Data-Trax2™

Data recording and analysis software & Lab-Trax™ data acquisition systems

INSTRUCTION MANUAL

WARNING: Do not connect the Lab-Trax hardware to your computer until AFTER the software installation is complete.

050707

World Precision Instruments

Introduction

Welcome

Thank you for choosing *WPI DataTrax2* Data Recording and Analysis Software. Developed over a 5-year period, *DataTrax2* software is the result of more than 70 man years of collective experience in data recording. We are confident that this software will make your data recording and analysis easier, and welcome suggestions for improving our products. Please contact us with comments, concerns or suggestions at 800-234-1757 or via email at: support@wpiinc.com.

How to Use This User's Guide

This User's Guide assumes you know how to operate your computer and are familiar with basic Windows terminology. The **Table of Contents** lists each chapter and its contents. You can use it or the index to locate pages on a particular subject. The **Appendix** provides some useful reference materials.

We have divided the subject matter into categories and explained each in detail, but if you'd rather cut to the chase, the **Quick Start** chapter should have you recording data in less than an hour.

System Requirements

DataTrax2 recording software requires a Pentium II-level computer running Windows 98, Windows ME, Windows 2000, Windows XP, Windows Vista or a Apple computer running OSX 10.3 or later, with at least 256 megabytes of RAM and at least 50 megabytes of free space on your hard drive.

The preferred processor is a Pentium III or Pentium 4 with AGP graphics card, 512 megabytes of RAM running Windows 98/Me or Windows 2000/XP. The minimum recommended system configuration is a Pentium II, 300 megahertz computer. The faster the processor, the smoother the screen scrolling, however, all data is recorded regardless of the screen scrolling display.

Installation

From CD   *DataTrax2* software is provided on a CDROM. If you don't have a CD drive, the software installer can be downloaded from our FTP area. Point your browser to http:/www.wpiinc.com, select Support from the directory, and sign up for the *WPI* Users' Area. All installers are available at the Users' Area.

Note: Contact your IT department before installing software. You will need to have **administrative priviledges**. Do not connect your WPI hardware to the computer until **AFTER** the software installation is complete.

To install the software using the CDROM:

1 Insert the *DataTrax2* installation CD.

2 A welcome screen will appear offering to install **Software.**

3 Follow the instructions presented by the **Install Wizard**. The **Install Wizard** will tell you when the installation is complete and will restart your computer.

4 Once you have completed the installation process, remove the installation CD from the drive, connect and turn on your *WPI* Hardware.

Note: *When your hardware is connected for the first time,* <u>Windows</u> *will advise you "New Hardware found" and proceed to load the driver automatically. If for some reason* <u>Windows</u> *cannot locate the driver, point the* **Install Wizard** *to the WPI/DataTrax2/**drivers folder** in the **Program Files folder***.

From Users' Area

To install using the downloaded installer from our FTP site:

1 Go the **Users' Area** on the wpiinc.com website. Select the proper installer for the <u>Windows</u> operating system on your computer. After downloading the *DataTrax2* software installer, double click on the downloaded file. It is a self extracting archive, which will automatically launch the installer after it extracts. (Files are normally stored in the **Windows Temp Folder**). The installer will then open the **Install Wizard** and walk you through the installation.

2 Go to step three above.

Registration and Updates

Be sure to register on-line at our website at: http://www.wpiinc.com. Registering your software ensures that you are notified of updates, new releases, and sent the free upgrades you are entitled to for as long as you use the program. Software and .pdf manuals are available from the password protected area of our website. Passwords are available to registered users only by submitting an email request.

Comments and Suggestions

WPI/CB Sciences understands that user feedback is critical to the improvement of any software product. If you have ideas, suggestions or criticisms, please contact us at: support@wpiinc.com.

Technical Support

If you cannot find an answer in this User's Guide, please check the list of FAQ's (Frequently Asked Questions) on our website. If you still cannot find a solution to your problem, technical support is available to all registered users at no charge via email, phone, fax or mail. When requesting technical support, please follow the steps listed below:

• Write down your question or problem and the actions you took that created the problem.

• Be prepared to duplicate the problem.

• Note any error messages.

- Note your computer model.

- Note your operating system version.

- Note the amount of RAM (Random Access Memory) in your computer.

- Note your *DataTrax2* Software version number.

  See the sample customer support form in the appendix of this manual.

  Following these steps will enable *WPI* support staff to quickly and efficiently address your issues.

Chapter 1: Quick Start

Start here if you are already familiar with data recording concepts. This chapter contains all you need to know to begin recording data quickly and easily.

