScope 1016, the trigger source can be External only.
- Note also that if you have a Multi-Card system installed, you will see one trigger source for each channel available in your system. For example, for a three-card CS8012A system, there will be six input channel trigger sources available (since each CS8012A has two input channels). If you do not see the right number of trigger sources, go to **Preferences** in the Tools menu. Make sure the **Independent Trigger** item is checked.

4 To change the Trigger Level, use the increment/decrement buttons to set a percentage of the current input range of the selected trigger source.

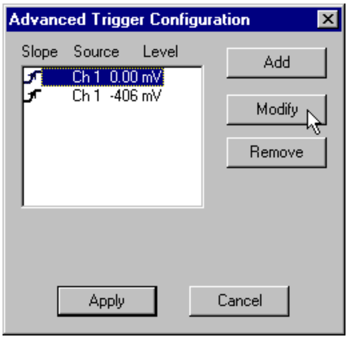
For instance, since our Trigger Source is set to Ch 1, and the current input range for Ch 1 is ±1V, if we select 50%, the trigger level will be set to +500 mV. If we select -10%, the trigger level will be set to -100 mV. The maximum positive setting is 100% and the maximum negative setting is -100%.

Set the Trigger Level to about **-40%**, or about -400 mV. You can click and hold the Down Increment button until it reaches the desired level.

5 Click **OK** to accept the new trigger source.

6 We will now modify our initial trigger source.

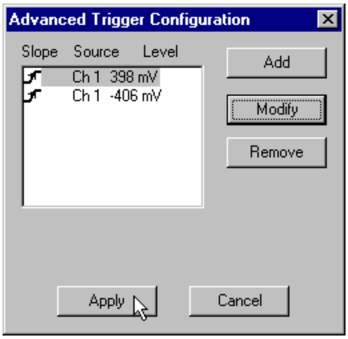
Click on the first item in the list to highlight it and then click **Modify**.



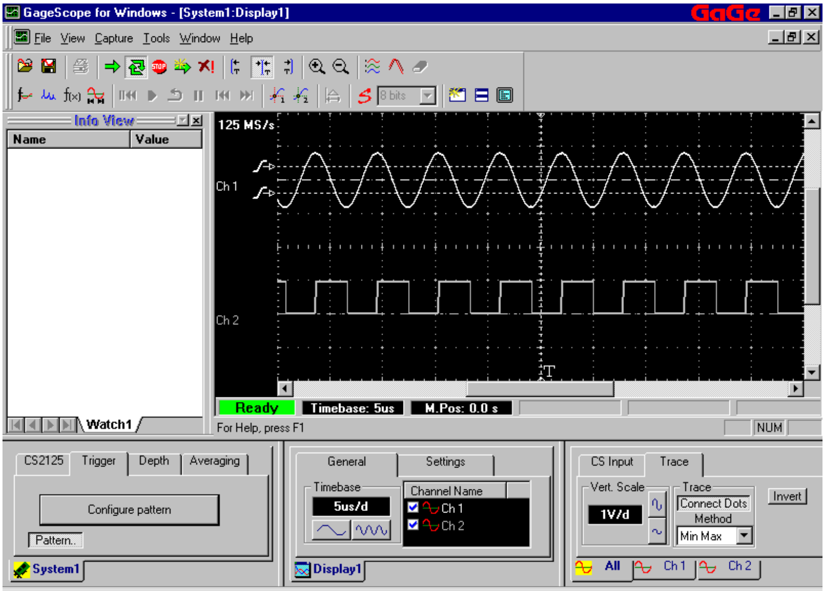
7 Change the trigger level from its current value, 0.00 mV, to **+40%**, equivalent to about +400 mV.

Click **OK** when finished.

8 Now click **Apply** in the Advanced Trigger Configuration dialog to apply the changes. Your changes will be updated in the Display Window.



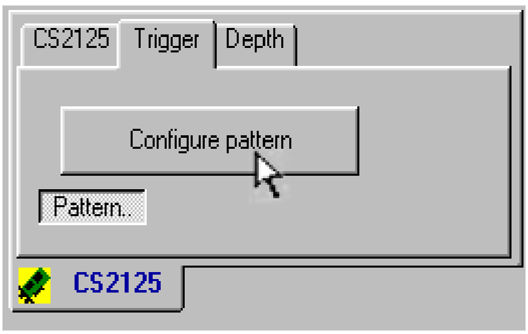
9 You will now see the signals in the Display Window moving somewhat jerkily to the left and right. This is because the signals are being triggered off the first trigger source and then the second, as the trigger level is constantly shifting from +400 mV to -400 mV.



Create a Trigger Source on each Input Channel

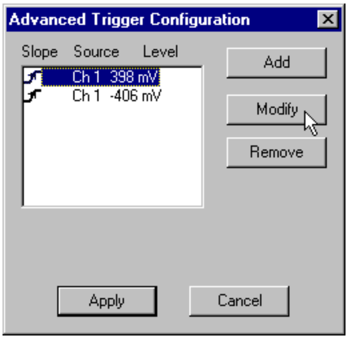
1 We have just created two trigger conditions on Ch 1. We will now create one trigger condition on Ch 1 and one on Ch 2.

Click the **Configure Pattern** button in the Trigger tab.



2 The Advanced Trigger Configuration dialog appears displaying our two trigger sources set to Ch 1.

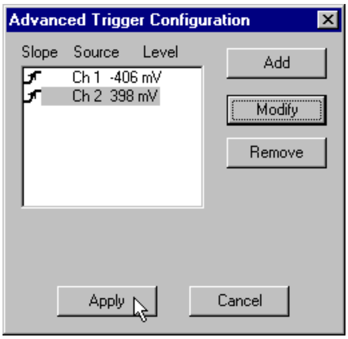
Select the item in the list with the positive trigger level to highlight it, and click **Modify**.



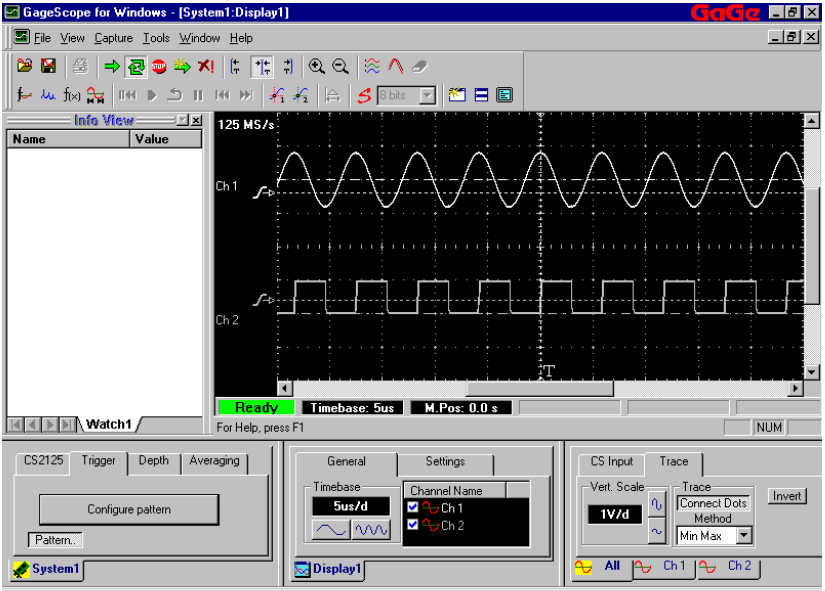
3 Change the Trigger Source to **Ch 2** by clicking the **Source** drop-down menu.

Click **OK** to accept the changes.

4 Click **Apply** to exit the Advanced Trigger Configuration dialog.

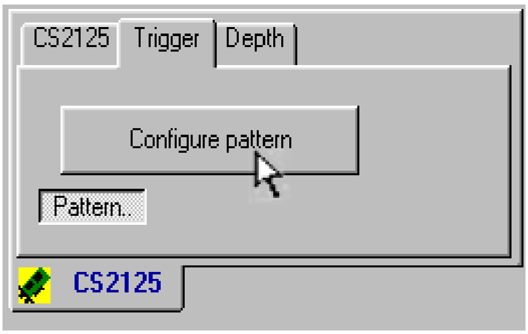


5 Note first that the second trigger has moved down to Channel 2. The signals are again moving jerkily to the left and right, this time because they are repeatedly triggering off Ch 1 and then Ch 2.

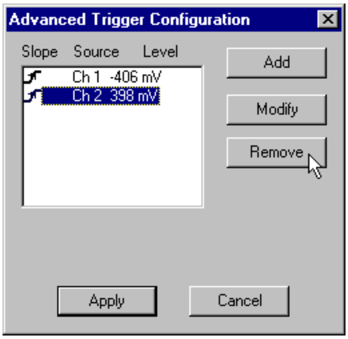


Remove the Second Trigger Source

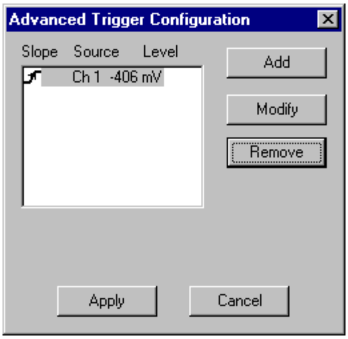
1 Click the **Configure Pattern** button in the Trigger tab.



2 In the Advanced Trigger Configuration Dialog, highlight the Ch 2 trigger source and click **Remove**.

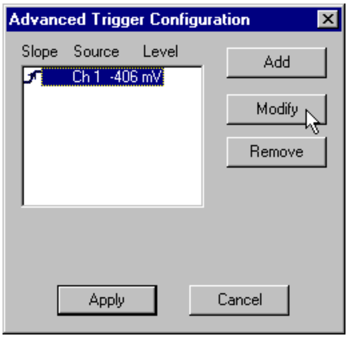


3 The trigger source will disappear from the list.

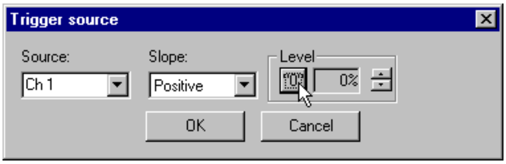


Change the Trigger Level Back to 0 mV

1 Click on the remaining trigger source and click **Modify**.

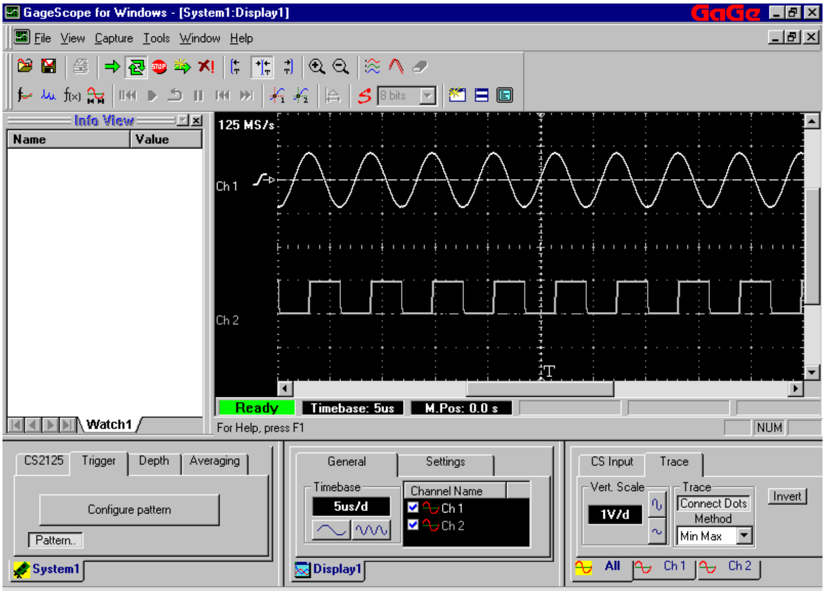


2 Click the **Level** button to change the trigger level back to **0 mV**.



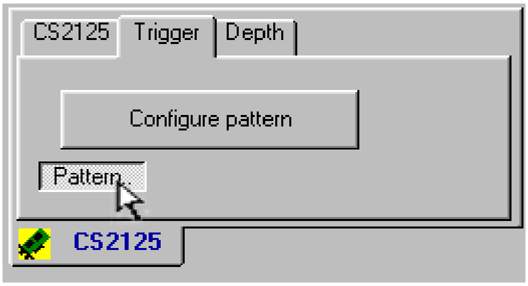
Click **OK** to accept the change.

3 Click **Apply** to exit the Advanced Trigger Configuration dialog. Our display shows that we are triggering off Ch 1 at 0 mV.



Exit the Trigger Pattern Configuration

1. To return to the regular Trigger Tab, click on the **Pattern** button so that it is no longer pressed in.



Notes

If you have already created two independent triggers and you attempt to add a third, you will not be able to select a Trigger Source—the drop-down menu will be blank. On CompuScope cards that support multiple independent triggers, you cannot set more than two trigger sources.

Triggering Summary

- System triggers are set in the Trigger Tab of the System Control as well as in the Pattern dialog. Not all CompuScope cards support multiple triggers.
- The **Trigger Align** buttons can be used to automatically center, left-align or right-align the trigger marker.
- A trigger is selected in the Trigger Tab of the System Control or in the Pattern dialog. Triggers can be set to channel 1, channel 2, or external, and can also be disabled.
- The trigger level can be changed by dragging the trigger level line in the Display Window, or by dragging the trigger level slider in the Trigger tab of the System Control.
- Trigger slope can be changed by double-clicking the trigger slope icon in the Display Window, or by right-clicking to bring up the Slope context menu and then left-clicking on the desired slope.
- Trigger timeout is set in the Depth Tab of the System Control. You can disable the timeout by clicking **Auto** so that it reads **Normal** and is not “pressed in.” When a timeout is reached, GageScope® issues a software trigger, i.e., it triggers at the moment of timeout.
- Timeout is automatically disabled when you click on the **One Shot** button. It is automatically re-enabled when you click on the **Continuous** button.
- The amount of captured data is set in the Depth tab of the System Control. The unit of measurement can be set in samples, percent of on-board memory, and time. The Total field reflects the total amount of data that can be stored in the on-board memory of the CompuScope card.
- Both pre- and post-trigger depths are set to 4096 by default.
- When the **Fine** button (Depth tab) is enabled, pre- and post-trigger depths are increased or decreased in increments of 64 sample points. With Fine Mode disabled, the changes are made in increments of 640 sample points.

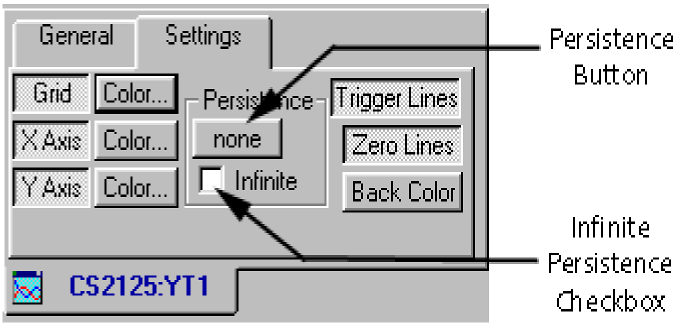
Tutorial 4: Working with Special Modes

This tutorial deals with three special modes available in GageScope®:

Persistence Mode

The **Persistence** setting allows the user to control the persistence of the signal display in the Display Window. By default, the signal does not persist in the Display Window. However, with the special Persistence Controls in GageScope®, you can specify any degree of persistence that you wish, from none to infinite.

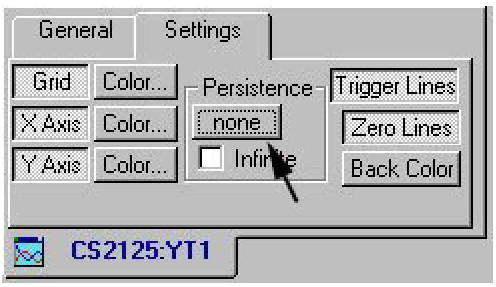
The **Persistence** button is located in the Settings tab of the Display Control.



Set Persistence to Infinite

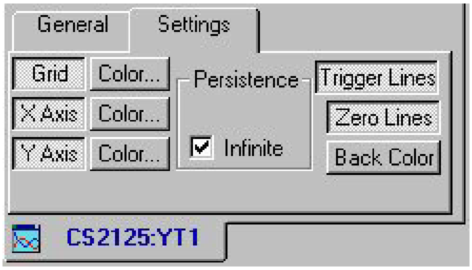
1 Make sure you are in **Continuous Capture** mode (the  button in the Toolbar should be "pressed in.")

2 By default, Persistence is disabled. The button underneath Persistence reads **None**. To enable persistence, click on **None**.

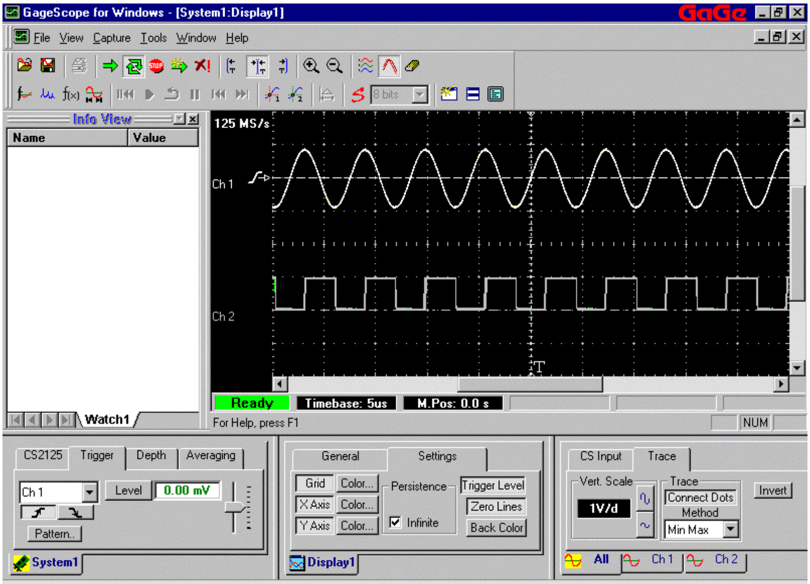


3 The Persistence field appears.

4 Click on the **Infinite** checkbox for infinite persistence. This is equivalent to clicking the **Overdraw Mode** button in the toolbar.



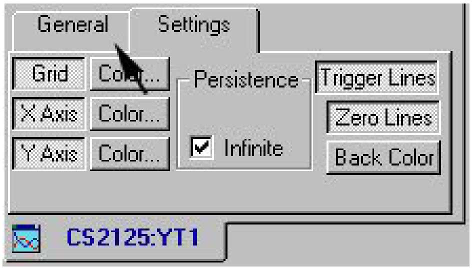
5 Our signals are now persisting on the screen. You may notice the signals getting slightly thicker.



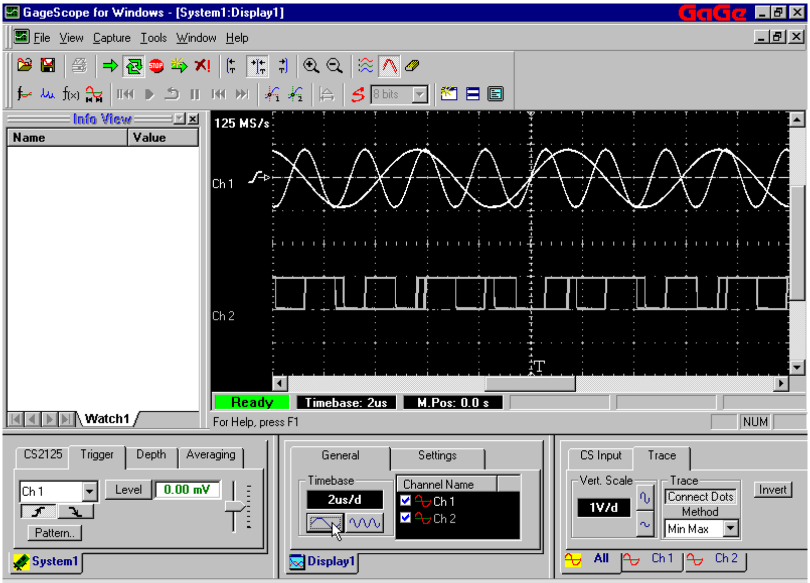
However, to truly see Persistence at work, we will change some settings that will affect the signal display.

Change the Timebase to 1 us/d

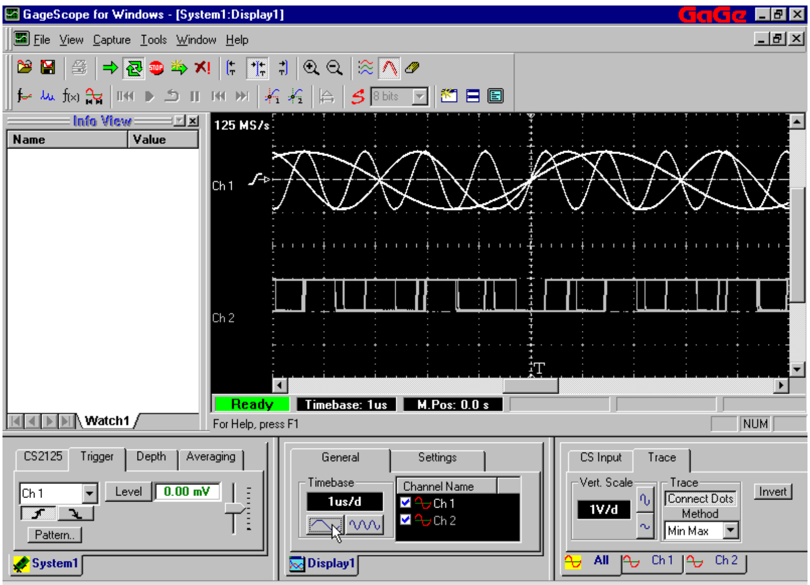
1 Click on the **General** tab of the Display Control to access the **Timebase** buttons.



2 Click on  once to change the timebase to **2us/d**. Watch how your signal display changes—the old signal persists while the new signal is being displayed.

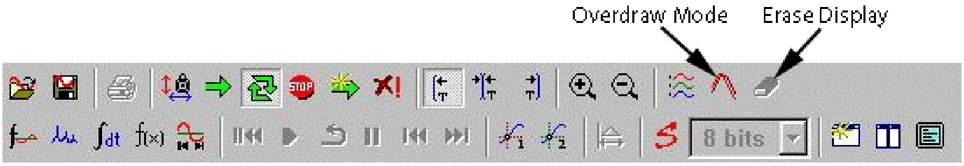


3 Click on  again so that the timebase becomes **1us/d**. The new signal will be displayed along with the previous two signals.

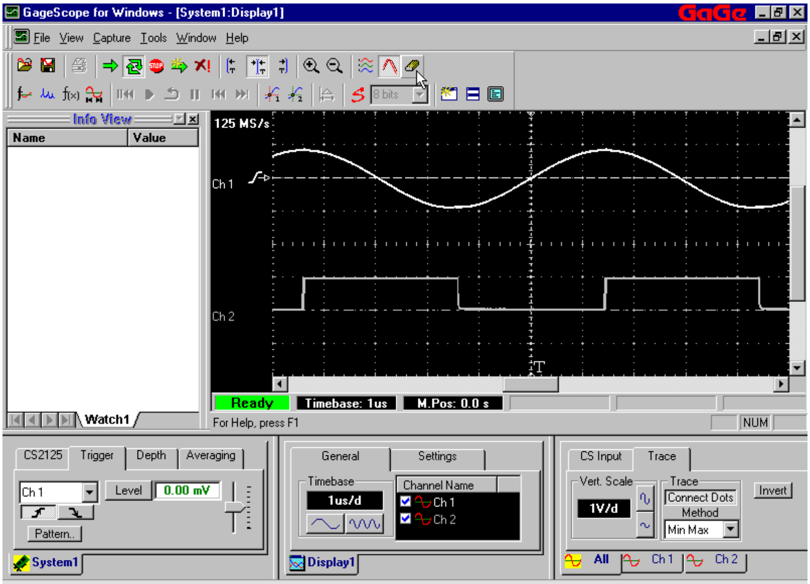


Erase the Display

1 An **Erase Display** button becomes available in the toolbar when Persistence mode is turned on.

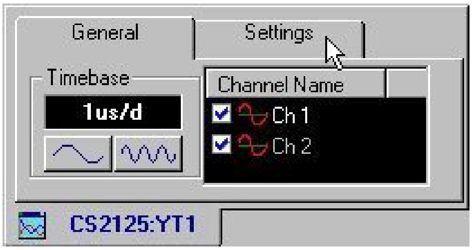


2 Click on [Erase Display button] to clear the previous signals. The latest signal with a timebase of 1us/d is the only one in the Display Window now.

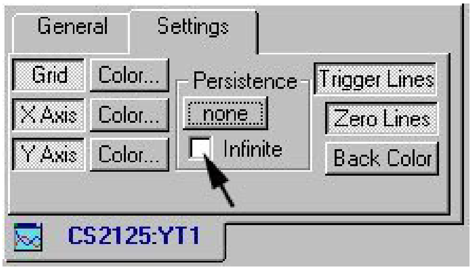


Set Persistence to 5 seconds

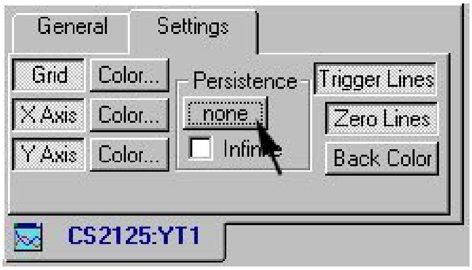
1 Click on the **Settings** tab of the Display Control so that it is again in front.



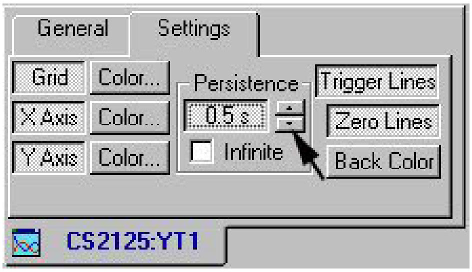
2 Turn off Infinite Persistence by clicking on the checkmark next to **Infinite**.



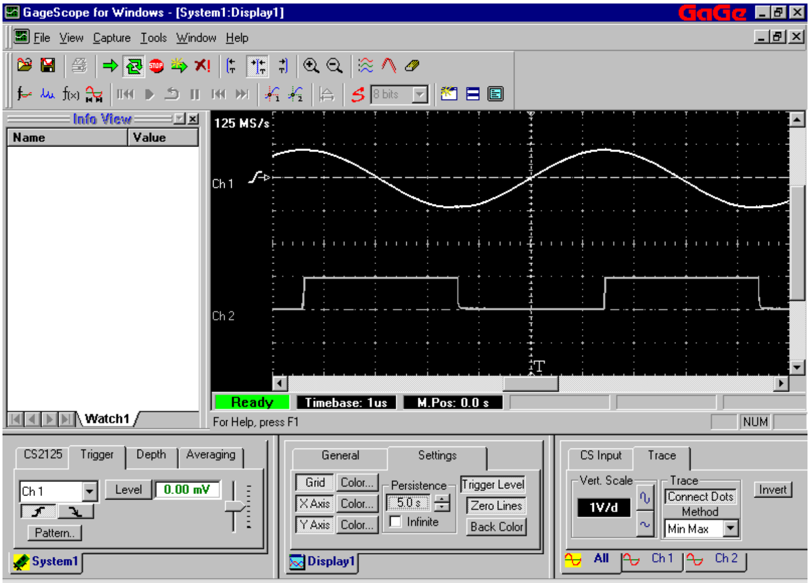
3 Turning off Infinite Persistence turns off Persistence altogether. To enable Persistence again, click on **None**.



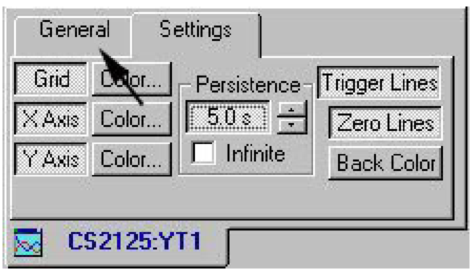
4 To change the length of time that the signal will persist on the screen, you can click on the **Increment Up** and **Increment Down** buttons.



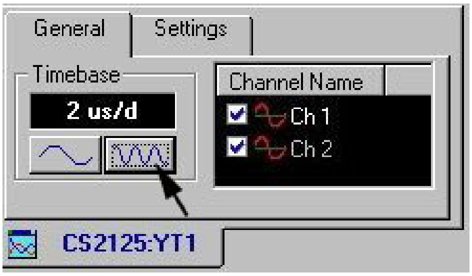
5 Click and hold the button until the Persistence is set to **5.0s**.



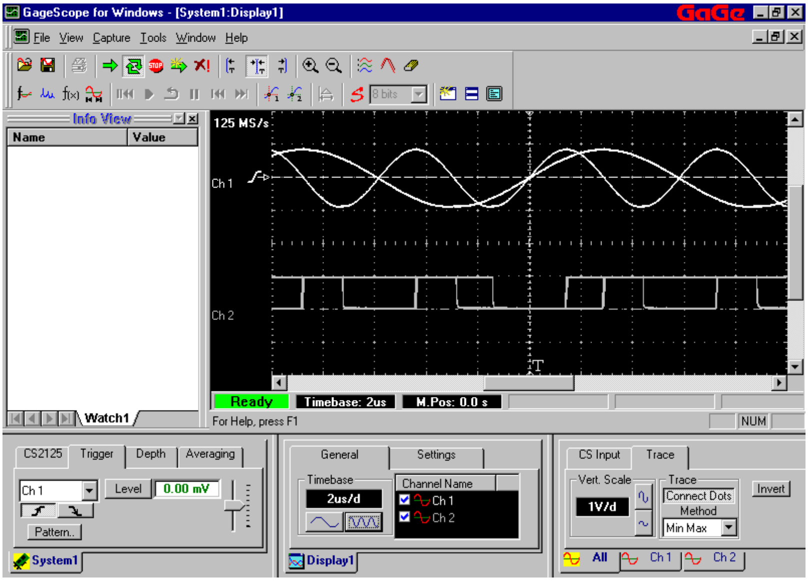
6 Now our signals will persist, but only for 5 seconds. To test this, we will again go to the **General** tab of the Display Control.



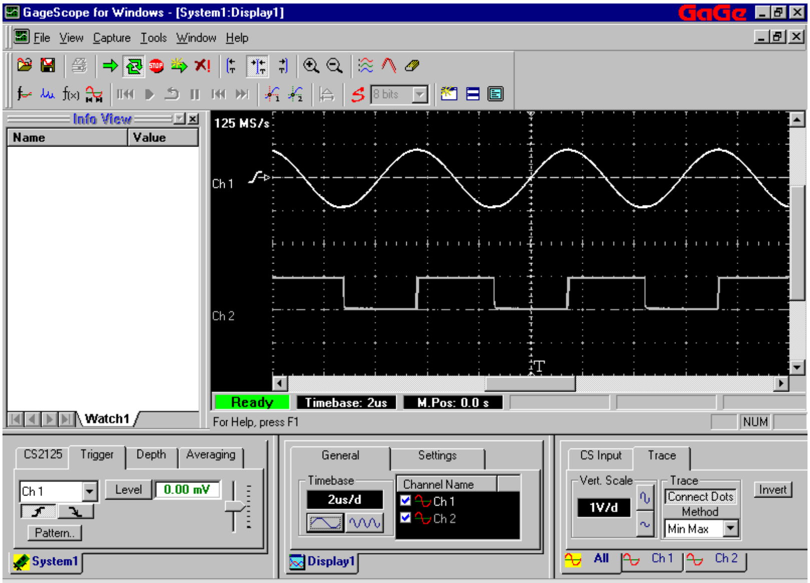
7 Change the Timebase to **2us/d** by clicking .



8 You will notice that the old signal briefly persists on the screen.



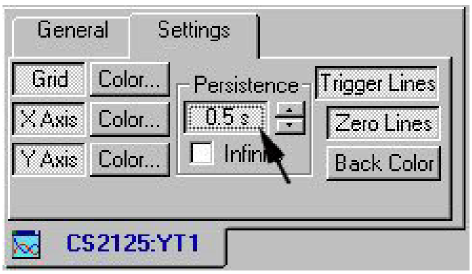
9 The display is erased after a short time, so that only the latest signal is preserved.



Setting the Persistence to 5 seconds has the same effect as clicking the **Erase Display** button every 5 seconds.

Turn Persistence Off

1 To disable Persistence Mode, go back to the **Settings** tab of the Display Control. Click on the **Persistence** field so that it reads **None**.



2 To get ready for the next part of the tutorial, change the Timebase back to **5 us/d** (click on [button] in the **General** tab of the Display Control).

SuperRes Mode

SuperRes Mode allows an averaging window to be defined within a single acquisition. All samples within the averaging window are accumulated, resulting in an effective increase in the sample bits. The averaging window is then shifted, over the current acquisition, by its width and the same averaging process is performed repeatedly.

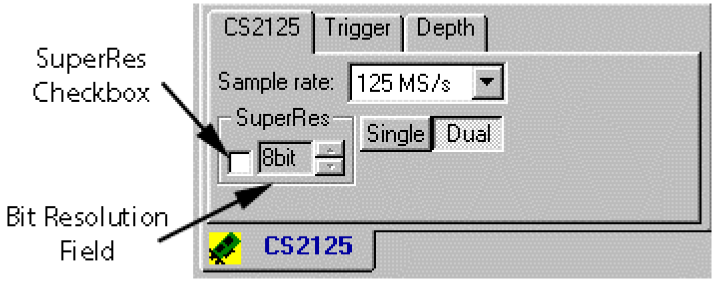
SuperRes allows you to specify the desired "Effective Resolution." For example, you can specify 12 bit resolution even if you are using the 8 bit CompuScope 8500. GageScope® will automatically perform digital filtering (averaging of oversampled data) to enhance the effective resolution at the expense of sample rate.

For example, in order to provide 12 bits for the CompuScope 8500 (an increase of 4 bits), GageScope® has to average $2^4 = 16$ adjacent points, hence reducing the sample rate by a factor of 16. This means that instead of 500 MS/s, the sample rate will be 500/16 = 31.25 MS/s.

Note

- The effective resolution mentioned here is **not** the same as Effective Number of Bits (ENOB). ENOB is a measurement of the Signal-to-Noise Ratio.

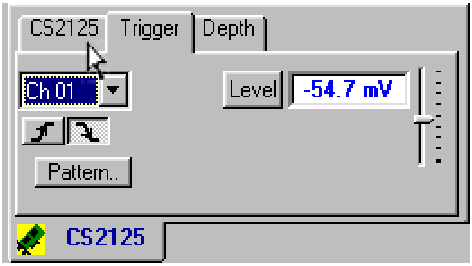
SuperRes is located in the CS tab of the System Control.



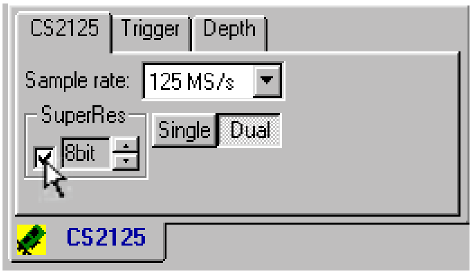
Enable SuperRes Mode

1 Make sure you are in **Continuous Capture** mode (the  button in the Toolbar should be "pressed in").

2 Click on the **CS** tab of the System Control to bring it to the front.

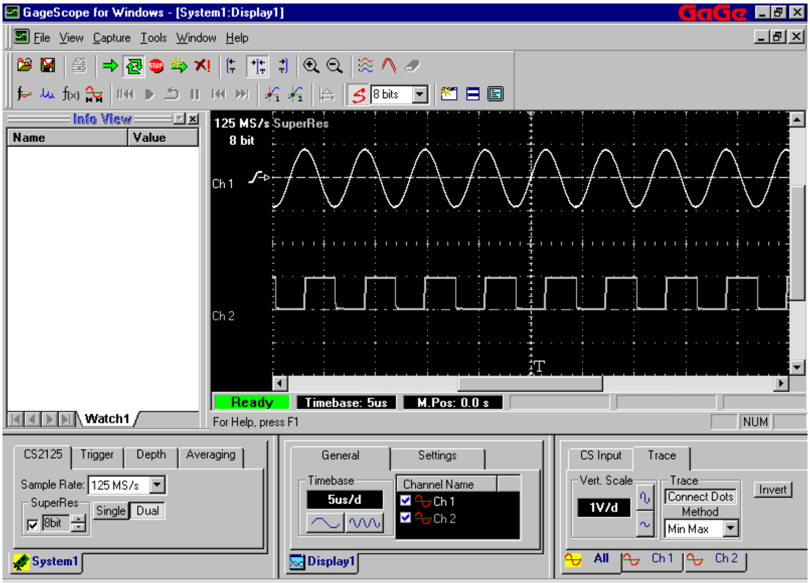


3 Click on the **SuperRes** checkbox to enable SuperRes mode.



You can also click the **SuperRes** button in the toolbar (it should be "pressed in").

4 Notice that **SuperRes** appears in the upper left of the Display Window, next to the sample rate.

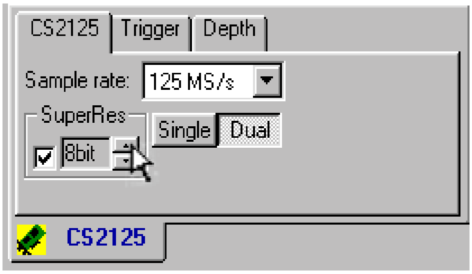


Increase the Bit Resolution by 6 or 7 Bits

1 We will now increase the Bit Resolution of the signal displayed on the screen.

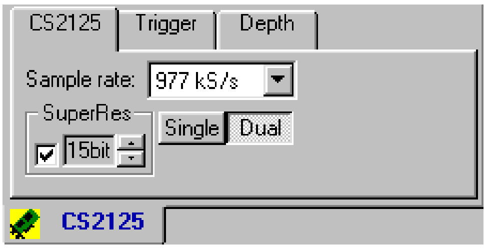
- 8 bit CompuScope cards: Increase the bit resolution to 15 bits
- 12 bit CompuScope cards: Increase the bit resolution to 18 bits

To increase the bit resolution, click on the **Up Increment** button on the CS tab of the System control.

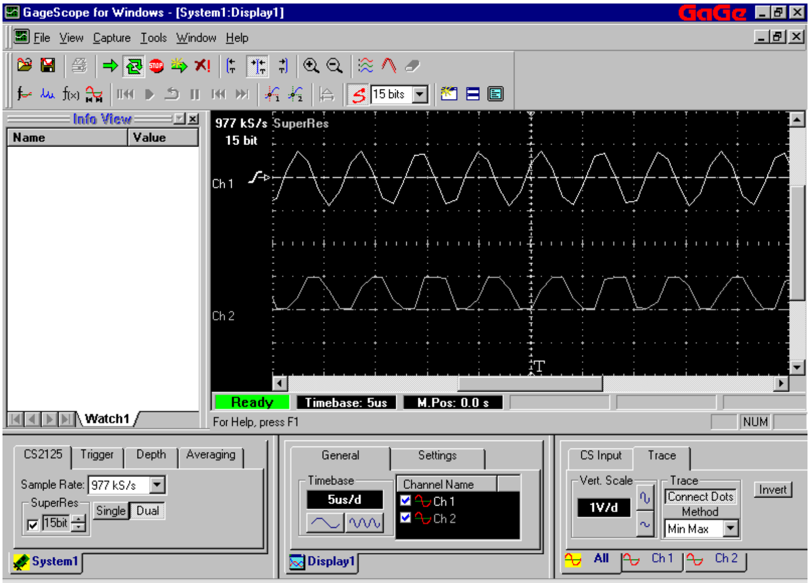


You can also click on the SuperRes drop-down menu in the toolbar.

2 Note that when you increase the bit resolution, the sample rate decreases. For instance, we have raised the bit resolution on our CS2125 to 15 bits and our sample rate has decreased to 977 kS/s from 125 MS/s. You will not be able to increase the sample rate until you decrease the bit resolution again.



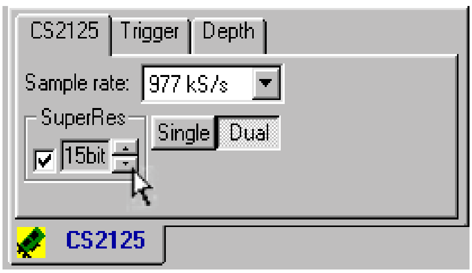
3 With such a comparatively low sample rate, our signal display becomes jagged.



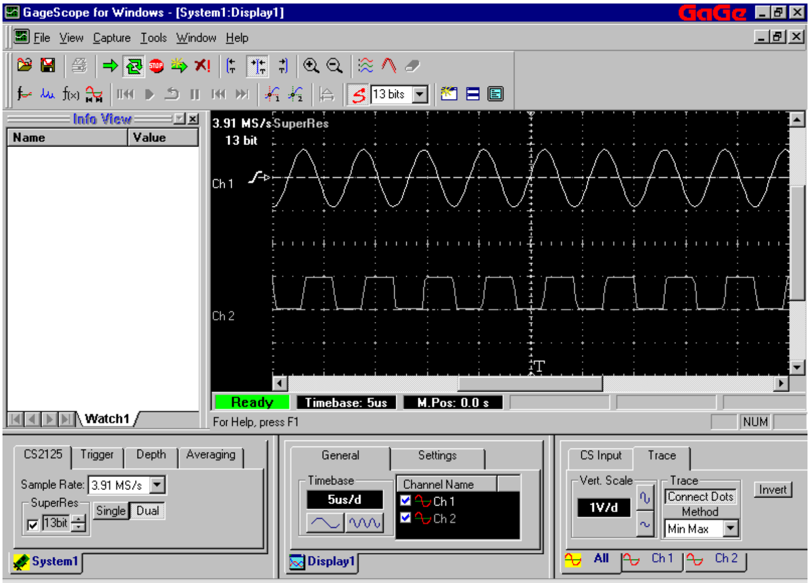
When the sample rate is low, the CompuScope card is acquiring fewer sample points per second. With fewer points available for building a representation of the signal, the resulting waveform looks jagged. Keep in mind that this illustration uses a slow capture rate. The true power of SuperRes is more evident with a faster sampling rate; while the waveform remains mostly unchanged, the signal to noise ratio improves dramatically thanks to the increased resolution of the acquisition.

Decrease the Bit Resolution by 2 Bits

1 Decrease the bit resolution by two bits by clicking the **Down Increment** button.

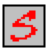


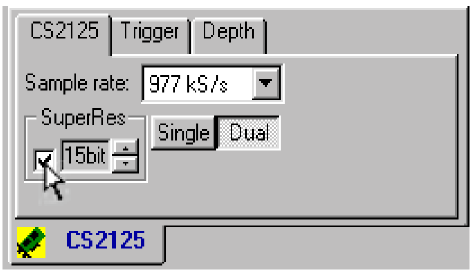
2 Notice that GageScope® automatically raises the sample rate to the maximum rate corresponding to the selected bit resolution—in our case, the sample rate rose to 3.91 MS/s when we decreased the bit resolution to 13 bits.



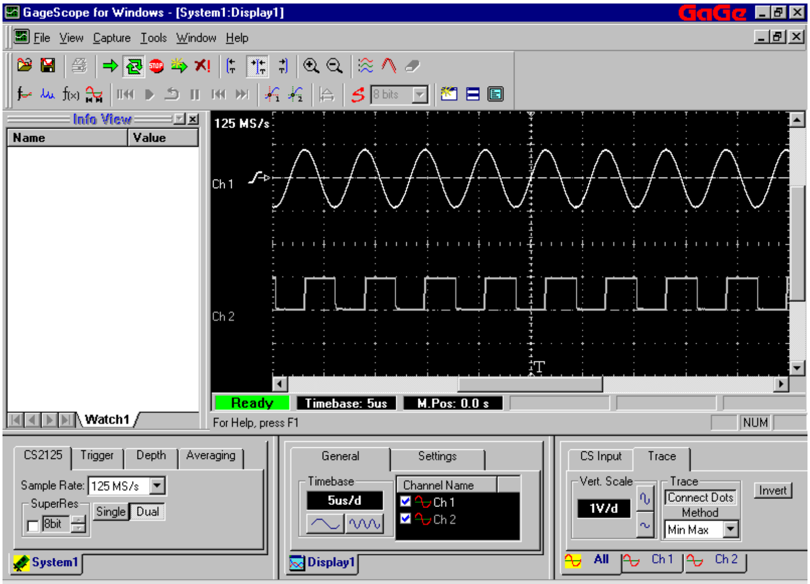
The signal looks less jagged than before, since more samples are being acquired per second.

Disable SuperRes Mode

1 To disable SuperRes mode, click on the **SuperRes checkbox** in the CS Input Tab (so that it is unchecked), or click on  in the toolbar (so that it is not pressed in).



2 The bit resolution has been returned to normal, and the sample rate is now at its maximum value for dual channel mode.



Multiple Record Mode

Multiple Record Mode takes advantage of the CompuScope card's deep memory buffers by allowing the hardware to stack captures in on-board memory, so that many small acquisitions can occur in a very short amount of time, with near-zero re-arm time. This feature is invaluable in applications in which trigger events are happening rapidly or unpredictably and A/D down-time must be minimized.

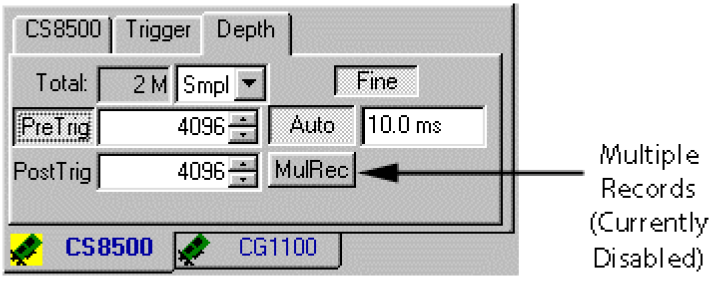
When Multiple Record Mode is enabled, the CompuScope card looks for a trigger event, acquires post-trigger data, then automatically re-arms itself to look for another trigger event. Data collected from each successive acquisition is "stacked" on top of the previous acquisition until the on-board buffer fills up.

For example, if a post-trigger depth of 1024 points is specified, the first acquisition stores data in addresses between 0 and 1023, the next acquisition from 1024 to 2047, and so on until the buffer is full.

Important Notes

- The number of records you can capture will depend on the amount of on-board memory your CompuScope card has, as well as other settings, such as the post-trigger depth and channel mode.
- With all currently available CompuScope cards, you can only capture post-trigger data in Multiple Record mode. When you click on the **MulRec** button, pre-trigger capture is disabled.
- Multiple Record Mode comes as a standard feature or as an option on most CompuScope cards, but is not available on the CompuScope LITE. If your CompuScope card does not have this capability, the **MulRec** button will not be available.

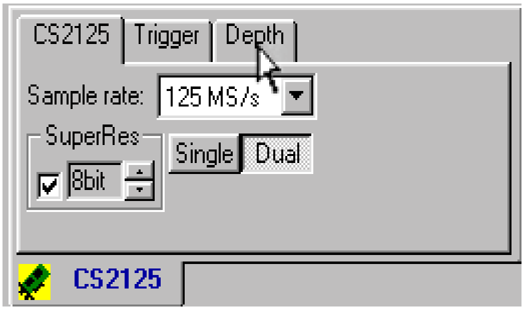
The Multiple Record setting is located in the Depth tab of the System Control.



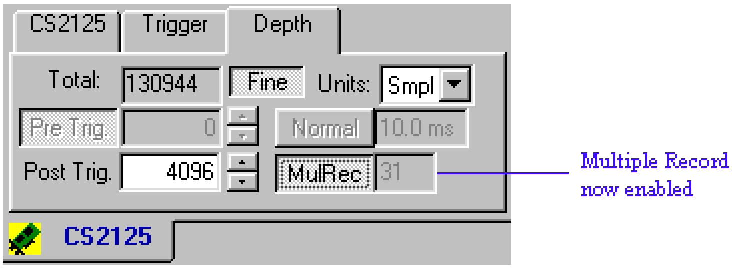
Perform a Multiple Record Capture

1 Make sure you are in **Continuous Capture** mode (the  button in the Toolbar should be “pressed in”).

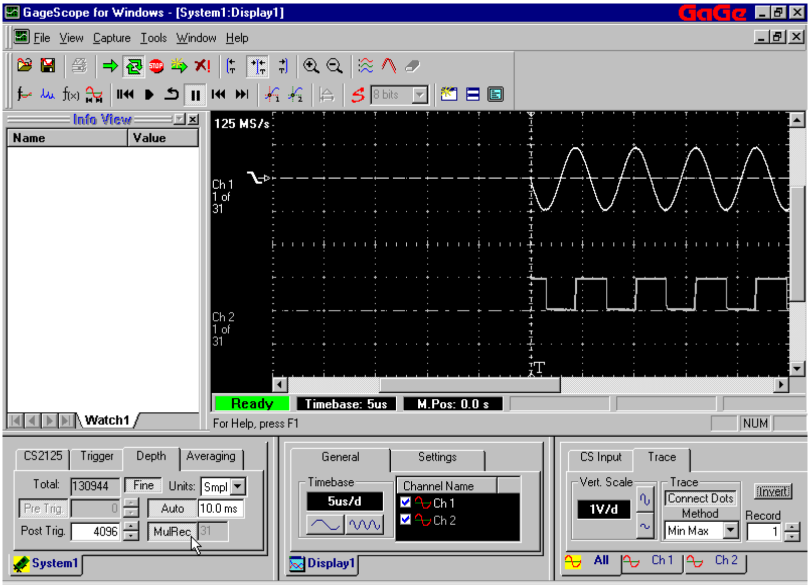
2 Click on the **Depth** tab of the System Control to access the Multiple Record controls.



3 Click on the **MulRec** button to enable Multiple Record Mode. Multiple Record Mode is enabled if its button is pressed in.



4 GageScope® automatically captures the post-trigger data for as many records as possible, depending on the current channel mode, sample rate, and post-trigger depth settings.



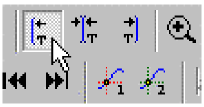
The number of records will be reflected in the Display Window underneath the channel number at the left. For example, if 31 records are captured as in our example, you will see "1 of 31" displayed underneath "Ch 1."

5 While you are performing a Multiple Record acquisition, you may briefly see the message "Transferring" in the Status bar.

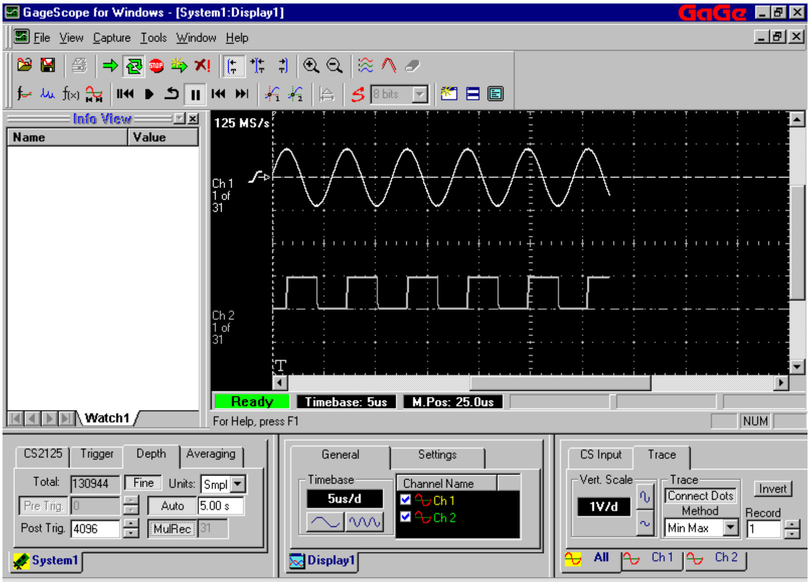
This means that GageScope® is transferring data from the CompuScope board to the internal buffers of the application.

Scroll Through the Multiple Record Data via the Toolbar

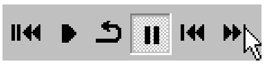
1 To see more of your Multiple Record data, click the **Trigger Left Align** button in the toolbar.



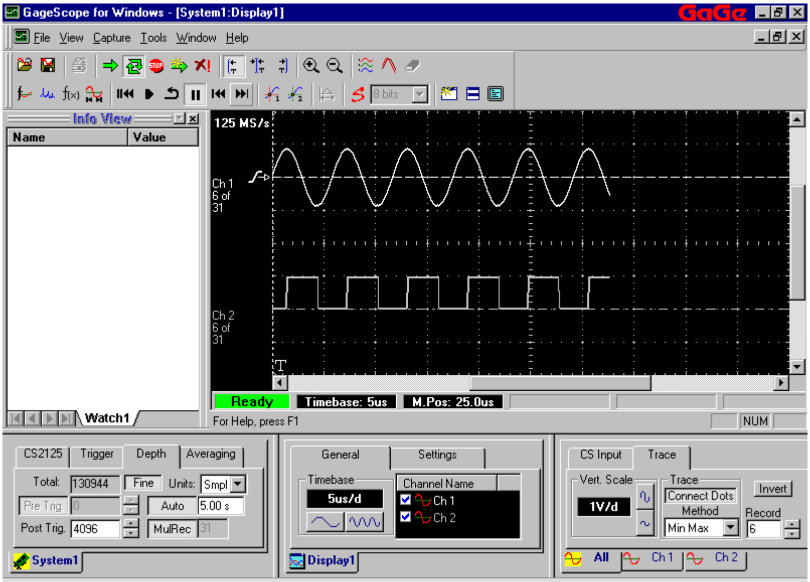
2 You should now be able to see the entire record.



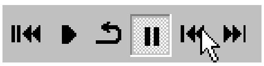
3 Use the **Go to Next Record** button in the toolbar to scroll through your records one by one.



4 Beside **Ch 1** and **Ch 2** in the Display Window, you can watch the record counters increase by one with each click.

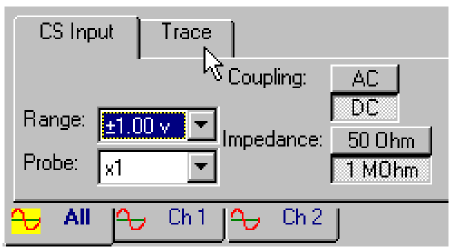


5 Click the **Go to Previous Record** button in the toolbar to scroll backward through your records until you get back to Record 1.

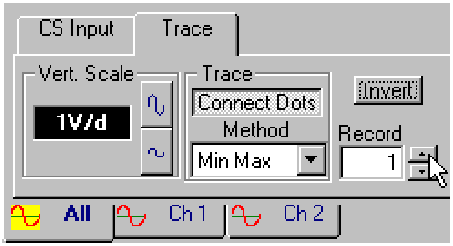


Scroll Through the Multiple Record Data via the Record Counter

1 Select the **Trace** tab in the Channel Control if it is not already in front.

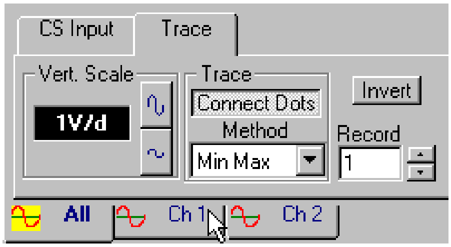


2 Use the  and  buttons to perform the same function as the **Go to Next Record** and **Go to Previous Record** buttons on the Toolbar.

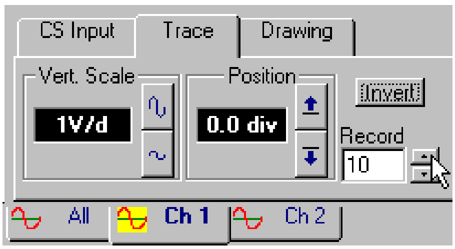


3 With these buttons, you can cycle through the records of all channels at once (if the All Channels Tab is current), or through the records of individual channels (if an individual channel tab is current).

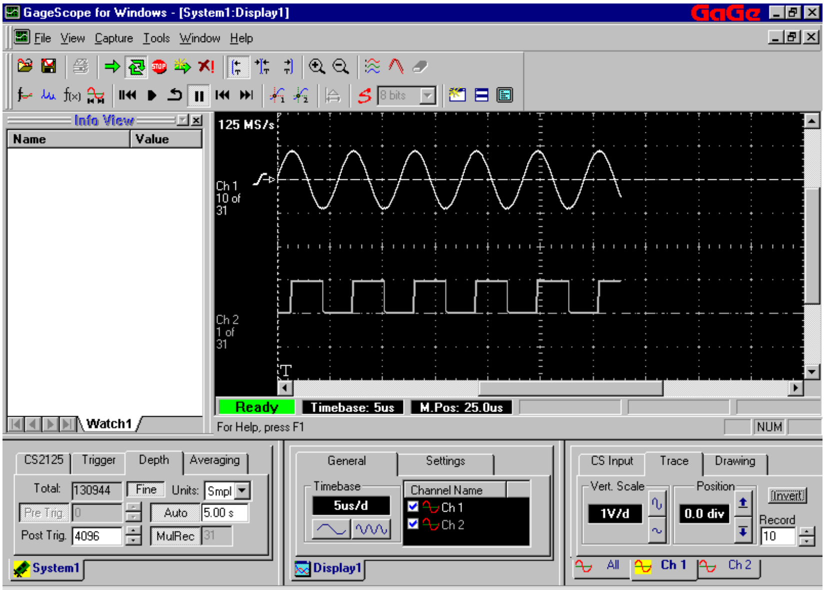
Click on the **Ch 1** tab of the Channel Control.



4 Use the Record Counter to go to **Record 10** of Channel 1.

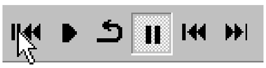


5 Now you are viewing Record 10 of Ch 1, while Ch 2 is still displaying Record 1.



Set all Records to Record 1 via the Toolbar

1 Set all channels to the first record with the  button in the toolbar.

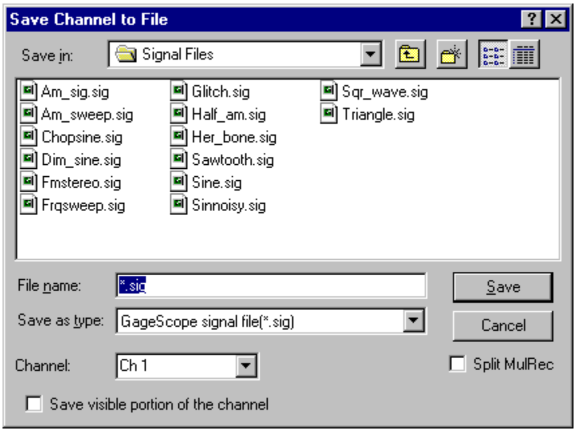


Save Channel 1 to Individual Files

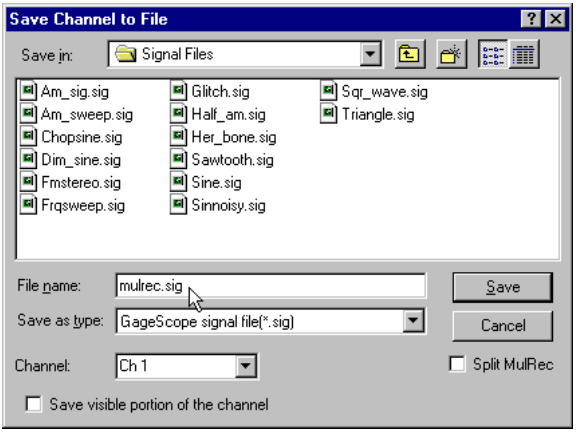
1 You can save a Multiple Record channel just as you would any other channel. However, there are a few options available for saving Multiple Record files which are not available with normal acquisitions. In this section, we will be saving each record in the acquisition to a separate file in one step.

Click **Save Channel**  in the toolbar.

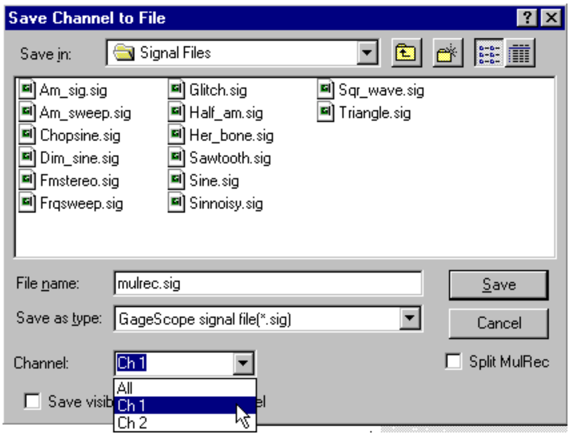
2 The Save Channel dialog appears.



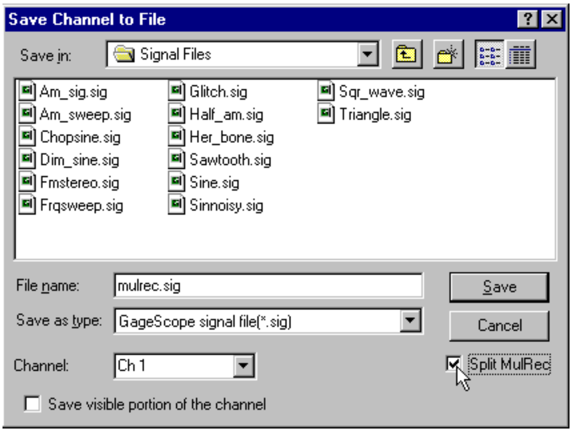
3 In the File Name field, type **mulrec.sig**.



4 **Ch 1** should already be selected in the Channel field. If it is not, select it from the drop-down menu.



5 To save each record in the acquisition to a separate file, check the **Split MulRec** checkbox.

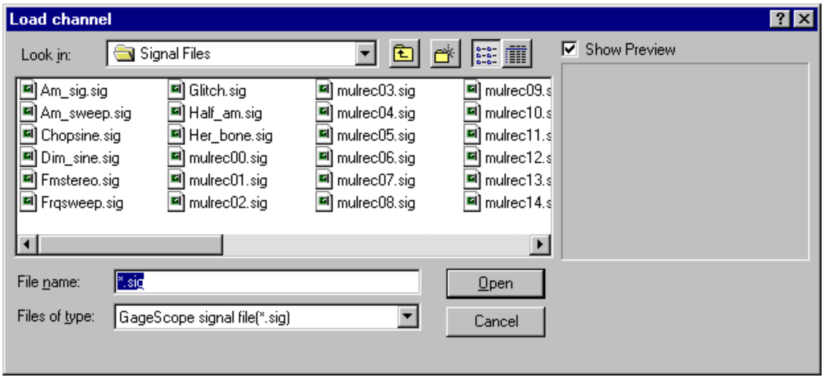


6 Click **Save**.

➧ Load one of the Saved Records to Ch 3

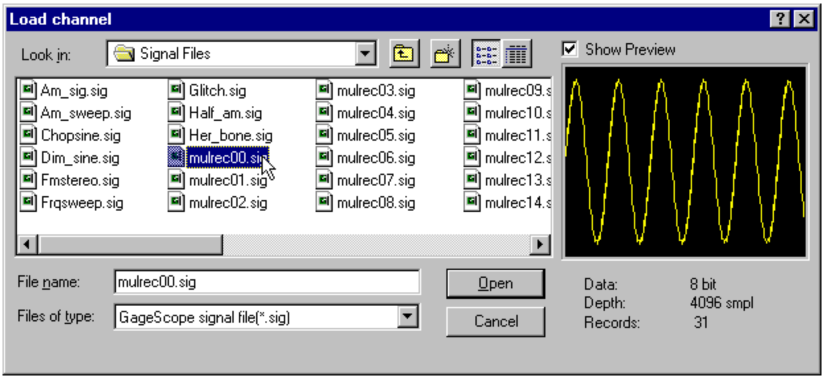
1 You can also load a Multiple Record channel as you would any other channel—by clicking  on the toolbar.

2 The Load Channel dialog appears.

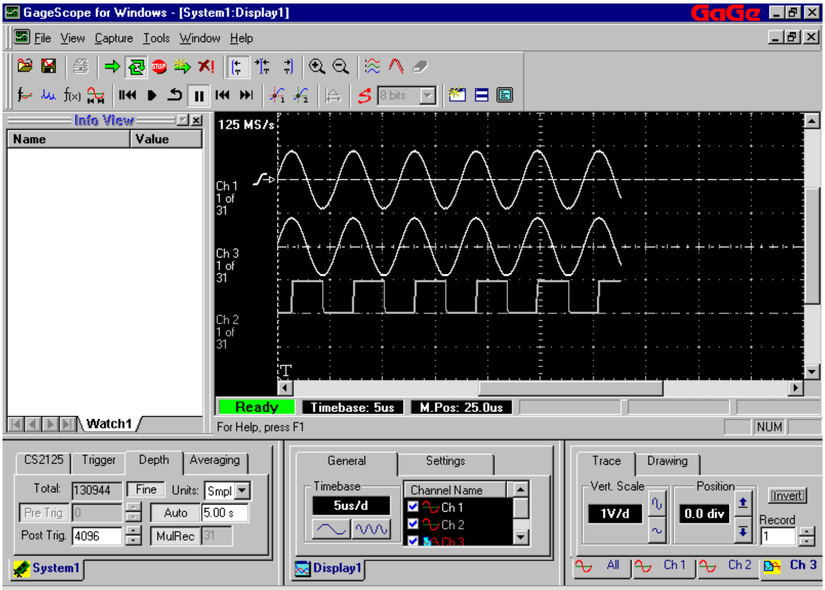


Notice that since we selected **Split MulRec** when saving our Multiple Record acquisition, and in our example there were 31 records in the acquisition, we now have 31 files named "mulrecxx.sig," where xx is a suffix attached by GageScope®, beginning at 00.

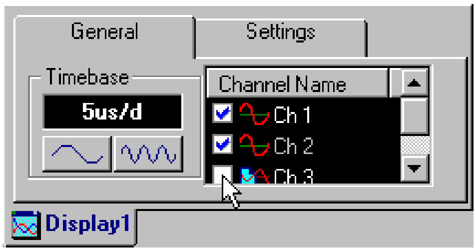
3 Click on the first of these files to select it, and then click Open.



4 Our signal file has been loaded as Ch 3, in between Ch 1 and Ch 2.



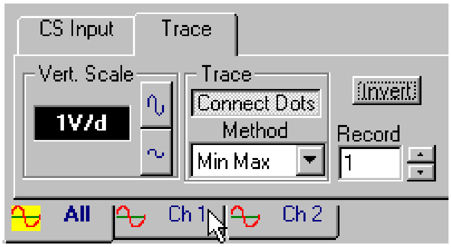
5 Hide Ch 3 by clicking the checkmark next to it in the Channel List (in the General tab of the Display Control).



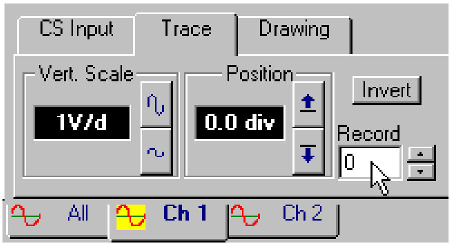
Look at All Records of Channel 1 Simultaneously

1 Up to now, we have been viewing one record at a time. GageScope® also allows you to view all records at once.

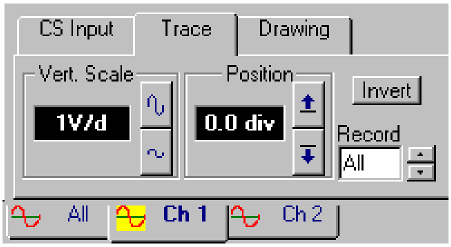
Click on the **Ch 1** tab of the Channel Control to bring it to the front again.



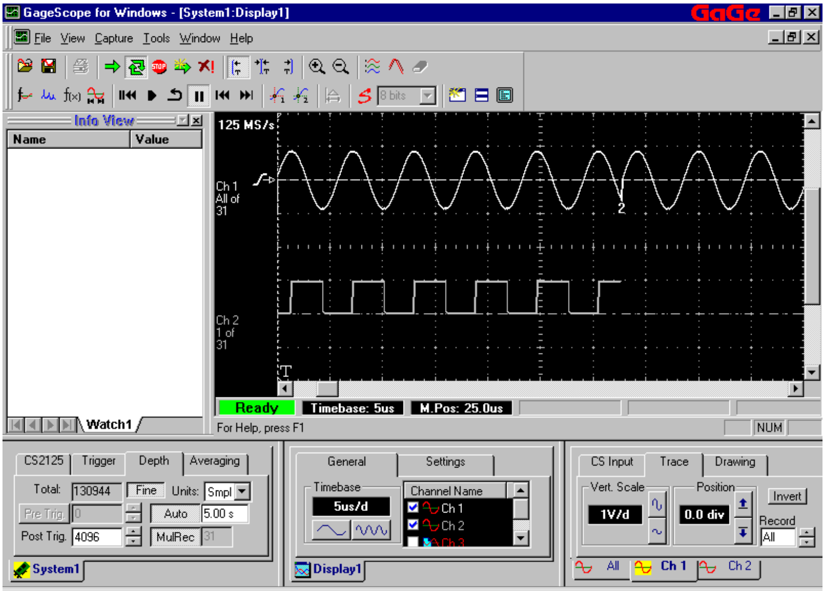
2 Click with the left mouse button in the **Record** field and press the Backspace to erase the value that's there. Then type **0** (zero).



3 Press Enter. The 0 will change to **All**.

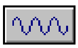


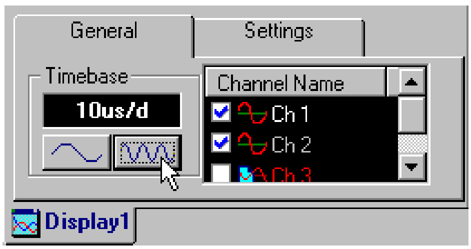
4 The display will now show all of your records at once. To the left of the slope icon and below “Ch1” on the screen below, notice the message “All of 31,” indicating that all 31 records are being displayed.



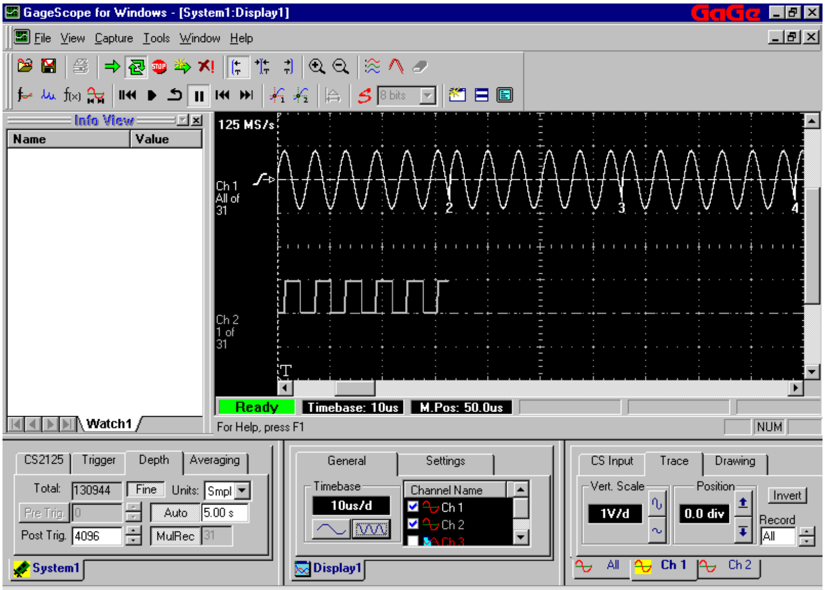
Beneath the start of each record in the signal, its record number is displayed.

Note that you are still seeing only one record of Ch 2. If you wish to see all records of both channels, you can click the **All** tab in the Channel Control and type 0 in the Record Counter, followed by Enter.

5 Increase the timebase by clicking on  in the **General** tab of the Display Control.



6 This allows you to see more records.

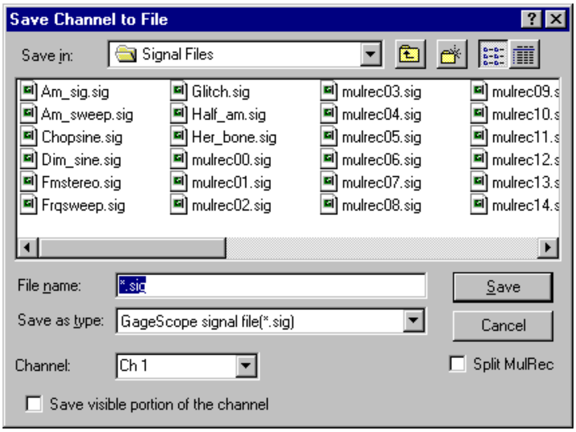


➧ Save All Channel 1 Records to One File

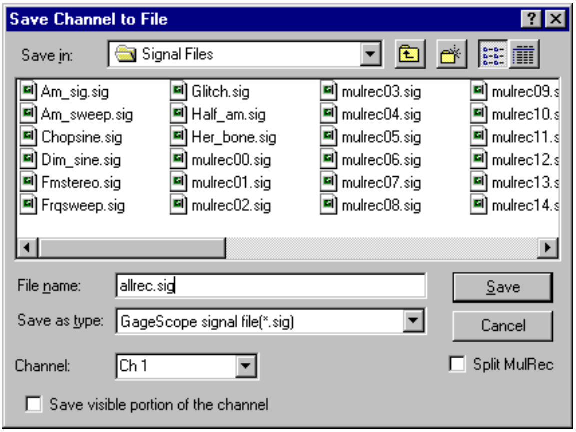
1 Now that we are viewing all of Channel 1’s records at once, we can also save all of the records to one file. This is only possible when **All** is displayed in the Record Counter for the channel you wish to save.

2 Click **Save Channel**  in the toolbar.

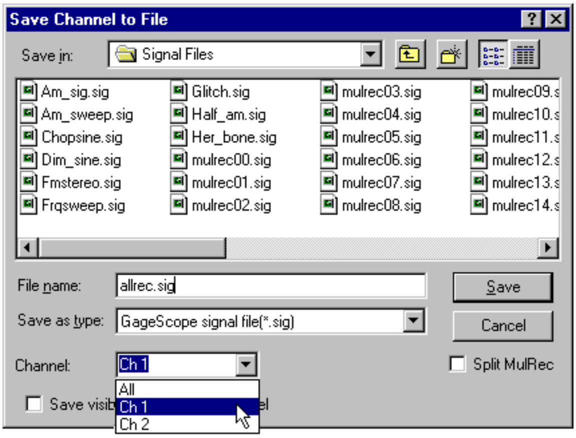
3 The Save Channel dialog appears.



4 In the File Name field, type **allrec.sig**.



5 **Ch 1** should already be selected in the Channel field. If it is not, select it from the drop-down menu.

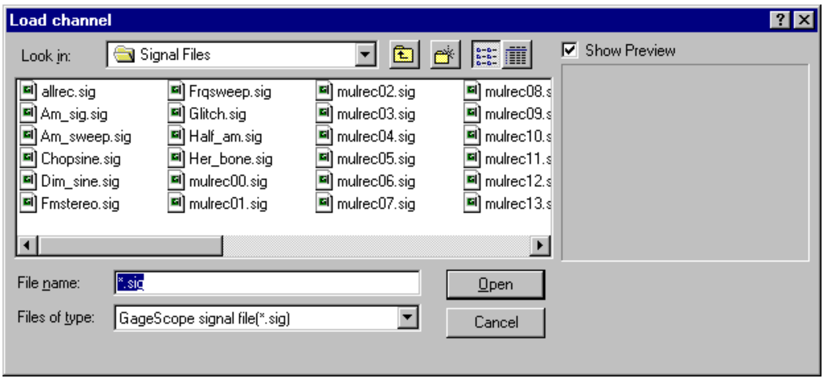


6 Click **Save**.

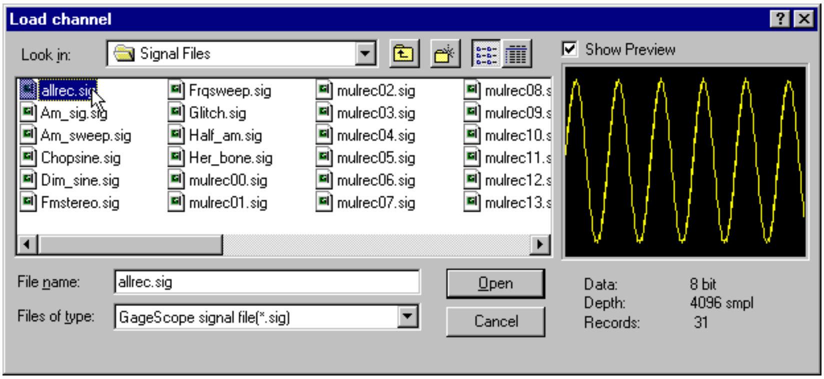
➧ Load the All Records File to Ch 4

1 Click  on the toolbar.

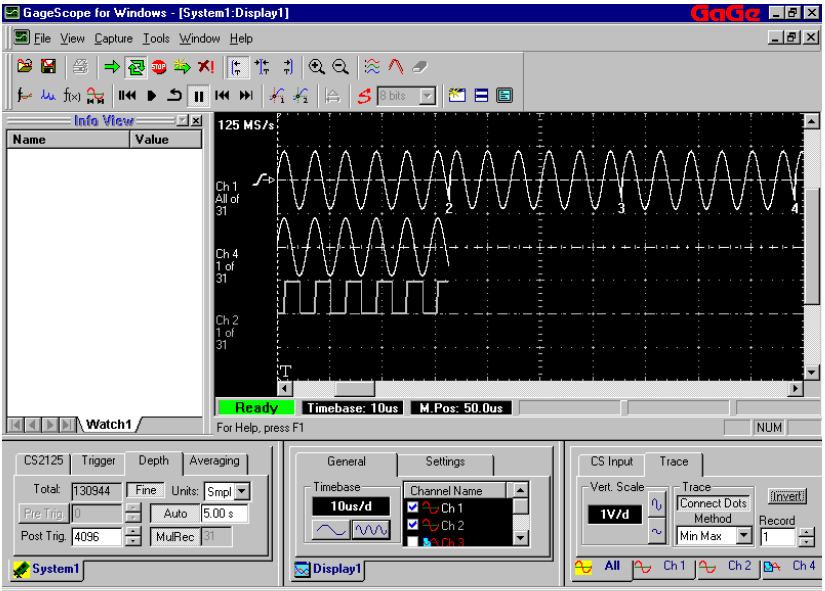
2 The Load Channel dialog appears.



3 Along with our multiple **mulrecxx.sig** records, there is also an **allrec.sig** file. Click on **allrec.sig** to select it, and then click **Open**.



4 Our signal file containing all of 31 records has been loaded as Ch 4, in between Ch 1 and Ch 2.



Note

You can also save only the visible portion of the channel by checking the box at the bottom of the Save Dialog (just as you can with a non-Multiple Record acquisition).

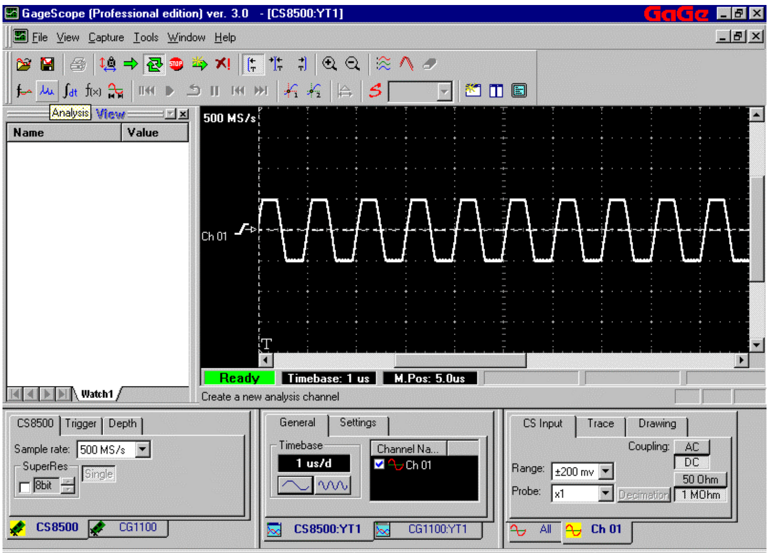
Scope Mode

The automatic sample rate and input range selection feature allows GageScope® to be setup like a conventional digital oscilloscope. When the Scope Mode icon is pressed in or the Scope Mode check box in Preferences dialog box is checked, GageScope® is said to be in the **Scope Mode**. In Scope Mode, GageScope® automatically selects the signal input range in accordance with the selected vertical scale in Volts/division. The sample rate for the input signal is also automatically selected and is linked to GageScope's timebase in seconds/division.

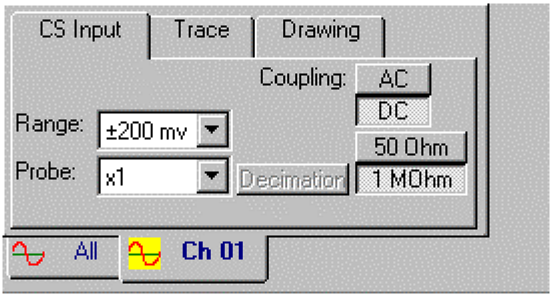
How it works

Automatic Input Range selection

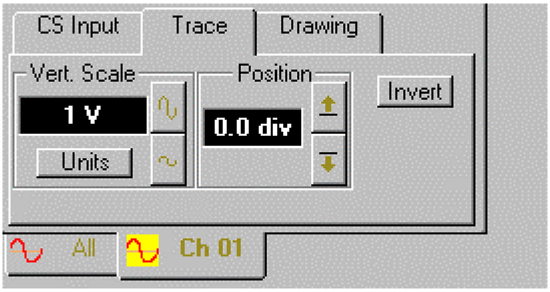
1 The figure below shows a 1 MHz sine wave being captured on Channel 1 of the CompuScope 8500 high speed A/D card.



2 The input signal amplitude is close to one volt. This signal has been clipped since the input voltage range setting is intentionally selected to be ±200mV. This is shown in the Channel Control setting for Channel 1 as follows:

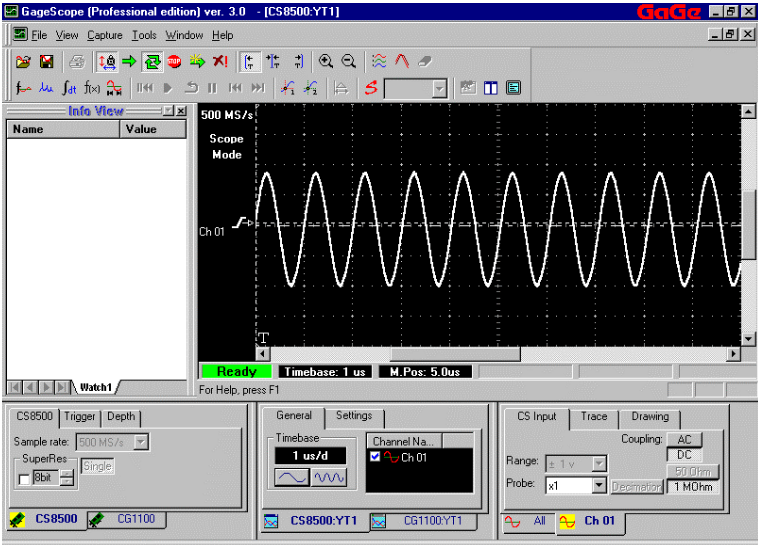


3 The display setting for channel 1 is set to be one volt per division. This can be viewed by clicking the **Trace** tab for channel 1 in the channel control which is located in the bottom right of the GageScope® display. The **Trace** tab property page is shown as follows:



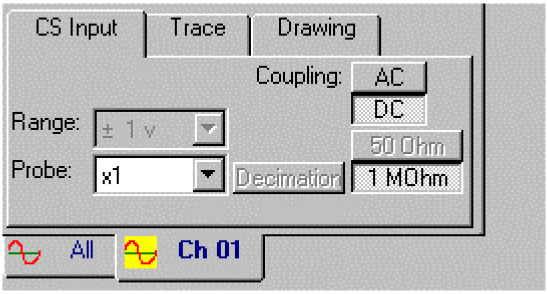
Note that GageScope® is not operating in the Scope Mode as yet.

4 Click on the **Automatic Sample Rate and Input Selection Rate** Icon. You will notice that the **Scope Mode** appears on the display underneath the 500MS/s sample rate text. This implies that the input mode of CompuScope is now the **Scope Mode**. As a result the GageScope® display will automatically select the input range of the input signal according to the currently selected vertical display range (1V/div). You will see the changes in display as follows:

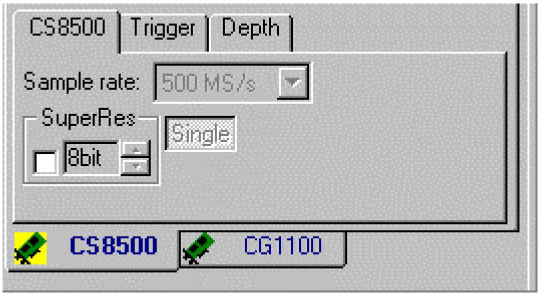


Note that the input signal now no longer appears clipped. ***The input voltage range is automatically adjusted to correspond to the currently selected vertical display range (1V/div***), resulting in displaying the original 1 MHz sine wave.

5 As a result of the CompuScope 8500 being in the Scope Mode, the **Range** setting under **CS Input** tab in the Channel control has been changed from ±200mV to ±1V. Note that the Range text box is disabled so it appears dimmed out. This is done to ensure that the user is unable to change the range while the Scope Mode is active.

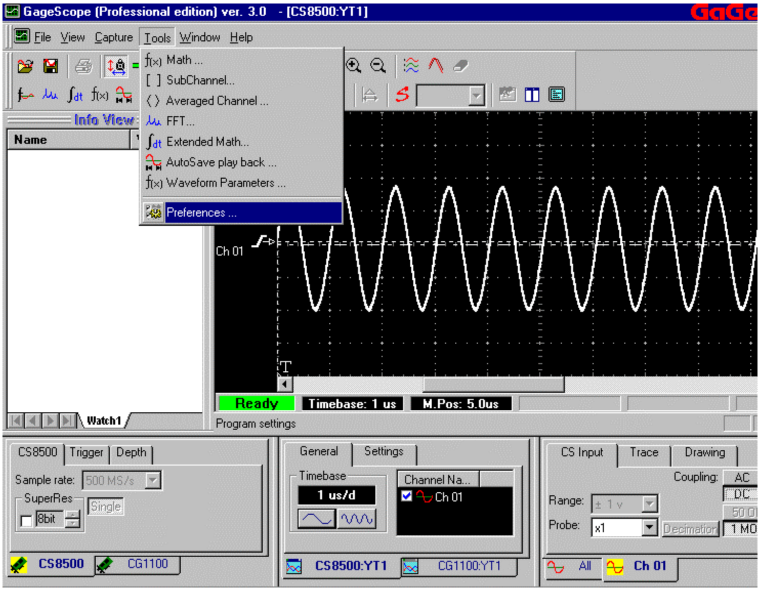


The **Sample Rate** Option under **CS8500** tab in the System Control is also disabled. The CS8500 tab is located in the System Control area in the left bottom corner of the GageScope® display.

