

Tutorials

About the Tutorials

This chapter consists of seven tutorials. You have selected the one that is highlighted in red:

- Tutorial 1 - Fundamentals
- Tutorial 2 - Working with Channels
- Tutorial 3 - Triggering
- **Tutorial 4 - Working with Special Modes**
- Tutorial 5 - Working with Subchannels
- Tutorial 6 - Arbitrary Waveform Generator
- Tutorial 7 - Using the Graphical Waveform Editor

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 4: Working with Special Modes

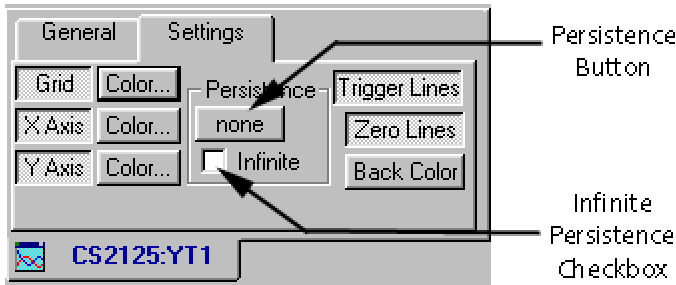
This tutorial deals with three special modes available in GageScope®:

- Persistence Mode — page 2
- SuperRes Mode — page 11
- Multiple Record Mode — page 17
- Scope Mode — page 34
- Decimation Mode — page 43


Persistence Mode

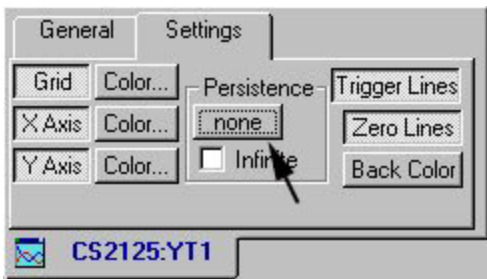
The **Persistence** setting allows the user to control the persistence of the signal display in the Display Window. By default, the signal does not persist in the Display Window. However, with the special Persistence Controls in GageScope[®], you can specify any degree of persistence that you wish, from 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**.



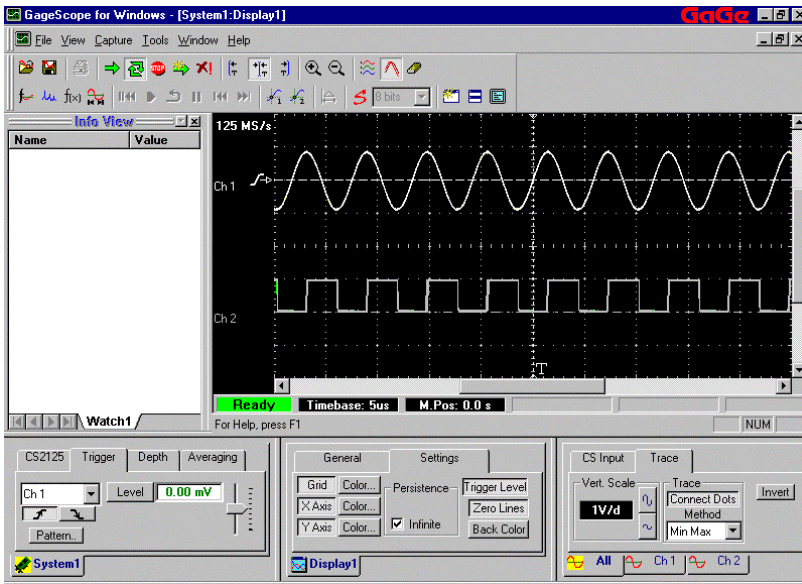
- 3 The Persistence field appears.



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



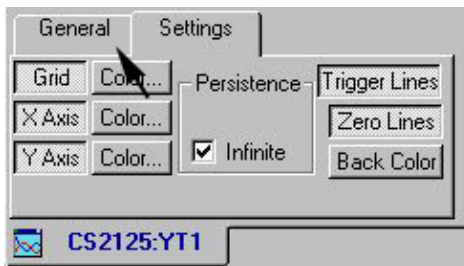
- 5 Our signals are now persisting on the screen. You may notice the signals getting slightly thicker.




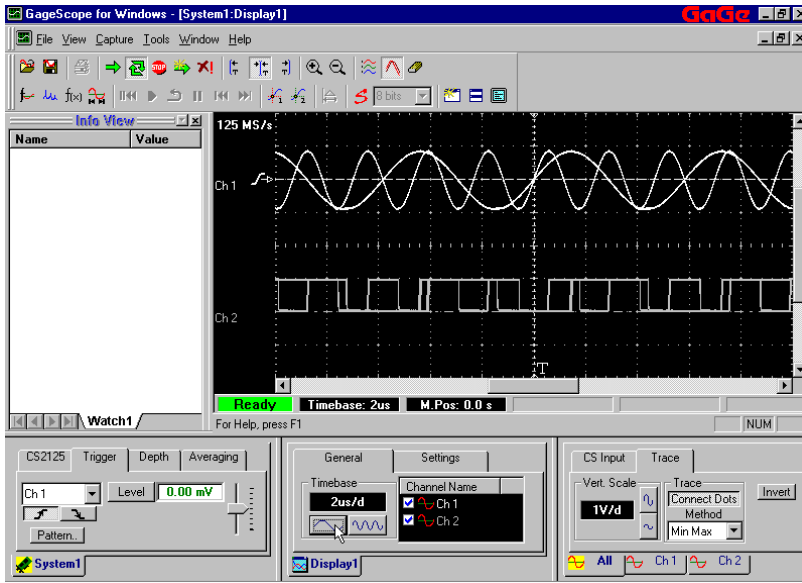
However, to truly see Persistence at work, we will change some settings that will affect the signal display.


Change the Timebase to 1 us/d

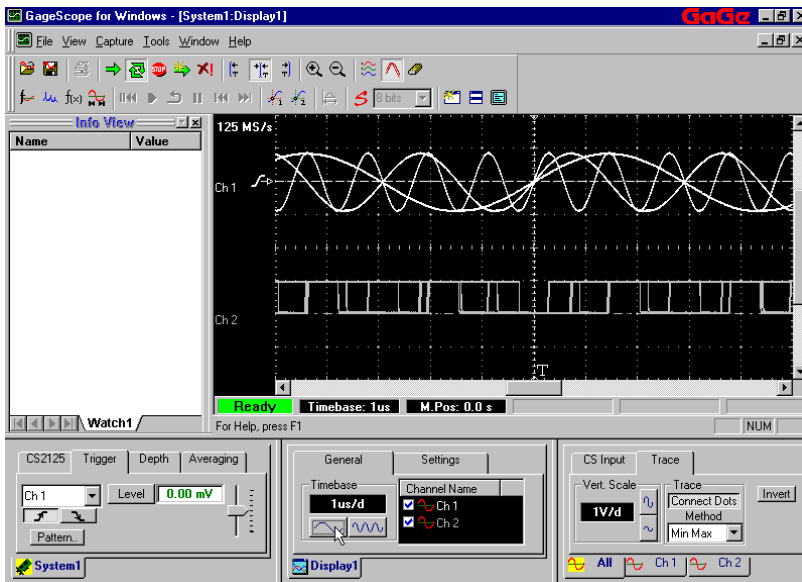
- 1 Click on the **General** tab of the Display Control to access the **Timebase** buttons.



- 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.

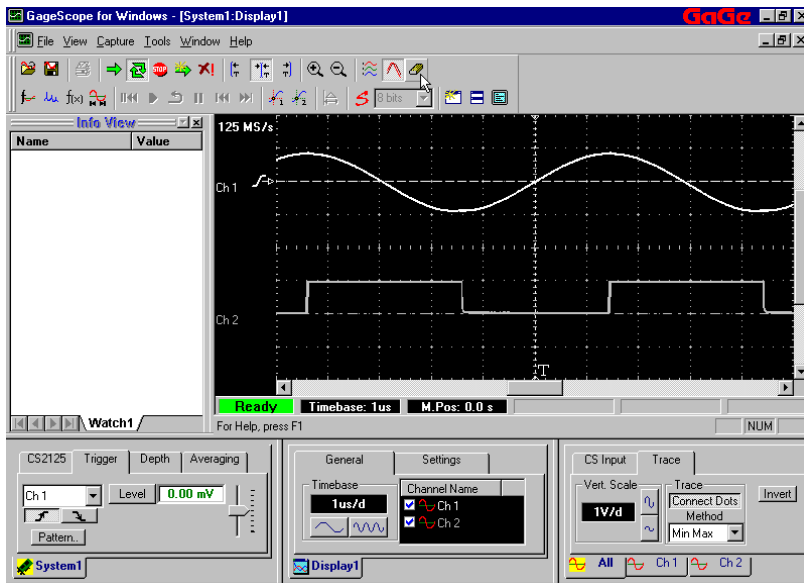


Erase the Display

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

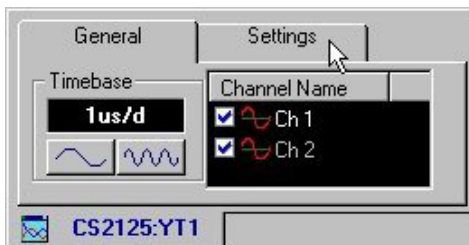


- 2 Click on  to clear the previous signals. The latest signal with a timebase of 1 μ s/d is the only one in the Display Window now.

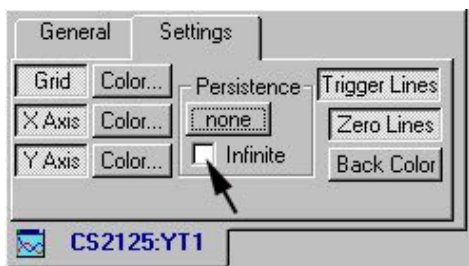


Set Persistence to 5 seconds

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



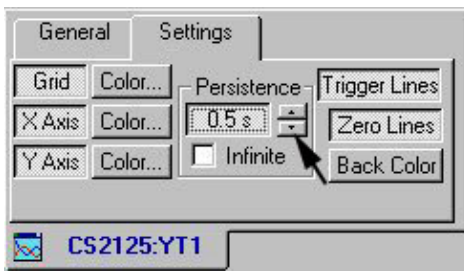
- 2 Turn off Infinite Persistence by clicking on the checkmark next to **Infinite**.




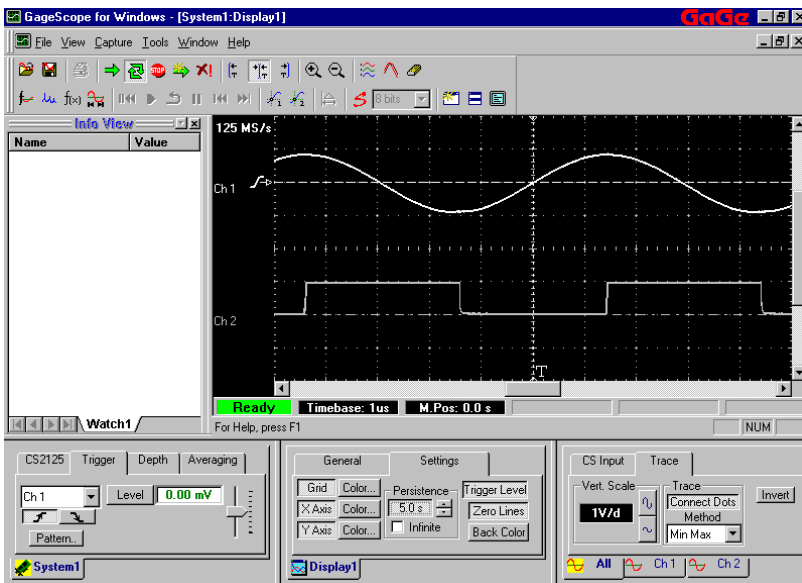
- 3 Turning off Infinite Persistence turns off Persistence altogether. To enable Persistence again, click on **None**.