Managing Signals

The management of recorded data is divided into three steps:

1 Recording data onto the computer,

2 Navigating the recorded data to find particular areas of interest, and

3 Outputting the data either in printed or analyzed form.

Recording Data

Recording data is the most basic of the three steps. It is also the only one that is time critical. That is, you must record data when it happens, whereas navigating, analyzing or printing data can occur anytime after the data has been recorded. The following section explains the basic concepts and controls required to record data using the *DataTrax2* software.

Start Recording

The most basic control in the *DataTrax2* software is the one that starts and stops the recording. This control is found in the upper right hand corner of the **Main** window. After ensuring that the source of your signal is properly connected to your data acquisition device, click the **Record** button to begin recording. While data is being recorded the **Record** button will change to a **Stop** button. Click **Stop** at any time to end data recording.

Displaying Data

The events you want to record using the *DataTrax2* software may be very slow, like the discharge curve of a 9-Volt battery, which can take hours; or very fast, like the QRS complex in a human electrocardiogram, which may take only a fraction of a second. *DataTrax2* software allows you to record both slow and fast events, while displaying the recorded results in a format that is easily interpreted. To manage the temporal display of your data in the *DataTrax2* software, a parameter called **display time** is used.

1 **Display time** is simply the amount of time represented by one full screen of data that you see on the monitor. When the program opens, the default display time is set to 10 seconds. You can change the **display time** by using the **Half Display Time, Zoom Between Cursors** or **Double Display Time** controls in the toolbar (Figure 1-1 on page 5).

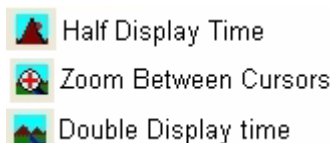*Figure 1-1: The Display Time icons on the* DataTrax2 *Main window toolbar.*

• Clicking the **Half Display Time** button (big mountain) "halves" the screen time. So, a 10-second, full-screen display becomes a five-second, full-screen display, effectively doubling the display resolution.

• Clicking the **Zoom Between Cursors** button fills the display window with the data located between the cursors.

• Clicking the **Double Display Time** button (little mountain) doubles the screen time. Therefore, a 10-second, full-screen display becomes a 20-second, full-screen display allowing twice as much data to be displayed on one screen.

2   To manage the vertical display of the data on the **y-axis** of the window, the *DataTrax2* software uses an **AutoScale** feature. The **AutoScale** function examines the recorded data in the current screen and adjusts the display so that the data is centered along the y-axis, and the amplitude is adjusted to allow the best possible view of all of the data. The **AutoScale** button is located on the channel bar of each data channel.

• Clicking the **AutoScale** button resizes the data and centers it on the current screen. The **AutoScale** button can be used as frequently as needed.

*Note: Clicking the **AutoScale** button, when large artifact spikes are present on the screen, will result in a smaller than optimal display of real data since the **AutoScale** function sets the highest and lowest data points close to the upper and lower limits of the screen.*

• Two additional controls, a **Zoom In** button and **Zoom Out** button, are positioned on either side of the **AutoScale** button. **Zoom In** expands the channel data and **Zoom Out** reduces its size.

• Further, the position of the waveform on the screen can be adjusted by clicking on it and dragging it up or down.

3   The **Value Display** box also appears on the channel bar above the channel window. The amplitude value of the data at the position of the cursor or the value of the data point that is currently being recorded is displayed in the box, in the units you specified. If you are using **2 Cursor Mode,** the difference between the amplitudes at **Cursor 1** and **Cursor 2** will be displayed.

4   Clicking and dragging on the channel bar allows movement of the channel bar up and down, Moving the channel bar changes the size of the channel window and the amount of space allocated to the adjacent channels.

Figure 1-2: The common controls of each channel in the DataTrax2 Main window.

Controls

The main controls for the *DataTrax2* program are located in the **Preferences** dialog window in the **Edit** menu. Controls are also found in the channel menu of each channel.

The Preferences Dialog Pages

From the **Preferences** dialog pages, parameters controlling the **Channels**, **Stimulator(s)**, **Views**, **Output Sequences,** and **Options and Events,** can be set. For example, the number of channels to be displayed in the **Main** window and their titles, the sampling rate, and the display width of the screen can be set from this dialog window. Many more parameters, some explained in other sections of this manual, can be set from the **Preferences** pages. These parameters can also be saved as a **settings file**, to be used again and again to perform the same experiment. The creation of groups of settings and the actual settings themselves is accomplished through the functions of the **Settings** menu.

Channel Mode   Clicking on **Channel Mode** in Raw Data displays the input option available for a given channel. On a computed channel, clicking on **Channel Mode** allows you to set up an existing function or to change and set up a new function on that channel.