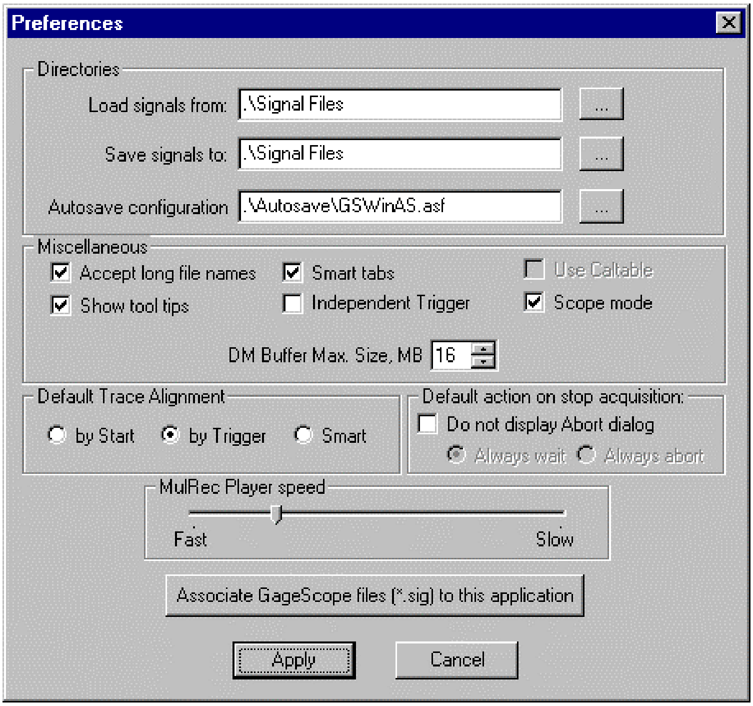


6 The **Scope Mode** can also be activated or deactivated by selecting Scope Mode in the **Preferences** dialog box.

Click **Tools** in the menu bar. The drop down menu appears:

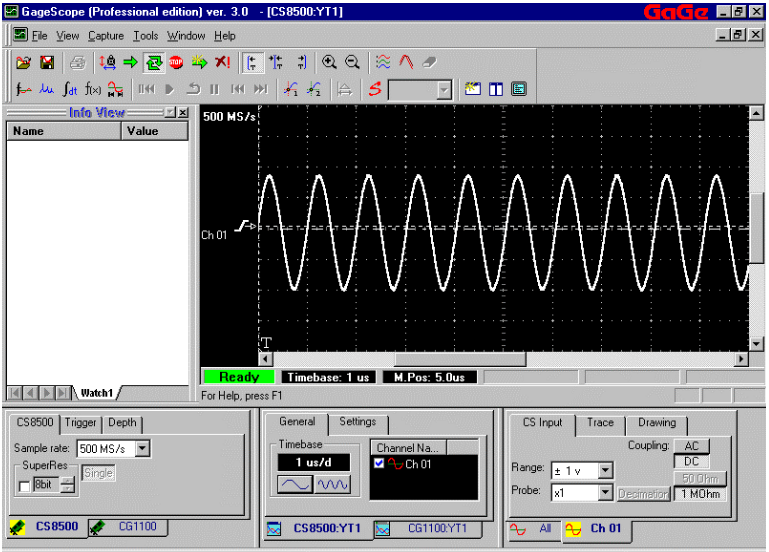


7 Click **Preferences** to view the Preferences dialog box:



The Check mark besides Scope Mode shows that the Scope Mode is currently active. When the Scope Mode is activated by clicking the Scope Mode Icon in the GageScope® toolbar, a check mark is simultaneously placed in front of the Scope Mode in the Preferences dialog box, signifying that the Scope Mode is active.

8 Deselect the Scope Mode by clicking inside the check box. The check mark disappears. Click **Apply**. You will see the following**:**

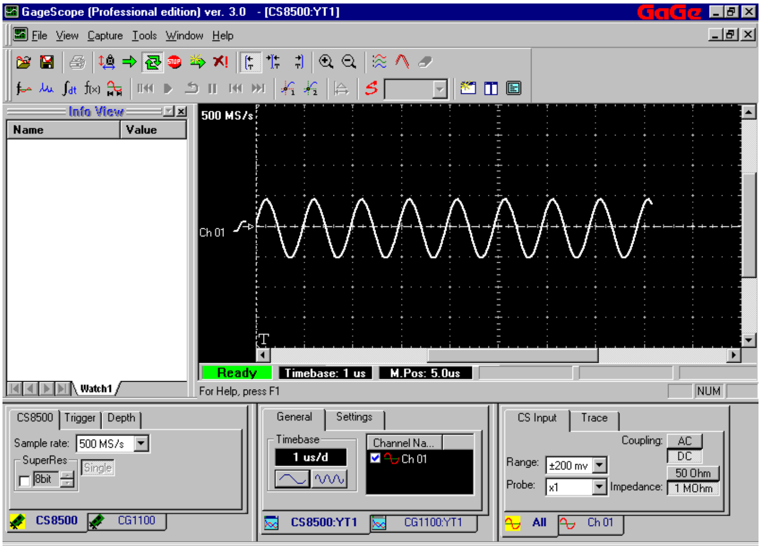


Note that Scope Mode text is now taken off the display indicating that the Scope Mode is no longer active.

Also note that the **Range** setting is not reverted to the original value of ±200mV and the Sample Rate and Range text box controls have been enabled.

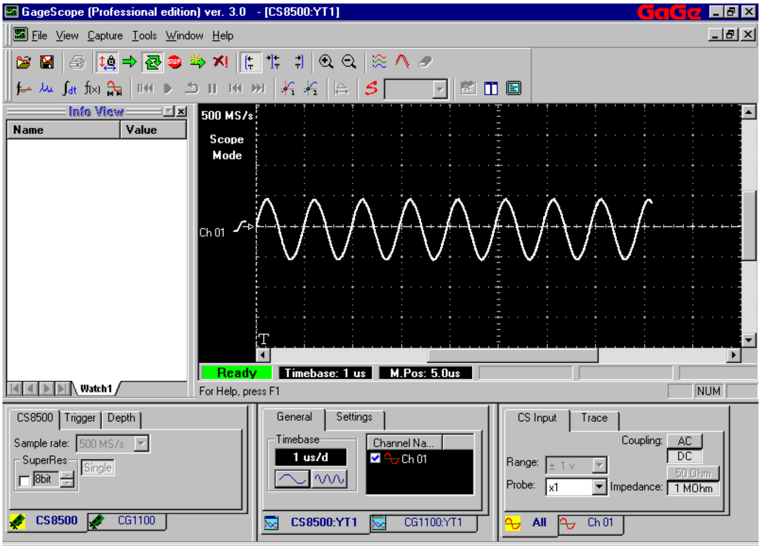
Automatic Sample Rate selection

1 A 1 MHz sine wave being captured on Channel 1 of the CompuScope 8500 high speed A/D card is shown below:

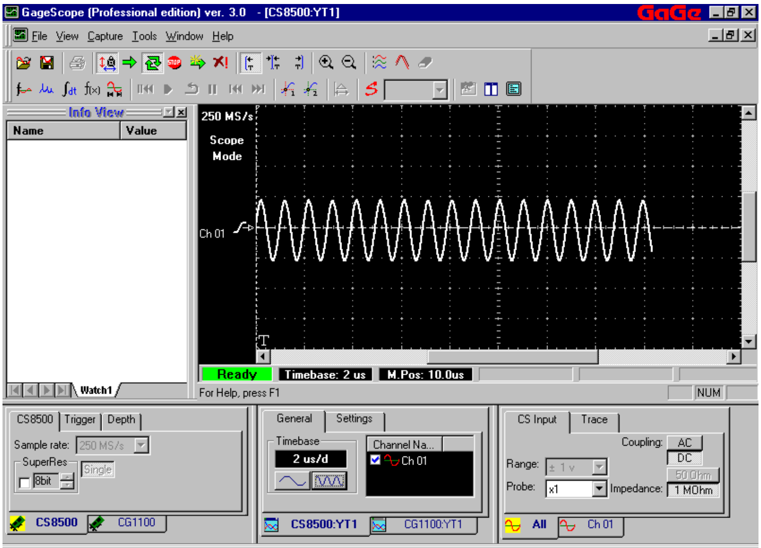


For the signal shown above, the post trigger depth is set to 4096 samples and the input range is ±200 mV. The timebase setting is set to 1 us/d (microsecond per division).

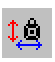
2 Launch the **Scope Mode** by clicking on the Scope Mode Icon in the toolbar. You can also put a check mark in the Scope Mode check box in the Preferences dialog box as explained in the previous section. The text **Scope Mode** appears on the left side of the display, as shown below:



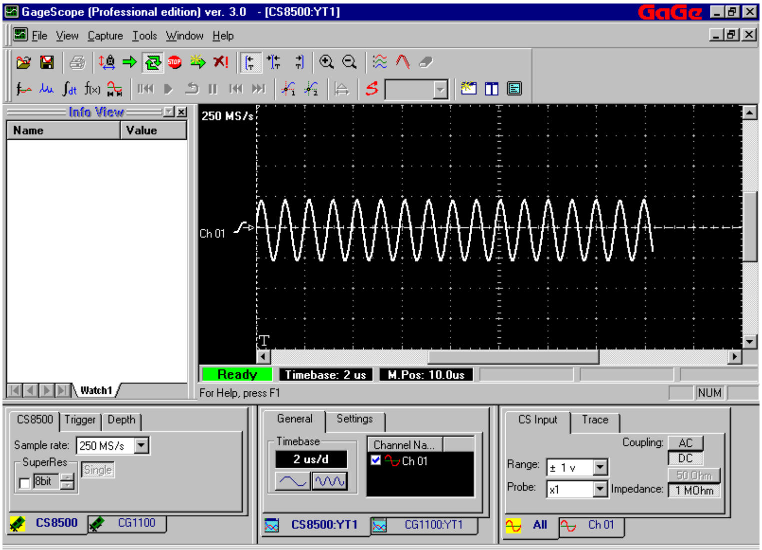
3 **Change the timebase to 2 us/d** by clicking once on the time base compression button,  . The GageScope® display will appear as follows:



Note that the sample rate has changed to from 500 MS/s to 250 MS/s. GageScope® automatically adjusted the sample rate to account for this change in the timebase setting from 1 us/d to 2 us/d. The input range was also changed from ±200mV to ±1V.

4 Click on the **Scope Mode** Icon  to deactivate it. You can also check off the Scope

Mode setting in the **Preferences** dialog box as explained in the previous section. GageScope® will no longer be in the Scope Mode as shown below:



Decimation Mode

Decimation is a feature by virtue of which a user can view large amounts of data within GageScope. The user must first specify the amount of RAM needed for viewing desired amount of data. The amount of data needed per channel is specified in megabytes in the **Deep Memory (DM) Buffer** setting in the **Preferences** dialog box.

The size of the deep memory buffer is part of the actual physical RAM present in your system. The DM Buffer size must be between 8-64 megabytes. Care must be exercised in specifying a value for the DM buffer size.

The Decimation Mode is enabled when the size of acquisition per channel is more than 1/4th the size of the DM buffer size. Let us assume that you have allocated 8 MB for the Deep Memory buffer. In this case, the Decimation Mode will be automatically activated whenever you specify more than 2 MB of total acquisition length in GageScope. The total acquisition length is the sum of the pre and post trigger data size. The Decimation command button will then be enabled, signifying that the Decimation mode is currently active.

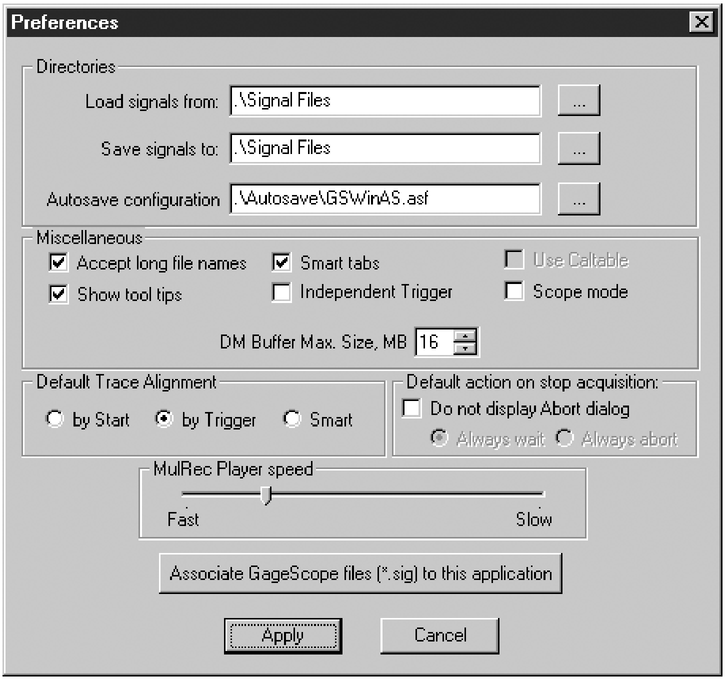
When working in Decimation mode, the user can select the starting sample and the number of samples to be downloaded from CompuScope acquisition memory. Only these data are then decimated and displayed.

Decimation functionality has also been incorporated in the Multiple Record mode. When working in Multiple Record mode, the user may now select the starting record and the number of records to be downloaded from CompuScope acquisition memory. Only these data are then decimated and displayed.

GageScope® uses skip samples, mean, and min max methods of decimating the data. The details of these methods and other decimation features are given in the following example:

How it Works

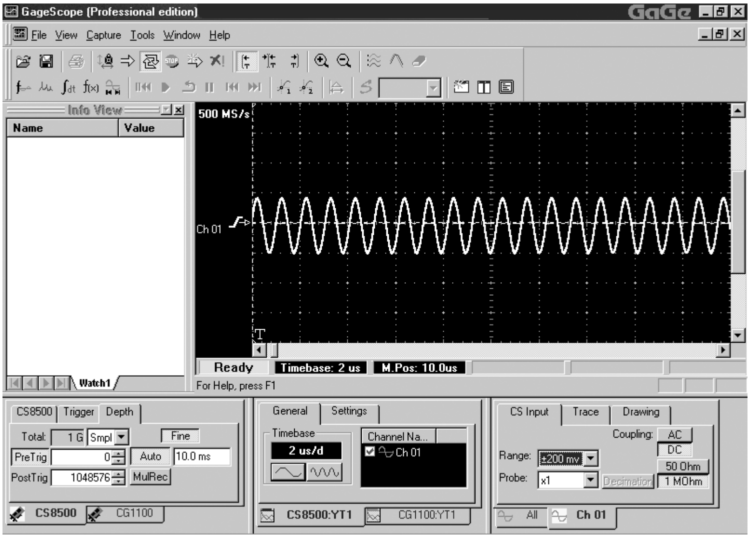
1 Open the **Preferences** dialog box by clicking on **Tools --> Preferences**. The Preferences dialog box will appear as follows:



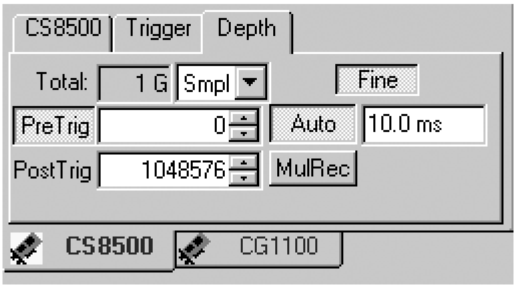
The Deep Memory Buffer size is set at 16 MB as shown in the DM text box. The maximum DM buffer size can be 64 MB.

Click **Apply**

2 Shown below is a 1 MHz sine wave captured by a CompuScope card and displayed as Channel 1.

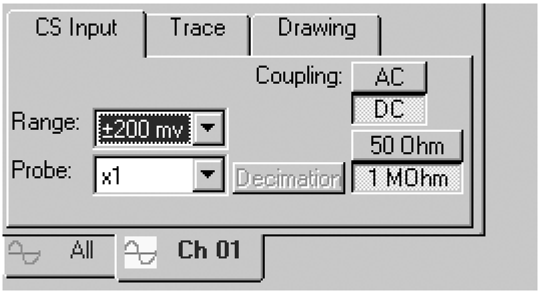


The acquisition post trigger depth is set at 1 Mega Samples (1 MS) as shown in the **Post Trig** text box in the System Control:



Note the total memory available for signal acquisition is 1 Giga Samples (1 G Smpl) as shown in the **Total** text box.

3 Let us now have a look at the Channel 1 property page, which is located in the right bottom corner of GageScope® screen:



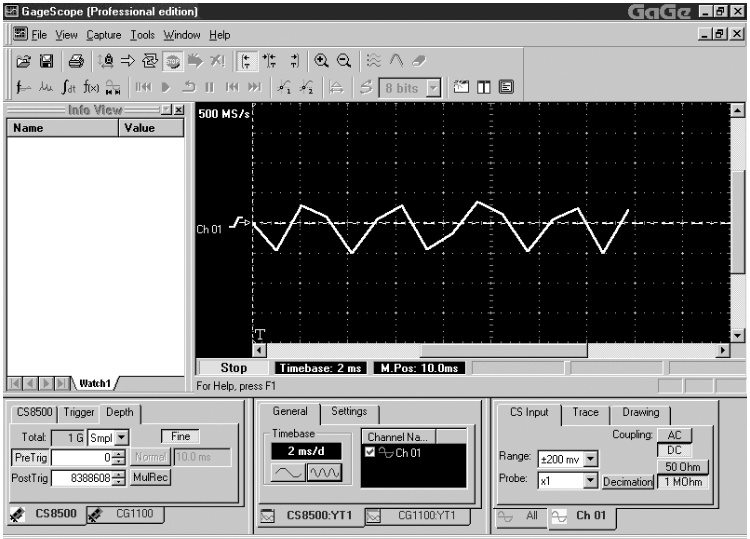
Observe that the Decimation command button is shown disabled implying that the Decimation mode is not currently invoked.

For the settings we have chosen for this acquisition, the criterion for the Decimation mode to be invoked is as follows:

GageScope® will decimate the input data whenever the acquisition length per channel (pre and post trigger) exceeds 4 MB.

Note that the 4 MB is **one quarter** of the DM Max. Buffer Size of 16 MB. The Decimation Mode is enabled whenever the size of acquisition per channel is more than 1/4th the size of the DM buffer size. Since the acquisition depth is currently set to 1 MB for Channel 1, the decimation Mode is not invoked.

4 Change the total acquisition depth for Channel 1 to **8 MB**. As soon as you change the total signal acquisition depth and click anywhere within the display, GageScope® will start decimating the input data and display it on Channel 1 as follows:

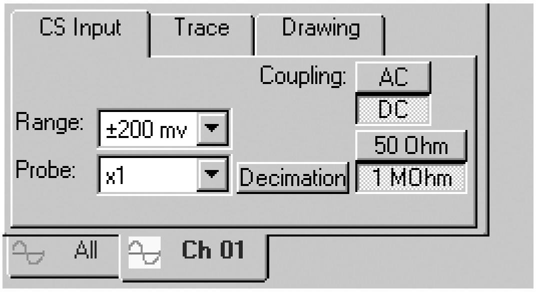


Note that the data on channel 1 is now being decimated. By default, a user gets to view the best representation of the input data.

One of the three methods is employed for decimating the input data: Skip Sample, Mean, or Min Max. A method of your choice can be selected from within the **Decimation** dialog box. The decimation dialog box opens up whenever the Decimation command button is clicked. Make sure that the Decimation button is enabled before you click on it.

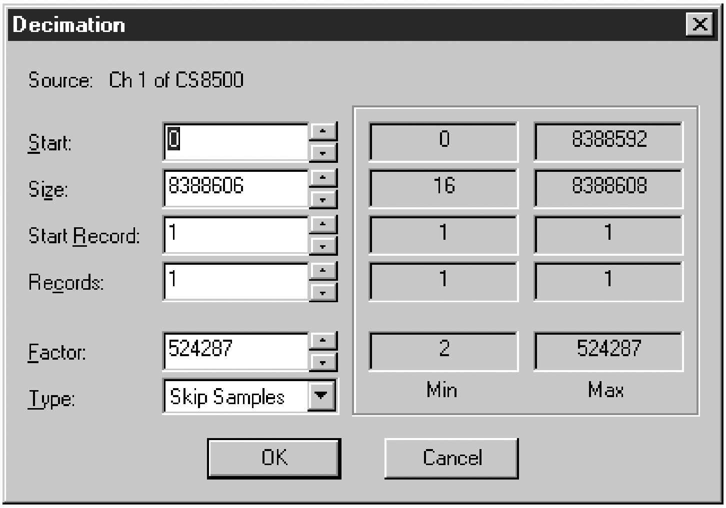
There are several other useful features available within the decimation dialog box. A detailed descriptions of these features and their use is given as follows:

5 Click on the **CS Input** tab of Channel 1 control. The property page appears as follows:



Note that the Decimation command button is shown enabled.

6 Click on **Decimation** button. The **Decimation** dialog box opens as shown below:



7 The **Decimation** dialog box options are as follows:

Source	Shows the channel number being decimated and the name of the CompuScope card. It can also be the name of the file, if the signal is loaded from a file.
Start	The starting sample of the data to be displayed. Note that the current range of data values spans from the Minimum Sample value of zero to the Maximum Sample value of 8388592. These are the values in the corresponding row as shown in the Min and Max columns on the right side of the text box. **Min** isthe minimum number that a user can select for the corresponding feature. For example the **Minimum Factor** a user can select is **2** as shown in the Decimation dialog box. **Max** is the maximum number that a user can select for the corresponding feature. For the current setup, **524287** is the **Maximum Factor** a user can select. The user can select any number from within this range. Since the maximum size of the signal to be displayed in the Decimation Mode is 16, **the Maximum Start Sample Number (Max.) is set to be signal acquisition depth minus 16.**
Size	The total number of samples to be displayed. The number in the size text box refers to the total length of the signal to be displayed. **Note that no less than 16 samples can be displayed on the decimated channel.** The maximum number is the acquisition depth per channel specified in the Pre and Post Trig text boxes in the System Control. In our case the total acquisition depth is 8 MB, which turns out to be 8388608 as shown in the corresponding column.

Start Record The number of the starting record to be displayed in a Multiple Record acquisition. Presently, there is only one single record being captured. When the Multiple Record Mode is invoked, you will see the Min and Max numbers in the corresponding columns.

Records The total number of records to be displayed in the Multiple Record Mode.

The Maximum number of records equals the total CompuScope memory available for acquisition divided by the acquisition depth per channel.

The minimum number of records to be displayed is obviously one.

Factor This is the Compression Ratio. Compression ratio can be chosen from the available range of minimum and Maximum values.

Decimation is a process of compressing the data so that it can be displayed in GageScope.

Note that the best compression ratio corresponds to maximum number in the corresponding row in **Max** column. In the present example, you can see that the factor is chosen to be the Max by default.

The minimum compression ratio will display the maximum number of samples. This will give the best possible representation of the entire acquisition.

The best compression ratio is the one that retains key features of the signal to be displayed. This number is left to the user's discretion.

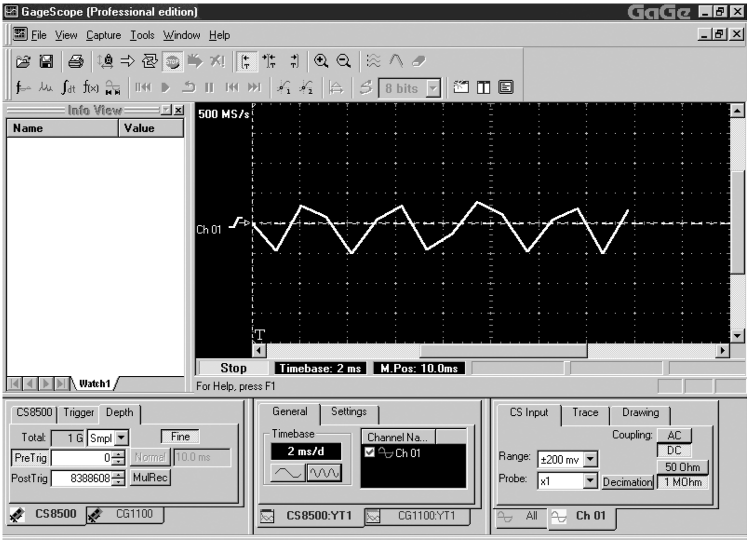
Type There are three types of decimation methods available:

Skip Samples: Skips samples automatically according to the **Factor** (Compression ratio)

Mean: Takes the mean value of a group of samples. The size of this group is governed by the number entered as the **Factor**.

Min Max: Depending on the **Factor**, takes the Minimum and Maximum values of a group of samples. These values are then displayed in a vetical line connecting the minimum value to the maximum value.

8 Getting back to our example, click **OK** in the Decimation dialog box. You will get back to the signal being acquired and decimated:



According to the current Decimation settings, the Skip Sample method is used to decimate the input data shown in the figure above.

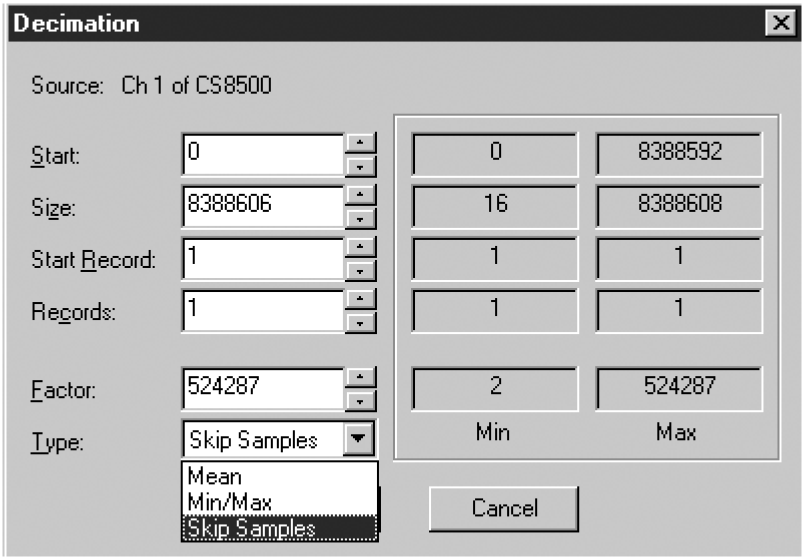
9 There are two other methods of decimating the data: MinMax and Mean.

The MinMax Method

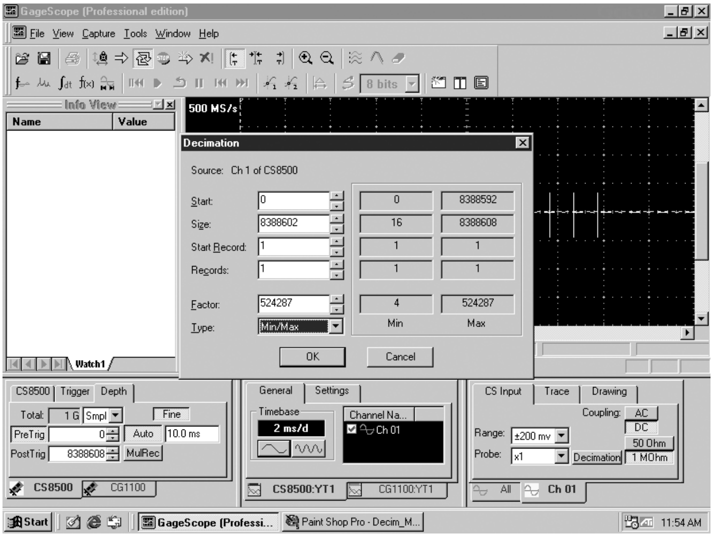
In MinMax method, as the first step, the entire acquisition is divided into groups of samples. Each group contains fixed number of samples. The number of groups and the size of a single group depend on the number entered in the **Factor** text box in the Decimation dialog box. User has no direct control in specifying the number of samples to be grouped.

Next, the minimum and the maximum value from each group is extracted and saved. Fro each group , a vertical line is then displayed. The two end points correspond to the minimum and maximum value of the selected group. There will be as many vertical lines as the number of groups.

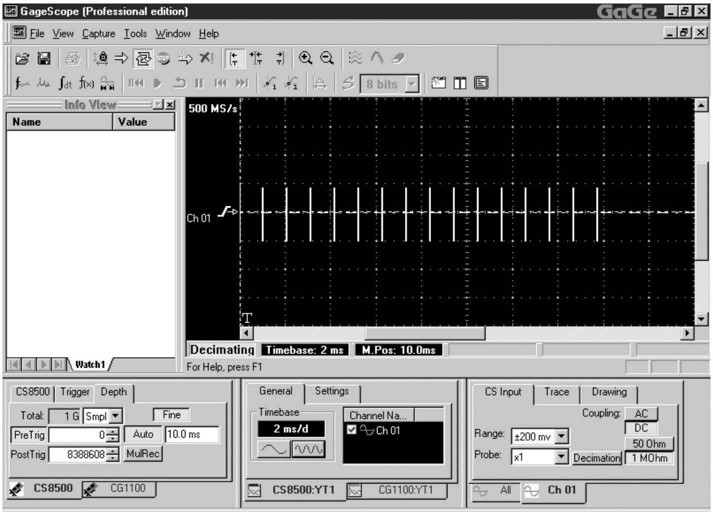
10 In order to use the **MinMax** method of decimating the data, click on the decimation button in the channel control. The decimation dialog box appears. Click on the Type drop down menu button. The **Type** menu will appear:



11 Click **MinMax** from the list. MinMax shows up in the Type text box:



12 Click **OK**. You will see channel 1 decimated input data as follows:



Observe that the data is displayed as vertical lines. The number of lines represent the size of the decimated data to be displayed. Divide the **Size** of the data to be displayed by the **Factor** to get the size of the decimated data.

In the current example, we have

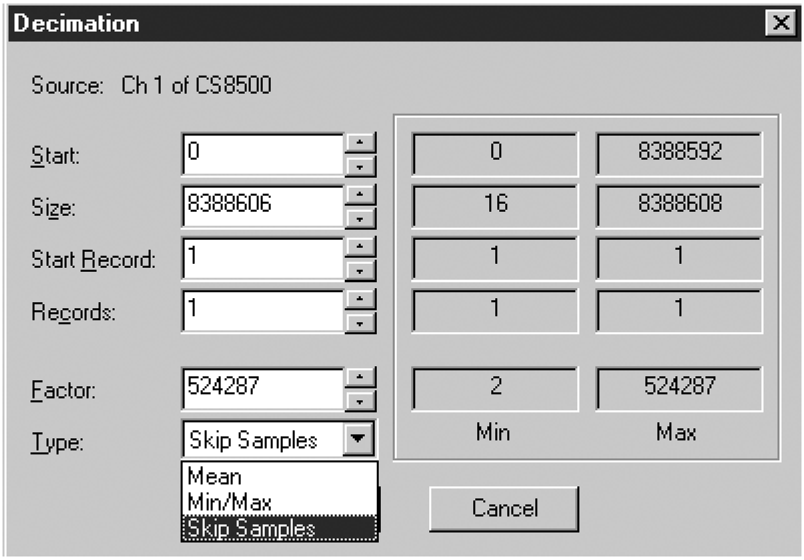
Size (data to be displayed) = 8388606
Factor = 524287
Decimated DataSize = **Size/Factor**
= 8388606/524287
= **16**

Now count the number of vertical lines. You will notice that there are 16 shown in the display above.

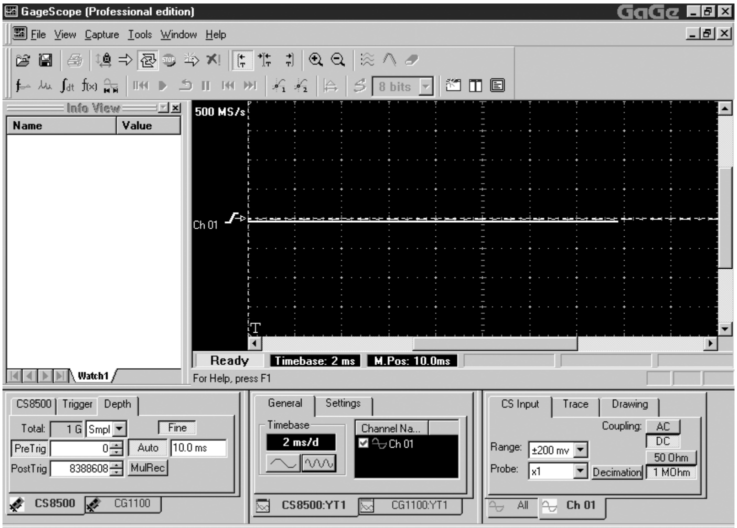
13 **The Mean Method**

In Mean method, same as in the MinMax method, the entire acquisition is divided in groups of fixed length. The mean value for the samples in each group is computed. These mean values for the entire acquisition are then displayed as the decimated data.

14 In order to use the **Mean** method of decimating the data, click on the decimation button in the channel control. The decimation dialog box appears. Click on the Type drop down menu button. The **Type** menu will appear:

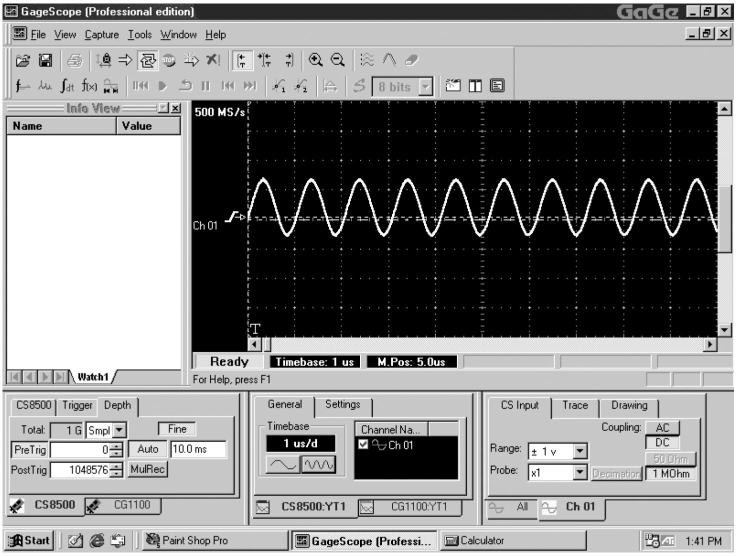


15 Click **Mean** from the list. Mean shows up in the Type text box. Click OK. You will the decimated signals as follows:

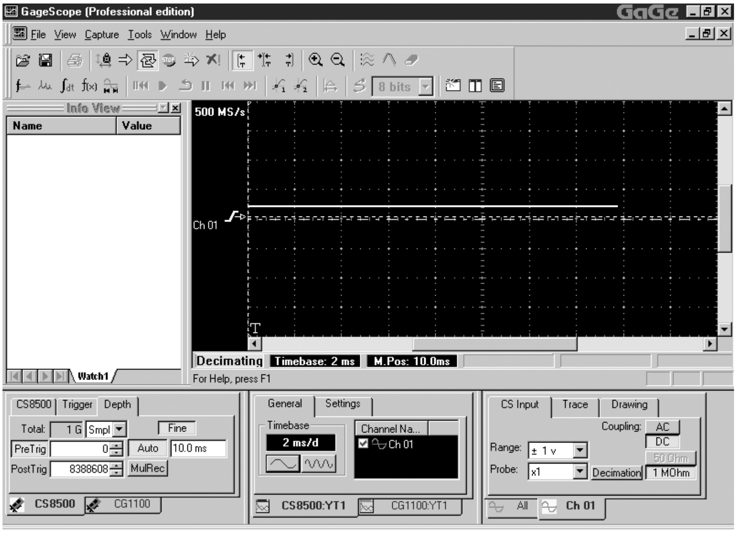


Observe that the decimated data according to the Mean method indicates that there is a small neative bias in the signal. Thus the decimated data canbe used to observe and even measure trends in the acquired signals.

16 In order to further elaborate the Mean method of decimation, shown below is a sine wave with a positive DC offset added to it:



17 Follow the steps in the previous sections to obtain the decimated data emplying the Mean method. The decimated data is shown as follows:



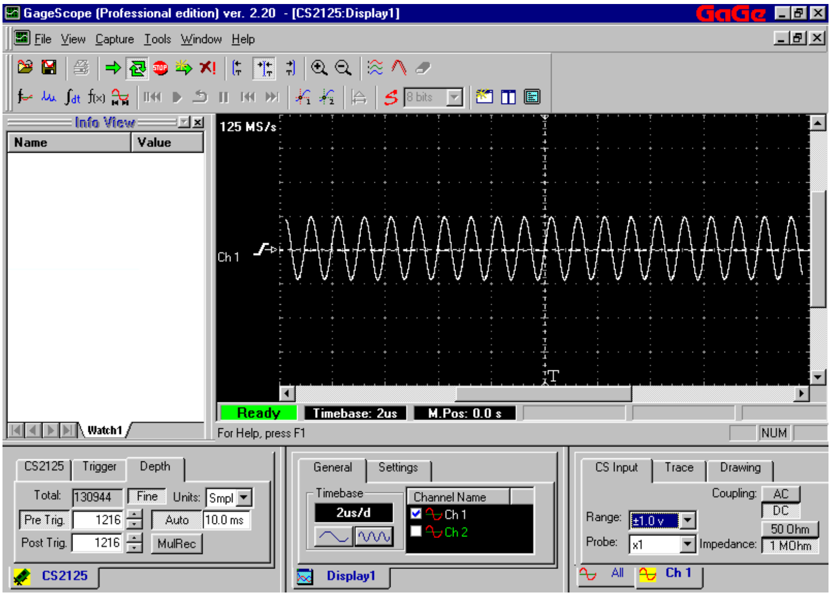
You can see that the positive DC offset now appears as the decimated data.

Tutorial 5: Working with Subchannels

The Subchannel feature is of tremendous use in applications such as ultrasonics where the reflected echoes play a key role in identifying several key features of the medium.

Illustrative Example

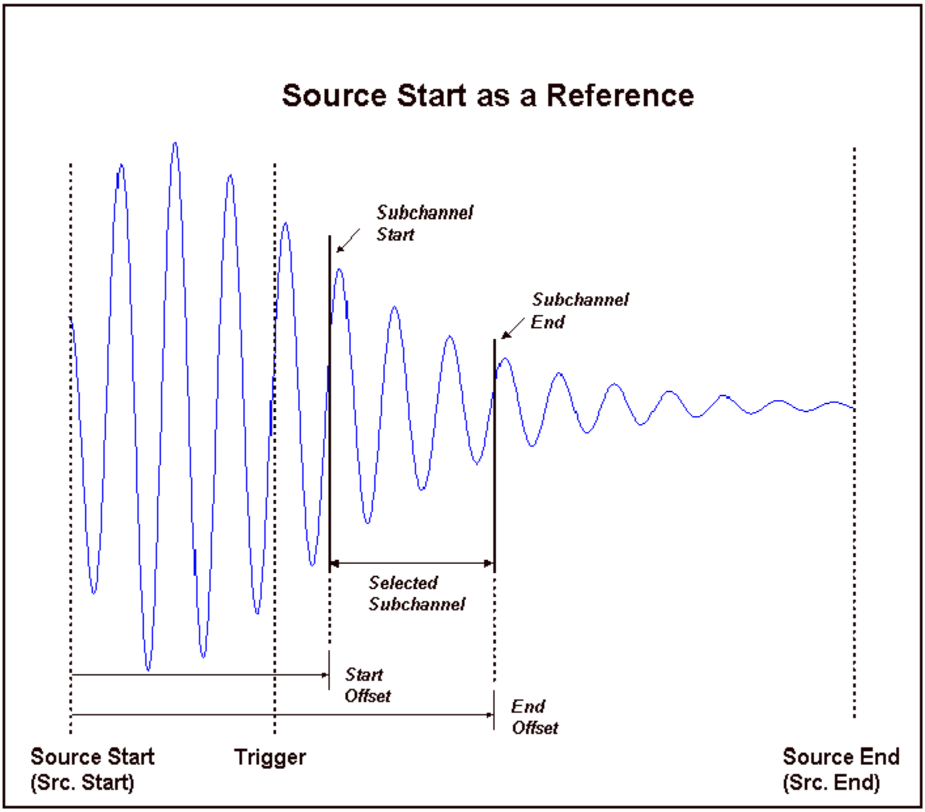
1 A 1MHz sine wave sampled at 125MHz is shown below:



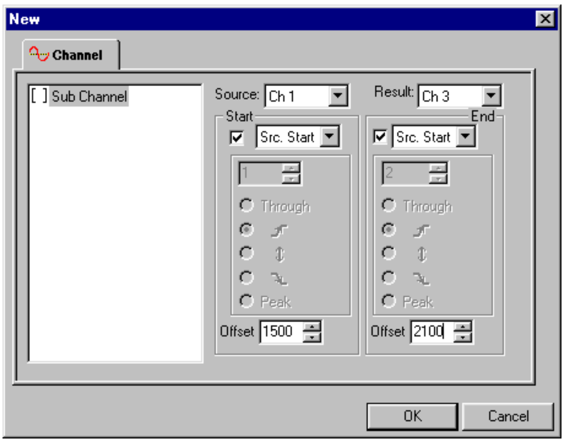
The acquisition contains 1216 pre-trigger and 1216 post trigger sample points. There are 10 cycles in the pre-trigger portion and 10 cycles in the post-trigger portion of the signal.

We will choose the portion of the signal between the sample point 1500 and 2100 such that the entire subchannel is extracted from the post-trigger part of the source channel 1. This means that the Start Offset is 1500 while the End Offset is selected to be 2100. Channel 3 is chosen as the resulting subchannel.

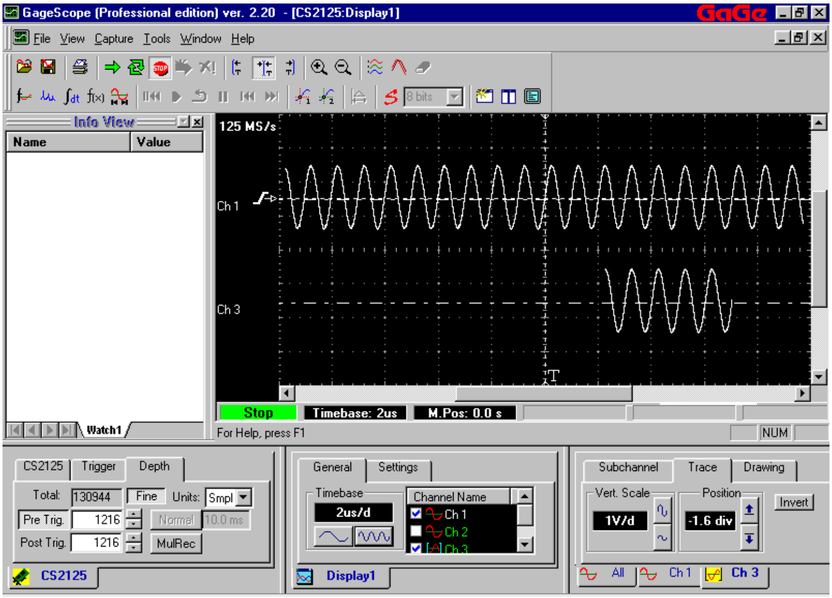
Figure below shows the acquisition parameters:



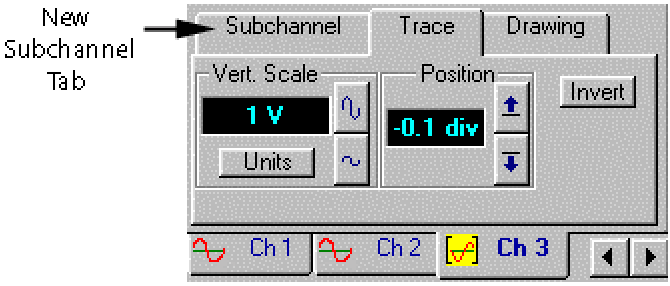
2 Settings in the subchannel window appear as:



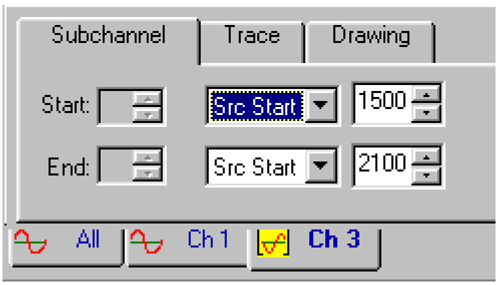
3 Click **OK**. The window with the Subchannel appears



4 Note that **Subchannel** now appears as a tab in the Channel Control area



5 Click on **Subchannel** to further explore the properties of the selected Subchannel

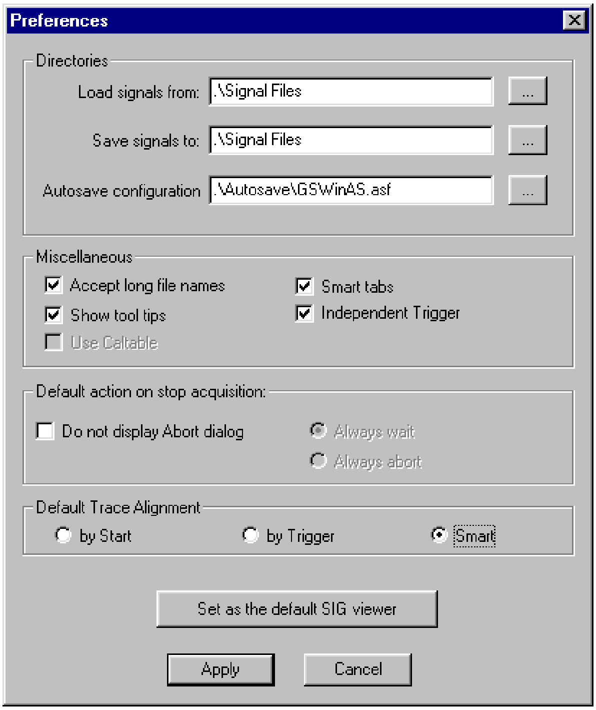


Note that the **Start** and **End** options are available in case the Subchannel is obtained using the features of the source (Trough, Rising edge, Falling edge, and Peak)

The drop down menu shows that the Subchannel starts at sample point 1500 and ends at sample point 2100 with reference to the first point of the source channel i.e. **Src Start**

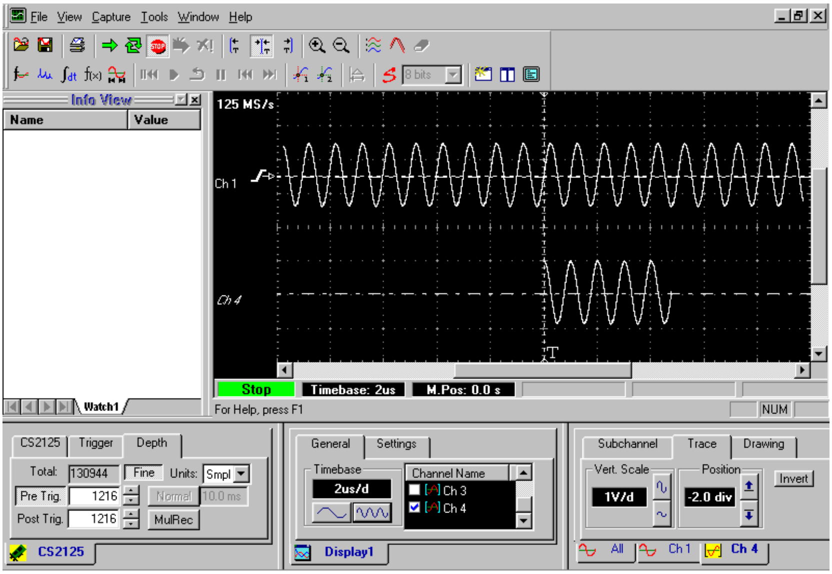
6 *The Subchannel can be aligned with the **Trigger**.*

From the **Tools** drop down menu, select **Preferences**. Now Select **Smart** in the **Default Trace Alignment** Box



Click **Apply**

7 ***Repeat steps 1 – 3***. The Subchannel appears aligned with the trigger position

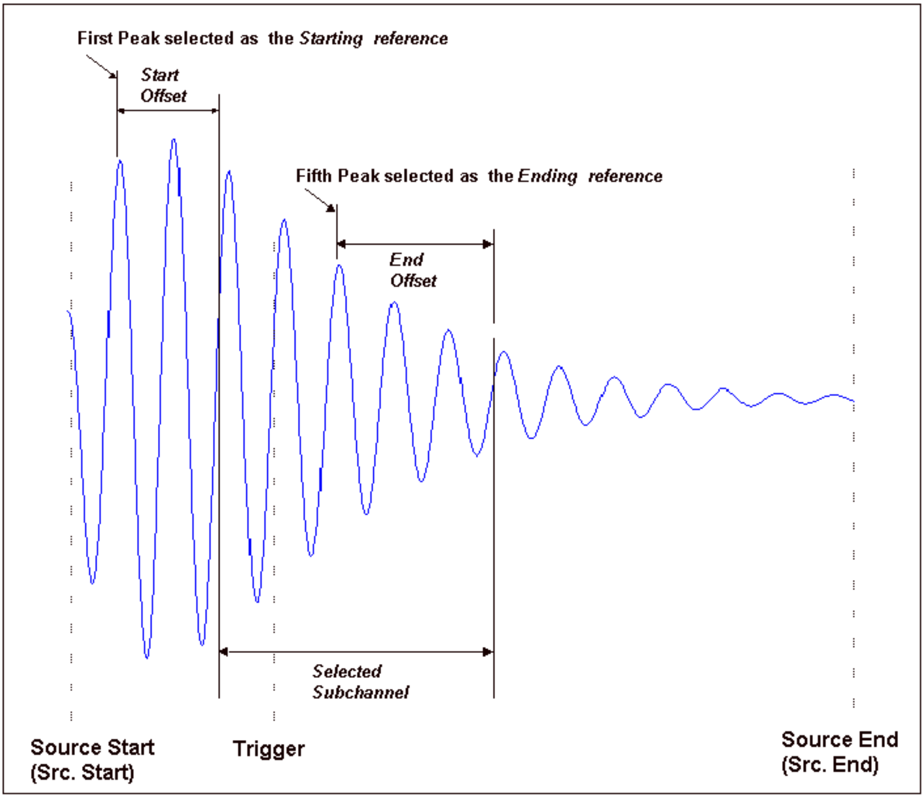


Note that the channel 3 appears as channel 4 as selected while repeating steps ***1 – 3.***

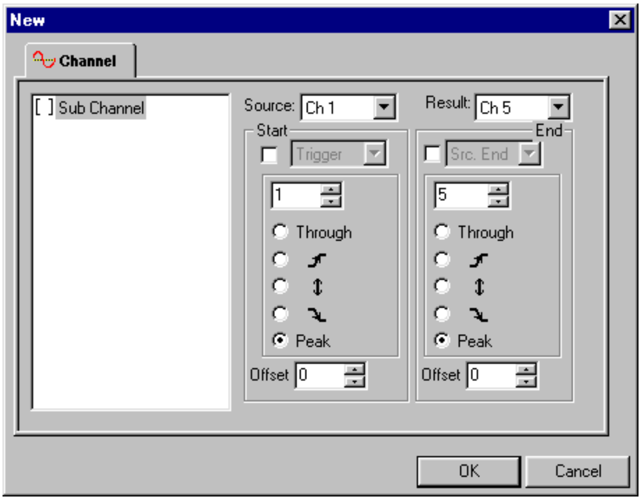
The Subchannel aligns itself with the trigger also when the **by Start** option is selected in the **Preferences** dialog box. Refer to page C-30 of the Reference section for more details on Preferences.

8 Follow the procedure outlined below in case you wish to obtain the Subchannel using one the feature-based references (Peak, Rising edge, Falling Edge, Either Edge, and Trough) of the source signal.

Select the Subchannel between the first and the fifth peak of the original signal. There are no offsets specified for the present example:



9 Click on **Tools** in the Toolbar. Select **SubChannel**



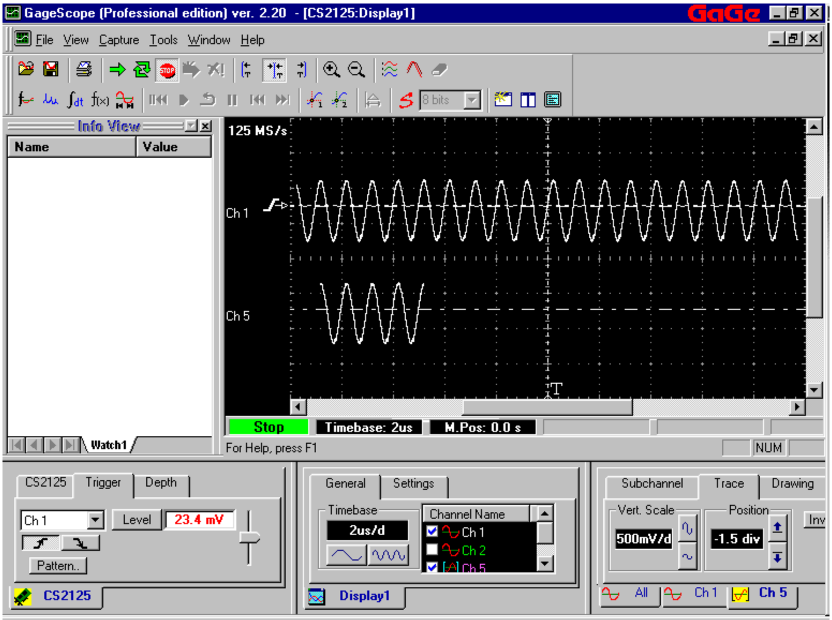
It can be seen that Channel 5 has been selected as the Subchannel. The Subchannel will conform to the following:

Sarting point of the Subchannel = First Peak + Offset

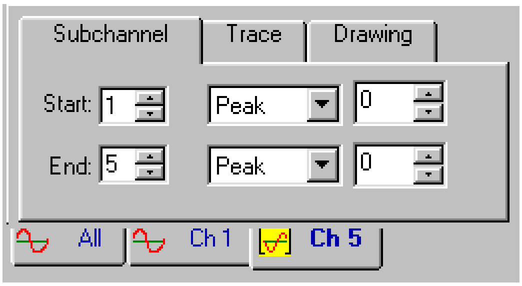
Ending point of the Subchannel = End Peak + Offset

In our case, since no offsets are applied, the Subchannel starts at the first peak and ends at the fifth peak.

10 Click **OK**. The Subchannel appears as Channel 5



11 Click on the **Subchannel** tab in the Channel Properties dialog box



The **Start** and the **End** options are now available.

Tutorial 6: Arbitrary Waveform Generator

GageScope® has the unique ability to connect to Arbitrary Waveform Generators (CompuGen cards) sold by Gage Applied Technologies Inc. The primary step in achieving this connectivity is to create a rich set of waveforms within the GageScope® software. The waveforms created in GageScope® can then be uploaded to an appropriate CompuGen card.

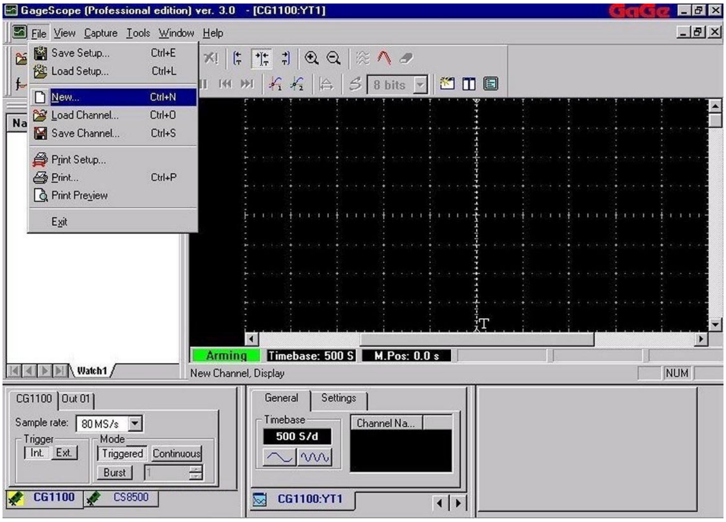
There are three different ways to create arbitrary waveforms from within GageScope®:

- Equation
- Predefined
- Standard Waveforms

In this tutorial, we will go through the unique features of each of the methods outlined above. It is assumed that you have launched GageScope® prior to starting this tutorial.

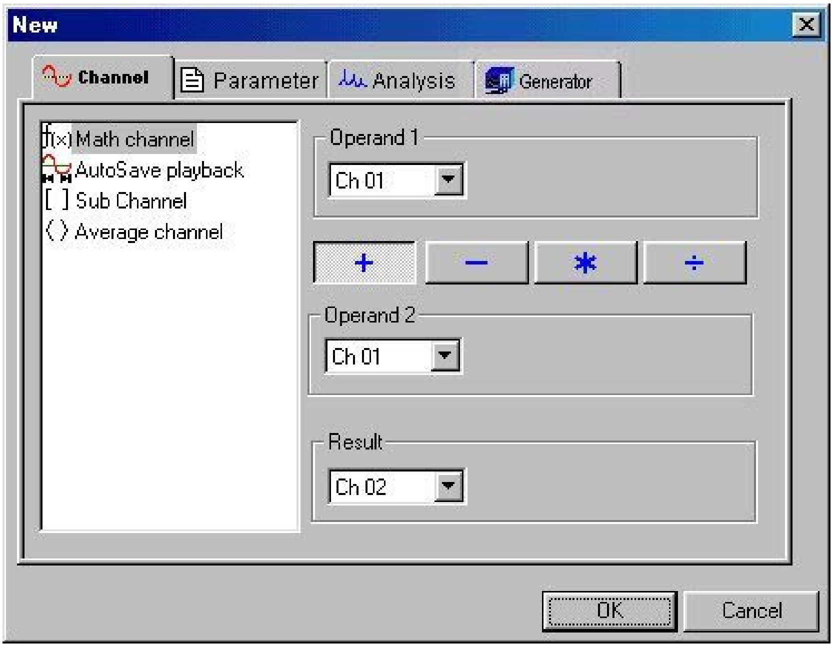
Waveform Generation using Equation as an input

1 Click **File** menu from the menu bar.

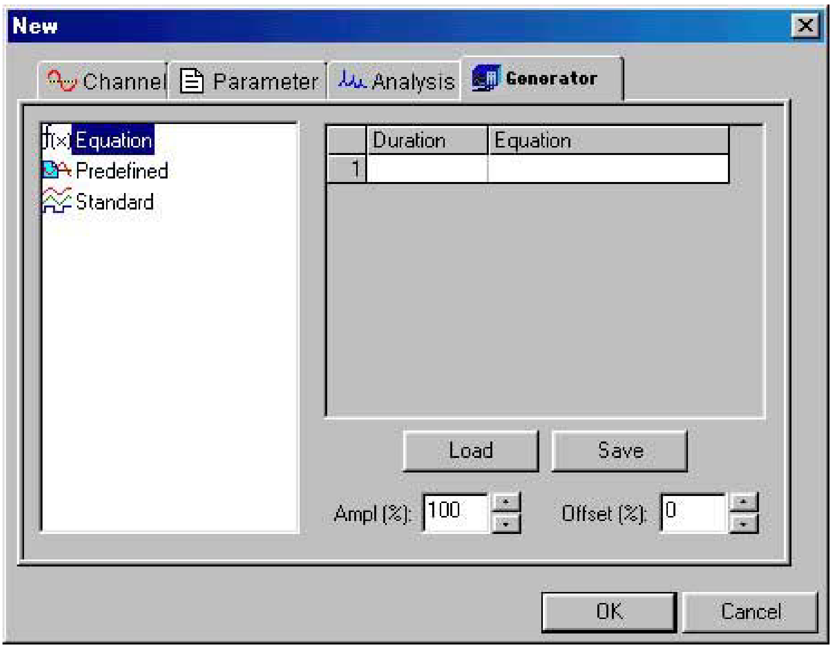


It can be clearly observed that the Channel Control at the bottom right hand of the GageScope® screen is blank. This is due to the fact that no channel is currently active.

2 Select **New** from the drop down menu. You will see the following window:

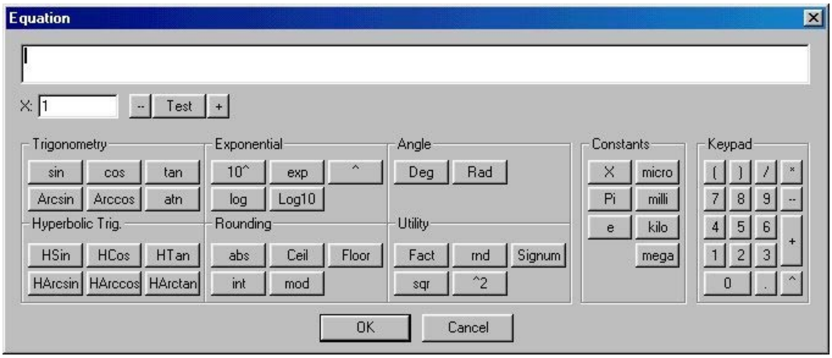


3 Click on the **Generator** tab. You will notice that the Equation text is highlighted as a default.



Using controls in window shown above, users can type in an equation to generate a specific signal.

4 Click inside the text box under the label **Equation**, brings up the Equation editor dialog box as shown below:

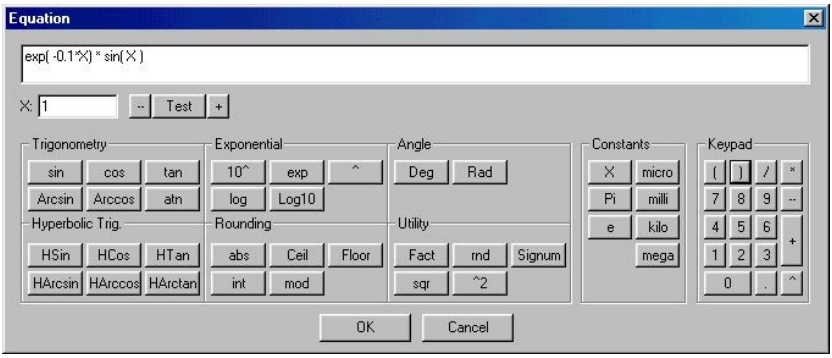


There is an entire range of functions provided here. See the **Reference** Section of this manual for a complete listing of these functions.

We will attempt to create a simple equation at this stage. Later on we will show that more complex signals can be created using controls provided in the Equation editor dialog box above.

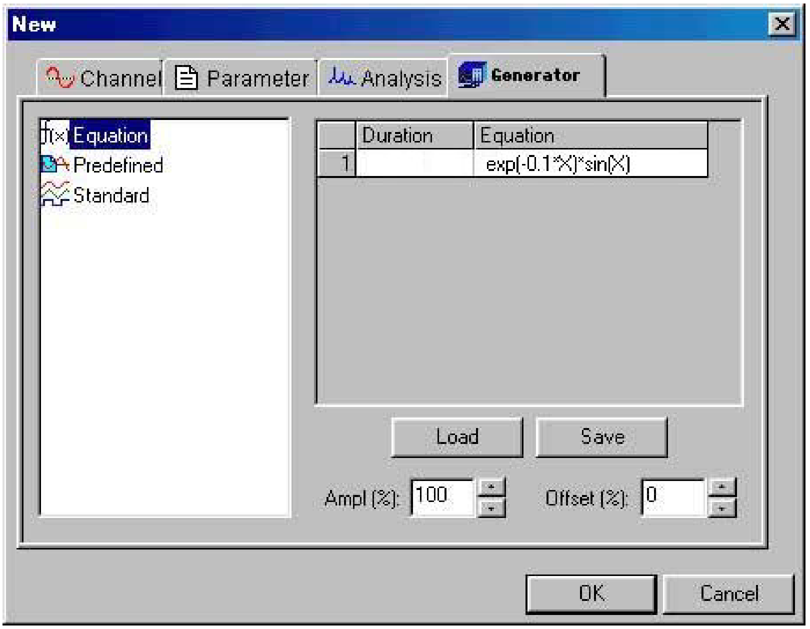
5 Let us now create the equation for generating a constant frequency sine wave.

Type **Exp[-0.1*X]** * **Sin[X]** in the text box either using keyboard or the buttons provided in Equation Editor dialog box. After entering the complete equation,

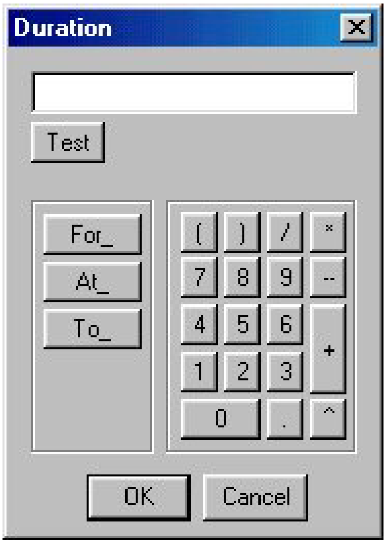


Click **OK**.

6 **New** dialog box appears again. Notice that the equation typed in the previous step, is shown in the text box beneath the **Equation** label.

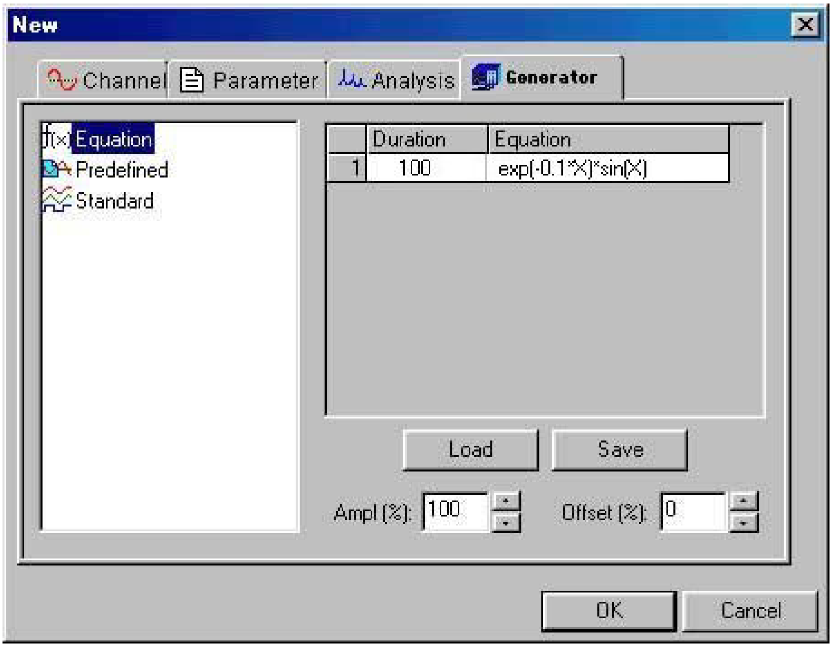


7 Click inside the text box on the left of the typed equation. This is the text box right under the **Duration** label.



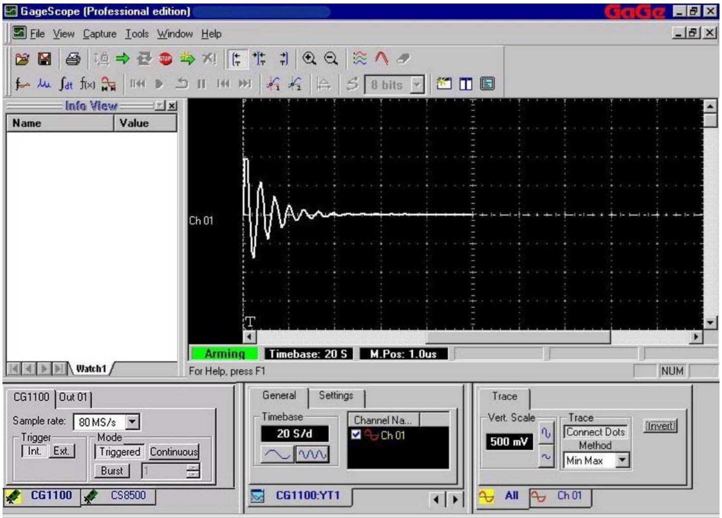
Type in 100 in the text box. These are the total number of samples for the waveform. Click **OK**.

8 The New dialog box appears, showing the duration to be 100 samples.



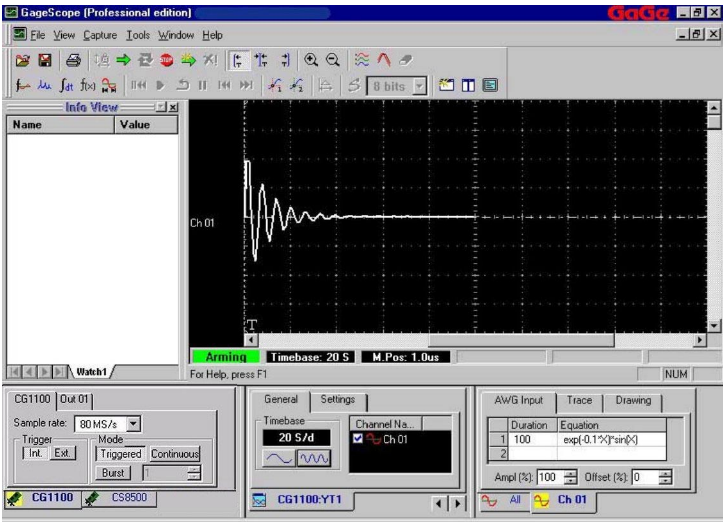
Click **OK**.

9 Damped sinusoid will be displayed in GageScope® as follows:



This signal can be saved as a sig file to be uploaded to an AWG.

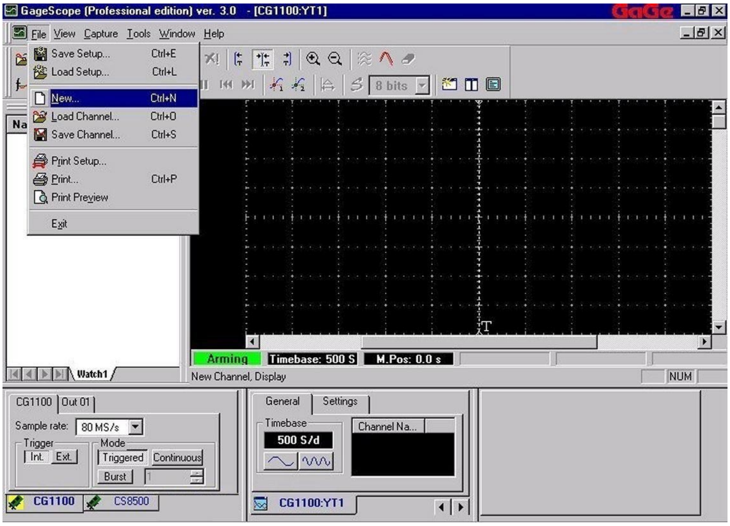
10 Click on **Ch 01** tab located in the bottom of Channel Control display. You will notice that **AWG Input** tab appears in the Channel Control as shown below:



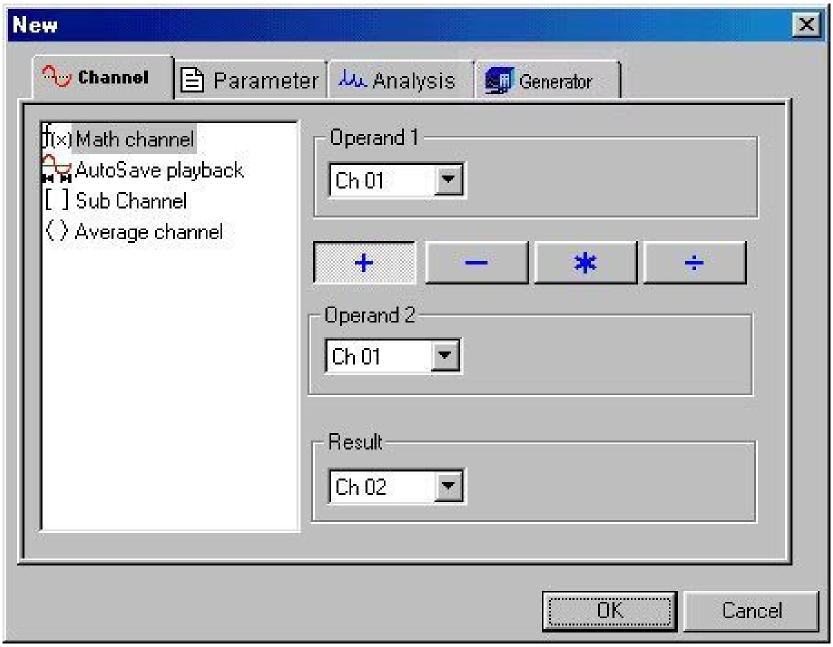
The parameters can be modified using controls in the channel control above.

Predefined Waveforms

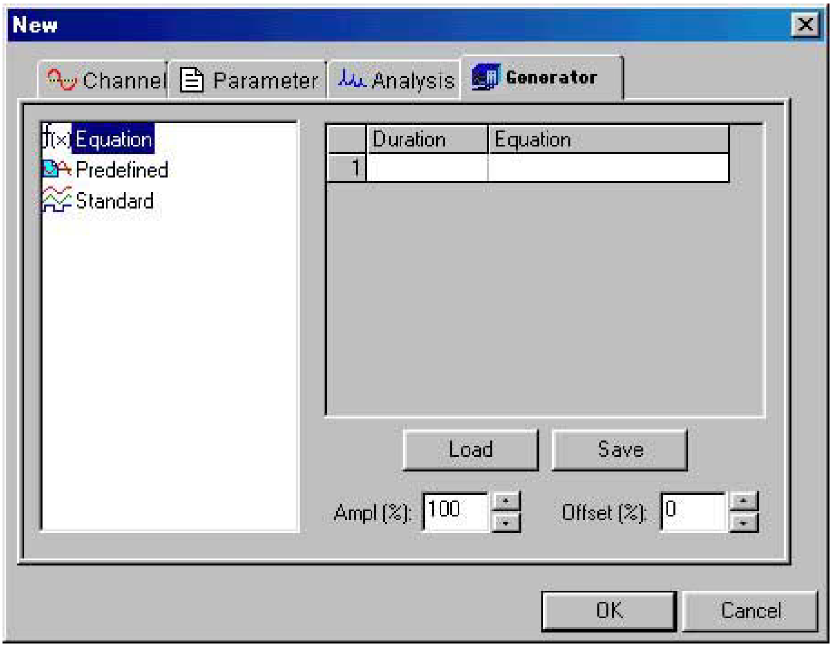
1 Click **File** menu from the menu bar.



2 Select **New** from the drop down menu. You will see the following window:

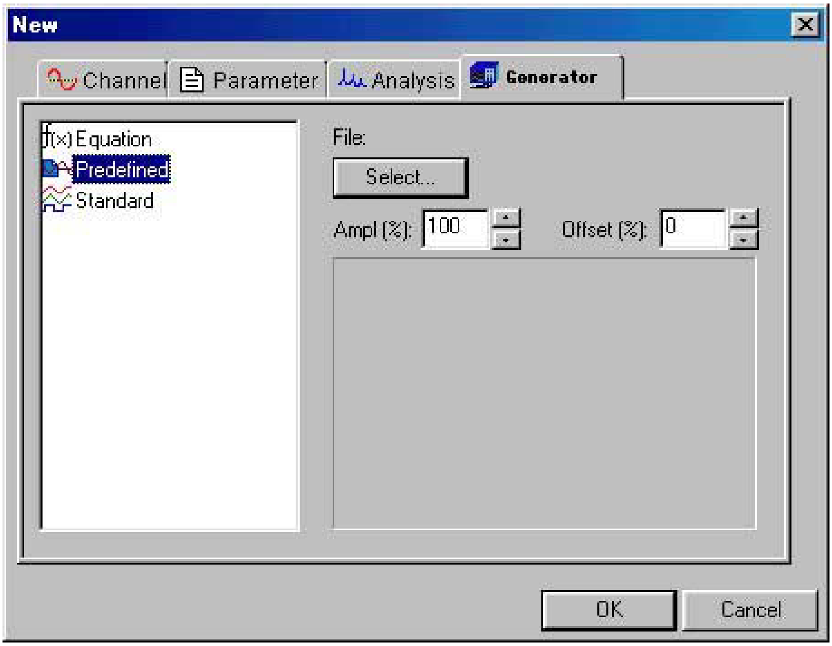


3 Click on the **Generator** tab. You will notice that the Equation text is highlighted as a default.

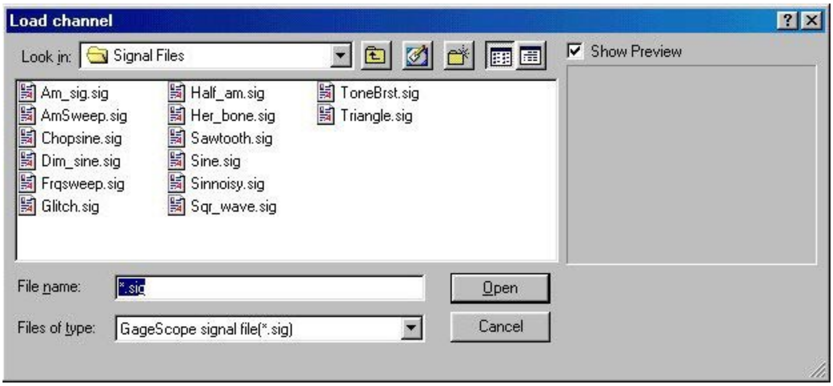


Using controls in window shown above, users can enter an equation to generate a specific signal.

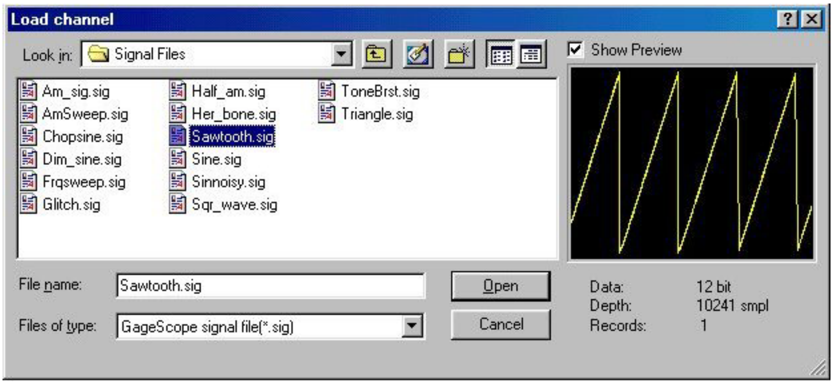
4 Click **Predefined** from the list in the dialog box. Click **OK**.



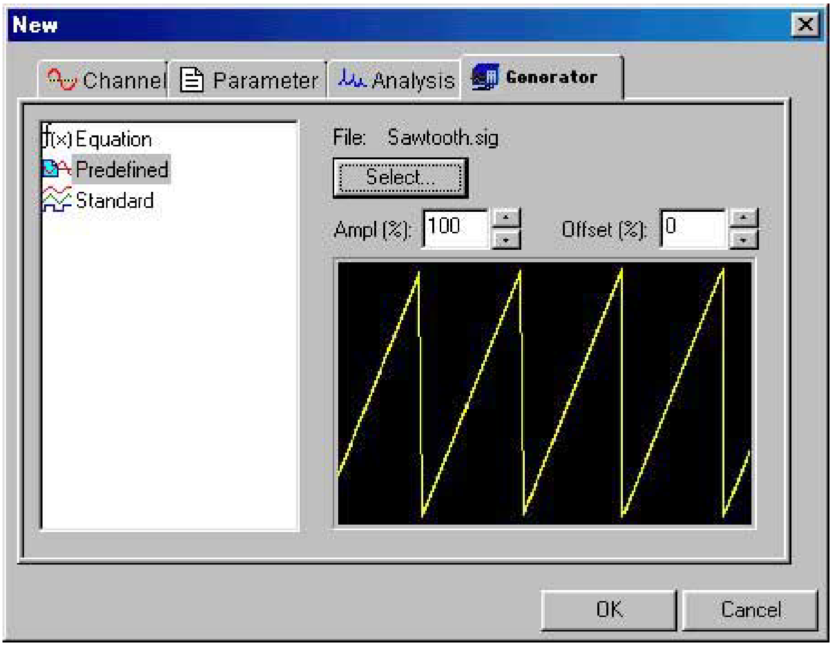
5 Click **Select**. You will see the **Load Channel** window as follows:



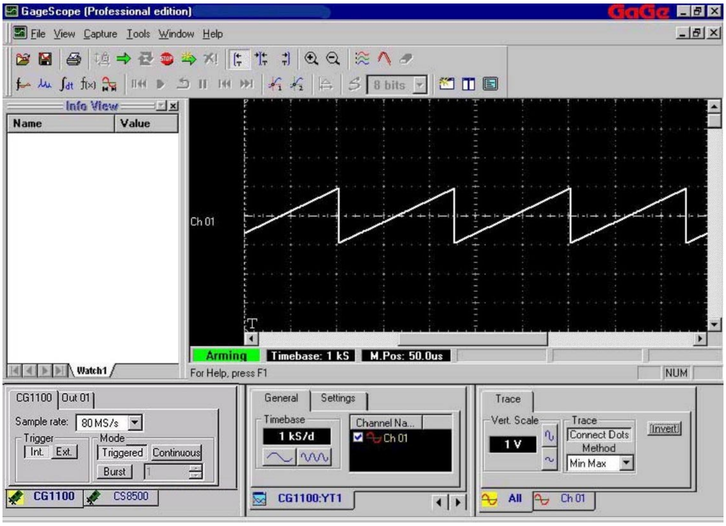
6 The dialog box shows that there are several types of signals stored for use. Click on **Sawtooth.sig**. The signal appears in the Preview window.



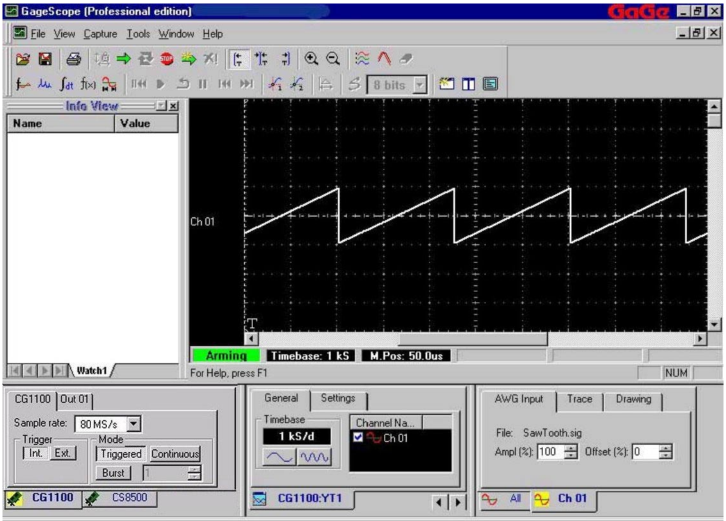
7 Click **Open**. The predefined signal appears in the New window as a Preview.



8 To load the Sawtooth signal file in GageScope, click **OK**. If you do not wish to load the signal at this point, click **Cancel**.

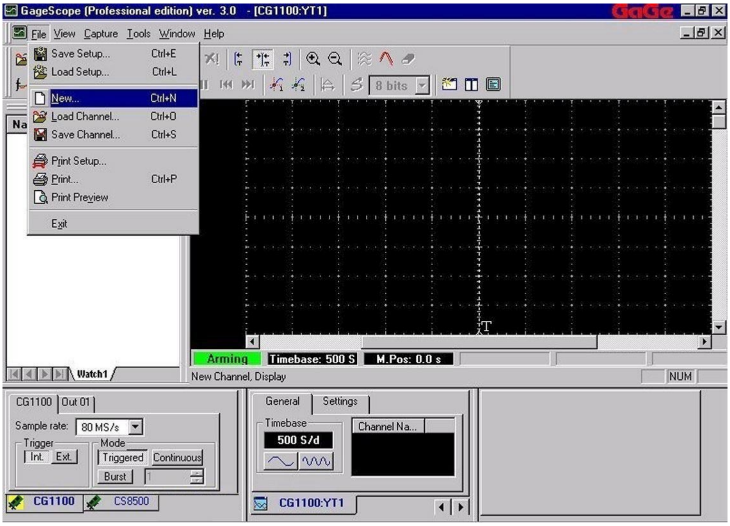


9 Clicking **Ch 01** tab in the channel control, brings up the **AWG Input** tab at the top of the control. Click **AWG Input** tab to view the associated property page as follows:

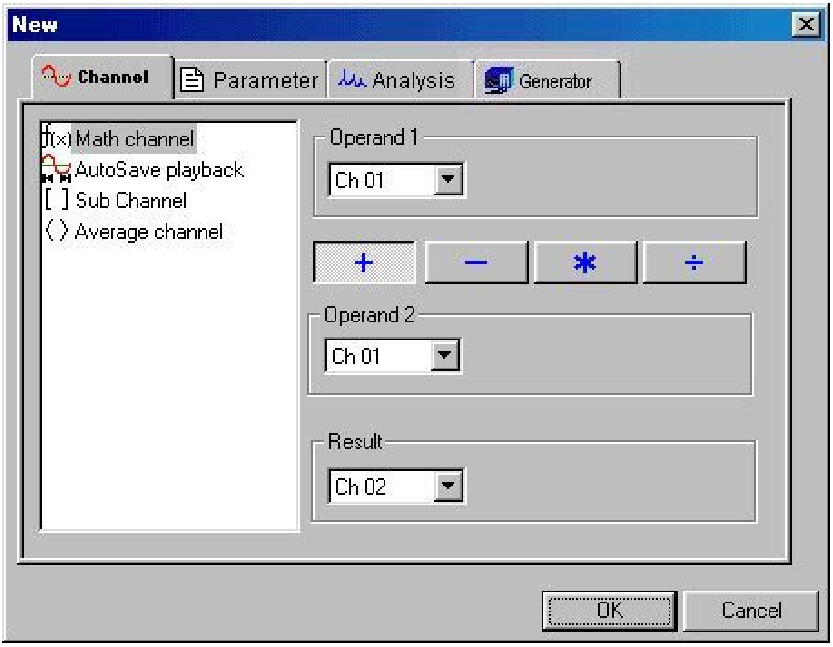


Standard Waveform Generation

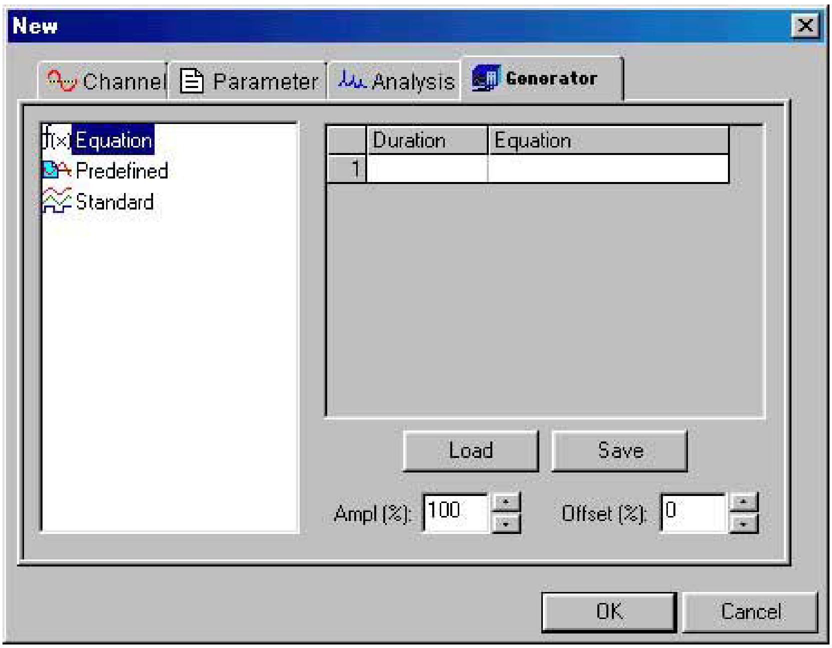
1 Click **File** menu from the menu bar.