- 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.




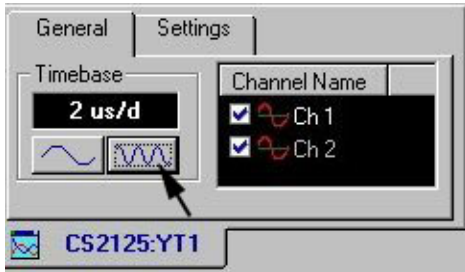
- 5 Click and hold the  button until the Persistence is set to **5.0s**.



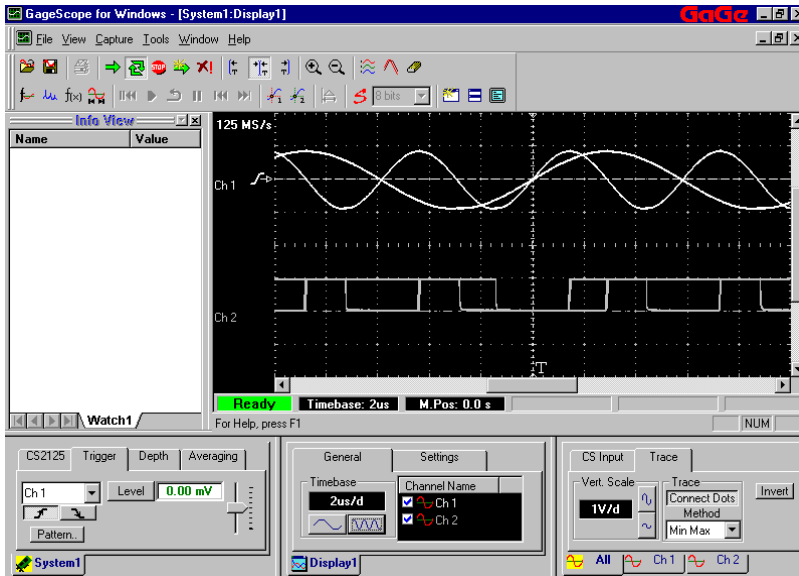
- 6 Now our signals will persist, but only for 5 seconds. To test this, we will again go to the **General** tab of the Display Control.



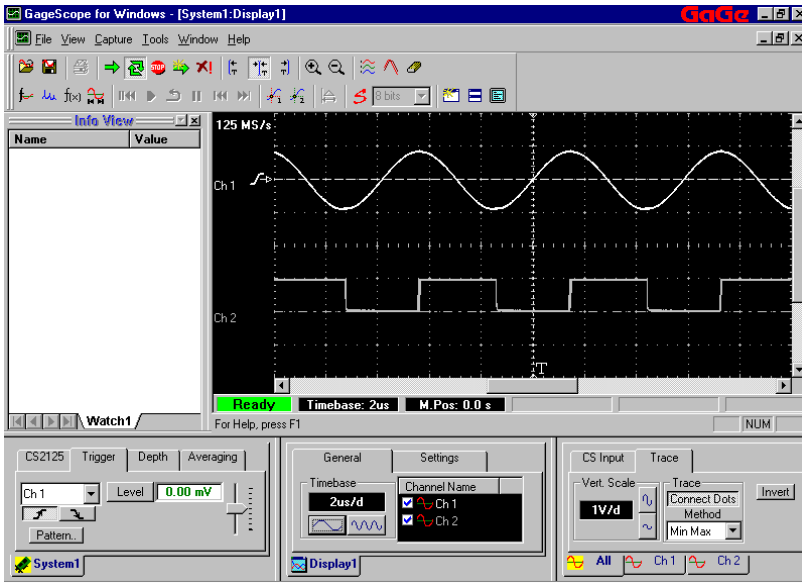
- 7 Change the Timebase to **2us/d** by clicking .



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



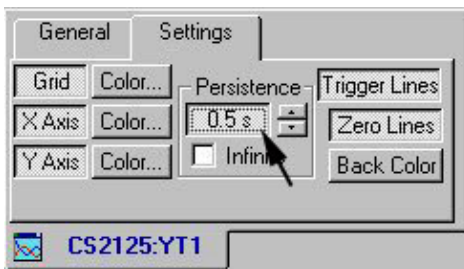
- 9 The display is erased after a short time, so that only the latest signal is preserved.




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  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.

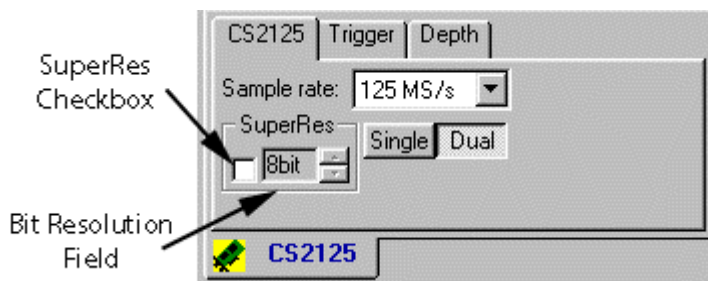
SuperRes allows you to specify the desired “Effective Resolution.” For example, you can specify 12 bit resolution even if you are using the 8 bit CompuScope 8500. GageScope® will automatically perform digital filtering (averaging of oversampled data) to enhance the effective resolution at the expense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

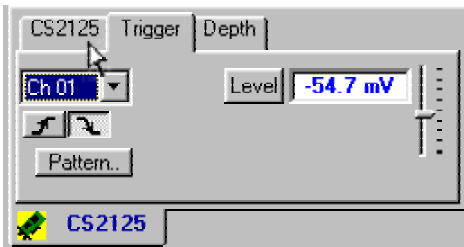
SuperRes is located in the CS tab of the System Control.



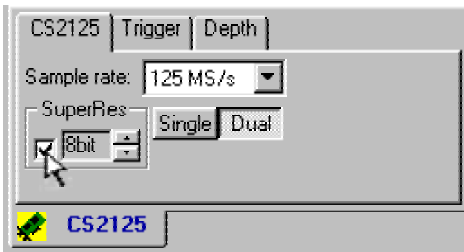
Enable SuperRes Mode

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

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

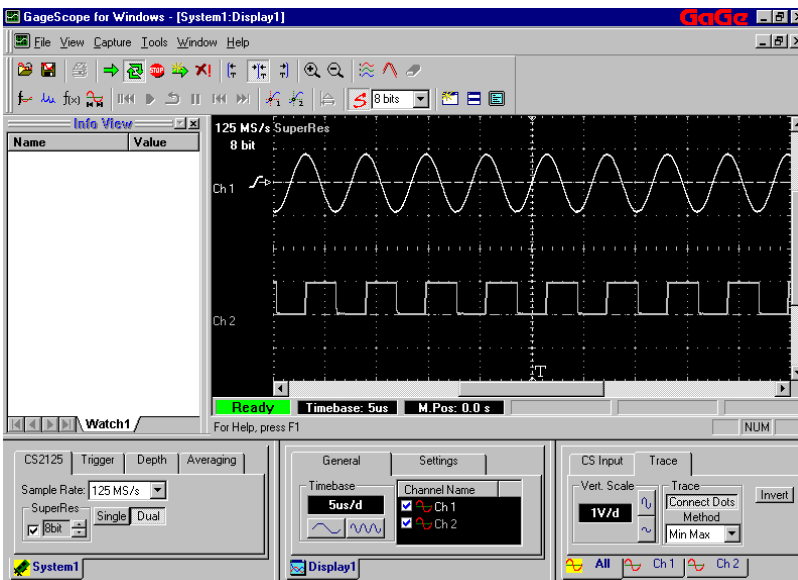


- Click on the **SuperRes** checkbox to enable SuperRes mode.



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

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

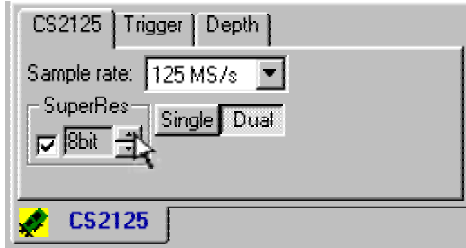


Increase the Bit Resolution by 6 or 7 Bits

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

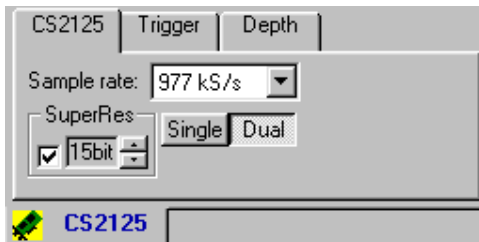
- 8 bit CompuScope cards: Increase the bit resolution to 15 bits
- 12 bit CompuScope cards: Increase the bit resolution to 18 bits

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

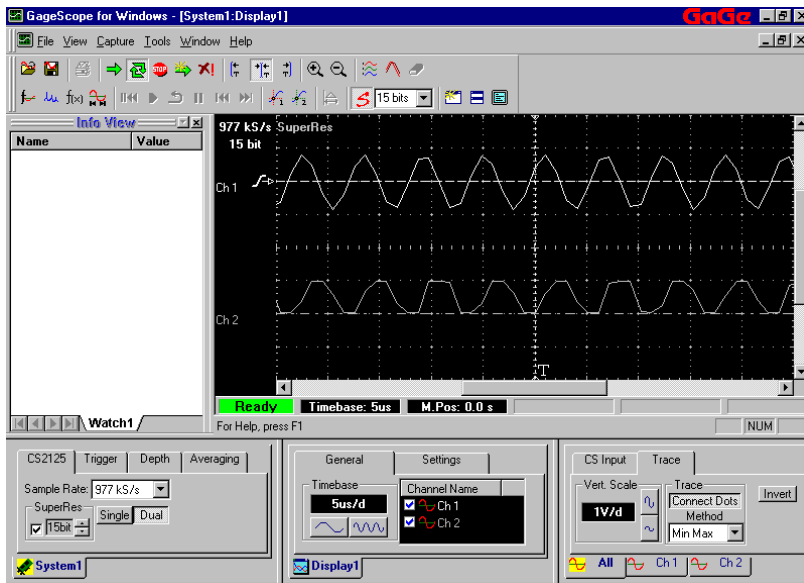


You can also click on the SuperRes drop-down menu in the toolbar.

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.




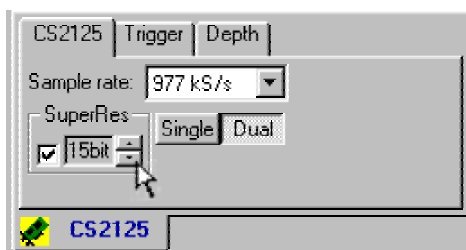
- 3 With such a comparatively low sample rate, our signal display becomes jagged.