The Channel Menu   From the channel menu, some of the parameters also controlled in the **Preferences** dialog pages, can be set. These include **Units**, **Scale** and **Title**.

Add Function   Raw data can be transformed by computed functions and displayed as values which provide a a clearer interpretation of the data. For example, periodic data, like that of an ECG, can be transformed by the **Rate** function to a plot of the subject's heart rate (BPM). Some of the functions available include: **Frequency, Period, Integral, Derivative, Channel Math, Spirometry Volumes,** and **EEG Power Spectrum**. Clicking on a Raw Data Channel's **Add Function** button displays a menu of available functions. Selecting one creates a new channel to display the function data.

Saving Data   Data recorded in a file can be saved, closed and opened from any computer at a later time for analysis.

To save the recorded data in a file, select **Save As** in the **File** menu. Name the file in the **file name** box. Choose a destination on the computer in which to save the file (e.g. the *WPI* or *class* folder). Click the **Save** button to save the file as an ***.iwxdata** file.

Navigating Data

Once recorded, data is "navigated" in a few ways:

Analysis Window   An area of the data can be selected for viewing and measuring in the **Analysis** window.

1   Use the screen time controls (**Half** or **Double Display Time**) to provide a better view of recorded events. Click on the **Double Display Time** button (little mountain in the toolbar, Figure 1-1 on page 5) until all of the data is visible in a single screen.

2   Once the data of interest is displayed in the **Main** Window, click the **Analysis** Window icon and the data is displayed in the **Analysis** Window.

Marks   **Marks** can also be used to navigate data. A **Mark** is a text annotation made while recording or after recording, that is locked to the data at a certain point in time.

1   To mark the record while you are recording, use the keyboard to type a message on the **Marks** line. Pressing **Return** or **Enter** on the keyboard will enter the text mark on the recording at that time.

2   When reviewing data, you can move directly to a mark from anywhere in the data using the **Marks** window. To open the **Marks** window, click on the **Marks** button in the toolbar or select **Marks** from the **Window** menu of *DataTrax2*. Select the mark of interest. Click the **GoTo** button to move directly to the area of data that contains the mark. Another way to navigate to a **Mark** is to click on the down arrow next to the **Mark** button and select the **Mark** of interest.

3   **Marks** can be entered after data recording is complete using the **Single Cursor** button (the button with

a single vertical line in Figure 1-1 on page 5).

4   The text attached to a mark can be seen at the bottom of the screen, just above the timeline. Text marks can be dragged and dropped anywhere in the record. For example, a mark may denote that a drug was administered at a certain time. Using the mouse, you can click on the text associated with a mark and drag it to the channel to which it corresponds.

Scroll Bar   Lastly, data can be navigated by using the scroll bar at the bottom of the **Main** window. Click on the right scroll arrow to move to data on the right side of the screen. Click on the left scroll arrow to move to data on the left side. Alternatively, clicking and dragging the waveform while holding the **Ctrl Key** down allows you to move the waveform to the left and right.

Getting Data Out

Printing Data   The most obvious method of exporting data is to print it. The **Print** command in the *DataTrax2* software prints the window (**Main, Analysis, ScopeView, Journal**) in the foreground.

To print the **Main** window, use the **Display Time** controls to adjust the amount of data on the screen. Pull down the **File** menu and select **Print**, which prints only data visible in the window.

Copying and Pasting Data   Data can be exported to the Journal or other word processing document using the **Cut, Copy** and **Paste** commands located in the **Edit** menu of the program.

The **Copy** command always looks at the window in the foreground. For instance, using the **Copy** command, while the **Main** window is in the foreground, copies the data in the **Main** window to the clipboard (as a picture). Adjusting the screen time affects what data will be included in the picture. Copying data is best done from the **Analysis** window; you have more control over which channels and time domain will be displayed and copied.

Exporting Data   Select areas of data in the file can be saved for export.

• Select the area of data to be saved and exported using the **2 Cursor** mode in the **Main** window.

• Click the **Analysis** button in the toolbar to view the selected data in the **Analysis** window.

• Open the **File** menu and select **Export**. A dialog box appears and provides for the naming and formatting of the file to be exported from the **Analysis** window. Files to be exported can be saved as pictures (**.bmp - bitmap** and **.png - portable network graphics** formats are currently supported), a text file (**.txt - ASCII** or **.atf - Axon Text**), as a **MatLab** file (**.mat**), or a **DADiSP** file (**.dat**).

• Complete data files can be saved, from the **Main** window, as a **MatLab** file or **DADiSP** file.