2 Select **New** from the drop down menu. You will see the following window:

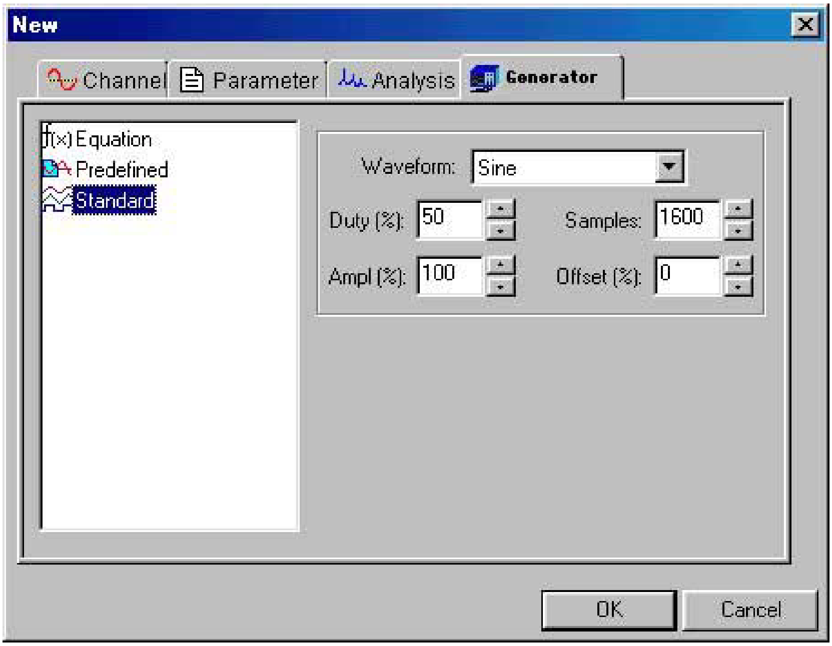


3 Click on the **Generator** tab. You will notice that the Equation text is highlighted as a default.

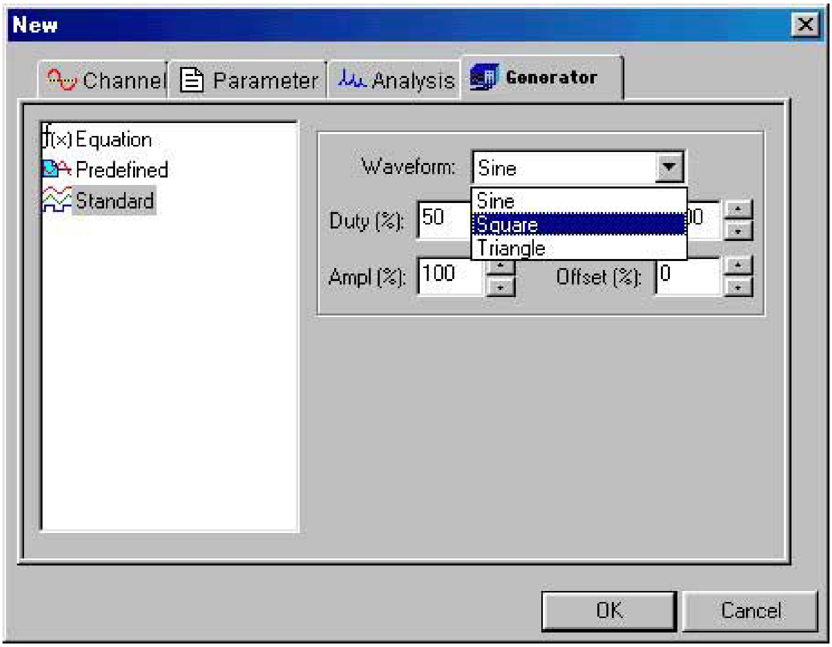


Using controls in window shown above, users can specify an equation to generate a specific signal.

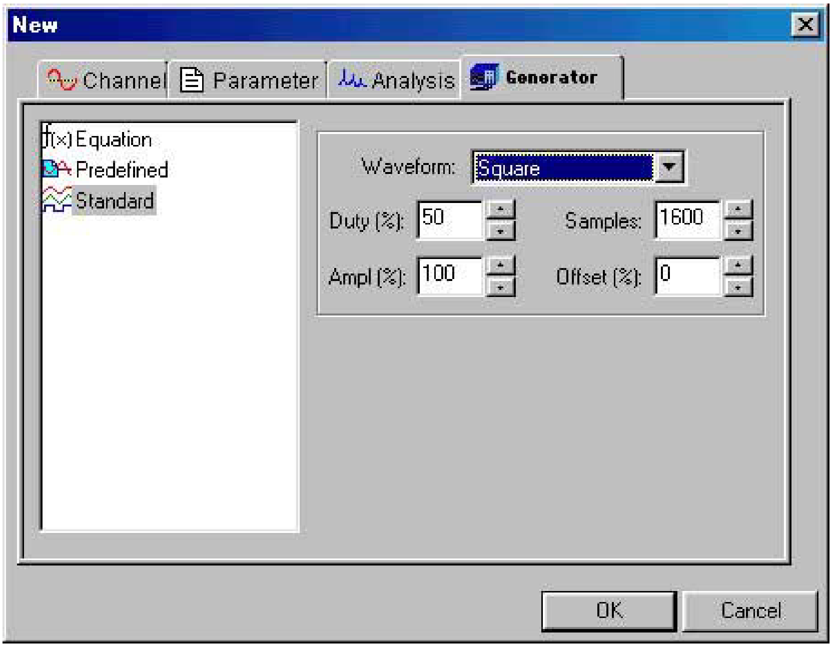
4 Click **Standard** from the list in the dialog box. Click **OK**.



5 One can observe that the Sine waveform is the default selection. Click on the scroll down arrow in the Waveform text box. You will find out that there are three different types of standard waveforms available for selection. These are **Sine**, **Square**, and **Triangle** as shown below:



6 Select **Square** from the drop down list. Click **OK**.

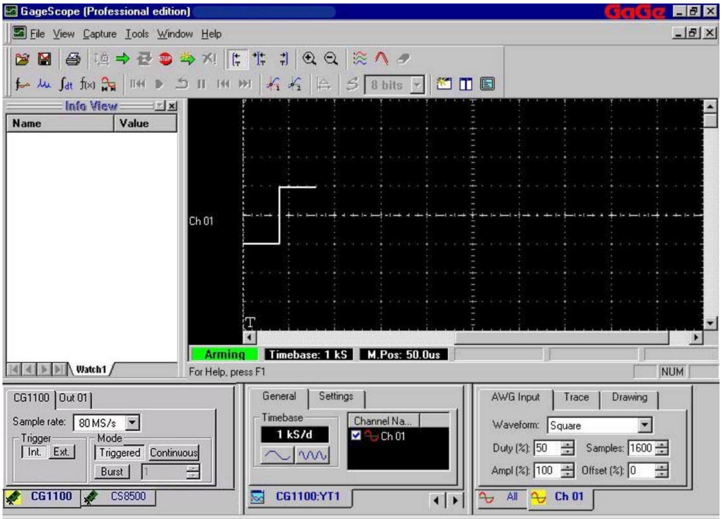


The resulting window shows the default settings for input parameters: **Duty**, **Samples**,

Amplitude and **Offset**. Users can choose to change these parameters depending on their particular application.

In the current setting, a single cycle of a Square wave will be generated with 50% duty cycle. The total number of samples within one cycle of this wave will be 1600. The amplitude of the wave is 100% referenced to the input range selected. There will be no DC Offset.

7 Click **OK**. A single cycle Square wave will be displayed in GageScope®:



Note that Channel 1 displays the generated Square wave. The AWG Input tab has been selected and consequently the AWG property page appears in the bottom right hand corner of GageScope® display.

Refer to the Reference section of this manual on further details for arbitrary waveform generation.

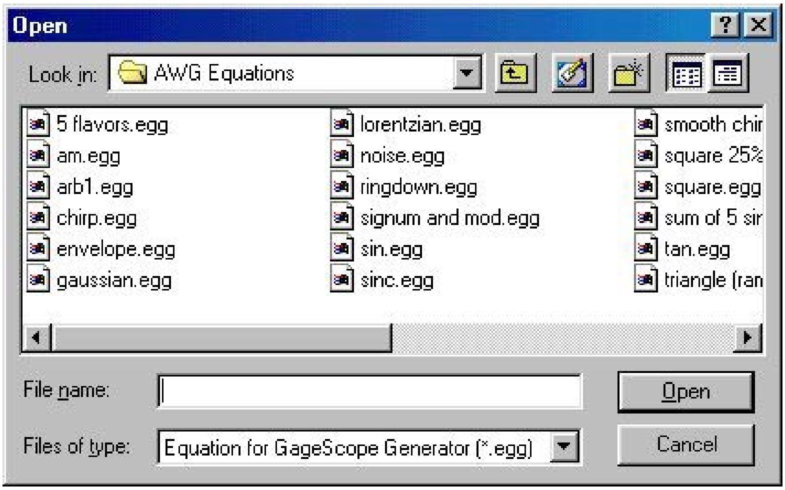
Tutorial 7: Using the Graphical Waveform Editor

The graphical waveform editor allows onscreen viewing and editing of waveforms. Complex signals can be created with relative ease using the graphics editing capabilities of GageScope® software. The complex waveforms created can then be uploaded to an arbitrary waveform generator for use in scientific applications. These features have been explained in an illustrative example below.

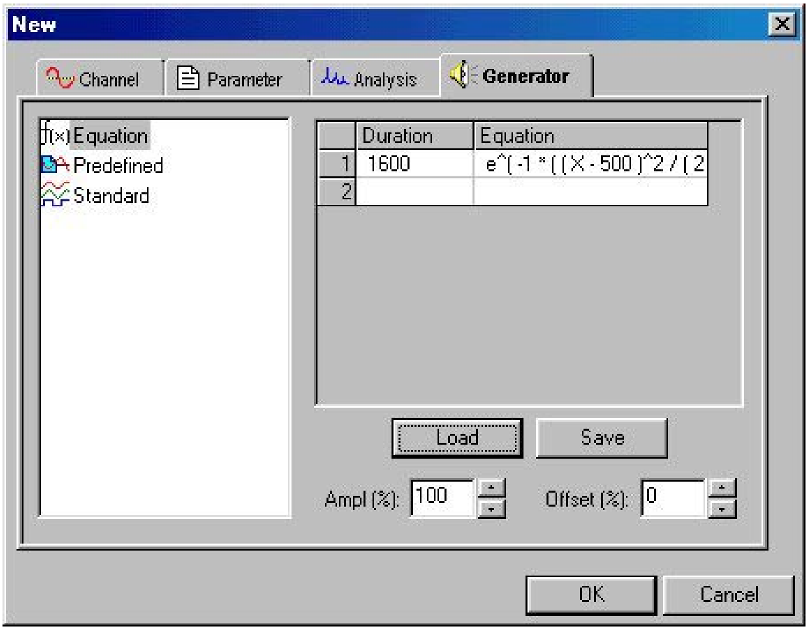
We will work on a gaussian pulse that has been created and saved as a .sig file in a local directory. The objective is to produce a non-gaussian shaped pulse using the graphical editor within GageScope. It has been assumed that GageScope® has been launched prior to following these steps:

Waveform Editing Procedure

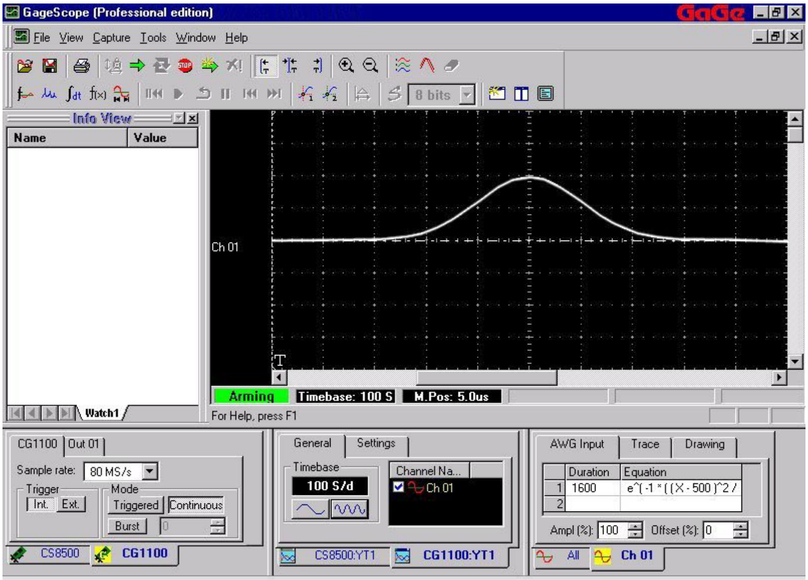
1 Click **File --> New --> Generator --> Equation --> Load**. The file **Open** dialog box appears:



2 Click **gaussian.egg** from the list in the dialog box. Gaussian.egg will appear in the **File name** text box. Click **Open**. The **New** dialog box opens as follows:



3 Click **OK**. GageScope® will display the gaussian pulse as follows:



Note: The pulse has been expanded in time and displayed at the center of the display window in Gagescope. Refer to the **System** and **Display Control** sections in the

Chapter titled, **Reference**, of this manual for details on how to adjust these settings.

We have selected a gaussian pulse to be edited using the graphical editor for the purpose of demonstration. You can either select any other stored waveform or create a new waveform using the mathematical equation editor.

4 Right click on the reference line of the displayed signal. You will observe that the cursor changes shape, the waveform changes color and a menu appears at the right click position.

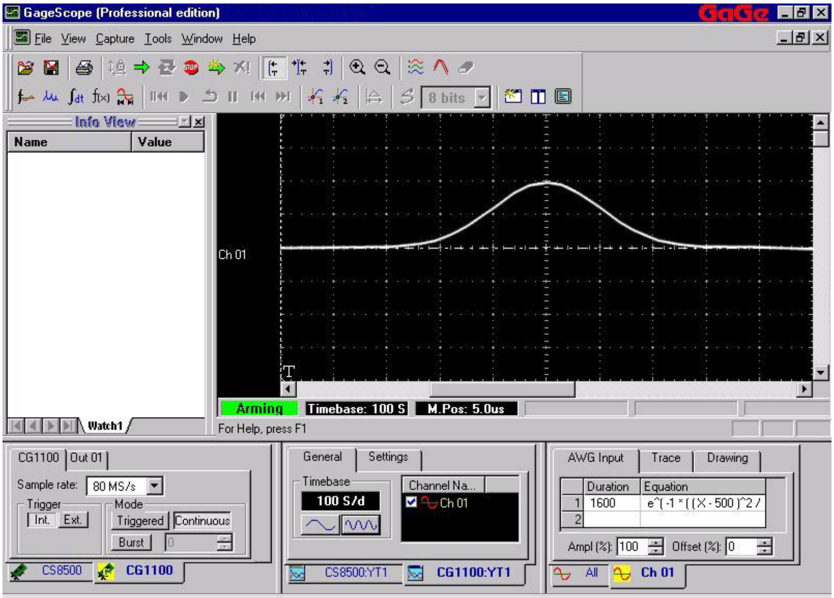
This menu is shown as follows:

5 Double click on **Edit** to enter the Graphical editor. At this stage, you may be warned that for the current time base, only limited editing is available.

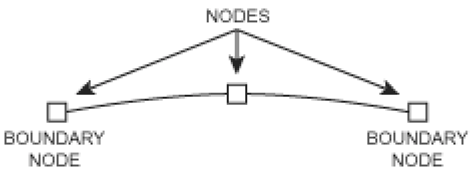
The warning message is displayed as follows:

If you enter **Yes**, basic editing features like Copy, Cut, and Paste will be available. The graphical editing to change the waveform will be available only when the time base is expanded to 2 microseconds per division.

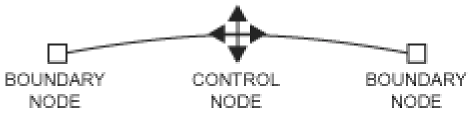
6 Since we are not ready for editing the waveform, click **No**. You will get back to the GageScope® display with the gaussian pulse shown on Channel 1.



7 At this stage, if you click **Yes**, you will enter the graphical editor but will not be able to use any of its features. You must see a few **Nodes** on the waveform to commence the editing process. A node is a point on the trace surrounded by a square.



The figure above shows a selected segment to be edited. Note that the edited segment shows three nodes. The end points of the segment are the **boundary nodes**. The middle node is the one that has been created (node creation to be explained later) or is an already existing node. The middle node is termed as the **control node**. The control node will be used to modify the selected line segment.



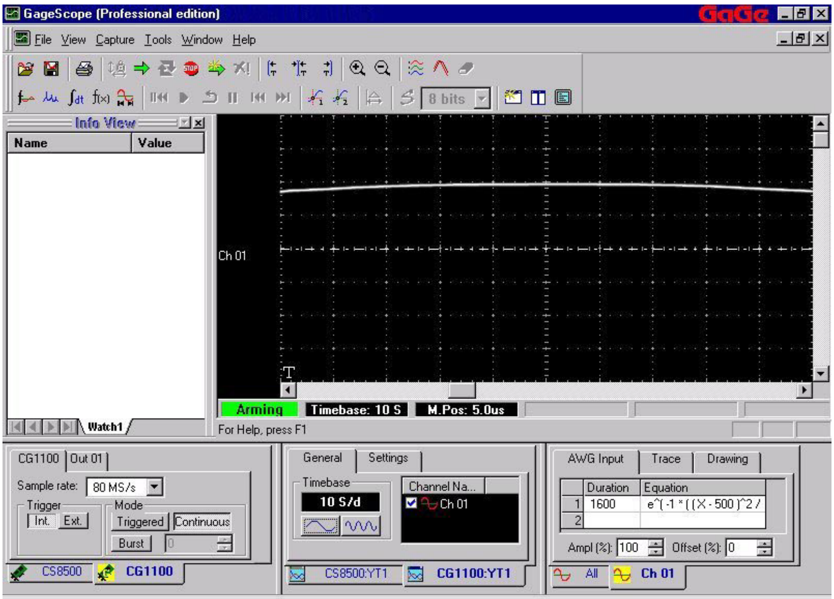
Note that the control node is shown with two orthogonal lines with arrows at the end. The mouse pointer changes shape to the one shown above when this node is selected for editing.

You can select a segment with just the boundary nodes and then create the control node anywhere within the segment OR you can select two segments with the middle node as the control node.

***The important point to remember is that we must have three nodes within a selected signal segment to modify the segment*.**

When cursor is placed on a node to be selected for editing, it changes into a four way arrow as ✣

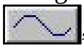
8 Expand the time base by clicking [button]. Keep redisplaying the portion of the signal to be edited by using GageScope® display's horizontal scroll bar every time you adjust the time base. The display below shows an expanded view of the gaussian pulse:



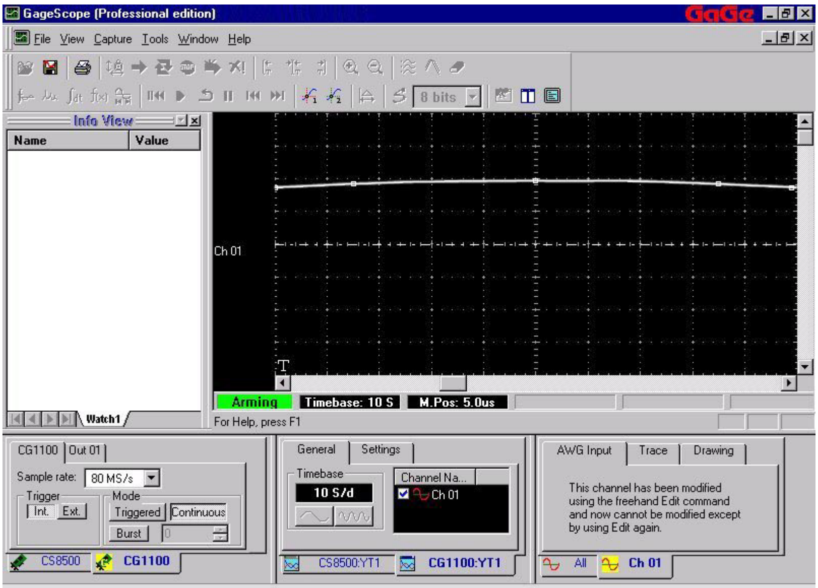
9 After expanding the time base and redisplaying the signal as shown above, make another attempt to launch the graphical editor.

Right click on the zero reference line. The cursor will change shape and the window display context menu will appear. Double click on **Edit** to launch the graphical editor. One of the following will happen:

- If the trace is still too dense, the GageScope® warning window will reappear. This indicates that the time base has not been expanded enough for the editor to function.

- In such a case, click **No**. You will get back to the expanded display. Click on the time base expand button, , to expand the time base even further. Try launching the graphical editor again by right clicking on the zero reference line and selecting **Edit** from the drop down menu.

- If the time base is proper for the editor to function, you will see the following display:

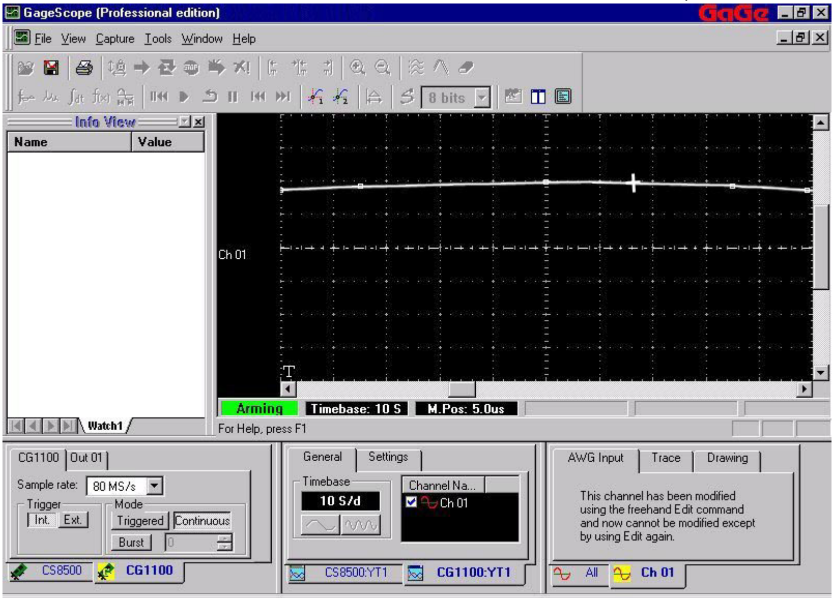


10 Note that the trace is now divided into four segments and hence there are five nodes shown on the trace. You can select any segment for editing purposes. Since the displayed portion of the waveform shows the nodes, it implies that we are now ready for graphical editing.

To start editing the displayed portion of the waveform, you can either use the existing Nodes or create new Nodes.

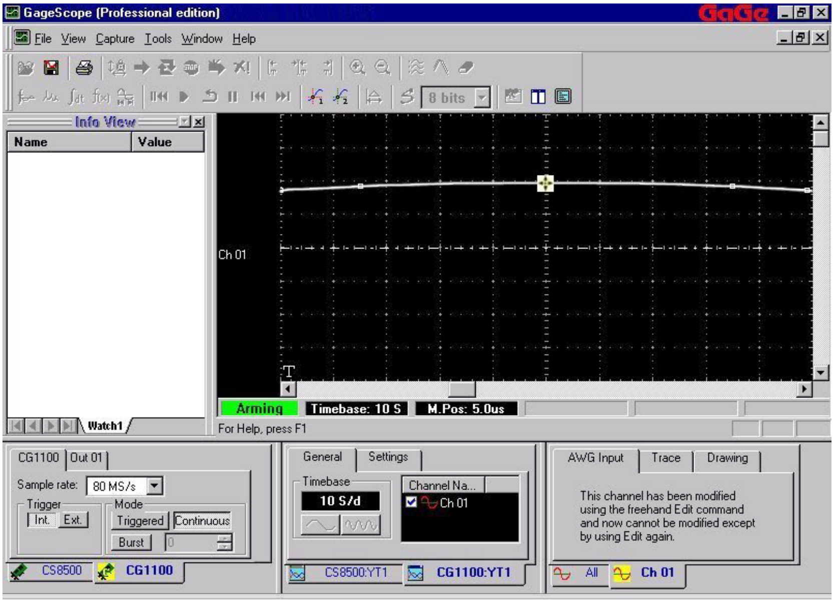
Using the Existing Nodes

11 Place the cursor on any portion of the waveform. The cursor changes from an arrow to be a plus sign +. This shows that a point on the trace has been selected, as depicted in the figure below:

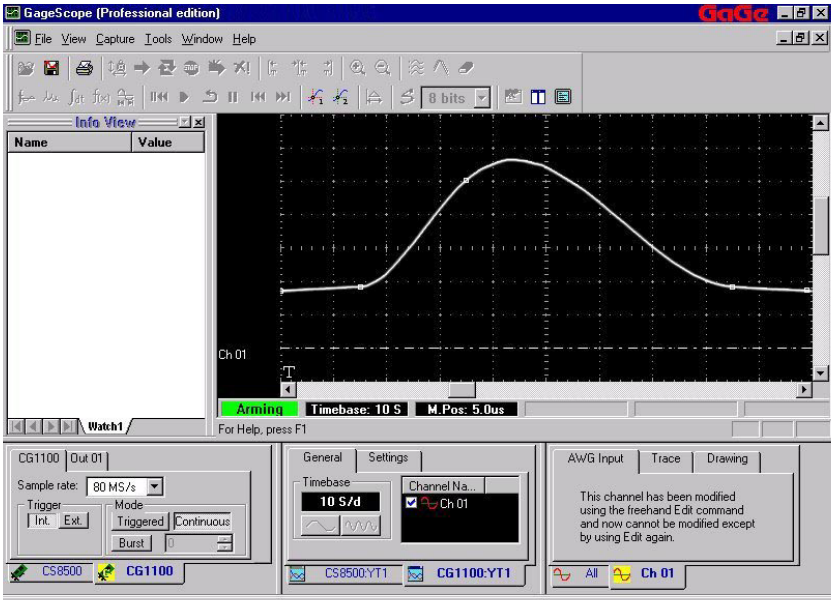


12 Move the hairline cursor closer to a Node. You do not have to move along the trace. You can go directly to the Node.

Once you are in the neighborhood of a Node, you will notice that the hairline cursor changes into a four-way arrow, ✣. This signifies that this particular node is now a control node and can be used to alter the trace. The figure below shows the trace with the control node:



13 To move the control node, click the left mouse button once. This action activates the four-way arrow and now we are ready for editing. Press the left mouse button again and move the cursor as desired. Since we wish to make a gaussian pulse, non-gaussian, we will move the peak point to the left as shown in the figure below:



Observe that the peak has been shifted to the left making the pulse non-gaussian.

Note that the procedure employed in the drag and drop editing process, is the **Cubic Spline** Interpolation, which has been selected as a default. The Cubic Spline interpolation procedure works on fitting a cubic polynomial within a selected interval. In the example above, the selected interval is the interval between the two nodes: the nodes on the left and right of the control nodes define the left and right boundaries of the selected portion of the waveform. When the user moves the control node, the editor fits a cubic curve in between the two boundary nodes. This curve passes through the selected node as well as the two boundary nodes.

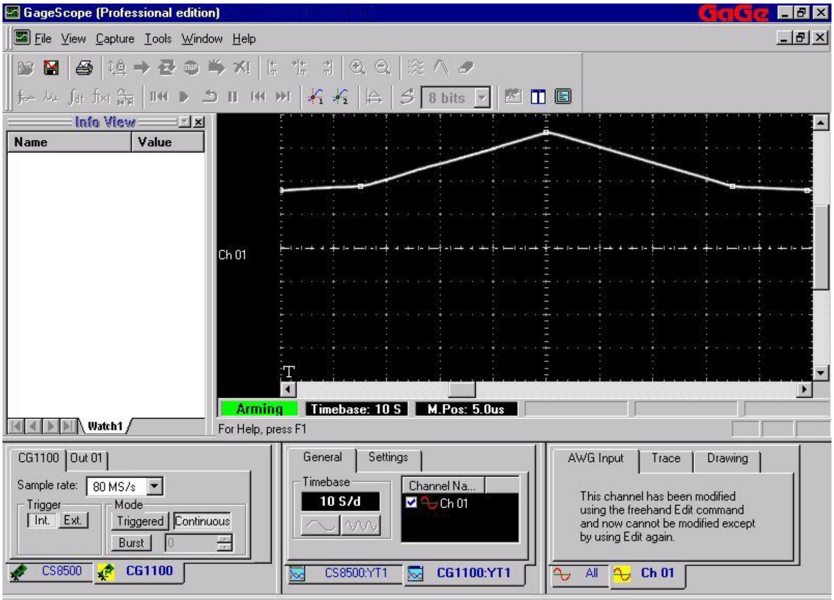
14 You can also select the straight line interpolation procedure too by selecting **Straight** from the interpolation submenu. While in the Edit mode, right click on any portion of the display. You will see following context menu:

Note that **Cut, Copy**, and **Paste** operations are not available since the portion of the waveform to be edited for cut and paste operations has not been selected as yet. These items will be available for use whenever a portion of waveform is selected by placing the cursors on the trace for editing. The cutting, pasting and copying procedure has been outlined in the section to follow.

15 Select **Interpolation** to view the submenu as follows:

Spline is shown as the default interpolation option.

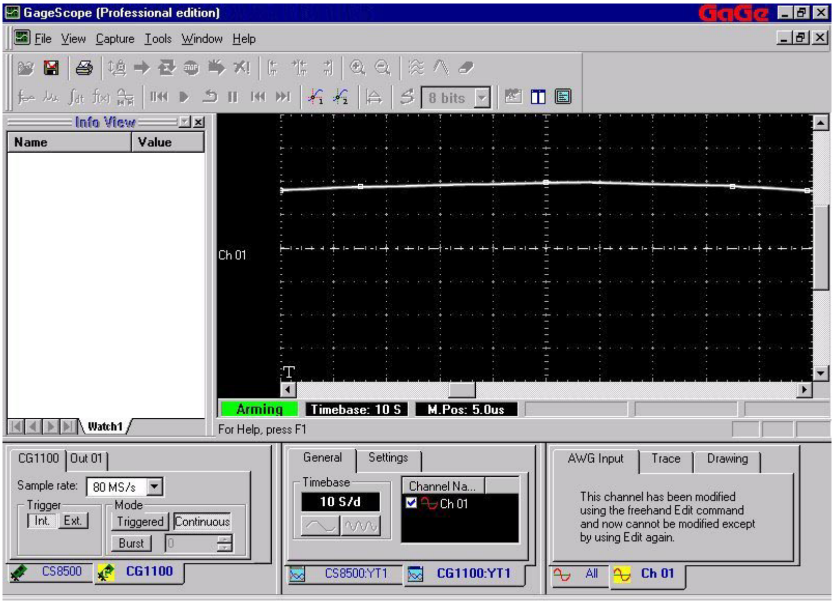
16 Select **Straight** option in the menu above. This option connects the three nodes, the control node, and the two boundary nodes, by straight lines as shown below:



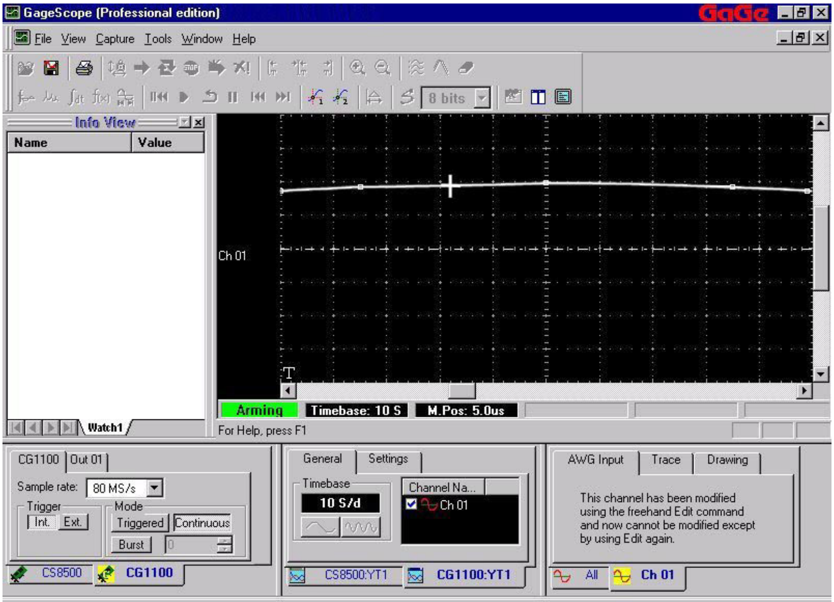
Creating New Nodes

17 New nodes can be created with absolute ease in the Graphical waveform editor. Newly created nodes impart user extreme flexibility to reshape the waveform arbitrarily.

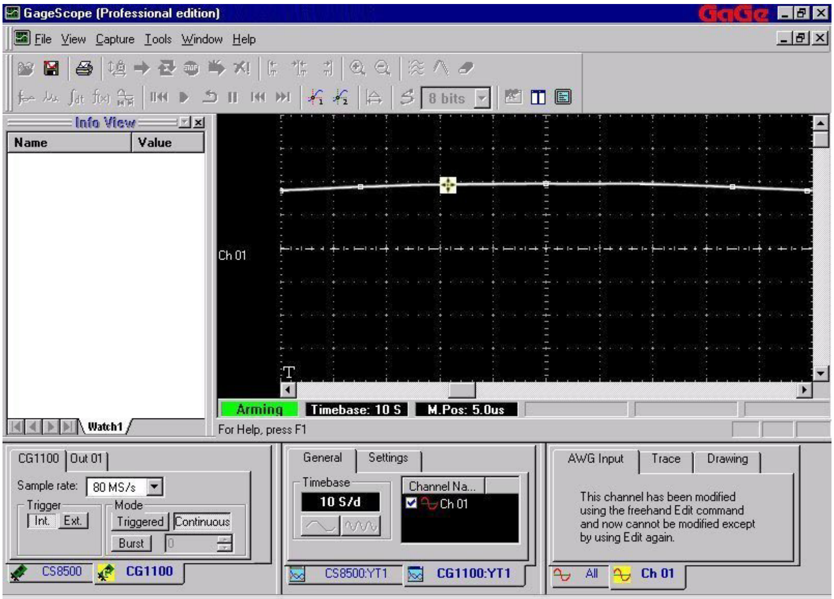
Follow steps **1-9** to get to the following displayed waveform in GageScope:



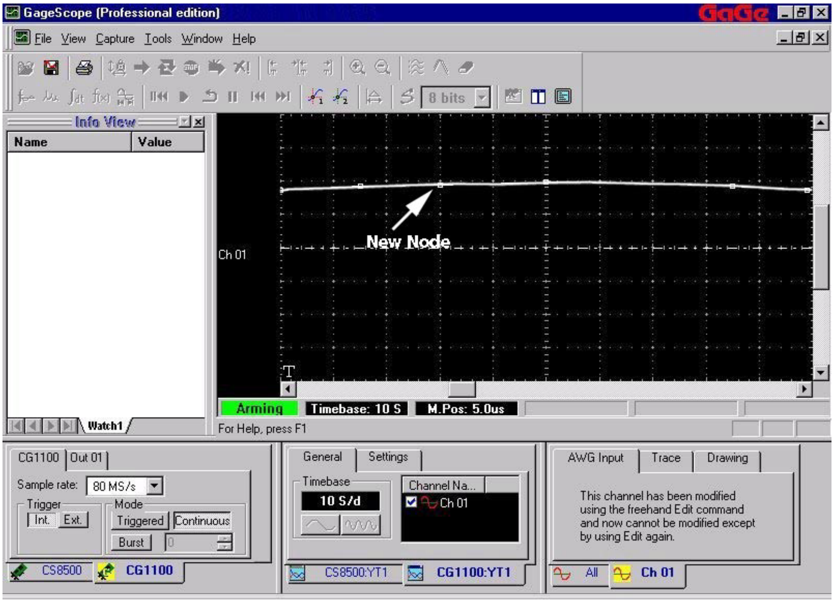
18 Move the cursor to the desired location on a selected portion of the waveform. This is the location where you wish to create a new node. You will note that the cursor changes shape to be a plus sign +.



19 Double click on the selected point and the cursor changes shape to look like .



20 At this juncture, you have created a new node at this location. To view the newly created node, move the cursor away. You will see the following:

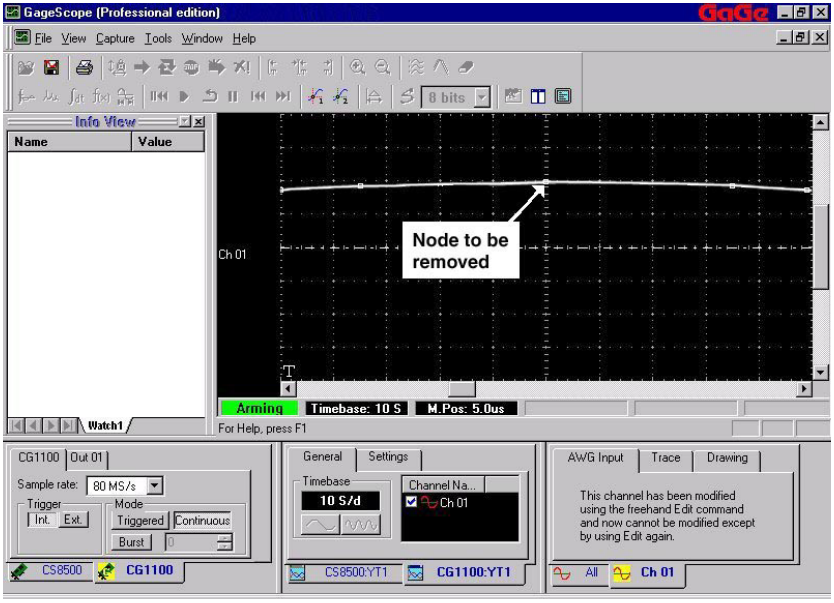


You can now use this new node to edit the selected portion of the trace.

Deleting Existing Nodes

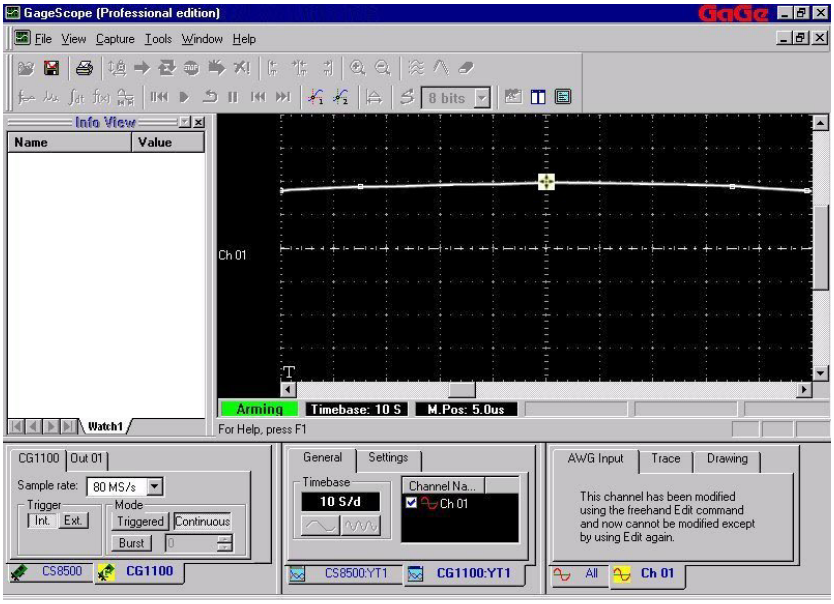
21 Existing nodes can be deleted by simply double clicking on the Node.

Follow steps **1-9** to get to the following displayed waveform in GageScope:

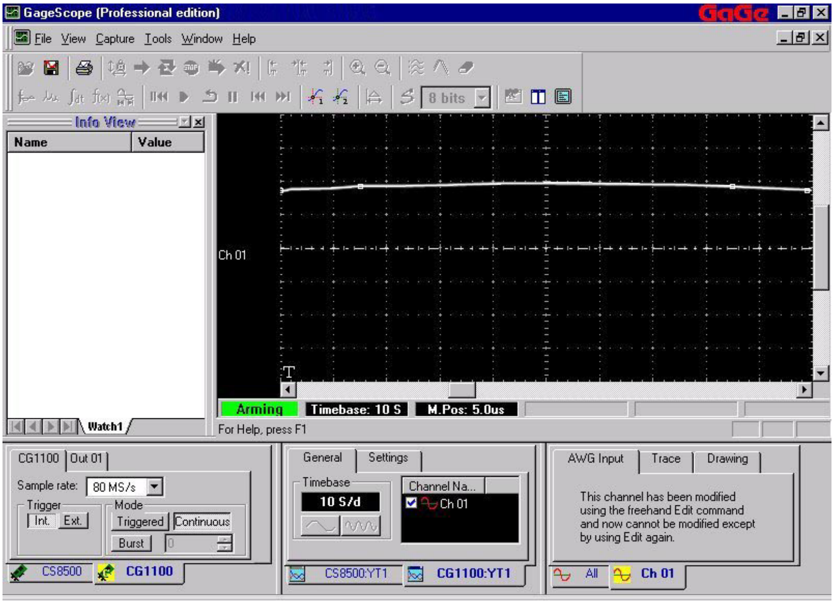


22 Move the cursor to the node that you wish to delete. We have selected the middle node to be deleted.

Move the cursor to the desired node. The cursor changes to be a four-way arrow.



23 Double click to remove the selected node.



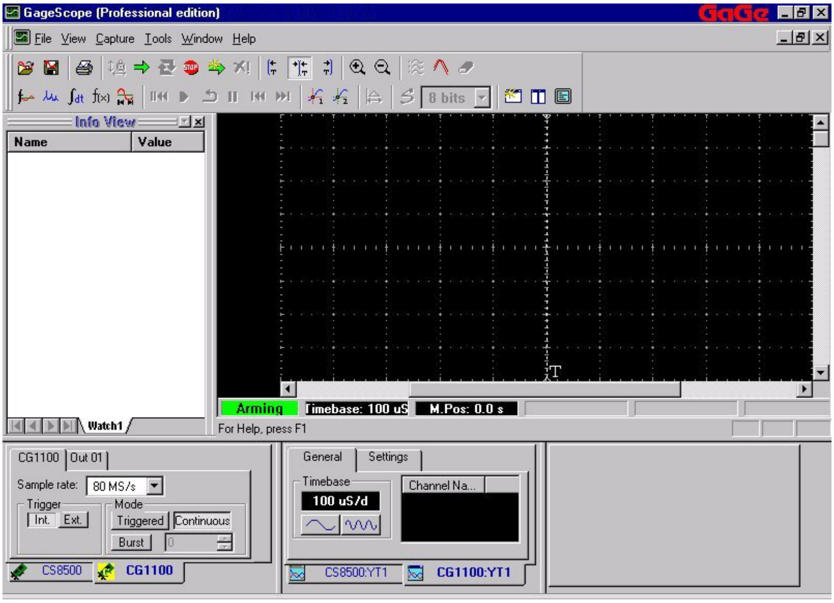
Cut, Copy and Paste Features

These features are available to the user whenever the desired portion of the waveform is selected using the cursors provided in GageScope® software.

Cut and Paste Operations

The example below illustrates how the Cut and Paste features work within the GageScope® software.

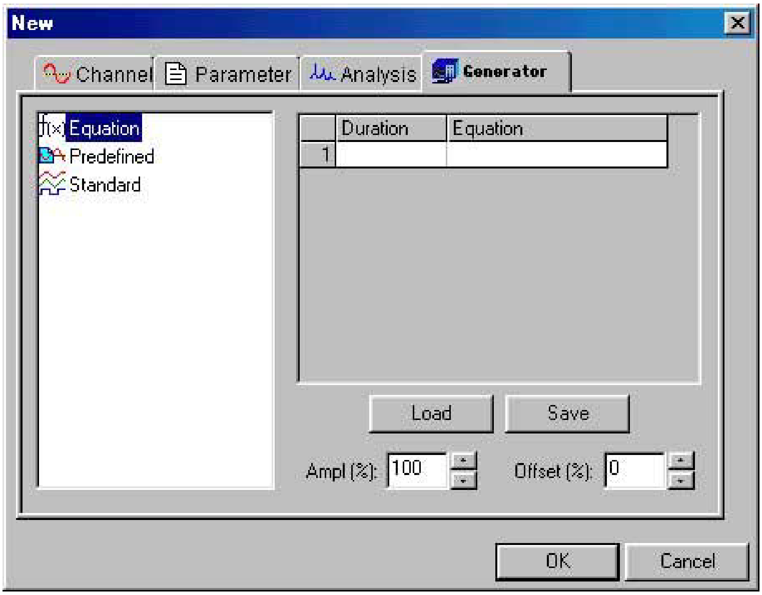
1 Before carrying out example below, make sure that GageScope® display is configured as follows:



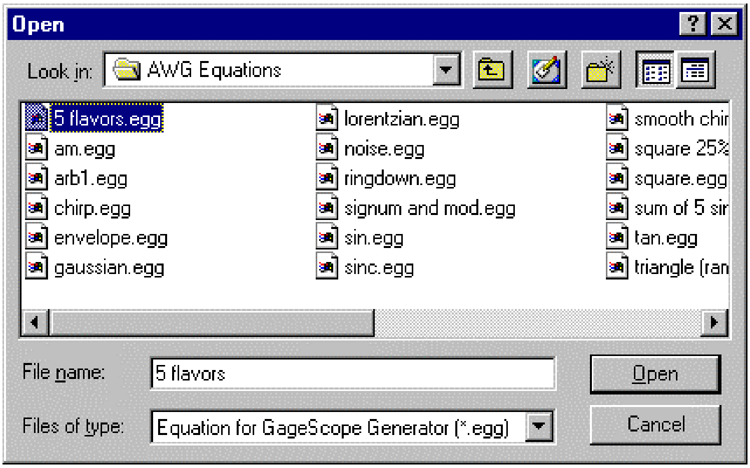
Note that the Channel control area in the lower right corner of the GageScope® display appears blank. This is because no arbitrary waveform generator related signal has yet been created or loaded. Also note that the CG1100 tabs in the System control and display controls have been selected and the CG1100 related display window is shown maximized.

2 Open the file titled **5 Flavors.egg** by carrying out the steps as follows:

Click **File --> New --> Generator --> Equation**. The **New** dialog box opens:

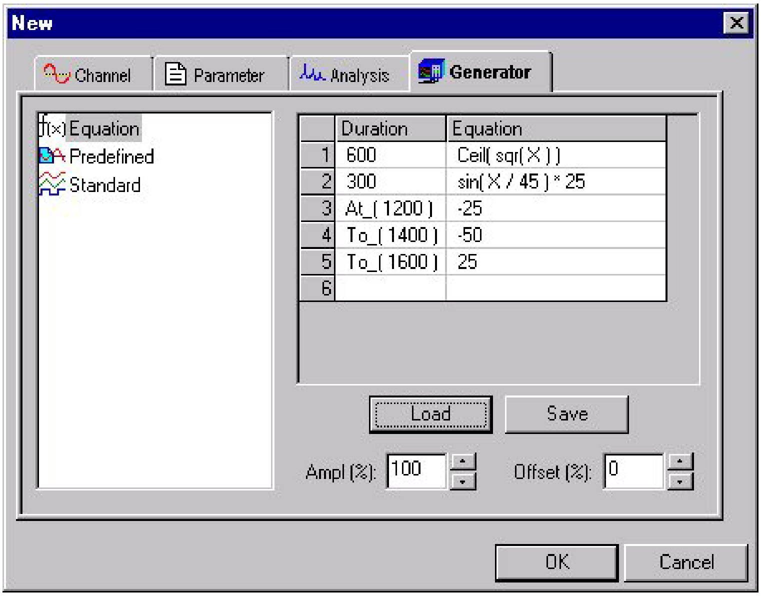


3 Click **Load**. The file **Open** dialog box appears:



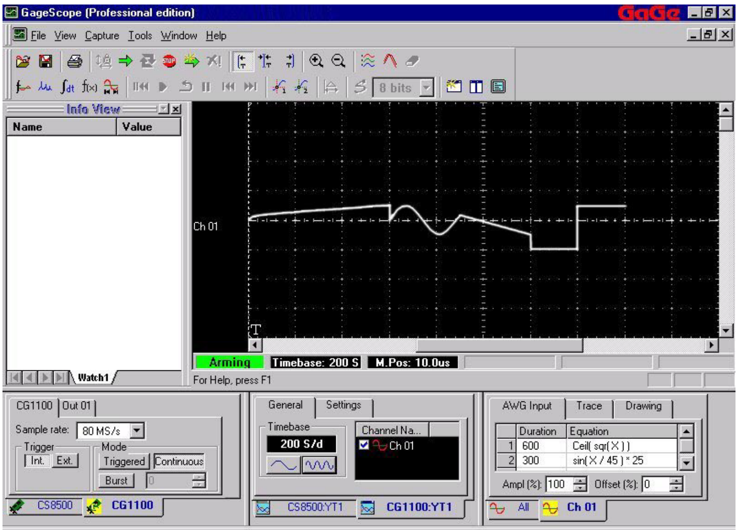
4 By default, the contents of the **AWG Equations** folder appear in the file **Open** dialog box as shown in the above.

Select the file named **5 flavors.egg**. The name of the selected file appears in the File name text box. Click **Open**. The **New** dialog box appears again:

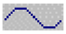


Note that the right hand portion of the **New** dialog box shows the exact composition of the signal to be displayed in GageScope. The signal certainly contains 5 types of waveforms.

5 Click **OK** to load the signal. GageScope® will display the signal as follows:

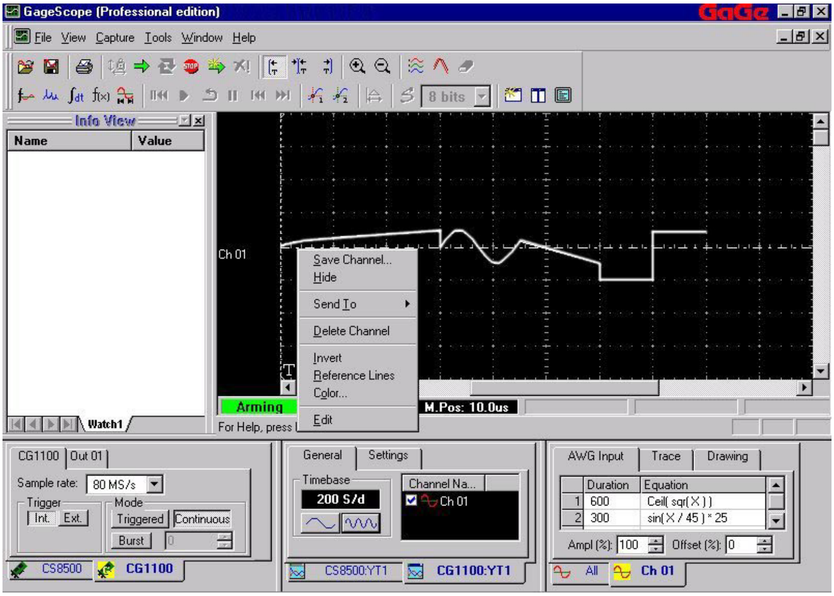


In case the signal does not appear exactly the same as in the figure above, you may have to align the trigger to the left and compress the timebase to make the signal match the one shown above.

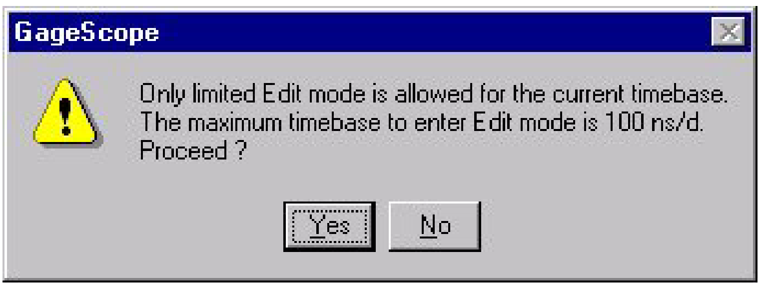
Click the left trigger align Icon  to align the trigger to the left. Click on the timebase expansion  or compression  buttons in the display control area to adjust to the timebase to exactly match the display settings as shown in the figure above.

6 The next step is to invoke the Edit Mode. In order to achieve this end, place the cursor on the signal zero reference line. The cursor will turn into a move pointer.

Right click to view the context menu:



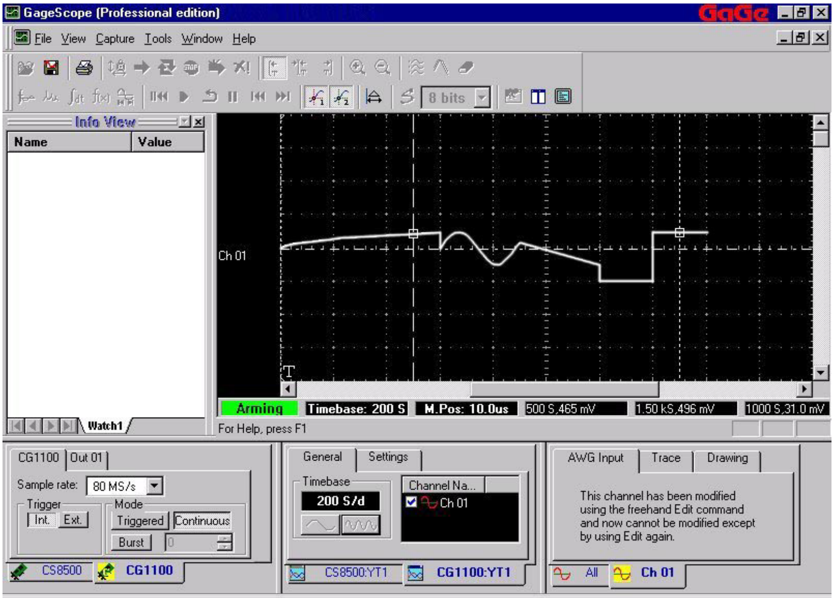
7 Note that **Edit** is the last option in the context menu. Click **Edit** to enter the graphical editing mode. The following dialog box appears:



This message informs you that the current timebase settings allow limited editing features. This means that you may have to expand the timebase to be able to utilize full editing capabilities. Since the timebase settings are frozen in the Edit mode, you will have to click **No** to go back to the non-edit mode and expand the timebase by clicking on button in the display control. After having expanded the timebase, you can again attempt to launch the graphical editor by clicking Edit in the windows context menu. This is an iterative procedure and we do not want to embark on this route.

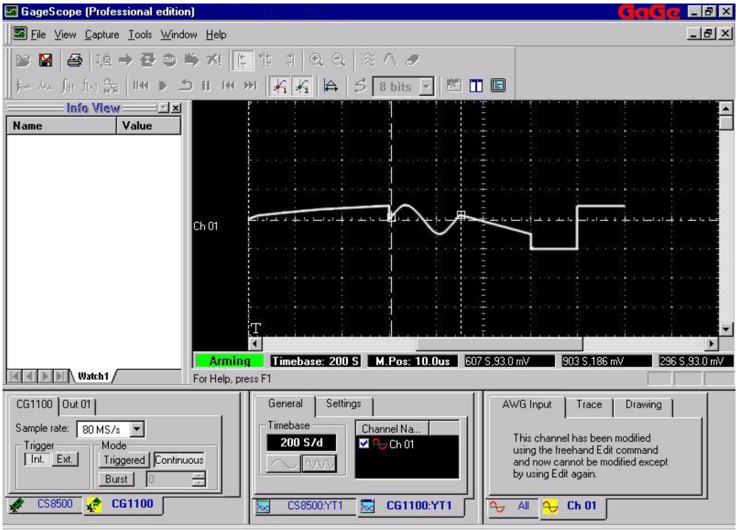
8 We have opted to select **Yes**. The waveform graphical editing mode is now invoked. You will view the original waveform.

At this stage, click one after the other on the left , and right cursor , icons in the toolbar. The GageScope® display will show both the cursors at default locations:



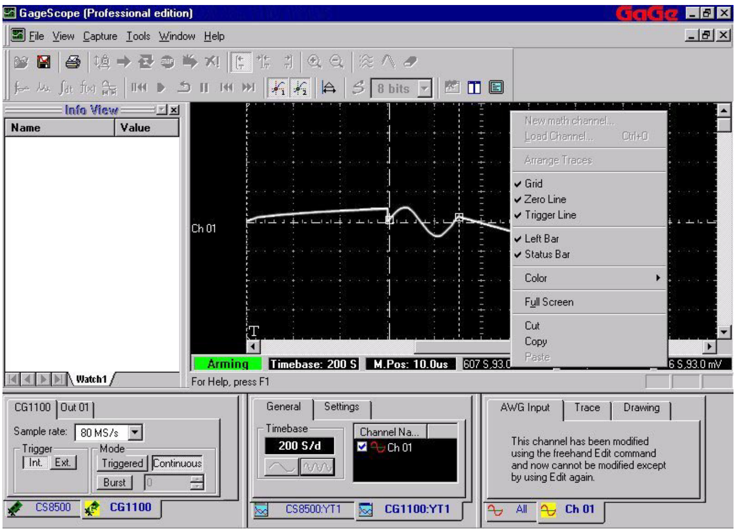
9 Place the mouse pointer anywhere on the left cursor. The mouse pointer will change shape to a move pointer. Hold down the left mouse button and drag the move pointer. The cursor will drag with the pointer. Release the left mouse button at the starting location of the waveform to be edited. Follow the same procedure for the right cursor.

In this example, we have decided to cut out the sine portion of the trace. This portion will be inserted in between the last part of the trace.



You can observe that the sine portion of the signal has been selected.

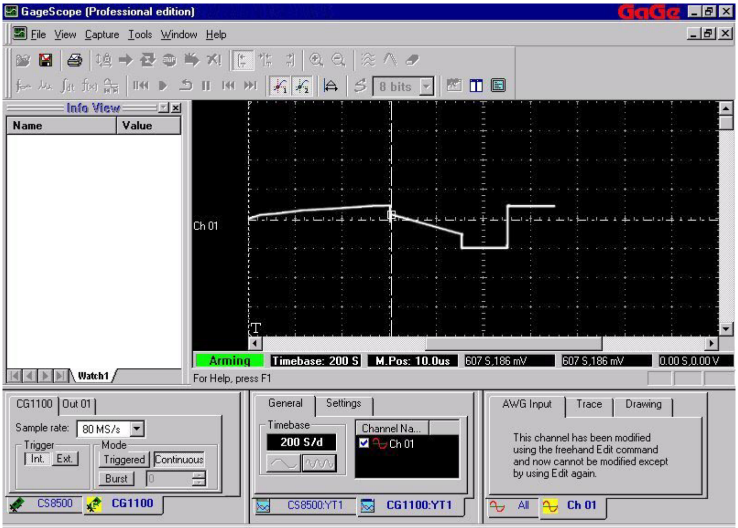
10 Right click anywhere on the display (except the zero reference line). You will see the display context menu as follows:



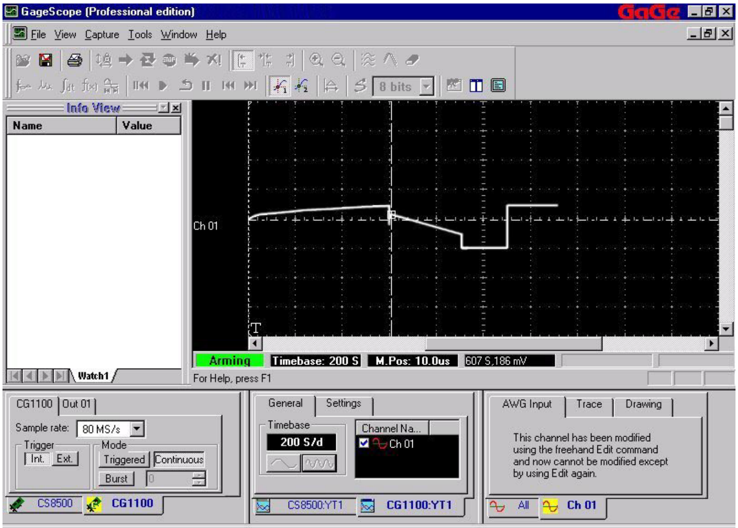
The **Cut** and **Copy** commands are grouped together at the bottom of this window.

11 Double click on **Cut** to select and execute the command. You will see that the selected portion of the signal has been removed from the display and both left and right cursors collapsed to show a single cursor. You must understand that both cursors exist as

separate entities. It's just that they seem to be one.



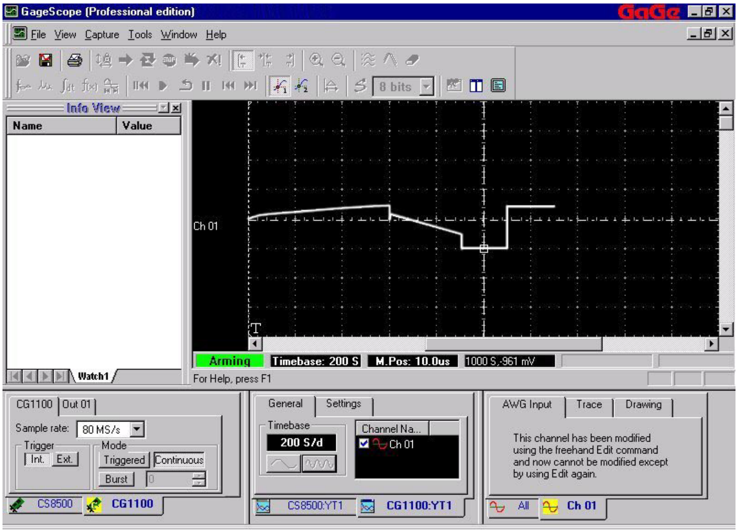
12 Hide the right cursor by clicking on the respective icon in the toolbar.



Observe that the sine portion of the signal has been removed. The portion to the right of the cut part has moved to the left to glue with the left part of the signal.

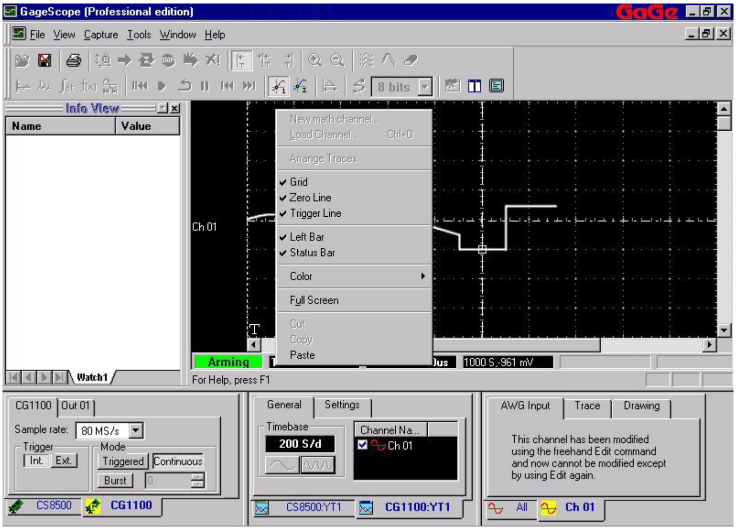
Note that the right cursor icon appears enabled in the toolbar.

13 Place the mouse pointer on the left cursor, hold down the left mouse button, and drag the cursor to the desired insertion point on the displayed waveform. Release the left mouse button. You will see the following display with the left cursor positioned at the desired insertion point:



Note that the insertion point has been arbitrarily selected. If you wish, you can select any other paste location.

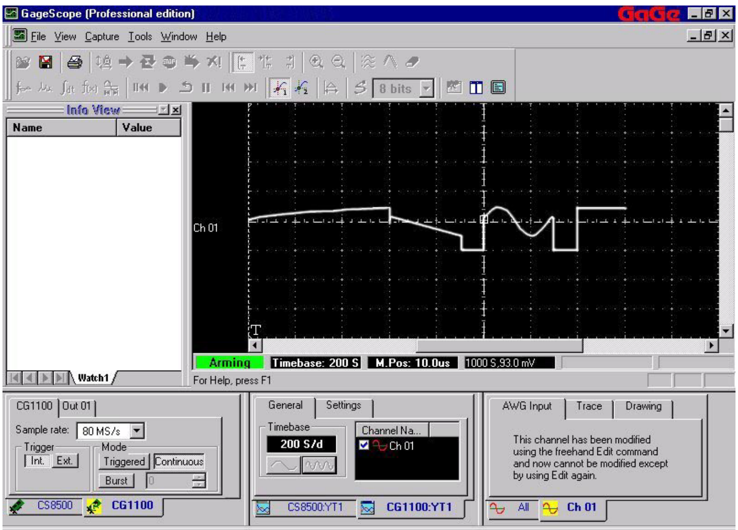
14 Right click anywhere within the display window. The display context window will appear again as shown below:



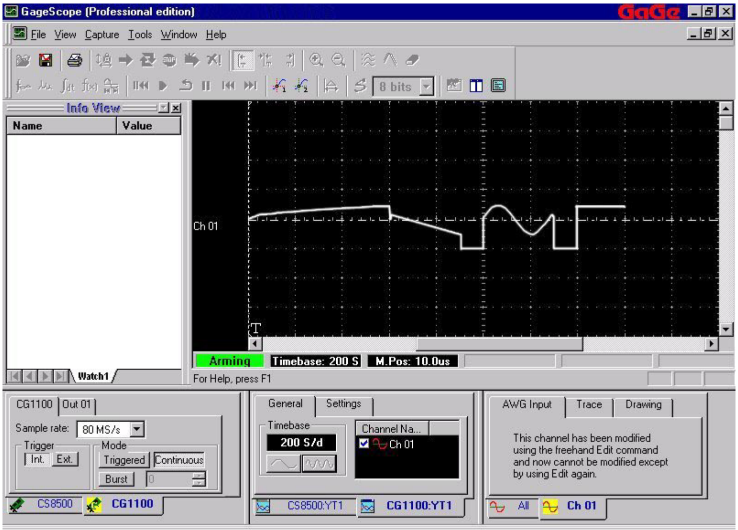
Make sure that you do not right click on the zero reference line. Right clicking on the zero reference line brings up the channel context menu.

15 You can see that the **Cut** and **Copy** commands have been disabled. Only the **Paste** command is shown enabled at the bottom of the display context menu.

Double click on Paste to insert the cut portion of the trace. The modified signal will be shown as follows:

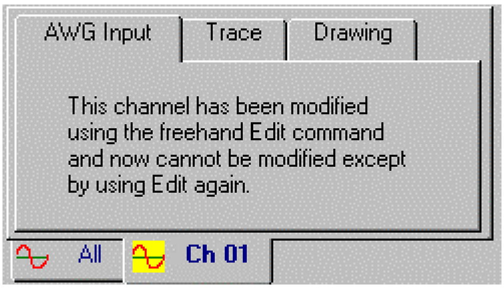


16 Hide the cursor by clicking on the left cursor button  in the toolbar:



17 You will note that while in the edit mode, most of the functionality in GageScope® remains disabled.

Click on the channel 1 tab in the bottom of the Channel Control. Click on the **AWG Input** tab to bring it to front. The AWG Input property page will appear and is shown as follows:



Channel 1 (**Ch 01**) contains the signal you have just modified. In order to get back to the normal functionality, you will have to leave the Edit Mode.

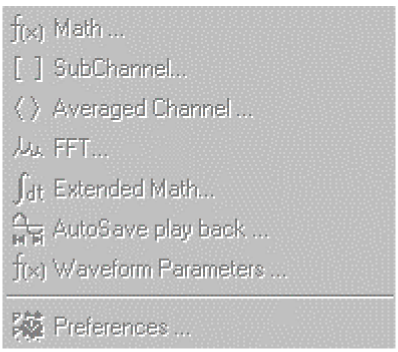
18 Click on the **File** menu in the menu bar. The File menu will be displayed:

Note that the Save Setup, Load Setup, New, and Load Channel commands have been disabled. These will be re-enabled once you leave the Edit mode.

19 Click on **Capture** in the main menu. The Capture menu will appear:

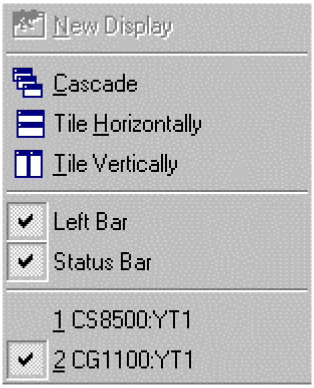
Note that the entire Capture menu has been disabled. These features are not needed while editing of signals is in progress.

20 Click on **Tools** in the main menu bar. The Tools menu will appear:



Note once again that the entire menu has been disabled since these functions are not required when editing of a waveform is in progress.

21 Click on **Windows** in the main menu bar. The Windows menu will appear:



You can see that the New Display command is disabled. While in the edit mode, you are not allowed to create another signal. In other words, only one signal can be modified at a time.

22 We have finished editing the waveform and now must exit the Edit Mode. In order to do so, move the mouse pointer to the zero reference line. The mouse pointer will turn into a move pointer. Right click on the zero reference line of the waveform. The zero reference context window will appear as follows:

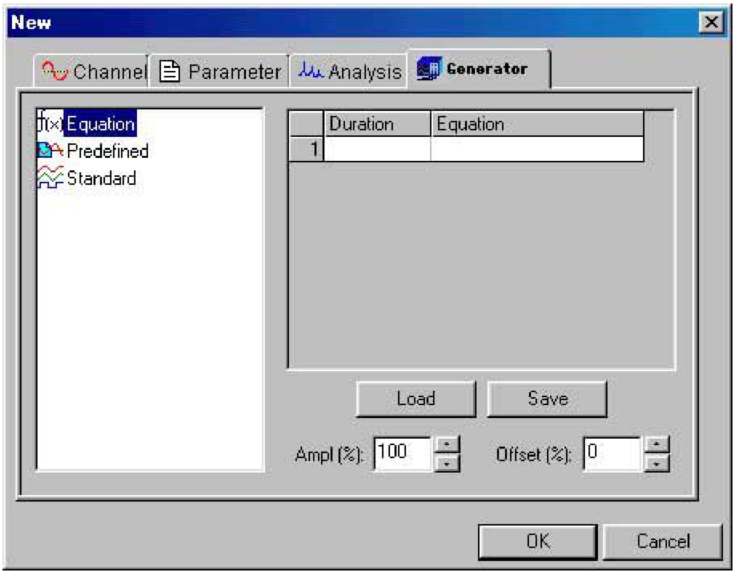
The check mark indicates that the Edit Mode is in force. Double click on Edit. This will take GageScope® out of the Edit Mode. You will note that the functions that were disabled will now be restored to their original settings.

Copy and Paste Operations

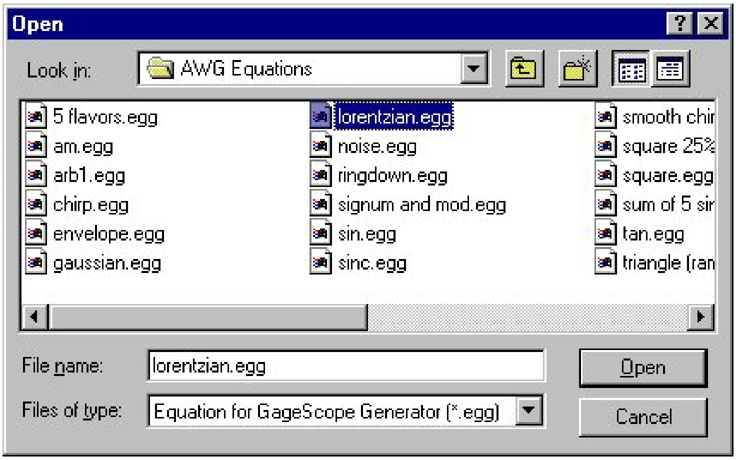
The example below illustrates how the Copy and Paste features work within the GageScope® software.

1 Open the file titled **Lorentzian.egg** by carrying out the steps as follows:

Click **File --> New --> Generator --> Equation**. The **New** dialog box opens:

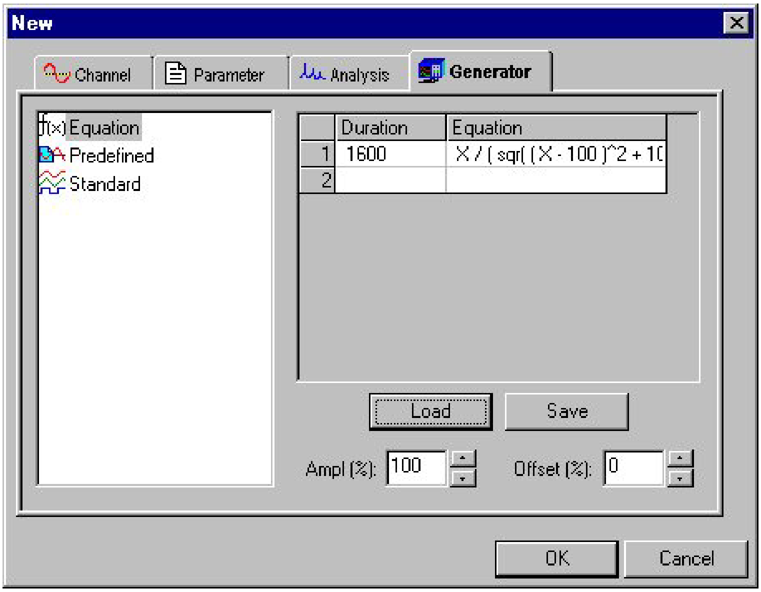


2 Click **Load**. The file **Open** dialog box appears:



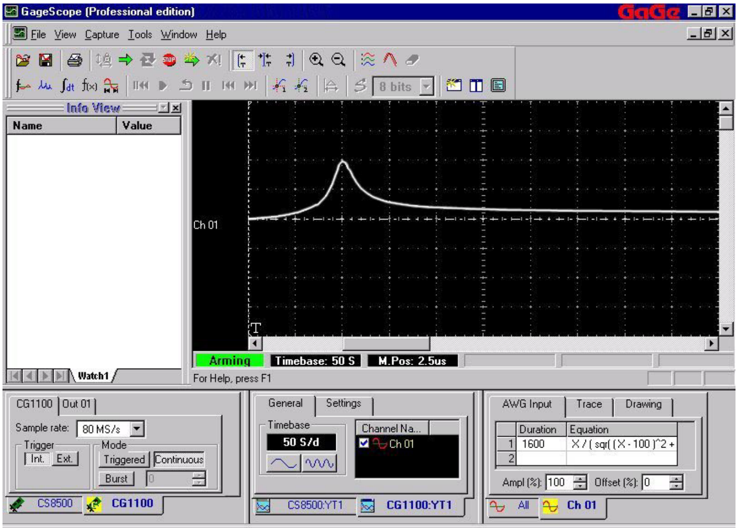
3 By default, the contents of the **AWG Equations** folder appear in the file **Open** dialog box as shown above.

Select the file named **Lorentzian.egg**. The name of the selected file appears in the File name text box. Click **Open**. The **New** dialog box appears again:

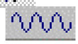


Note that the right hand portion of the **New** dialog box shows the exact composition of the signal to be displayed in GageScope.

4 Click **OK** to load the signal. GageScope® will display the signal as follows:



In case the signal does not appear exactly the same as in the figure above, you may have to align the trigger to the left and compress the timebase to make the signal match the one shown above.

Click the left trigger align Icon  to align the trigger to the left. Click on the timebase expansion  or compression  buttons in the display control area to adjust to the timebase to exactly match the display settings as shown in the figure above.

5 The next step is to invoke the Edit Mode. In order to achieve this end, place the cursor on the signal zero reference line. The cursor will turn into a move pointer.

Right click to view the context menu:

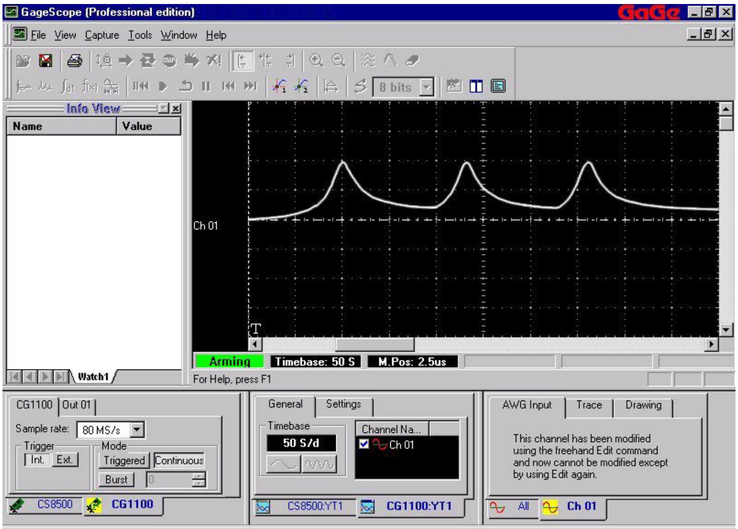
6 Note that **Edit** is the last option in the context menu. Click **Edit** to enter the graphical editing mode. The following dialog box appears:

Choosing Yes will invoke the graphical editor.

We have already discussed the implications of limited Edit mode capabilities in the previous section.

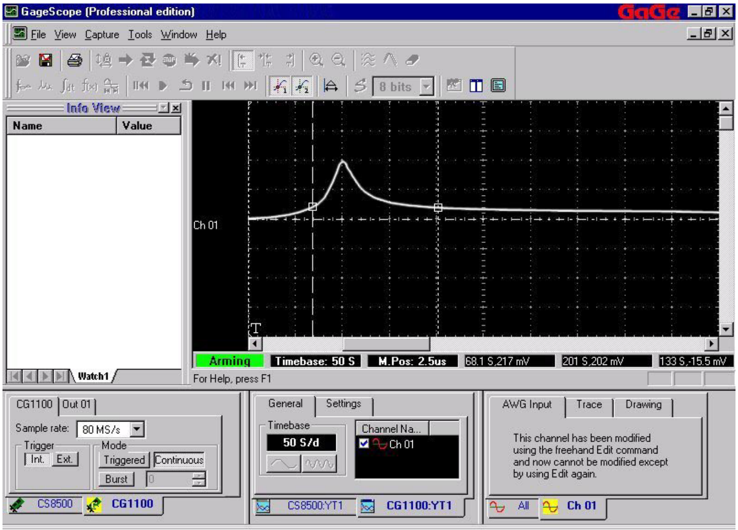
7 Click **Yes**. The waveform graphical editing mode is now invoked.

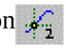
Following the procedure given in the previous section, select the portion of the trace to be copied. This is achieved by positioning the cursors at the desired locations on the trace. The display below shows one such selection:



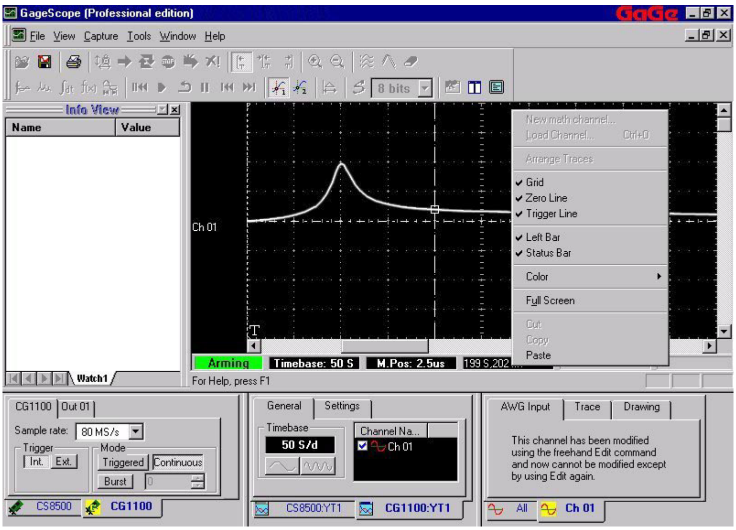
Observe that we have selected the middle portion of the signal to be copied.

8 Right click anywhere on the display (except the zero reference line). You will see the display context menu. Click on **Copy**. The screen will appear the same but the selected portion of the trace will be copied to the memory buffer:



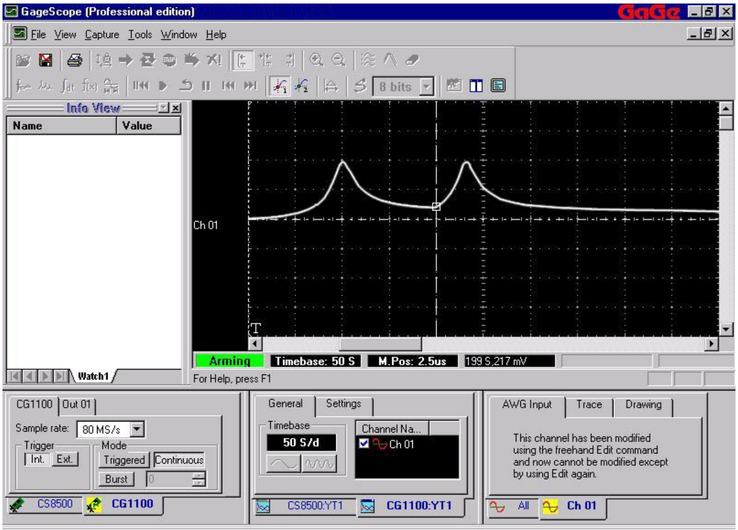
9 Hide the right cursor by clicking on the respective icon  in the toolbar. After hiding the right cursor, drag the left cursor to place it at the insertion point on the trace.

Next, right click anywhere on the display except the zero reference line. The reference context menu will appear:

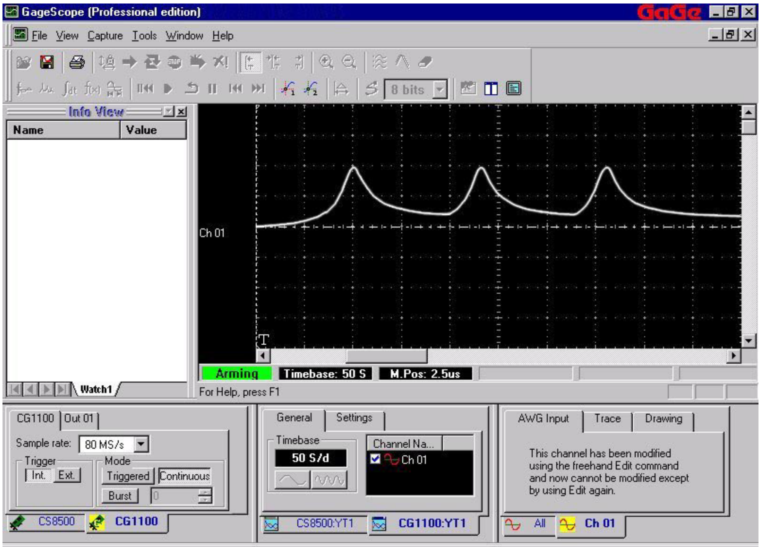


10 You can see that the **Cut** and **Copy** commands have been disabled. Only the **Paste** command is shown enabled at the bottom of the display context menu.