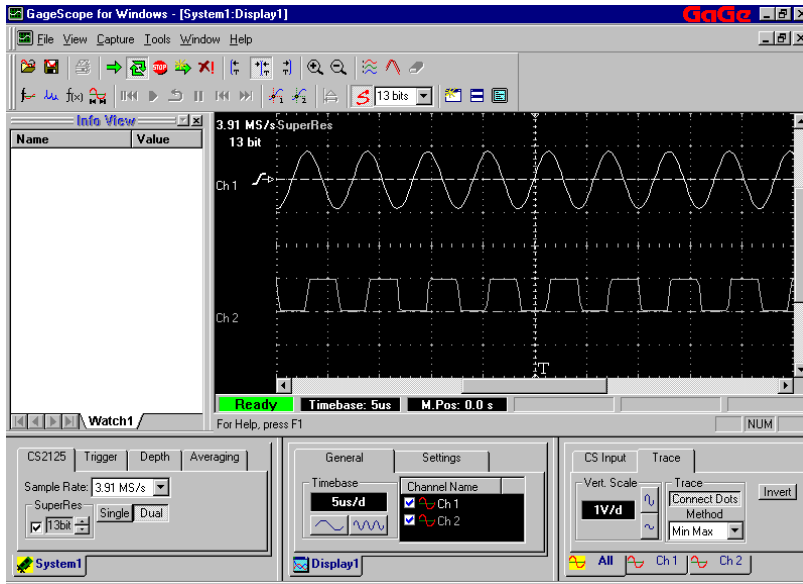
When the sample rate is low, the CompuScope card is acquiring fewer sample points per second. With fewer points available for building a representation of the signal, the resulting waveform looks jagged. Keep in mind that this illustration uses a slow capture rate. The true power of SuperRes is more 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 .




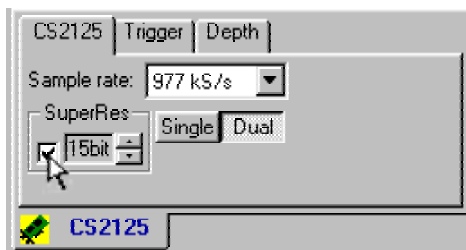
- 2 Notice that GageScope[®] automatically raises the sample rate to the maximum rate corresponding to the selected bit resolution—in our case, the sample rate rose to 3.91 MS/s when we decreased the bit resolution to 13 bits.



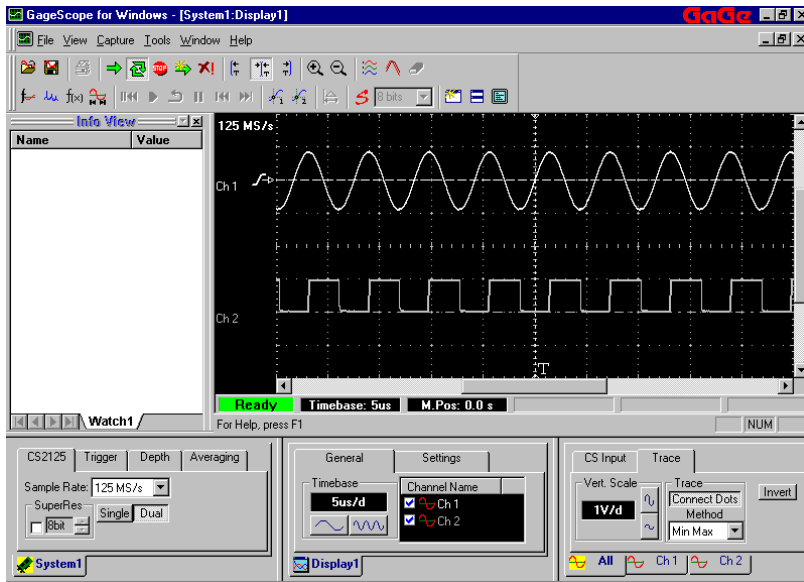
The signal looks less jagged than before, since more samples are being acquired per second.

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.

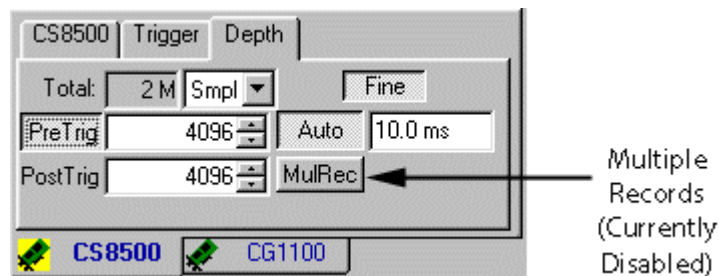
When Multiple Record Mode is enabled, the CompuScope card looks for a trigger event, acquires post-trigger data, then automatically re-arms itself to look for another trigger event. Data collected from each successive acquisition is "stacked" on top of the previous acquisition until the on-board buffer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

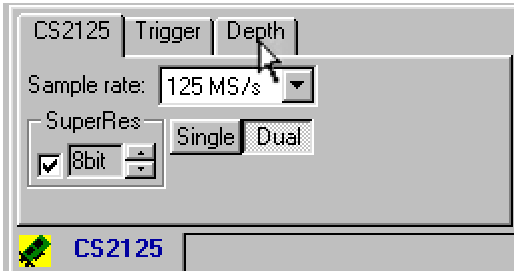
- The number of records you can capture will depend on the amount of on-board memory your CompuScope card has, as well as other settings, such as the post-trigger depth and channel mode.
- With all currently available CompuScope cards, you can only capture post-trigger data in Multiple Record mode. When 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.

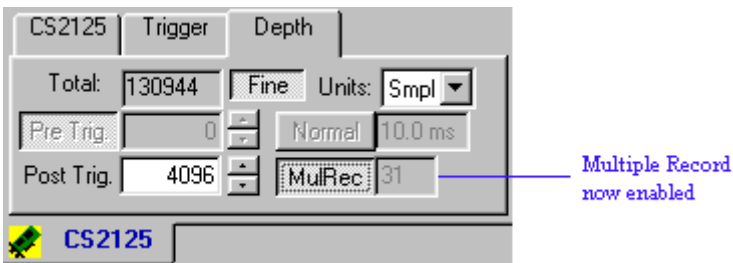


Perform a Multiple Record Capture

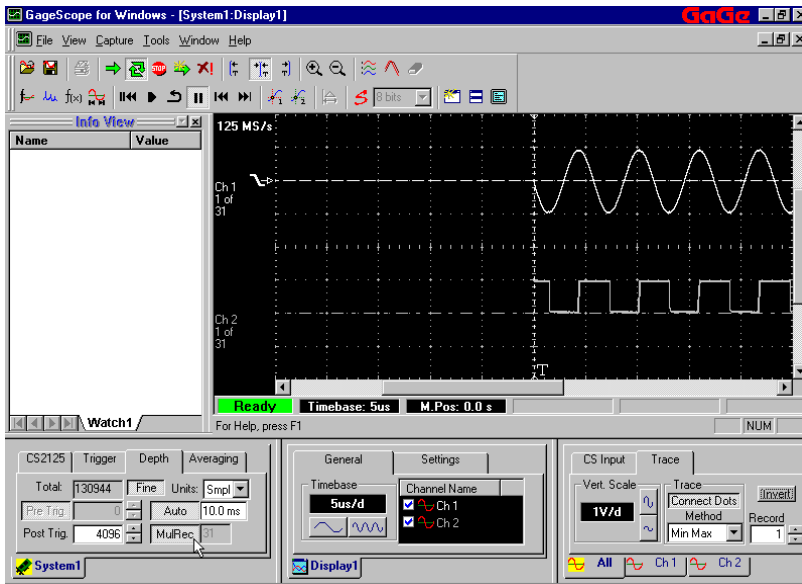
- 1 Make sure you are in **Continuous Capture** mode (the  button in the Toolbar should be “pressed in”).
- 2 Click on the **Depth** tab of the System Control to access the Multiple Record controls.



- 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.”

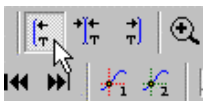
- 5 While you are performing a Multiple Record acquisition, you may briefly see the message “Transferring” in the Status bar.

Transferring Timebase: 10us M.Pos: 50.0us

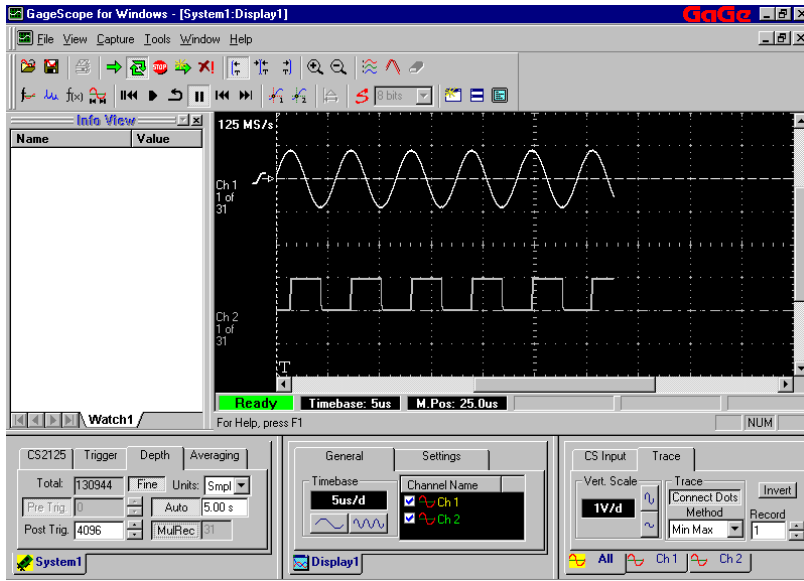
This means that GageScope® is transferring data from the CompuScope board to the internal buffers of the application.

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.



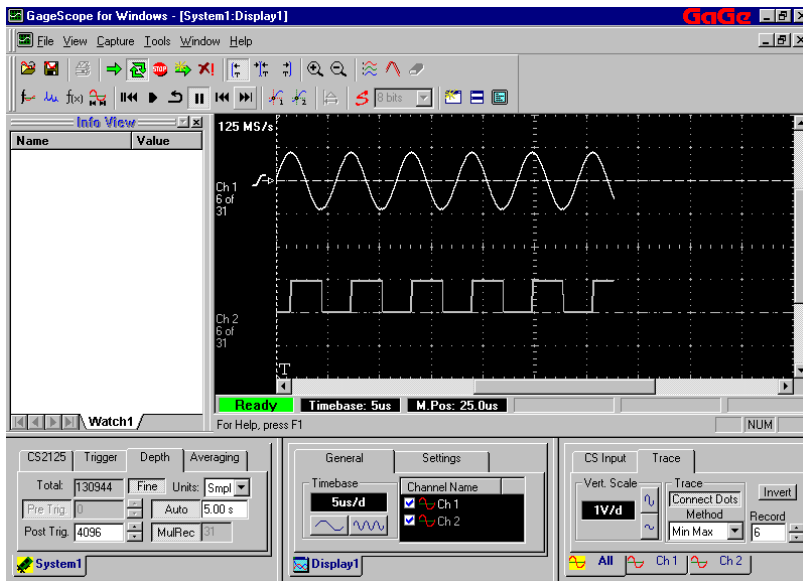
- 2 You should now be able to see the entire record.



- 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.

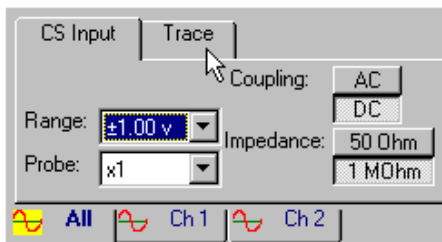




- 5 Click the **Go to Previous Record** button in the toolbar to scroll backward through your records until you get back to Record 1.