Analysis   Apart from the cursor measurements available in the *DataTrax2* **Main** window, analysis occurs in two places. Data manipulation, close examination, and a variety of calculations are handled in the **Analysis** window, and wave to wave transforms are available in the **Main** window. For specific information on the analyses that each of these categories can perform, refer to the **Analysis** chapter in this manual. The basics are summarized in this chapter.

On a given selection of data displayed in the **Analysis** window, the program can calculate and report the selected parameters. These are displayed across the top of each channel in the channel bar. By selecting the **Send To Journal** items in the **Tools** menu (or **right-click** menu), these values for the selected parameters can be sent to a part of the *DataTrax2* program called the **Journal**. From any of the **Channel** menus (or **right-click**) you can add data and titles to the **Journal**. Copying of the channel display is accomplished using the **Copy Graph** option.

Journal

The **Journal** is an internal word processing application capable of accepting calculated numbers from the **Analysis**, **FFT**, or **XY** window, as well as copied graphs from any window. In addition to values and pictures, text can be put into the **Journal** using the keyboard. This allows lab reports or explanatory documents to be composed without having to leave the *DataTrax2* program.

Calculations

In the **Analysis** window, data can be reduced using mathematical operators like **mean** or **maximum**. These operators generally take a selection of data, operate on it and produce a single number as a result.

A second and quite different type of analysis requires that an entire set of data be transformed, point for point, into a completely different, but related set of data. Transforming data into its first derivative is a good example of this type of analysis. *DataTrax2* software includes several of these **Wave-to-Wave** transforms in the **Add Function** menu of each channel.

The list of available functions is always expanding. *DataTrax2* V2.0 includes: **Periodic Functions**, **Integrals**, **Derivatives**, **Channel Math**, **Filter**, **Smoothing**, **Power**, **Spirometry**, **Cardiac**, **EEG**, **Multipoint Calibration** and **Gain Telegraph.**

These calculations are also available in the **Function** list on the **Channel Page** of the **Preferences** dialog window. Functions typically used on signals recorded in a biology lab are listed. Functions can be changed and these modifications can be saved in new **settings files** stored in the **Settings Groups**.

Chapter 2: Acquisition

The Basics

Time, Temporal Resolution, Sample Speed, and Aliasing

Temporal resolution in digitally recorded data is determined by the sampling rate, also known as sampling speed. To select a suitable sampling rate, you will need some background on the theory and use of analog-to-digital converters (ADC).

The ADC in the *WPI* computer interface takes simple voltage measurements at regular intervals. The voltage measurements (Y-axis values) and the times at which the voltages were recorded (X-axis values) are sent by the *WPI* interface to the *DataTrax2* software as a pair of (X,Y) coordinates. The *DataTrax2* software displays these coordinates as data points by plotting them. The software then connects the data points with a line to create the smooth, graphical appearance of an analog chart recording; but, only the data points themselves are hard measurements. The lines that are drawn to connect the data points are "educated guesses" made by the software and may be misleading. For example, if the sampling speed being used is not fast enough for the data being recorded, the display will not accurately represent the real data. This misleading behavior of data acquisition software, can be avoided by using a sampling speed that is fast enough for the data being recorded.

When the sampling speed is set in *DataTrax2* software, the hardware is being told what the time interval between samples will be. For example, 100 samples per second equals one sample every 10 milliseconds. To avoid creating displays that may be misleading, a simple question about the sampling interval needs to be asked: Is there a chance that something important has occurred between the sample points?

If the answer is "yes," then the sampling speed must be increased in order to see and record that event. In the example presented in Figure 2-1 on page 10, a sine wave with a frequency of 100Hz is sent to the ADC. In the screen on the left, the data points, recorded by an ADC set to the same sampling speed as the frequency of the sine wave, are marked.
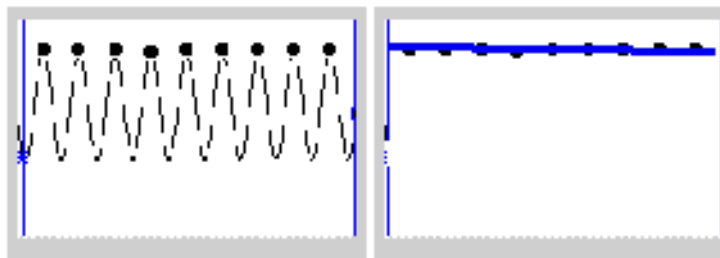

Figure 2-1: A 100Hz sine wave recorded at 100Hz. The samples coincide with the peaks of waves; so, the data points can be connected by a straight line.