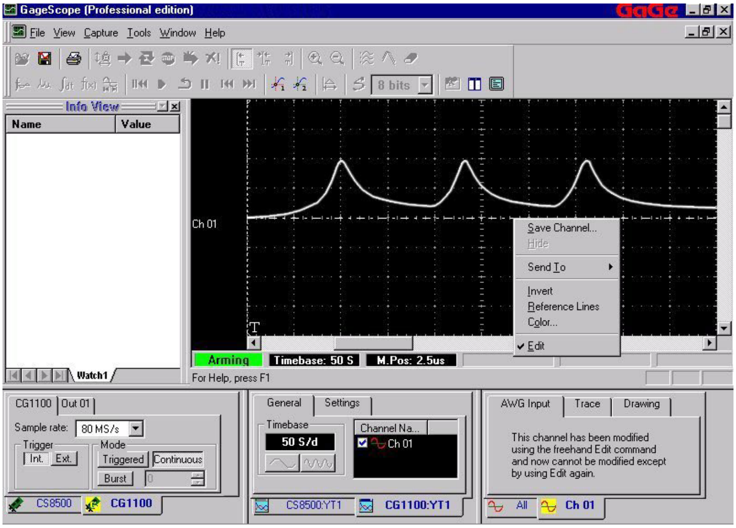
Double click on **Paste**. The copied portion of the trace will be pasted at the cursor location as shown below:



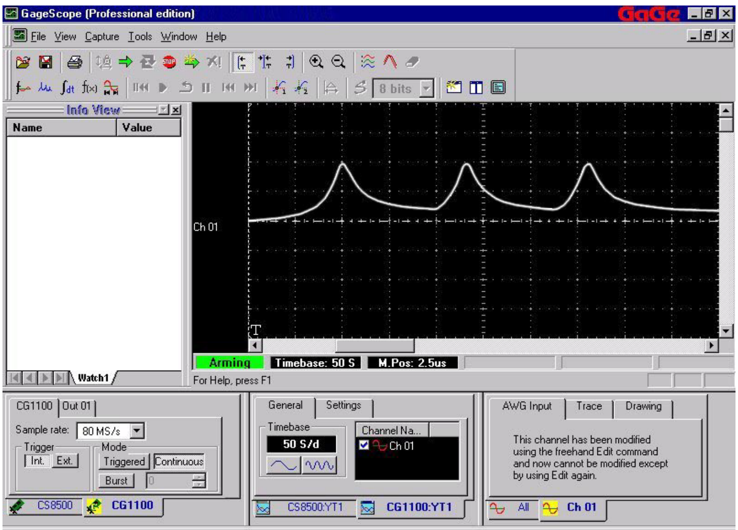
11 The part of the trace copied is again pasted at the end of the signal. Hide the left cursor by clicking on the left cursor button in the toolbar. The GageScope® display will show as follows:



12 When finished, you must exit the Edit mode. Right click on the zero reference line. The reference context menu will appear. The check mark besides **Edit** command shows that the Edit mode is currently active.



13 Click on **Edit** to deactivate the Edit Mode. The GageScope® window will appear as follows:



Observe that the normal operation has commenced since all disabled controls have been enabled.

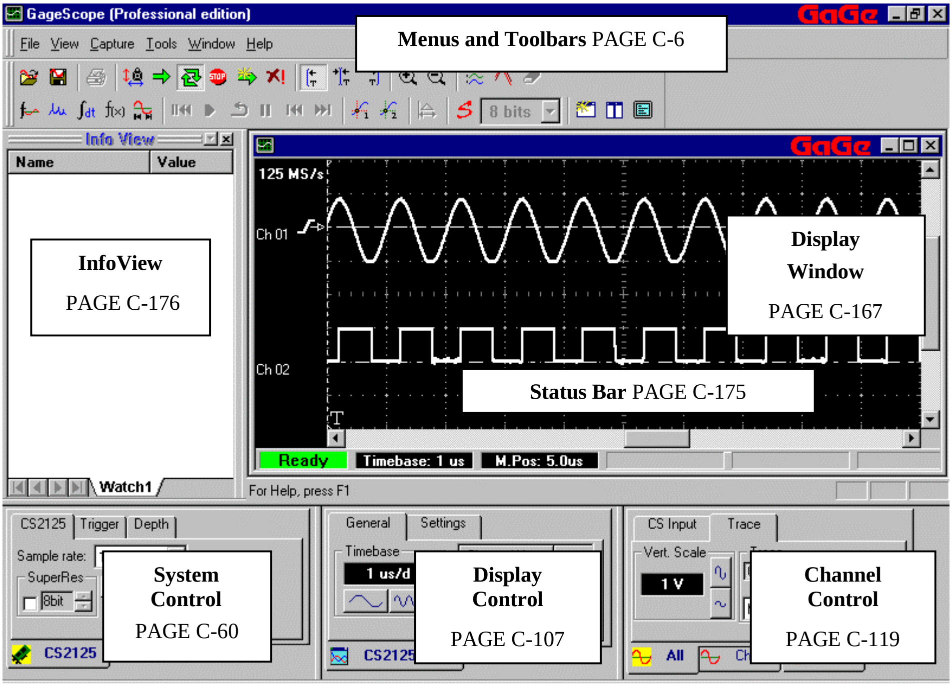
Reference

The Reference section contains information on every feature of GageScope®. If you are new to GageScope®, it is recommended that you try out the tutorials (part B of this manual).

Contents

Quick Reference

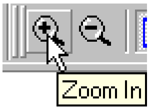
Use the diagram below to quickly find the reference section you are interested in.



Menu Commands

The following pages contain descriptions of menu commands (some of which have equivalent buttons in the toolbar). For descriptions of commands found in the toolbar only (and not in menus), see page C-47.

To quickly find out what a toolbar button does, position the mouse pointer over a button—a “tool tip” will appear with a description of the button.



File: Save Setup

The Save Setup command saves current GageScope® settings in a Setup file with the extension .INI.

Upon startup, GageScope® chooses several default settings in order to display a signal. As so many settings can be changed, it can be tedious to have to repeatedly fix everything to your liking. To solve this problem, GageScope® offers the ability to save all the current settings in a Setup file.

What is Saved?

Setup files record the following information:

- CompuScope card and CompuGen card's Display Settings: Channel mode, sample rate
- Channel Settings: input range, coupling, position, vertical scale, polarization, channel name, comments
- Trigger Settings: Trigger type, trigger level, timeout, trigger slope, depth units, post-trigger depth, pre-trigger depth
- Display Settings: Timebase, channel visibility

GageScope® for DOS Setup File Compatibility

The GageScope® setup file format (.INI) is not compatible with the GageScope® for DOS setup file format (.SET).

How it Works

1. Choose **Save Setup** from the File menu. The Save Setup dialog appears.
2. In the "File name" field, type in a file name.
3. Go to the folder where you would like to save your Setup file.
4. Click **Save** to save the Setup file.

File: Load Setup

The Load Setup command loads a previously saved Setup file. A Setup file contains the GageScope® settings which were current at time of saving. The Setup file has the extension .INI.

For details on what is saved in a setup file, see Save Setup on page C-7.

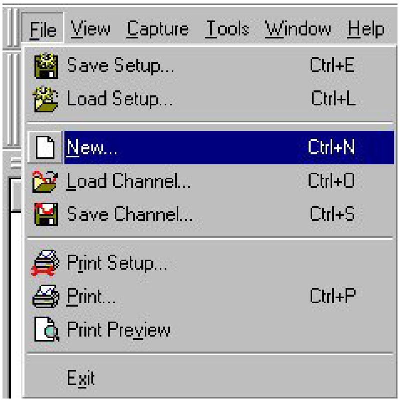
How it Works

1 Click **Stop**  in the Toolbar to stop the current acquisition.

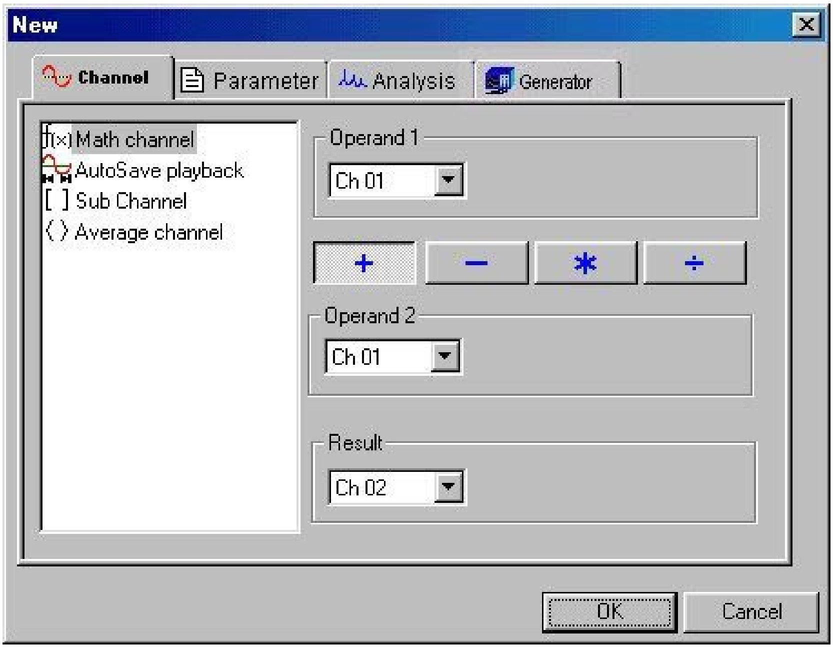
2 Choose **Load Setup** from the File menu. The Load Setup dialog appears.

3 Go to the folder where you saved your Setup files and choose a file from the list.

File: New



The **New** command opens the New dialog box. It imparts user the ability to use advanced computational features of GageScope® like Averaging, Parameter Extraction, Advanced Math, FFT analysis, etc.



How it works

1 The dialog box opens with **Cancel** tab selected as a default. Following options are available to the user:

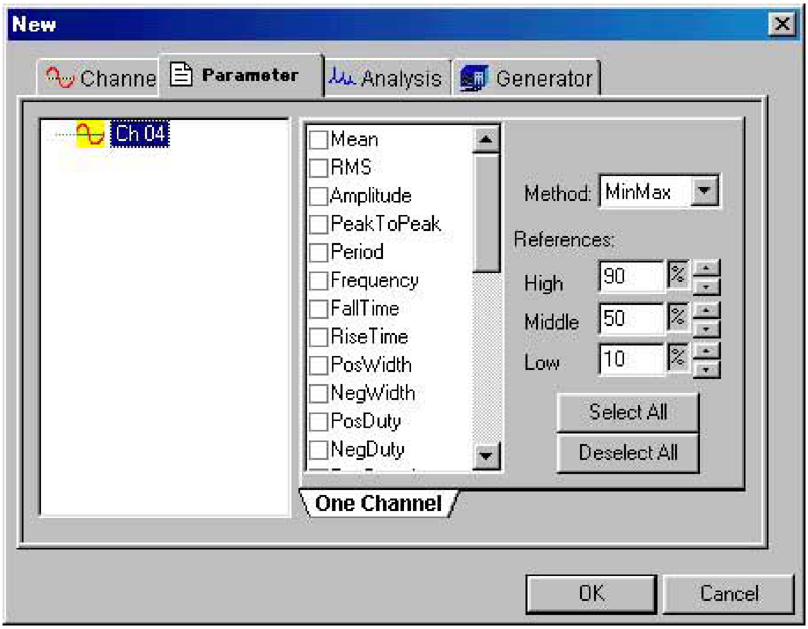
- **Math Channel** : Performs basic mathematical operations of addition, substraction, multiplication and division. Refer to discussion on page C-33 for further details on this option.

- **AutoSave Playback** : This command brings up the AutoSave Playback dialog, which allows you to locate the previously saved AutoSave records on your system as well as the channels to be played back. See AutoSave tool on page C-178 for more details.

- **SubChannel** : Subchanneling allows part of signal to be extracted based on a certain feature of interest. Refer to Channel Control Section on page C-122 and the Tutorial section on page B-105 for detailed discussion on **Subchannel**

- **Average Channel** : Acquired signals can be averaged using the Average Channel feature using several methods. See Averaging tool on page C-191 for more details.

2 To use the waveform parameter feature of GageScope®, click on **Parameter** tab in **New** dialog box. The following window appears:

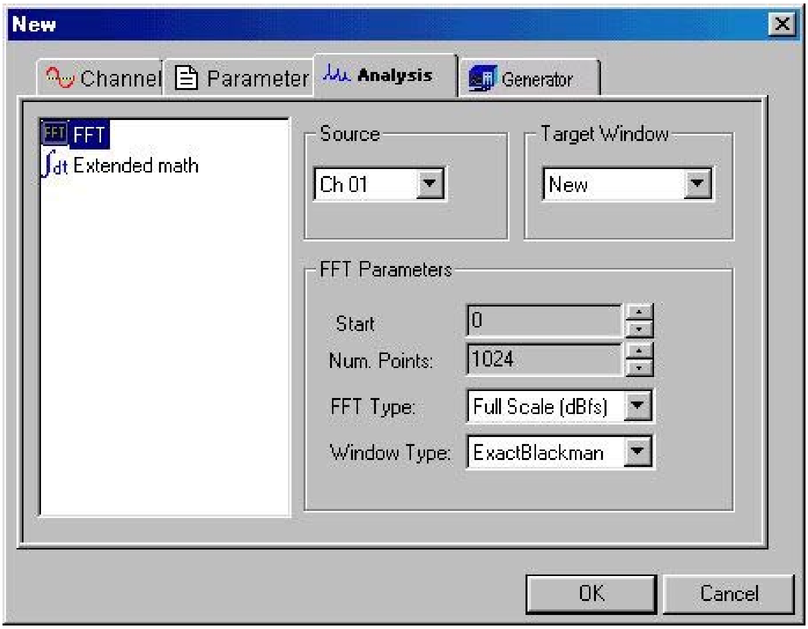


Parameter dialog box opens giving user options to choose from a list of signal parameters to be acquired. Parameters such as Frequency, Mean, RMS, Peak, Amplitude, etc can be computed automatically during signal acquisition.

This feature is only partially available in the Standard Edition of GageScope®. Professional Edition provides full Waveform Parameter measurement capability.

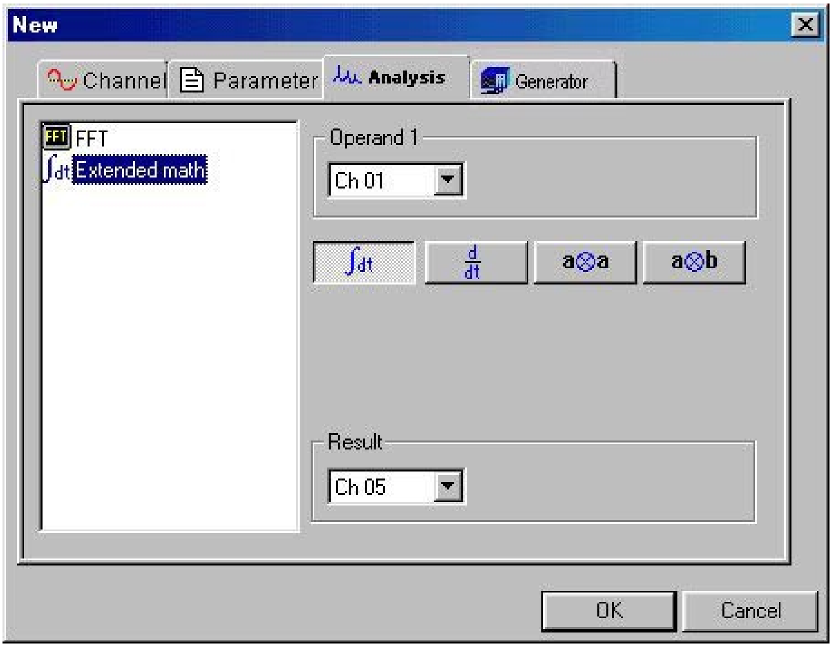
For further details and a comprehensive list of all parameters that are offered, see Waveform Parameters tool on page C-205.

3 Click **Analysis** tab to view the screen as follows:



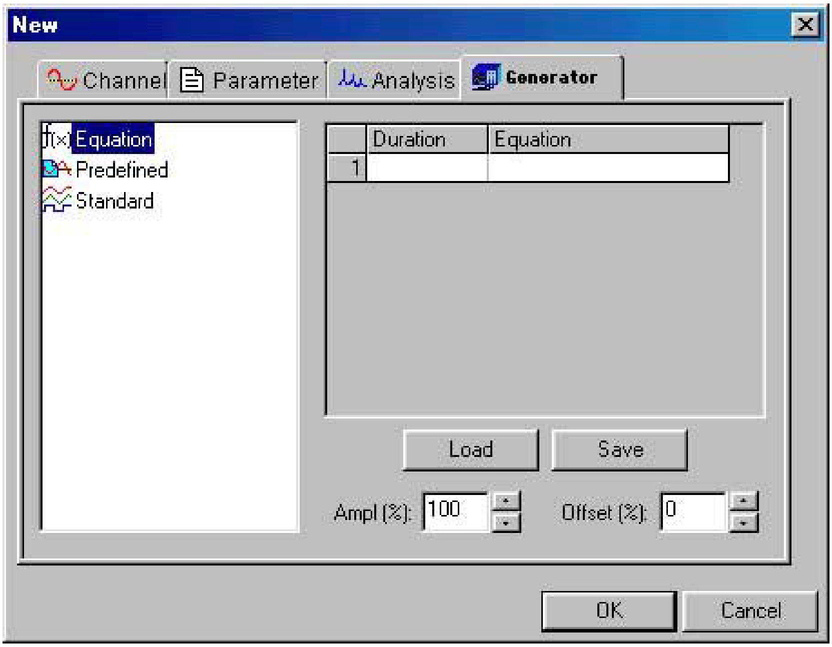
The two options available are the FFT and the Extended Math. Dialog box shows FFT as the default option. Refer to FFT Analysis tool on page C-215 for more details.

4 Clicking **Extended Math** option opens the Extended Math dialog:



A complete list of Extended Math functions is provided in the Extended Math tool on page C-220

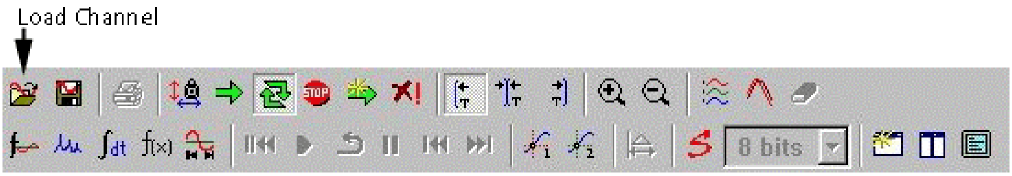
5 Click on **Generator** tab to open the Generator dialog box:



The **New** dialog box allows signals to be created using **Equation**, **Predefined** and **Standard** Methods.

Refer to Chapter 6 in the Tutorials or the AWG section of this guide for further details.

File: Load Channel



The Load Channel command opens a previously saved GageScope® signal file. GageScope® signal files have the extension .SIG. The channel is placed in the Display Window and assigned a unique channel number.

How it works

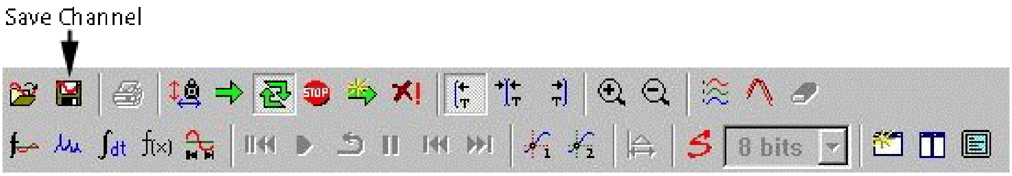
1. Choose **Load Channel** from the File menu to bring up the standard Windows Open dialog box; or, click on the Load Channel button in the toolbar.
2. Move to the folder that contains the .SIG file you want to open.
3. Click on the file.
4. Click **Open**.

Compatibility

GageScope® and GageScope® for DOS share the same .SIG file format. GageScope® can load .SIG files saved by GageScope® for DOS and vice-versa.

Neither GageScope® nor GageScope® for DOS can read an ASCII file. This format should be used for export purposes only.

File: Save Channel



The Save Channel command writes a channel to disk in the GageScope® signal file format. GageScope® signal files have the extension .SIG. You also have the option to save the signal as an ASCII text file, which you can import into a third-party program for further analysis.

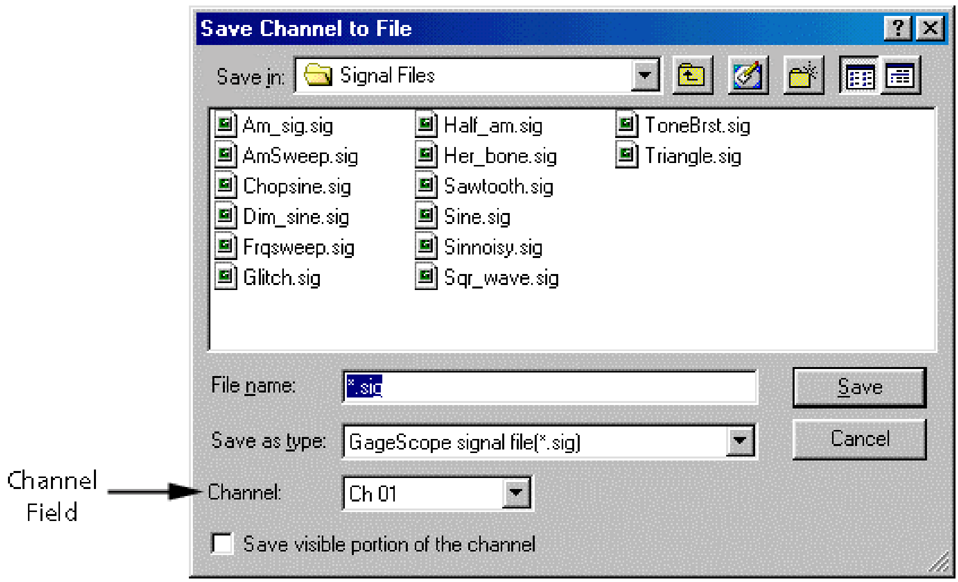
For detailed information on what is saved in a .SIG file, see Appendix A.

How it works

1 Choose **Save Channel** from the File menu to bring up the standard Windows Save As dialog; or, click on the Save Channel button in the toolbar.

You also can right-click on a channel to bring up the channel context menu, then choose **Save Channel**.

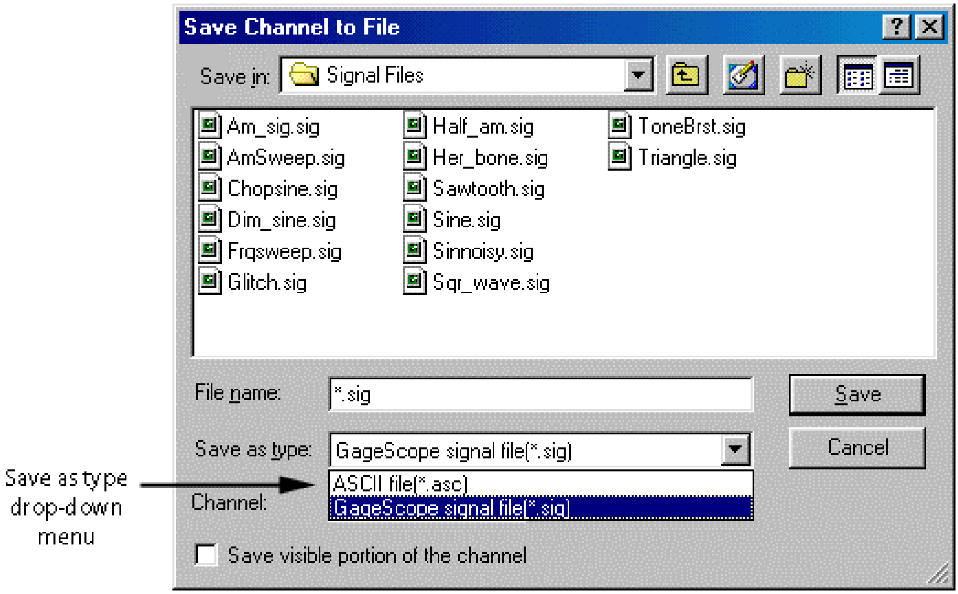
2 Notice the "Channel" field at the bottom of the dialog. This field tells GageScope® which channel to save. If necessary, select the channel you wish to save from this list.



3 If you chose **Save Channel** from the File menu or the toolbar, channel 1 is listed by default in the channel field. If you used the channel context menu (by right-clicking on a channel and choosing **Save Channel**), the channel you selected appears in this field automatically.

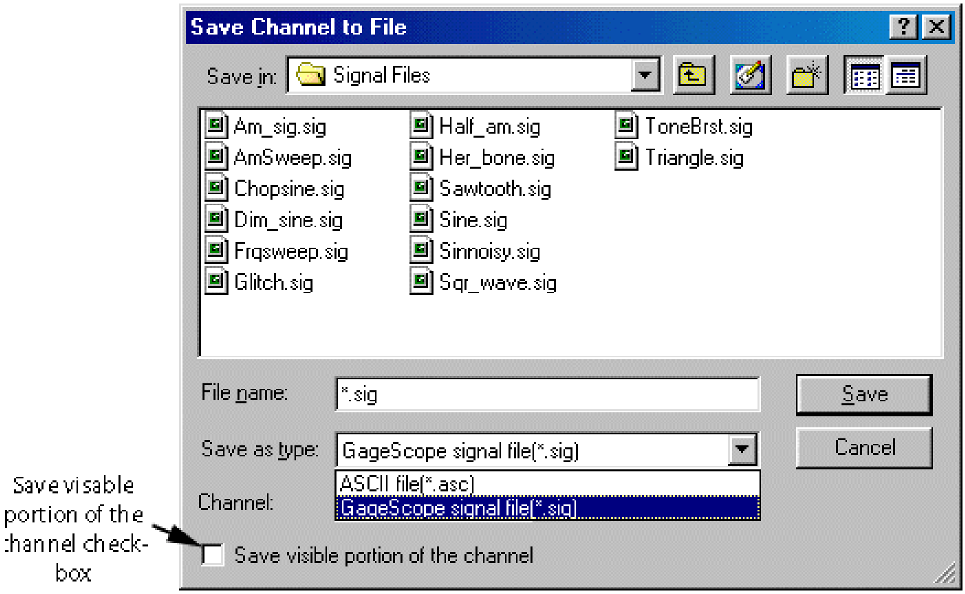
Note that you can also save all visible channels at once by selecting **All** in the Channel list.

4 When saving, the GageScope® signal file format (extension .SIG) is chosen by default. If you wish to save the signal as an ASCII text file, choose "ASCII File" from the **Save as type** drop-down menu.



Enter a name in the **File name** field. There is no need to enter the extension, as GageScope® will add the extension .SIG or .ASC automatically.

5 You have the option of saving only that portion of the channel that is currently visible in the Display Window. To specify this option, click the **Save visible portion of the channel** checkbox.



6 Move to the folder in which you wish to save the file. By default, GageScope® will save files to the Signal Files folder.

Click **Save**.

Compatibility

GageScope® and GageScope® for DOS share the same .SIG file format. GageScope® can open .SIG files saved with GageScope® for DOS and vice-versa.

Neither GageScope® nor GageScope® for DOS can read an ASCII file. This format should be used for export purposes only.

File: Print Setup

Print Setup allows you to change the orientation of the paper and other settings.

How it Works

1. Choose **Print Setup** from the File menu. The Print Setup dialog appears.
2. To change the orientation of the paper, click on the **Portrait** or **Landscape** radio buttons in the Orientation area of the dialog.
3. Choose a printer from the drop-down list. To change the printer’s settings, click on the **Properties** button.
4. To change the size of the paper, click on the **Size** drop-down menu in the Paper area of the dialog.
5. To change which tray to take the paper from, click on the **Source** drop-down menu in the Paper area of the dialog.

File: Print

The Print command sends a copy of the Display Window to the printer.

How it Works

1. Click **Stop** in the Toolbar or in the Capture menu. (You will not be able to print unless GageScope® has stopped acquiring data.)
2. Choose **Print** from the File menu or the toolbar. The Print dialog appears.
3. Choose a printer from the drop-down list. To change the printer's settings, click on the **Properties** button.
4. To print more than one copy, enter a number in the **Number of Copies** field.
5. Click **OK** to begin printing.

Note

To get the largest possible printout, it is recommended that you change the page orientation to landscape using the **Print Setup** command.

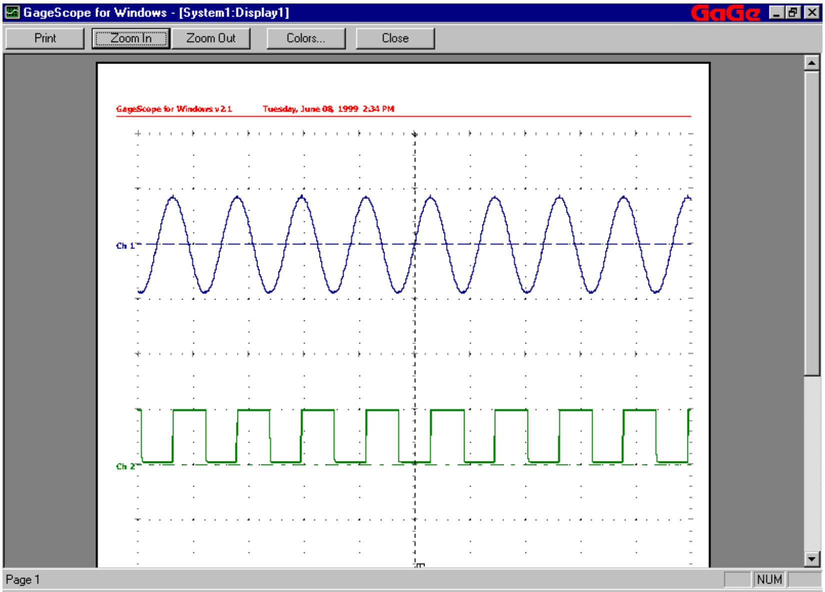
File: Print Preview

Print Preview allows you to preview how a file will look when you print it, based on the current Print Setup.

How it Works

1 Choose **Print Preview** from the File menu.

Your current signals will be displayed as they will be printed.

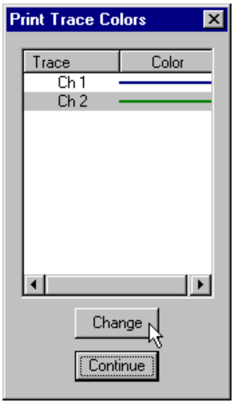


2 Click **Print** to accept the current print settings.

3 Click **Zoom In** or **Zoom Out** to view different parts of the previewed page.

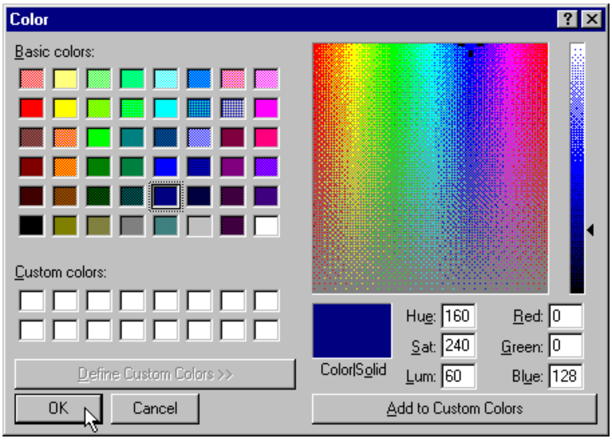
4 If you have a color printer installed on your system, click **Colors** to edit the colors to be used for each channel trace. If you do not have a color printer installed, this option will not be available.

5 You will see the Print Traces dialog.



To edit the colors, click on a channel to highlight it and then click **Change**.

6 The Color dialog appears.



Click on another color and click **OK** to accept. The new colors will be displayed in the Print Preview window.

7 Click **Close** to exit Print Preview.

File: Exit

The Exit command quits GageScope®.

How it works

1 Click on the **File** menu.

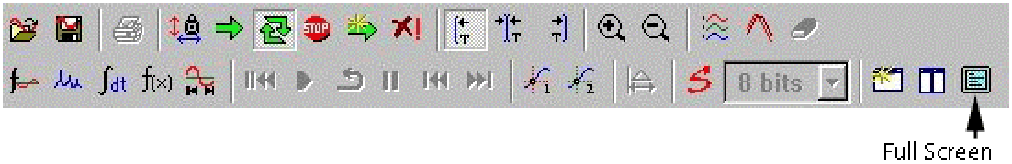
2 Click on **Exit**.

You can also click the **X** at the top right of the application window.

Note

GageScope® will not ask you if you wish to save any unsaved channels. If you exit without first saving unsaved channels, those channels will be lost.

View: Full Screen



The Full Screen command hides the System, Display, Channel, and InfoView controls, allowing the Display Window to expand to fill the screen. The menus, toolbar and status bar remain visible.

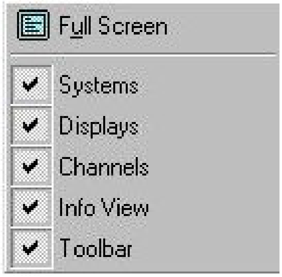
How it works

1 Click on the **Full Screen** button in the toolbar; or, choose **Full Screen** from the View menu. All controls are hidden and the Display Window expands to take up the remaining space.

2 To return GageScope® to normal, click on the **Full Screen** button once more.

View: Systems, Displays, Channels, InfoView, Toolbar

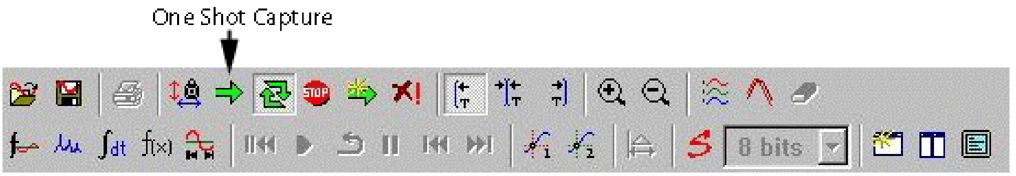
Click **View** on the menu bar. The View submenu appears:



These commands hide or show the System Control, Display Control, Channel Control, InfoView Window and Toolbar.

A checkmark next to an item indicates that the display element is currently visible; absence of a checkmark indicates that the element is hidden.

Capture: One Shot



In one shot mode, GageScope® waits for a trigger event, acquires and displays the data, then stops. This allows for unattended one shot capture of both pre- and post-trigger data.

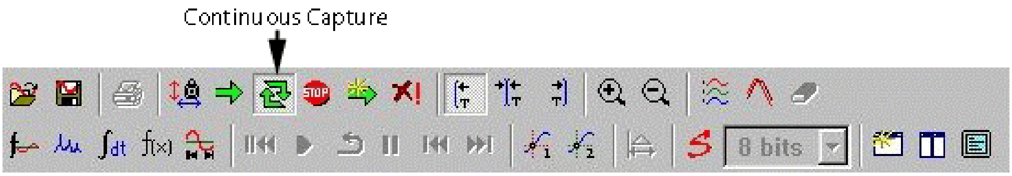
- To set GageScope® to one shot capture, select **One Shot** from the Capture menu; or, click on the **One Shot** button in the toolbar. The capture mode is already set to one shot if there is a checkmark next to **One Shot** in the Capture menu, or if the **One Shot** button is enabled.
- To cancel a one shot capture, select **Stop** from the Capture menu, or click on the **Stop** button in the toolbar.
- When you choose one shot mode, GageScope® automatically disables the Timeout setting in the Depth tab of the System Control.

About One Shot Mode and Stop

While in one shot mode, if you click on **Stop** in the toolbar, GageScope® completes the current acquisition, then stops.

If you select **Stop** while in one shot mode but a trigger event has yet to occur, GageScope® issues a software trigger, i.e., it immediately acquires the current signal, then stops. The data you see is that which existed when you clicked on **Stop**.

Capture: Continuous



In Continuous Capture Mode, GageScope® repeatedly waits for a trigger event, acquires and displays the data, then resets the CompuScope card to wait for another trigger event. This is the default capture mode.

- To set GageScope® to Continuous Capture, select **Continuous** from the Capture menu; or, click on the **Continuous** button in the toolbar.

- To cancel continuous capture, select **Stop** from the Capture menu; or, click on the **Stop** button in the toolbar.

When you select Continuous mode, GageScope® automatically enables the Timeout setting in the Depth tab of the System Control.

About Continuous Mode and Stop

If you click on **Stop** in the toolbar while in continuous mode, GageScope® completes the current acquisition, then stops.

If you select **Stop** while in continuous mode but a trigger event has yet to occur, GageScope® issues a software trigger, i.e., it triggers immediately, acquires the current signal, then stops. The data you see is that which existed when you chose **Stop**.

About Continuous Mode and the Trigger Timeout

In the event of a timeout while in continuous capture mode, GageScope® issues a software trigger, i.e., it triggers immediately upon timeout, acquires the current signal, then resets the CompuScope card to await another trigger event.

Capture: Stop

Choose **Stop** from the Capture menu or click on the **Stop** button in the toolbar to interrupt the current capture mode. Capturing has already stopped if there is a checkmark next to **Stop** in the Capture menu or if the **Stop** button is enabled.

When an acquisition is stopped, GageScope® completes the current acquisition, then stops. If a trigger event has yet to occur, GageScope® triggers immediately, acquires the current signal, then stops. The data you see is that which existed when you clicked on **Stop**.

By contrast, pressing **Abort** (see below) means that the current acquisition will not be completed.

Capture: Force Trigger

Choose **Force Trigger** from the Capture menu or click on the **Force Trigger** button in the toolbar to force a trigger.

Note on Force Trigger and Capture Modes

- When you are in Continuous Capture mode and GageScope® is waiting for a trigger, if you click Force Trigger, the Status Bar will very briefly display the message “Ready” on a blue background. Thereafter, the Status Bar will display “Stop” on a blue background, indicating a trigger was forced.
- When you are in One Shot capture mode and you click Force Trigger, the Status Bar will display “Stop” on a blue background, indicating a trigger was forced.

Capture: Abort

Choose **Abort** from the Capture menu or click on the **Abort** button in the toolbar to abort the current acquisition immediately. While pressing **Stop** causes GageScope® to complete the current acquisition first, pressing **Abort** means that the current acquisition will not be completed. This is helpful for very long acquisitions. The data you see is that which existed when you clicked on **Abort**.

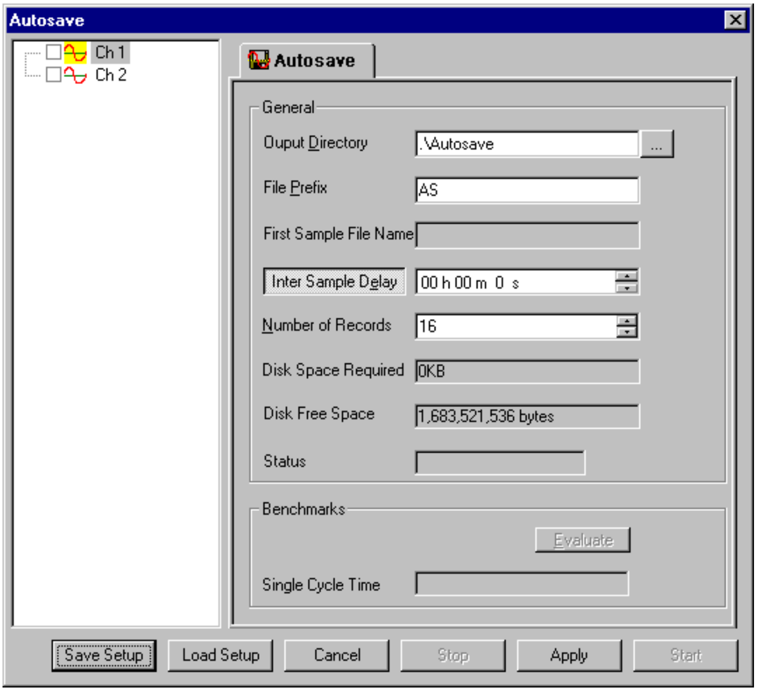
See page C-44 for more information on customizing the **Abort** command.

Capture: AutoSave Setup

Choose **AutoSave Setup** from the Capture menu to specify settings for an AutoSave acquisition.

How it works

1 Choose **AutoSave Setup** from the Capture menu. The AutoSave Setup dialog appears.



The AutoSave dialog allows you to select the channels for which AutoSave should be set up, as well as benchmark your settings to determine how long the system will need to perform a single AutoSave cycle using your settings.

2 To benchmark your AutoSave session, select the channels you intend to save to disk and then click on **Evaluate**. GageScope® will determine how long it will take to perform the specified AutoSave session.

3 After you have specified the desired AutoSave settings, click **Start** to begin the AutoSave process.

4 To save your current AutoSave setup, click **Save Setup**.

5 You will see the Save Setup dialog, allowing you to specify the directory for the setup file. The default directory is **C:\Gage\Gagescope\AutoSave**, and the default AutoSave setup file name is **GagescopeAS.asf**.

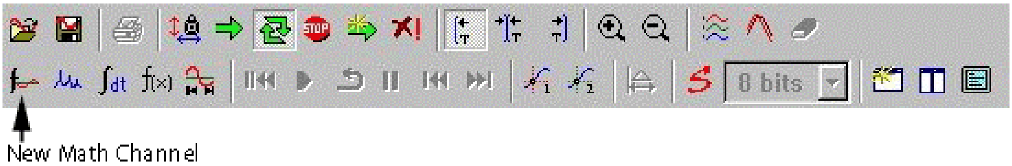
6 To load a previously saved AutoSave setup file, simply click **Load Setup** in the AutoSave Setup dialog. This brings you to an Open dialog, allowing you to select an .asf file to load.

Note

If you wish to change hardware settings before beginning the AutoSave, you must exit the AutoSave Setup dialog. To preserve the AutoSave setup changes you have made, click **Apply**, then **Cancel**. Make the necessary hardware changes. When you re-open the AutoSave Setup dialog, your settings will have been retained.

For more information on the AutoSave tool, see page C-178.

Tools: Math

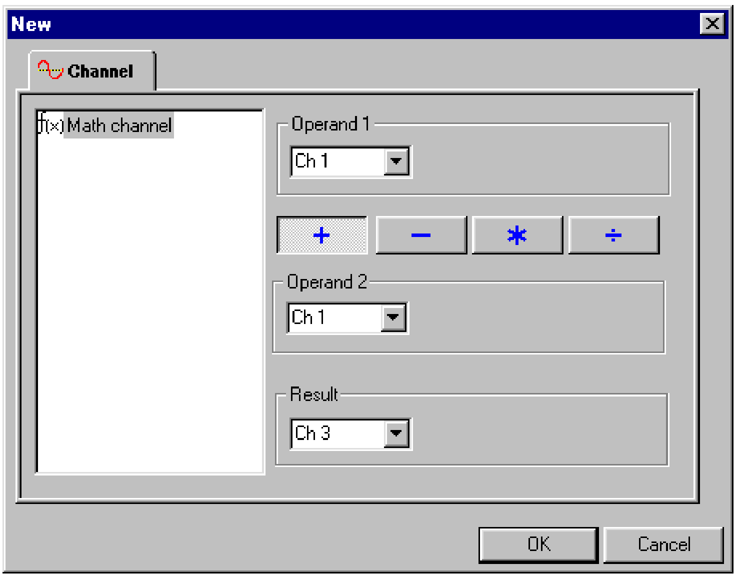


The Math command creates a new channel by mathematically combining two existing channels.

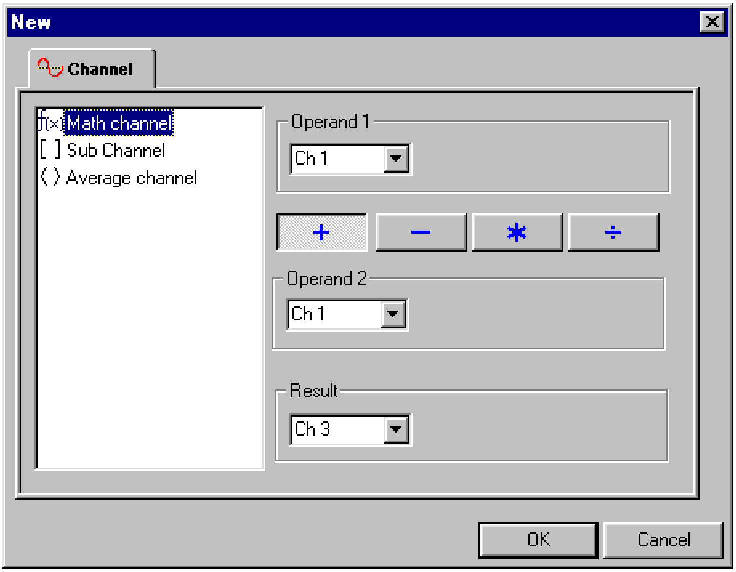
How it works

1 Click on the Tools menu, then click on **Math**. Or, you can click on  in the toolbar.

2 When **Math** is clicked from the drop down menu, following Channel dialog appears:



3 When you click on  in the toolbar, the following dialog box appears:



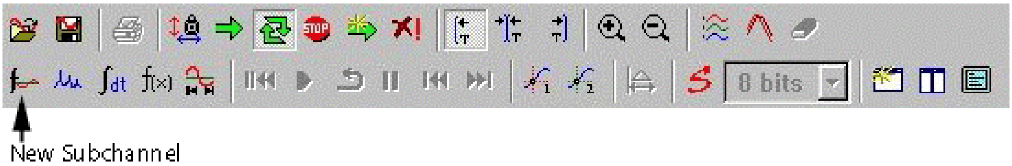
4 In either case, choose a channel from the **Operand 1** drop-down menu

5 Click on one of the four buttons:
+ (addition), - (subtraction), * (multiplication), ÷ (division).

6 Choose a channel from the **Operand 2** drop-down menu.

7 Choose a result channel from the **Result** drop-down menu.Click **OK** when finished or **Cancel** to abort the changes.

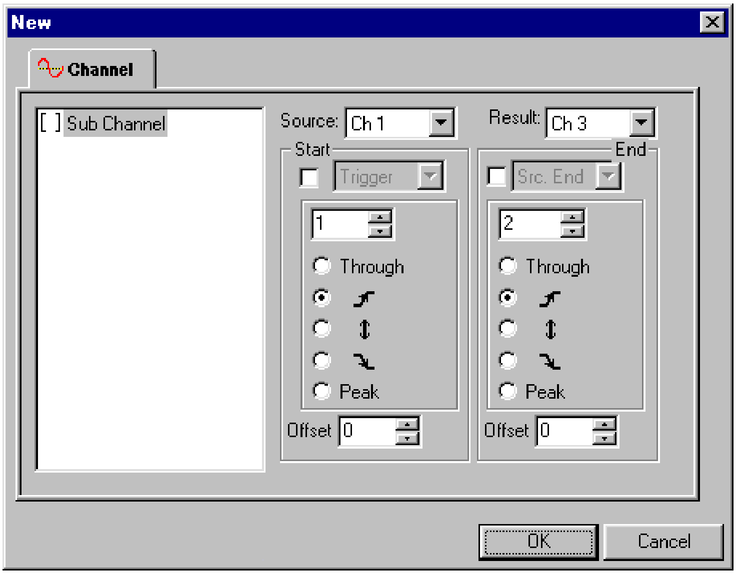
Tools: SubChannel



This command brings up the new SubChannel dialog box, which allows the user to pick out a part of the signal depending on the feature of interest.

How it works

1 Click on the **Tools** menu and select **Subchannel**. You can also select  button in the Toolbar. The New Channel Dialog will appear:



2 Choose from the available options that appear in the new channel dialog box

3 Select a channel from the **Result** drop down menu

4 Click **OK** when finished or **Cancel** to abort the changes.

Refer to **Channel Control** Section on page C-122 and the Tutorial section for detailed discussion on **Subchannel**.

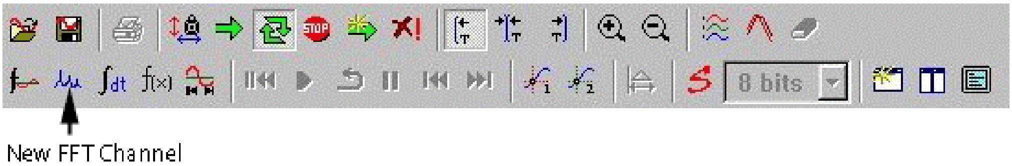
Tools: Averaged Channel

This command brings up the new Averaged Channel dialog box, which allows the user to carry out the averaging of the selected channels using several methods available.

Note: The full functionality of the Averaging analysis tool is available only when you have purchased and installed the Professional Edition of GageScope® software. There, however, are a few features included in the GageScope® Standard Edition.

See Averaging tool on page C-191 for more details.

Tools: FFT

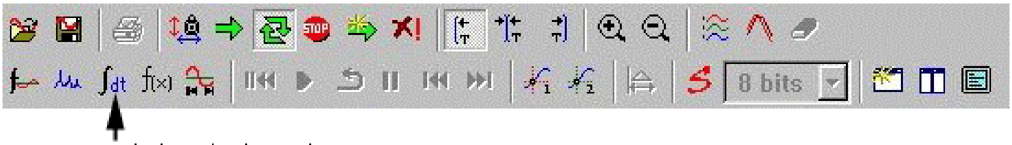


This command brings up the New FFT Channel dialog, which allows you to specify the settings for a new FFT signal analysis.

Note: The full functionality of the FFT analysis tool is available only when you have purchased and installed the Professional Edition of GageScope® software. There are a few features included in the Standard Edition of GageScope®.

See FFT Analysis tool on page C-215 for more details.

Tools: Extended Math



New Extended Math Channel

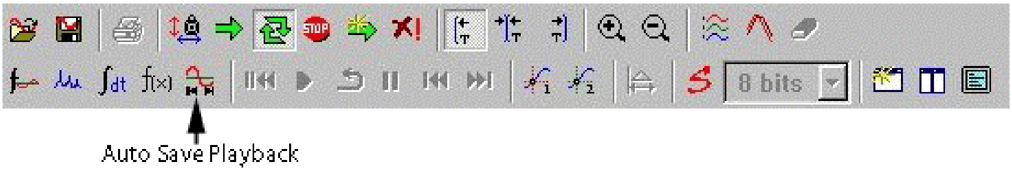
This command brings up the New Extended Math dialog, which allows you to choose from one of the following functions:

- Integration of the selected channel
- Differentiation of the selected channel
- Autocorrelation of the selected
- Cross-correlation of the selected channel

Note: You must have purchased and installed the Professional Edition of GageScope® software in order to make use of this feature. This advanced tool is not available in the Standard Edition of GageScope®.

See Extended Math tool on page C-220 for more details.

Tools: AutoSave Playback

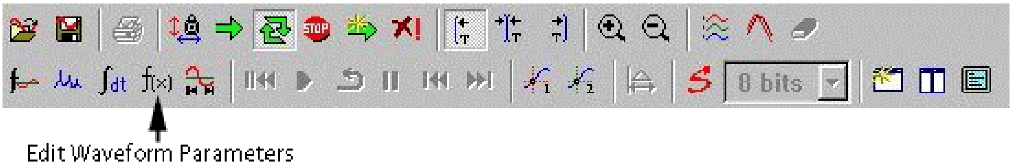


This command brings up the AutoSave Playback dialog, which allows you to locate the previously saved AutoSave records on your system as well as the channels to be played back.

Note: The full functionality of the AutoSave tool is available only when you have purchased and installed the Professional Edition of GageScope® software. There are a few features included in the Standard Edition of GageScope®.

See AutoSave tool on page C-178 for more details.

Tools: Waveform Parameters



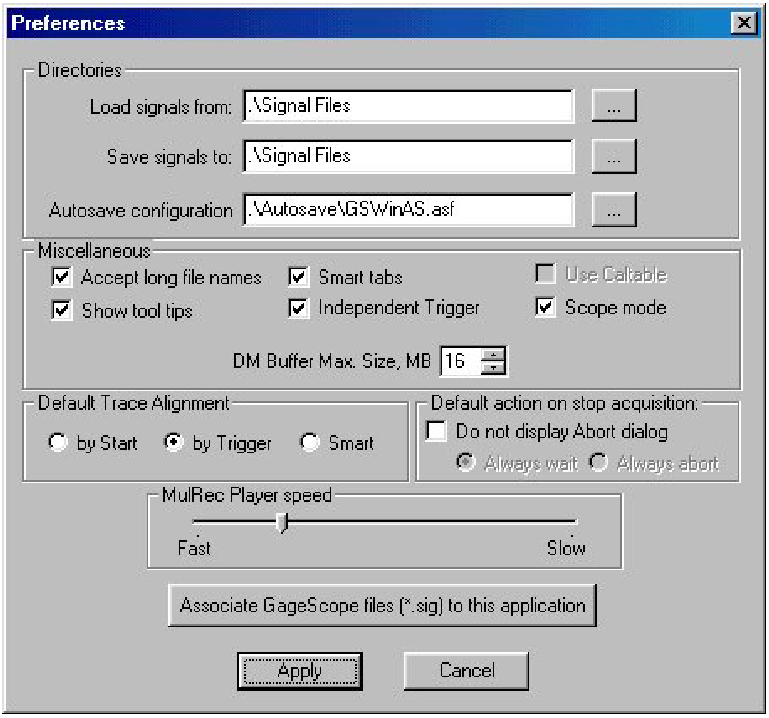
This command brings up the Edit Waveform Parameters dialog, which allows you to change which parameters are shown in the InfoView windows.

Note: The full functionality of the Waveform Parameters tool is available only when you have purchased and installed the Professional Edition of GageScope® software. There are a few features included in the Standard Edition of GageScope®.

See Waveform Parameters tool on page C-205 for more details.

Tools: Preferences

This command brings up the Preferences dialog, which allows you to change GageScope® preferences.



Directories

Allows you to specify the directories for loading signals, saving signals, and saving AutoSave configuration files (*.asf). The defaults are:

- **Load signals from:** C:\Gage\Gagescope\Signal Files
- **Save signals to:** C:\Gage\Gagescope\Signal Files
- **AutoSave configuration file name:** C:\Gage\Gagescope\AutoSave\GagescopeAS.asf

To change any of these settings, click on .

Miscellaneous

- **Accept long file names:** When checked, GageScope® will allow long file names when saving files. By default, this item is checked.

- **Show tool tips:** When checked, GageScope® will display “Tool Tips” when the mouse is positioned over a toolbar button. By default, this item is checked.

- **Use Caltable:** When checked, GageScope® will use a **Calibration Table** to set gains and offsets on the CompuScope card(s) instead of using Auto-Calibration. Not recommended for normal use.

  This item is available only if GageScope® detects a Calibration Table file (filename caltable.dac) in the proper folder. For Windows NT, this file should be kept in the Windows\System32\Drivers folder. For Windows 95/98, this file should be kept in the Gage\GageScope folder.

- **Smart tabs:** When checked, GageScope® will “remember” which Channel Control tab you were browsing when you move from one channel tab to another. By default, this item is checked.

- **Independent Trigger:** For Master/Slave systems of selected CompuScope cards only. When this item is checked, you will be able to set not only the Master board channels as trigger sources, but also the Slave board channels. This is very useful in applications that require a multi-board system to trigger from any one of the channels. By default, this item is checked.

- **Scope Mode:** When selected, this option links the sample rate and input range settings to the timebase and vertical scale selected by the user. This provides a control method very similar to that of stand-alone digital oscilloscopes on the market.

  Input range is set to be greater than or equal to +/- 4 times the vertical scale.

  Sample rate is set to be greater than the value required to display more than 4 samples per division.

  You can also set Scope Mode by clicking on the tool.

- **DM Buffer Max. Size:** This option is used whenever you have a CompuScope card with more than 16 Meg of acquisition memory.

  The Deep Memory Buffer Max. size refers to allocating, for every channel, actual physical memory present in the system, that will be used for displaying the acquired signals in GageScope® .

  The minimum and maximum DM Buffer size are 8 and 64 Mega Bytes (MB) per channel respectively.

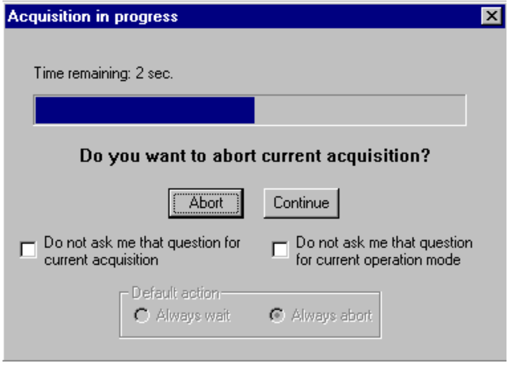
  Care must be exercised in allocating a value in the DM Buffer max size text box. You must make sure that the total memory available in the PC is significantly greater than the DM maximum buffer size.

  Please refer to the chapter titled **Decimation** in the tutorial section of this manual for detailed discussion on how to use the Deep Memory option.

Default action on stop acquisition

- **Do not display Abort dialog:** When this item is unchecked, GageScope® will display the Abort dialog when the user issues a command during a very long acquisition. By default, this item is unchecked.

  When the Abort dialog is displayed, you must choose between **Abort** and **Continue**. Clicking **Continue** means that the current acquisition will be completed before carrying out further commands.



  When **Do Not Display Abort Dialog** is checked, you may select between the following options:

- **Always wait:** GageScope® will always wait until the acquisition is completed before carrying out further commands. This is the default setting.

- **Always abort:** GageScope® will always abort the acquisition in order to carry out further commands.

Default Trace Alignment

When displaying a saved signal that does not cross the trigger marker, you can specify where on the screen that signal will be displayed.

- **by Start:** GageScope® will display the signal at the Start (at the Trigger Marker, or time zero).
- **by Trigger:** GageScope® will display the signal at its original time position relative to time zero. This is the default setting.
- **Smart:** If the signal crosses the trigger marker, GageScope® will display the signal at its original time position relative to time zero. If the signal does not cross the trigger marker, GageScope® will display the signal at the Start (at the Trigger Marker, or time zero).

Align by Trigger and **by Start** are also available in the Drawing tab of the Channel Control when a signal is displayed which does not cross the trigger marker.

MulRec Player Speed

When playing back the Multiple Records, set the speed to range between **Fast** and **Slow**. Use the speed tab to set an appropriate speed as desired.

Note: Use relatively Fast MulRec player speeds when you need to quickly browse through the records. Use Slower playback speeds when you wish to look in the records for some feature of interest.

Associate GageScope® files to this application

When selected, this option associates GageScope® files to the current application.

Window: Window Menu Commands

The Window menu and the Window toolbar buttons give you access to commands that control the Display Window.

New Display	Creates a new Display Window. Each Display Window is independent of the others, in that it can have its own timebase, colors, etc.
Cascade	Rearranges and resizes all the Display Windows, offset, one on top of the other.
Tile Horizontally	Rearranges and resizes Display Windows horizontally.
Tile Vertically	Rearranges and resizes all the Display Windows vertically.
Left Bar	Hides or shows the left bar of the Display Window.
Status Bar	Hides or shows the status bar of the Display Window.

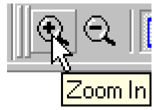
Notes

- A list of all Display Windows is located at the end of the Window menu.
- To close a Display Window, click on the small **x** at the top right of the window.
- You cannot close the last open window.

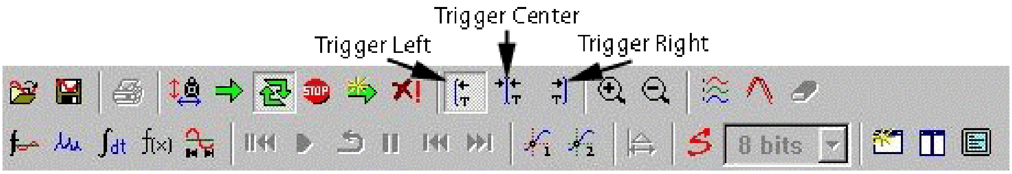
Toolbar Commands

The following pages contain descriptions of commands found only in the toolbar (and not in any menu).

To quickly find out what a toolbar button does, position the mouse pointer over a button—a “tool tip” will appear with a description of the button.



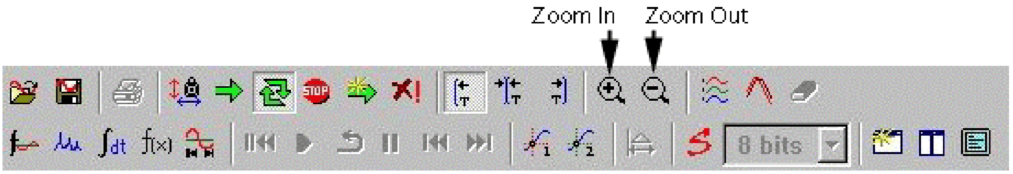
Toolbar: Trigger Align Buttons



The trigger align buttons automatically repositions the contents of the Display Window so that the trigger marker is either at the left, center, or right.

- The **Trigger Left Align** button positions the trigger marker to the left of the Display Window.
- The **Trigger Center Align** button positions the trigger marker to the center of the Display Window.
- The **Trigger Right Align** button positions the trigger marker to the right of the Display Window.

Toolbar: Zoom In / Zoom Out



The zoom tools allow you to magnify or demagnify an area of the Display Window. This allows for inspection of particular parts of a waveform.

Zoom In

The **Zoom In** tool magnifies the area you select. To use this tool:

1. Click on the **Zoom In** button in the toolbar.
2. Position the mouse pointer to the left of the desired portion of the waveform.
3. Hold down the mouse button. A vertical line will appear where you first click.
4. Drag the mouse to the right of the portion of the signal you wish to see. A second line will follow the movement of the mouse.
5. Let go of the mouse button. The area you selected is magnified.

When magnifying a particular area, the zoom tool actually decreases the timebase in order to display the data within your selection, then centers the area in the Display Window. Note that zooming changes the horizontal scale only—it does not affect the vertical scale of the channels.

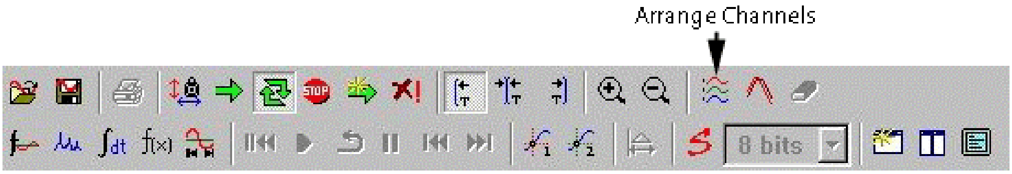
Zoom Out

The **Zoom Out** tool reduces the magnification (and consequently increases the timebase) of the Display Window. To use this tool:

1. Click on the **Zoom Out** button .
2. Click somewhere in the Display Window. This decreases the timebase by one increment. Continue clicking with the Zoom Out tool to further reduce the magnification.

Alternatively, you can use the Increase Timebase button in the Display Control to reduce the magnification of the Display Window.

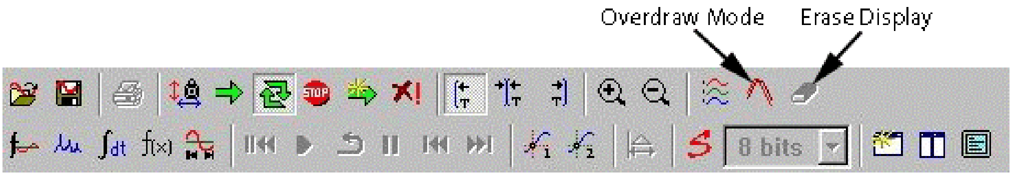
Toolbar: Arrange Channels Button



Clicking on the Arrange Channels button automatically re-organizes visible channels in order of their channel number. Hidden channels are ignored.

You can also right-click in the Display Window and choose **Arrange Traces** from the context menu.

Toolbar: Overdraw Mode and Erase Display

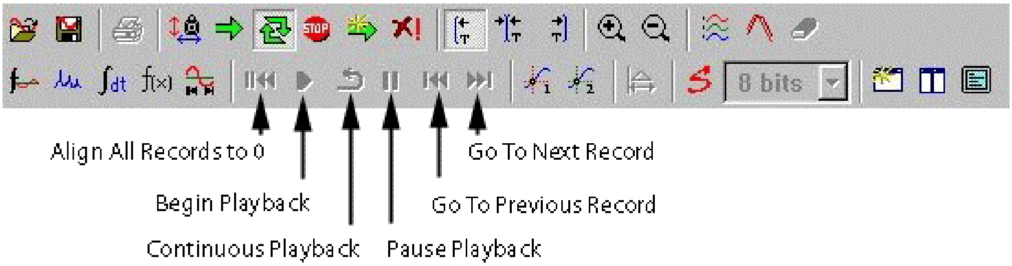


In overdraw mode, GageScope® does not erase a trace before updating the Display Window. This causes two or more traces to show where there would normally be one.

Click on the **Overdraw Mode** button to enable this feature. Disabling Overdraw Mode removes all the extra traces.

When Overdraw Mode is active, the **Erase Display** button (located to the right of the Overdraw Mode button) is available. Erase Display will erase all the duplicate traces without turning Overdraw Mode off. This button is grayed out when Overdraw Mode is disabled.

Toolbar: Multiple Record Buttons



The Multiple Record buttons allow you to move from one record to another when viewing signals captured in Multiple Record mode.

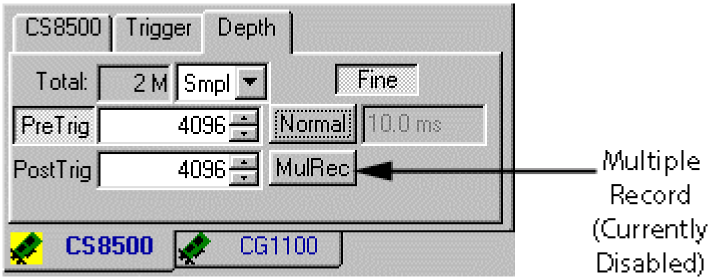
 Align all channels to first record
 Start playback
 Continuous playback
 Pause playback
 Go to the previous record
 Go to the next record

Note: These buttons are also used for AutoSave Playback. See the AutoSave tool on page C-178.

About Multiple Record

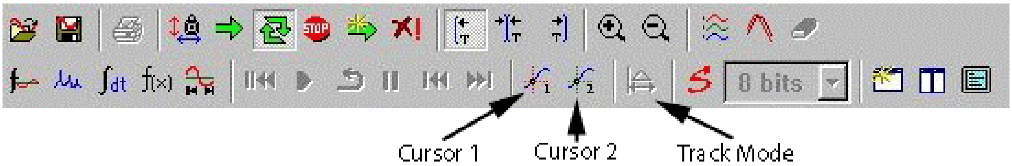
Multiple Record comes as a standard feature or as an option on most CompuScope cards, and is not available on the CompuScope LITE. If your CompuScope card does not have this capability, these buttons will be disabled.

The Multiple Record setting is located in the Depth tab of the System Control.



For more information on this topic, see Multiple Record on page C-88.

Toolbar: Cursor Buttons



Cursors allow the user to measure the difference between two locations on the same signal or on different signals.

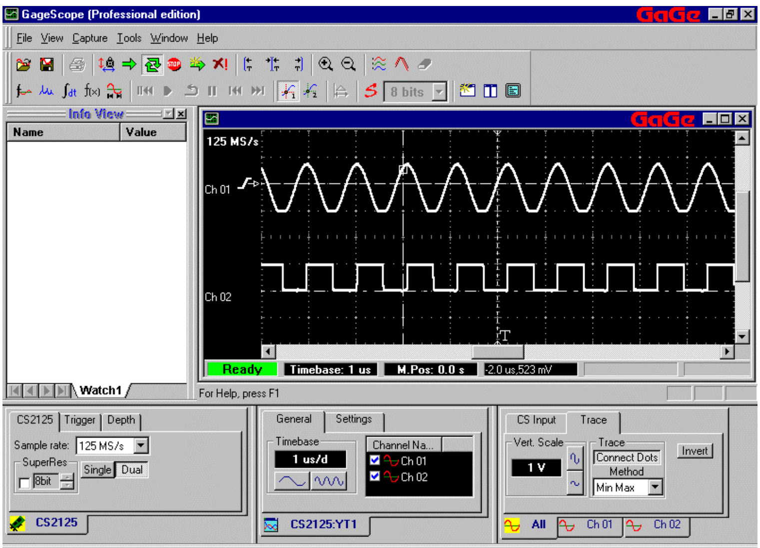
Enabling Cursors

Cursors are disabled by default. To enable a cursor, click on either the **Cursor 1** or **Cursor 2** buttons in the Toolbar.

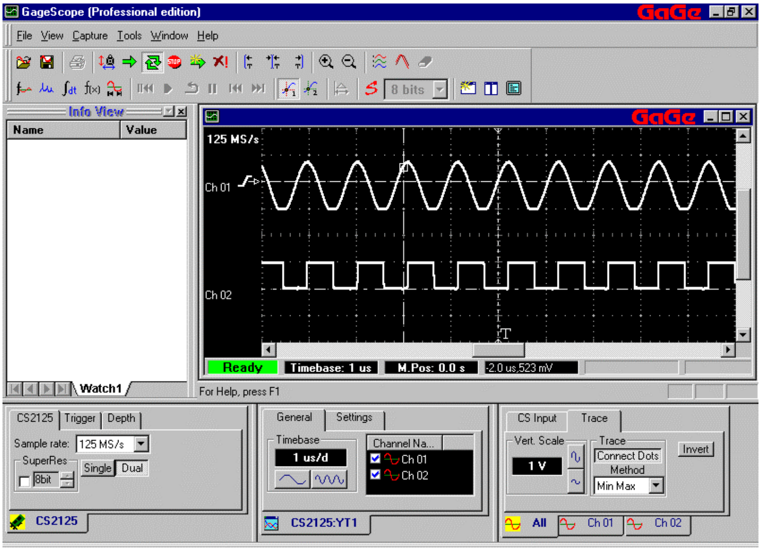
Once a cursor is enabled, a cursor (represented by a small box) riding on the waveform appears on channel 1, and is positioned halfway between the center line and left (Cursor 1) or right (Cursor 2) edge of the display.

Moving Cursors

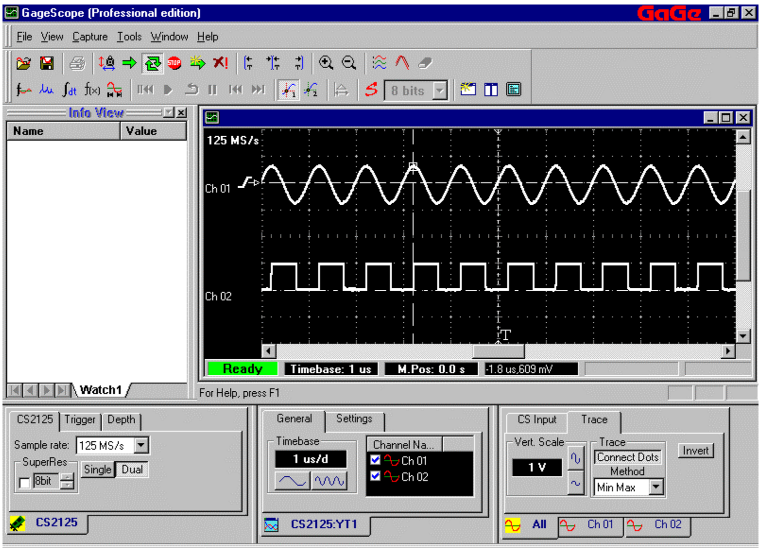
1 Click on one of the cursor buttons  to enable a cursor, if you have not already done so.



2 Position the mouse pointer somewhere on the cursor line. The pointer should change to a drag pointer. Note: You do not have to be on the small box – anywhere on the line will do.

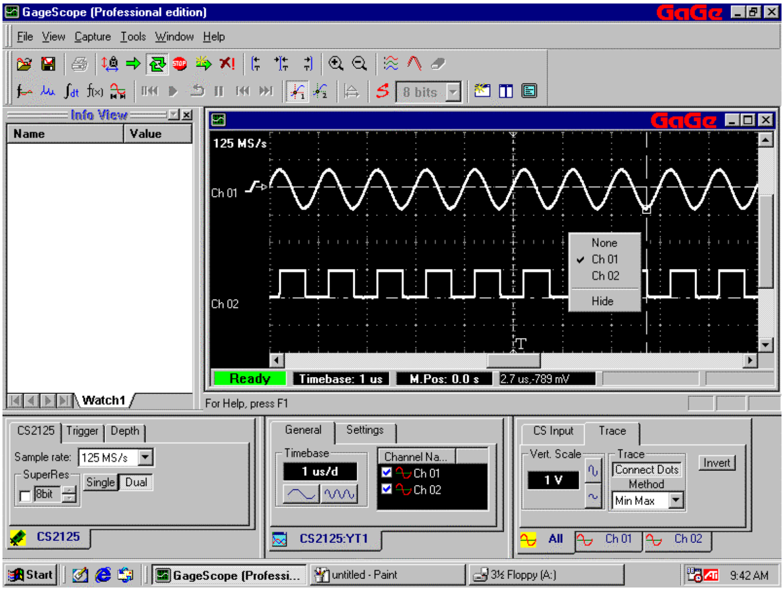


3 Drag the cursor line to the left or right of the trigger marker line. The cursor will follow.



Attaching the Cursor to a Different Channel

1 Click with the right mouse button on the cursor line. This brings up the Cursor context menu.



2 Click with the left mouse button on the channel you wish to use. A checkmark next to the channel indicates which channel the cursor is on.

The **None** option will leave the cursor line enabled without being attached to any particular channel. You can still move the cursor’s vertical line.

The **Hide** option will turn the cursor off, as if you had clicked the **Cursor 1** or **Cursor 2** buttons.

Getting the Cursor Back After Scrolling Off-screen

With the exceptionally deep memory buffers offered on CompuScope cards, it is easy to make a cursor measurement and then scroll far away from this area to inspect another portion of the display.

The cursors you had set are still in place and the readouts are still displayed. If you scroll back, you will find the cursors again.

However, if you want to make cursor measurements in a different part of the signal, all you have to do is the following:

1. Disable the Cursor by clicking on the **Cursor 1** or **Cursor 2** button.
2. Enable the Cursor by clicking again on the same button.

   The cursor will now be on your screen and you will not have to drag it from its previous location.

Cursor Values

The values of the current cursors are reflected in the Display Window's status bar to the right of M.Pos.

Cursor 1's values are in the first cursor status box; Cursor 2's values are in the second cursor status box, and the third cursor status box shows the absolute difference between the cursor values, if both cursors are enabled.

The first field in each cursor status box is the X co-ordinate, measured in seconds, relative to the trigger marker. A positive value indicates the cursor is to the right of the trigger marker; a negative value means the cursor is to the left of the trigger marker.

The second field in each cursor status box is the Y co-ordinate, measured in Volts, relative to the zero line of the channel. A positive value indicates the cursor lies above the channel's zero line; a negative value means the cursor lies below the channel's zero line.

Cursor Deltas

Once two cursors have been set, GageScope® automatically calculates the absolute differences between them. This is shown in the third set of values in the status bar.

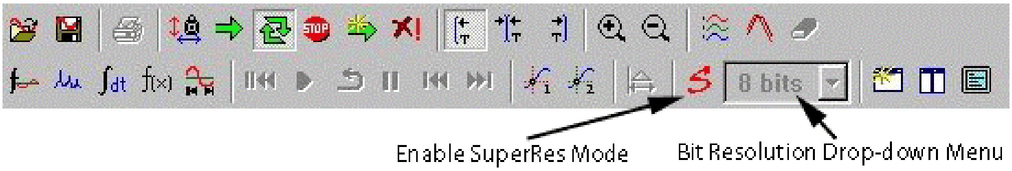
Delta Track Mode

Track mode “locks” the distance between two cursors. As such, when you move one cursor, the other moves in tandem.

1 Click on the Track Mode  button to enable track mode.

2 As a test, try to move Cursor 1 toward Cursor 2.

Toolbar: SuperRes Mode



Click on the **SuperRes** button to enable SuperRes mode. Click on the **Bit Resolution** Drop-down Menu to select the desired bit resolution.

About SuperRes Mode

SuperRes Mode allows you to increase the effective resolution of the measurement at the expense of signal bandwidth.

The mathematical operation done as a result of SuperRes selection is that an averaging window is defined within a single acquisition. All samples within the averaging window are accumulated, resulting in an effective increase in the sample bits. The averaging window is then shifted, over the current acquisition, by its width and the same averaging process is performed repeatedly.

SuperRes allows the user to specify the desired "Effective Resolution." For example, you can specify 12 bit resolution even if you are using the 8 bit CompuScope 8500. GageScope® will automatically perform digital filtering (averaging of oversampled data) to enhance the effective resolution at the expense of sample rate.

For example, in order to provide 12 bits for the CompuScope 8500 (an increase of 4 bits), GageScope® has to average $2^4 = 16$ adjacent points, hence reducing the sample rate by a factor of 16. This means that instead of 500 MS/s, the effective sample rate will be 500/16 = 31.25 MS/s.

Notes

- The effective resolution mentioned here is **not** the same as Effective Number of Bits (ENOB). ENOB is a measurement of the Signal-to-Noise Ratio.
- For more on SuperRes Mode, see page C-68

System Control

The System Control is located at the lower left corner of the screen. It contains the control settings for CompuScope and CompuGen cards.

For CompuScope cards, the system control contains the following tabs and settings:

CS Tab – Page C-62

The title of the CS tab depends on the model of your CompuScope card. For example, if you have a CompuScope 82G, this tab will display **CS82G**.

- Sample rate
- Single/dual channel mode
- SuperRes mode
- External Clock (if available)

Trigger Tab – Page C-70

- Trigger source
- Trigger slope
- Trigger level
- Trigger pattern configuration

Depth Tab – Page C-82

- Total memory
- Pre- and post-trigger depth
- Fine
- Depth units
- Timeout (Auto/Normal)
- Multiple Record (if supported by your CompuScope)

Tabs and settings for CompuGen cards are as follows:

CG 1100 Tab – Page C-95

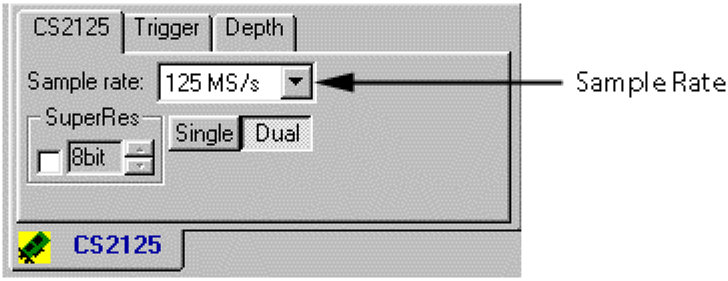
- Sample Rate
- Trigger (Internal/External)
- Mode (Triggered/Continuous/Burst)

Out 01 Tab – Page C-100

- Source Channel
- Gain
- Offset
- Filter

System Control: CS Tab: Sample Rate

The Sample Rate is located in the CS tab of the System Control.



The sample rate sets how many sample points per second should be acquired. A high sample rate will result in more sample points acquired per second, resulting in a smooth and accurate waveform. A lower sample rate will result in fewer sample points per second, resulting in a less smooth signal representation on the screen.

Changing the Sample Rate

- To change the sample rate, click on the sample rate drop-down menu and choose a sample rate from the list. This menu will differ depending on the model of CompuScope card and whether or not it is set to single or dual channel mode.

  For example, a CompuScope 82G can run at 2 GS/s in Single Channel mode, but the maximum is 1 GS/s in dual channel mode. GageScope® adjusts the list of available sample rates depending on these factors.

Notes

- Upon startup, GageScope® sets the sample rate to the highest available. When you switch to dual channel mode, GageScope® checks if the sample rate is higher than what is possible, and, if necessary, reduces the sample rate.

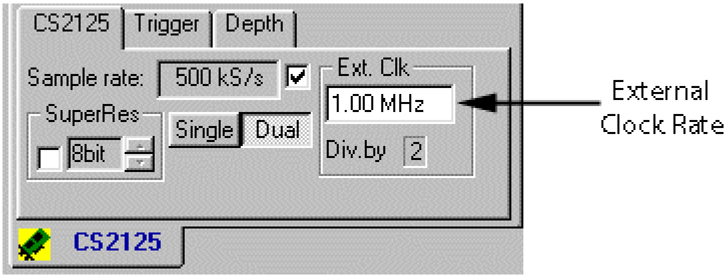
- You should normally use a sample rate that is twice (or more) the frequency of the input signal. For more information on this topic, see **Nyquist Theorem** in the *Glossary* section of this manual.

  Obviously, it is preferable to have as many samples per cycle of the input frequency as possible. The industry standard is to have 5 to 8 samples per cycle. In other words, sample rate should be 5 to 8 times the input frequency to get a good representation of the signal on-screen.

System Control: CS Tab: External Clock

External Clock appears as an additional entry in the Sample Rate Drop-down menu, located in the CS tab of the System Control.

Please note that External Clock is an optional hardware upgrade on the older CompuScope cards and must be purchased from the factory. If External Clock is not installed on your CompuScope card(s), this entry will not be available.



GageScope® automatically senses the presence of the External Clock modification on the installed PCI bus CompuScope board(s).

For enabling External Clocking on ISA bus cards, you must provide either of the following command line switches prior to running GageScope®:

/extclk

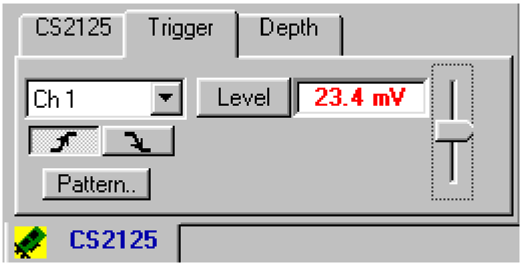
or

/extclkmin=sample_rate_number

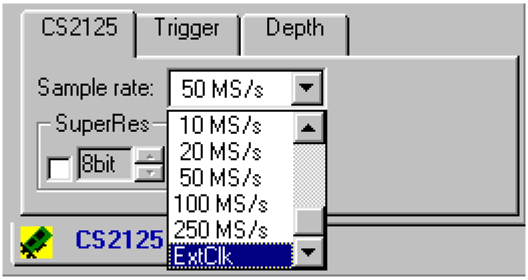
where sample_ rate_ number is the minimum expected external clock frequency, in Hertz.

How it works

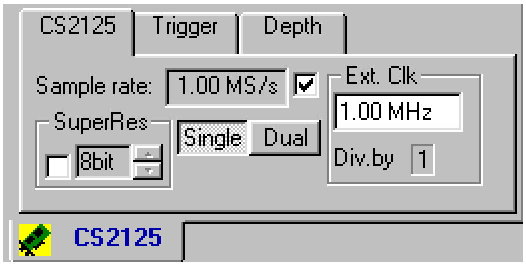
1 Click on the **CS** tab in the System Control to bring it to the front, if it's not already visible.



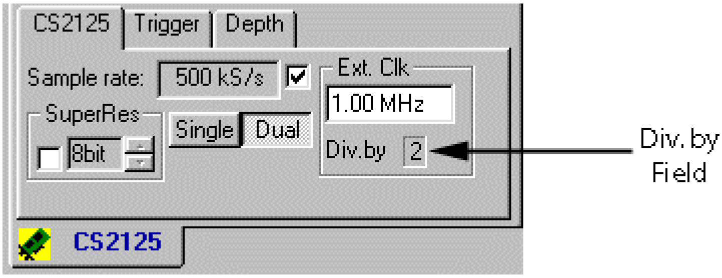
2 Click the **Sample Rate** drop-down menu and scroll down to the bottom of the list to **ExtClk**.



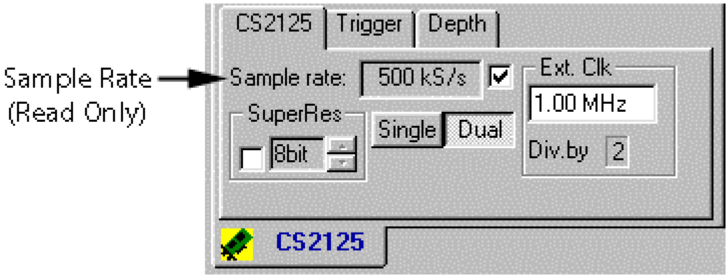
3 The new External Clock options appear on the CS tab.



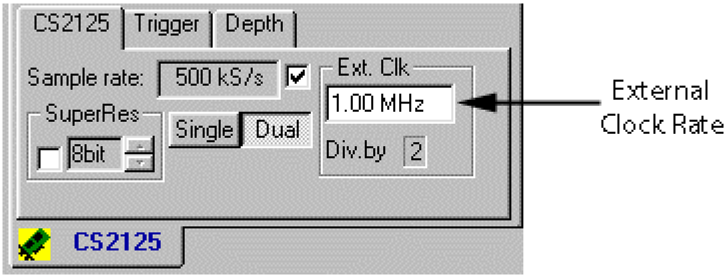
4 Note that when you are in dual channel mode, the **Div by** field reads 2. When you are in single channel mode, the **Div by** field reads 1.



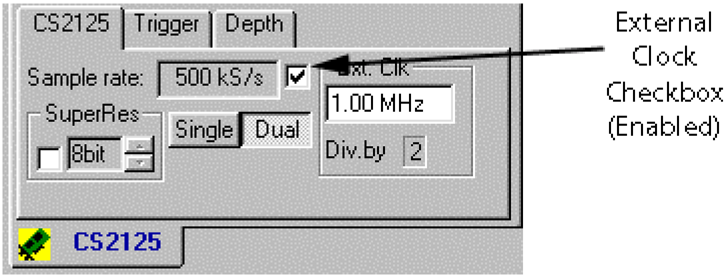
5 In addition, the Sample Rate (which is a read-only field when External Clock is enabled) in single channel mode is double that in dual channel mode.



6 You can change the external clock rate by typing in the **Rate** field.



7 To disable External Clocking, click on the checkbox to the right of the Sample Rate drop-down menu. The regular CS tab will re-appear.

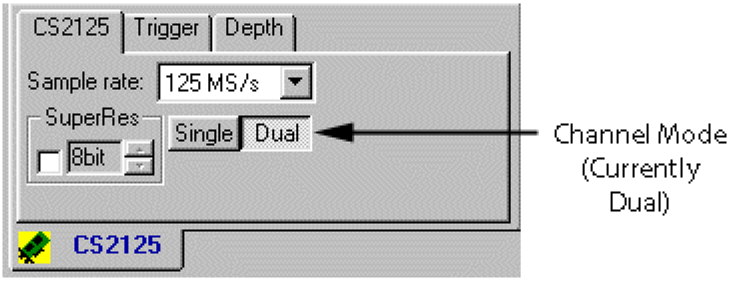


Notes

- You must have an external clock connected to your system. If you do not, while you will still be able to select External Clock from the Sample Rate drop-down menu, GageScope® will display “Waiting” in the Status Bar and a trigger event will not occur.

System Control: CS Tab: Channel Mode

The Channel Mode is located in the CS tab of the System Control.



The Single and Dual Channel Mode buttons allow you to switch between channel modes. This feature is not available if your CompuScope card is single-channel only (such as the CompuScope 8500) or dual channel only (such as the CompuScope 1602, CompuScope 512 and CompuScope 512/PCI).

Changing Channel Modes

- To change modes, click on the desired button. The current mode is indicated by the button that is pressed in.