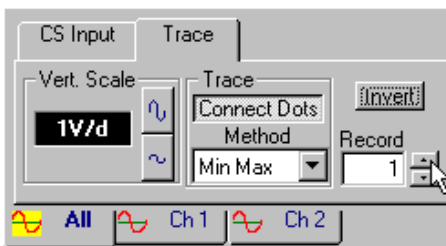


Scroll Through the Multiple Record Data via the Record Counter

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

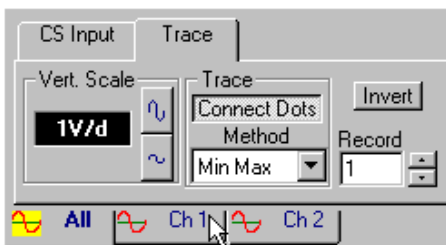


- 2 Use the  and  buttons to perform the same function as the **Go to Next Record** and **Go to Previous Record** buttons on the Toolbar.

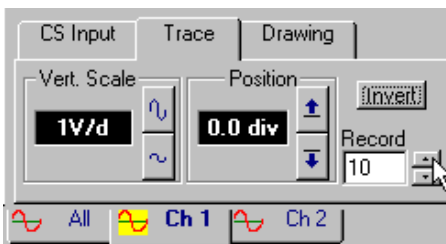


- 3 With these buttons, you can cycle through the records of all channels at once (if the All Channels Tab is current), or through the records of individual channels (if an individual channel tab is current).

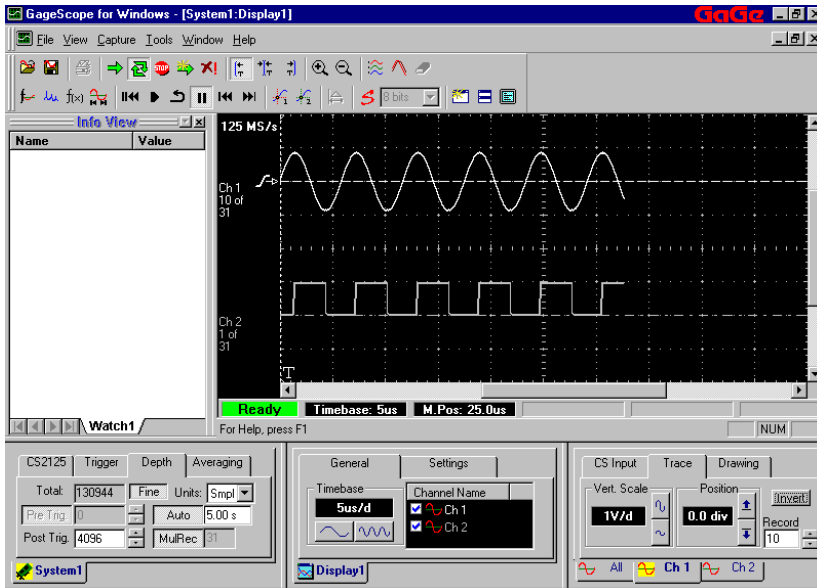
Click on the **Ch 1** tab of the Channel Control.



- 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.

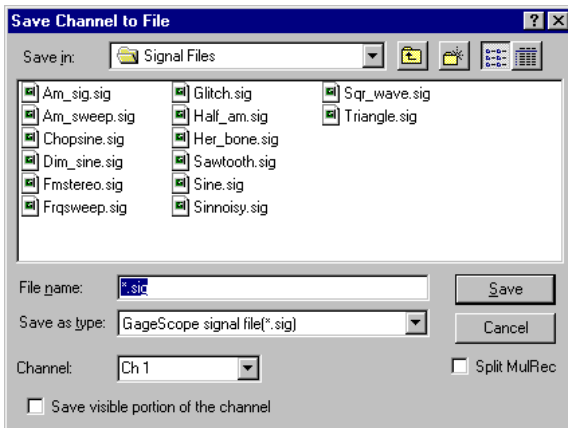


Save Channel 1 to Individual Files

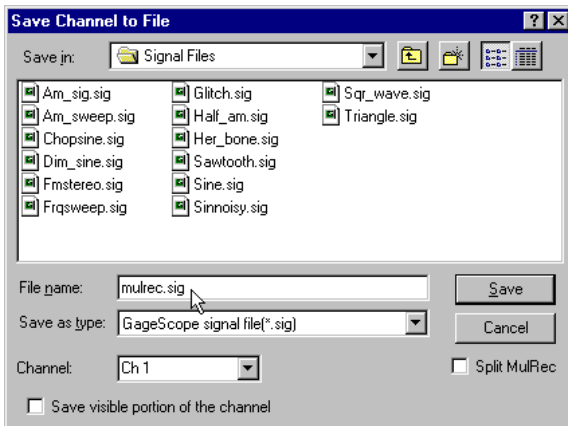
- 1 You can save a Multiple Record channel just as you would any other channel. However, there are a few options available for saving Multiple Record files which are not available with normal acquisitions. In this section, we will be saving each record in the acquisition to a separate file in one step.

Click **Save Channel**  in the toolbar.

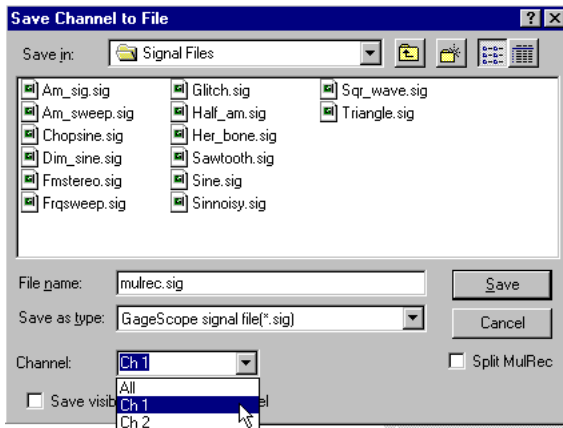
- 2 The Save Channel dialog appears.



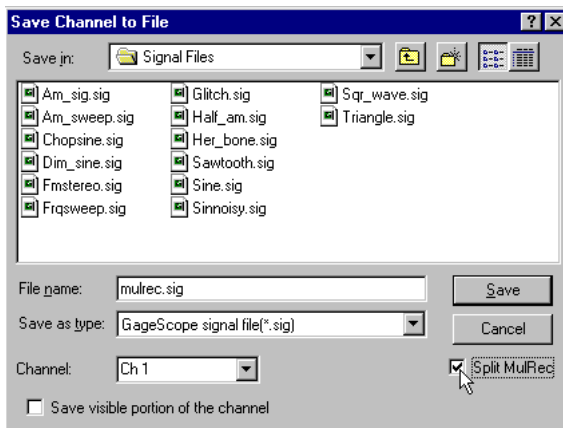
- 3 In the File Name field, type **mulrec.sig**.



- 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.

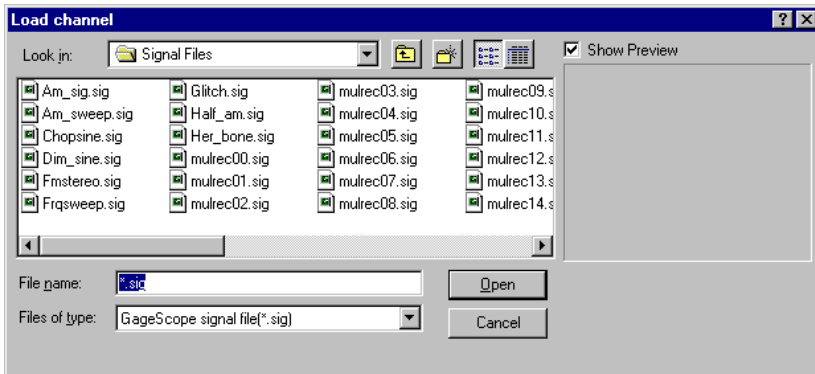


- 6 Click **Save**.

◆ Load one of the Saved Records to Ch 3

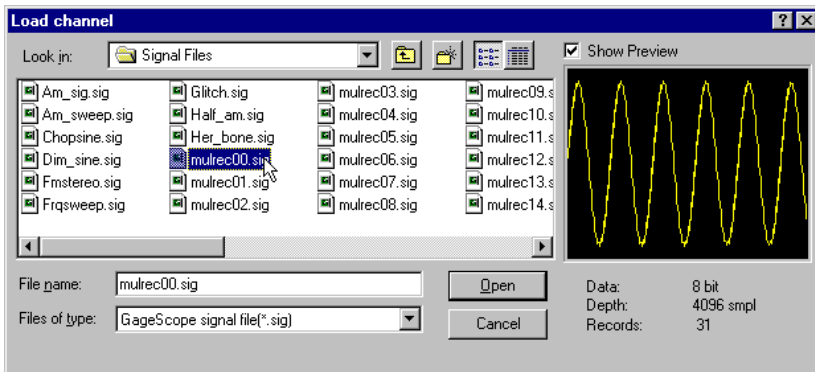
- 1 You can also load a Multiple Record channel as you would any other channel—by clicking  on the toolbar.

2 The Load Channel dialog appears.

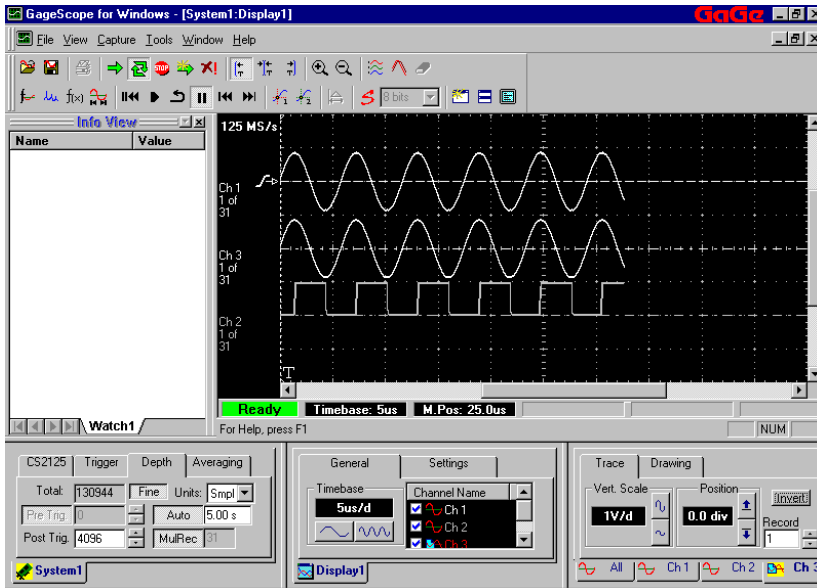


Notice that since we selected **Split MulRec** when saving our Multiple Record acquisition, and in our example there were 31 records in the acquisition, we now have 31 files named “mulrecxx.sig,” where xx is a suffix attached by GageScope®, beginning at 00.

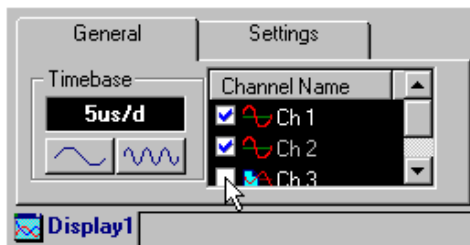
3 Click on the first of these files to select it, and then click Open.