Because the sine wave has the same frequency as the sampling rate, each recorded data point occurs at the same place in each cycle of the sine wave. In the screen on the right, a straight line is the result when the data points are connected. The line is a false representation of the real data, which is a sine wave. To fix the problem and render a better picture of the data, a faster sampling rate is required. If the sampling rate is increased to 110 samples a second, the data points would be a millisecond closer to each other than at 100 samples a second, but a 100Hz sine wave would still have only one data point for each cycle. When the data points are

connected by a line, a sine wave with a frequency of 0.1Hz is displayed on the screen. Again, the wave displayed on the screen is not a true representation of the real wave because the sampling speed is too slow.

So, how fast does the sampling speed need to be to record a reasonable representation of a real waveform? A general rule of thumb is to sample at a rate that is a minimum of five times faster than the fastest frequency of interest in the waveform.

To determine the optimal sampling frequency, find the shortest event in a sample recording. For example, the **R** wave has the shortest duration of any event in the ECG and should be used to determine the sampling speed needed to record a reasonable representation of the ECG. Next, find the rise time (in seconds) of the **R** wave. The rise time is the time it takes the wave of interest to go from its start to about 2/3 of full value. In the case of the **R** wave in an ECG, this value is about 20 milliseconds (0.020 seconds).

Substitute the value 0.02 for the rise time in the following equation to determine the bandwidth of the event (R wave):

$$0.159/\text{Rise Time} = \text{Bandwidth}$$

$$0.159/0.020 = 8$$

Finally, multiply the bandwidth by five to determine the minimum sampling frequency needed, which is 40Hz in this example. Higher sampling rates give better pictures, at the expense of making data files larger. Therefore, choosing a proper sampling speed must be a balance between the accurate portrayal of the data and the unwieldy size of the data file.

The default sampling speed set in the *DataTrax2* software is 200 samples per second. This is adequate to execute most of the laboratory experiments presented in the laboratory manuals furnished with the *WPI* teaching kits.
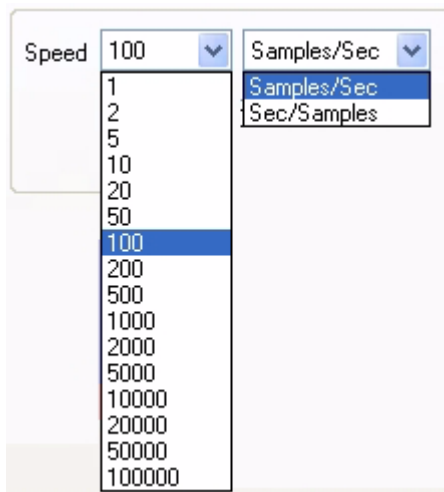
If sampling speeds are set manually, it is always best to start sampling the data at the fastest speed possible. Then, slowly reduce the sampling rate to a speed where data initially begins to degrade. Finally, set the sampling speed just above the rate that initially cause degradation. See the examples of pulse waves sampled at different frequencies in the appendix.

Sampling rates are set from the **Speed** menu on the **Channels** page of the **Preferences** dialog window. The **Preferences** dialog window is called by selecting **Preferences** from the **Edit** menu.

The speeds displayed in the **Speed** menu are rates per second per channel. This menu is dynamic, the maximum available sampling speed will adjust depending on how many channels are open.

Make sure the **Display Time**, as programmed on the **Channels** page of the **Preferences** dialog window, is set to 10 seconds before doing the following exercise.

1 Plug the DIN connector on the cable of the pulse transducer (PT-104) furnished with your kit into Channel 3 on the *WPI/214* interface. Wrap the plethysmograph (pulse transducer) firmly around the end of the subject's middle finger.

2 Open the **Channels** page in the **Preferences** dialog window and set the sampling speed to 1,000 samples per second. Record 10 seconds of data.

3 Next, set the sampling speed to 500 samples/second and record an additional 10 seconds of data.

4 Repeat this procedure for sampling speeds of 200, 100, 50, 20 and 10 samples/second.

Display a section of each recording block in the **Main** window, then switch to the **Analysis** window. Closely examine the recorded data as displayed in the **Analysis** window. Notice that the signal becomes progressively more coarse as the sampling rates go down, until eventually the signal is unrecognizable.

Vertical Resolution

As stated earlier in this chapter, the ADC takes measurements at regular time intervals to produce data points with X,Y coordinates and the *DataTrax2* software later plots and connects the data points with lines.

The temporal resolution of the incoming signal was the topic of the previous section of this chapter. In this section, the vertical (voltage) or amplitude resolution of the signal will be discussed.

When making a measurement of length with a ruler, the accuracy of the measurement is determined by how many gradations are printed on the ruler. Clearly, a ruler with gradations every eighth of an inch is more precise than a ruler with a gradation every inch. The more lines or gradations there are per unit of measure, the more accurate the ruler.