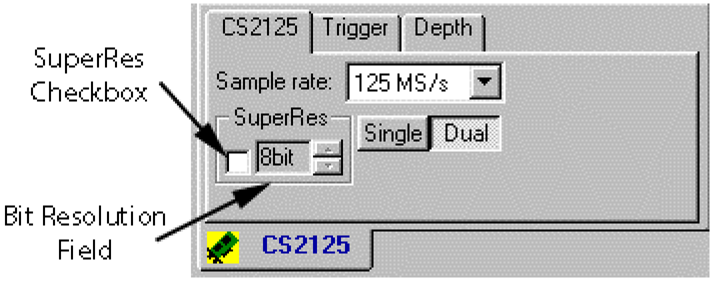
Notes

- Upon startup, GageScope® sets this option to dual channel, if two channels are available on your CompuScope card.

- When you switch to dual channel mode, GageScope® reduces the sample rate if necessary. When you switch to single channel mode, GageScope® returns to the sample rate which had been set previously in single channel mode.

System Control: CS Tab: SuperRes

SuperRes is located in the CS tab of the System Control.



Click on the **SuperRes** button to enable SuperRes mode. Click on the **Bit Resolution** increment/decrement buttons to select the desired bit resolution.

About SuperRes Mode

SuperRes Mode allows you to increase the effective resolution of the measurement at the expense of signal bandwidth.

The mathematical operation done as a result of SuperRes selection is that an averaging window is defined within a single acquisition. All samples within the averaging window are accumulated, resulting in an effective increase in the sample bits. The averaging window is then shifted, over the current acquisition, by its width and the same averaging process is performed repeatedly.

SuperRes allows the user to specify the desired "Effective Resolution." For example, you can specify 12 bit resolution even if you are using the 8 bit CompuScope 8500. GageScope® will automatically perform digital filtering (averaging of oversampled data) to enhance the effective resolution at the expense of sample rate.

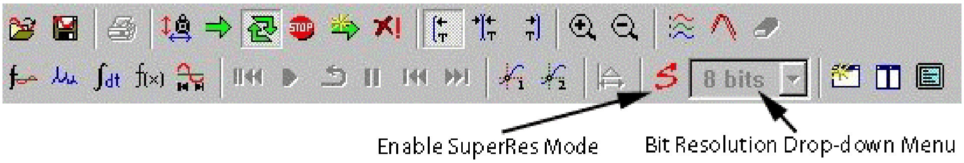
For example, in order to provide 12 bits for the CompuScope 8500 (an increase of 4 bits), GageScope® has to average 24 = 16 adjacent points, hence reducing the sample rate by a factor of 16. This means that instead of 500 MS/s, the effective sample rate will be 500/16 = 31.25 MS/s.

Notes

- The effective resolution mentioned here is **not** the same as Effective Number of Bits (ENOB). ENOB is a measurement of the Signal-to-Noise Ratio.

How it works

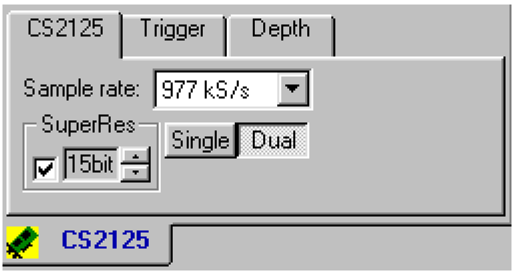
1 Click the SuperRes checkbox to enable SuperRes mode. Or, you can click on the **SuperRes** button in the toolbar (it should be “pressed in”).

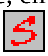


You will also notice that **SuperRes** appears in the upper left of the Display Window, next to the sample rate.

2 To change the bit resolution, click on the SuperRes drop-down menu in the toolbar, or click on the Increment/Decrement buttons on the CS tab of the System control.

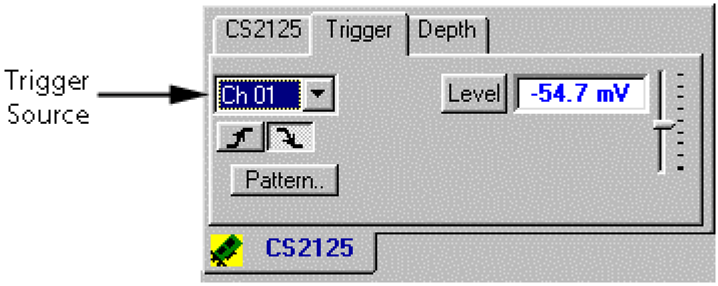
Note that when you increase the bit resolution, the effective sample rate decreases. For instance, we have raised the bit resolution on our CS2125 to 15 bits and our effective sample rate has decreased to 977 kS/s from 125 MS/s.



3 To disable SuperRes mode, click on the SuperRes checkbox in the CS Input tab (so that it is unchecked), or click on  in the toolbar (so that it is not pressed in).

System Control: Trigger Tab: Trigger Source

The Trigger Source setting is located in the Trigger tab of the System Control.



The trigger source setting tells GageScope® where to look for a trigger event.

Click on the trigger source drop-down menu to choose from four settings:

- Channel 1
- Channel 2 (if your card has two channels)
- External (through the External Trigger connector on the CompuScope card, if supported by your card)
- Disabled

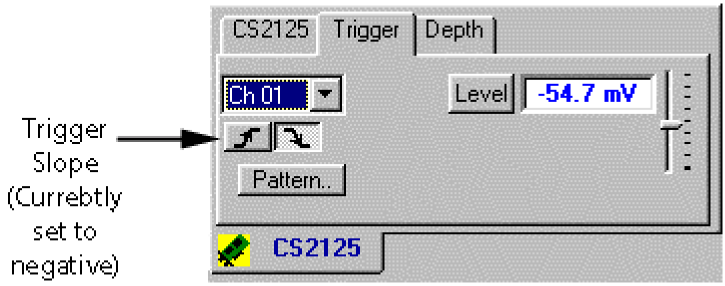
Notes

- If you have a Multi-Card system installed and the **Independent Trigger** item is checked in the **Preferences** dialog (accessed via the Tools menu), you can access additional trigger sources via the **Pattern** Dialog. See page C-74.

- You can also choose the trigger source via the trigger context menu, by right-clicking on the trigger slope icon in the Display Window.

System Control: Trigger Tab: Trigger Slope

The Trigger Slope setting is located in the Trigger tab of the System Control.



The slope icon to the left of a channel in the Display Window indicates the current slope setting.

 means positive slope, i.e., the trigger occurs on the signal’s rising edge.

 means negative slope, i.e. the trigger occurs on the signal’s falling edge.

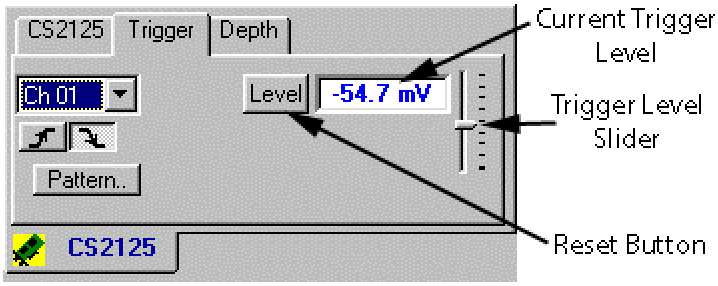
Changing the Slope

There are three ways to change the slope:

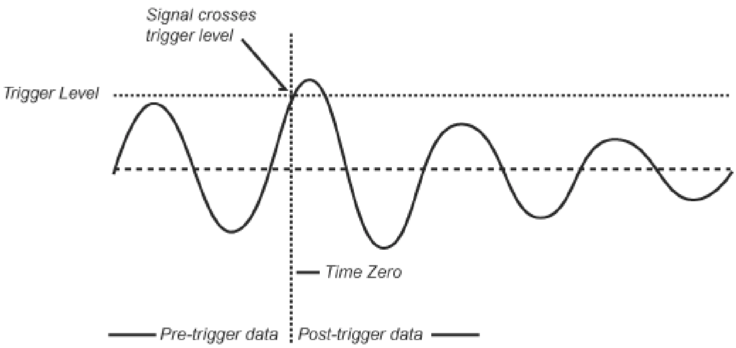
- In the **Trigger** tab, click on the desired slope icon.
- Double-click on the slope icon in the Display Window.
- Right-click on the slope symbol in the Display Window to bring up the slope context menu, then click on either **Positive** or **Negative**.

System Control: Trigger Tab: Trigger Level

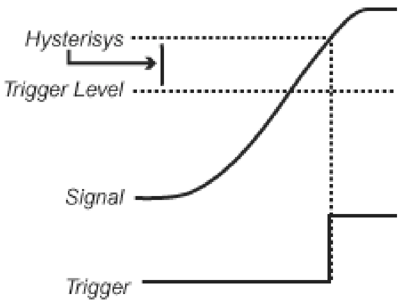
The Trigger Level setting is located in the Trigger tab of the System Control.



The trigger level is the level in Volts a signal must cross in order for a trigger event to occur. For example, if the trigger is set to +1V, and the signal peaks at +900 mV, a trigger event does not occur. But if the signal rises to +1.1V, the trigger level will have been crossed, invoking a trigger event and subsequent data acquisition.



You may notice that the level of the signal at the trigger line is slightly higher than the level set by you. This is because of trigger hysterisys implemented on CompuScopes which protects against triggering on simple electronic noise.



In the Display Window, the trigger level is represented by a slope icon to the far left of a channel with a dashed horizontal line connected to it.

The current trigger level value is shown in the **Trigger** tab next to the **Level** button. By default the trigger level is set to the zero line of channel 1.

Raising or Lowering the Trigger Level

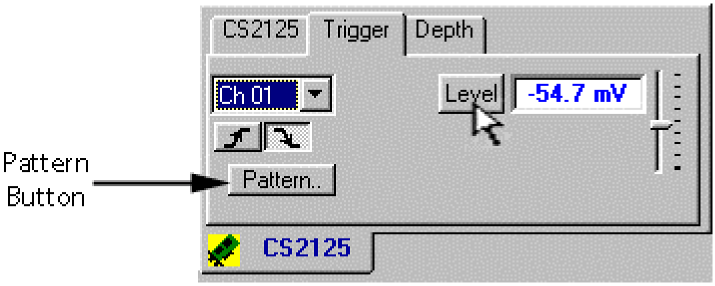
- To raise or lower the trigger level directly in the Display Window, drag the slope icon up or down.
- You can also drag the slider located in the **Trigger** tab.

Resetting the Trigger Level to Zero

- To quickly reset the trigger level to 0 mV, click on the **Level** button.

System Control: Trigger Tab: Pattern

The Pattern setting is located in the Trigger tab of the System Control.

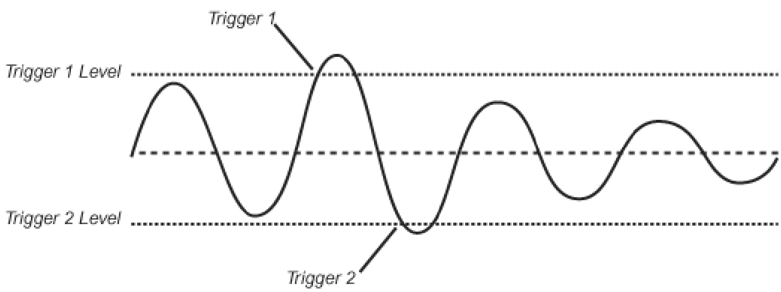


The Trigger Pattern allows configuration of multiple independent triggers. Trigger source, slope and level can be modified, and trigger sources may be added and removed. Multiple triggers are not available for all CompuScope models.

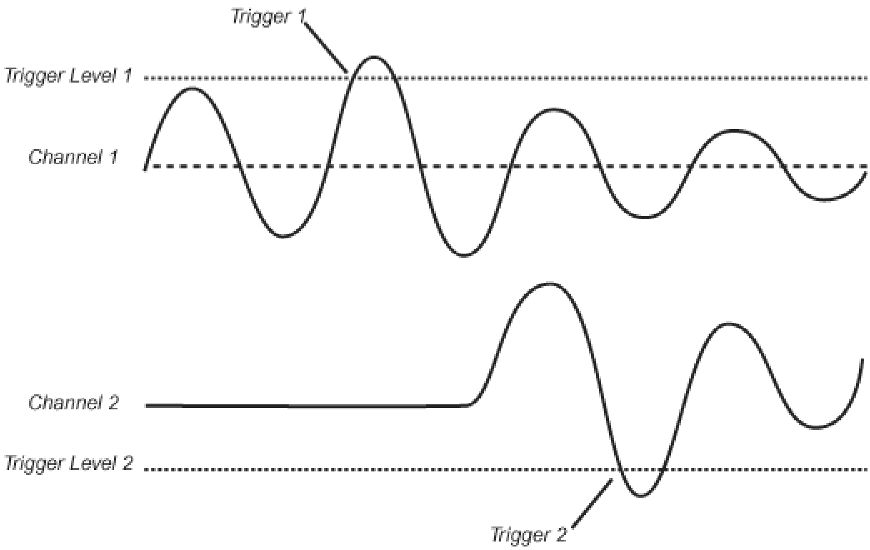
Windowed Triggering

Certain CompuScopes support “Windowed Triggering,” meaning that you can set two independent trigger conditions which are OR’ed together. Windowed triggering allows you to:

- Set two different triggers on the same channel

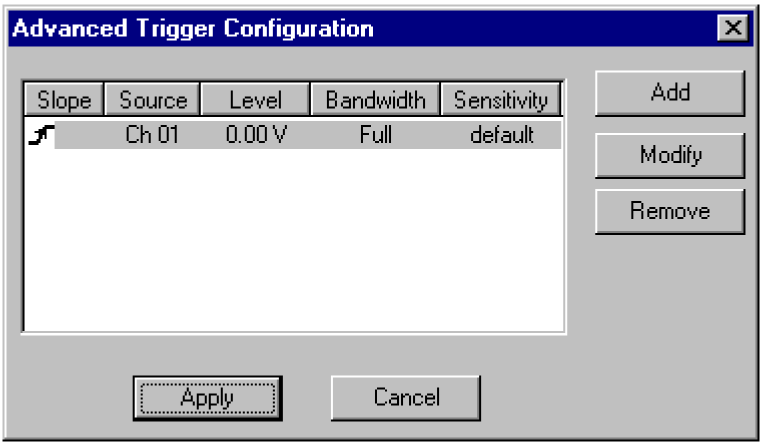


- Or set one trigger condition on each input of a CompuScope



How it works

1 Clicking on the **Pattern** button brings up the Advanced Trigger Configuration dialog.

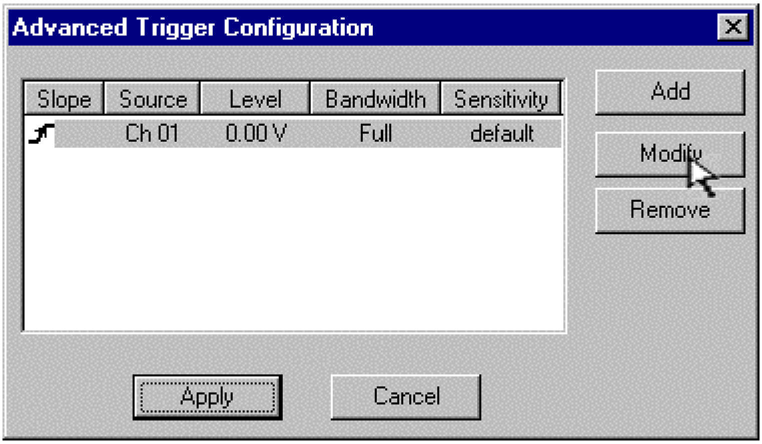


This dialog lists all of the currently specified trigger sources.

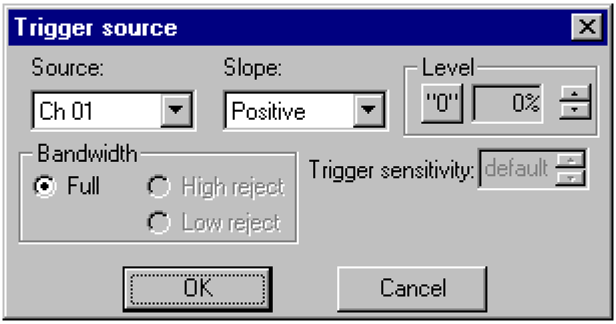
Modifying an Existing Trigger Source

2 To change existing trigger settings, first make sure the trigger source you wish to modify is highlighted. If it is not, click on it. (If there is only one trigger source in your list, there is no need to highlight it first.)

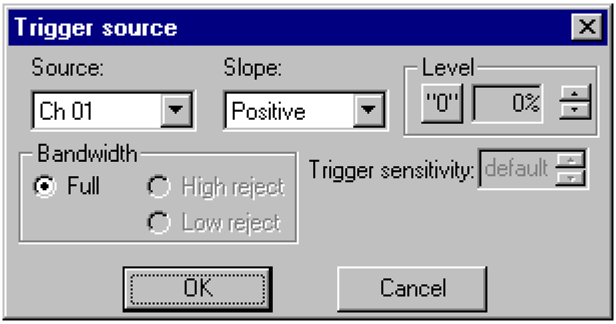
Click **Modify**.



3 This brings up the Trigger Source dialog.

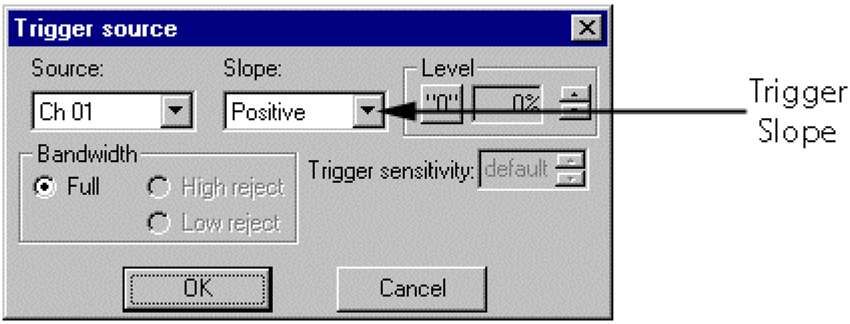


4 To change the Trigger Source, click on the **Source** drop-down list. The available trigger sources are input signals (“Ch 1,” “Ch 2,” etc.) and “External.” (Disabled is not available from this dialog; to disable a trigger, you must select **Disabled** from the **Source** drop-down menu in the main Trigger tab.)

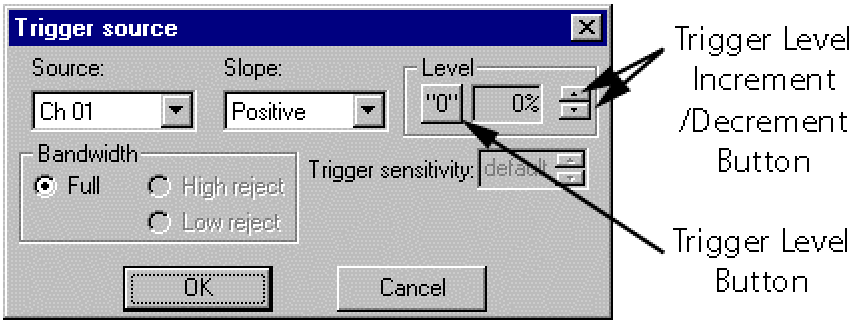


- Note that on CompuScope 1016, the trigger source can be External only.
- Note also that if you have a Multi-Card system installed, you will see one trigger source for each channel available in your system. For example, for a three-card CS8012A system, there will be six input channel trigger sources available (since each CS8012A has two input channels). If you do not see the right number of trigger sources, go to **Preferences** in the Tools menu and make sure the **Independent Trigger** item is checked. See page C-42 for details.

5 To change the Trigger Slope (either positive or negative), click on the **Slope** drop-down list.

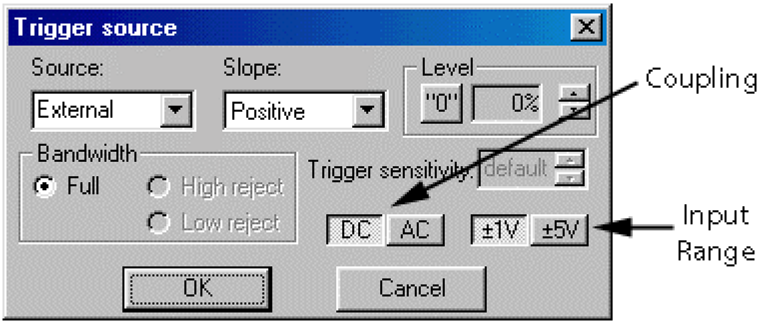


6 To change the Trigger Level, use the increment/decrement buttons to set a percentage of the current input range of the selected trigger source.



For instance, if your Trigger Source is set to Ch 1, and the current input range for Ch 1 is ±1V, if you select 50%, the trigger level will be set to 500 mV. If you select -10%, the trigger level will be set to -100 mV. The maximum positive setting is 100% and the maximum negative setting is -100%.

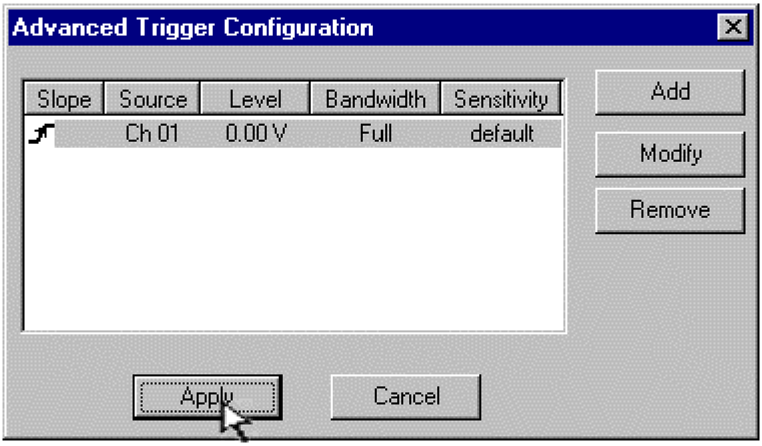
7 If your Trigger Source is set to External, the Trigger Source dialog looks slightly different.



In addition to Source, Slope and Level, this dialog includes the **Input Range** and **Coupling** settings as well.

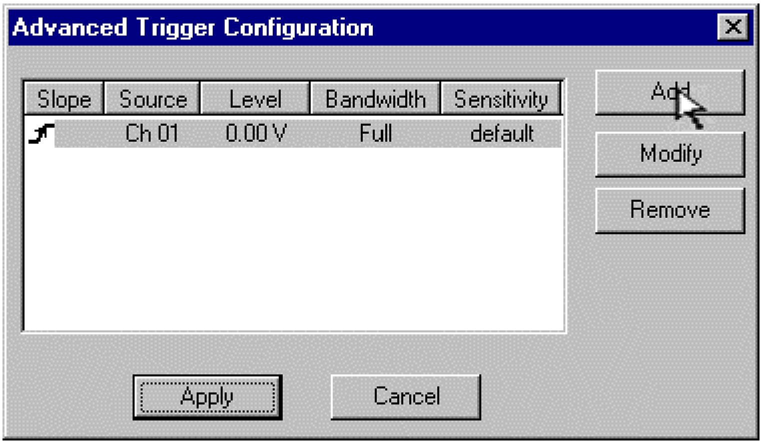
8 Click **OK** when finished.

9 Now click **Apply** in the Advanced Trigger Configuration dialog to apply the changes. Your changes will be updated in the Display Window.

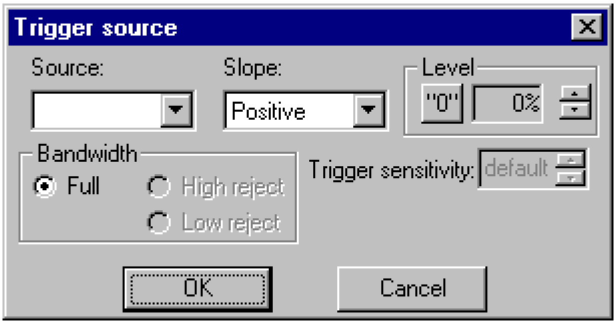


Creating a New Trigger Source

1 In the Advanced Trigger Configuration dialog, click **Add**.



2 This brings up the Trigger Source dialog with a blank Source field.



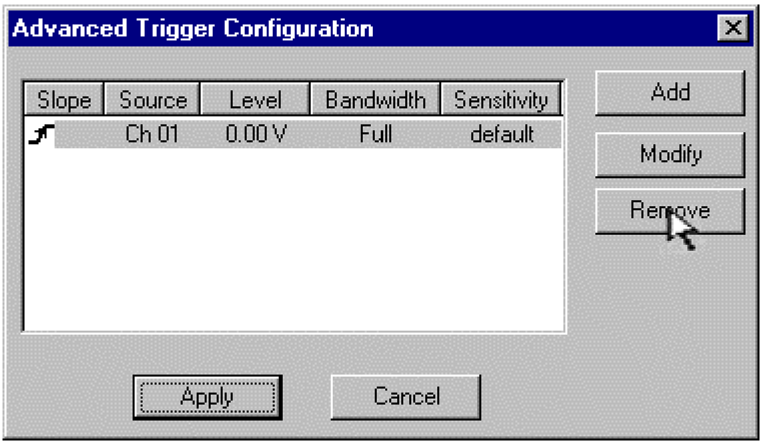
3 Now you can proceed as described in "Modifying an Existing Trigger Source" above.

Note

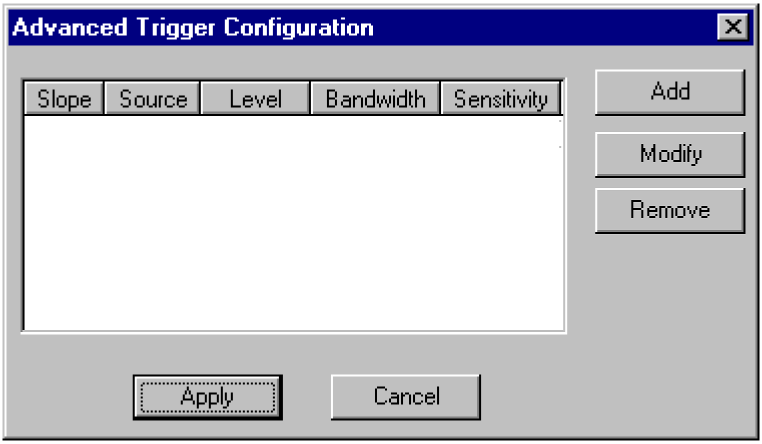
- If you have already created two independent triggers and you attempt to add a third, you will not be able to select a Trigger Source—the drop-down menu will be blank. On CompuScopes that support multiple independent triggers, you cannot set more than two trigger sources.

Removing a Trigger Source

1 In the Advanced Trigger Configuration Dialog, highlight the trigger source you wish to delete, and click **Remove**.

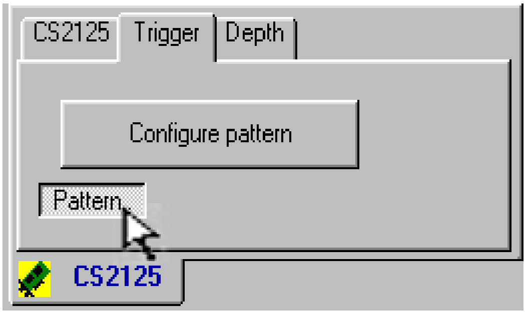


2 The trigger source will disappear from the list.



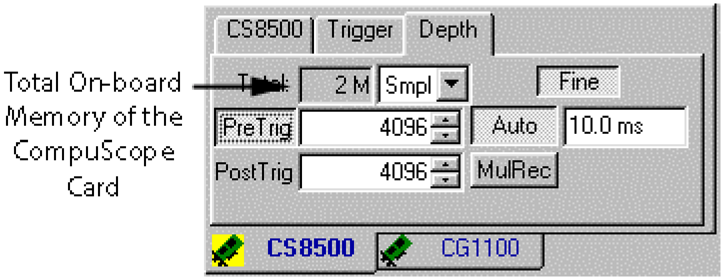
Exiting the Trigger Pattern Configuration

3 To return to the regular Trigger tab, click on the **Pattern** button so that it is no longer pressed in.



System Control: Depth Tab: Total

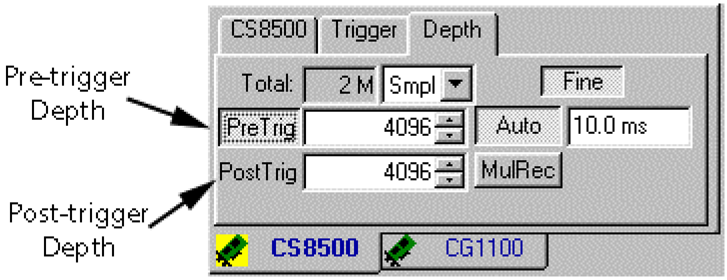
The Depth Total is located in the Depth tab of the System Control.



The **Total** field displays the total amount of on-board memory available on the CompuScope card. This value represents the maximum number of samples that can be captured at one time using this card.

System Control: Depth Tab:
Pre Trigger and Post Trigger Depth

The Pre- and Post-Trigger Depth settings are located in the Depth tab of the System Control.



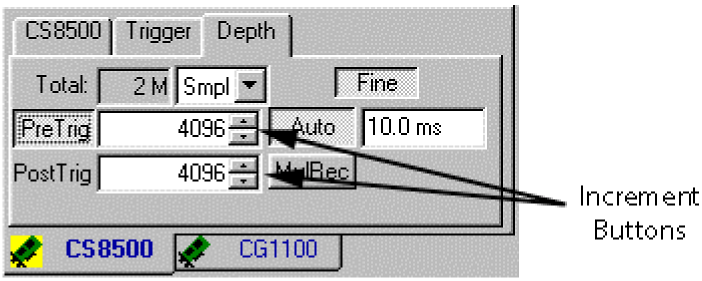
Pre-trigger data is data that leads up to a trigger event. In the Display Window, the pre-trigger signal is located to the left of the trigger marker line (the vertical “T” line).

Post-trigger data is data that follows a trigger event. In the Display Window, the post-trigger signal is located to the right of the trigger marker.

By default, pre- and post-trigger depths are set to 4096 samples.

Changing the Pre- and Post-trigger Depths

To change the amount of pre- or post-trigger data, click on the **Increment Up** or **Increment Down** button.



By default, **Units** is set to samples and **Fine** mode is on. As such, when clicking on an increment button, the depth settings initially change by 64 samples. See **Fine Mode and Increment Buttons** on the following pages for more information on how the depth settings are affected by these buttons.

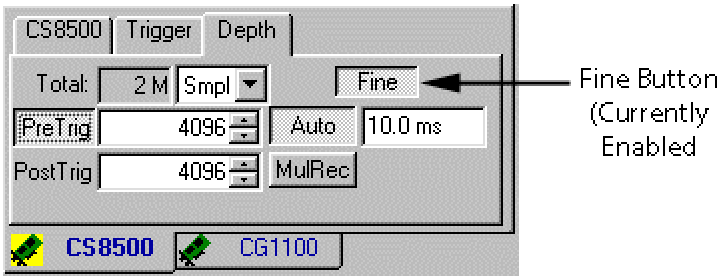
You can also type the number of pre- and post-trigger points directly in the dialog box. GageScope® will verify that the entry meets the requirements of the hardware and will correct it, if necessary.

Disabling Pre-trigger Depth

If the **Pre-Trig** button is enabled, pre-trigger capture is on. To turn off pre-trigger capture, click on the **Pre-Trig** button.

System Control: Depth Tab: Fine Mode and Increment Buttons

The Fine Button is located in the Depth tab of the System Control.

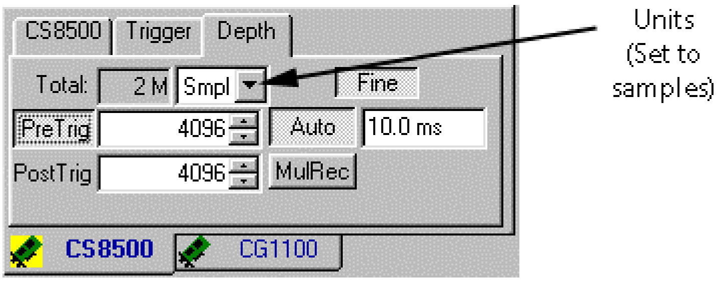


When Fine Mode is enabled, changes to depth settings occur in increments of 64 samples. When Fine Mode is off, changes occur in increments of 640 samples.

Fine Mode is enabled by default.

System Control: Depth Tab: Units

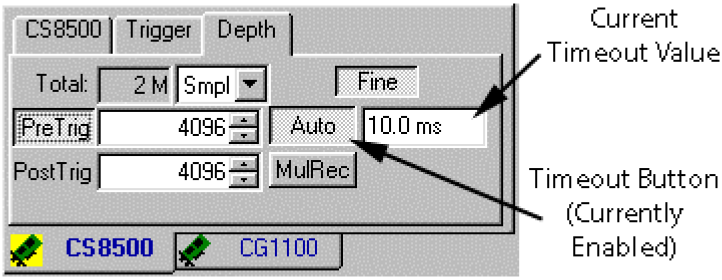
The Depth Units field is located in the Depth tab of the System Control.



The unit is always set to Samples.

System Control: Depth Tab: Auto / Normal

The Auto/Normal setting is located in the Depth tab of the System Control.



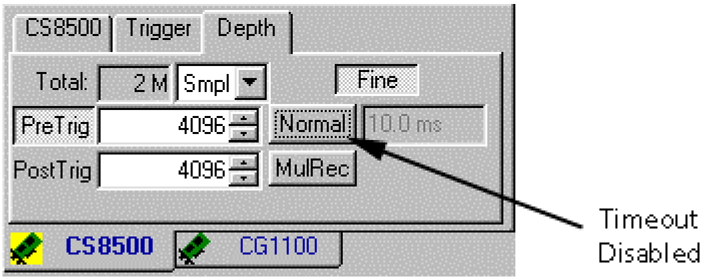
The Auto setting is the amount of time GageScope® waits for a trigger event to occur. The default setting is **10 ms**.

If the **Auto** button in the Depth tab is pressed in, it is enabled. If it is not pressed in, it reads **Normal**.

When a timeout occurs in Continuous Mode, the message **Ready** appears in the status bar on a yellow background. Timeout is automatically disabled in One Shot Mode.

Disabling the Timeout

To disable the timeout, click on the **Auto** button so that it reads **Normal**.



Setting a Timeout

1. Enable the timeout (make sure the button reads **Auto** and is pressed in).
2. Click in the Auto field.
3. Press **Backspace** or **Delete** to erase the existing value.
4. Enter a new value followed by a unit of measurement. For example, type in **1 ms** for 1 millisecond. You can also use **s** for seconds. Use only lowercase letters.

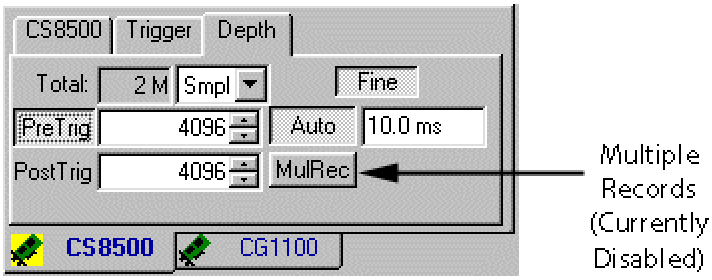
How Timeout Affects Capture Modes

In the event of a timeout while in Continuous capture mode, GageScope® forces a software trigger, i.e., it triggers immediately upon timeout, acquires the current signal, then resets the CompuScope card to await another trigger event.

When you change to One Shot capture mode, timeout is automatically disabled. When you change back to Continuous capture mode, timeout is automatically set back to the value it had before you entered One Shot capture mode.

System Control: Depth Tab: Multiple Record

The Multiple Record setting is located in the Depth tab of the System Control.



Multiple Record takes advantage of the CompuScope card's deep memory buffers by allowing the hardware to stack captures in on-board memory, so that many small acquisitions can occur in a very short amount of time, with near-zero re-arm time. This feature is invaluable in applications in which trigger events are happening rapidly or unpredictably and A/D down-time must be minimized.

When Multiple Record is enabled, the CompuScope card looks for a trigger event, acquires post-trigger data and automatically re-arms itself to look for another trigger event. Data collected from each successive acquisition is "stacked" on top of the previous acquisition until the on-board buffer fills up.

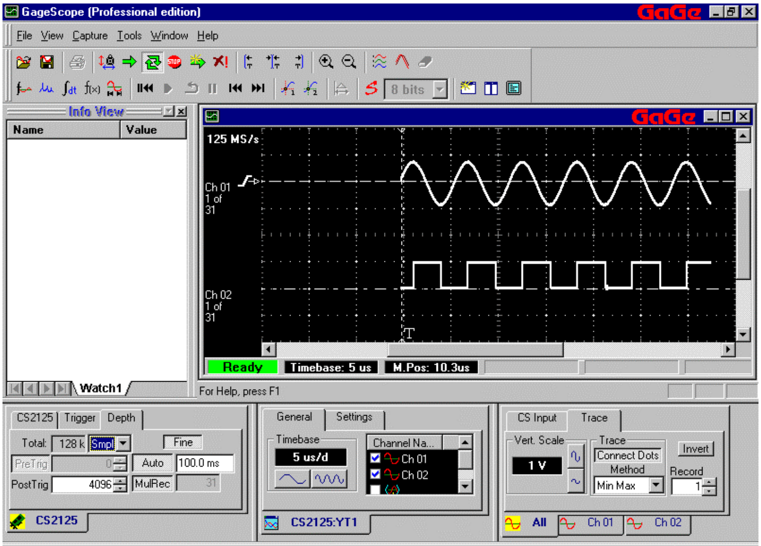
For example, if a post-trigger depth of 1024 points is specified, the first acquisition stores data in addresses between 0 and 1023, the next acquisition from 1024 to 2047, and so on until the buffer is full.

Important Notes

- The number of records you can capture will depend on the amount of on-board memory your CompuScope card has, as well as other settings, such as the post-trigger depth and channel mode.
- With all currently available CompuScopes, you can only capture post-trigger data in Multiple Record mode. When you click on the **MulRec** button, pre-trigger capture is disabled.
- Multiple Record comes as a standard feature or as an option on most CompuScope cards, but is not available on the CompuScope LITE. If your CompuScope card does not have this capability, the **MulRec** button will not be available.

Performing a Multiple Record Capture

1 Click on the **MulRec** button to enable Multiple Record. Multiple Record is enabled if its button is pressed in.



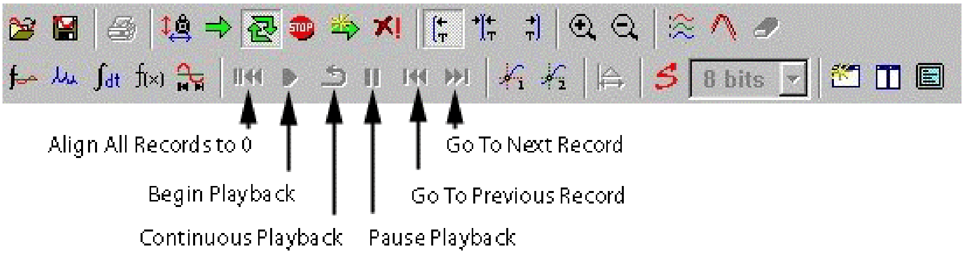
2 GageScope® automatically captures the post-trigger data for as many records as possible, depending on the current channel mode, sample rate, and post-trigger depth settings.

Note: If your CompuScope hardware supports this feature, you can also set the desired number of records to be captured.

The number of records will be reflected in the Display Window underneath the channel number at the left. For example, if 31 records are captured as in our example, you will see "1 of 31" displayed underneath "Ch 1."

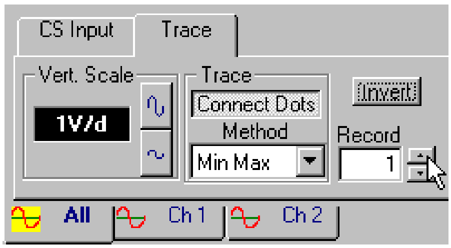
Viewing the Multiple Record Data

1 To move from record to record, click the Multiple Record Playback buttons located in the toolbar.



Note that the toolbar buttons will always cycle through all channels at once.

2 You can also use the Record Increment/Decrement buttons on the **Trace** tab of the Channel Control.

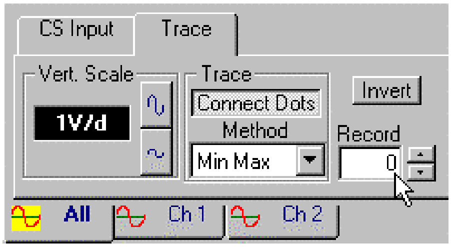


With these buttons, you can cycle through the records of all channels at once (if the All Channels tab is current), or through the records of individual channels (if an individual channel tab is current).

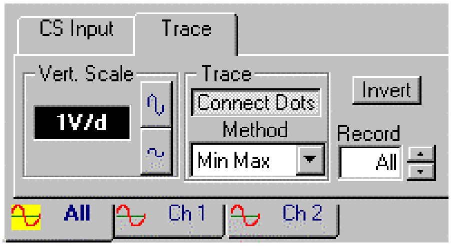
Remember that you can always set all channels to the first record with the  button in the toolbar.

Viewing All Records of the Acquisition Simultaneously

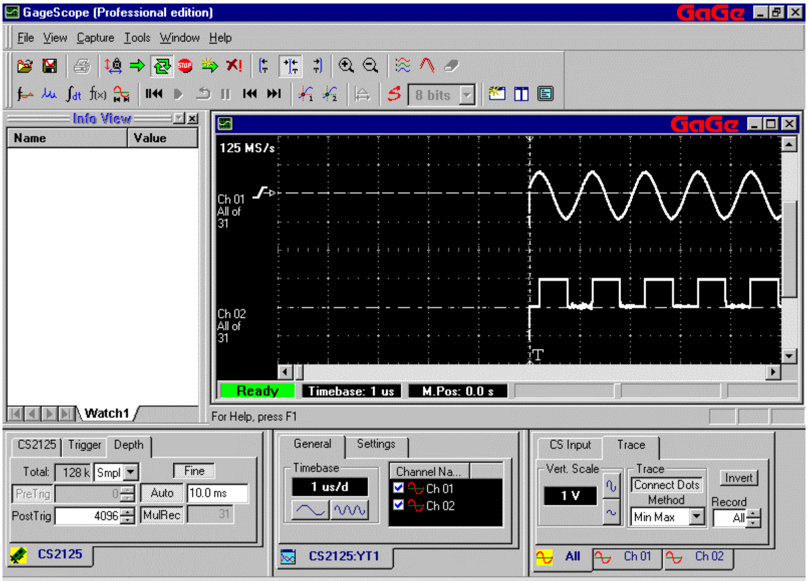
1 To view all records in the acquisition at once, click with the left mouse button in the **Record** field and type 0 (zero).



2 Press **Enter**. The 0 will change to **All**.



3 The display will now show all of your records at once. To the left of the slope icon and below “Ch1” and “Ch2” on the screen below, notice the message “All of 31,” indicating that all 31 records are being displayed.



Notes

- The **View All Records** command is available whether the **All Channels** tab or an individual channel tab is current. If the **All Channels** tab is current, as it was in our example above, you will see all records of all current channels.

- If you wish to see all records of Channel 1, for example, but only a specific record of Channel 2, make sure the proper channel tab is current when you are changing the **Record** field in the Trace tab of the Channel Control.

"Transferring" Message in the Status Bar

When you are in Continuous Capture mode (the button in the toolbar is "pressed in") and Multiple Record mode is enabled, you may see the message "Transferring" in the Status bar.

This means that GageScope® is transferring data from the CompuScope board to the internal buffers of the application.

Saving a Multiple Record Channel

1. You can save a Multiple Record channel just as you would any other channel—by clicking **Save Channel** in the File menu, by clicking in the toolbar, or by right-clicking with the mouse on the zero line of the channel display.

2. The Save Channel dialog appears.

3. Type a name for the file and select the channel to be saved from the Channel drop-down list.

4. By default, only the current record is saved. However, there are several options for saving a Multiple Record acquisition:

   - The **Split MulRec** checkbox (checked in the sample screen above) allows you to save each record of your Multiple Record acquisition as an individual file. If this box is not checked, the entire acquisition will be saved to one file.

   - To save all of the records in the acquisition to one file, make sure that all records are visible onscreen by following the steps under the heading Viewing All Records of the Acquisition Simultaneously. Then, save the file as usual.

   - You can save only the visible portion of the channel by checking the box at the bottom of the Save Dialog (just as you can with a non-Multiple Record acquisition).

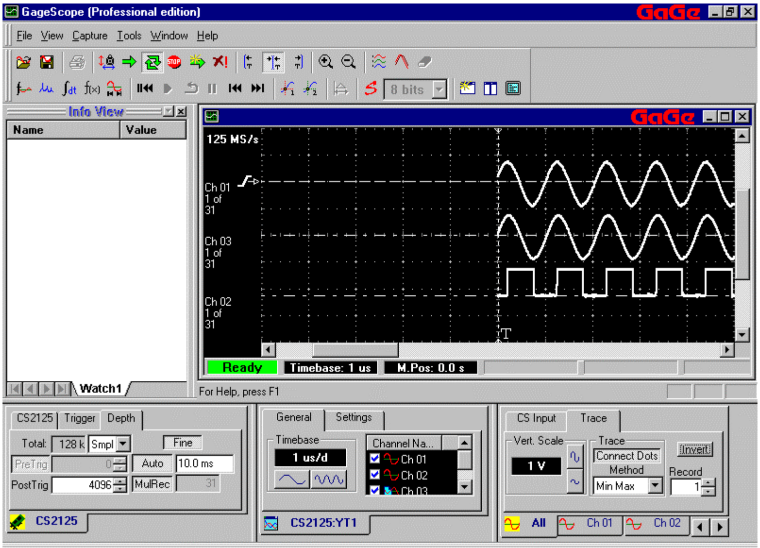
5. Click **Save** when you have specified all of the options you wish.

Loading a Multiple Record Channel

1. You can also load a Multiple Record channel as you would any other channel—by choosing Load Channel from the File menu or by clicking on the toolbar.

2. The Load Channel dialog appears.

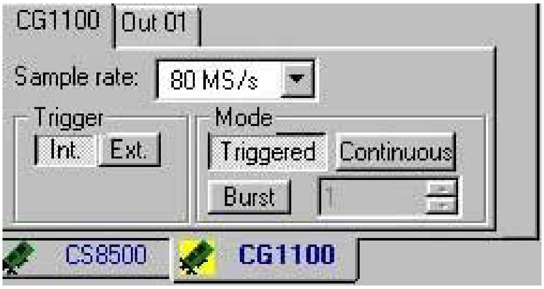
3 Click on one of the files to select it, and press **Open**.



Our signal file has been loaded as Ch 3, in between Ch 1 and Ch 2.

System Control: CG1100 Tab: Sample Rate

The CG1100 tab in System Control contains the Sample Rate setting:

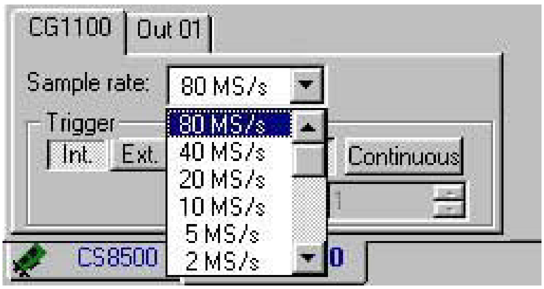


The Sample Rate specifies the number of sample points to be generated per second. The total number of sample points to be generated is specified in the **AWG Input** tab of Channel Control. Refer to the channel control section of this chapter for further details.

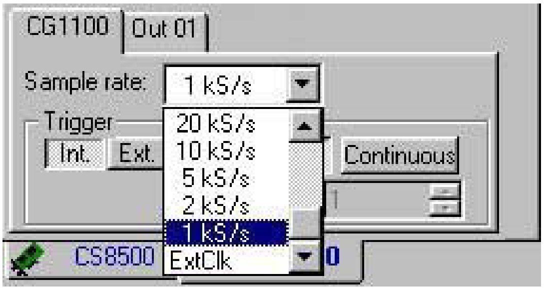
Changing the Sample Rate

- To change the sample rate, click on the sample rate drop-down menu and choose a sample rate from the list.

  For example, CompuGen 1100 can generate at the maximum rate of 80MS/s:



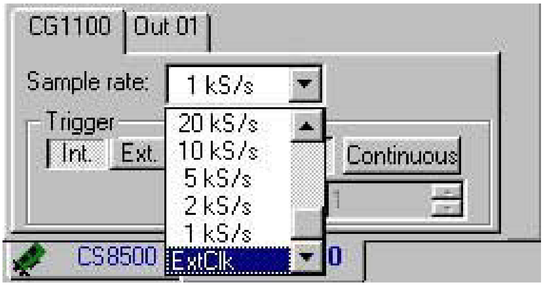
  and a minimum of 1KS/s:



Notes

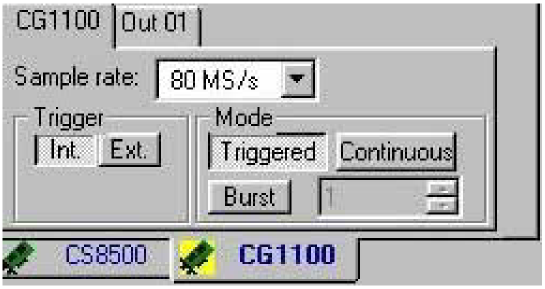
- Upon launch, GageScope® sets the sample rate to the highest available. Therefore in case a CompuGen 1100 is present in the system, you will notice that a sample rate of 80MS/s shows up as a default whenever the CG1100 tab is selected.

- To select an External Clock for your CompuGen 1100, you must choose **Ext Clock** from the sample rate drop down list.



System Control: CG1100 Tab: Trigger

The Trigger setting is located in the CG1100 tab in System Control:



The Trigger Setting indicates the type of triggers available to a generator card. As shown in the figure above, CompuGen 1100 can be triggered either internally or externally.

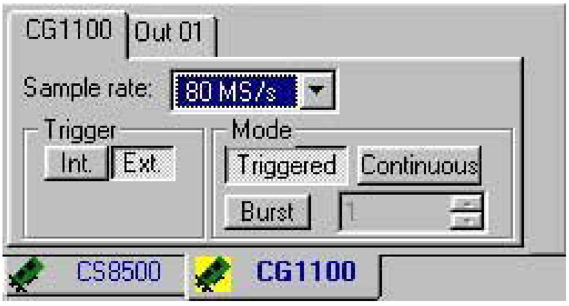
Internal Trigger

1 Internal triggering shows up as the default option when the CG1100 tab is selected. This is depicted in the figure shown above.

2 Note that when internal trigger option is selected, the GageScope® Screen indicating that the card is being internally triggered.

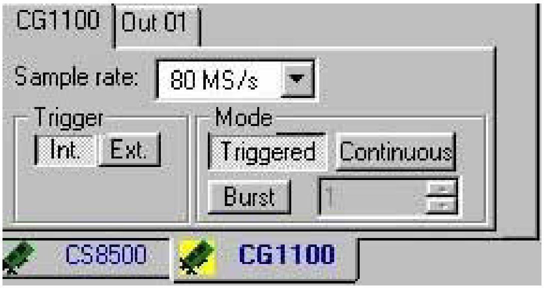
External Trigger

3 External triggering option can be selected by pressing the **Ext.** command button in the Trigger Group. This is shown in the figure below:



System Control: CG1100 Tab: Mode

The Mode setting is located in the CG1100 tab in System Control:



The Mode Setting governs the manner in which the data is going to be uploaded to the CompuGen card. Following choices are available:

Modes

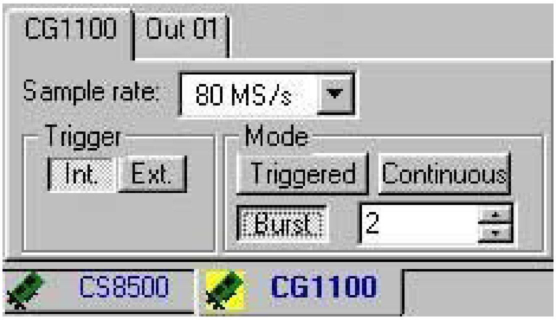
- **Triggered** In Triggered mode, the displayed waveform pattern is output from the CompuGen only once after a trigger event has been received.
- **Continuous** In Continuous mode, the displayed waveform pattern is output from the CompuGen in an endless loop.

  Note that Continuous is the default Mode setting.

- **Burst**

In Burst mode, the displayed waveform pattern is output from the CompuGen an exact number of times.

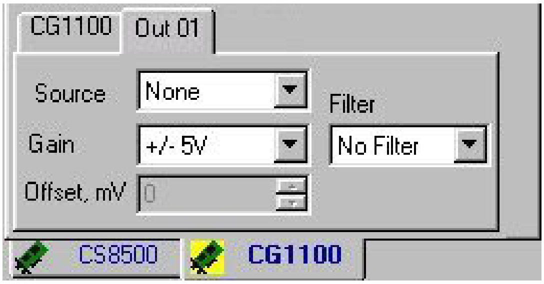
Left click the command button with Caption **Burst**. The button will look pressed in. The text box can now be used to enter the number of sequences to be generated.



Note that the default number of samples to be uploaded is **2**. You can change the value in the text box by either directly entering the desired number of sequences or by using the increment and decrement buttons at the right end of the text box.

System Control: Out 01 Tab: Source

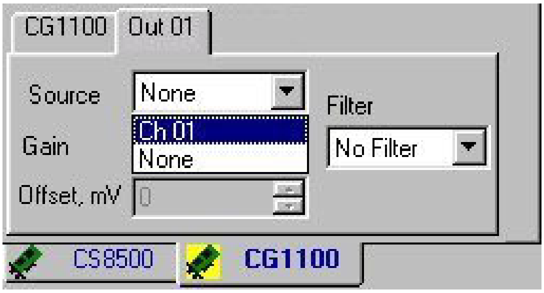
The Source setting is located in the Channel Number tab in System Control:



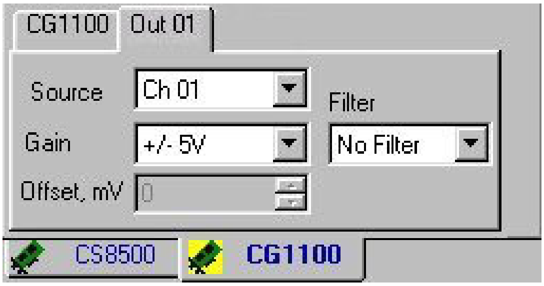
Source Settings refer to the display channel number that will be used to generate the desired waveform on the output Channel, labeled as **Out 01**. The **Source** is linked to the display channel containing the waveform created in GageScope.

Selecting a Source Channel

1 Click on the down arrow to view the drop down Source list:



2 Select **Out 01** from the list to choose channel 1 as a Source Channel for signal generation. Out 01 now appears as the source channel in the Source text box.

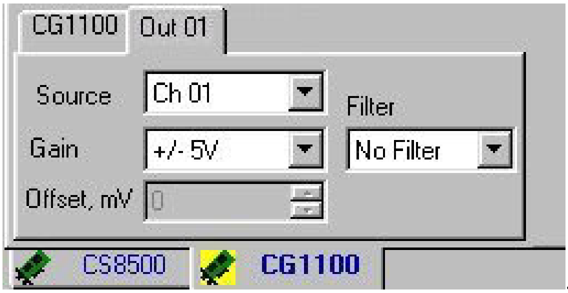


Note that a waveform has already been created in GageScope® and is represented as **Out 01**. This is the source waveform that shows up as **Out 01** in the Source drop down list. You will only see **None** in the drop down list if no AWG Channel exists. We are assuming at present that there is at least one AWG channel, **Out 01**, available to be uploaded to the generator.

Several options exist for creating waveforms in GageScope. The controls associated with the user defined, predefined or standard waveform generation, are part of the **AWG Input** tab in the Channel Control. Refer to Channel Control section of this manual to obtain further details about the **AWG Input** options.

System Control: Out 01 Tab: Gain

The Gain setting is located in the Channel Number tab in System Control:

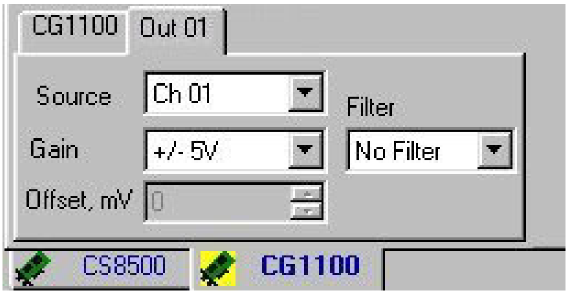


The Gain Setting imparts the user ability to change the output gain of the signal to be generated. Following Gain settings are available:

- ± 5 Volts
- ± 2 Volts
- ± 1 Volts
- ± 200 milli Volts
- ± 100 milli Volts

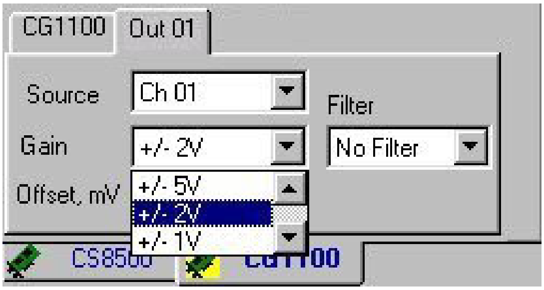
Selecting a Gain Value

1 Click on the down arrow in the text box associated with label **Gain**, to view the drop down list:



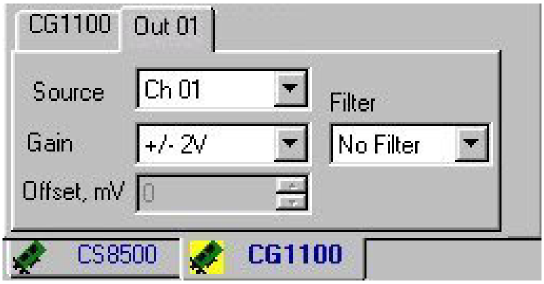
Note that +/- 5V appears as a default Gain setting.

2 To change the Gain to be +/- 2V, select the corresponding value from the list:



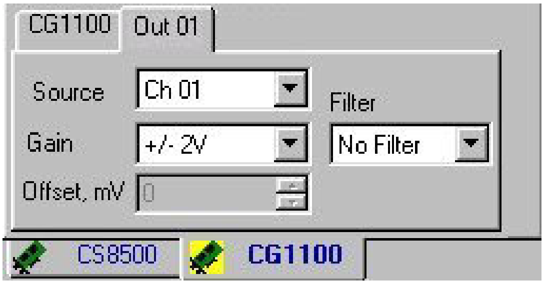
Note that Ch 01 displayed in GageScope® contains the waveform to be generated.

3 Click anywhere in the user interface or double click on your select to appear in the Gain text box.



System Control: Out 01 Tab: Offset

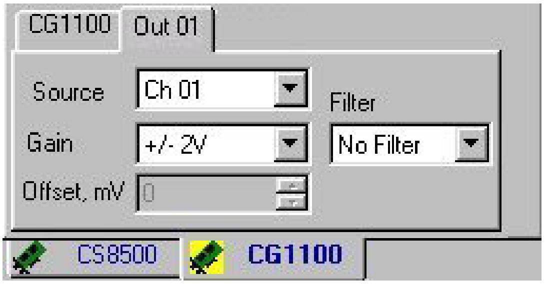
The Offset setting is located in the Channel Number tab in System Control:



The Offset Setting allows the user to enter the DC Offset in milli Volts for the output signal to be generated by the arbitrary waveform generator. Offset is linked to the Offset setting in the Channel Control property page.

System Control: Out 01 Tab: Filter

The Gain setting is located in the Channel Number tab in System Control:

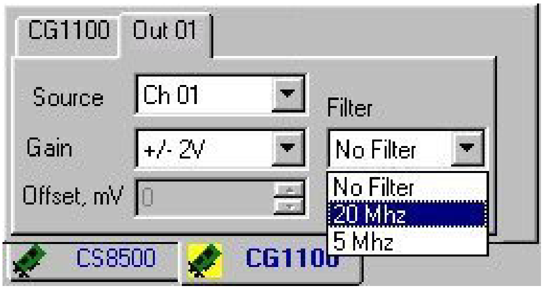


The Filter Setting allows the user to filter the output to be generated by the arbitrary waveform generator. A 20MHz filter, for example, means that all frequencies above 20 MHz will not be present in the output. Three settings are currently available for CompuGen 1100:

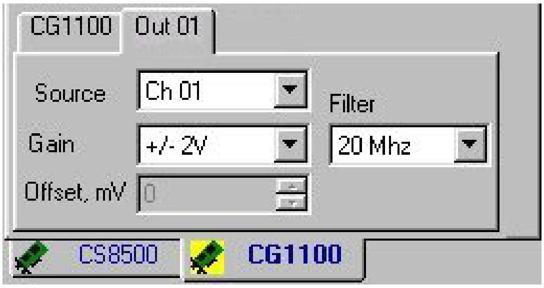
- No Filter
- 20 MHz Filter
- 5 MHz Filter

Selecting a Filter Value

1 Click on the down arrow in the text box associated with label **Filter**, to view the drop down list:



2 To change the Filter frequency to be 20MHz, select the corresponding value from the list and double click:



Display Control

The Display Control contains options for changing the look of the current display, such as the timebase and channel enable/disable, along with display element properties such as grid color. Display settings do not affect the captured data.

The Display Control contains the following tabs and settings:

General Tab – Page C-108

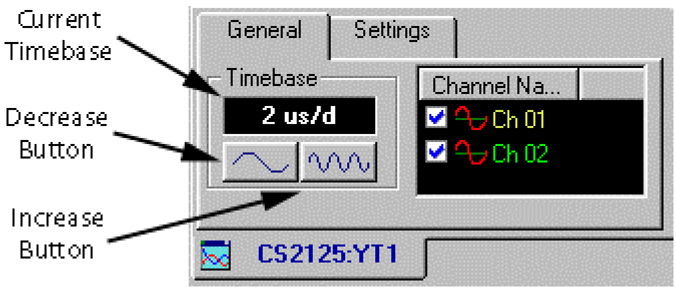
- Timebase
- Channel list and visibility

Settings Tab – Page C-110

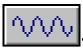
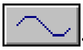
- Grid Button
- X Axis Button
- Y Axis Button
- Persistence Button
- Trigger Level Button
- Zero Lines Button
- Display Window Background Color

Display Control: General Tab: Timebase

The Timebase setting is located in the General tab of the Display Control.



The Timebase is a display function only and is measured in *time per division*, a division being one square in the grid in the Display Window. For example, a timebase of **5us/d** equals five microseconds of data per division.

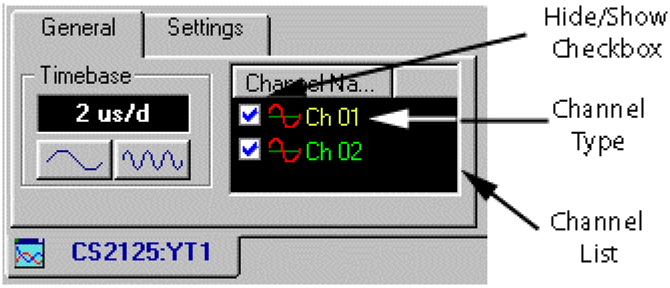
- To increase the timebase, click .
- To decrease the timebase, click .

Notes

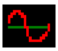
- Changes to the timebase affect all channels in the current Display Window. The size of the bar in the horizontal scroll bar is also affected.
- It is possible to set the timebase to any value; however, if there is a timebase conflict with the minimum display requirements, the timebase will be changed to the smallest timebase value that does not conflict at display time.
- If you want to see the same set of signals using different timebases, simply create a new display, select the channels you want displayed, and set the timebase to the desired value. If you tile the two Display windows, you will be able to see the same signals with different timebases.

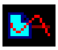
Display Control: General Tab: Channel List

The Channel List is located in the General tab of the Display Control.



The Channel List shows all channels currently available, including channels loaded from disk and channels created mathematically. The icon next to each channel indicates the channel's type.

 indicates that the signal originates from an input of a CompuScope card.

 indicates that the channel has been loaded from disk.

 indicates that the channel has been created mathematically.

If more than three channels are available, a vertical scroll bar appears, allowing you to move up and down the list.

Hiding and Showing Channels using the Channel List

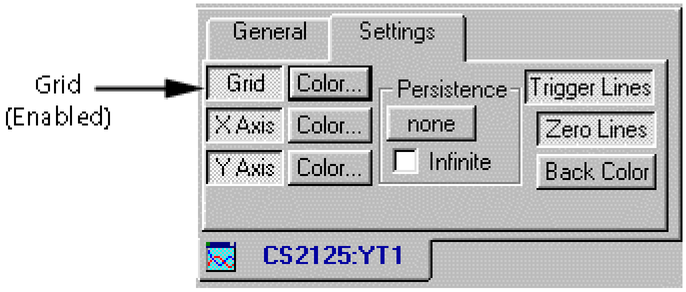
The checkbox next to each channel in the list indicates the channel's visibility. A checkmark means a channel is visible; lack of checkmark means a channel is hidden. To toggle a channel's visibility, click on this checkbox.

Hiding and Showing Channels directly in the Display Window

1. Position the mouse pointer on a blank area of the Display Windows, i.e., where there is no signal.
2. Click on the right mouse button to bring up the Display Window context menu.
   A checkmark next to a channel number indicates it is currently visible.
3. Click on **Ch 1** to hide or show channel 1, **Ch 2** for channel 2, etc.

Display Control: Settings Tab: Grid Button

The Grid button is located in the Settings tab of the Display Control.



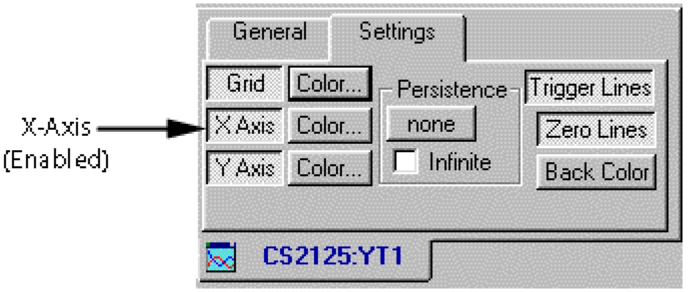
The grid is a set of vertical and horizontal lines drawn at regular intervals in the Display Window. The resulting grid has 8 squares from top to bottom and 10 squares from left to right. Each square is counted as a “division” and is used as a reference in other parts of GageScope® (such as the Position setting in the Parameter tab of the Channel Control). The grid is a permanent fixture in the Display Window and is not affected by movement of the scroll bar.

To toggle the grid on and off, click on the **Grid** button. You also have the option of changing the color of the grid by clicking on the **Color** button next to the **Grid** button.

The Grid may not be that noticeable as the default Grid color is a subtle gray. To make the grid stand out more, change its color to white by clicking on the grid’s **Color** button.

Display Control: Settings Tab: X Axis Button

The X Axis button is located in the Settings tab of the Display Control.

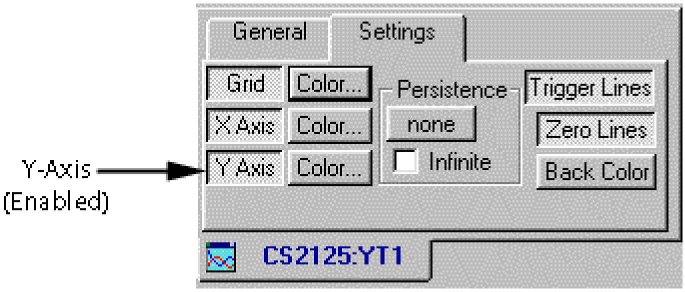


The X axis is a dashed horizontal line that divides the Display Window into two equal parts. The X Axis is a permanent fixture in the Display Window and does not move with changes to the vertical scroll bar.

To toggle the X axis on and off, click on the **X Axis** button. You also have the option of changing the color of the X axis by clicking on the **Color** button next to the **X Axis** button.

Display Control: Settings Tab: Y Axis Button

The Y Axis button is located in the Settings tab of the Display Control.

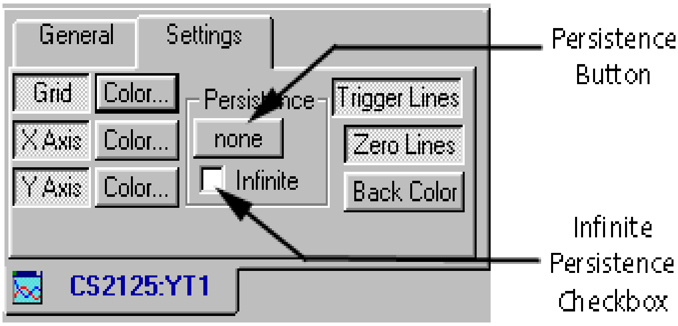


The Y Axis is a dashed vertical line that divides the Display Window into two equal parts. The Y Axis is a permanent fixture in the Display Window and does not move with changes to the horizontal scroll bar.

To toggle the Y axis on and off, click on the **Y Axis** button. You also have the option of changing the color of the Y axis by clicking on the **Color** button next to the **Y Axis** button.

Display Control: Settings Tab: Persistence

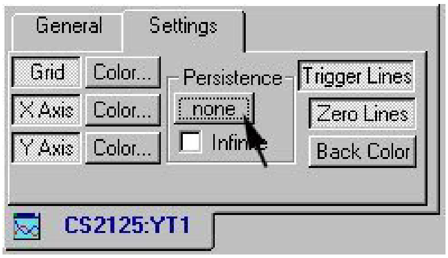
The Persistence button is located in the Settings tab of the Display Control.



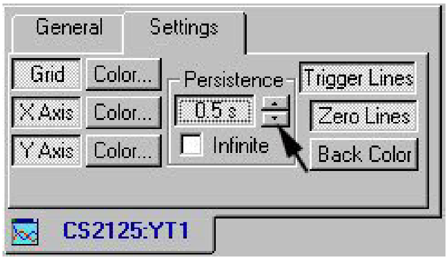
The Persistence setting allows the user to control the persistence of the signal display in the Display Window.

How it works

1 By default, Persistence is disabled. The button underneath Persistence reads **None**. To enable persistence, click on **None**.



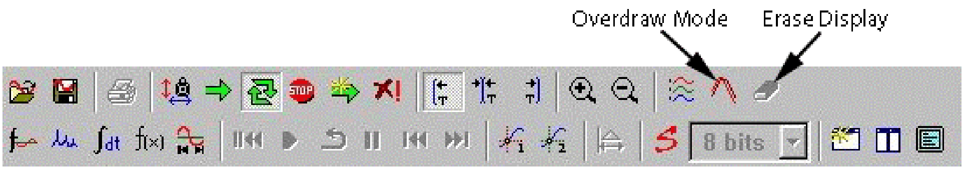
2 The Persistence field appears, allowing editing of the length of time that the signal will persist on the screen. The default value is **0.5s**.



To change the degree of persistence, click on the Persistence increment/decrement buttons to change the time value.

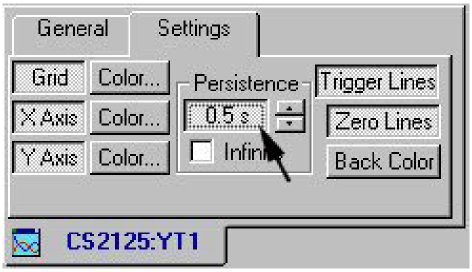
3 Or, click on the **Infinite** checkbox for infinite persistence. This is equivalent to clicking the **Overdraw Mode** button in the toolbar.

4 An **Erase Display** button becomes available in the toolbar when Persistence mode is turned on.



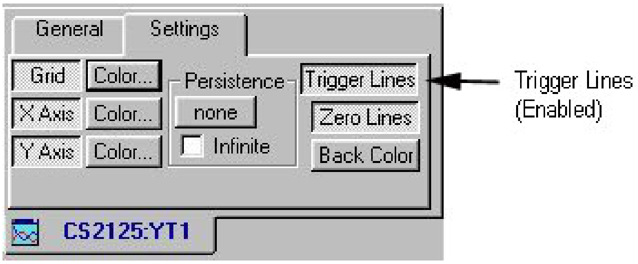
Use this button to clear the display.

5 To disable Persistence Mode, click on the **Persistence** field so that it reads “None.”



Display Control: Settings Tab: Trigger Lines Button

The Trigger Lines button is located in the Settings tab of the Display Control.

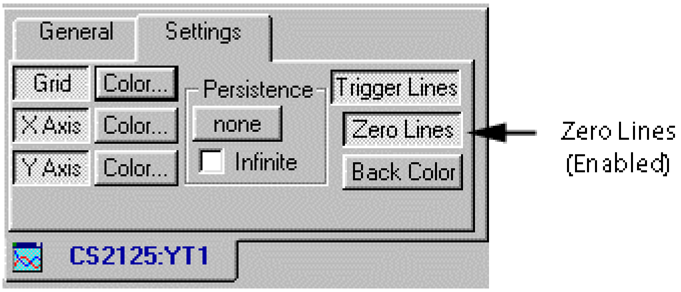


The trigger line is shown as a dashed horizontal line with a slope icon to the far left of a channel. The trigger line indicates the level the trigger signal must cross in order for a trigger event to occur. By default the trigger level is set to the zero line of channel 1.

To toggle this line on and off, click on the **Trigger Lines** button. The color of the Trigger Line is always set to the same color as the channel it is set to trigger on. For more information on the Trigger Lines, see page C-72.

Display Control: Settings Tab: Zero Lines Button

The Zero Lines button is located in the Settings tab of the Display Control.

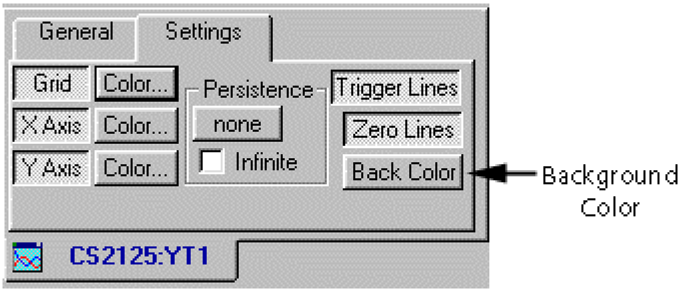


The zero line of a channel is the dashed horizontal line that runs near the middle of the signal, and indicates the 0 Volts reference. To toggle the zero lines on and off, click on the **Zero Lines** button.

The color of a Zero Line is always set to the same color as its channel.

Display Control: Settings Tab: Background Color Button

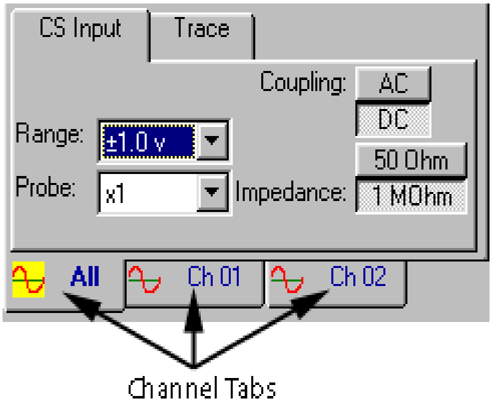
The Background Color button is located in the Settings tab of the Display Control.



Click on the **Back Color** button to change the background color of the Display Window. By default the background color is set to black.

Channel Control

The Channel Control contains options for changing the settings of individual channels, such as position, vertical scale, color, coupling, and polarization. When changing channel settings, make sure the changes are being made to the intended channels by clicking on the appropriate channel tab at the bottom of the control.



The **All Channels** tab allows you to control certain settings for all current channels simultaneously. These settings are:

- Vertical Scale
- Polarization
- Current Record Number in Multiple Record Mode and AutoSave
- Connect Dots
- Input Range
- Probe
- Coupling
- Impedance
- Null Channel Input

The channel control contains the following tabs and settings:

Subchannel Tab – Page C-122

- Starting and Ending of the Subchannel
- References
- Start and End Offset

Averaging Tab – Page C-128

- Acquisitions
- Average on Multiple Record
- Averaging vs. Co-adding
- Intermediate results
- Running Average

CS Input Tab – Page C-135

- Input Range
- Probe
- Coupling
- Impedance
- Null Channel Input
- Decimation

Trace Tab – Page C-142

The options available on this tab vary depending on whether the **All Channels** tab or an individual channel tab is current. The list below contains all of the possible options.

- Vertical Scale
- Position
- Connect Dots
- Method
- Polarization

Drawing Tab – Page C-151

This tab is not available when the **All Channels** tab is current. Drawing tab elements apply to individual channels only.

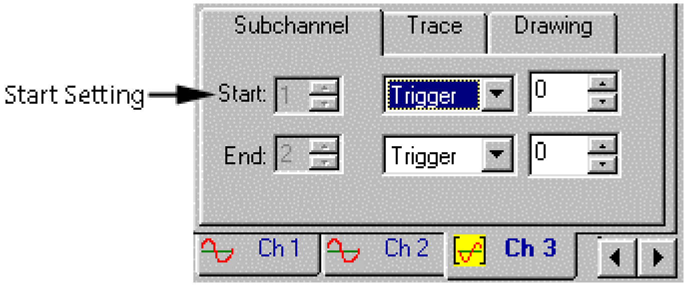
- Method
- Connect dots
- Format
- Color
- Align by Start (available only when a channel is loaded which does not cross the Trigger Marker)
- Align by Trigger (available only when a channel is loaded which does not cross the Trigger Marker)

AWG Input Tab – Page C-160

- Duration
- Equation
- Ampl
- Offset
- Waveform
- Duty

Channel Control: Subchannel Tab: Start

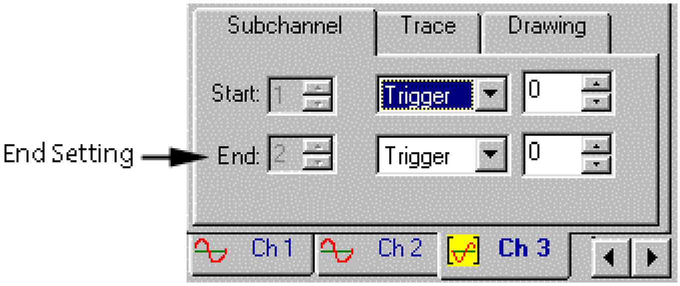
The **Start** setting is located in the Subchannel tab of the Channel Control.



Start This setting allows the user to select an appropriate reference for the starting point of the Subchannel.

Channel Control: Subchannel Tab: End

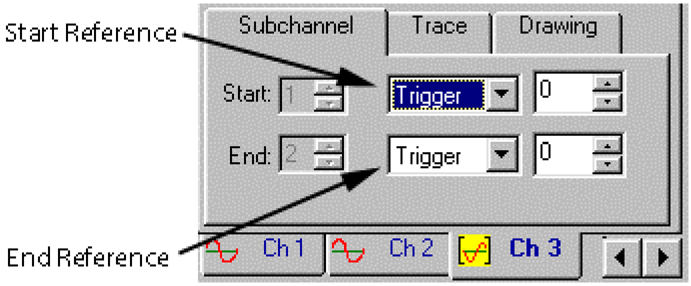
The **End** setting is located in the Subchannel tab of the Channel Control.



End Allows you to select the ending point of the subchannel with respect to a selected reference.

Channel Control: Subchannel Tab: Reference

The reference is located in the Subchannel tab of the Channel Control.



The references are available both for the **Start** and **End**. This allows the user to select any portion of the acquired channel with relative ease and flexibility. Following options are available from the drop down menu:

Src Start is the first sample point of the source channel which can be used as a reference for the subchannel signal starting and ending samples. Source signal refers to the original channel to be subchanneled.

Trigger refers to time when the trigger event occured. The selected subchannel can be referenced to the trigger.

Src. End is the last point of the source signal. The first and the last sample points of the subchannel can be referenced to the end of the source signal.

Rise specifies the selected rising edge to be the starting reference

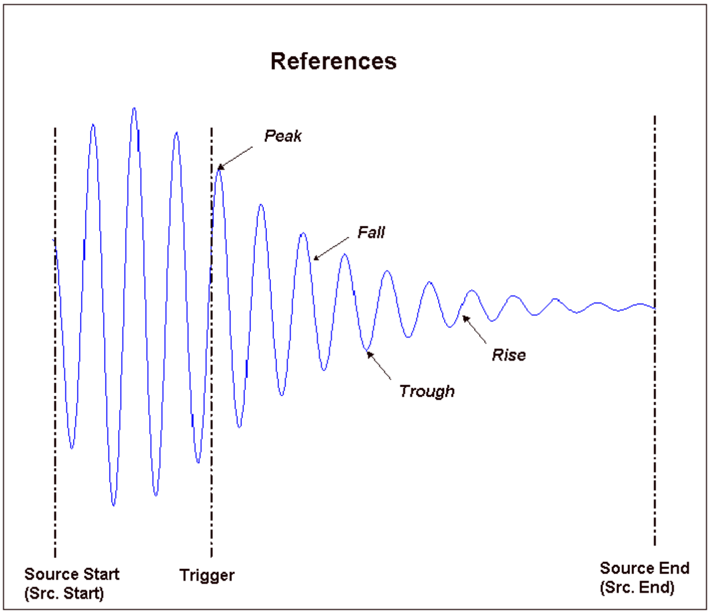
Fall specifies the selected falling edge to be the starting reference

Edge specifies the selected rising or falling edge to be the starting reference

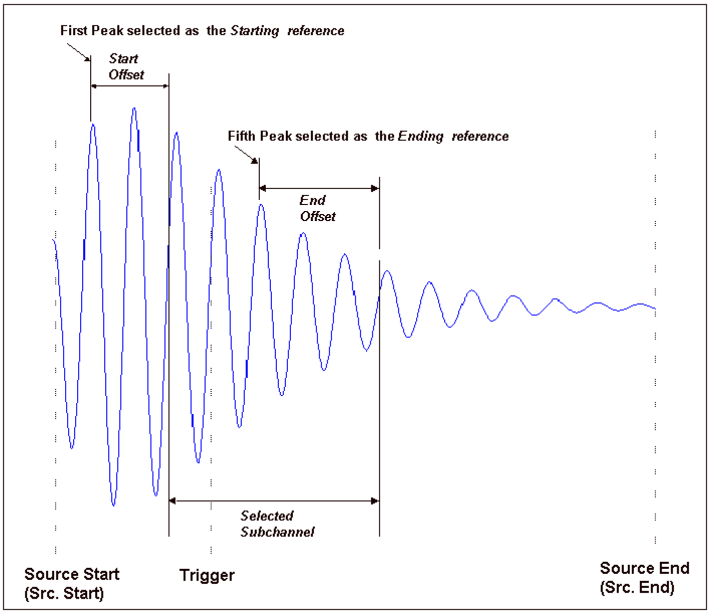
Peak specifies the selected peak to be the starting reference

Trough specifies the selected trough to be the starting reference. Trough is the valley in a sine wave.

Figure below shows all the available references

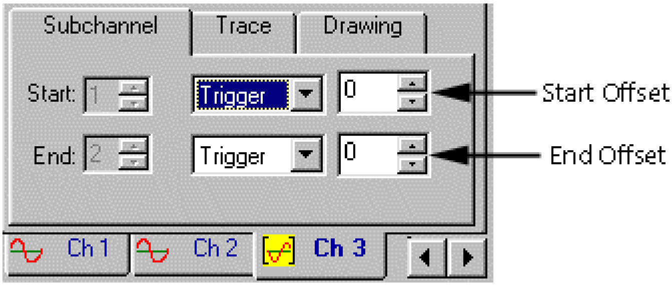


Shown below is the first peak taken as the starting reference while the fifth peak is chosen to be the end reference. Note that any combination of these references can be used to select the Subchannel.



Channel Control: Subchannel Tab: Offset

The offset setting is located in the Subchannel tab of the Channel Control.

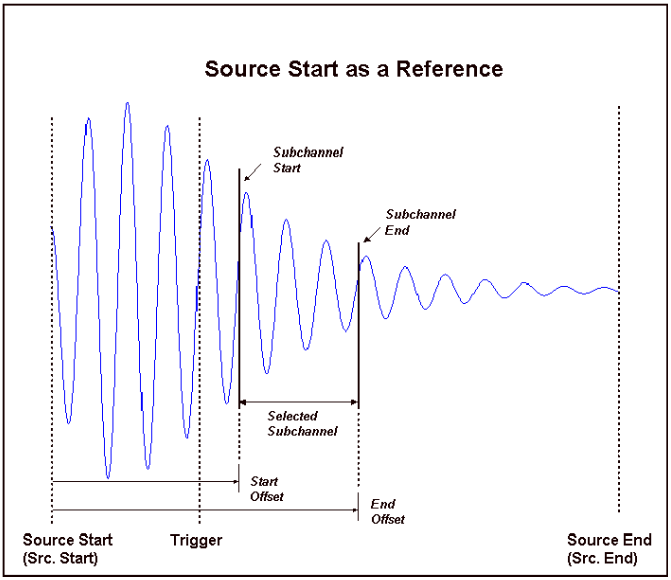


Offset The number entered in this box specifies the first sample of the subchannel displaced with respect to the selected reference. This is the case when the offset appears in the **Start** dialog area.

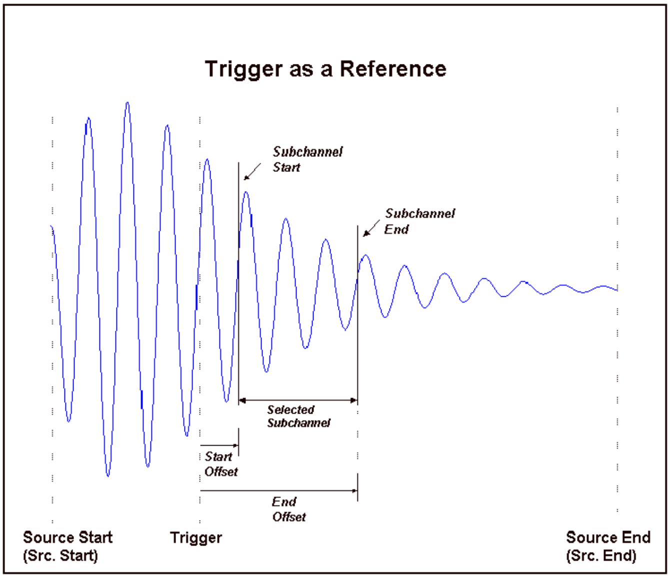
When a number is entered in this field in the **End** dialog area, the last sample of the subchannel will be selected with respect to the reference.