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



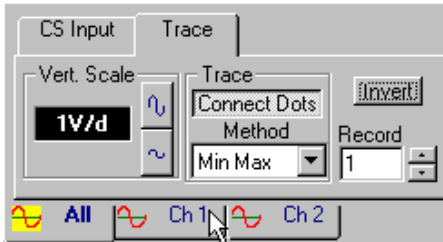
- 5 Hide Ch 3 by clicking the checkmark next to it in the Channel List (in the General tab of the Display Control).



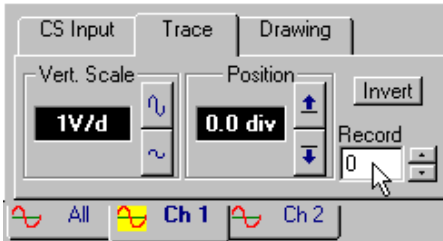
Look at All Records of Channel 1 Simultaneously

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

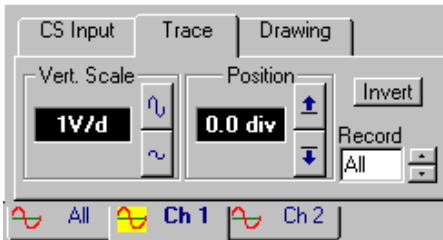
Click on the **Ch 1** tab of the Channel Control to bring it to the front again.



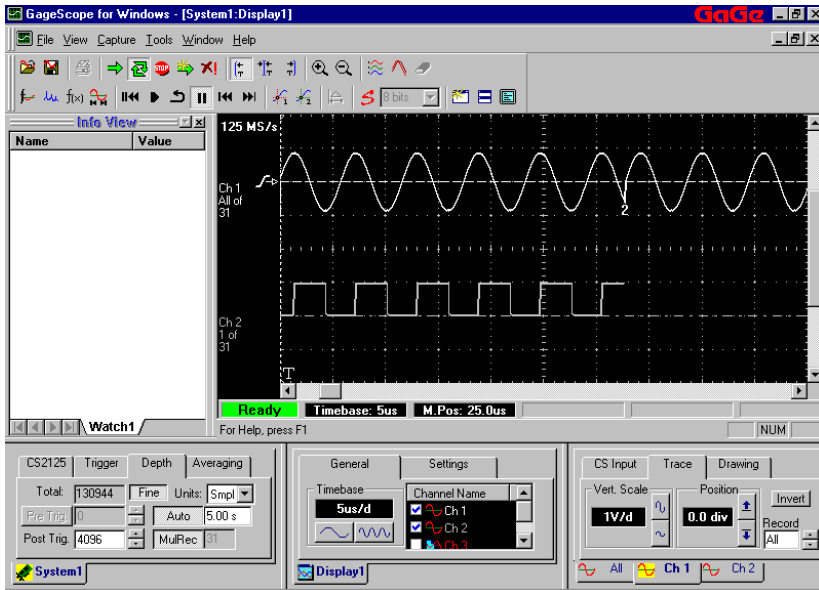
- 2 Click with the left mouse button in the **Record** field and press the Backspace to erase the value that's there. Then type **0** (zero).



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



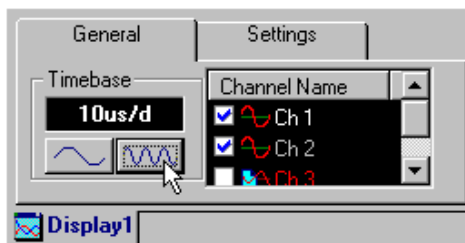
- 4 The display will now show all of your records at once. To the left of the slope icon and below “Ch1” on the screen below, notice the message “All of 31,” indicating that all 31 records are being displayed.



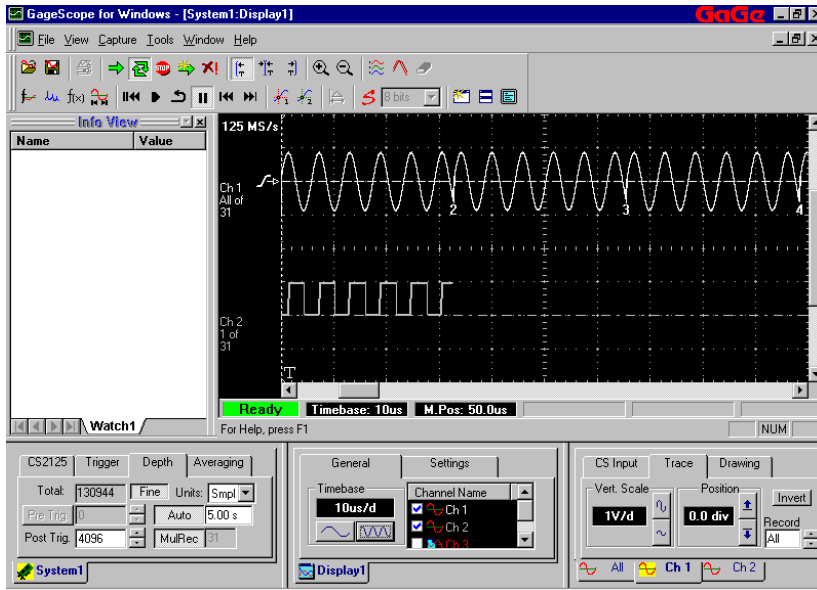
Beneath the start of each record in the signal, its record number is displayed.

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.



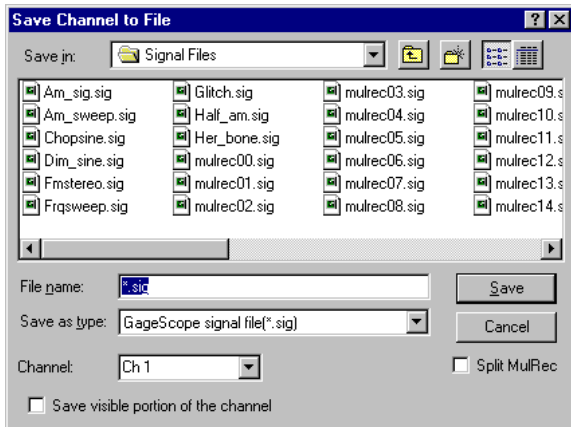
- 6 This allows you to see more records.



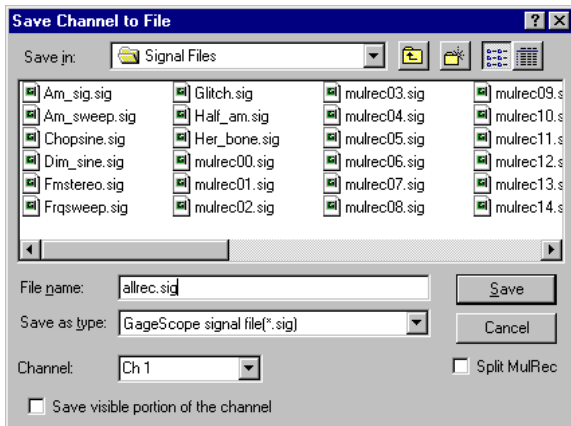
► Save All Channel 1 Records to One File

- 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.

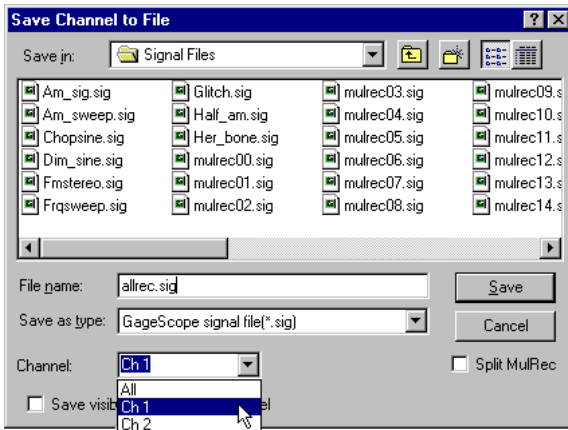
- 3 The Save Channel dialog appears.



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




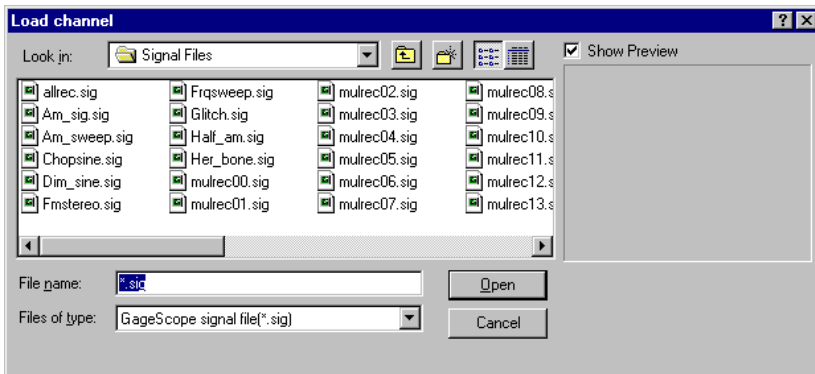
- 5 **Ch 1** should already be selected in the Channel field. If it is not, select it from the drop-down menu.



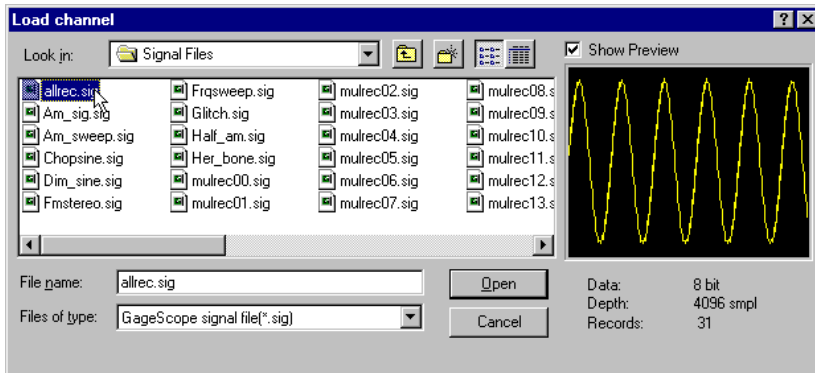
- 6 Click **Save**.

► Load the All Records File to Ch 4

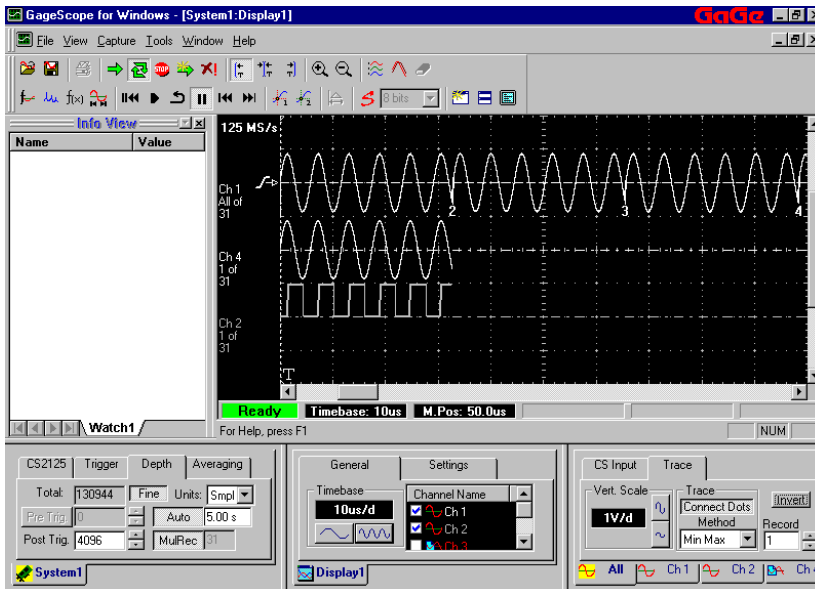
- 1 Click  on the toolbar.
- 2 The Load Channel dialog appears.