If the ADC is considered to be a ruler for voltage, then its resolution (its number of gradations) is determined by a parameter called "bit depth."

In an 8-bit word, or byte, there are 256 (2^8) different possibilities for the value of the byte. A 9-bit word has twice that many possibilities (512), a 10-bit word has four times as many (1024) and so on. LabTrax 4/16 and the WPI 118 use a 16-bit ADC providing 65,536 different possibilities, while the LabTrax 24 series uses a 24-bit ADC providing for 16,777,216 possibilities. The Digital Resolution of the ADC is the range divided by the number of possible ADC steps. This is the minumum measurement possible with the ADC.

It is important to realize that it is not possible to make a measurement with more precision than the Digital Resolution unless the bit depth is increased or the input range is narrowed. Changing the bit depth requires a different ADC to be placed in the unit. However, the input range can be easily narrowed by applying amplification (gain) to the signal before the signal is presented to the ADC. If a 1000X gain is applied to the incoming signal by an amplifier, the minimum resolution of the ADC improves

Note: The actual resolution of the system depends on both the analog and the digital resolution. It is important to always consider the resolution of the sytem as a whole.

Signal Conditioning

Gain

As pointed out in the section on vertical resolution, there is a minimum voltage beneath which the ADC cannot read. This voltage ±304μV for the WPI 118, LabTrax 4/16 and ±1.2nV for the Labtrax 24 series. If the signal being measured is very small, additional gain can be applied to the signal before it is presented to the ADC by using a bioamplifier.

For example, consider an ECG, whose total peak to peak amplitude is only 2 mV. The number of steps that can be resolved with an 16 bit ADC is 2mV/0.600uV or a little more than three steps, which is not sensitive enough to see any meaningful changes. If an amplifier is placed between the signal and the analog to digital converter (ADC), the "raw" signal can be amplified by a selected **gain**. For example, if the gain is 100 times (X100), the 2mV ECG signal becomes a 200mV signal. Now, when the amplified signal is presented to the ADC, it is 100 times bigger. The minimum signal now would represent only 1/658th of the total signal (304µV/200mV). It can be seen that a 16-bit ADC is much more tolerant of small signals, making the gain required to view a small signal with a 16-bit unit less critical.

Adding gain to the recording system improves the signal to noise ratio of the measuring system, but the cost is that the ADC cannot see any signal above the +10V or below the -10V input limits. In the case where X100 gain is applied to a 0.2V signal, the amplified signal becomes 20V. This is above the +10V input limit of the ADC, so the signal is **out of range**. In fact, the **effective** input range of *WPI* ADC units is equal to ±10V/Gain. So, if a gain of X10 is used on an amplifier, the effective input range of the ADC drops to ±1V (±10V range/X10 gain). Any input signal larger than 1 volts will be out of range. If a gain of X100 is applied to the input signal, the effective input range of the ADC is restricted to ±0.1V, and so on.

DIN 8 Inputs   The DIN 8 inputs on *WPI* analog to digital converters can apply up to X1000 gain. This is accomplished through the placement of a gain programming resistor within the DIN 8 connector of the transducer or cable that can be plugged into the DIN 8 inputs of the *WPI* units. Gain programming resistors are ALREADY PRESENT IN ALL *WPI* TRANSDUCERS. Gain programming resistors can be installed on NON-*WPI* transducers by rewiring the connector. Consult the hardware section in the appendix of this manual for the pin configurations and diagrams of the DIN 8 connectors used with *WPI* ADCs.

Offset

Offset is sometimes referred to as "positioning." Some recorders, amplifiers, and transducers have a knob that positions the baseline of the recording on the screen. Positioning control permits the centering of the signal which makes measurements more convenient, or the lowering of the baseline to accommodate the display of a signal that has more gain applied to it. The availability of the **AutoScale** feature in the *DataTrax2* software reduces the need for a positioning knob. In fact, the very low noise of the *WPI* Recorders makes positioning controls unnecessary as signals are automatically centered and expanded to fill the recording screen when the **AutoScale** button is clicked. Positioning of the waveform may be accomplished by clicking on the waveform and dragging it to the desired position within the channel.

Filters

Filters can be set to remove certain frequencies from signals. Hopefully, the frequency information removed when the filters are applied is noise and not part of the signal of interest. There are two basic types of electronic filters: **Low Pass** and **High Pass**. When used in combination with each other, these filters can create either **Band Pass** or **Band Reject** (**Notch**) filters.