The figure below shows the Start and End Offsets for the available references:

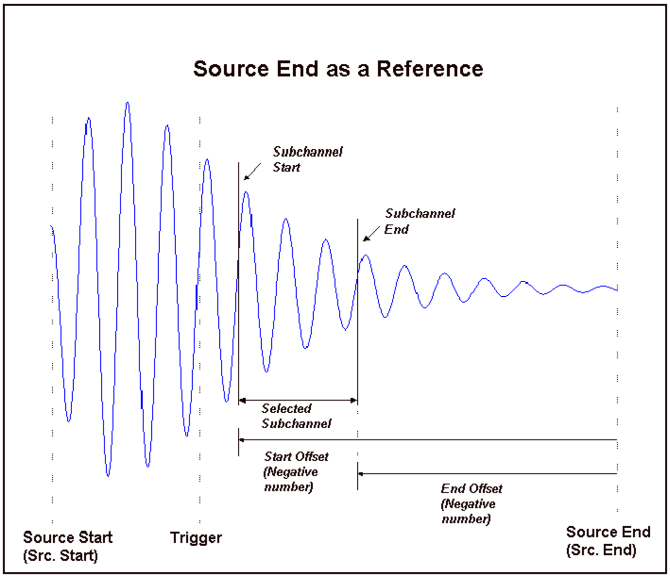
- The Source Start Reference



- Trigger Reference



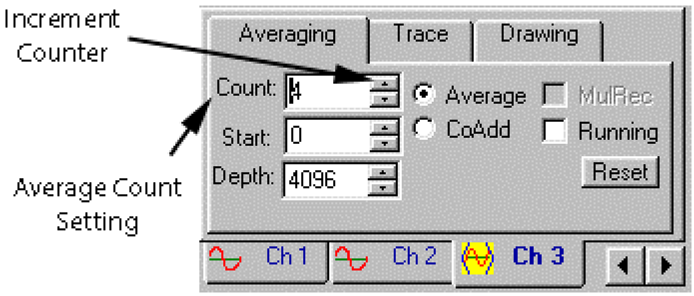
- Source End Reference



See the Tutorial Section of this manual for further details on Subchanneling.

Channel Control: Averaging Tab: Count

The Count setting is located in the Averaging tab of the Channel Control.



The number in the Count field specifies the number of averages to be performed by the GageScope®. The default number of averages is 1.

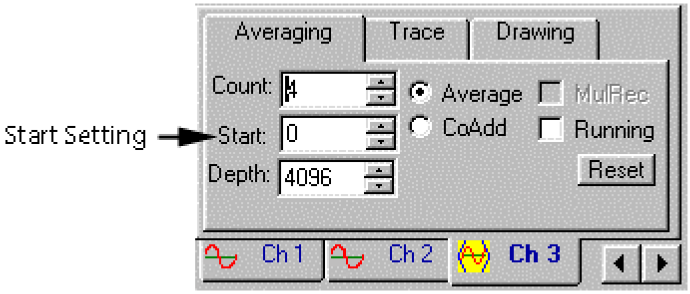
To change the number of count, do one of two things:

- Click on the increment/decrement buttons to change the value
- Click in the Counter field, press Backspace to erase the current value, and type in a new value.

For more details on Count, see Averaging tool on page C-191.

Channel Control: Averaging Tab: Start

The Start setting is located in the Averaging tab of the Channel Control.

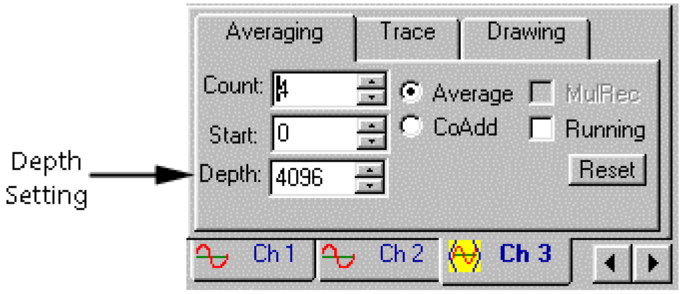


The Start setting specifies the first sample of the averaged region relative to the trigger position. For a selected portion of the acquisition to be averaged, the user can select the starting point of the signal by entering a number in this field.

For more details on Start, see Averaging tool on page C-191.

Channel Control: Averaging Tab: Depth

The Depth setting is located in the Averaging tab of the Channel Control.

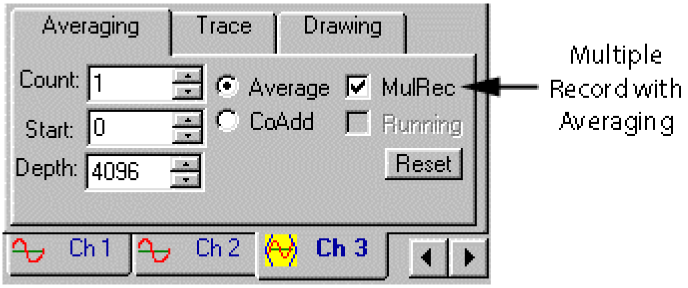


The Depth setting specifies the total length of the signal to be averaged. Use the increment buttons to set the desired length.

For more details on Depth, see Averaging tool on page C-191.

Channel Control: Averaging Tab: MulRec

The Multiple Record setting is located in the Averaging tab of the Channel Control.



The Multiple Record setting with the Average radio button selected, specifies the type of averaging to be performed when Multiple Record mode is enabled.

When in Multiple Record mode, averaging can be performed in one of the following ways:

- A single "MulRec" record can be used to produce a single averaged non-MulRec record. This method is best when the incoming signals are non-repetitive, such as in Imaging applications.
- A specified number of MulRec records can be averaged to produce a single averaged record. This method is best for applications like Mass Spectrometry, where the signals are highly repetitive.
- The running average of any single "MulRec" record can be accomplished.

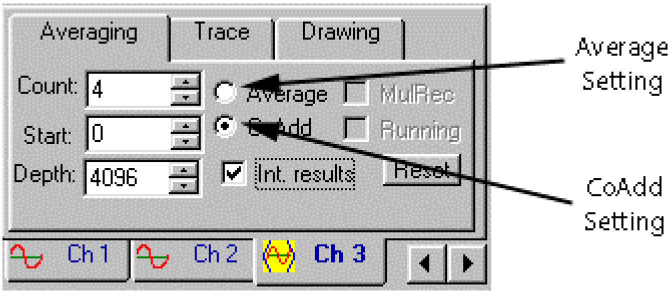
Note

When **Mulrec** is checked, the Record Counter disappears from the Trace tab of the Channel Control.

For more details on Averaging, see Averaging tool on page C-191.

Channel Control: Averaging Tab: Average vs. CoAdd

The Average and CoAdd settings are located in the Averaging tab of the Channel Control.



When Averaging is enabled, the first point of the first acquisition is added to the first point of the second, third, fourth..., nth acquisition to produce the first point of the "Result." The same is done for all other points in the record.

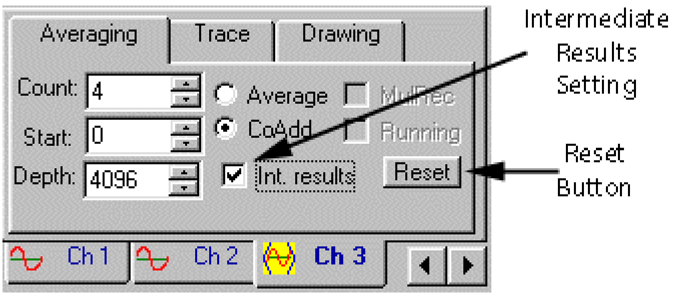
Users are provided an option to either average or simply co-add the data.

- **Averaging**: The result is divided by "n."
- **Co-Adding**: No division is done on the result. The data resulting from co-adding is 32 bit data.

For more details on Averaging, see Averaging tool on page C-191.

Channel Control: Averaging Tab: Intermediate Results and Reset

The Intermediate Results and Reset settings are located in the Averaging tab of the Channel Control.



When acquisitions are co-added, the Averaging tool allows the user to request the display of intermediate results. This means that GageScope® will display the results while it performs the adding process, rather than displaying just the final result.

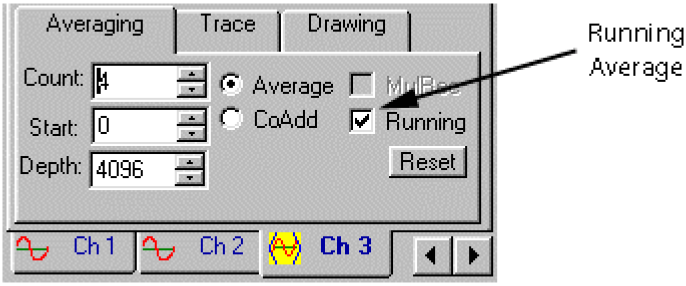
- To display intermediate results, click the **Intermediate Results** check-box.
- By clicking the **Reset** button, the Averaging or the CoAdding procedure starts all over again.

Notes

- Please note that display of Intermediate Results may slow the co-adding process.
- For more details on Averaging, see Averaging tool on page C-191.

Channel Control: Averaging Tab: Running

The Running average settings are located in the Averaging tab of the Channel Control.



The running average is performed on the acquisitions. The N most recent acquisitions are averaged and the result is stored in the results channel.

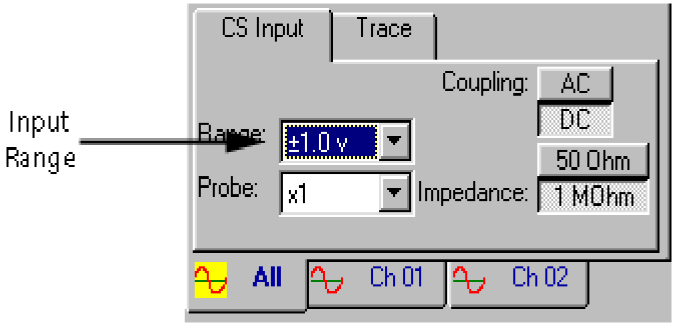
This option is also available for the MulRec channels, however, only one MulRec channel can be averaged at a time.

Notes

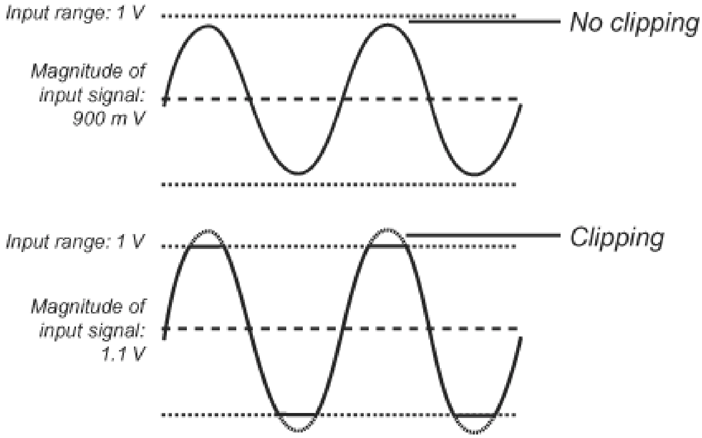
- The maximum number that can be entered for the Count in the Running average mode is 256 in the GageScope® Professional Edition. The Running average is not available in the Standard Edition of GageScope®.
- For more details on Running, see Averaging tool on page C-191.

Channel Control: CS Input Tab: Input Range

The Input Range setting is located in the CS Input tab of the All Channel Control.



The input range defines the range in volts the CompuScope card should measure when converting analog signals. For example, if the input range is set to ±1 V, then a signal that reaches 900 mV will be captured in its entirety, while the peaks of a signal measuring 1.1 V will be cut off or clipped.



Changing the Input Range

1. Click on the CS Input tab of the Channel Control.

2. Click on the Input Range drop-down list.

3. Select a value from the list.

Notes

- The list that appears in the input range will depend on the type of CompuScope card being used.

- The input range for the CompuScope 1016 is fixed at ±2V, so if you are using this card you will not be able to change this setting.

Note that Input Range is not linked to the Vertical Scale setting, unless you select Scope Mode.

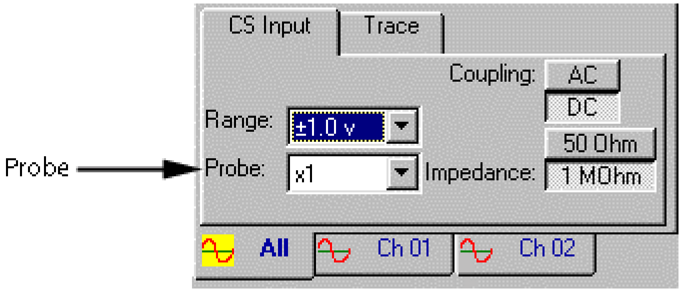
We have decoupled vertical scale from the input range in order to allow you to take advantage of the high vertical resolution offered by CompuScope cards. You can capture a signal with ±1V range and still view it with a vertical scale of 50 mV/div.

All digital oscilloscopes on the market are limited to 8 bit resolution and can digitize the signal with 256 levels only. This way, they can fit all 256 pixels on screen. CompuScopes, which offer up to 16 bit resolution, digitize the signal with up to 65,536 levels. There is no screen in the world that can display that many pixels. Hence, the ability to display only a portion of the capture signal’s dynamic range is necessary.

If, however, you are more familiar with the traditional linkage of vertical scale and input range, simply select **Scope Mode**.

Channel Control: CS Input Tab: Probe

The Probe setting is located in the CS Input tab of the All Channel Control.



The Probe setting is used to define the gain of the oscilloscope probe used to capture external signals. The available values are ±1V, ±2V, ±5V, ±10V, ±20V, ±50V, ±100V, ±200V, ±500V, ±1000V, ±2000V and ±5000V.

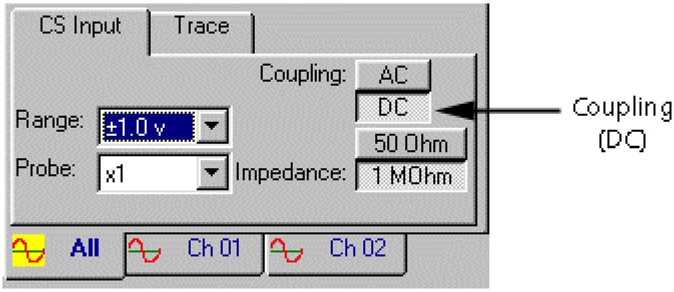
It should be noted that GageScope® does not read the probe settings used. It is entirely up to the user to enter the correct probe setting.

Changing the Input Range

1. Click on the **CS Input** tab of the Channel Control.
2. Click on the **Probe** drop-down list.
3. Select a value from the list.

Channel Control: CS Input Tab: Coupling

The Coupling setting is located in the CS Input tab of the All Channel Control.



By default GageScope® sets all channels to DC coupling. This option will not be available if your CompuScope card does not support this feature.

Toggling the Coupling via the Channel Control

1 Click on the **CS Input** tab in the Channel Control.

2 To change to AC coupling, click on the **AC** button. To change back to DC coupling, click on the **DC** button.

Toggling the Coupling via the Channel Context Menu

1 Position the mouse pointer on a channel's zero line. The channel will turn white to indicate that it has the current focus.

2 Click on the right mouse button to bring up the channel context menu.

3 Click on **Coupling** with the left mouse button. The Coupling sub-menu appears. The current setting has a bullet next to it.

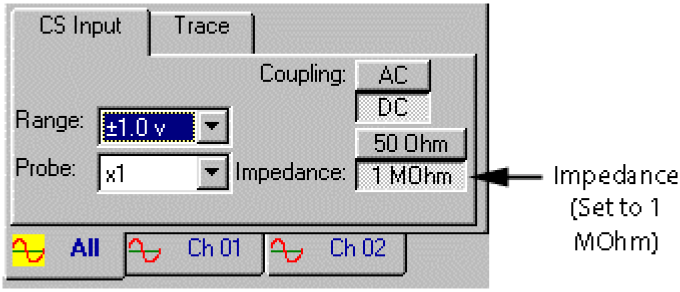
4 Click on **AC** or **DC**.

Notes

- This setting will not be available if you are using a CompuScope 1016, as this card is DC only.
- For most CompuScope cards, only DC coupling is available when impedance is set to 50 Ohm.

Channel Control: CS Input Tab: Impedance

The Impedance setting is located in the CS Input tab of the All Channel Control.



By default GageScope® sets all channels to 1 MOhm impedance. This option will not be available if your CompuScope card does not support this feature.

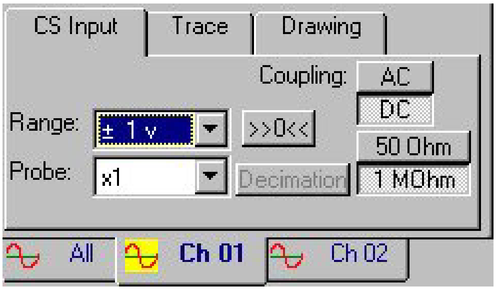
Changing the Impedance

1 Click on the **CS Input** tab in the Channel Control.

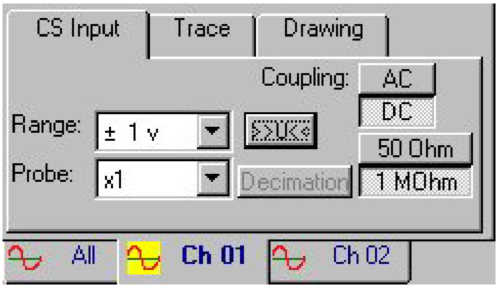
2 To change to 50 Ohm impedance, click on the **50 Ohm** button. To change to 1 MOhm impedance, click on the **1 MOhm** button.

Channel Control: CS Input Tab: Null Channel Input

The Null Channel Input setting is located in the CS Input tab of the **Al**l Channel Control. It also appears in the CS Input tab of the individual channel control.



Click the Null Channel input command button, >>0<<, to activate this feature. In the activated state the button will appear disabled:



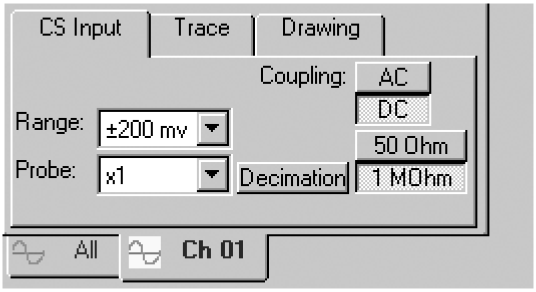
When clicked, the Null Channel Input button will force recalibration of the hardware taking the average value of the current input as a new reference for the zero level.

Notes

- Changing hardware settings such as sample rate, acquisition mode or input range will cancel the previously issued Null Channel Input command. Changing impedance or coupling will not change the calibration of the card however it can affect captured data and may require recalibration.

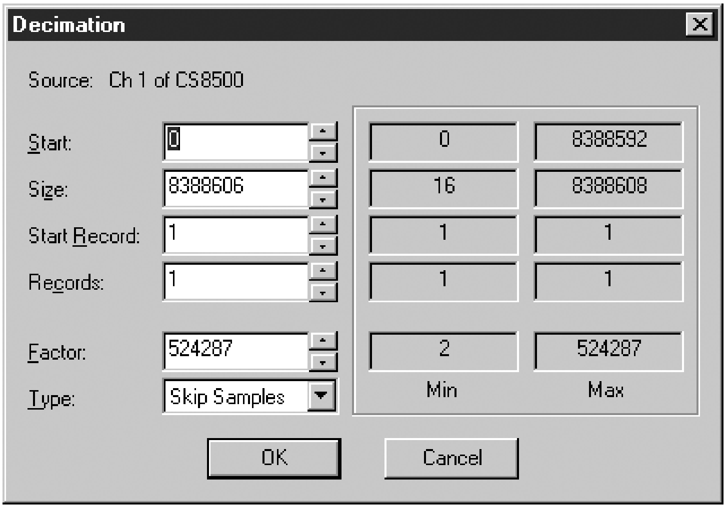
Channel Control: CS Input Tab: Decimation

The decimation setting is located in the CS Input tab of the Channel Control.



Decimation is a feature by virtue of which a user can view large amounts of acquired data within GageScope.

Click on the **Decimation** command button view the Decimation dialog box:

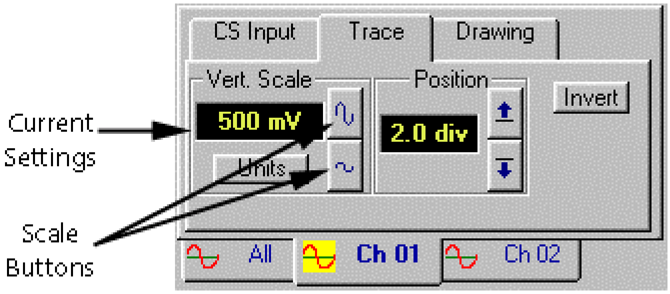


The settings in the decimation dialog box and the deep memory buffer size setting in the **Preferences** dialog box, controls the decimation procedure.

Refer to the Tutorial section of this manual for a detailed explanation of how to use the decimation features.

Channel Control: Trace Tab: Vertical Scale

The Vertical Scale setting is located in the Trace tab of the Display Control.



This setting is available whether the All Channels tab or an individual channel tab is current.

The vertical scale of a channel is measured in terms of *Volts per division*, a division equaling one square in the grid. (For more information on the grid, see page C-110.)

Changing the Scale

1 Click on the **Trace** tab in the Channel Control.

2
- To increase the scale, click . This shrinks the signal.
- To decrease the scale, click . This makes the signal taller.

Notes

- The default vertical scale is **1V/div**. In other words, for every 1V of signal, GageScope® shows 1V per grid square. If you increase the vertical Scale to **2V/div**, a 1V signal displays as half of one grid square. If you decrease the vertical scale to **500mV/div**, a 1V signal displays at the height of two grid squares.

- If you increase the vertical scale so that the signal peaks go off screen, you can use the vertical scroll bar to scroll up or down to reach the peaks.

Note that Input Range is not linked to the Vertical Scale setting, unless you select Scope Mode.

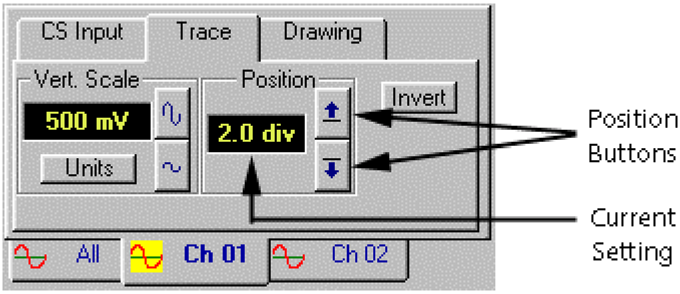
We have decoupled vertical scale from the input range in order to allow you to take advantage of the high vertical resolution offered by CompuScope cards. You can capture a signal with ±1V range and still view it with a vertical scale of 50 mV/div.

All digital oscilloscopes on the market are limited to 8 bit resolution and can digitize the signal with 256 levels only. This way, they can fit all 256 pixels on screen. CompuScopes, which offer up to 16 bit resolution, digitize the signal with up to 65,536 levels. There is no screen in the world that can display that many pixels. Hence, the ability to display only a portion of the capture signal's dynamic range is necessary.

If, however, you are more familiar with the traditional linkage of vertical scale and input range, simply select **Scope Mode**.

Channel Control: Trace Tab: Position

The Position setting is located in the Trace tab of the Channel Control.



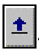
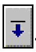
This setting is available only if an individual channel tab is current (i.e., Ch1, Ch2…).

The position of a channel is measured in terms of divisions relative to the X axis, a division equaling one square in the grid.

It should be noted that changing the “position” of a channel only changes the display offset for that channel: no hardware offset is induced due to position changes. (For more information on the grid, see page C-110.)

Moving a Channel via the Position Buttons

1 Click on the **Trace** tab in the Channel Control.

2
- To raise a channel, click on .
- To lower a channel, click on .

Moving a Channel Directly in the Display Window

1 Position the mouse pointer on a channel’s zero line. The channel will turn white to indicate that it has the current focus, and the mouse pointer will change to a drag pointer.

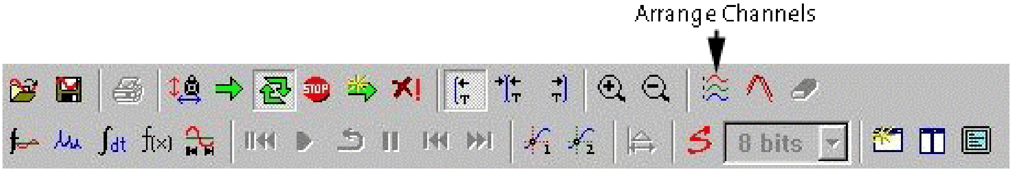
2 Hold down the mouse button.

3 Move the mouse up or down.

Notes

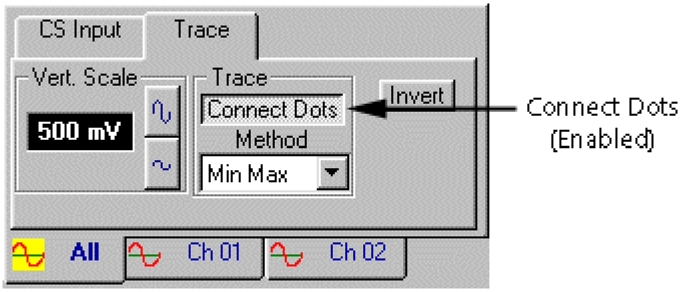
- If you have one channel available upon startup, GageScope® places this channel at 0.0 divisions (in other words, on the X Axis). If you have two channels available at startup, GageScope® places channel 1 at 2.0 divisions and channel 2 at –2.0 divisions.

- To re-arrange the position of all channels quickly, use the Arrange Channels button in the toolbar.



Channel Control: Trace Tab: Connect Dots

The Connect Dots button is located in the Trace tab of the Channel Control.



This setting is available only if the **All Channels** tab is current. If an individual channel tab is current, the Connect Dots setting appears under the Drawing tab.

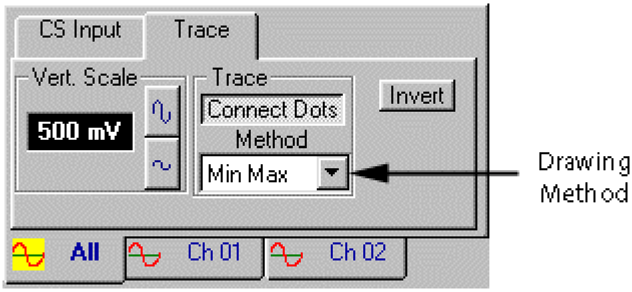
By default GageScope® draws a line between each sample point in order to display a continuous waveform. To have GageScope® display as single points, you can turn off the **Connect Dots** setting.

Toggling the Connect Dots Button

1. Click on the **Trace** tab of the Channel Control (make sure the All Channels tab is current).
2. Click on the **Connect Dots** button to toggle this option on and off.

Channel Control: Trace Tab: Method

This setting is located in the Trace tab of the Channel Control.



This setting is available only if the **All Channels** tab is current. If an individual channel tab is current, the Method setting appears under the Drawing tab.

The Method setting defines how GageScope® displays traces based on the acquired data. The options are Mean and MinMax.

MinMax When two or more samples are to be displayed on the same screen pixel column, MinMax will plot the minimum and maximum values, then draw a line between them. This method allows glitch detection even at very low timebases.

This is the default setting.

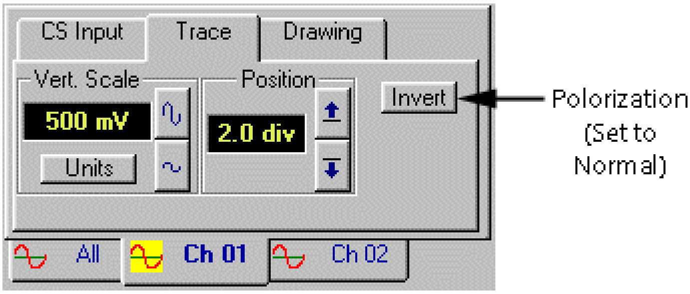
Mean When two or more samples are to be displayed on the same screen pixel column, Mean will plot the average of the samples. In display terms, the Mean method reduces the apparent signal noise when capturing slow signals at high sample rates and then displaying many points on the screen. The Mean display method allows over-sampled data to be digitally filtered before display, thereby improving the signal to noise ratio.

Changing the Method via the Trace Tab

1. Click on the **Trace** tab of the Channel Control.
2. Click on the **Method** drop-down list.
3. Select a method from the list.

Channel Control: Trace Tab: Polarization

The Polarization setting is labeled “Invert” and is located in the Trace tab of the Channel Control.



This setting is available whether **All Channels** or an individual channel is current.

Polarization refers to how the trace is displayed relative to the X axis. When polarization is normal, the channel is displayed exactly as captured. Inverting the polarization inverts the signal about the X axis for Display purposes only, without affecting the actual data.

If a channel is inverted, the color of the channel identifier at the far left of the signal will be reversed as well. For example, if channel 1 is inverted and its color is yellow, then the channel identifier will be black text in a yellow box.

Toggling the Polarization via the Trace Tab

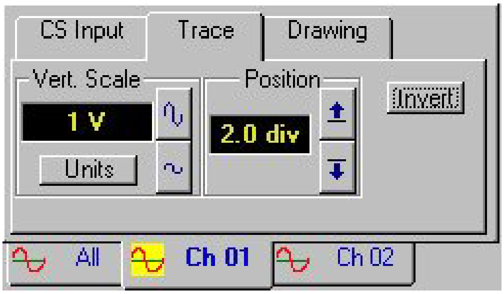
1. Click on the **Trace** tab in the Channel Control.
2. Click on the **Invert** button to toggle this option on and off. If the **Invert** button is enabled, polarization is inverted; otherwise it is normal.

Changing the Polarization via the Channel Context Menu

1. Position the mouse pointer on a channel’s zero line. The channel will turn white to indicate that it has the current focus, and the mouse pointer will change to a drag pointer.
2. Click on the right mouse button to bring up the channel context menu.
3. Click on **Invert** to toggle this setting. If **Invert** has a checkmark next to it, polarization is inverted; otherwise it is normal.

Channel Control: Trace Tab: Units

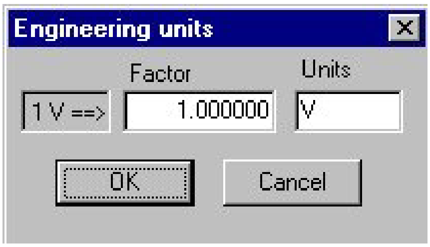
The Units setting is labeled "Unit" and is located in the Trace tab of the Channel Control.



Engineering units are now available in GageScope. This feature allows the user to specify a conversion factor and a base units text string for each channel. The factor must be entered in base units (e.g. A or Pa) and not in derived units (e.g. mA or kPa). GageScope® will automatically perform metric scaling.

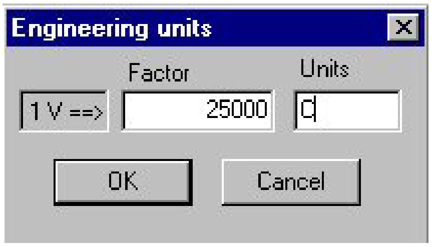
How To Invoke the Engineering Units

1 Click on the **Units** tab in the Channel Control. The Engineering Units dialog box appears:

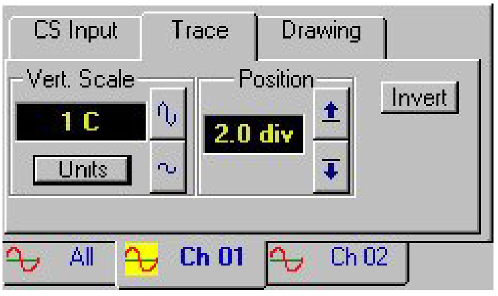


2 Assume that the user has his CompuScope connected to a thermocouple that produces a 40 microVolt DC voltage for every 1 degree Celsius change in temperature. In this case, the user would enter "C" as the **Units** text string. The conversion **Factor** would be: 1 / (40 x 10^ -6) = 25000.

Enter 25000 for the Factor and C for the Units in the respective text boxes.



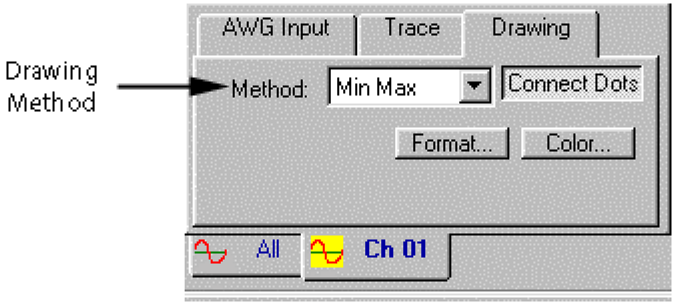
3 Click **OK**. You will notice that the Vertical Scale now measures in degree Celsius.



Note that depending on the conversion factor, you may have to use the amplitude expansion or contraction buttons to view the signal properly.

Channel Control: Drawing Tab: Method

The Method button is located in the Drawing tab of the Channel Control.



The Drawing tab is not available when the **All Channels** tab is current. Drawing tab elements apply to individual channels only. If the **All Channels** tab is current, the Method setting appears in the Trace tab.

The Method setting defines how GageScope® displays traces based on the acquired data. The options are MinMax and Mean. These options affect the screen representation of the data only and do not change the actual signal data.

MinMax When two or more samples are to be displayed on the same screen pixel column, MinMax will plot the minimum and maximum values, then draw a line between them. This method allows glitch detection even at very low timebases.

This is the default setting.

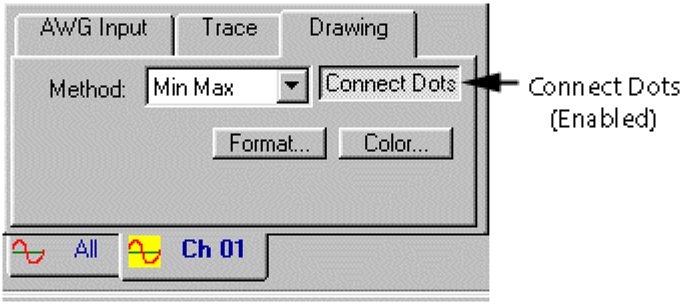
Mean When two or more samples are to be displayed on the same screen pixel column, Mean will plot the average of the samples. In display terms, the Mean method reduces the apparent signal noise when capturing slow signals at high sample rates and then displaying many points on the screen. The Mean display method allows over-sampled data to be digitally filtered before display, thereby improving the signal to noise ratio.

Changing the Method via the Trace Tab

1. Click on the **Drawing** tab of the Channel Control.
2. Click on the **Method** drop-down list.
3. Select a method from the list.

Channel Control: Drawing Tab: Connect Dots

The Connect Dots button is located in the Drawing tab of the Channel Control.



The Drawing tab is not available when the **All Channels** tab is current. Drawing tab elements apply to individual channels only. If the **All Channels** tab is current, the Connect Dots setting appears in the Trace tab.

By default GageScope® draws a line between each sample point in order to display a continuous waveform. To have GageScope® display as single points, you can turn off the **Connect Dots** setting.

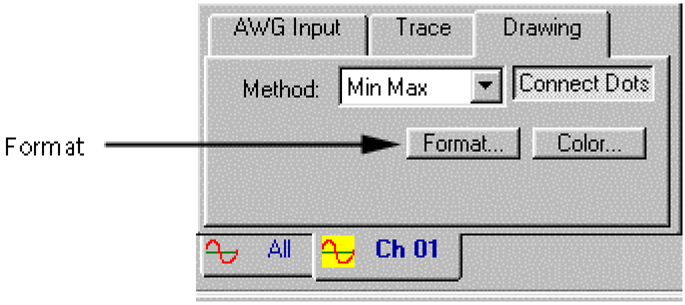
Toggling the Connect Dots Button

1 Click on the **Drawing** tab of the Channel Control (make sure the All Channels tab is not current).

2 Click on the **Connect Dots** button to toggle this option on and off.

Channel Control: Drawing Tab: Format

The Format setting is located in the Drawing tab of the Channel Control.



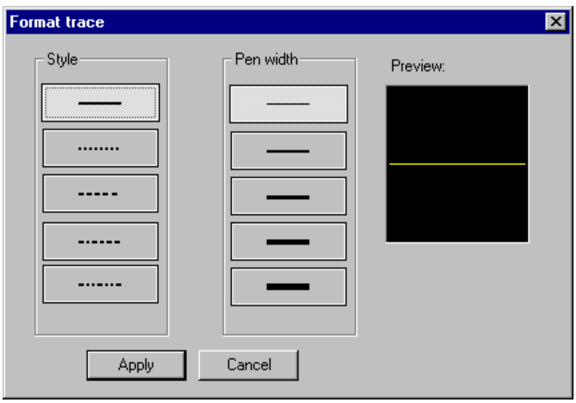
The Drawing tab is not available when the **All Channels** tab is current. Drawing tab elements apply to individual channels only.

The Format setting defines how GageScope® will draw a trace.

Changing the Format

1 Click on the **Drawing** tab of the Channel Control (make sure the **All Channels** tab is not current).

2 Click on the **Format** button. The Format trace dialog appears.



3 Select a style from the list on the left and a pen width from the list on the right. GageScope® will show you a preview of your selection.

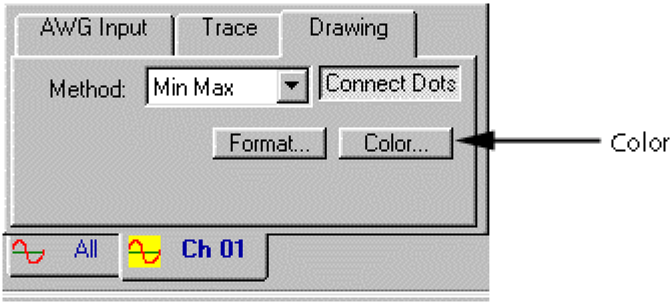
4 Click **Apply** to confirm your selection.

Notes

If **Connect Dots** is turned off, your changes to the Format settings will not be visible on the screen. The line style does not have any effect on the size of the dots but the pen width does change the dot size.

Channel Control: Drawing Tab: Color

The Color button is located in the Drawing tab of the Channel Control.

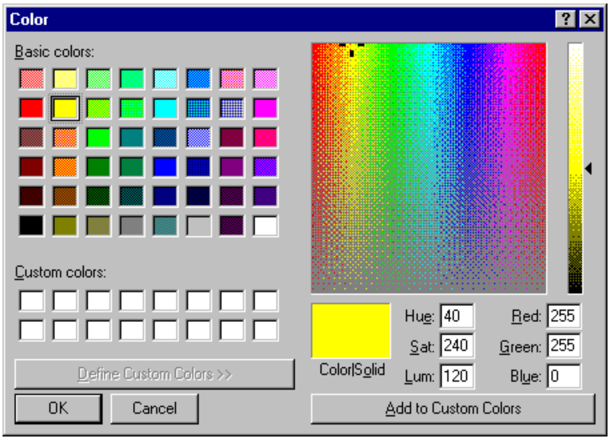


The Drawing tab is not available when the **All Channels** tab is current. Drawing tab elements apply to individual channels only.

The Color Button allows you to change the color of a channel.

Changing the Color via the Drawing Tab

1 Click on the **Color** Button in the Drawing tab. The color dialog appears.



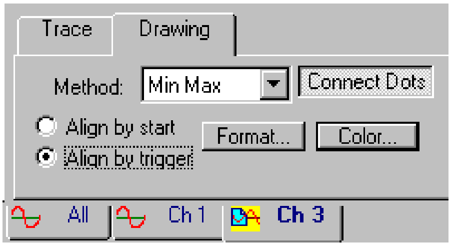
2 Click on a color in the dialog box, then click **OK**.

Changing the Color via the Channel Context Menu

1. Position the mouse pointer on a channel’s zero line in the Display Window. The channel will turn white to indicate that it has the current focus, and the mouse pointer will change to a drag pointer.
2. Right-click to bring up the channel context menu.
3. Click on **Color** with the left mouse button.
4. Click on a color in the dialog box, then click **OK**.

Channel Control: Trace Tab: Align by Start / Align by Trigger

This setting is located in the Drawing tab of the Channel Control when a channel has been loaded which does not cross the Trigger Marker (start, or time zero).



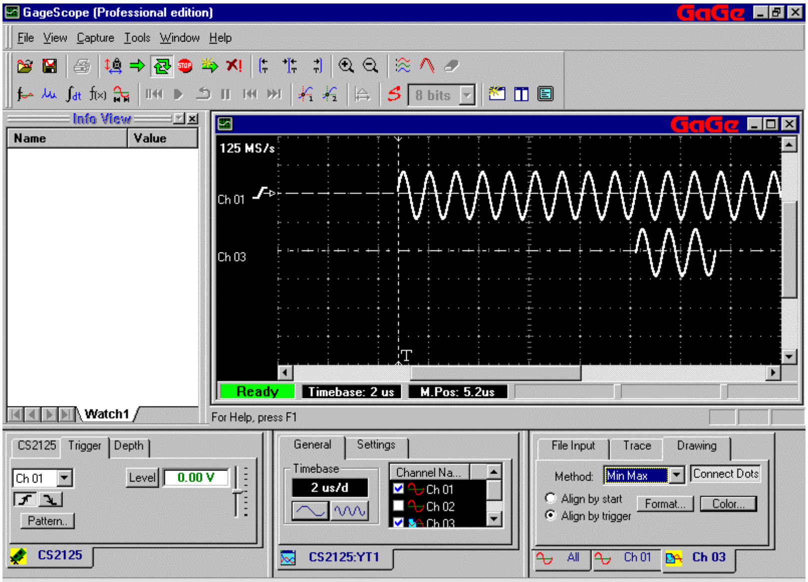
When displaying a saved signal that does not cross the trigger marker, you can specify where on the screen will that signal be displayed.

- **Align by Start:** GageScope® will display the signal at the Start (at the Trigger Marker, or time zero).

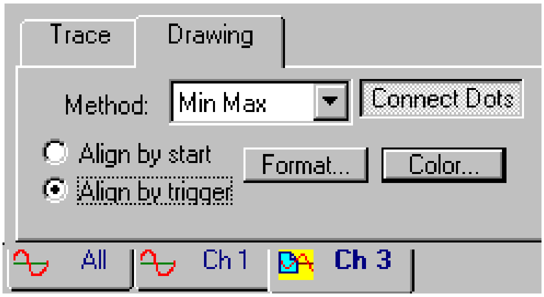
- **Align by Trigger:** GageScope® will display the signal at its original time position relative to time zero. This is the default setting.

How it works

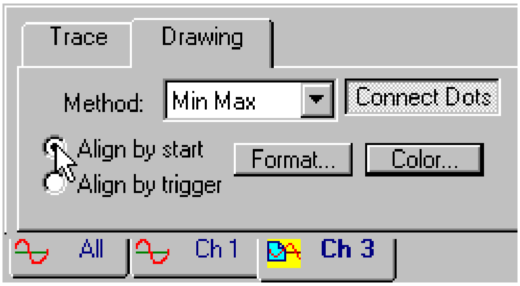
1 When you load a channel that does not cross the Trigger Marker, by default it will be **Aligned at Trigger** (at its original time position in the grid). Below, Channel 3 is aligned at Trigger.



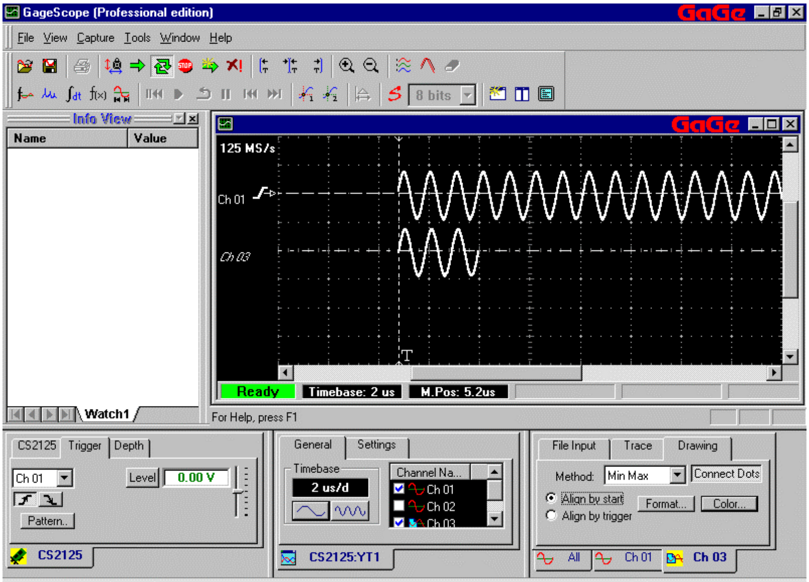
2 Click on the appropriate Channel tab in the Channel Control. Then click on the **Drawing** tab.



3 Click on **Align by Start**.



4 The signal moves to line up with the Start (the Trigger Marker). Below, Channel 3 is aligned by Start.

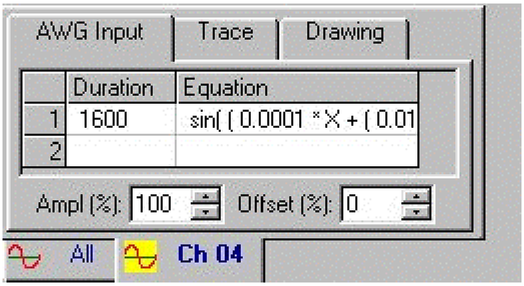


Notes

- This setting can also be specified via the Preferences dialog in the Tools Menu. See page C-41.

Channel Control: AWG Input Tab: Duration

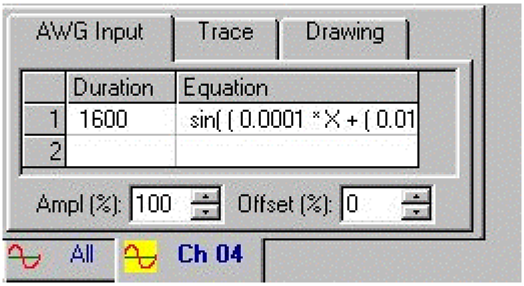
The duration settings are located in the AWG Input tab of the Channel Control.



The duration setting refers to the total length, in Samples, of the signal being created. The current setup shows that the duration of the signal under creation is **1600** Samples. Refer to the Tutorial section of this guide for further details.

Channel Control: AWG Input Tab: Equation

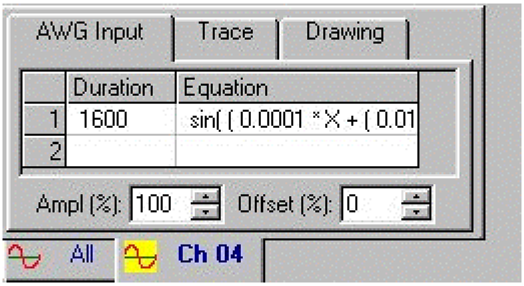
The equation settings are located in the AWG Input tab of the Channel Control.



You can enter or modify the equation to be created using the setting in the AWG Input control. You can observe that the equation for a **sine** function appears in the equation text box. Refer to the Tutorial section of this manual for further detail on entering or modifying equations to create user defined signals for arbitrary waveform generation.

Channel Control: AWG Input Tab: Ampl

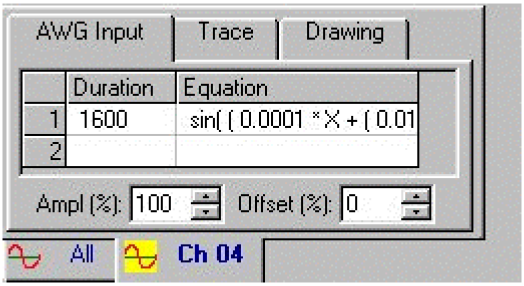
The Amplitude (**Ampl**) settings are located in the AWG Input tab of the Channel Control.



The **Ampl**itude is the amplitude of the signal to be generated. The amplitude can be specified as the percentage of the selected output range. You can modify the amplitude of the signal to be created by using the increment/decrement buttons. You can also change the amplitude by clicking inside the amplitude text box and entering the value.

Channel Control: AWG Input Tab: Offset

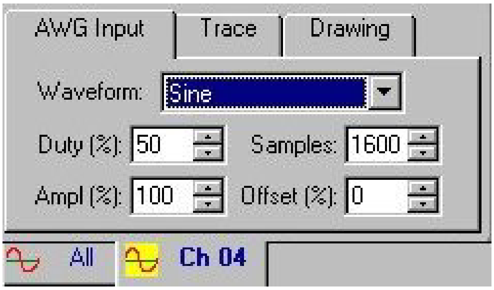
The Offset settings are located in the AWG Input tab of the Channel Control.



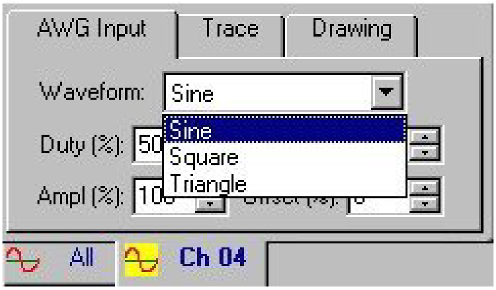
The **Offset** is the DC offset that can be specified as a percentage of the selected range for the signal to be generated. You can modify the Offset of the signal to be created by using the increment/decrement buttons. You can also change the Offset by clicking inside the amplitude text box and entering the value.

Channel Control: AWG Input Tab: Waveform

The waveform settings are located in the AWG Input tab of the Channel Control.



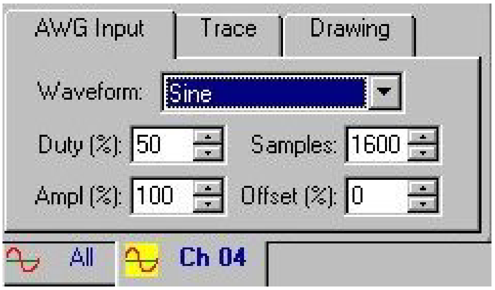
There are three types of standard signals that can be created for generation using the AWG capabilities of GageScope. Click on the Waveform drop down button to view the list of available standard signals as follows:



Click on the desired waveform to be selected for generation. The generation process is discussed in the System Control section on page C-98.

Channel Control: AWG Input Tab: Duty

The duty settings are located in the AWG Input tab of the Channel Control.

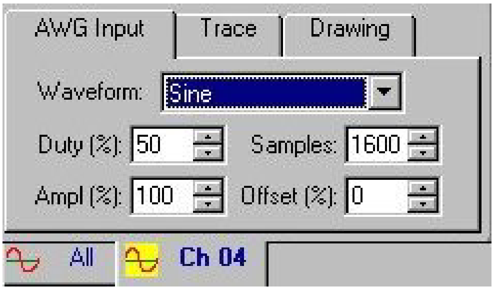


The Duty setting provides a means for specifying the duty cycle for the signal to be created and later generated. Note that the number entered in the duty cycle text box is a certain percentage of the total number of samples in the cycle. The total number of samples is specified in the Samples text box.

In the example above, a duty cycle of 50% is desired. The total number of samples in the cycle is given to be 1600. Therefore a 50% duty cycle for a sine wave will create an evenly balanced sine wave.

Channel Control: AWG Input Tab: Samples

The samples settings are located in the AWG Input tab of the Channel Control.



For standard waveform generation, the **Samples** setting specifies the total number of samples for a single period of the wave. Looking at the settings in the figure above, a 1600 samples sine wave will be created. The period of the sine wave will be determined by the generation rate specified in the system control. Refer to the system control section of this chapter for details on the generation procedure (page C-98).

Display Window

The Display Window shows live signals being acquired from a CompuScope card as well as signals loaded from disk. You can move signals directly in this window as well as change certain settings via context menus.

In this Section…

Channel Context Menu

Many options found in the controls and menus are also available through the channel context menu.

1 Position the mouse pointer on a channel's zero line. The channel will turn white to indicate it has the current focus, and the mouse pointer will change to a drag pointer.

2 Click the right mouse button. The channel context menu appears.

3 Click on a command using the left mouse button.

Options

Save Channel This is equivalent to saving a channel via the Save Channel command in the File menu.

Hide Hides the selected channel. This is equivalent to hiding a channel via the Channel List in the General tab of the Display Control.

Send To You can save the signal files to a floppy drive, My Briefcase folder on your hard disk, or post these directly on a web site.

Delete Channel Removes the channel from the Display Window. No equivalent. Only math channels and channels originating from a file may be deleted, not live signals originating from a CompuScope card. Note that all channels that were created from the channel selected for deletion will also be deleted. For example, if Ch 5 is a math channel that was created by adding Ch 3 and Ch 4, and Ch 3 is deleted, Ch 5 will be deleted also. You will be notified of this via the Confirm Deletion dialogs.

Invert Inverts the polarization. This is equivalent to inverting the polarization via the Trace tab of the Channel Control.

Reference Lines The reference settings **High**, **Middle** and **Low** are used to determine the High Level, Middle Level, and Low Level when calculating certain time parameters. For example, the **Rise Time** is the length of time for a signal's rising edge to go from the low level to the high level.

Color This is equivalent to changing the channel color via the Drawing tab of the Channel Control.

Edit Select this option to perform basic editing operations like Cut, Copy, and Paste on signals.

Display Window Context Menu

Several of the general display options, such as channel visibility and full screen view, are located in the Display context menu.

1 Position the mouse pointer on an empty area of the Display Window (not on a channel or other item).

2 Click with the right mouse button to bring up the Display context menu.

Note that few options are disabled depending on GageScope® settings.

3 Click on a command using the left mouse button.

Options

New Math Channel	Creates a new channel by combining other channels mathematically. This is equivalent to creating a channel via the Math command in the Tools menu.
Load Channel	This is equivalent to loading a channel via the Load Channel command in the File menu.
Arrange Traces	Rearranges the visible channels in the Display Window in order of channel number. This is equivalent to clicking on the Arrange Traces button in the toolbar.
Ch 1 **Ch 2** **…**	Hides or shows a channel. This is equivalent to hiding a channel via the Channel List in the General tab of the Display Control.
Grid **Zero Line** **Trigger Line**	Hides or shows the Grid, Zero Line or Trigger Line. This is equivalent to hiding these display elements via the Settings tab of the Display Control.
Left Bar	Hides the left bar of the Display Window, which contains the slope icon, the channel label, the sample rate indicator and the number of records captured in a multiple record acquisition. This is equivalent to selecting Left Bar from the Window menu.
Status Bar	Hides the status bar of the Display Window. This is equivalent to selecting Status Bar from the Window menu.
Color > Background **Color > Grid**	The options in the Color sub-menu allow you to change the background and grid color. This is equivalent to changing the color via the Settings tab of the Display Control.
Full Screen	Hides all controls to all the Display Window to expand to a full screen view. This is equivalent to selecting Full Screen from the View menu.
Cut, Copy, Paste	Cut, Copy and Paste are the three options available to edit the signals.

Trigger Context Menu

Trigger slope can be changed via the trigger context menu.

1 Position the mouse pointer on the trigger slope icon, located to the left of the channel.

2 Click with the right mouse button to bring up the trigger context menu.

3 Click on a command using the left mouse button.

Options

Positive Slope **Negative Slope**	This is equivalent to double-clicking on the slope icon to change the slope, or clicking on the slope buttons via the Trigger tab of the Systems Control.

A bullet next to an option indicates the setting currently being used.

General Display Window Shortcuts

Moving Channels

1 Position the mouse pointer on a channel’s zero line. The channel will turn white and the mouse pointer will change to a drag pointer, to indicate that the channel has the current focus.

2 Hold down the mouse button, then move the mouse up or down.

Raising the Trigger Level

1 Position the mouse pointer on the slope symbol at the far left of a channel. This is the trigger level.

2 Hold down the mouse button, then move the mouse up or down.

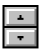
Changing the Trigger Slope

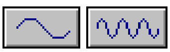
1 Position the mouse pointer on the slope symbol at the far left of a channel.

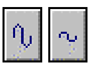
2
- Double-click to toggle the slope
  or
- Click the right mouse button to bring up the Slope context menu; click on either Positive or Negative to select the slope.

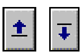
AutoRepeat Buttons

1 Position the mouse pointer on any button in the System, Display and Channel Controls which provides an increment/decrement function. These include:


All Increment/Decrement buttons


Timebase buttons


Vertical Scale buttons


Position buttons

2 Press and hold the left mouse button. The button will repeat its function for as long as you hold the button.

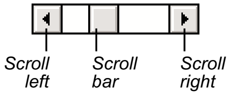
Scrolling using the Trigger Marker

1 Position the mouse pointer on the trigger marker line (the vertical line with the **T** at the bottom).

2 Press down the mouse button, and move the mouse left or right.

Scrolling using the Scroll Bars

The horizontal and vertical scroll bars allow you to scroll through the Display Window in order to see different parts of what was captured. To move toward the right, for example, you can click on the horizontal scroll bar's right scroll arrow.



You can also drag the scroll bar—the Display Window will update dynamically.

Scroll Bar Size

The size of the bar in the vertical and horizontal scroll bar indicates the size of the Display Window relative to the overall width of the signal from end to end. If the scroll bar is very small, the portion of the signal you are seeing is small compared to the whole signal. Conversely, if the scroll bar is two-thirds the size of the Display Window, then you are seeing two-thirds of the whole signal.

The scroll bar size is affected by the timebase. If the timebase is large (100us/d compared to the default 5us/d, for example), then the scroll bar gets larger, as you are able to see more of the waveform in the same-size Display Window. The Scroll bar always remains visible.

Status Bar

The Status Bar is located at the bottom of the Display Window.

Current Settings Reflected in the Status Bar

- Triggering activity
- Timebase
- M.Pos (distance from the trigger marker to the current position, in seconds)
- Cursor 1 and/or Cursor 2 values (if a cursor is enabled) and Cursor deltas (if both cursors are enabled) – see Cursors on page C-54 for more information.

Triggering Activity Messages

The text and background color have special meanings in the Status Bar. While the text indicates the current hardware status, the background color indicates the status of the currently displayed acquisition.

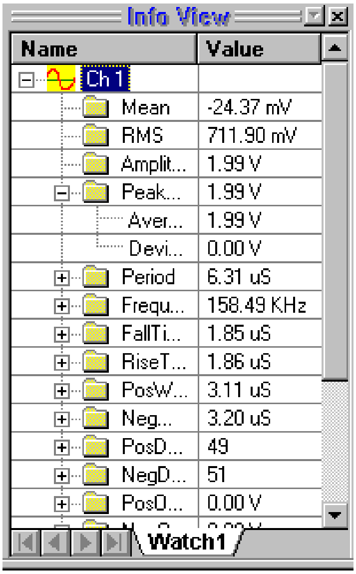
Waiting	The message "Waiting" means the system is currently waiting for a trigger to occur in order to perform a capture. "Waiting" is displayed on a magenta background.
Ready	The message "Ready" means a trigger event has occurred and a capture is being performed. A green background means a trigger occurred normally. A yellow background means a timeout occurred and GageScope® forced a trigger. A blue background means a trigger was forced in Continuous Capture mode.
Stop	The message "Stop" means GageScope® has stopped the current acquisition. A blue background means a trigger was forced; a green background means a trigger occurred normally.
Transferring	The message "Transferring" means GageScope® is transferring data from the board to the internal buffers of GageScope®.
Arming	The message "Arming" means that GageScope® is transferring data from the internal buffers to the board for generation.

InfoView

The InfoView Window truly differentiates GageScope® from all other oscilloscope programs on the market. Using the advanced analysis tools, GageScope® can analyze the signal data and display the results in the InfoView window. InfoView is what the marriage of instruments and computers is all about.

Any results calculated in GageScope® are displayed in InfoView.

Below is a sample of what the Waveform Parameters tool displays. For more information on advanced analysis tools, see page C-177.



Advanced Analysis Tools

Advanced analysis tools are now an integral part of GageScope® program that provide additional functionality, such as measurement and display of waveform parameters.

Following are the analysis tools currently available:

These advanced analysis tools are installed automatically when you install the GageScope® software program. To be able to exploit the maximum capabilities offered by these advanced analysis tools, you should purchase the Professional Edition of GageScope® .The Standard Edition of GageScope® also provides a selected capability of these advanced features.

AutoSave

AutoSave is a very powerful tool for data acquisition of transient signals. It allows unattended data capture to disk with accurate time and date stamping of when the trigger occurred..

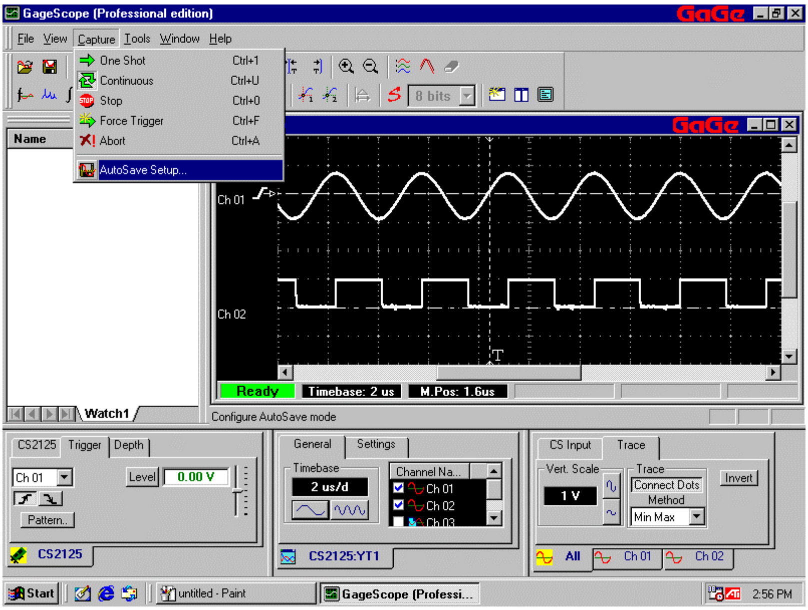
The AutoSave tool allows the periodic output of a channel to file. Each such file represents a single AutoSave record. The channels are saved to file using the current hardware configuration defined in the GageScope® session.

The AutoSave tool also allows benchmarking of AutoSave settings. The benchmarks allow a user to determine the time required to perform a single AutoSave cycle on a given system.

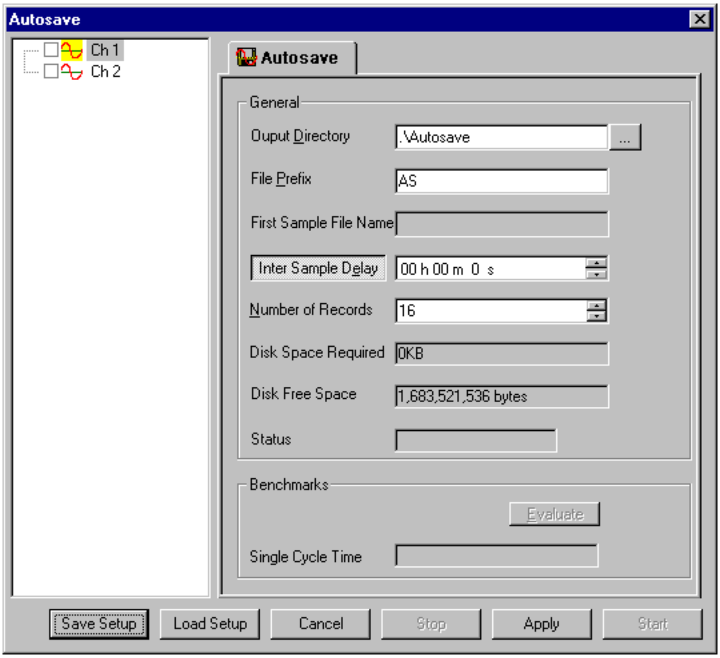
The AutoSave tool defines a new file extension, *.asf. ASF files contain AutoSave settings as defined inside GageScope®. Those settings are the inter-sample delay, the number of records, the AutoSave file prefix, and the output directory. Note that long file names are permitted. GageScope® can save and restore AutoSave setups to make them available for future sessions.

Setting up for an AutoSave Session

1 To set up AutoSave inside GageScope®, select **AutoSave Setup** from the Capture menu.



2 The AutoSave dialog will appear.



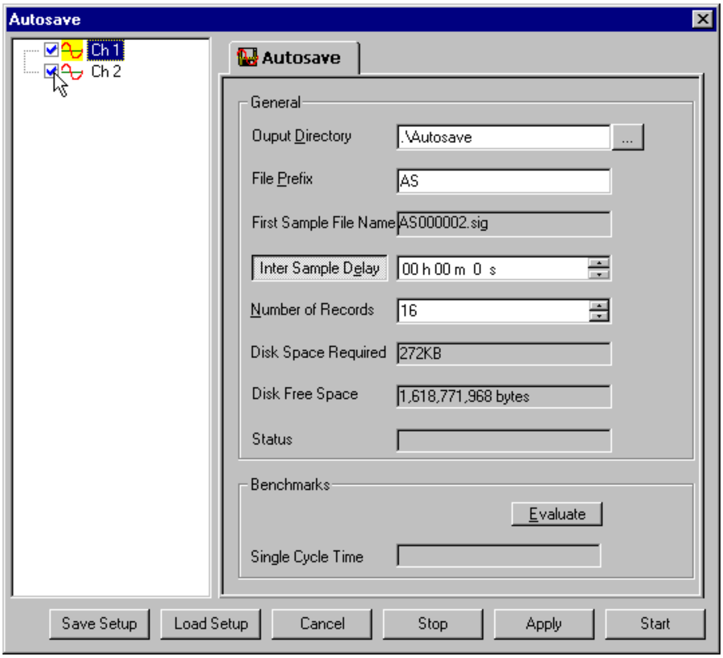
This dialog allows you to:

- select the channels for which AutoSave should be set up
- specify where the files will be saved
- specify the prefix for each file in the series
- specify the InterSample Delay, or the length of time between samples
- specify the number of records to be saved
- benchmark how long the system will need to perform a single AutoSave cycle using current settings
- save the current setup
- load a previously saved setup
- apply the current settings, allowing you to exit the dialog, make changes to your hardware settings, and return to the dialog with your changes preserved

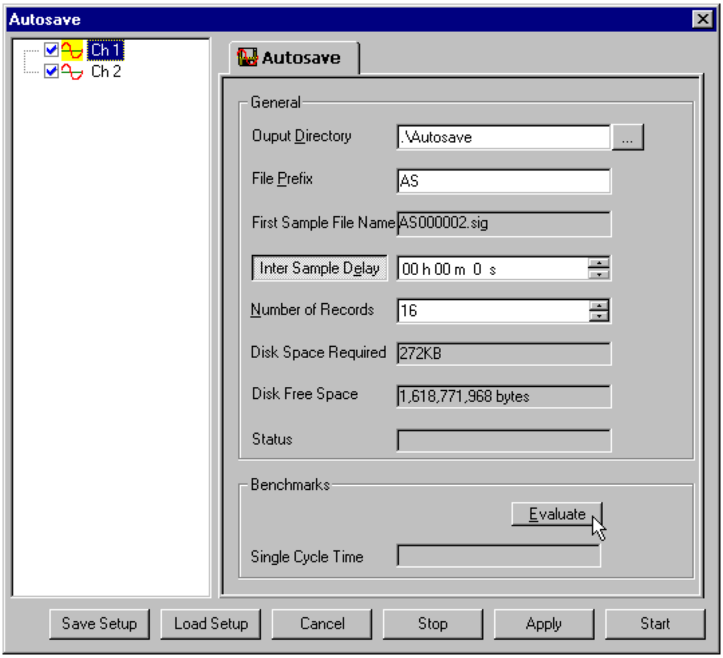
3 This dialog also provides the following information:

- disk space required for the AutoSave files, based on current settings
- free disk space on the current hard drive
- status of the AutoSave session (“Done” is displayed after a successful Benchmark or AutoSave)
- Time taken by the AutoSave session, either projected (benchmark) or actual
- See below for more details.

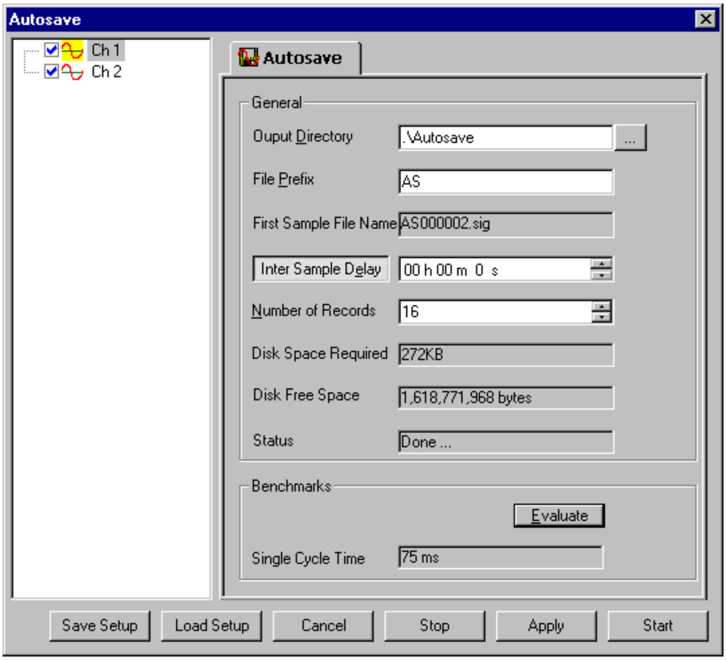
4 Click on one or more of the current channels to be included in the AutoSave session.



5 To determine how long it will take GageScope® to perform the AutoSave session based on current settings, select the channels you want to save and click on **Evaluate**.

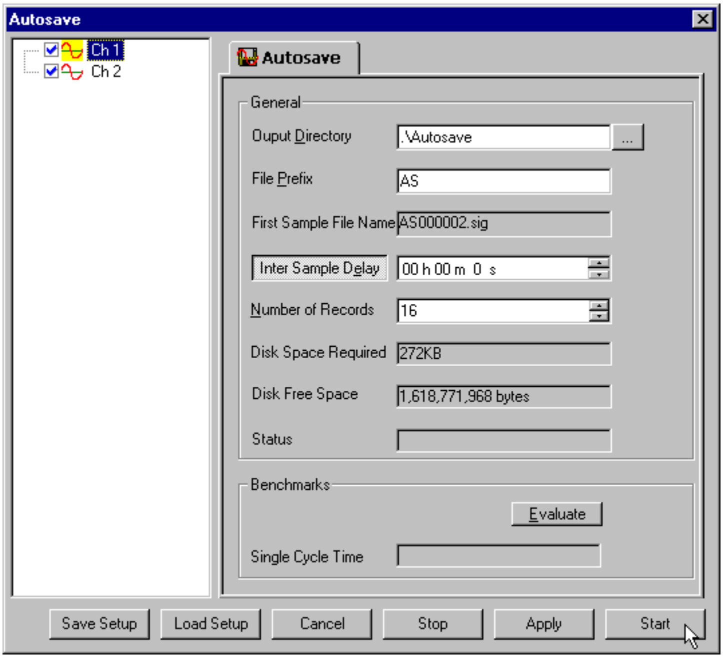


6 The screen below shows a completed Benchmark. Note that the Status field reads "Done." GageScope® has determined that the specified AutoSave session will take 75 ms and will require 272KB of hard disk space.

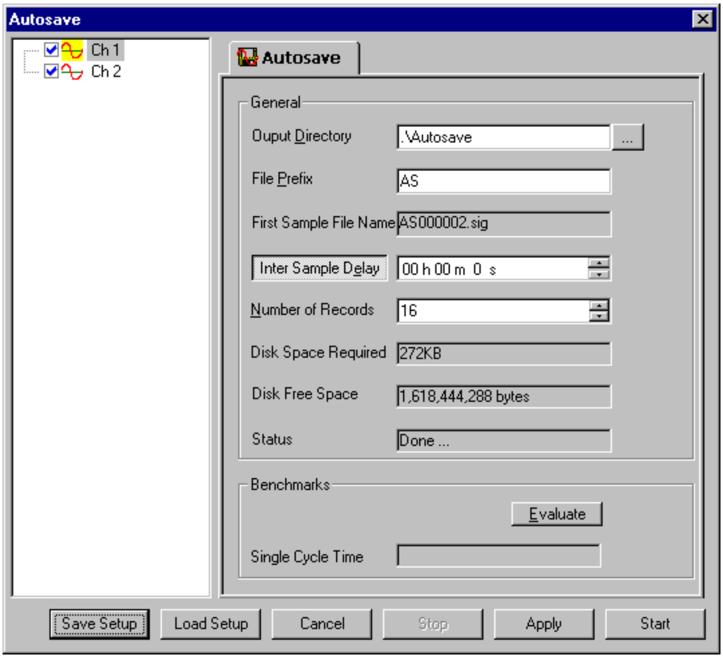


7 Note that if you wish to change hardware settings before beginning the AutoSave, you must exit the AutoSave Setup dialog. To preserve the AutoSave setup changes you have made, click **Apply**, then **Cancel**. Make the necessary hardware changes. When you re-open the AutoSave Setup dialog, your settings will have been retained.

8 After you have specified the desired AutoSave settings, click **Start** to begin the AutoSave process.

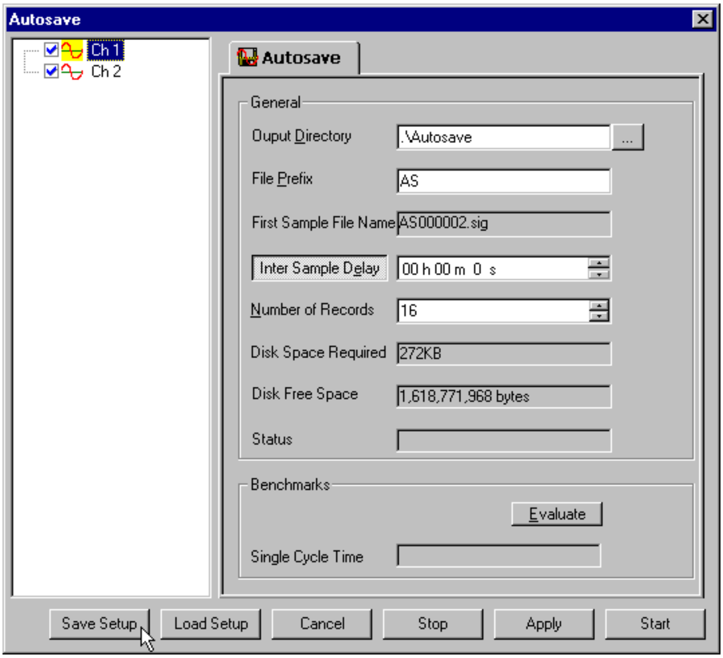


9 When the process is complete, the message “Done” will appear in the Status field.

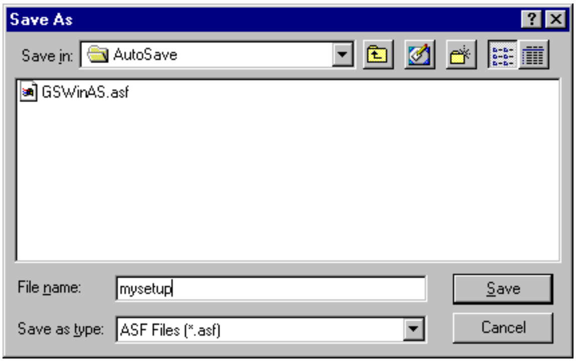


Saving and Loading AutoSave Setups

1 You can save the current AutoSave setup. To save, click **Save Setup**.



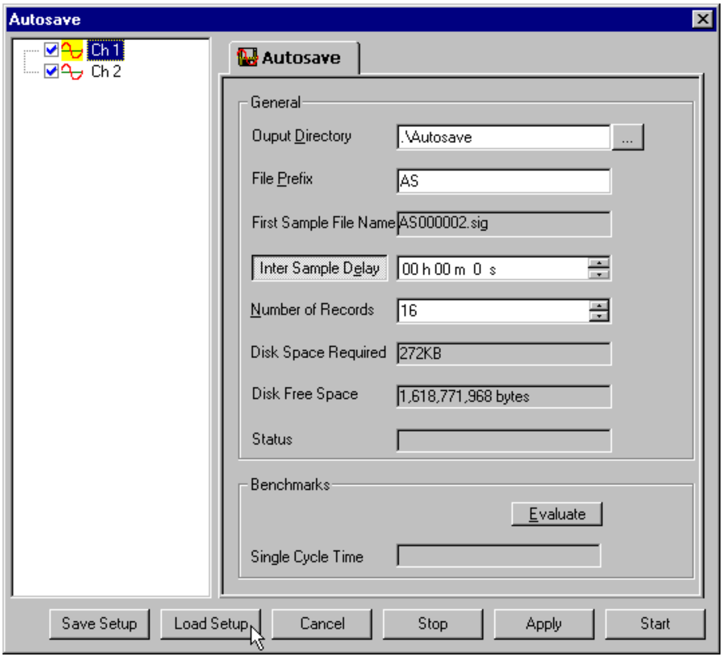
2 You will see a Save Setup dialog, allowing you to specify the location and name for the setup file.



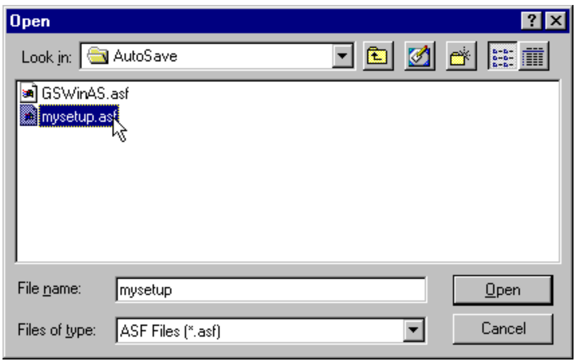
The default directory is **C:\Gage\Gagescope\AutoSave**, and the default AutoSave setup file name is **GagescopeAS.asf**. To save a new file, type in the File name field. Click Save.

Note: If you save more than one series of AutoSave files with the same prefix, GageScope® will create additional AutoSave directories underneath the main AutoSave directory. To prevent this, choose a different prefix, or save to another directory.

3 To load a previously saved AutoSave setup file, simply click **Load Setup** in the AutoSave Setup dialog.



4 This brings you to an Open dialog, allowing you to select an .asf file to load.



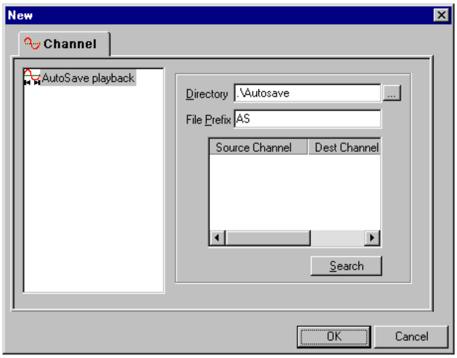
Click on the setup file you wish to load and then click **Open**.

AutoSave Playback

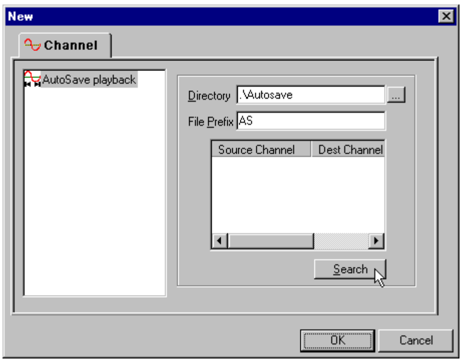
1 When an AutoSave session is executed, it can be played back by selecting the **AutoSave Playback** button on the main toolbar.

You can also select **AutoSave Playback** from the Tools Menu.

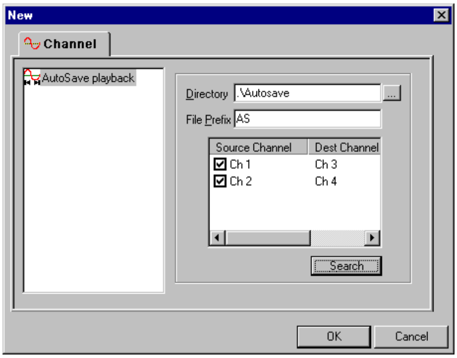
2 A dialog box then allows you to locate the AutoSave records on your system as well as the channels that should be played back.



3 Press the **Search** button to locate your AutoSave records.

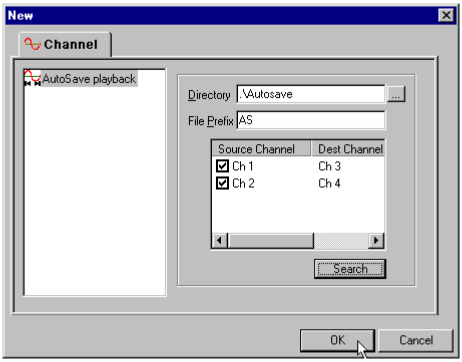


4 GageScope® will automatically look in the Gagescope\AutoSave directory for saved records. (If you saved in another directory, click ... to browse.) You will then see a list of channels to be played back, along with the destination channel, which will be used to display the AutoSave records on-screen.

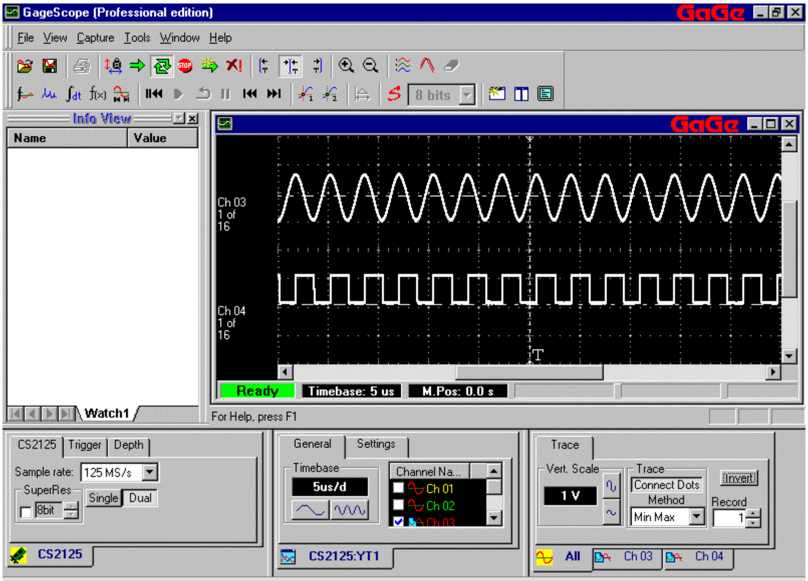


You can choose which channels will be displayed by clicking in the checkbox next to the channel in the list.

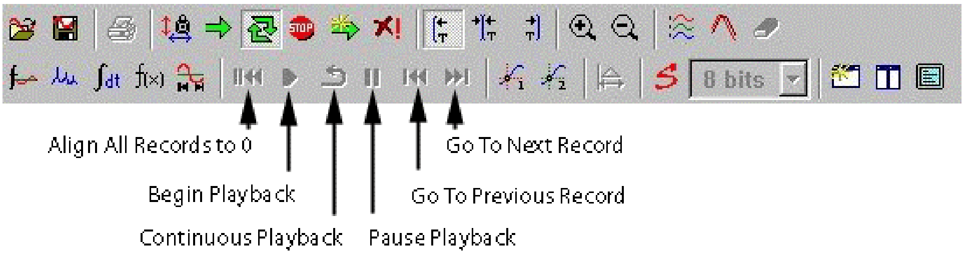
5 Now click **OK** to display the first of your AutoSave records.



6 The record number is displayed to the left of the signal, for example, **1 of 16** is shown in the screen below.



7 To move from record to record, use the Toolbar buttons shown below.

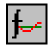


If you choose **Continuous Playback**, GageScope® will cycle through all of the AutoSave records and loop back to 1 for another cycle. You can see the record counter to the left of the signal, underneath the channel number (for example, 1 of 16, 2 of 16, 3 of 16…).

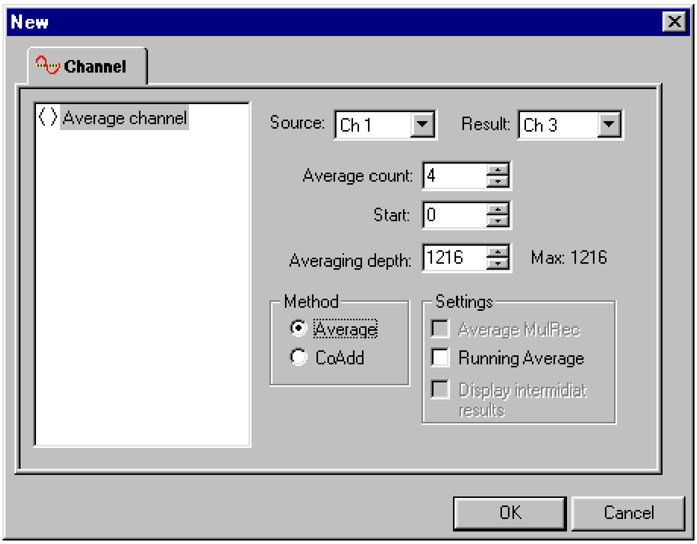
Averaging

This advanced tool allows the user to capture many records and average them for display. This data can also be saved to disk as averaged data. You can control how many acquisitions are performed before they are displayed, as well as how these acquisitions are combined into a single trace display.

Using the Averaging Tool

1 Click on the Tools menu, then click on **Averaged Channel** or, you can click on  in the toolbar.

2 When **Averaged Channel** is clicked from the drop-down menu, the following Channel dialog appears:



3 Choose a channel from the **Source** combo box. You can specify the following:

- **Average Count:** This is the number of acquisitions to be averaged.
- **Start:** It specifies the first sample of the averaged region relative to the trigger position. For a selected portion of the acquisitions to be averaged, the user has the option to select the starting point of the acquisition to be averaged.
- **Averaging Depth:** This is the length of the portion of the signal to be averaged. The maximum averaging depth, **Max** is 8192 in the GageScope® Professional Edition when the **running average** option is selected.

4 Select an averaging method from the **Method** menu. Users are provided with the following options:

- **Average:** The first point of the acquisition is added to the first point of the second acquisition and the process is repeated till the number of additions equal the number specified as the average count. This number is now divided by the average count and is then identified as the first point of the results channel. The process is repeated for the average length of the acquisition.

- **CoAdd:** When CoAdd is selected, all steps are followed as in the Average option, except that after adding the specified number of acquisitions, the result is stored as a sum and is not divided by the average count. By selecting the intermediate display option, one can view the results of the intermediate CoAdd steps.