- 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).

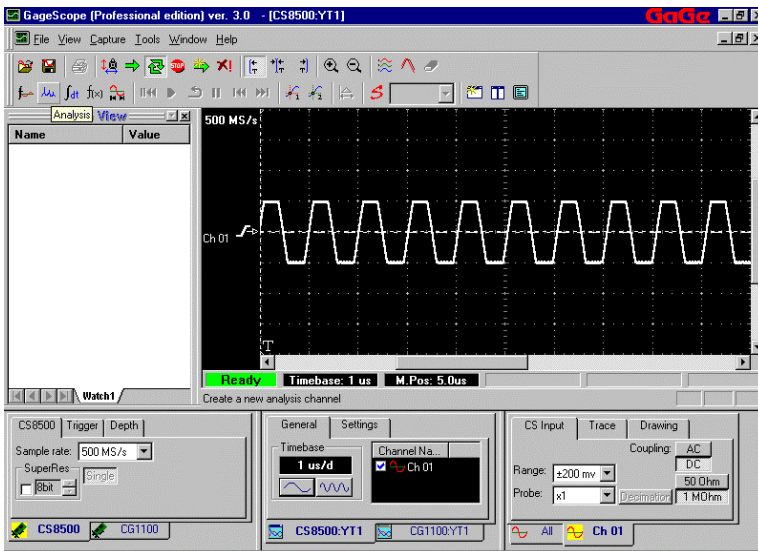
Scope Mode

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.

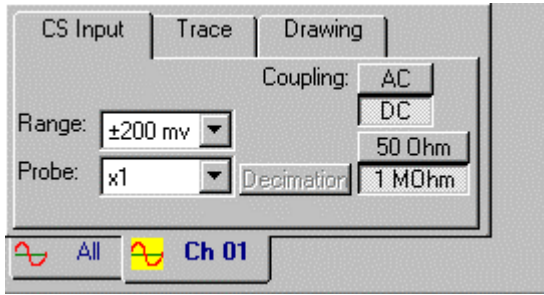
How it works

Automatic Input Range selection

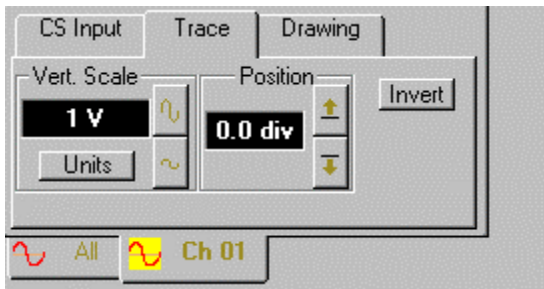
- 1 The figure below shows a 1 MHz sine wave being captured on Channel 1 of the CompuScope 8500 high speed A/D card.



- 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 $\pm 200\text{mV}$. This is shown in the Channel Control setting for Channel 1 as follows:

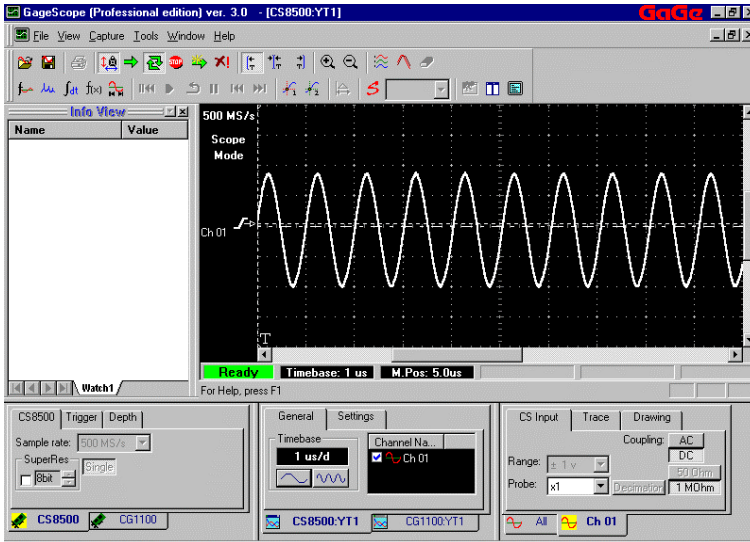


- 3 The display setting for channel 1 is set to be one volt per division. This can be viewed by clicking the **Trace** tab for channel 1 in the channel control which is located in the bottom right of the GageScope® display. The **Trace** tab property page is shown as follows:



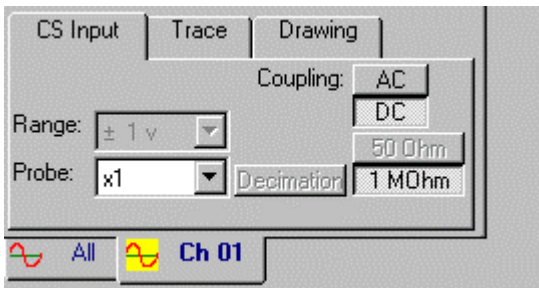
Note that GageScope® is not operating in the Scope Mode as yet.

- 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:

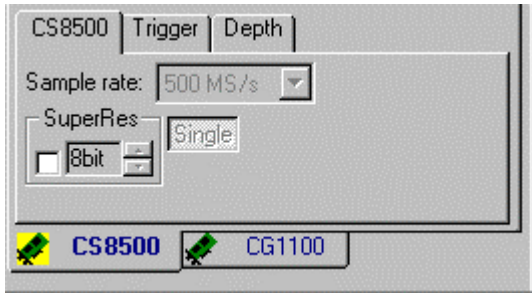


Note that the input signal now no longer appears clipped. *The input voltage range is automatically adjusted to correspond to the currently selected vertical display range (1V/div), resulting in displaying the original 1 MHz sine wave.*

- 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 $\pm 200\text{mV}$ to $\pm 1\text{V}$. 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.

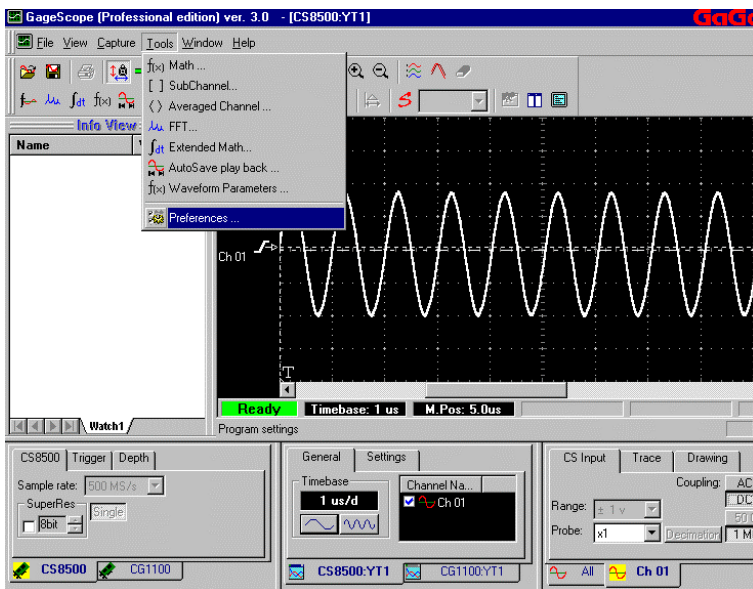


The **Sample Rate** Option under **CS8500** tab in the System Control is also disabled. The CS8500 tab is located in the System Control area in the left bottom corner of the GageScope® display.

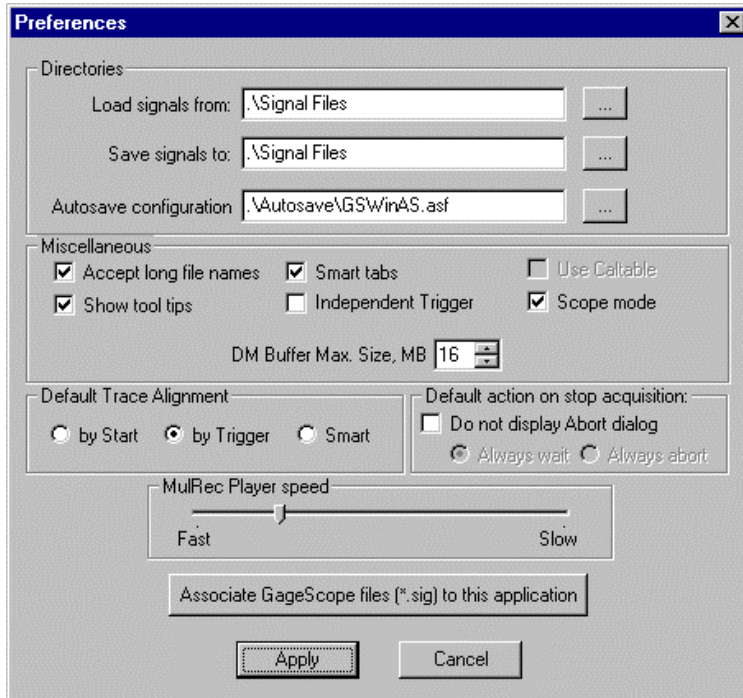


- 6 The **Scope Mode** can also be activated or deactivated by selecting Scope Mode in the **Preferences** dialog box.

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

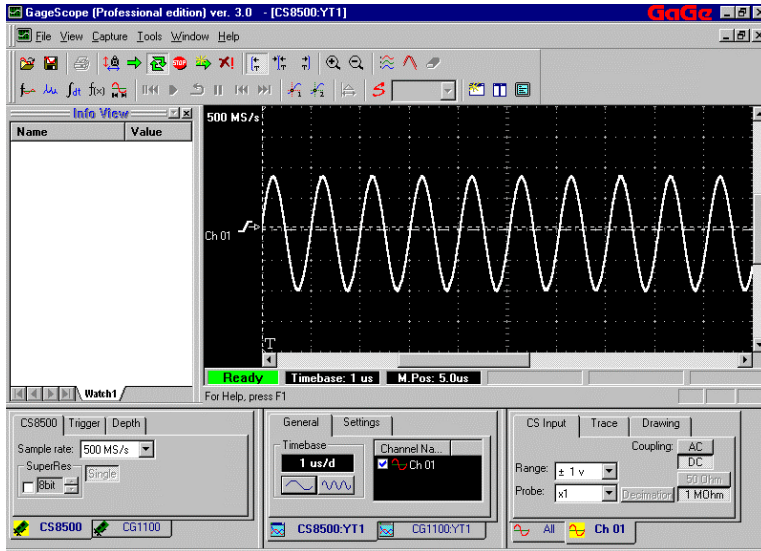


- 7 Click **Preferences** to view the Preferences dialog box:



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:

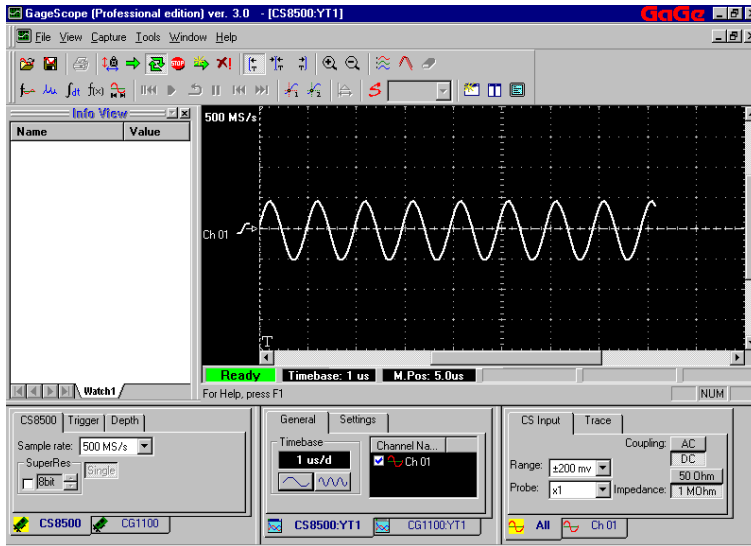


Note that Scope Mode text is now taken off the display indicating that the Scope Mode is no longer active.