By definition, **Low Pass** filters PASS only those frequencies beLOW the set frequency. For example: in an electrocardiogram, a large percentage of the ECG signal information is contained in frequencies below 40Hz. A significant noise source in such recordings is the 60Hz line voltage (110V AC power, or mains) used to power equipment and lights. A 50Hz **Low Pass** filter would allow all frequencies below 50Hz (including most ECG information) to pass to the recorder, but would exclude all frequencies above 50Hz, including the 60Hz noise from the

mains. The end result of using the 50Hz Low Pass filters is the creation of a quieter, more readable ECG. In general, the application of a low pass filter "quiets" the baseline and improves the signal to noise ratio of the recording.

On the other hand, **High Pass** filters PASS frequencies HIGHer than the set frequency. These filters can remove low frequency interference, such as baseline drift or a standing offset voltage, so that the user sees a more stable baseline.

High Pass, **Low Pass** and **Band Pass** filters can also be applied to each channel from the **Filter** function available from the **Add Function** button on the **Channel Bar** of each channel. These filters are not "Hardware" filters; they are executed in software and work on all ADC units running *DataTrax2*. Unlike hardware filters, software filters can be applied AFTER the data is recorded. These filters also work in real time and can be applied to data as it is being recorded.

Averaging

In addition to filtering, the *DataTrax2* software can apply another quieting technique to data recorded at speeds less than 200 samples per second. In the section on temporal resolution, it was demonstrated that fast events require fast sampling speeds. What happens if slow events are recorded at fast sampling speeds? More information is collected than is needed to accurately display the signal. What should be done with the extra data? At sampling speeds of 100 samples per second or less, the *DataTrax2* program operates on the "extra data" using a technique similar to the oversampling employed by CD players to reduce noise. The amount of oversampling that occurs is determined by the sample speed that you set. The end result is that small signals recorded at slow speed will appear less noisy than small signals recorded at high speed.

Outboard Conditioning

WPI Data Acquisition Systems with built-in amplifiers and filters, can be controlled by the *DataTrax2* software.These amplifiers and filters should be adequate for most applications. In cases where additional or custom signal conditioning is required, outboard devices can be used to condition the signal before it is presented to the *WPI* Data Acquisition System. *WPI* makes a full range of amplifiers for this purpose, but any amplifier with an analog output can be used. This allows other amplifiers, like those from Thornton or Grass, or special purpose devices, like those made by Warner Instruments, Axon instruments, or World Precision Instruments, to be used with *WPI* Data Acquisition Systems. Outputs from these and other devices can be connected to the available BNC inputs on the front panel of *WPI* Data Acquisition Systems. Outputs of external devices can also be connected to the DIN 8 inputs with an adapter cable available from *WPI*.

Chart Mode

Once the signal source is connected to the *WPI* unit and the signal conditioners are configured, data recording can begin. The most basic controls required are the ones that turn the recording on and off. *DataTrax2* has three ways to **Start** a recording and two ways to **Stop** it.
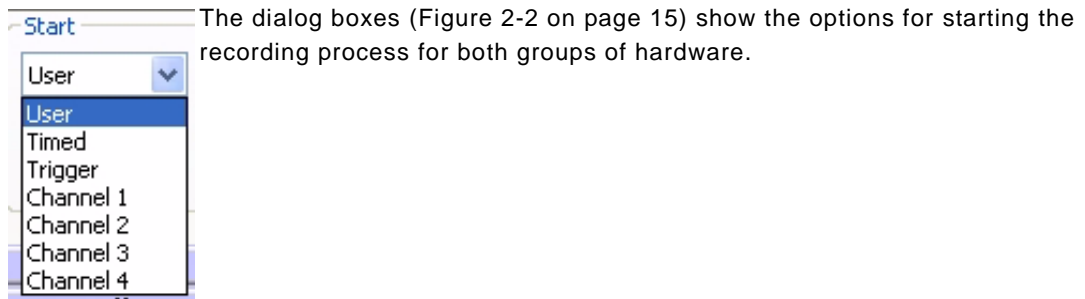
Starting

The dialog boxes (Figure 2-2 on page 15) show the options for starting the recording process for both groups of hardware.

Figure 2-2: The DataTrax2 *Start dialog boxes*

User

The easiest away to start the recording is for the **User** to manually press the **Record** button in the upper right hand corner of the *DataTrax2* **Main** window. **User** is the default setting in *DataTrax2*. Recording begins when the **User** presses the **Record** button, and will continue until one of the **Stop** conditions is met. **Pretriggering** is not possible in the **User** (manual) mode.

Timed

If desired, the recording of data can be delayed for a user specified fixed duration.