5 Choose an item from the **Settings**

- **Running Average:** The N most recent acquisitions are added and then averaged. The result is stored in the results channel. One can notice that this result is continuously refreshed since the average is the running average.

  Note:

  1. *The running average is not available in the Standard Edition of GageScope®. **N cannot exceed 256** for the Professional Edition.*

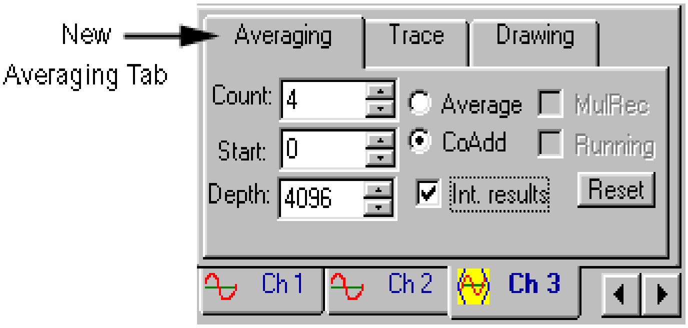
  2. *The **Maximum Averaging Depth is 8192** in the running average mode.*

- **Display Intermediate Results**: When acquisitions are co-added, the averaging tool allows the user to request the display of intermediate results. Note that this intermediate display may slow the co-adding process.

- **Average Multiple Record:** The user must have the multiple record activated to make use of this option. See the details of this option in the Multiple Record Section on page C-88.

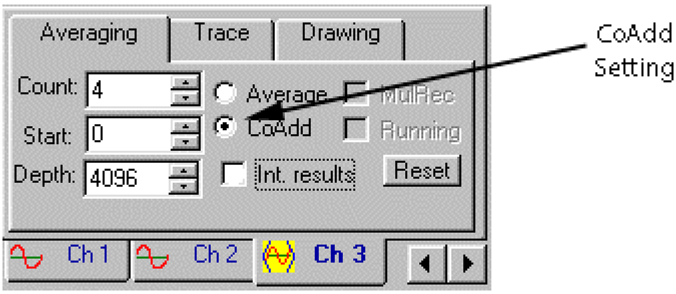
6 After entering the parameters for your application, Click **OK**.

Averaging appears up as a tab in the **Channel Properties** dialog box:

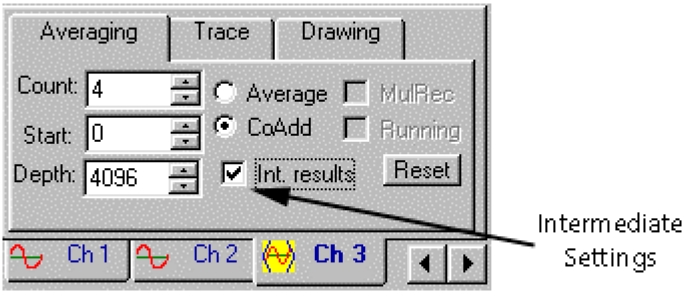


Details

1 The selected number of acquisitions can either be averaged or co-added. To **Average** the acquisitions, click the radio button next to Average. To **Co-add** the acquisitions, click the radio button next to CoAdd.

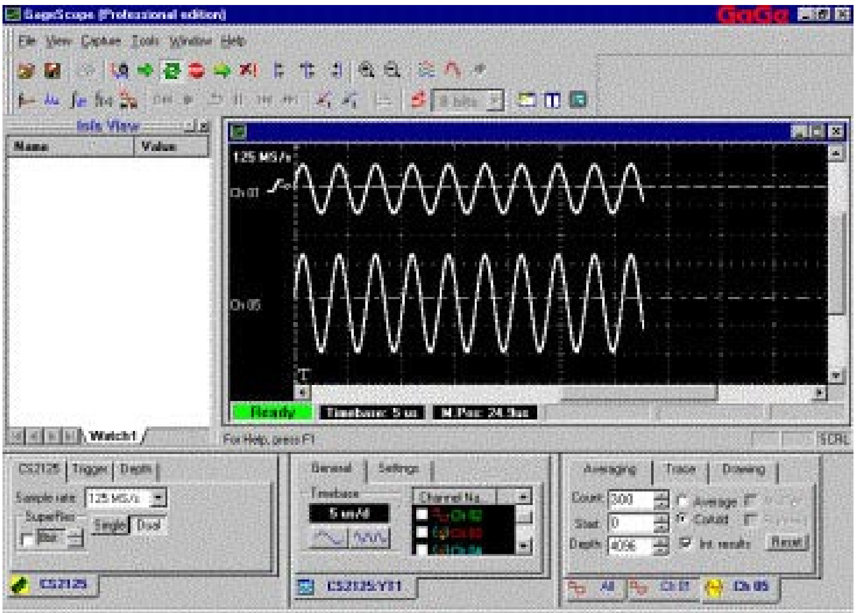


2 To show Intermediate Results, click the **Int. Results** check box. This means that GageScope® will display the results while it performs the adding process, rather than displaying just the final result.

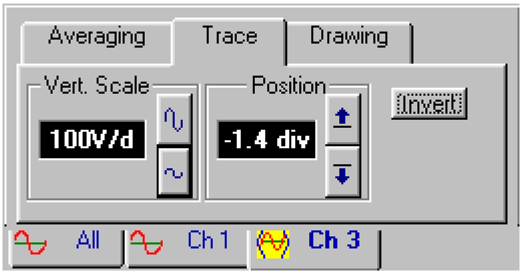


3 When the **Int. results** box is checked, the **Reset** button is available. This button allows you to set the counter back to 1.

The screen below shows a Co-add with 300 acquisitions, showing intermediate results



4 Click the **Trace** tab in the Channel Properties dialog. Note that the peak-to-peak (p-p) amplitude of the CoAdded Channel 3 is 300 volts. The signal on channel 1 was originally 1 Volt p-p.

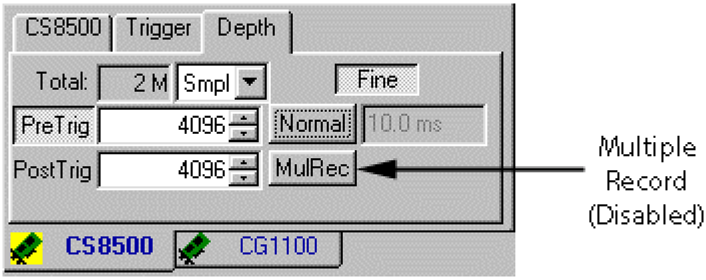


Averaging in Multiple Record Mode

When in Multiple Record mode, averaging can be performed in three ways.

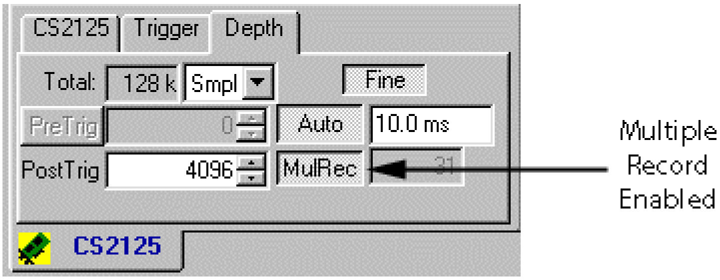
- A single “MulRec” record can be used to produce a single averaged MulRec record. This method is best when the incoming signals are non-repetitive, such as in Imaging applications.
- A specified number of MulRec records can be averaged to produce a single averaged record. This method is best for applications like Mass Spectrometry, where the signals are highly repetitive.
- The running average of a single “MulRec” record.

1 To perform averaging in Multiple Record mode, a Multiple Record acquisition must first be performed. The **MulRec** item is available only if **MulRec** is enabled in the Depth tab of the System Control, as shown below:



For more details on Multiple Record, see page C-88.

2 In this Multiple Record acquisition, we will acquire 4096 samples. GageScope® calculates that we can acquire 31 records, each containing 4096 samples, given the amount of memory on-board our CS2125 card.



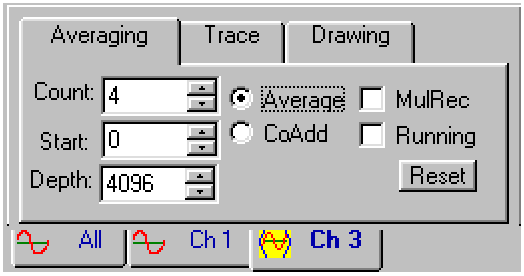
Averaging when MulRec is Not Checked

1 MulRec appears at two places:

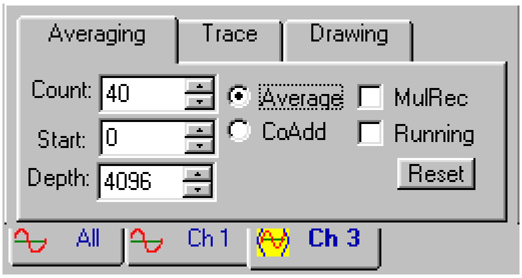
- In the System Control, MulRec appears under the Depth tab.
- In the Channel Control, MulRec appears on the Trace tab once Multiple Record Option is selected in the System Control and an acquisition has been made.

It is important to remember that we have set the Multiple Record Mode by selecting the MulRec button in the System Control and now referring to the MulRec under the Channel Control.

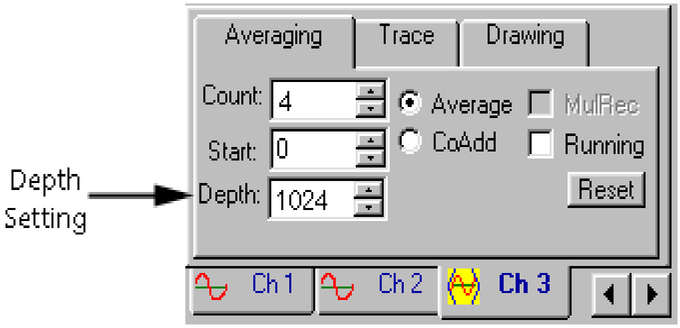
By default, the **MulRec** item is unchecked in the Averaging Mode. In this mode, GageScope® takes a MulRec record and averages it over a specified number of acquisitions to produce an averaged MulRec record.



2 For instance, if we set the **Count** to **40** via the Increment buttons, GageScope® will acquire the current records or their portion forty times and average the records in each of the forty acquisitions to produce an averaged MulRec record.

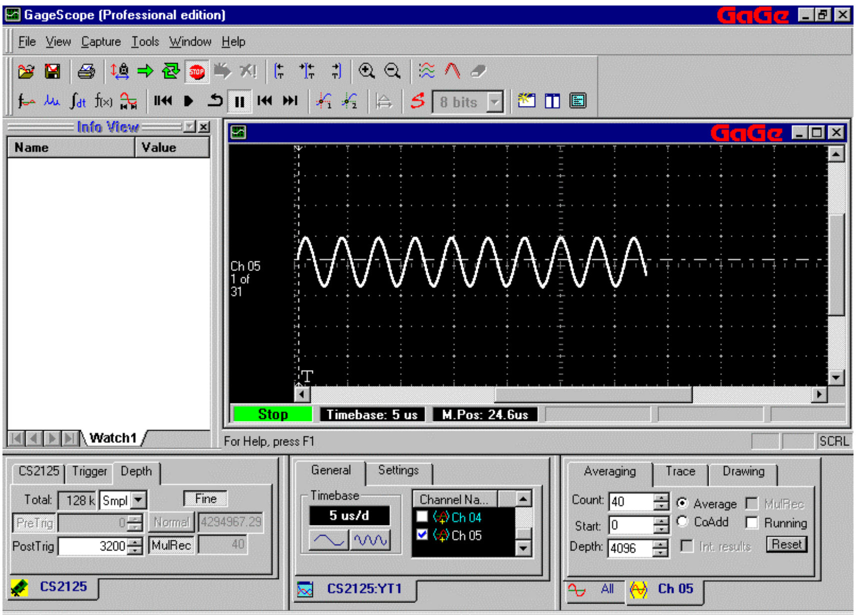


3 In this case, although the multiple record depth was specified to be 4096 samples, we have changed the averaging depth to be the first 1024 samples.

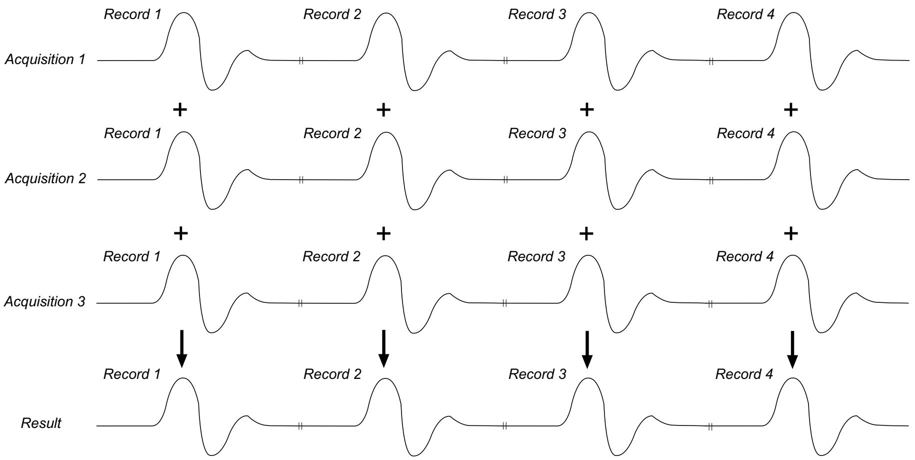


Note that by entering an integer number in the Start field and the averaging depth, we can select any arbitrary portion of the acquisition to be averaged.

4 Change the count to 40. The result of the 40 averages for the selected portion of the acquisition is shown as

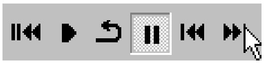


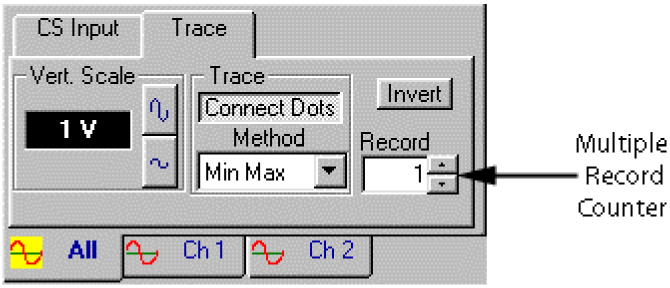
This is shown diagrammatically below. A single MulRec record is averaged over **n** acquisitions to produce an averaged MulRec record.



Result, Record 1 = Acquisition 1, Record 1 + Acquisition 2, Record 1 + Acquisition 3, Record 1 ... + Acquisition n, Record 1

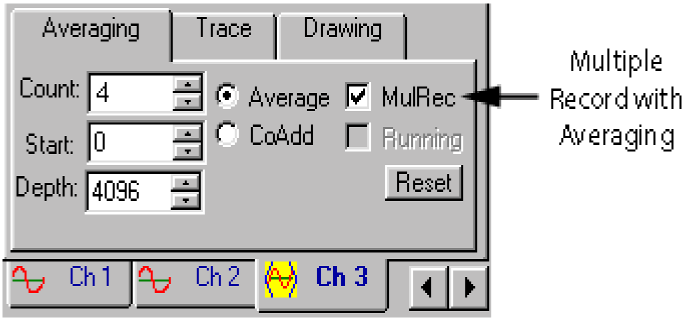
You can specify which resultant record is averaged and displayed via the **Record Counter** in the Trace Tab of the Channel Control, or the **Multiple Record Playback** buttons in the toolbar.



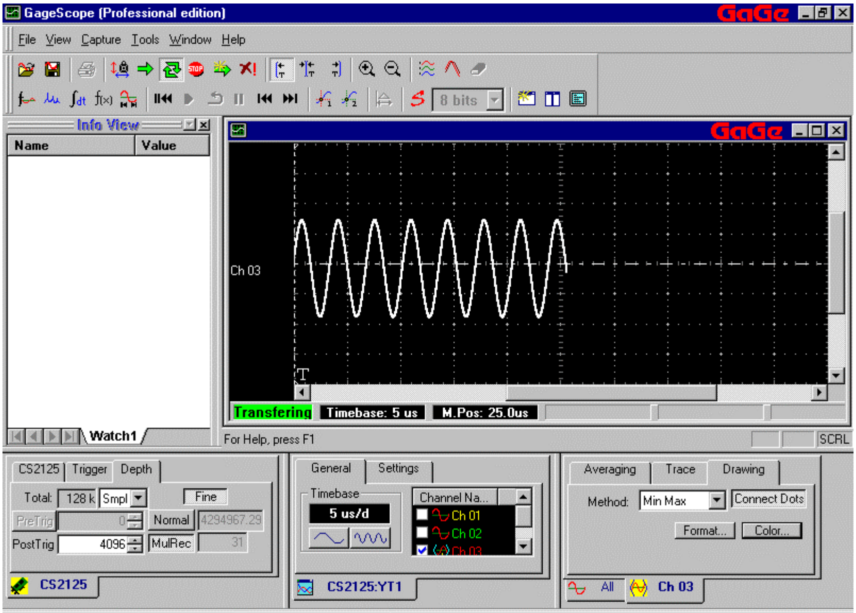


Averaging when Mulrec is Checked

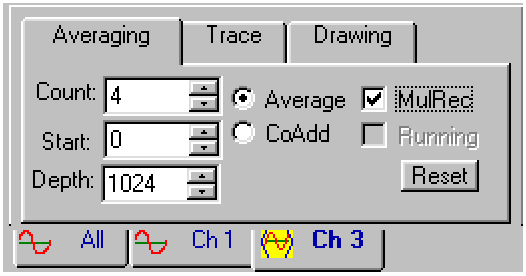
1 Now we will perform the second type of averaging possible in Multiple Record mode. Click the **MulRec** checkbox in the Averaging tab.



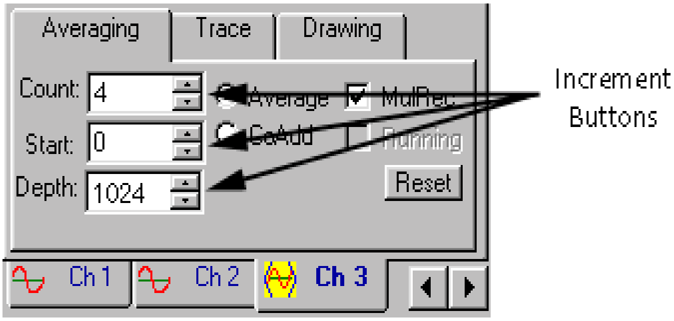
2 In this mode, GageScope® averages a specified number of records in one Multiple Record acquisition to produce a single averaged result.



3 Notice that the Averaging **Depth** has been changed to 1024

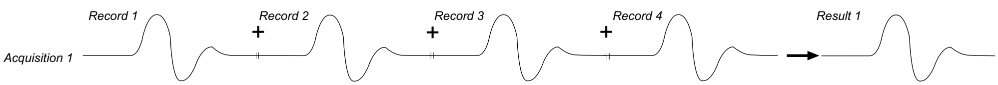


4 The number of records to be averaged is specified in the **Count**. This field can be set to any number from 2 up to the total number of records in the Multiple Record acquisition. In our example, our Multiple Record acquisition contains 31 records, so we can average anywhere from 2 to 31 records to produce a single resultant averaged record.



If you attempt to set the counter to a number higher than the current number of records in your acquisition, GageScope® will automatically set the field to the maximum number allowed.

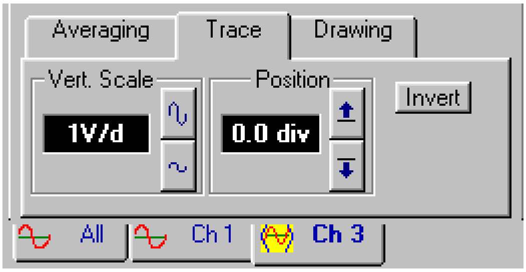
This is shown diagrammatically below. GageScope® averages **n** records in the current acquisition to produce a final result, which is displayed in the Display Window.



Result 1 = Acquisition 1, Record 1 + Acquisition 1, Record 2 + Acquisition 1, Record 3 ... + Acquisition 1, Record n

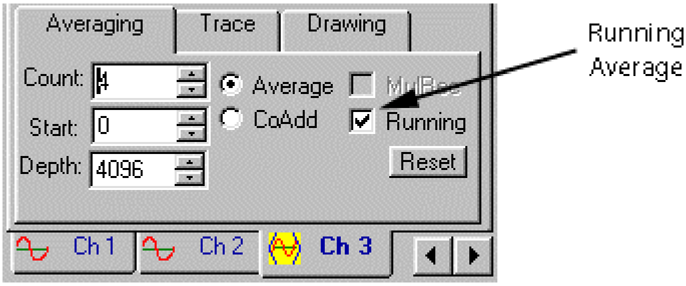
Note

When **Mulrec** is checked, the Record Counter disappears from the Trace tab of the Channel Control, after the averaging operation is completed.

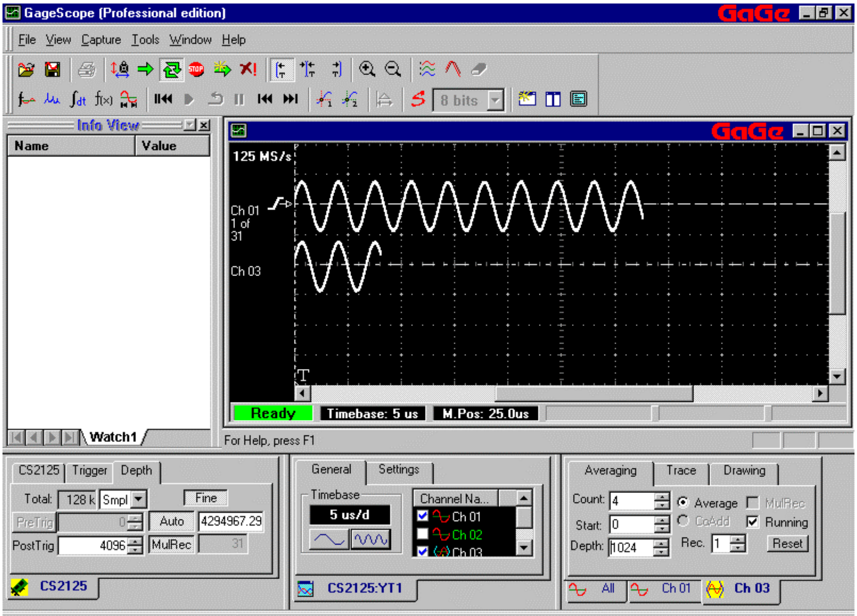


Averaging when Running is Checked

1 When **Running** is checked, the running average of the selected multiple record is taken.



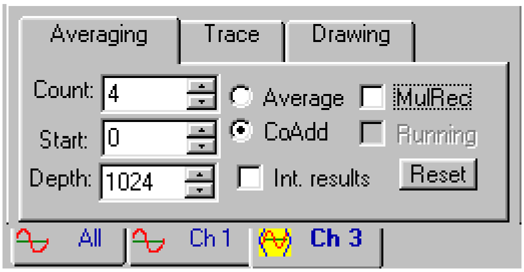
2 The result of the running average of the first multiple record is shown as



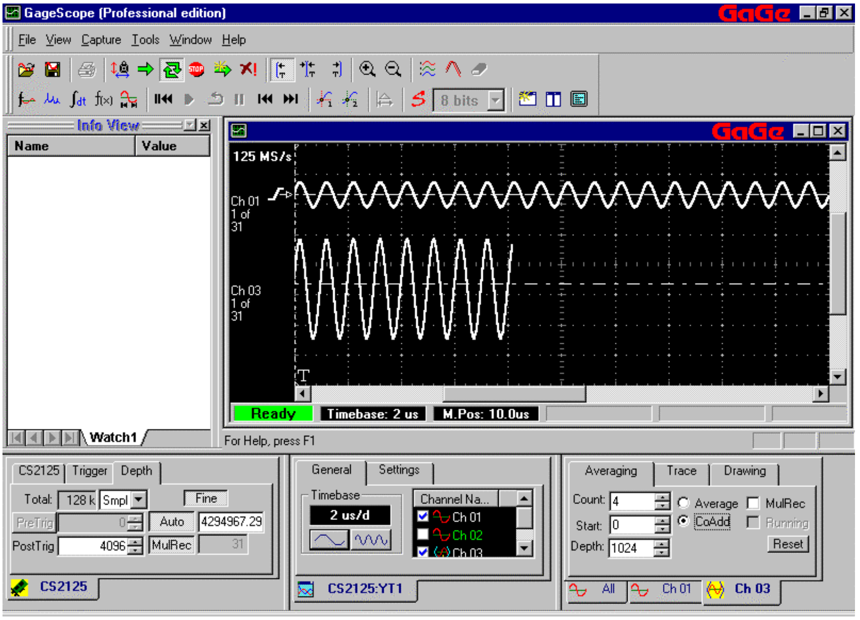
Note that although the record length is 4096, yet the length of the average channel is being chosen as 1024.

CoAdding Multiple Record Acquisitions

1 A single multiple record acquisition can be CoAdded a specific number of times. Select the CoAdd radio button to activate this feature while in the Multiple Record Mode. Remember that Multiple Record Mode is selected via the System Control.



2 A single multiple record on channel 1 is being CoAdded 4 times.



Note

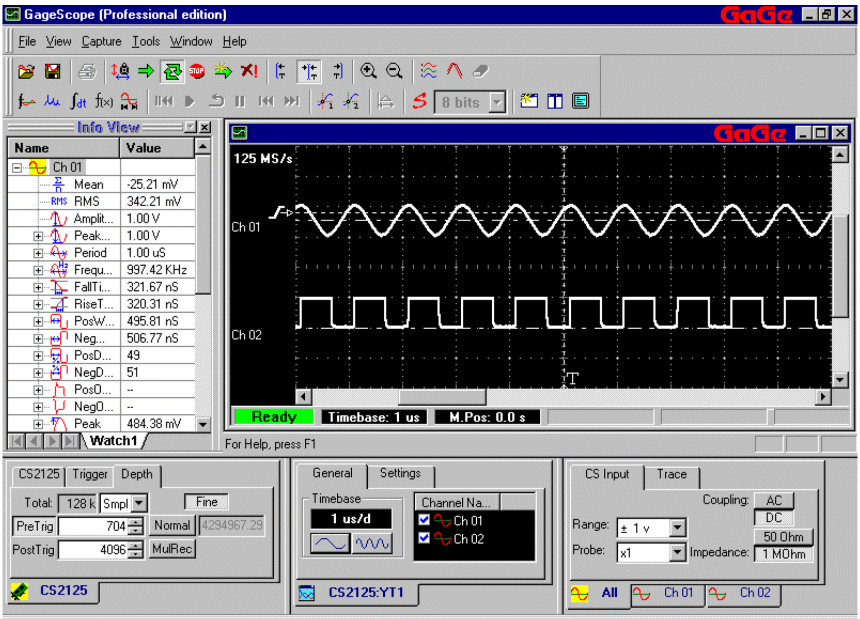
- Record 1, out of the 31 records, is a 1MHz sine wave with a record length of 4096 post trigger points
- The peak-to-peak amplitude of the sine wave is 1 Volt
- The length of the CoAdded signal is chosen to be 1024.
- The peak-to-peak amplitude of the CoAdded result shown as Channel 3 is 4 Volts

Waveform Parameters

The Waveform Parameters tool automatically calculates various voltage and time parameters of a signal, such as mean, amplitude, and rise time. In all, 28 parameters are available:

- Mean
- RMS
- Amplitude
- Peak to Peak
- Period
- Frequency
- Fall Time
- Rise Time
- Positive Width
- Negative Width
- Positive Duty
- Negative Duty
- Positive Overshoot
- Negative Overshoot
- Peak
- Trough
- Top Value
- Bottom Value
- TAAPos
- TAANeg
- TAA
- Pk-Pk time
- Pk-Thr time
- Tr-Pk time
- Tr-Thr time
- Ext-Ext time
- Base line
- Delta Base line

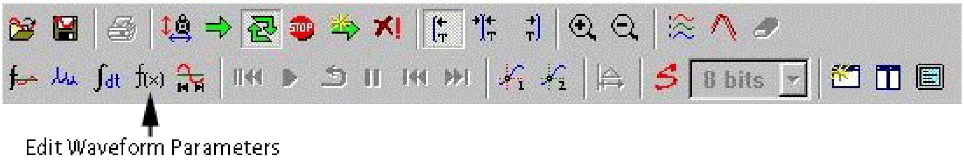
Parameters are displayed in the InfoView window at the far left of the GageScope® screen.



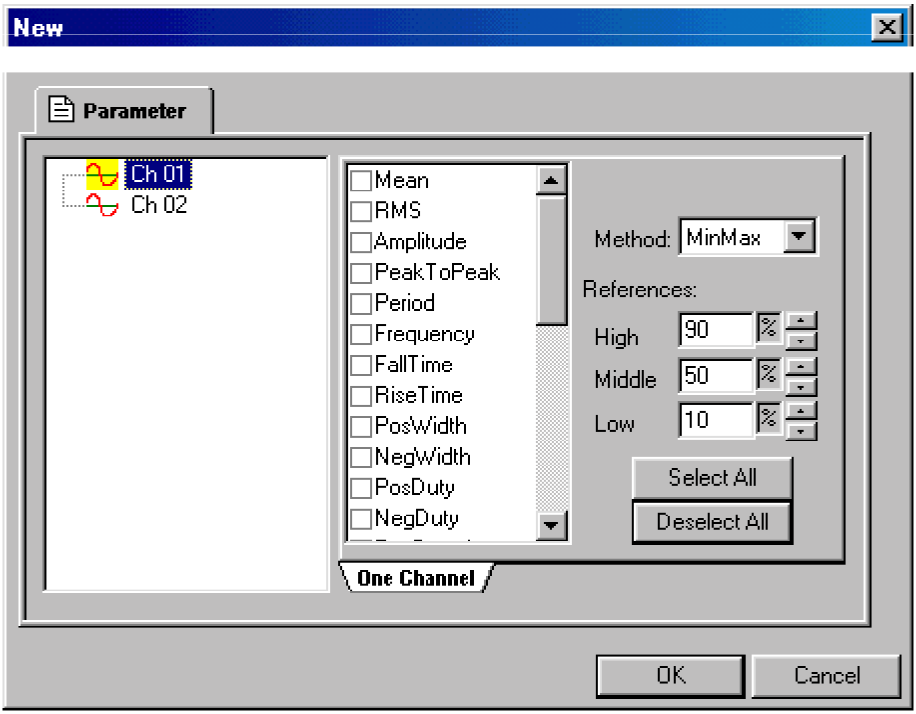
For definitions of what each parameter measures, see **About the Parameters** on page C-212.

Adding Parameters to the InfoView

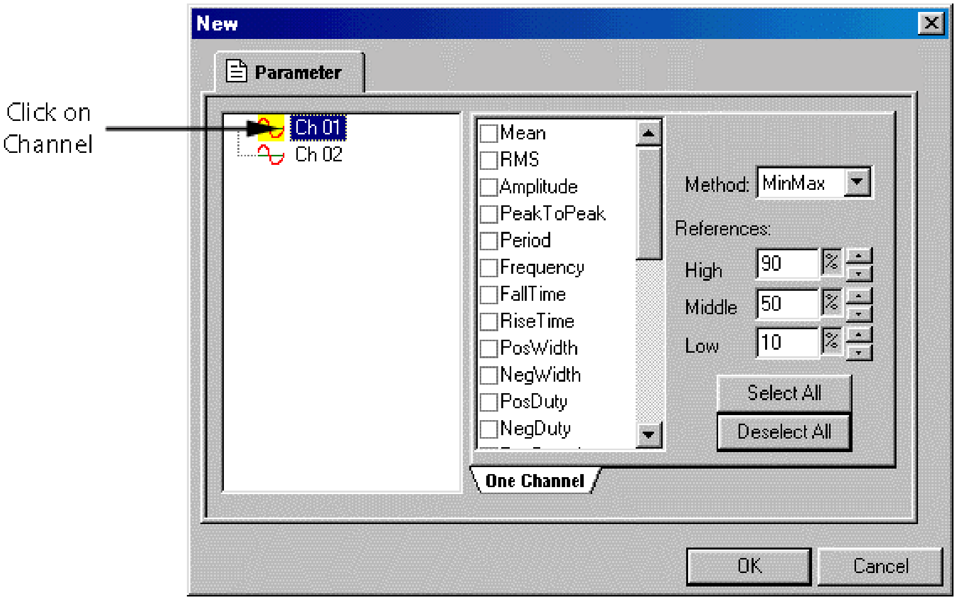
1 To display a parameter, right-click on the InfoView window and choose **Edit Waveform Parameters**. You can also choose Waveform Parameters from the Tools menu or the  button in the Toolbar.



The Parameters dialog box appears.

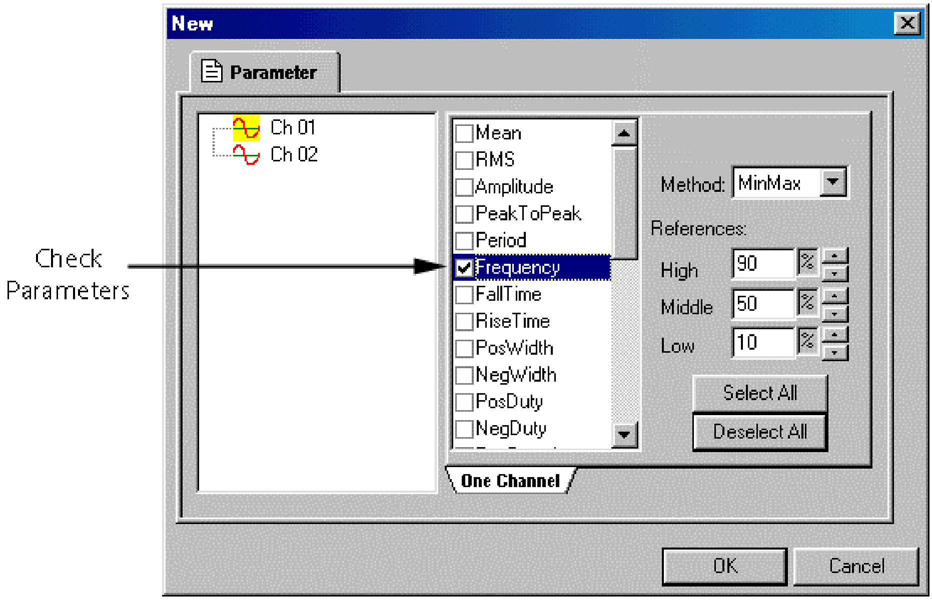


2 Click on the channel number you want to measure.



3 Click on the checkbox of each parameter you want to display in the InfoView window. You may have to scroll down the list to find the parameter you want.

A checkmark indicates the parameter will be shown; the lack of a checkmark indicates the parameter will not appear.



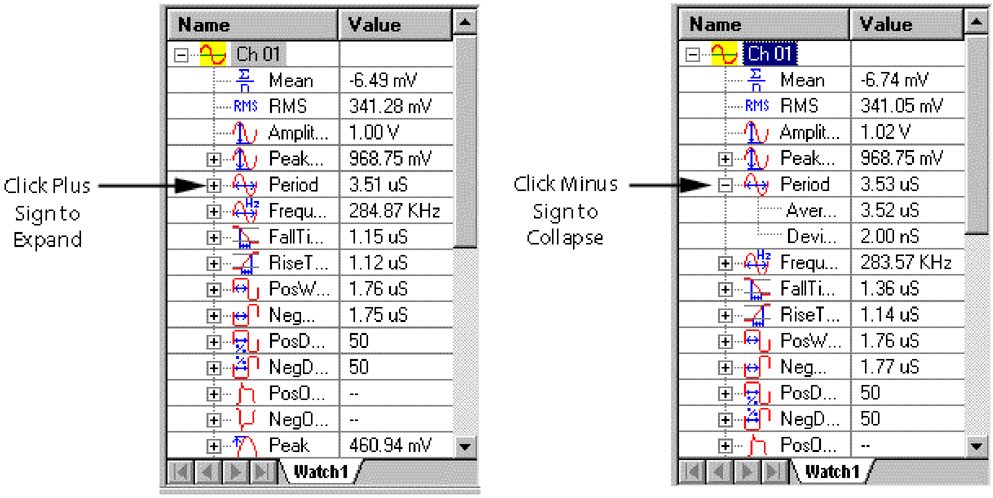
To enable all parameters, click on the **Select All** button. To disable all parameters, click on the **Deselect All** button.

4 Click **OK** when finished or **Cancel** to abort the changes.

Once you click **OK**, the parameters appear in the InfoView window.

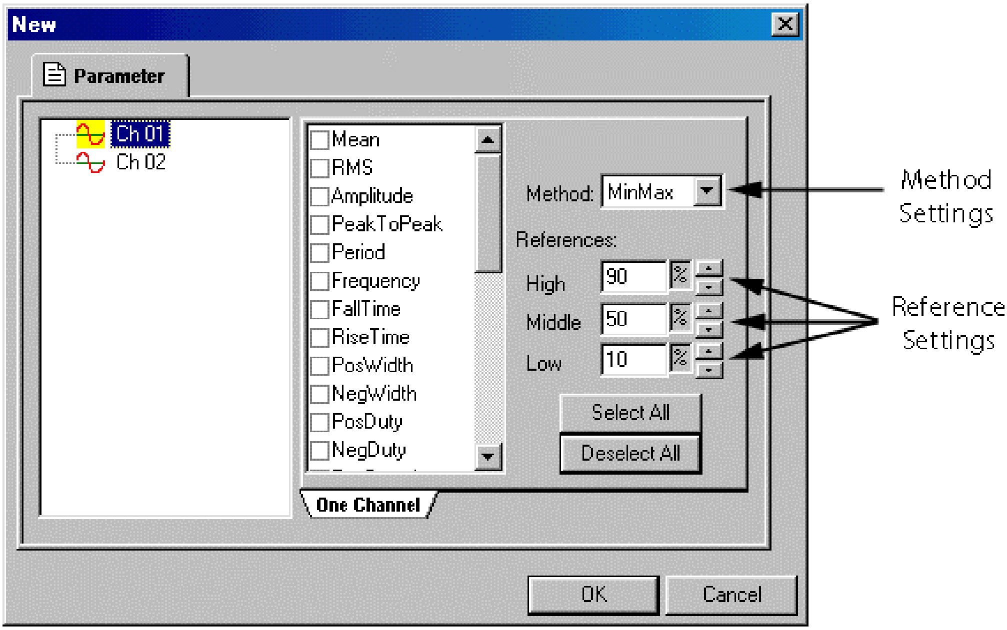
Expanding and Collapsing Parameters

Many of the parameters can display the average value and standard deviation. Such parameters have a plus sign to the left of their name in the list. To expand a parameter and see the average values and standard deviations, click on the plus sign to the left of a parameter. To collapse the parameter back to its original state, click on the minus sign.



Method and Reference Settings

When calculating certain parameters, the Waveform Parameters tool relies on the **Reference** and **Method** settings located in the Parameters dialog.



For information on how particular parameters are affected by these settings, see **About the Parameters** on page C-212. Not all parameters make use of these settings.

The Method Setting

The options for Method are **MinMax** and **Histogram**. To change the **Method** setting, select a value from the drop-down menu. The default is **MinMax**.

The **Histogram** Method is preferred for pulse-based signals such as digital logic (CMOS, TTL, ECL, etc.).

The **MinMax** Method is preferred for other signals, such as sine waves.

The Reference Settings

The reference settings **High**, **Middle** and **Low** are used to determine the High Level, Middle Level, and Low Level when calculating certain time parameters. For example, the **Rise Time** is the length of time for a signal's rising edge to go from the low level to the high level.

To change the **Reference** settings, click on the Increment buttons next to High, Middle and Low.

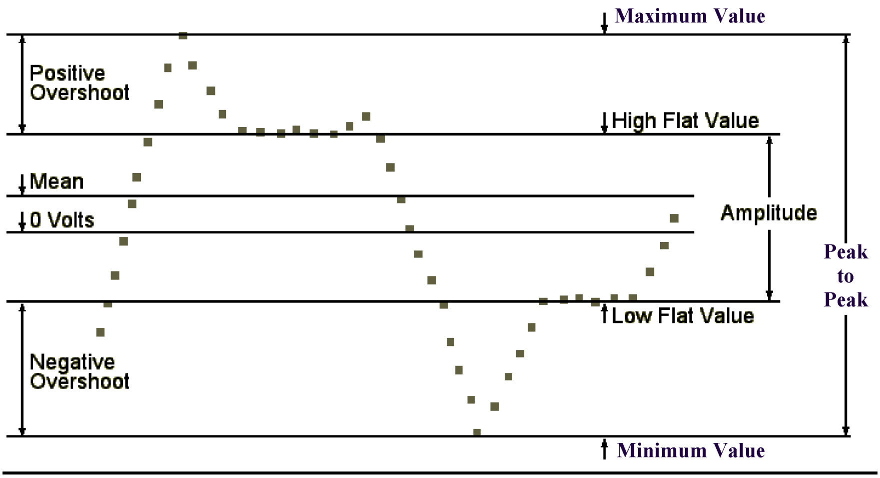
The definitions for High Level, Middle Level, and Low Level are shown below. Reference:High%, Reference:Middle%, Reference:Low% refer to the values in the Reference settings in the Parameters dialog box.

High Level = LowFlatValue + (HighFlatValue - LowFlatValue) x Reference:High%
Middle Level = LowFlatValue + (HighFlatValue - LowFlatValue) x Reference:Middle%
Low Level = LowFlatValue + (HighFlatValue - LowFlatValue) x Reference:Low%

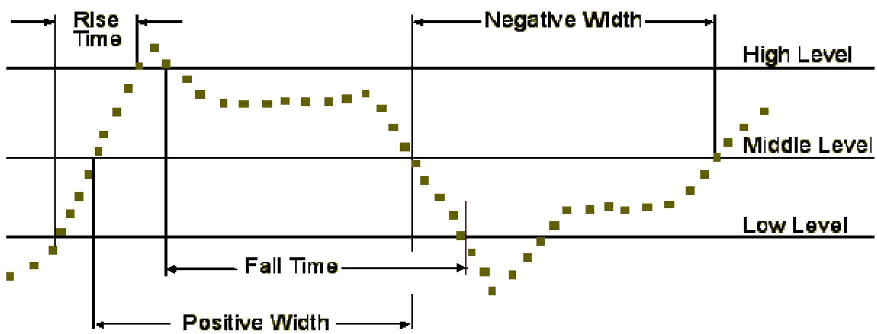
About the Parameters

Use the diagrams below to get an idea of what each parameter measures. More detailed explanations are offered on the pages following the diagram.

Voltage Parameters



Time Parameters



The parameters are defined as follows:

Mean	Mean is the average of all the points in the signal.
RMS	RMS is the Root Mean Square voltage.
Amplitude	When the **MinMax** method is chosen, Amplitude is equal to the **Peak to Peak** value. When the **Histogram** method is chosen, the Amplitude is the difference between the **High Flat** and **Low Flat** values, in Volts.
PeakToPeak	Peak-to-Peak is the absolute difference between the **Maximum** and **Minimum** values, in Volts.
Period	Period is the time it takes for a signal to complete one cycle and is equal to the sum of the **Positive** and **Negative Widths**.
Frequency	Frequency is the number of cycles per second in a signal, measured in Hz, with 1 Hz equaling 1 cycle per second, and is equal to **1 / Period**.
FallTime	Fall Time is the length of time for a signal's falling edge to go from the **High Level** to **Low Level**. (These two levels are specified in the Reference:High and Reference:Low settings in the Parameters dialog box.)
RiseTime	Rise is the length of time for a signal's rising edge to go from **Low Level** to **High Level**. (These two levels are specified in the Reference:High and Reference:Low settings in the Parameters dialog box.)
PosWidth	Positive Width is the length of time between the rising and falling edge of the portion of the signal above the **Middle Level**. (This level is specified in the Reference:Middle setting in the Parameters dialog box.)
NegWidth	Negative Width is the length of time between the falling and rising edge of the portion of the signal below the **Middle Level**. (This level is specified in the Reference:Middle setting in the Parameters dialog box.)
PosDuty	Positive Duty is a percentage of the time it takes for the positive portion of a signal to complete, compared to one whole cycle. It is defined as:

(Positive Width ÷ Period) x 100

NegDuty Negative Duty is a percentage of the time it takes for the negative portion of a signal to complete, compared to one whole cycle. It is defined as:

(Negative Width ÷ Period) x 100

PosOvershoot Positive Overshoot is measured in Volts and is defined as:

Maximum Value - High Flat Value

When the method is set to MinMax, Positive Overshoot is 0.

NegOvershoot Negative Overshoot is measured in Volts and is defined as:

Minimum Value - Low Flat Value

When the method is set to MinMax, Negative Overshoot is 0.

Top Top is the average of the highest values recorded for all the cycles in the signal.

Bottom Bottom is the average of the lowest values recorded for all the cycles in the signal.

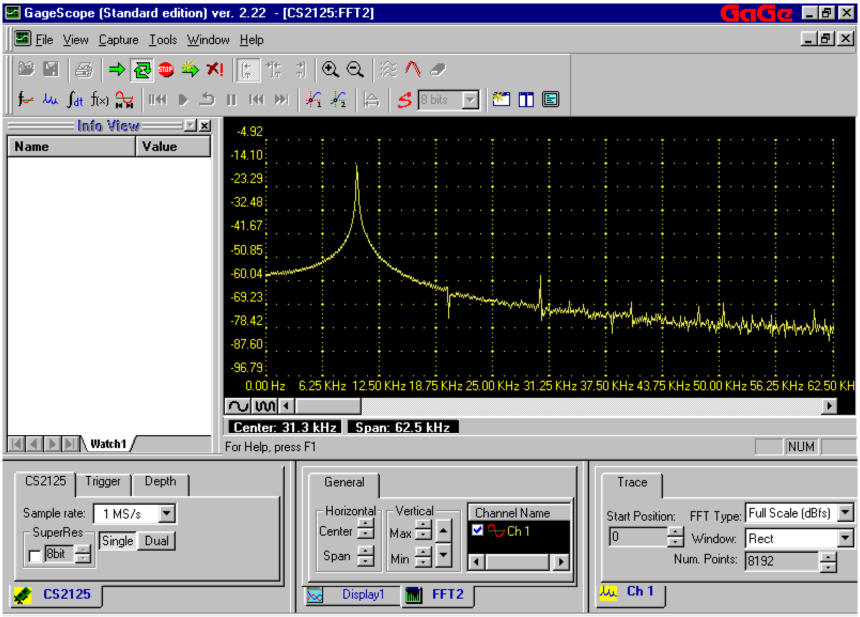
Minimum Minimum is the lowest voltage recorded for the signal.

Maximum Maximum is the highest voltage recorded for the signal.

NumCycles Number of cycles is the total number of cycles in the signal.

FFT Analysis

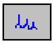
The FFT Analysis tool for GageScope® provides multiple-channel spectral analysis with simultaneous viewing of time and frequency domain data, all within the easy-to-use framework of GageScope®.

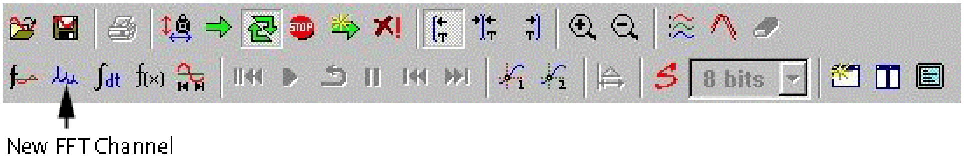


Notes

- You must set your screen to 256 colors (minimum) in order to use the FFT Analysis tool, otherwise your screen display may be garbled.

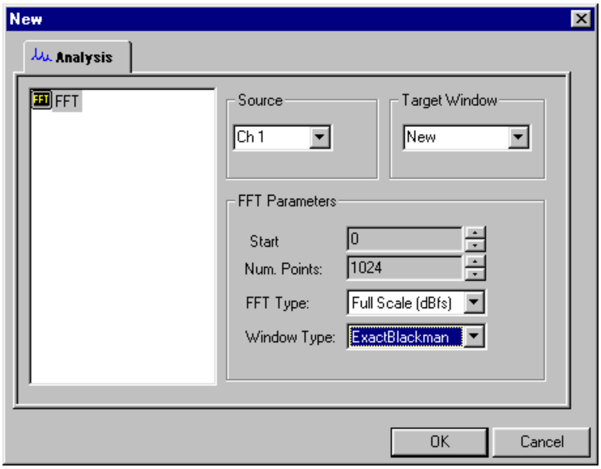
Using the FFT Analysis Tool

1 To initiate an analysis of a signal, select the  button in the Toolbar.



You can also select **FFT…** from the Tools menu.

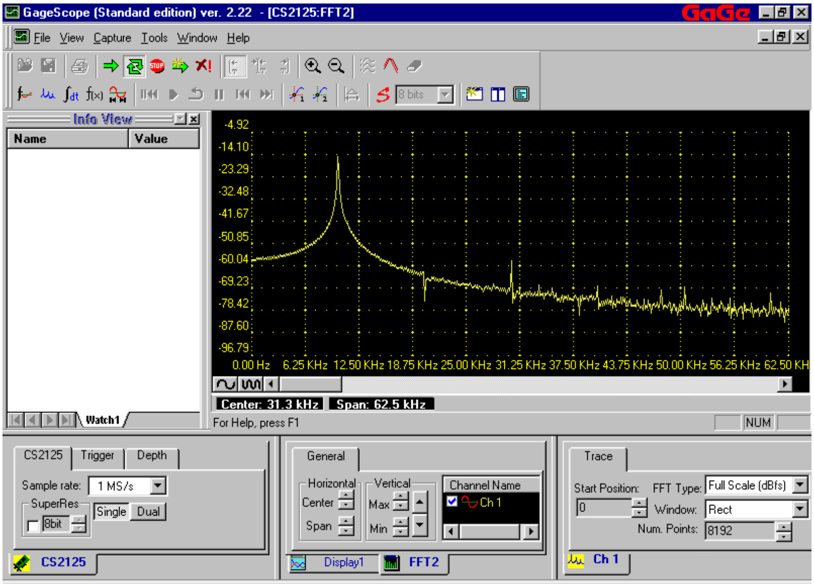
2 The FFT Dialog will appear.



3 Choose from the available options that appear at the right.

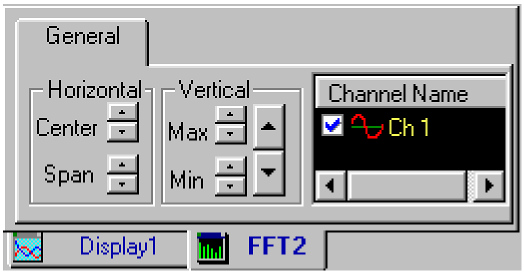
Source:	The channel the FFT analysis will be performed on.
Target Window:	The window that FFT analysis will be displayed in.
Start:	The starting sample point for the analysis. 0 means the point at which a trigger occurred and capture began.
Num. Points:	The number of points to use in the analysis. The maximum number of points supported in the Professional Edition of GageScope® is **4 Million**.
FFT Type:	Choice of FFT Type includes Full Scale (dBfs), 1 milliwatt (dBm) and 1 Volt (dBV).
Window Type:	Choice of window type includes Rect, Parzen, Hanning, Welch, Hamming, ExactBlackman and BlackmanHarris.

4 Click **OK** when ready. A new window opens with the FFT on display.

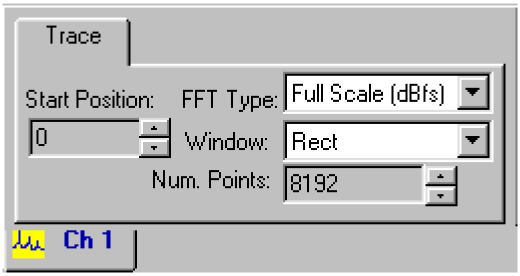


Once you have created an FFT window, the Display and Channel Controls change to offer new options.

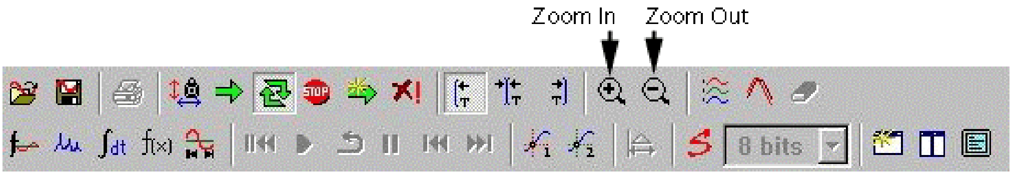
Specifically, a new tab appears at the bottom of the Display Control called **FFT2**. This tab allows you to switch from the normal signal window to the FFT window. Several new options appear here as well. The **Center** and **Span** settings in the Combo box titled **Horizontal** allow you to change the horizontal scale of the FFT Display Window.The **Max** and **Min** settings in the Combo box titled **Vertical** allow you to change the vertical scale of the FFT Display Window. The **Channel Name** is used to display/hide the FFT corresponding to a particular Channel.



If the FFT Display Window is being shown, the Channel Control changes to display all the settings you specified in the FFT dialog. You can change the settings in this control at any time in order to change the FFT Display.



You can also zoom in on a specific portion of the FFT by using the toolbar's Zoom Tools.



Cursors can also be used to make precise readings of frequencies and power. Upon activation of a cursor, you can immediately read frequencies, but to get the dB level, you must first attach the cursor to the FFT trace. This is done by right-clicking on the cursor and selecting the channel to which the cursor is to be attached.

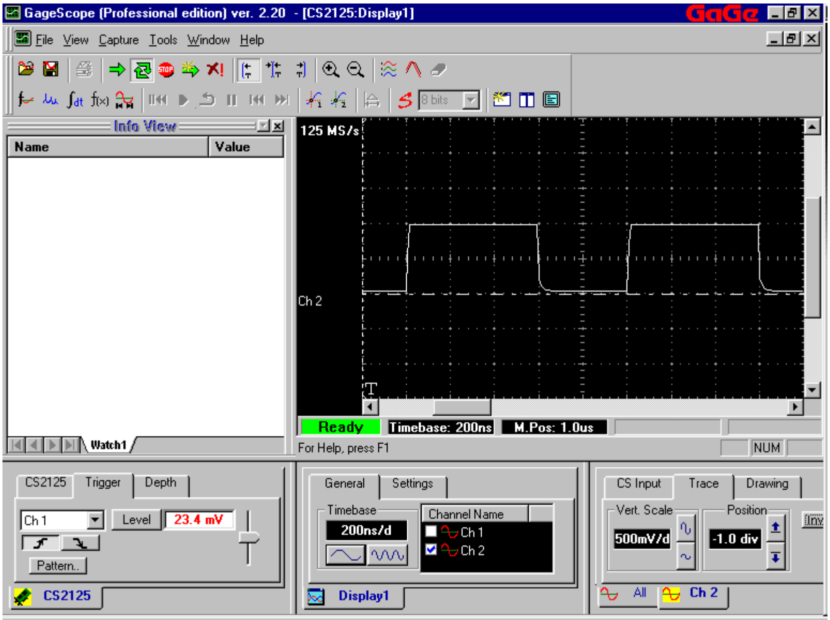
Extended Math

The Extended Math tool gives GageScope® the ability to perform the following operations on the acquired signals:

- Integration
- Differentiation
- Autocorrelation
- Crosscorrelation

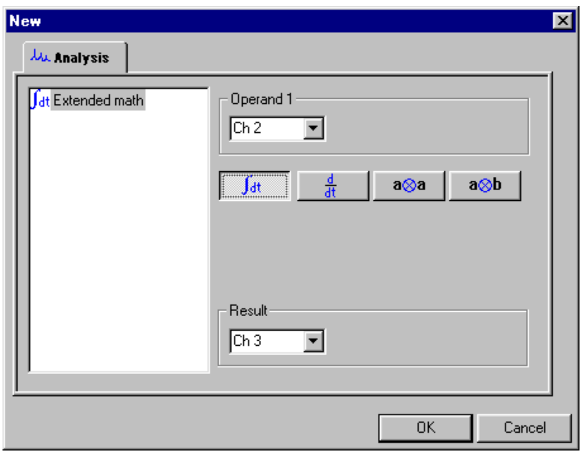
Integration

1 The signal of interest is as follows:



The acquisition is a series of output pulses with of 1MHz frequency from a frequency generator. We will perform the integration operation on this channel.

2 From the **Tools** menu, select **Extended Math** or select  from the toolbar. The New Analysis Dialog box will appear

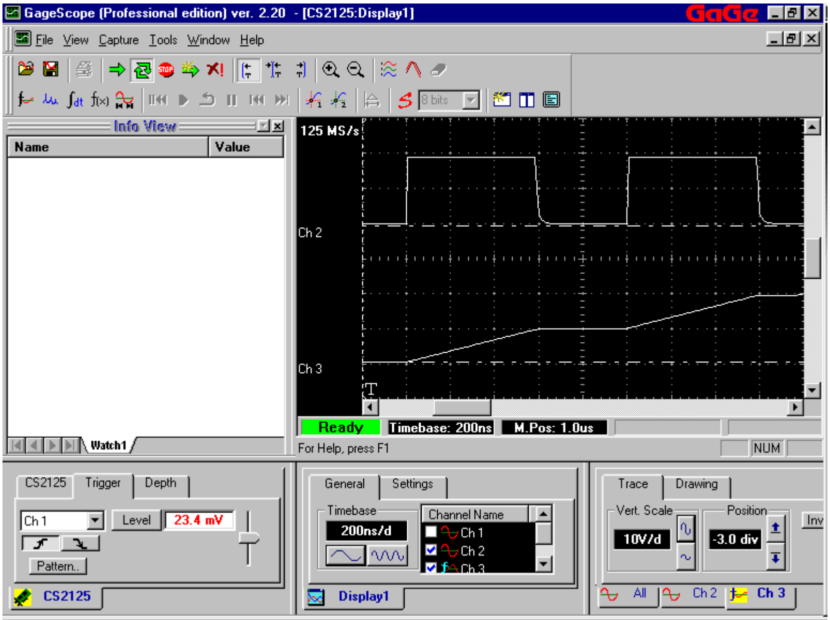


3 Select Channel 2 from the **Operand 1** drop down menu.

4 Click on the  button.

5 Select channel 3 for the **Result** drop down menu.

6 Click **OK**. The result of the integration appears will appear as

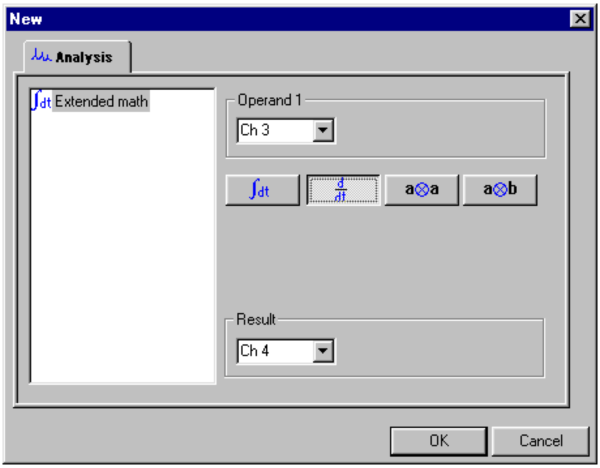


The result of integrating a constant amplitude function is a linear function. Note that the scaling of the integration operation is not preserved. Current analysis will be useful in applications where the relative and not the absolute value of the integration is desired.

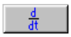
Differentiation

1 The signal of interest in this case will be the integrated signal shown as Channel 3 in step 6.

2 From the **Tools** menu, select **Extended Math** or or select  from the toolbar. The New Analysis Dialog box will appear

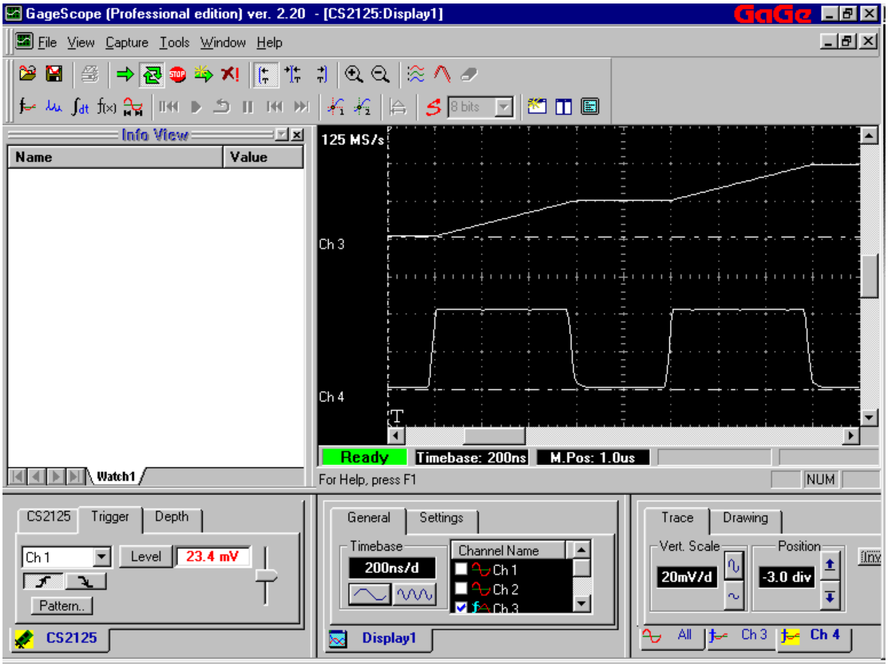


3 Select Channel 3 from the **Operand 1** drop down menu.

4 Click on the  tab.

5 Select channel 4 for the **Result** drop down menu.

6 Click OK. The result appears as



The result of differentiating an linear function should be a constant amplitude function. The scaling is however not preserved.

Autocorrelation

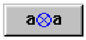
The Correlation function shows

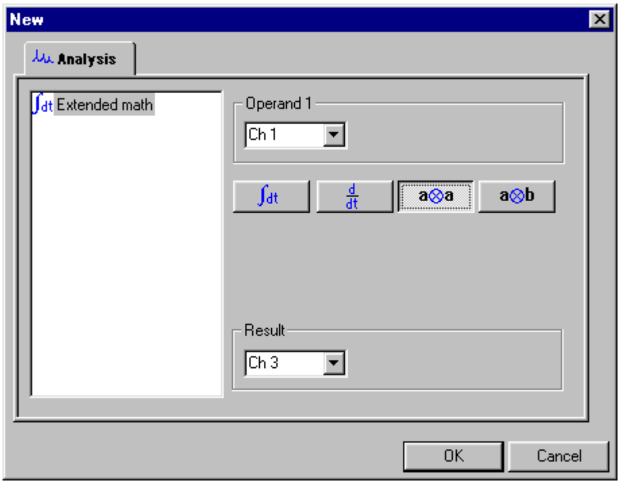
- How similar two functions are
- How long these functions remain similar when one is shifted with respect to the other

Correlating a signal with itself is called Autocorrelation. Autocorrelation is a fundamental analysis tool in applications such as ultrasonics, vibrations, disk drive testing, communications, etc.

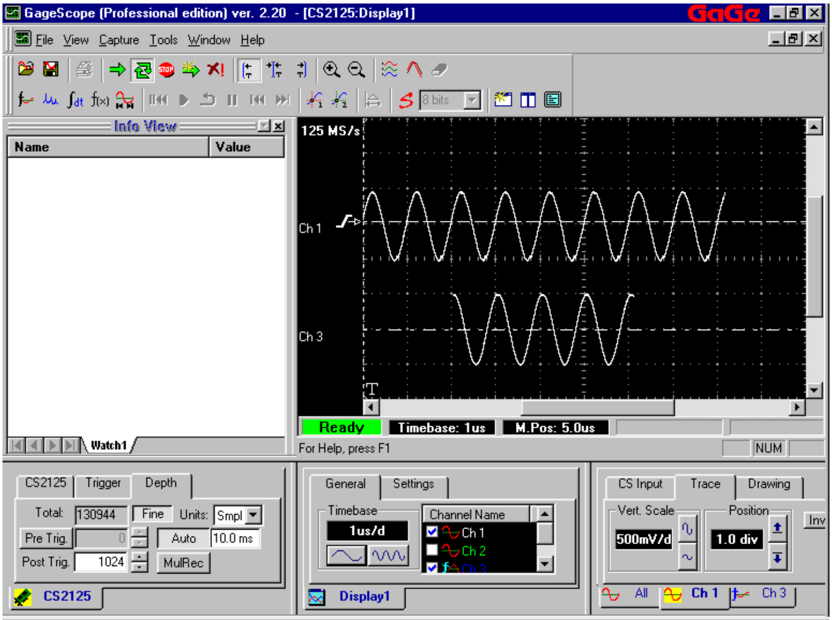
1 From the **Tools** menu, select **Extended Math** or select 

from the toolbar. The New Analysis Dialog box will appear.

Select **Operand 1.** Click the autocorrelation button 



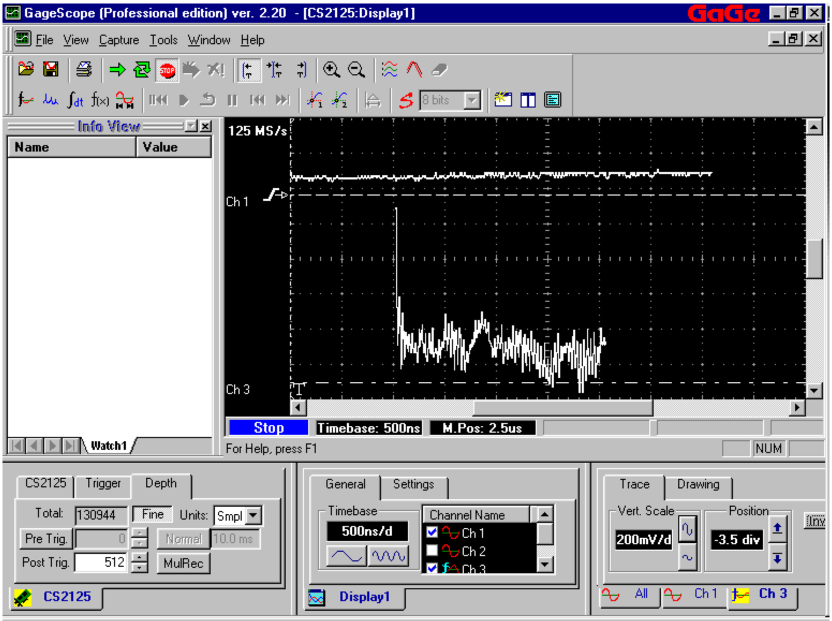
2 Choose a channel for the **Result**



Channel 1 is a 1024 point sine wave. The autocorrelation function is shown as Channel 3. In taking the autocorrelation of the acquired signal, the acquired signal was divided into half. This half portion of the signal was then slided on the original signal. The entire length of the slided portion was contained in the original signal at all times.

Note that phase information is not preserved under this operation.

3 The autocorrelation of random noise appears as follows:



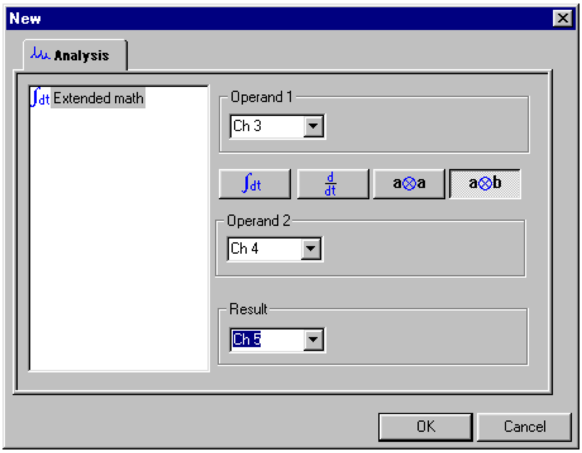
Notice the huge spike which seems like a dirac delta function at the start of the result in channel 3.

Crosscorrelation

It is well known in the test and measurement world that the response of a system can be characterized by subjecting the system to a random excitation. The concept of Crosscorrelation is a useful analysis tool in such applications.

1 From the **Tools** menu, select **Extended Math** or select  from the toolbar. The New Analysis Dialog box will appear.

Select **Operand 1.** Click the Crosscorrelation button 

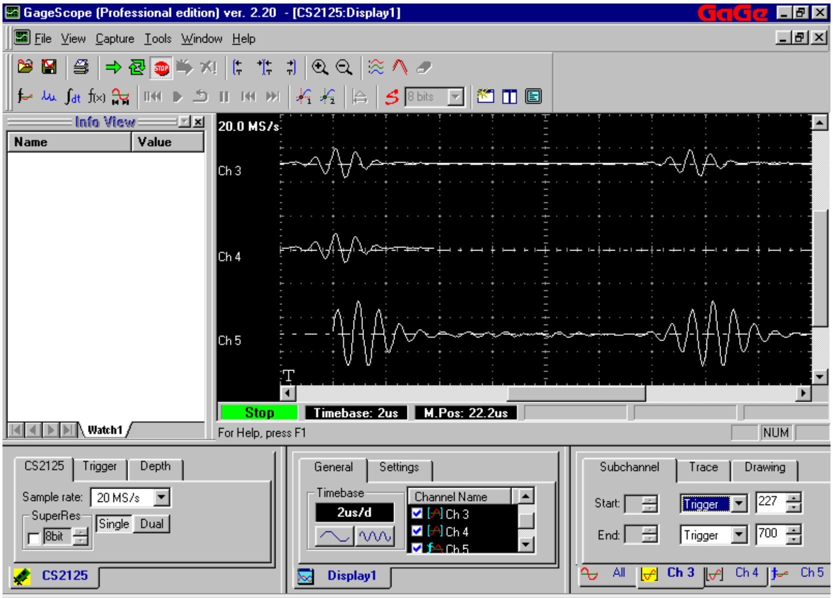


2 Choose **Operand 2**

3 Choose a channel for the **Result.**

As an example, Channel 3 below shows a simulated two echo system while channel 4 is the single echo which is extracted out of Channel 3 to perform the crosscorrelation operation between the two channels.

The result of Crosscorrelation is shown as Channel 5.


GageScope (Professional edition) ver. 2.20 - [CS2125:Display1]
File View Capture Tools Window Help
Info View
Name
Value
20.0 MS/s
Ch 3
Ch 4
Ch 5
Stop
Timebase: 2us
M.Pos: 22.2us
For Help, press F1
NUM
Watch1
CS2125
Trigger
Depth
Sample rate:
20 MS/s
SuperRes
8bit
Single
Dual
General
Settings
Timebase
2us/d
Channel Name
Subchannel
Trace
Drawing
Start:
End:
227
700
All
Display1

Troubleshooting

Q. GageScope® launched properly, but my signal is a flat line.

A. There are many reasons this could happen.

- Do you have a signal generator connected to your CompuScope card?
- Is the signal plugged into channel A or B of the CompuScope card, and not the other connectors (EXT for example)?
- Is the signal generator turned on?
- If you have several CompuScope cards in your machine, is GageScope® looking at the right card?
- If you are using a CompuScope card that requires a mezzanine power cable, is that power cable connected?

Q. How can I easily find the settings I want to change?

A. If you are looking for a particular feature you can always search the online help. Click on the **Help** menu, then **Help Topics**, then on the **Find** tab.

Q. I pressed the **One Shot** button but GageScope® acts as if I had clicked on **Stop**.

A. In Continuous mode, GageScope® looks for a trigger event, performs a capture, then rearms the CompuScope card for another trigger. In One Shot mode, GageScope® looks for a trigger event, performs a capture, then stops.

If a trigger situation existed the moment you clicked on the **One Shot** button, GageScope® performs a capture immediately, then stops. This could happen so quickly it may look like it went directly to **Stop**.

Q. The sample rates I have on my screen don't match what's in the manual.

A. The available sample rates and other settings depend on the model of CompuScope card. The manual was written using a CompuScope 2125, which has a maximum sample rate of 250 MS/s in single channel mode. If you have a CompuScope 8012A, the maximum sample rate in single channel mode is 100 MS/s.

Q. My card is supposed to have a maximum sample rate of 100 MS/s. The maximum in the sample rate list is 50 MS/s.

A. You are in dual channel mode. In dual channel mode, the maximum sample rate is half that of single channel mode. To resolve this problem, click on the single channel button in the CS tab of the System Control.

Q. I know I have a channel in the Display Window, but I can't see it.

A. This could happen for a few reasons.

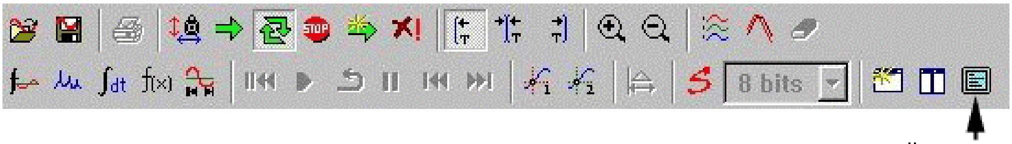
- The channel is hidden. Check the channel list in the General tab of the Display Control (lower middle of the screen).
- The channel is off-screen (outside of the Display Window). Click on the Arrange Channels button in toolbar to clean up the display.
- You are in single channel mode. In single channel mode, you do not get access to channel 2, if your CompuScope card supports a second channel. Click on Dual Channel in the CS tab of the System Control.
- The channel is the same color as the background color. For example, if the background color is black (the default), and the channel is black, it will appear invisible. To fix this problem, click on the tab of the channel you are looking for in the Channel Control (lower right of the screen), click on the Drawing tab, click on the **Color** button, and choose a new color.

Q. Impedance is listed in the manual, but I don't see it when I run GageScope® on my computer.

A. The Impedance setting normally appears in the CS Input tab of the Channel Control (lower right of the screen). If your CompuScope card does not support Impedance, the Impedance setting will not show up in the Channel Control.

Q. I want to hide all controls. Is there a quick way to do it?

A. You can hide controls individually by selecting them from the View menu, or you can select **Full Screen** from the View menu to hide everything at once. When you want to return the display to normal, click on the Full Screen button in the toolbar.



Q. How can I turn off some of the Display Window settings (such as the grid)?

A. The visibility and color of the grid and most other Display Window elements can be changed through the Settings tab of the Display Control.

Q. How can I quickly get to the trigger marker (where the trigger event occurred)?

A. Use the trigger align buttons in the toolbar.

Q. After I press the **MulRec** button in the Depth tab of the System Control, nothing happens.

A. Actually, quite a bit is happening, but the results usually aren't visible right away. Depending on the depth settings, your CompuScope card can fit a maximum number of records into its onboard memory. It takes a few moments to capture all these records. Click the Multiple Record buttons in the toolbar to move from record to record to check if GageScope® has finished capturing data.

Q. How do I tell how many records I have captured in Multiple Record mode?

A. After finishing all the captures, GageScope® displays the number of records underneath each channel's identifier at the left of the signal, in the form of 1/32, where 1 is the current record and 32 is the total number of records.

Q. I changed the channel mode (in the CS tab of the System Control) from single to dual, but acquisition stopped and the **Stop** button became active.

A. In certain cases you may find that when you change the channel mode from single to dual mode or vice-versa, continuous capture stops and the **Stop** button becomes active. This usually happens when the threads in GageScope® have to be stopped to achieve proper synchronization. You can start continuous capture again by clicking on the **Continuous** button .

Do you have a question?

Visit Gage's web site for more help on GageScope®.

http://www.gage-applied.com/

Or submit a support question through our online form.

www.gage-applied.com/support.asp

Appendices

Appendix A: The Instrument Manager Utility

The Instrument Manager configures the following:

- PC-based Instruments

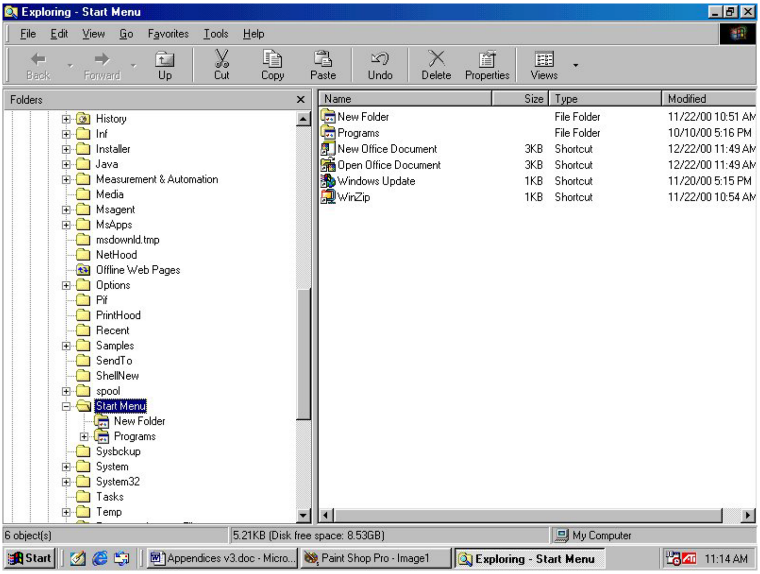
  These include the Gage high speed input and output, analog and digital data acquisition cards such as Compuscope 85G, 5 GigaSamples per second analog input card, which is the world's fastest PC based A/D card in the market today.

PC-based Instruments

Follow the steps outlined below to configure PC-based instruments such as Gage high speed data acquisition cards.

1 Right click **Start** on your computer's desktop.

2 Click **Explore**. Windows Explorer will launch and you will see the following window with Start Menu selected as a default:

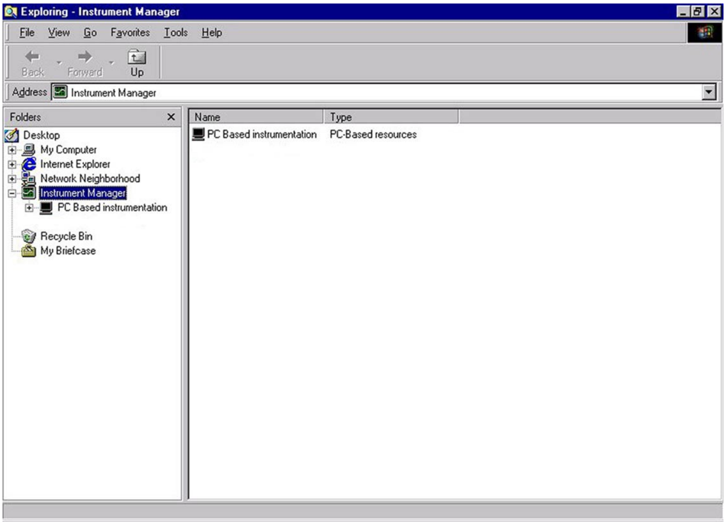


Note:

- Depending on the settings on your computer, you will see different contents in the explorer's left pane.
- You can also start Windows **Explorer** by clicking **My Computer** Icon from your computer's desktop.

3 To look for the Instrument Manager folder, scroll down the window in the left pane. If the Instrument Manager has been installed, you will find a folder named **Instrument Manager.**

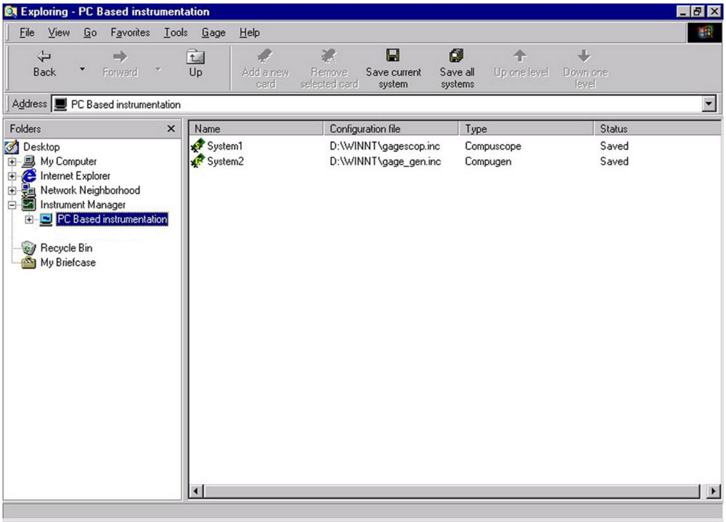
Click **Instrument Manager** Icon  to launch the Instrument Manager Explorer. You can also get directly to this step by double-clicking the Instrument Manager Icon on your computer's desktop.



Note that the active Window title now reads **Exploring: Instrument Manager**.
The contents of Instrument Manager folder appear in the right pane of the Instrument Manager Explorer. The right pane is divided into two fields: **Name** and **Type**.

Name shows the name of the folder while **Type** specifies the type of resource contained in the corresponding named folder.

4 Click **PC-based Instrumentation** Icon, . The PC-based Explorer opens:

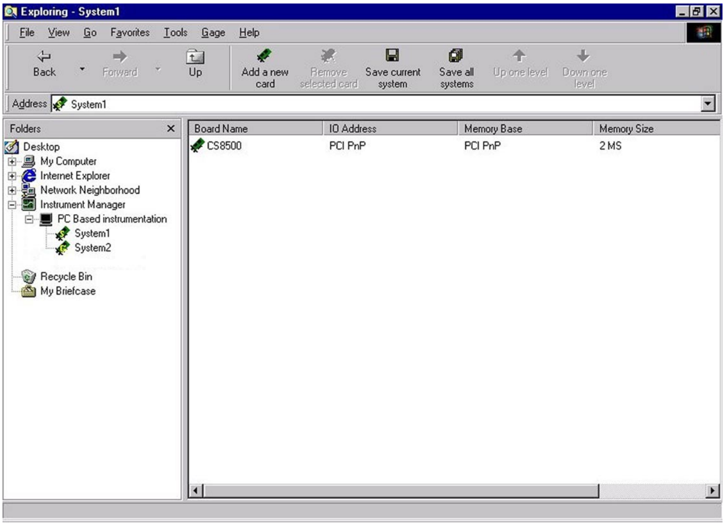


Note that the right pane displays the PC-based resources currently present in the system.

Four fields appear on the right hand side:
Name
Configuration Info
Type
Status

Name	The Name appears as **System1**, **System2**,…This list corresponds to the total number of Systems present in the PC. A system may comprise of a single or multiple Compuscope or Compugen cards.
Configuration File	Specifies the location of the configuration file in your personal computer. The configuration file is saved with **.inc** extension in the Windows directory in your PC's local drive. For example, the name assigned to the configuration file will be **Gagescop.inc**, in case a CompuScope card is present in the PC. For a CompuGen card present in your machine, the name of the configuration file will be **Gage_gen.inc**
Type	The type of resource, e.g. **CompuScope**, which refers to the Gage A/D fast data acquisition card and **CompuGen** which is the 12 bit arbitrary waveform generator by Gage Applied Technologies, Inc.
Status	This field shows the Status of the configuration file. **Saved** is displayed in the Status field whenever the system configuration has been saved. Otherwise the Status field is left blank.

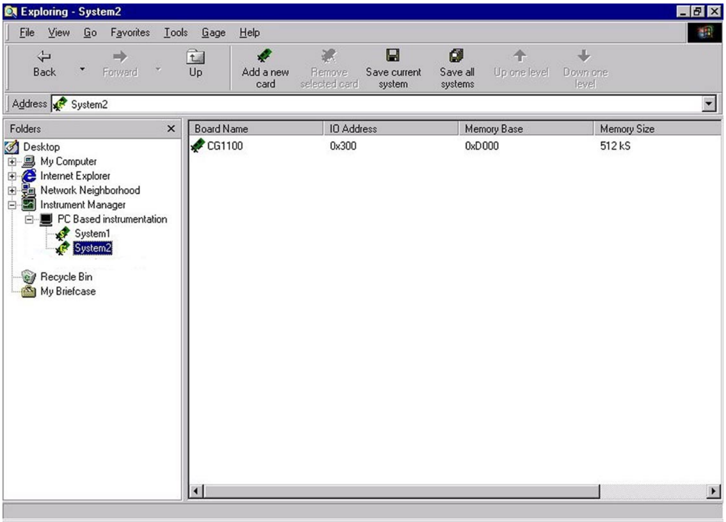
5 Double click on **System1** to view the System1 configuration as follows:



You can see that the CompuScope card, CS8500, present in the system, has 2 MegaSamples of onboard memory. Note that the CS8500 configuration settings are saved as shown in the Status field. In case the Status field does not show Saved, click on **Save configuration** icon from the toolbar or the drop down **Gage** menu.

6 Follow the same procedure for the CompuGen card to view and save the configuration settings for that card.

Double click on **System2** to view the System2 configuration as follows:



You can view the configurations settings for a single CompuGen card, CG1100, present in your PC. Click on **Save configuration** icon from the toolbar or the drop down **Gage** menu.

7 For the PC-based resources, the available fields in the right pane are

Board Name
IO Address
Memory Base
Memory
Status

The description of each of these fields is given as follows:

Board Name	This field identifies the name of the board present in the system. In the window shown above, CS8500 refers to CompuScope 8500 ultra high speed A/D card manufactured by Gage Applied Technologies, Inc.
IO Address	The IO address corresponds to the | nput/Output address of the card specified as System1. **PCI PnP** in this field shows that the CS8500 card is a Plug-and-Play device. For such cards, the IO address is automatically assigned by the Operating System. However, for an ISA card present in the system, the IO address of the card must be correctly specified. This is due to the fact that the Plug-and-Play feature is not supported by the ISA bus cards.
Memory Base	Memory Base specifies Base Memory address of the card in the system. For a list of permissible base memory addresses, refer to the Appendix D section of this chapter. Note that the Base Memory address of a PCI bus card is automatically set by the Operating System. Hence, PCI PnP appears in the Memory Base field for the CS8500 card. As for the IO address of the ISA bus cards, the Base memory address of an ISA bus card needs to be specified by the user.
Memory	The number in this field signifies the total amount of onboard memory in Samples present on the card. For CS8500 presently configured, 2M means that the total onboard memory available is two Mega Samples. For 8 bit cards, the memory specified in Samples is the same as the number of Bytes since a sample for 8 bit Gage cards is a Byte. For 12 and 16 bit Gage cards, the memory specified in Samples is to be multiplied by two to obtain the memory size in Bytes.