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

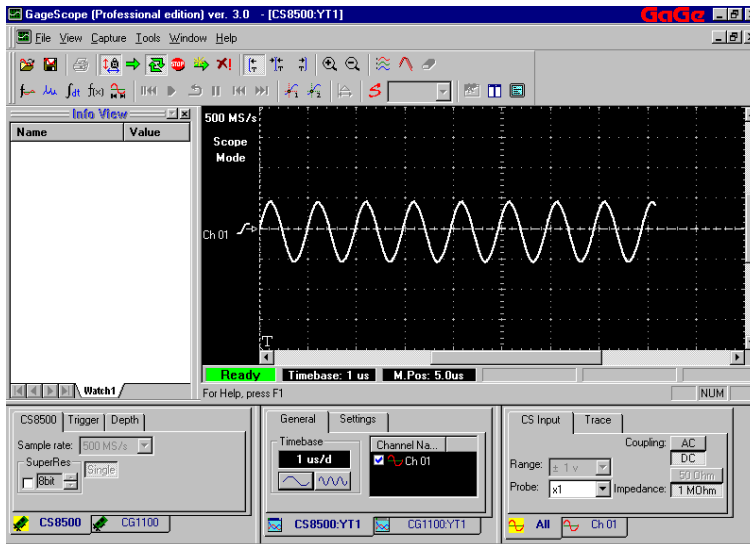
Automatic Sample Rate selection


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

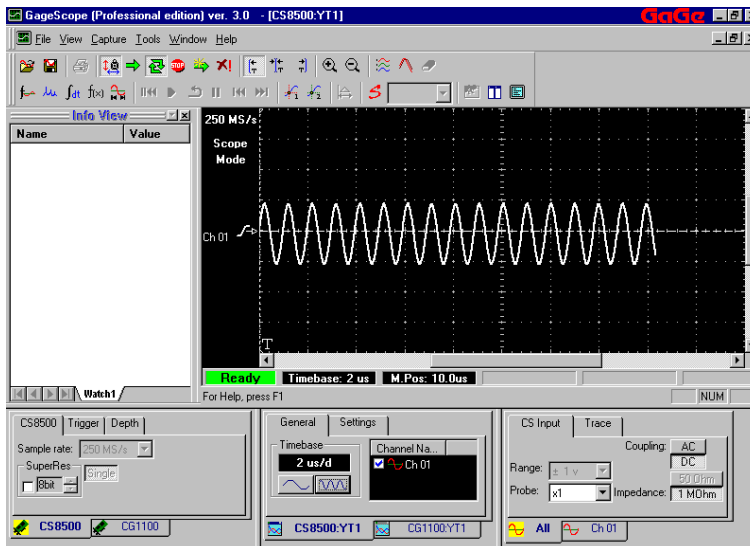


For the signal shown above, the post trigger depth is set to 4096 samples and the input range is ± 200 mV. The timebase setting is set to 1 us/d (microsecond per division).


- 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:



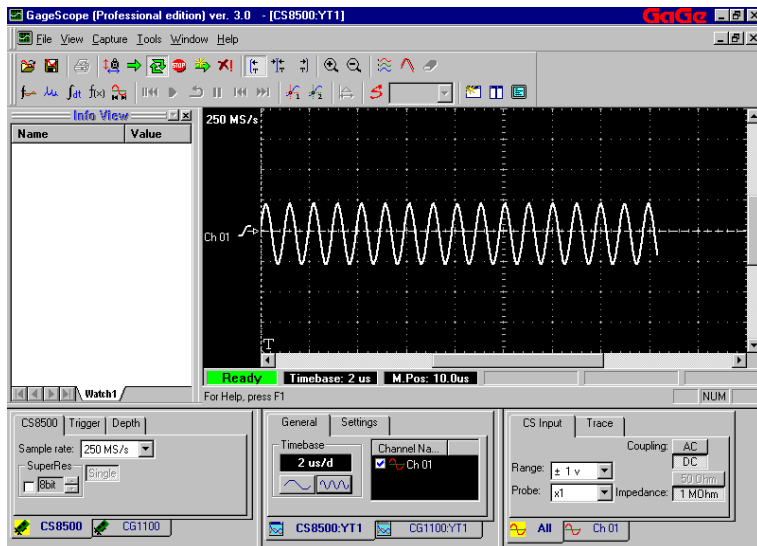
- 3 Change the timebase to 2 us/d by clicking once on the time base compression button, . The GageScope® display will appear as follows:



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 $\pm 200\text{mV}$ to $\pm 1\text{V}$.

- 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 $1/4^{\text{th}}$ 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.

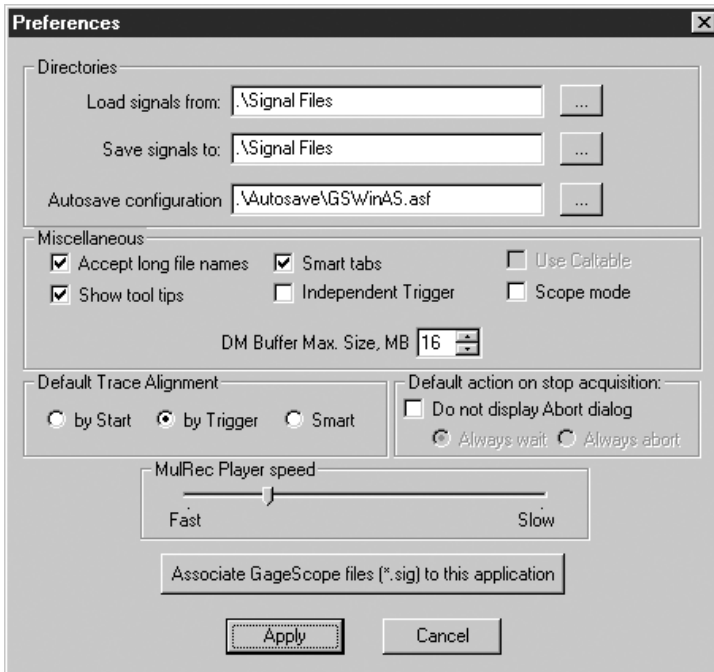
When working in Decimation mode, the user can select the starting sample and the number of samples to be downloaded from CompuScope acquisition memory. Only these data are then decimated and displayed.

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

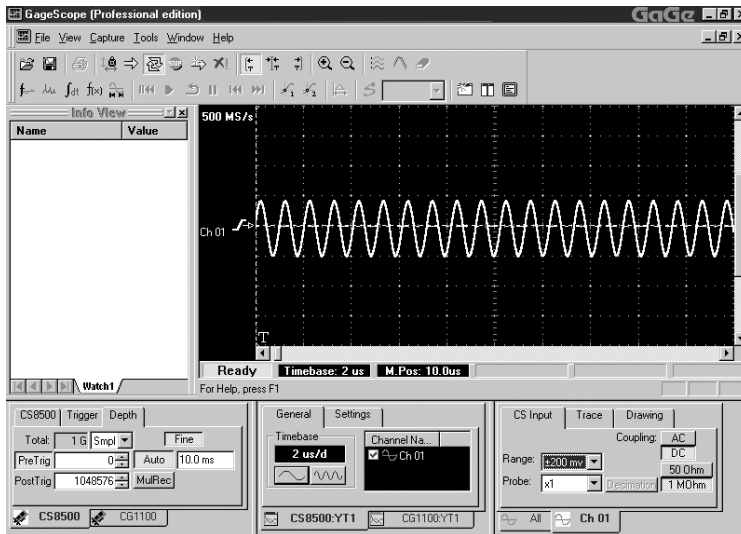
- 1 Open the **Preferences** dialog box by clicking on **Tools --> Preferences**. The Preferences dialog box will appear as follows:



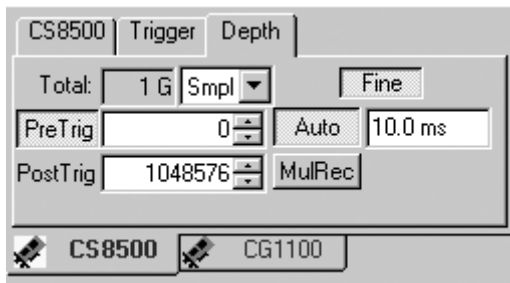
The Deep Memory Buffer size is set at 16 MB as shown in the DM text box. The maximum DM buffer size can be 64 MB.

Click **Apply**

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

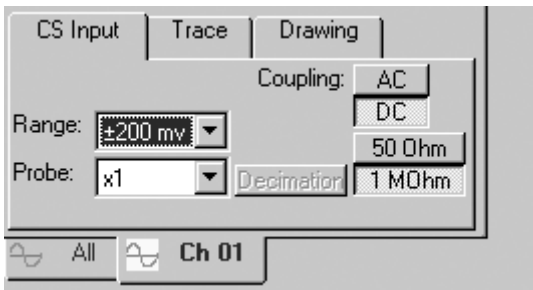


The acquisition post trigger depth is set at 1 Mega Samples (1 MS) as shown in the **Post Trig** text box in the System Control:



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:



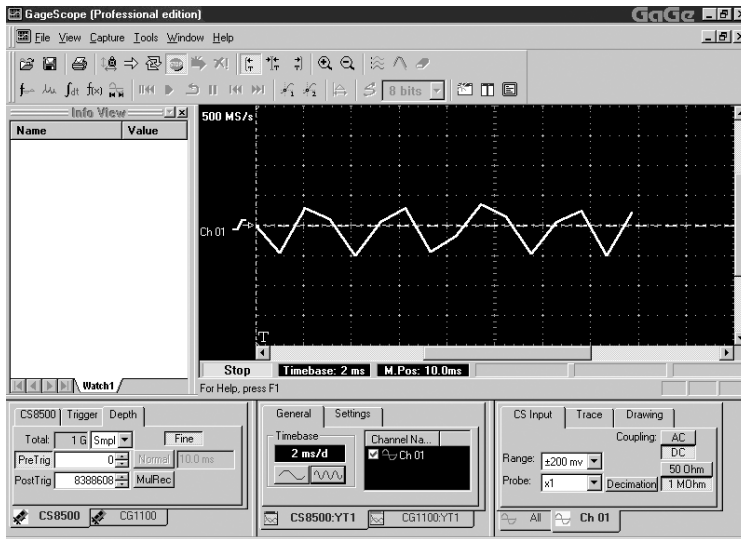
Observe that the Decimation command button is shown disabled implying that the Decimation mode is not currently invoked.

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:

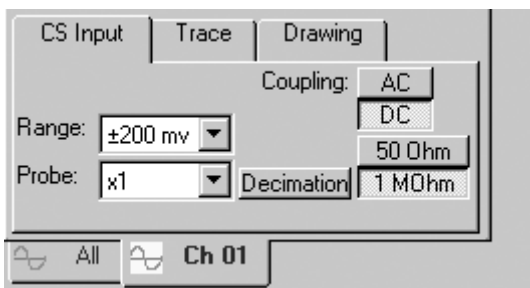


Note that the data on channel 1 is now being decimated. By default, a user gets to view the best representation of the input data.

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.

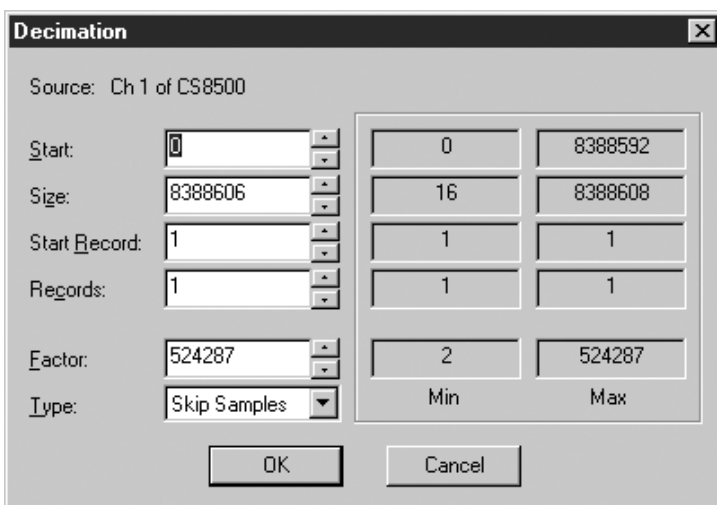
There are several other useful features available within the decimation dialog box. A detailed descriptions of these features and their use is given as follows:

- 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 is the 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	<p>The total number of records to be displayed in the Multiple Record Mode.</p> <p>The Maximum number of records equals the total CompuScope memory available for acquisition divided by the acquisition depth per channel.</p> <p>The minimum number of records to be displayed is obviously one.</p>
Factor	<p>This is the Compression Ratio. Compression ratio can be chosen from the available range of minimum and Maximum values.</p> <p>Decimation is a process of compressing the data so that it can be displayed in GageScope.</p> <p>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.</p> <p>The minimum compression ratio will display the maximum number of samples. This will give the best possible representation of the entire acquisition.</p> <p>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.</p>

Type

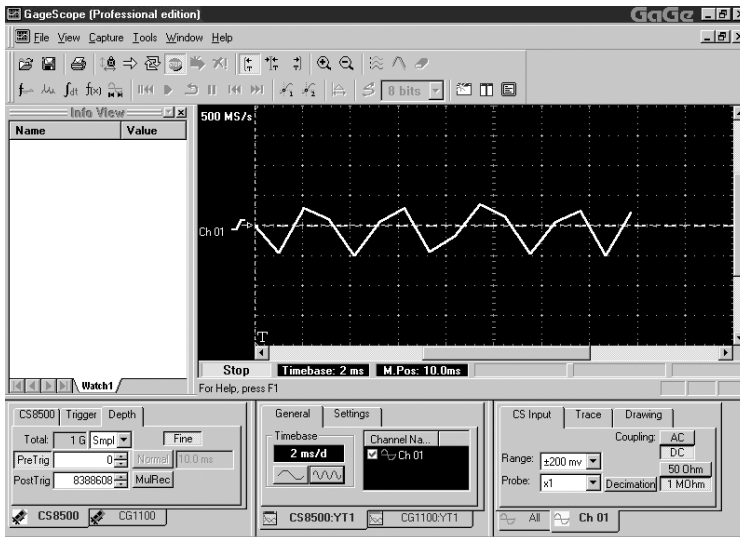
There are three types of decimation methods available:

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 vertical 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:

Decimation

Source: Ch 1 of CS8500

Start:0

Size:8388606

Start Record:1

Records:1

Factor:524287

Type:Skip Samples

0

16

1

1

2

Min

8388592

8388608

1

1

524287

Max

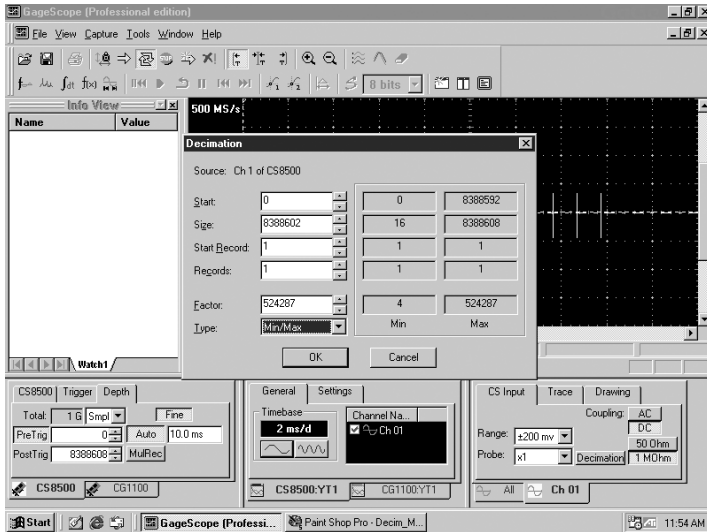
Mean

Min/Max

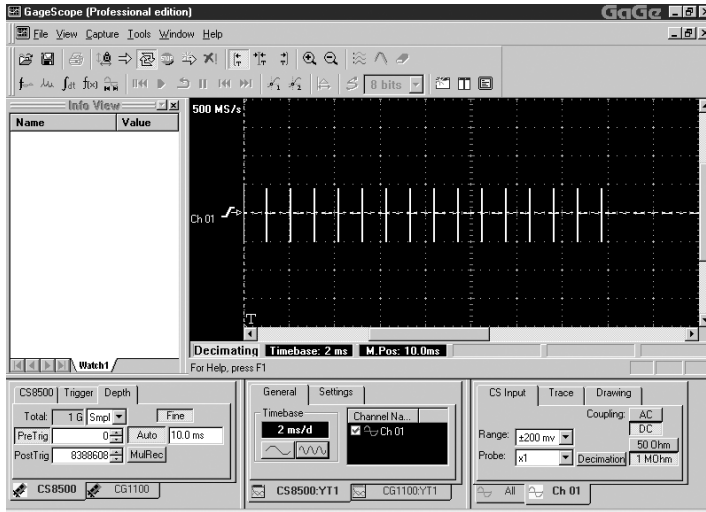
Skip Samples

Cancel

- 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

$$\begin{aligned}\text{Size (data to be displayed)} &= 8388606 \\ \text{Factor} &= 524287 \\ \text{Decimated DataSize} &= \text{Size/Factor} \\ &= 8388606/524287 \\ &= 16\end{aligned}$$

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:

The image shows a 'Decimation' dialog box with the following settings:

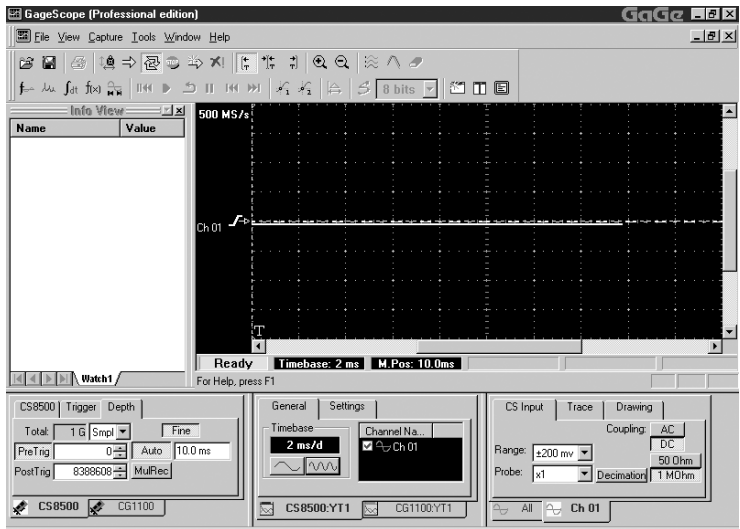
- Source: Ch 1 of CS8500
- Start: 0
- Size: 8388606
- Start Record: 1
- Records: 1
- Factor: 524287
- Type: Skip Samples (selected in the dropdown menu)

The 'Type' dropdown menu is open, showing the following options:

- Mean
- Min/Max
- Skip Samples (highlighted)

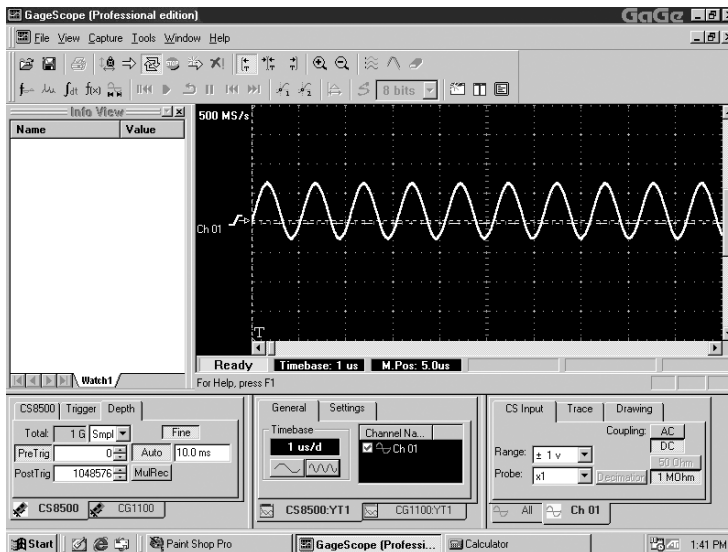
There is also a 'Cancel' button at the bottom right of the dialog box.

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

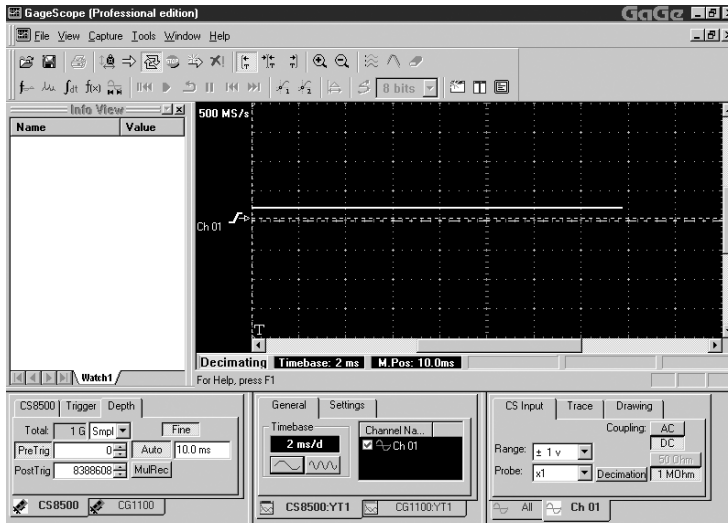


Observe that the decimated data according to the Mean method indicates that there is a small neative bias in the signal. Thus the decimated data canbe used to observe and even measure trends in the acquired signals.

- 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 employing the Mean method. The decimated data is shown as follows:



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