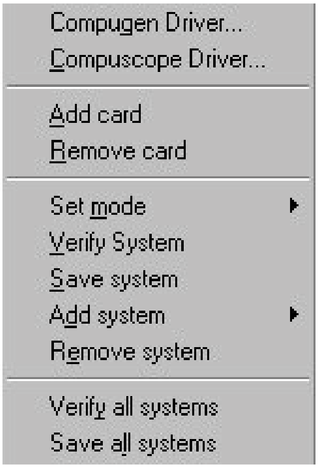
Status	Status shows the status of the configuration file, **Saved** and **Error** are the two possible entries for Status. **Saved** correspond to successful installation of drivers for the data acquisition card. Saved means that the configuration settings for Compuscope or Compugen cards have been saved in the Windows directory. **Error** status means that a card does not exist in the system.

Gage Menu

Gage Menu in the Menu bar allows users to configure newly installed Gage cards, set various options like adding, removing, saving file configurations for Gage CompuScope and CompuGen cards. Follow the steps below to explore Gage menu items:

1 View Gage Menu drop down list by selecting **Start --> Explore --> Instrument Manager --> PC-based Instruments --> Gage**

The Gage menu drop down list appears:

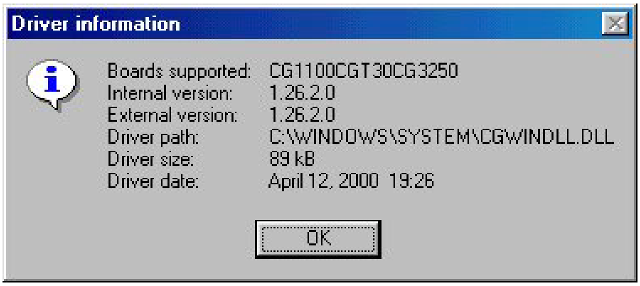


2 Gage menu is subdivided into four classes:

- **Driver Information**

Compugen Driver

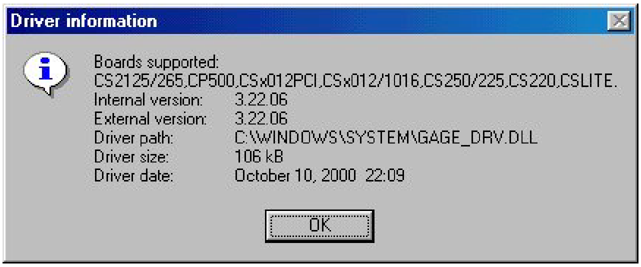
Select **Compugen Driver** from the menu. The Compugen Driver information dialog box appears:



The Compugen Driver information dialog box shows the types of boards supported, the version of the driver, location of the driver file in your machine, the driver size and date of creation.

Compuscope Driver

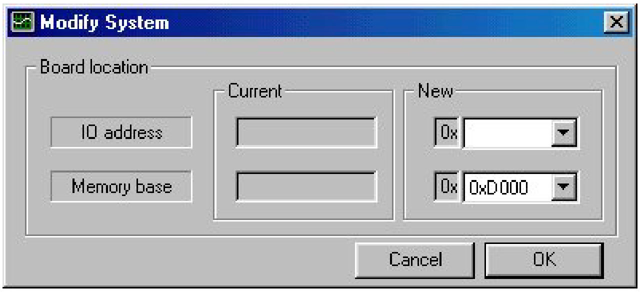
Select **Compuscope Driver** from the menu. The Compuscope Driver information dialog box appears:



The Compuscope Driver information dialog box shows the types of boards supported, the version of the driver, location of the driver file in your machine, the driver size and date of creation.

Add Card

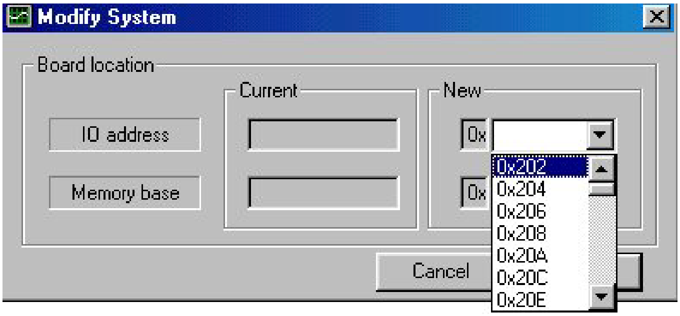
To add a CompuScope or a CompuGen card, click **Add Card** from the drop down menu. The **Modify System** dialog box appears:



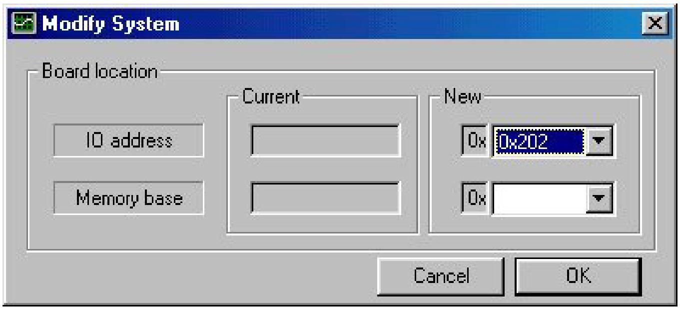
Two important pieces of information required to add a new card are the IO and Memory base addresses.

Adding an ISA bus card

To specify the IO address for the new ISA card, click on the display list symbol next to the text box under New Group. This is the text box, which is located in the first row. Select the address of the card from the drop down list. When you click on the appropriate address value, it shows in the IO address New text box.

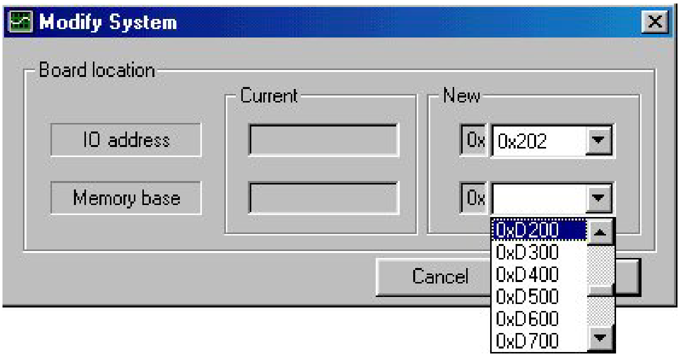


Click on the selected IO address to view the following:



The selected IO address appears in the IO address text box in the **New** Group.

Repeat the same procedure for the Base Memory address for the ISA card. Click on the drop down list button besides the text box in the **New** Group. The list of available memory base addresses appear:

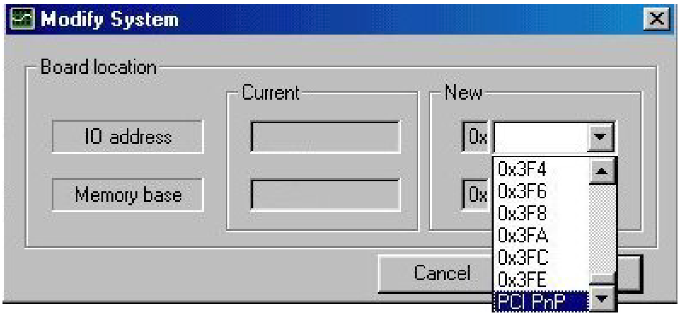


4

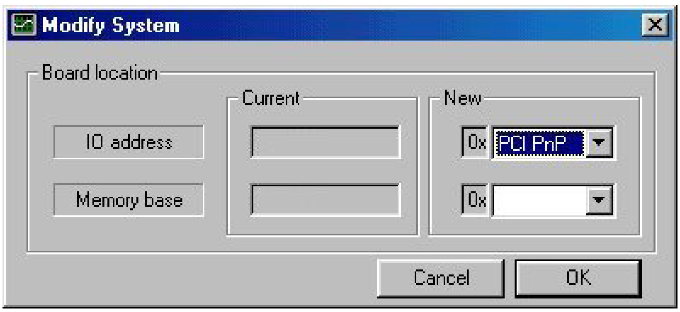
Adding a PCI bus card

To specify the IO address for the new PCI card, click on the display list symbol next to the text box under New Group. This is the text box, which is located in the first row. Select **PCI PnP** from the drop down list. When you click on the selected value, PCI PnP appears in the IO address New text box.

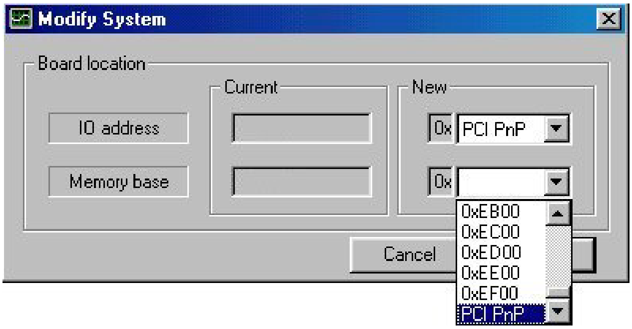
Scroll down to the bottom of the list. Select **PCI PnP.**



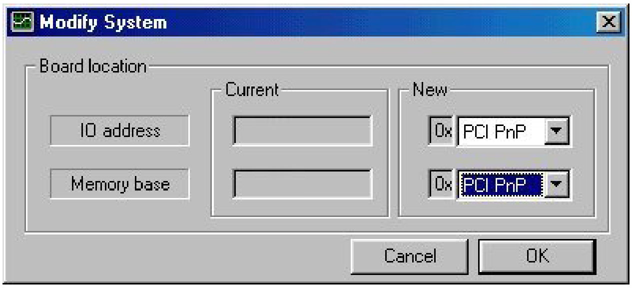
PCI PnP appears in the IO address text box in the **New** Group.



Repeat the same procedure for the Base Memory address for the PCI card. Click on the drop down list button besides the text box in the **New** Group. The list of available memory base addresses appear:



Scroll down to the bottom of the list. Select **PCI PnP**.



Note that the text boxes under the **Current** setting are greyed out since the new configuration has not been saved as yet.

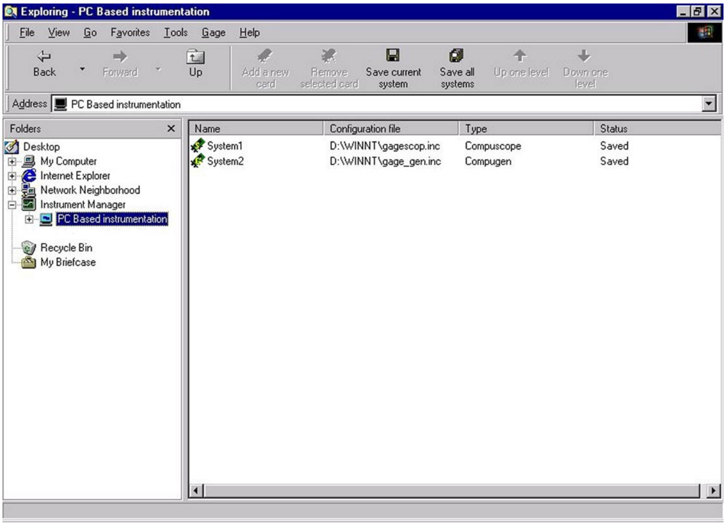
5 **Remove Card**

To be able to remove a card from the system, you must select an already installed card from within the Instrument Manager Explorer.

Follow the following sequence of steps to get to the PC-based Instruments folder in the Instrument Manager Explorer:

Start --> Explore --> Instrument Manager --> PC-based Instruments

You will see the following window:



Click on System1 to select it. You will notice that the entire row will be selected.

Now, right click on Gage Menu and select **Remove card** command. **System1** will be deleted and you will see only System2 in the right pane.

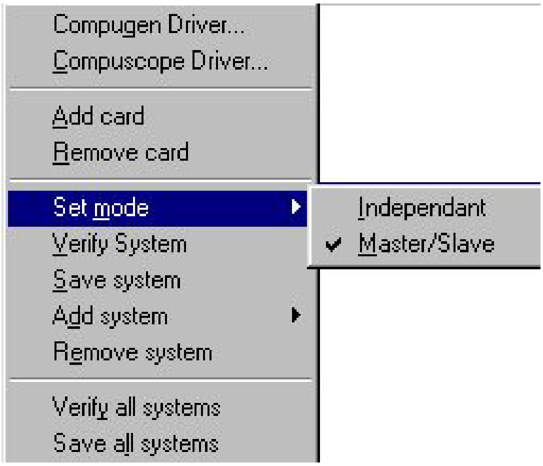
6 • **System Options**

Set Mode

Set Mode Option specifies whether a card is to be used in an **independent** unit or part of a **Master/Slave** system.

As implied by independent, a Gage data acquisition card is to be used as a single card contrary to the Master/Slave Mode in which two or more Gage cards are set up as a multi channel system.

Click **Set Mode** to select either of the two options, depending on the boards in your system:



7 **Verify System**

Once the new cards are configured properly, these need to be verified for proper operation. Click **Verify System** to make sure that the newly installed card(s) are verified.

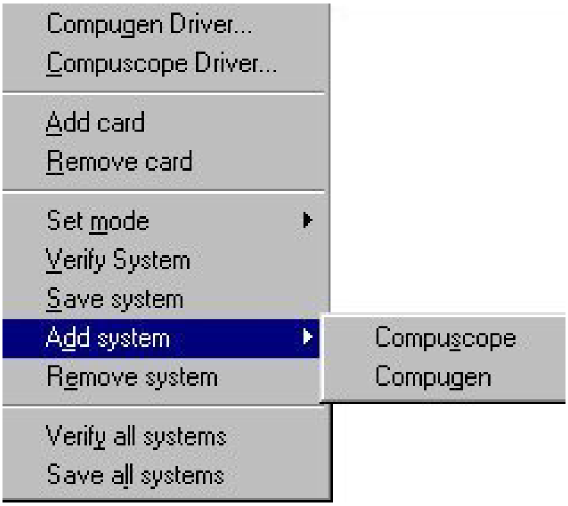
Save System

Save System command allows the Gage system configuration to be saved in the Windows directory as a GageScop.inc or Gage_ gen.inc file.

8 **Add System**

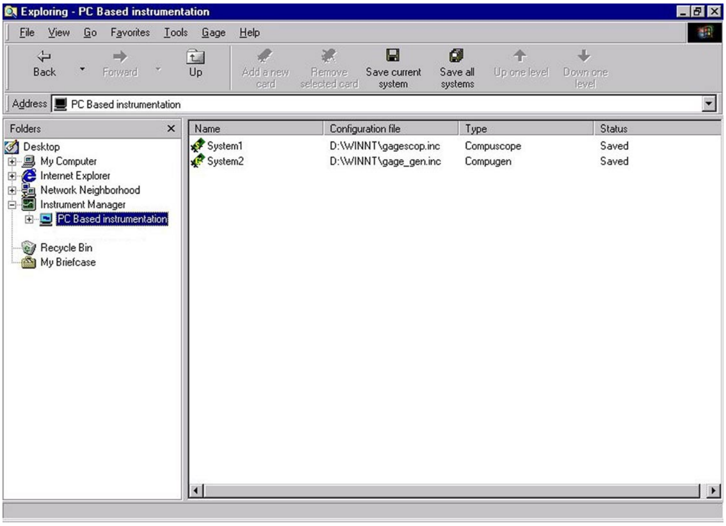
Add System command allows you to add two or more Compuscope or Compugen cards cards to your system.

Click on **Add System** to view the two options. Select either of the two depending on the type of card you are trying to install.



As soon as an option is selected, the Instrument Manager creates a new folder with an appropriate name in the PC-based Instrumentation folder.

As an example, we have a CompuScope and a Compugen card configured in our PC. These appear as **System1** and **System2** under **PC-based Instrumentation** folder in the Instrument Manager:



System1 comprises of a Compuscope card while **System2** shows that this system contains a Compugen card.

When **Add System** command is selected from the Gage drop down menu, Instrument Manager creates a new System folder. The Name of the folder depends on whether the System belongs to Compuscope or the Compugen cards.

In the present setup, if you add a CompuScope card as a new system, the name of the folder will be **System2** and the Compugen system folder name will be automatically changed to be System3 instead of System2.

In case the system being added comprises of a single or multiple Compugen cards, the name of the new System will be **System3**.

Note that the cards in the new System need to be configured properly after the creation of the System folder.

9 **Remove System**

Click on the System to be removed from the setup.

Click Gage Menu to view the drop down list. Select **Remove System** to remove the selected System from your PC.

10 Verify All Systems

When this option is selected, all Systems in your personal computer are verified.

11 Save All Systems

When this option is selected, all Systems in your personal computer are saved.

List of Instruments Supported by GageScope®

Type	Model/Family	Vendor
DSO	CompuScope 85G	Gage
DSO	CompuScope 82G	Gage
DSO	CompuScope 8500	Gage
DSO	CompuScope 14100	Gage
DSO	CompuScope 1450	Gage
DSO	CompuScope 12100	Gage
DSO	CompuScope 1250	Gage
DSO	CompuScope 1602	Gage
DSO	CompuScope 8012	Gage
DSO	CompuScope 8012A	Gage
DSO	CompuScope 6012	Gage
DSO	CompuScope 1012	Gage
DSO	CompuScope 512	Gage
DSO	CompuScope 2125	Gage
DSO	CompuScope 265	Gage
DSO	CompuScope 250	Gage
DSO	CompuScope 225	Gage
DSO	CompuScope 220	Gage
DSO	CompuScope LITE	Gage
DSO	CompuScope 8012A/PCI	Gage
DSO	CompuScope 8012/PCI	Gage
DSO	CompuScope 6012/PCI	Gage
DSO	CompuScope 1012/PCI	Gage
DSO	CompuScope 512/PCI	Gage
AWG	CompuGen 1100	Gage

Note

For an updated list of supported instruments, visit the GageScope® web site at http://www.gagescope.com

Appendix B: GageScope® Signal File Format (.SIG)

GageScope® signal files are binary files which contain a 512 byte header followed by any number of data points. The first 512 bytes of a GageScope® signal file contain the header, the exact format of which is listed below:

File Index	Field Type	Field Size	Field Variable	Field Description
0	char Note 1.	14	file_ version	Either GS V.1.20, GS V.2.00, GS V.2.05, GS V.2.10, GS V.2.15, GS V.2.20, GS V.2.25, GS V.2.50, GS V.2.60, GS V.2.65, GS V.2.70, GS V.2.75, GS V.2.80, GS V.2.85, GS V. 2.95 or GS V. 3.0.
14	int16	2	crlf1	A carriage return line feed pair.
16	char	9	name	The channel name when stored.
25	int16	2	crlf2	A carriage return line feed pair.
27	char	256	comment	The channel comment when stored.
283	int16	2	crlf3	A carriage return line feed pair.
285	int16	2	control_ z	A control Z, artificial end of file.
287	int16	2	sample_ rate_ index	Index to the sample rate table. Note 2.
289	int16	2	operation_ mode	1 = single channel, 2 = dual channel.
291	int32	4	trigger_ depth	Number of samples after the trigger point.
295	int16	2	trigger_ slope	1 = positive slope, 2 = negative slope.
297	int16	2	trigger_ source	1 = chan A, 2 = chan B, 3 = external, 4 = automatic, 5 = keyboard.
299	int16	2	trigger_ level	Stored as an int, actually a byte with the same format as the data. Note 5.
301	int32	4	sample_ depth	Number of samples stored in the signal section of the file.
305	int16	2	captured_ gain	Index to the input range table. Note 3.
307	int16	2	captured_ coupling	1 = DC, 2 = AC.

File Index	Field Type	Field Size	Field Variable	Field Description
309	int32	4	current_ mem_ ptr	Where display started when signal was stored.
313	int32	4	starting_ address	The first point in the data.
317	int32	4	trigger_ address	The point in the data where trigger occurred.
321	int32	4	ending_ address	The last point of the captured data.
325	uInt16	2	trigger_ time	The time when the trigger event occurred. Note 7.
327	uInt16	2	trigger_ date	The date on which the trigger event occurred.
329	int16	2	trigger_ coupling	1 = DC, 2 = AC. For the external trigger input.
331	int16	2	trigger_ gain	Index to the input range table. Note 3.
333	int16	2	probe	Index to the probe table. Note 4.
335	int16	2	inverted_ data	0 = normal data, 1 = inverted data (CS220), 2 = inverted and flipped data (CS220).
337	uInt16	2	board_ type	The CompuScope board type on which the saved data was captured. Note 6.
339	int16	2	resolution_ 12_ bits	0 = 8 bit file format, 1 = 12/16 bit file format.
341	int16	2	multiple_ record	The mode that the saved data was captured in: 0 = normal mode, 1 = Hardware multiple record, 2 = Software multiple record. **Note**: 5 = Multiple record with least significant bits saved and 6 = Software multiple record with least significant bits saved are not available in GageScope®. They are currently only available in the drivers with a hardware upgrade.
343	int16	2	trigger_ probe	Index to the probe table. Note 4.

File Index	Field Type	Field Size	Field Variable	Field Description
345	int16	2	sample_ offset	Used to offset the data for display and conversion to real voltages. Normally 127 for 8-bit CompuScopes and -1 for 12-bit CompuScopes. File versions before GS V. 2.95 will have sample offset=0 for 12-bit CompuScopes and 128 for 8-bit CompuScopes.
347	int16	2	sample_ resolution	Used to scale the data for display and conversion to real voltages. Normally 128 for 8-bit CompuScopes and 2048 for 12-bit CompuScopes.
349	int16	2	sample_ bits	Number of bits in the sampled data. Normally 8 for 8-bit CompuScopes and 12 for 12-bit CompuScopes.
351	uInt32	4	extended_ trigger_ time	The time when trigger event occurred. Note 7.
355	int16	2	imped_ a	Impedance for Channel A. 0 = 1 MegaOhm0x10 = 50 Ohm.
357	int16	2	imped_ b	Impedance for Channel B. 0x0 = 1 MegaOhm; 0x10 = 50 Ohm.
359	float	4	external_ tbs	Time between samples in nanoseconds when using external clock.
363	float	4	external_ clock_ rate	Minimum sample rate when using external clock.

File Index	Field Type	Field Size	Field Variable	Field Description
367	int32	4	file_ options	01 = GAGE_ SINGLE_ CHAN 02 = GAGE_ DUAL_ CHAN 04 = GAGE_ MODE_ EXT_ CLK_ ADJ . Can be ORed with either GAGE_ DUAL_ CHAN or GAGE_ SINGLE_ CHAN. 08 = GAGE_ MODE_ MR_ J ITTER_ ADJ Can be ORed with either GAGE_ DUAL_ CHAN or GAGE_ SINGLE_ CHAN. 10 = GAGE_ FAST_ RAM_ ADJ . Can be ORed with either GAGE_ DUAL_ CHAN or GAGE_ SINGLE_ CHAN. 20 = GAGE_ X012X_ VERS_ ADJ UST. Can be ORed with either GAGE_ DUAL_ CHAN or GAGE_ SINGLE_ CHAN.
371	uInt16	2	version	Version of the CompuScope hardware, 4 BCD digits.
373	uInt32	4	eeprom_ options	Hardware options stored in the CompuScope board’s on-board EEPROM.
377	uInt32	4	trigger_ hardware	Auxiliary trigger hardware in use. 0 = none; 1 = Trigger Marker Board (TMB).
381	uInt32	4	record_ depth	The size of each Multiple Record record when operating in Pre-trigger Multiple Record (PTM) mode.
385	uInt8	127	padding	0 filled section to complete the 512 byte header.

File Index	Field Type	Field Size	Field Variable	Field Description
512	uInt8 / int16 See note	var	signal	RAM image of the CompuScope memory at the time the signal file was stored. This data is in two different formats. The first format is when "operation_ mode" is equal to two. The data is stored contiguously as a binary image of the saved channel's signal storage space (one-half the memory depth). The second format is when "operation_ mode" is equal to one. The data is interleaved as a binary image of the complete signal storage space for the single channel mode (full memory depth). Interleaved in this example means the data is in two sections after the header. The first section is the data that was stored in the CompuScope memory for converter A and the second section of data is the memory assigned to converter B. All even addresses are in the area for converter A while the odd addresses are in the same place as the next smaller address, but offset into the converter B data area. In both cases, the stored addresses for the signals are indexes into the data. When extracting data, care must be taken to "wrap the pointers around" at the end of the file. **NOTE:** If the "resolution_ 12_ bits" flag equals zero then the data is stored as unsigned 8 bit bytes. Note 5. Otherwise, if the "resolution_ 12_ bits" flag equals one then the data is in the 12/16 bit format which is stored as 16 bit signed integers (in the 12 bit mode the sampled data is sign extended to 16 bits).

Note 1: Memory types

For portability reasons, the signal files use the following predefined memory types:

char	=	ASCII character
int 8	=	signed char (or byte)
uInt 8	=	unsigned char (or byte)
int 16	=	signed 16-bit integer
uInt 16	=	unsigned 16-bit integer
int 32	=	signed 32-bit integer
uInt 32	=	unsigned 32-bit integer

Note 2: Miscellenous

- GageScope® files are always stored in the dual channel format i.e. (Op Mode == 2)
- GageScope® can also save files in the 32 bit format

Note 3: The "sample_rate_index" to the sample rate table

For all file versions		Version 2.65 and below		Version 2.70 and above		Version 2.85 and above	
Index	**Sample Rate**	**Index**	**Sample Rate**	**Index**	**Sample Rate**	**Index**	**Sample Rate**
0	1 Hz	18	1 MHz	18	1 MHz	18	1 MHz
1	2 Hz	19	2 MHz	19	2 MHz	19	2 MHz
2	5 Hz	20	5 MHz	20	4 MHz	20	4 MHz
3	10 Hz	21	10 MHz	21	5 MHz	21	5 MHz
4	20 Hz	22	20 MHz	22	10 MHz	22	10 MHz
5	50 Hz	23	25 MHz	23	20 MHz	23	12.5 MHz
6	100 Hz	24	40 MHz	24	25 MHz	24	20 MHz
7	200 Hz	25	50 MHz	25	30 MHz	25	25 MHz
8	500 Hz	26	100 MHz	26	40 MHz	26	30 MHz
9	1 kHz			27	50 MHz	27	40 MHz
10	2 kHz			28	60 MHz	28	50 MHz
11	5 kHz			29	100 MHz	29	60 MHz
12	10 kHz			30	120 MHz	30	65 MHz
13	20 kHz			31	125 MHz	31	80 MHz
14	50 kHz			32	150 MHz	32	100 MHz
15	100 kHz			33	200 MHz	33	120 MHz
16	200 kHz			34	250 MHz	34	125 MHz
17	500 kHz			35	300 MHz	35	130 MHz
				36	500 MHz	36	150 MHz
				37	1 GHz	37	200 MHz
				38	2 GHz	38	250 MHz
				39	5 GHz	39	300 MHz
				40	External Clock	40	500 MHz
				39*	4 GHz	41	1 GHz
				40*	5 GHz	42	2 GHz
				41*	External Clock		

* For File Version 2.80 and above

For all file versions		Version 3.00 and above		Version 3.00 and above	
Index	**Sample Rate**	**Index**	**Sample Rate**	**Index**	**Sample Rate**
0	1 Hz	18	1 MHz	33	120 MHz
1	2 Hz	19	2 MHz	34	125 MHz
2	5 Hz	20	2.5 MHz*	35	130 MHz
3	10 Hz	21	5 MHz	36	150 MHz
4	20 Hz	22	10 MHz	37	200 MHz
5	50 Hz	23	12.5 MHz	38	250 MHz
6	100 Hz	24	20 MHz	39	300 MHz
7	200 Hz	25	25 MHz	40	500 MHz
8	500 Hz	26	30 MHz	41	1 GHz
9	1 kHz	27	40 MHz	42	2 GHz
10	2 kHz	28	50 MHz	43	4 GHz
11	5 kHz	29	60 MHz	44	5 GHz
12	10 kHz	30	65 MHz	45	8 GHz
13	20 kHz	31	80 MHz	46	10 GHz
14	50 kHz	32	100 MHz	47	External Clock
15	100 kHz				
16	200 kHz				
17	500 kHz				

*This entry was previously 4 MHz. Used on CSx012 and CSx012/PCI boards except for CS512 and CS512/PCI. Support for this sample rate has been dropped.

Note 4: The "captured_gain" and "trigger_gain" input ranges

If "file_version" equals "GS V.1.20"	Index	Input Range
	0	+/- 1v
	1	+/- 200mv

If "file_version" is greater than or equal to "GS V.2.00"	Index	Input Range
	0	+/- 10v
	1	+/- 5v
	2	+/- 2v
	3	+/- 1v
	4	+/- 500mv
	5	+/- 200mv
	6	+/- 100mv

Note 5: The "probe" connected to the input channel at time of capture.

If "file_version" is less than or equal to "GS V.2.10"	Index	Probe Multiplier
	0	x1
	1	x10
	2	x20
	3	x100
	4	x200

If "file_version" is greater than or equal to "GS V.2.15"	Index	Probe Multiplier
	0	x1
	1	x10
	2	x20
	3	x50
	4	x100
	5	x200
	6	x500
	7	x1000

Note 6: The "trigger_level" and the "captured_data" 8 bit uInt8 format

The byte stored as the trigger level is in the following format: the data on the CS8500, CS2125, CS265, CS250, CS225 and CSLITE is represented as an 8-bit unsigned integer, where the smallest value (0) represents -1 V and the largest value (255) represents +1 V. On the other hand, for the captured data, smallest value (0) represents +1 V and the largest value (255) represents -1 V. The data is normalized into a signed floating-point representation. The input range and probe are then multiplied by the return value to produce the actual level either sent to the hardware as the trigger level or returned from it after capture.

If the "resolution_ 12_ bits" flag equals zero then the data is stored as unsigned 8 bit bytes. Otherwise, if the "resolution_ 12_ bits" flag equals one then the data is in the 12/16 bit format which is stored as 16 bit signed integers (in the 12 bit mode the sampled data is sign extended to 16 bits). For the 12 bit boards, the smallest value (-2047) represents –1V while the largest value (+2047) represents +1V for the trigger level whereas the smallest and the largest values represent +1V and –1V respectively for the captured data.

Note 7: The "board_type" constants

"board_type"	CompuScope Hardware
0x0000	Unknown (pre "file_ version" GS v2.25)
0x0001	CompuScope 265
0x0002	CompuScope 8500
0x0004	CompuScope 8012-TYPE*
0x0008	CompuScope 8012
0x000E	CompuScope 12100, CompuScope 12130, CompuScope 1250
0x0010	CompuScope PCI
0x001C	CompuScope 8012/PCI
0x0020	CompuScope 512
0x0040	CompuScope 225
0x0080	CompuScope 250 v1.8+
0x0100	CompuScope LITE (pre-hardware version 1.5)
0x0200	CompuScope 220
0x0400	CompuScope 250
0x0800	CompuScope LITE (hardware version 1.5 & up).
0x1000	CompuScope 1012
0x1010	CompuScope 1012/PCI
0x2000	CompuScope 6012
0x2010	CompuScope 6012/PCI
0x4000	CompuScope 2125
0x8000	CompuScope 1016
0x8006	CompuScope 1610

* CompuScope 8012-TYPE refers to CSx012 cards, version 2.0+.

Note 8: Extended trigger time

GageScope® now supports an extended time stamp for the trigger event time of day. The resolution of this timer is in hundredths of a second. However, on some computer systems the real-time clock does not support complete accuracy at this resolution. If the normal trigger time in the signal file is zero, then the extended trigger time stamp is being used. The encoded long integer is hhhhhmmmmmmsssssddddddd, where dddddd is hundredths of a second. Note also that the new time supports full seconds.

Appendix C: Default Settings

The following settings are used upon startup of GageScope®.

CompuScope Card and Display Settings

Capture Mode:	Continuous
Channel Mode:	Dual (if two channels are available)
Sample Rate:	Maximum allowed by your CompuScope
SuperRes Mode:	Off

Channel Settings

Identification:	Channel A of the CompuScope is channel 1 Channel B of the CompuScope is channel 2
Colors:	Channel 1 set to yellow, channel 2 set to green
Input Range:	All channels set to ±1V (±2V for CS1016)
Position:	Depends on the number of channels. If there are two channels, channel 1 is displayed in the top half of the Display Window, and channel 2 is displayed in the bottom half. If there is only one channel, it is displayed in the middle of the Display Window
Vertical Scale:	All channels set to 1 Volt per division
Method:	Min Max
Trace:	Connect Dots
Impedance:	1 MOhm (if the CompuScope allows it)
Probe:	X1

Trigger Settings

Trigger:	Set to channel 1
Trigger Level:	0 mV
Timeout:	10 ms
Slope:	Positive
Pre-trigger depth:	4096 Samples
Post-trigger depth:	4096 Samples

Display Settings

Timebase:	5 us/d
Channels Shown:	All channels visible. A CompuScope channel that is not connected to a signal will display as a flat line
Persistence:	Off
Grid:	On
X/Y Axis:	On
Zero Lines:	On

Appendix D: CompuScope I/O Settings

This appendix contains brief information on solving I/O address conflicts encountered when configuring your CompuScope cards using the Instrument Manager. It is assumed that the cards have been properly installed in your computer. For more details on installing your CompuScope card, consult your CompuScope hardware manual or Start-up Guide.

Before changing the I/O address on your CompuScope card, ensure that the I/O address entered in Instrument Manager matches that set via the switches or jumpers, as outlined in the *Configuring PC-based cards using the Instrument Manager*.

Alternate I/O Addresses

Note about PCI-based Boards

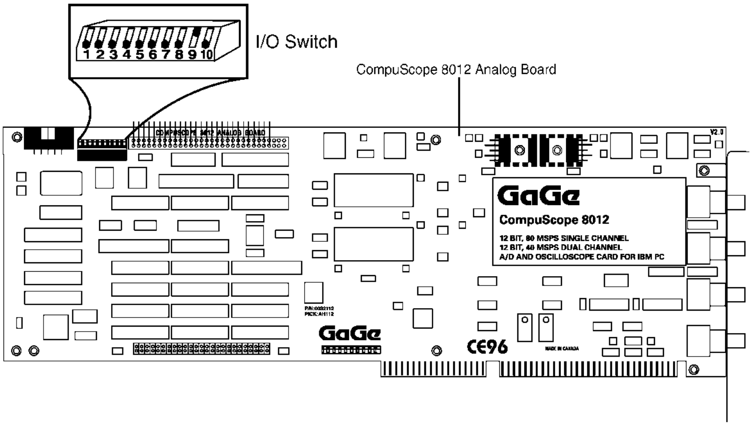
I/O addresses are not specified by the user for following PCI-based boards:

- CompuScope 85G
- CompuScope 82G
- CompuScope 1602
- CompuScope 14100
- CompuScope 12100
- CompuScope 1250
- CompuScope 8500

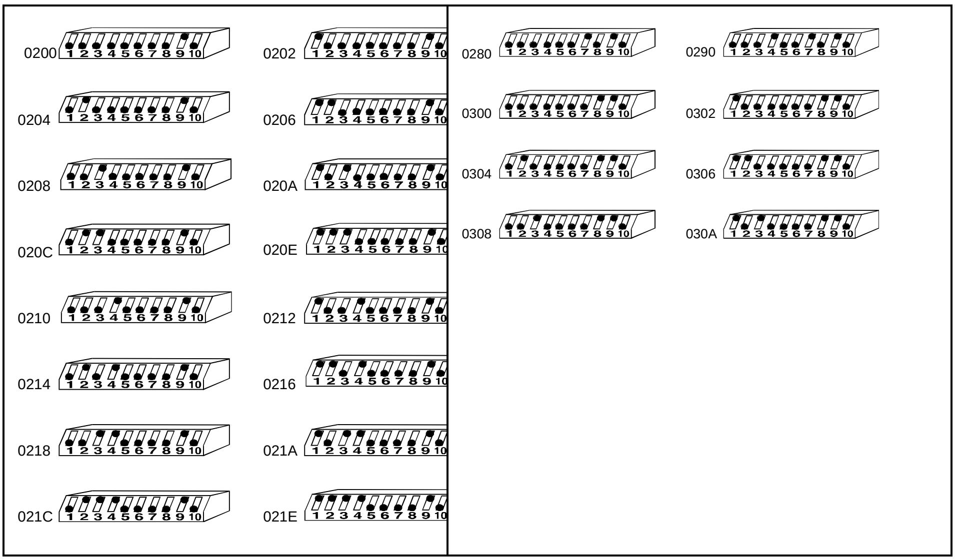
The assignment of I/O addresses for these boards is handled by the PCI BIOS at boot time. Please refer to the *Gage Driver Installation Guide for CompuScope Cards* for further details.

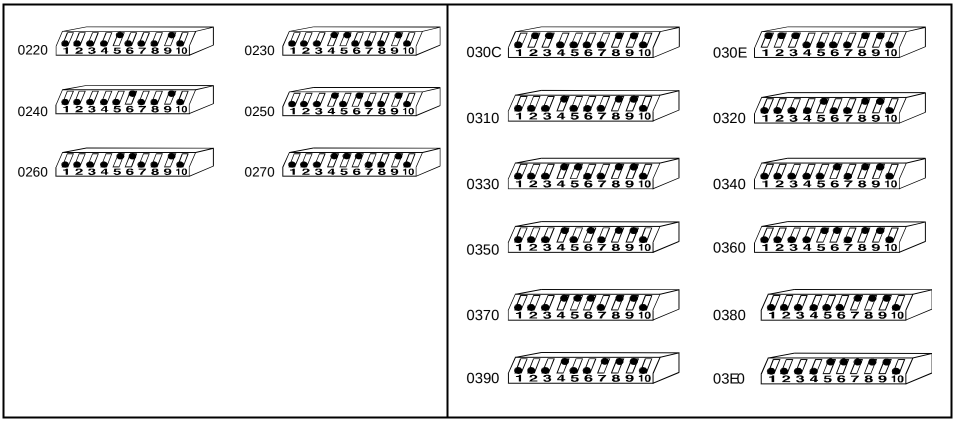
CompuScope 8012A/PCI, 8012/PCI, 6012/PCI, 1012/PCI and 512/PCI

The DIP switch SW1, as shown in the diagram below, defines the base I/O address of the CS8012A/PCI, CS8012/PCI, CS6012/PCI, CS1012/PCI and CS512/PCI boards.



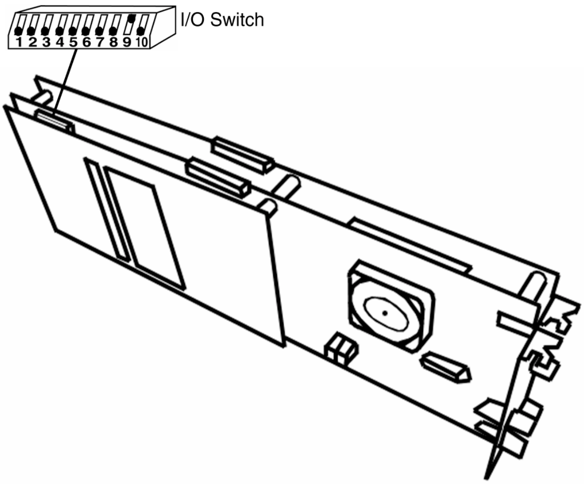
Each of the switch positions A1 through A9 corresponds to an address line A1 through A9 from the ISA bus. As the CS8012A/PCI, CS8012/PCI, CS6012/PCI, CS1012/PCI and CS512/PCI use two I/O locations, there is no need to configure a switch for A0. A10 is not used.



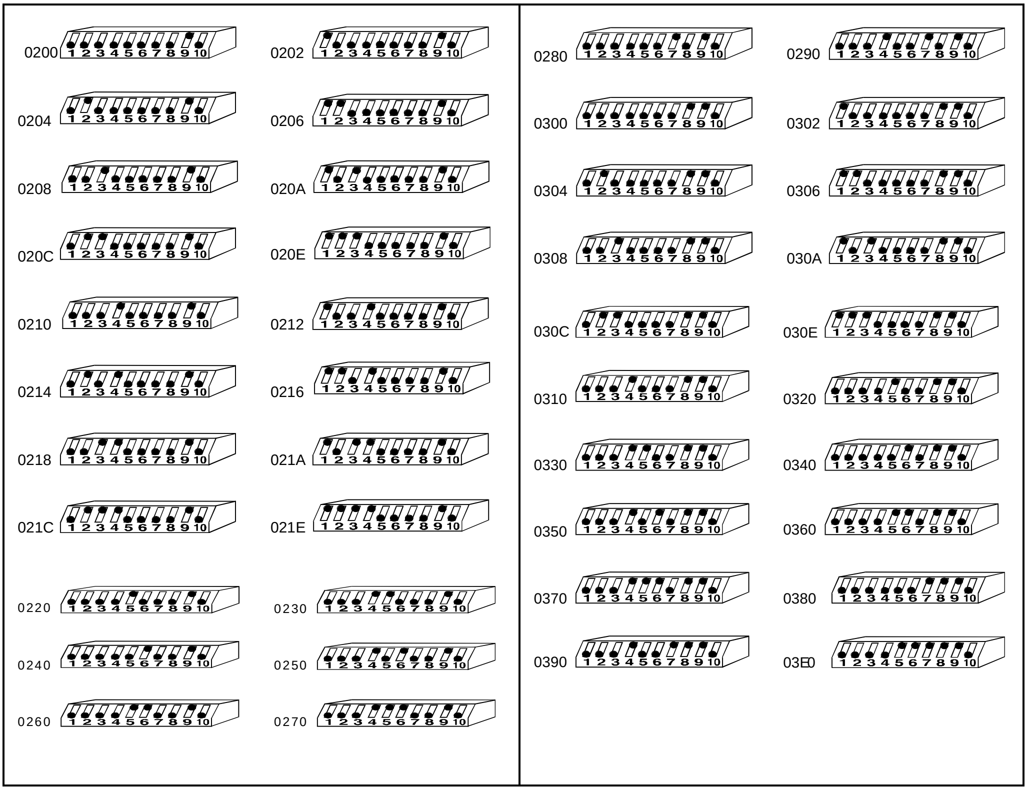


CompuScope 1016

The SIP switch SW1, as shown in the diagram below, defines the base I/O address of the CS1016 board. (Note: The illustration below shows CS1016 v1.1, but the I/O Switch SW1 has the same location on CS1016 v1.0.)

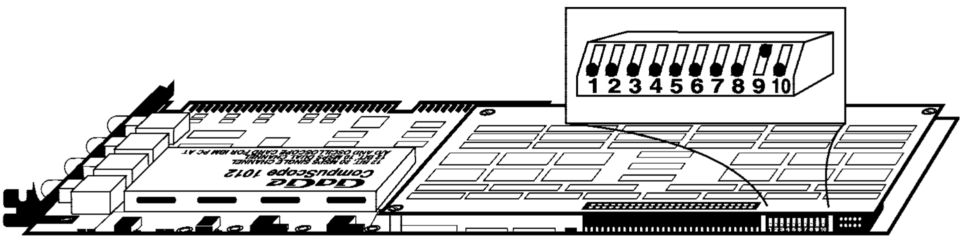


Each of the switch positions A1 through A9 corresponds to an address line A1 through A9 from the ISA bus. As the CS1016 uses two I/O locations, there is no need to configure a switch for A0. A10 is not used.

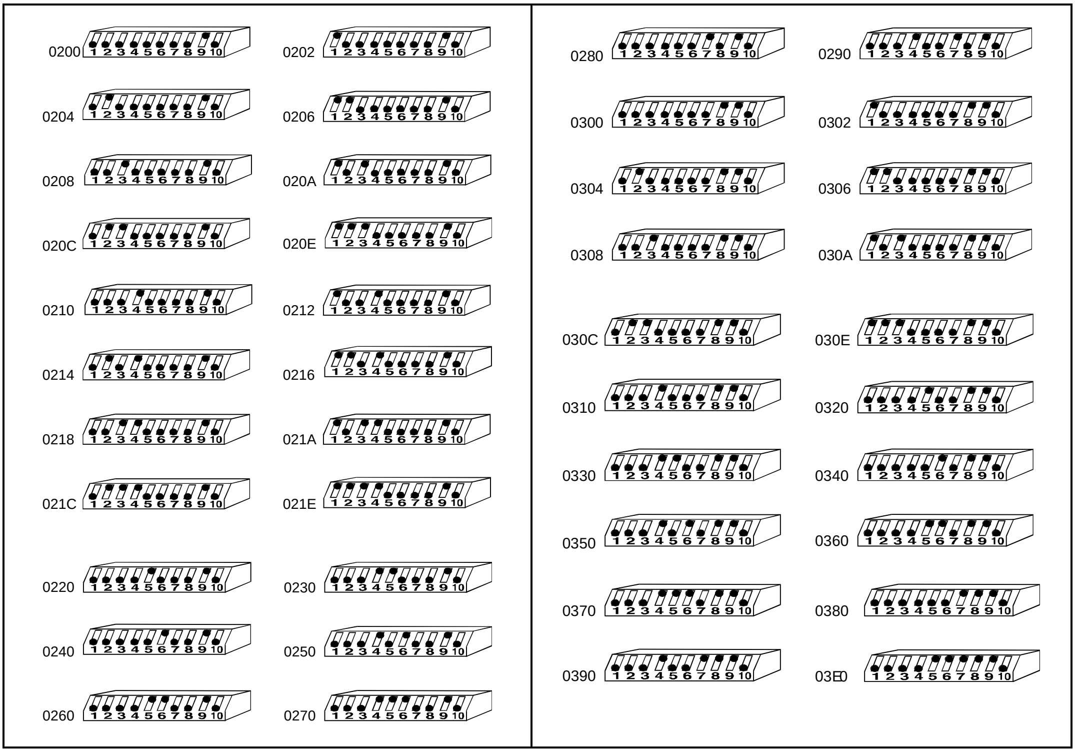


CompuScope 8012A, 8012, 6012, 1012, and 512

The DIP switch SW, as shown in the diagram below, defines the base I/O address of the CS8012A, CS8012, CS6012, CS1012 and CS512 boards. (See the following page for I/O Addresses for CS8012A v3.0.)

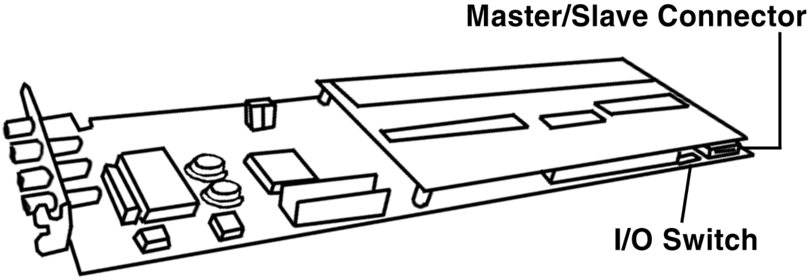


Each of the switch positions A1 through A9 corresponds to an address line A1 through A9 from the ISA bus. As the CS8012A, CS8012, CS6012, CS1012 and CS512 use two I/O locations, there is no need to configure a switch for A0. A10 is not used.


0200
0202
0204
0206
0208
020A
020C
020E
0210
0212
0214
0216
0218
021A
021C
021E
0220
0230
0240
0250
0260
0270
0280
0290
0300
0302
0304
0306
0308
030A
030C
030E
0310
0320
0330
0340
0350
0360
0370
0380
0390
03E0

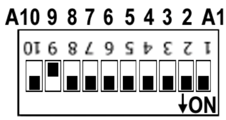
CompuScope 8012A Board Version 3.0

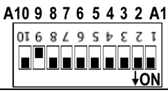
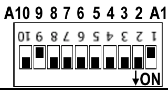
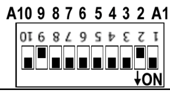
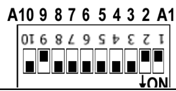
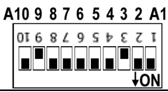
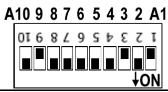
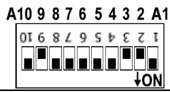
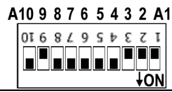
The DIP switch SW1, as shown in the diagram below, defines the base I/O address of the CS8012A v3.0 board.

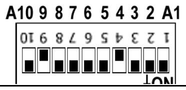
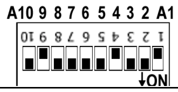
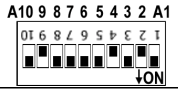
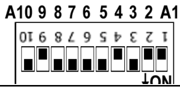
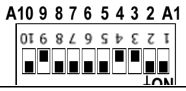
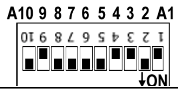
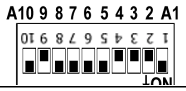
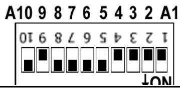


Each of the switch positions A1 through A9 corresponds to an address line A1 through A9 from the ISA bus. As the CS8012A uses two I/O locations, there is no need to configure a switch for A0. A10 is not used. To change the switch setting, a 1 mm (1/16 in.) jeweler's screwdriver is recommended.

NOTE FOR MULTI-CARD SYSTEMS: If you need to change the I/O switch setting, you must remove the Interconnect first in order to access the I/O switch.

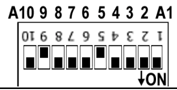


0200 9 UP 	0202 1,9 UP 	0204 2,9 UP 	0206 1,2,9 UP 
0208 3,9 UP 	020A 1,3,9 UP 	020C 2,3,9 UP 	020E 1,2,3,9 UP 

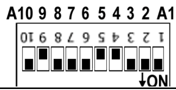
0210 4,9 UP 	0212 1,4,9 UP 	0214 2,4,9 UP 	0216 1,2,4,9 UP 
0218 3,4,9 UP 	021A 1,3,4,9 UP 	021C 2,3,4,9 UP 	021E 1,2,3,4,9 UP 

CS8012A v3.0 I/O Switch Settings (Continued)

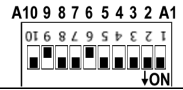
0220 5,9 UP



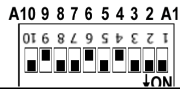
0230 4,5,9 UP



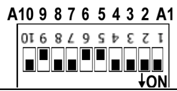
0240 6,9 UP



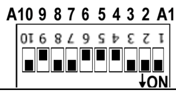
0250 4,6,9 UP



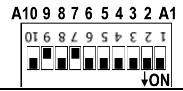
0260 5,6,9 UP



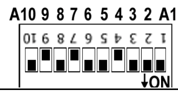
0270 4,5,6,9 UP



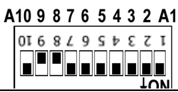
0280 7,9 UP



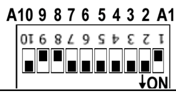
0290 4,7,9 UP



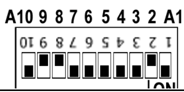
0300 8,9 UP



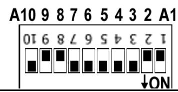
0302 1,8,9 UP



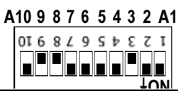
0304 2,8,9 UP



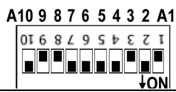
0306 1,2,8,9 UP



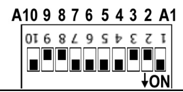
0308 3,8,9 UP



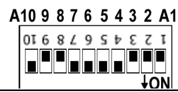
030A 1,3,8,9 UP



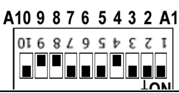
030C 2,3,8,9 UP



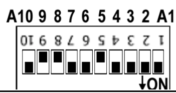
030E 1,2,3,8,9 UP



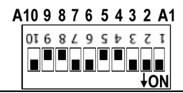
0310 4,8,9 UP



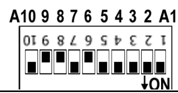
0320 5,8,9 UP



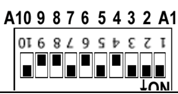
0330 4,5,8,9 UP



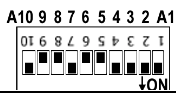
0340 6,8,9 UP



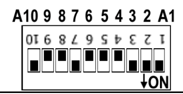
0350 4,6,8,9 UP



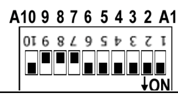
0360 5,6,8,9 UP



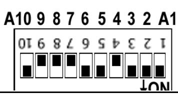
0370 4,5,6,8,9 UP



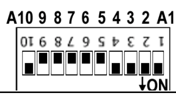
0380 7,8,9 UP



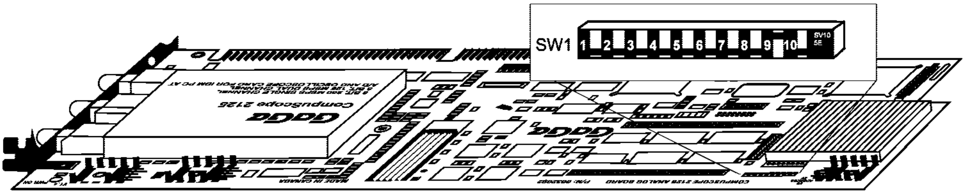
0390 4,7,8,9 UP



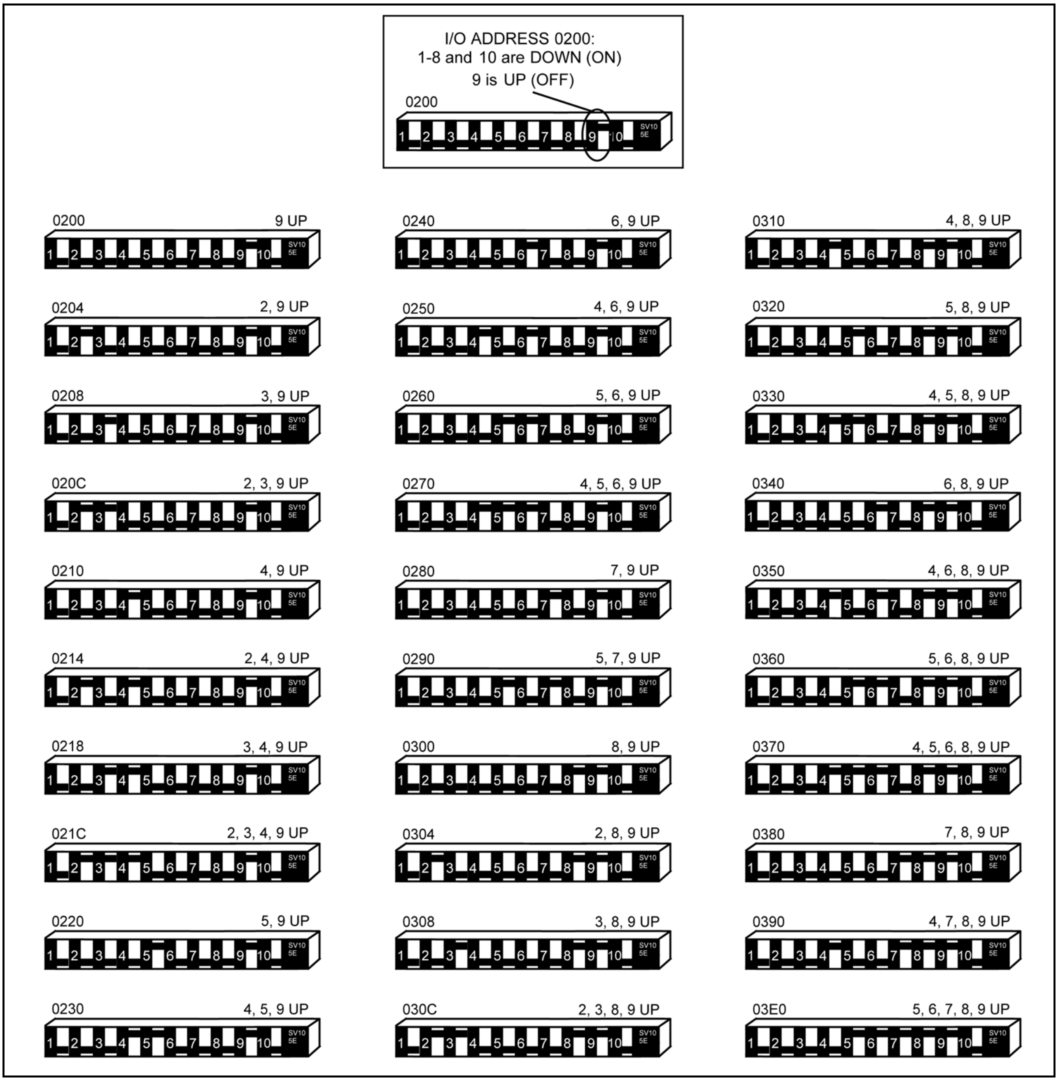
03E0 5,6,7,8,9 UP



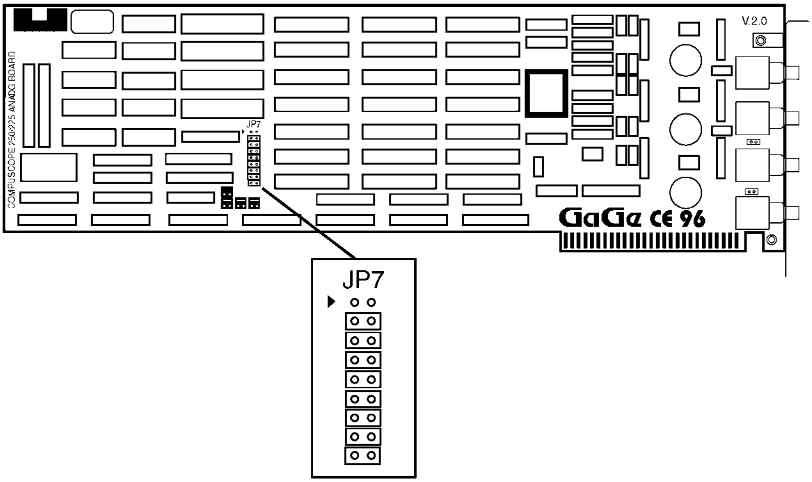
The SIP switch SW1, as shown in the diagrams below, defines the base I/O address of the CS2125 and CS265 boards.



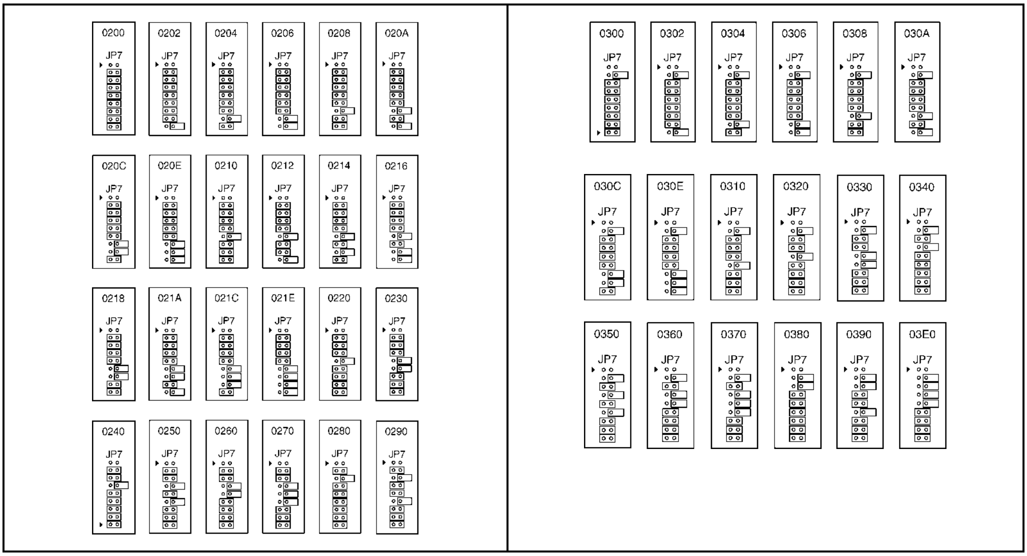
Each of the switch positions A1 through A9 corresponds to an address line A1 through A9 from the ISA bus. As the CS2125 and CS265 use two I/O locations, there is no need to configure a switch for A0. A10 is not used.


I/O ADDRESS 0200:
1-8 and 10 are DOWN (ON)
9 is UP (OFF)
0200
0200 9 UP
0204 2, 9 UP
0208 3, 9 UP
020C 2, 3, 9 UP
0210 4, 9 UP
0214 2, 4, 9 UP
0218 3, 4, 9 UP
021C 2, 3, 4, 9 UP
0220 5, 9 UP
0230 4, 5, 9 UP
0240 6, 9 UP
0250 4, 6, 9 UP
0260 5, 6, 9 UP
0270 4, 5, 6, 9 UP
0280 7, 9 UP
0290 5, 7, 9 UP
0300 8, 9 UP
0304 2, 8, 9 UP
0308 3, 8, 9 UP
030C 2, 3, 8, 9 UP
0310 4, 8, 9 UP
0320 5, 8, 9 UP
0330 4, 5, 8, 9 UP
0340 6, 8, 9 UP
0350 4, 6, 8, 9 UP
0360 5, 6, 8, 9 UP
0370 4, 5, 6, 8, 9 UP
0380 7, 8, 9 UP
0390 4, 7, 8, 9 UP
03E0 5, 6, 7, 8, 9 UP

The connector J P7, as shown in the diagram below, defines the base I/O address of the CS250 or CS225 board.

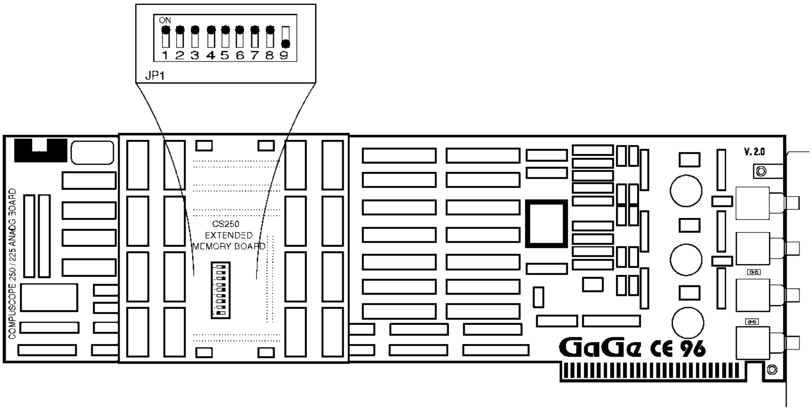


The switch position SW9 is indicated by the arrow at the top of J P7. Each of the switches SW1 through SW9 corresponds to an address line A1 through A9 from the ISA bus. As the CS250 and CS225 use two I/O locations, there is no need to configure a switch for A0.

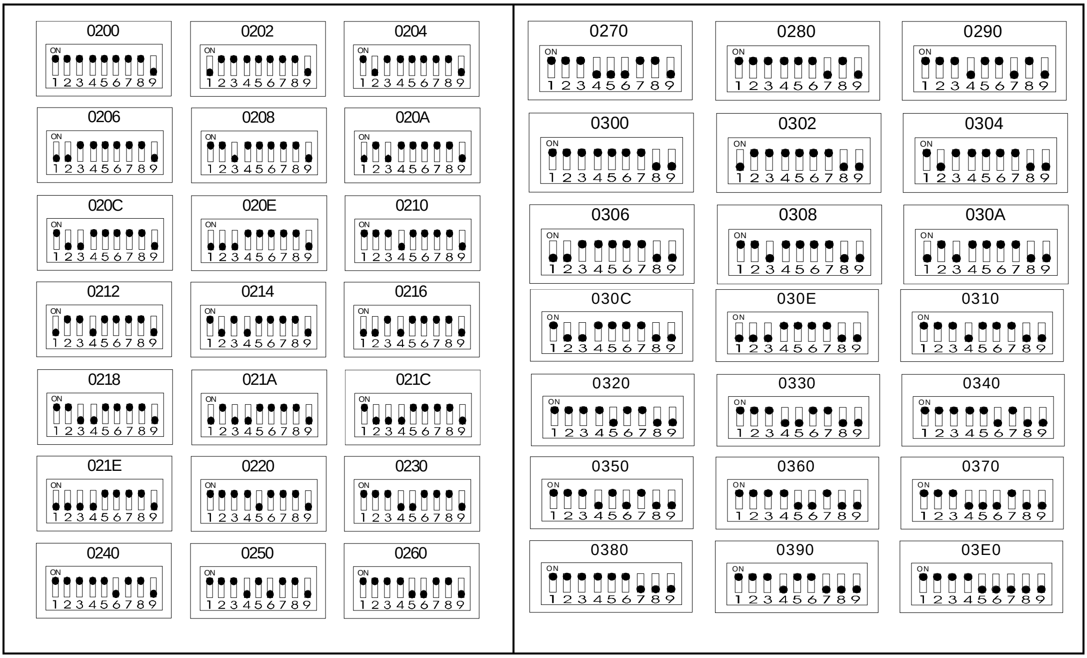


CompuScope 250 and 225 with Extended Memory

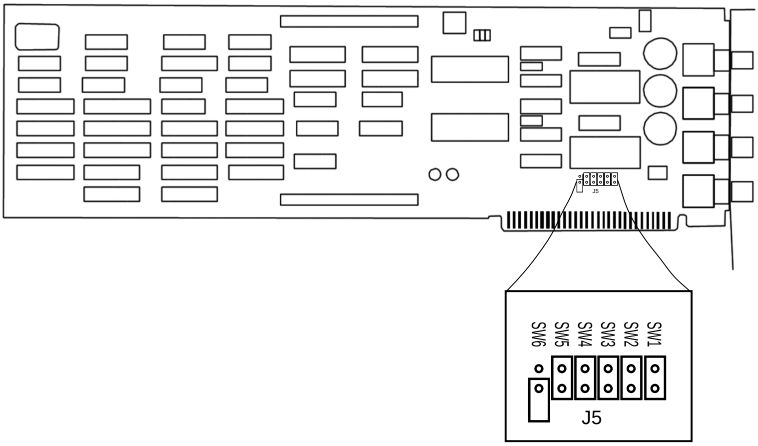
The DIP switch J P1, as shown in the diagram below, defines the base I/O address of the CS250 or CS225 board.



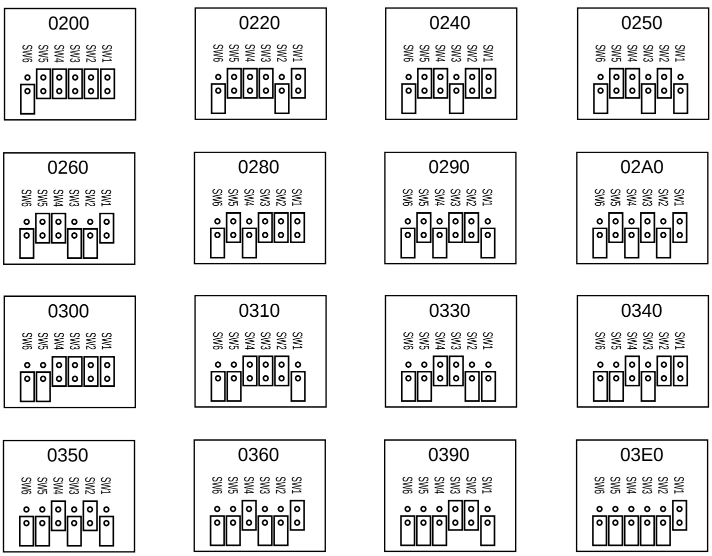
Each of the switch positions A1 through A9 corresponds to an address line A1 through A9 from the ISA bus. As the CS250 and CS225 use two I/O locations, there is no need to configure a switch for A0.


0200
0202
0204
0206
0208
020A
020C
020E
0210
0212
0214
0216
0218
021A
021C
021E
0220
0230
0240
0250
0260
0270
0280
0290
0300
0302
0304
0306
0308
030A
030C
030E
0310
0320
0330
0340
0350
0360
0370
0380
0390
03E0

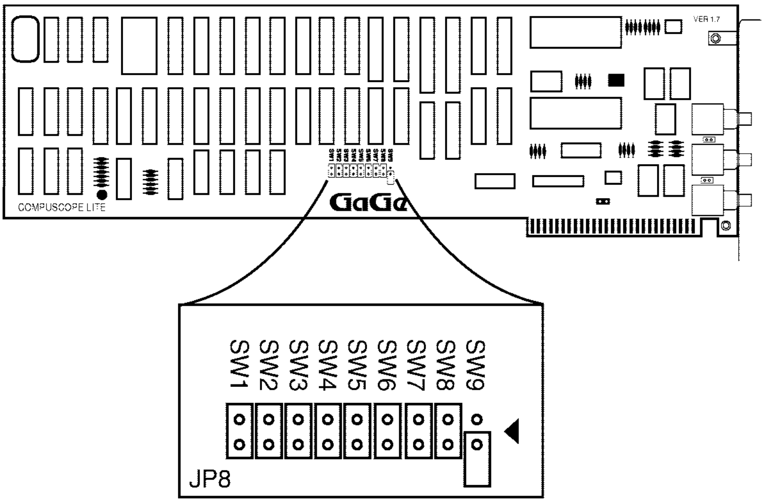
The connector J 5, as shown in the diagram below, defines the base I/O address of the CS220 board:



Each of the switches SW1 through SW6 corresponds to an address line A4 through A9 from the ISA bus. As the CS220 uses 16 I/O locations, there is no need to configure switches for A0 to A3.



The connector J P8, as shown in the diagram below, defines the base I/O address of the CSLITE board:



Each of the switches SW1 through SW9 corresponds to an address line A1 through A9 from the ISA bus. As the CSLITE uses two I/O locations, there is no need to configure a switch for A0.


0200
0202
0204
0206
0208
020A
020C
020E
0210
0212
0214
0216
0218
021A
021C
021E
0220
0230
0240
0250
0260
0270
0280
0290
0300
0302
0304
0306
0308
030A
030C
030E
0310
0320
0330
0340
0350
0360
0370
0380
0390
03E0

Appendix E: Using the Test Output Connector

If you do not have a signal generator in order to use GageScope®, you can use the test output connector found on certain CompuScope cards. These are:

- CompuScope 8012A/PCI
- CompuScope 8012/PCI
- CompuScope 6012/PCI
- CompuScope 1012/PCI
- CompuScope 512/PCI
- CompuScope 8012A
- CompuScope 8012
- CompuScope 6012
- CompuScope 1012
- CompuScope 512
- CompuScope 2125
- CompuScope 265
- CompuScope 250
- CompuScope 225

If your card is not listed here, consult your user's manual to see if it is equipped with a test output connector.

If your CompuScope card is equipped with such special options as Gated Digitization or External Clock, which make use of the test output BNC connector, you must use a signal generator to provide a test signal.

Note that the test output only works at 20 MHz or less in the CompuScope 8012A/PCI, CompuScope 8012A, CompuScope 8012/PCI, CompuScope 8012, CompuScope 6012/PCI, and CompuScope 6012.

Setting up the Test Signal

1. Set up an external probe to capture the test signal from the connector at the rear of the board.
2. Connect the probe's BNC connector to the BNC connector of the Channel A Input at the rear of the board.
3. Plug the other end of the probe into the TEST output BNC connector.

CompuScope LITE

You do not need external probes to capture the test signal from the CSLITE, since this board is equipped with an Internal Test Output.

Glossary

Aliasing

Aliasing occurs when a signal is sampled at a frequency lower than twice its highest frequency component ($2 * f_{max}$). If this happens, the resulting digital representation will be incorrect. Typically, the signal would appear to have a frequency very different from the actual. No amount of digital filtering can correct errors induced by aliasing. See **Nyquist Theorem**.

Amplitude Resolution and ENOB

The A/D converter used in a digitizing scheme typically has a fixed vertical resolution. This resolution is limited by the number of on-chip comparators for a flash converter (like the ones used on the CompuScope boards).

The **Signal to Noise Ratio (SNR)** of a digitizer system is another measure of the vertical resolution. The SNR is measured in dB and the ideal maximum is approximately 6 dB per 1 bit of resolution.

In practice, the SNR is always less than this ideal number due to aperture jitter, EMI and general system noise. The **Effective Number of Bits (ENOB)** of a digitizer system is calculated using the following equation:

ENOB = (SNR - 1.76) / 6.02

Analog Bandwidth

The analog bandwidth of a digitizer system depends upon the bandwidth of the front-end amplifier or the A/D converter, whichever is less.

The Full Power Analog Bandwidth is defined as the frequency at which the output is 3 dB below the input when the input is at its full scale.

The Small Signal Analog Bandwidth is defined as the frequency at which the output is 3 dB below the input when the input is 20 dB below the full scale.

Analog to Digital Conversion

Analog to digital (A/D) conversion is a process by which an analog signal is sampled and converted to a digital pattern of *n* bits where each *n*-bit "sample" represents the amplitude of the analog signal at a given time. This conversion is necessary as computers do not understand analog signals.

Flash A/D Converters are the most popular devices for use in a High Speed A/D Conversion system. An *n*-bit flash ADC contains 2*n* comparators. Each comparator compares the input signal with a fixed voltage level generated by a resistor ladder. The 2*n* outputs are encoded to an *n*-bit word by the on-chip encoder.

Capture Modes

GageScope® has two capture modes: Continuous and One Shot. In Continuous mode, GageScope® repeatedly waits for a trigger event, acquires and displays the data, then waits for another trigger event.

In One Shot mode, GageScope® waits for a trigger event, acquires and displays the data, then stops. This allows for unattended capture of both pre- and post-trigger data.

Channel

A channel on a CompuScope card refers to the card's input connectors, named Channel A and, if the card is equipped with two channels, Channel B.

A channel in GageScope® refers to any one of the waveforms available for processing, whether they originated from a CompuScope card, a channel file, or a mathematical formula.

Channel Mode

Most, but not all, CompuScope cards are equipped with two input channels. Dual channel mode refers to having access to both channels at the same time for data acquisition. Single channel mode refers to "shutting off" the second channel and acquiring data only from the first channel.

The advantage of single channel mode is that the sample rate is twice that of dual channel mode. For example, the CS8012A can sample at 50 MS/s in dual channel mode and 100 MS/s in single channel mode.

Continuous Capture Mode

See **Capture Modes**.

Cursors

Cursors allow the user to measure the distance between two locations on the same signal or on different signals. In GageScope®, a cursor is represented by a small **x** in the display window.

Display Method

The Display Method is the method GageScope® uses to draw traces when two or more pixels are to be displayed in the same pixel column.

In many situations, the number of screen pixels available is much smaller than the number of sample points to be displayed. For example, with screen depth set to 800x600 pixels, GageScope® might have to display 6000 points in a display window 600 pixels wide. In order to draw the waveform, GageScope® must make a choice as to which pixel to use.

See **Mean Display Method** and **MinMax Display Method**.

DLL (Dynamically Linked Library)

A DLL is a library of routines dynamically linked at run-time under Windows operating system.

Dual Channel Mode

See **Channel Mode**.

Effective Aperture Delay

The finite period of time between the rising edge of the CONVERT clock and the instant at which the sample is taken.

ENOB

Effective Number Of Bits, defined using the following equation:

ENOB = (SNR - 1.76) / 6.02

See **Amplitude Resolution and ENOB**. See also **Signal to Noise Ratio (SNR)**.

Equivalent Time Sampling

With Equivalent Time Sampling, the A/D system repeatedly samples the input at a rate $\mathbf{F_s}$, but delays sampling every time by time $\mathbf{t_d}$ such that:

$\mathbf{t_d = 1 / (N * F_s)}$.

This yields a sampling rate of $\mathbf{N * F_s}$ with the condition that, *in an Equivalent Time Sampling system, the input signal must be repetitive.*

If the input signal changes even by a very small margin for a short period of time, Equivalent Time Sampling method will either not capture the change or capture it and give an erroneous result.

See **Real Time Sampling**.

Full Power Analog Bandwidth

Full Power Analog Bandwidth is defined as the frequency at which the output is 3 dB below the input when the input is at its full scale. See **Analog Bandwidth**.

Gated Digitization

An optional hardware upgrade for CompuScope cards that allows the user to select the portion of the input signal to be digitized. When nothing is connected to the Gate, or Gate is made HIGH, data is stored in the on-board memory.

Grid

The Grid is a set of vertical and horizontal lines drawn at regular intervals in the display window. The resulting grid has 8 squares from top to bottom and 10 squares from left to right (although this can be changed by the user). Each square is counted as a “division” and is used as a reference in other parts of GageScope® (such as the Position setting in the Parameter tab of the Channel Control). The grid is a permanent fixture in the display window and is not affected by movement of the scroll bar.

Input Range

The Input Range is the range in Volts that GageScope® uses to acquire data. The current range of a signal can be measured by eye using the Display Window grid—by default, each grid square measures 1V high.

Inter Sample Delay

The time that must elapse between two successive high speed data captures. This feature is important for long term monitoring of signals.

Interpolation

A signal might become pointed or jagged when the number of samples per grid square becomes very small. When GageScope® performs interpolation on a signal, it uses a mathematical formula to add extra sample points in order to smoothen its appearance. See **Spline Interpolation**.

ISA and PCI Bus

The ISA bus is the computer bus found inside all IBM PC AT, 386, 486 and Pentium systems. It is by far the most popular 16 bit bus in the industry and has become the standard for PC-based data acquisition and instrumentation. With a standard 8 MHz clock speed, the ISA bus can transfer data with a throughput of up to 1.5 Megawords per second.

The PCI bus is a processor-independent, 32 bit local bus featured on almost all computers built around the Pentium, PowerPC and Alpha microprocessors. With its 33 MHz clock speed, PCI has the capability of burst transfer rates as high as 133 MB/s. This number can be doubled if the 64 bit extension of PCI is used.

Mean Display Method

When two or more samples are to be displayed on the same pixel column, Mean averages all the data points in order to display one single point. This is useful when the number of samples per grid square becomes very large (such as when the timebase is increased).

Mean increases the **Effective Resolution** when capturing and displaying low frequency signals at high sample rates, i.e., when a signal is being oversampled.

See **Display Method**.

Megasamples

A Megasample is 1,000,000 **Samples**.

Memory Depth

The A/D data has to be stored in digital memory so the microprocessor can read it at a later time. The amount of memory available for this purpose is called the memory depth of the A/D system. See **On-board Memory**.

MinMax Display Method

When two or more samples are to be displayed on the same pixel column, MinMax plots the minimum and maximum values, then draws a line between them. This method allows glitch detection even at very low timebases.

See **Display Method**.

Multi-Card System

A Multi-Card System is a system with multiple CompuScope or CompuGen cards that can be used with a common clock and trigger, to provide multiple input or output channels at very high speed.

Multiple Record

A feature supported by a number of CompuScope cards which lets the user take advantage of the very deep on-board buffers.

The user is able to set the number of samples to be captured after a trigger, but, unlike the normal capture sequence, the CompuScope card acquires the specified number of points and then re-arms itself to acquire more data after another trigger. Data collected from each successive acquisition is "stacked" on top of the previous acquisition until the on-board buffer fills up.

For example, if a post-trigger depth of 1024 points is specified, the first acquisition stores data in addresses between 0 and 1023, the next acquisition from 1024 to 2047, and so on until the buffer is full.

This feature is invaluable in applications in which trigger events are occurring rapidly or unpredictably, and A/D down-time has to be minimized.

Nyquist Theorem

The Nyquist Theorem states that a function, v(t), whose highest frequency component is f_{max}, must be sampled at a frequency F_s such that

$$\mathbf{F_s = 2 * f_{max}}$$

In other words, if an A/D device samples the input signal at a maximum frequency of F_s, the signal frequency can not be determined unless it is guaranteed that the maximum frequency component of the input signal is half the sampling frequency (**$0.5 * F_s$**). For example, in the case of CompuScope 250, F_s is 100 MHz, therefore the input signal must not have frequency components higher than 50 MHz.

Note: To reconstruct the signal in the time domain, the sample rate should be at least five times the maximum frequency component **($F_s = 5 * f_{max}$)**.

On-board Memory

In CompuScope cards, on-board memory refers to the amount of high-speed memory the card has. On-board Memory is measured in **Megasamples**.

As an ultra-fast data acquisition device, the CompuScope card can convert analog signals must faster than the resulting data can be transferred to the PC's host memory. To solve this problem, the CompuScope card can be instructed to store the data in its on-board memory, until such time that the data can be transferred later.

The amount of this high-speed memory determines the **Record Length** of data capture, i.e., how long a snapshot the CompuScope can take.

See **Sample Rate**, **Record Length**, and **ISA and PCI Bus**. See also **Memory Depth** and **Samples**.

One Shot Capture Mode

See **Capture Modes**.

PCI Bus

See **ISA and PCI Bus**.

Points

See **Samples**.

Polarization

Polarization refers to how the trace is displayed relative to the X axis. When polarization is normal, the channel is displayed exactly as captured. Inverting the polarization inverts the signal about the X axis for purposes of the display, without affecting the actual data.

Post-trigger Data

Post-trigger data is the data acquired after a **trigger event** occurs. Post-trigger data is located to the right of the **trigger marker**.

Pre-trigger Data

Pre-trigger data is the data acquired before a **trigger event** occurs. Pre-trigger data is located to the left of the **trigger marker**.

Real Time Sampling

Real Time sampling, as used in the CompuScope cards, is defined as the sampling of the input waveform every **T** seconds, such that the sampling rate **Fs** is equal to **1** / **T**.

This is the best method of digitizing any signal, be it transient or continuous. Samples are taken on every clock cycle and stored in a memory buffer. The signal is then recreated in software with a guarantee that no transient event which satisfied the Nyquist Criterion has been missed.

See **Equivalent Time Sampling**.

Record Length

The total amount of data that can be stored in a CompuScope card's on-board memory, measured in units of time. Record Length is defined as:

On-board Memory Depth (samples) / Sample Rate (MS/s)

For example, a CS250 with 32K memory sampling at 100 MS/s provides a record length of:

(32,768 bytes) / (100,000,000 8-bit samples / second)
(32,768 bytes) / (100,000,000 bytes / second)
(32,768 / 100,000,000) seconds
327.6 microseconds

See **Memory Depth** and **Sample Rate**.

Sample Rate

The sample rate is the number of times per second analog signals are converted to digital signals. The sample rate is measured in Megasamples per Second (MS/s). The timing is controlled by an internal or external clock.

See also **Record Length** and **Memory Depth**.

Samples

A sample is a single value measured in Volts converted at a single point in time from an analog signal. A sample is also referred to a sample point or a point.

Note that the terms samples and bytes, or megasamples and megabytes, **cannot** be used interchangeably. Although it is true that a sample acquired through an 8-bit CompuScope card takes up 1 byte, a sample acquired through a 12-bit card (such as the CS8012) or 16-bit card (such as the CS1016) takes up **2 bytes**. Furthermore, a megasample is equal to 1,000,000 samples, while a megabyte is 1,048,576 bytes.

SIG File

A SIG file is the signal file format shared by GageScope® and GageScope® for DOS. SIG files record the voltage levels vs. time as recorded during a data acquisition. SIG file are in binary format so as to keep the files as small as possible.

Signal to Noise Ratio (SNR)

Signal to Noise Ratio (SNR) is a frequency domain parameter measured at specific input frequencies. Using FFT analysis, the power spectrum of a known input sine wave is calculated. The contents of all frequency bins containing frequencies other than the fundamental (input frequency) and its harmonics are added. The ratio of the content of the fundamental frequency bin to this sum is the SNR of the system.

This parameter is more useful for specifying the performance of a high-speed A/D device than the conventional DC accuracy figures used for oscilloscopes.

Also see **Total Harmonic Distortion** and **Effective Number Of Bits**.

SINAD

SINAD is the ratio of the power in the fundamental bins to all other bins, including harmonics.

Single Channel Mode

See **Channel Mode**.

Slope

See **Trigger Slope**.

Small Signal Analog Bandwidth

The **Small Signal Analog Bandwidth** is defined as the frequency at which the output is 3 dB below the input when the input is 20 dB below the full scale. See **Analog Bandwidth**.

SNR

See **Signal to Noise Ratio (SNR)**.

Software Trigger

A trigger event issued by GageScope® when a timeout limit is reached.

Spline Interpolation

When GageScope® performs Spline Interpolation, intermediate sample points are added to a signal in order to smoothen its appearance. See **Interpolation**.

Timeout

The timeout limit is the amount of time GageScope® waits for a trigger event. A timeout occurs when the timeout limit is reached before a trigger event occurs. In the event of a timeout, GageScope® issues a **software trigger**, i.e. it triggers at the moment of timeout.

Total Harmonic Distortion

Total Harmonic Distortion (THD) is measured in a similar fashion to SNR (**Signal to Noise Ratio**), except for the fact that in the analysis, the fundamental is compared to the size of the sum of the harmonics. If some of the harmonics lie outside the **0** to $\mathbf{F_s / 2}$ range, it is the aliased versions of the harmonics that are seen in the base band.

Trace

A Trace refers to the actual line displayed to represent a waveform in GageScope®.

Trigger Event

A trigger event is the moment when a trigger is detected by GageScope®, causing it to begin a data acquisition. For example, if the **trigger source** is set to channel 1, and the **trigger level** is set to 600 mV, then a trigger event will only occur if the input signal of channel 1 rises past 600 mV. The detection of a trigger is affected by a variety of settings in GageScope®.

See **trigger level** and **trigger source**. Also see *Basic Concepts* of the Triggering tutorial.

Trigger Level

The trigger level is the level in Volts a signal must cross in order for a trigger event to occur. For example, if the trigger is set to 500 mV, and the signal reaches 400 mV, a trigger event does not occur. But if the signal rises to 600 mV, the signal will have crossed the trigger level, invoking a trigger event and subsequent data acquisition.

In the display window, the trigger level is represented by a slope icon to the far left of a channel, with a dashed horizontal line connected to it.

Trigger Marker

The vertical line with a small **T** in the display window marking time-zero in a data acquisition. The trigger marker points to when the trigger occurred. Data to the left of the trigger marker is **Pre-trigger Data**; data to the right is **Post-trigger Data**.

Trigger Source

The source GageScope® looks at when determining if a trigger event has occurred. In GageScope® the choices are input signals (channel 1 and channel 2), External Trigger, and **Free Run**.

Trigger View

Trigger View is a powerful feature of GageScope® software which allows the user to view the signal being captured by a CompuScope card in real-time. This feature differentiates Gage products from all other PC based and stand-alone digital oscilloscopes.

Vertical Resolution

Vertical resolution is the number of bits with which an A/D system can estimate the amplitude of the analog signal. It must be noted that the vertical resolution is not the same as the **Effective Number of Bits**.

X Axis

The X axis is a dashed horizontal line that divides the display window into two equal parts. The X Axis is a permanent fixture in the display window and does not move with changes to the vertical scroll bar.

X-Y Mode

X-Y mode allows channel display on both the X and Y axes, with up to 4096-point, real-time display for channels from the same board, or unlimited X-Y display in post-processing mode.

Y Axis

The Y Axis is a dashed vertical line that divides the display window into two equal parts. The Y Axis is a permanent fixture in the display window and does not move with changes to horizontal scroll bar.

Zero Line

The zero line of a channel is the dashed horizontal line that runs through the middle of the signal. The color of a Zero Line is always set to the same color as its channel.

Index

A

B

C

D

E

F

G

Q

R

S

T

V

W

X

Y

Z