External Trigger

Sometimes, it is necessary to synchronize the beginning of the recording with the beginning of an external event. The *DataTrax2* software can be configured to start recording when the *WPI* hardware detects a voltage pulse (+3 to +5V amplitude) through the BNC connector of the **Trigger** input. Many devices have **Trigger** or **TTL** outputs that are capable of starting the *DataTrax2* recording software; these devices include: stimulators, relays, pumps, valves and cameras.

Triggered from Channel

For various reasons, data recording may not need to begin until the data itself reaches certain criteria. For example, it may be necessary to record an animal's body temperature only if the base temperature exceeds 100°F. The *DataTrax2* software could be programmed to begin recording when the data on the temperature channel exceeded 100°F. Thresholds above and below a certain data value can be set to start the recording.
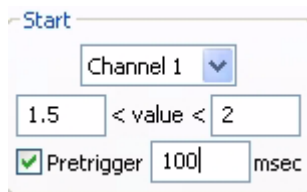
Open the **Preferences** dialog window from the **Edit** menu. Click on the tab for the **Channel** page. Pull down the **Start Mode** menu and select the **channel (Ch)** that contains the data to be used as a trigger. Set the values that the data must meet before recording is triggered. The diagram demonstrates that *DataTrax2* is programmed to look for a trigger from the data on Channel 1 when the amplitude of the data is above 1.5V and below 2 V.

Pretriggering
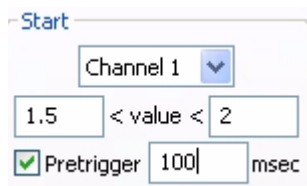
When an **External Trigger** or a trigger from a data channel (**Ch**)is used, the data just before the trigger occurs may be important to record as well. For example, if the **R** wave is used to trigger the recording of an ECG, the **P** wave and other parts of the ECG that occur before the **R** wave can be recorded. The **Pretrigger** feature of the *DataTrax2* software can be used to look back in time and display

a small piece of data prior to the trigger. To enable pretriggering, check the box next to the label **Pretrigger** (in the **Start Mode** box on the **Channel** page of the **Preferences** dialog window). Enter the amount of pretrigger time needed in the edit box to the right of the **Pretrigger** label.

Stopping

Once recording begins, the *DataTrax2* programs offers two different ways to halt the recording: **User** or **Timed**

.

Figure 2-3: The Stop dialog boxes.

User

User is the default **Stop** value for *DataTrax2* and can be reset from the **Stop Mode** box on the **Channel** page in the **Preferences** dialog window. In **User** mode, the **Stop** button in the upper right hand corner of the *DataTrax2* **Main** window will stop the recording when clicked. The **Record** button changes to the **Stop** button after the **Record** button is clicked to begin the recording. The **Stop** button remains visible until it is clicked. Clicking the **Stop** button will change it back to **Record**.Timed

When **Timed** is selected as the stop mode, the *DataTrax2* software will stop recording automatically after a predetermined time (in seconds), which is entered into the edit box.

Pausing Display

It is sometimes desirable to pause the scrolling display of data while at the same time continuing to record data. In *DataTrax2*, this is accomplished by clicking the **Pause Display** button in the lower left hand corner of the **Main** window.

Scope Mode

When to Use the Scope Mode

The first devices used to record data were electromechanical. These devices used a stylus or very fine pen that was moved by a sensitive motor. The frequency response of these machines was very low; the fastest events that they could record were on the order of tenths of a second. As science progressed, it became obvious that there were faster things in nature than pen recorders could visualize. The oscilloscope or "scope" is able to take a fast, brief snapshot of an event, but the continuous chart-like recording is sacrificed. An oscilloscope capturing a fast moving electrical signal, like an action potential, is analogous to high speed, stop motion photography capturing the movement of a hummingbird's wing.

To effectively use the **Scope** mode in the *DataTrax2* program, the length of the snapshot (display time) and the sampling rate need to be set to accurately visualize the event. Events that are best captured using the **Scope** mode are brief and repetitive. Action potentials and most neurophysiological signals are good examples of signals that are best recorded in **Scope** mode.

In addition to being able to capture a very brief event in time, a proper trigger is needed to begin the recording of the event at the moment it takes place. Using the example of stop motion photography, if the high speed camera snapped the picture too soon or too late, the picture may

not show the event that was trying to be captured. So the acquisition of data at the right moment requires a proper trigger. In the **Scope** mode either of the previously discussed "non-manual" modes, **External Trigger** or **Trigger from Channel**, can be used to start the sweep.

Acquiring Data in the Scope Mode

Scope mode can be enabled from the **Channel** page in the **Preferences** dialog window, which is accessible under the **Edit** menu. Data can be acquired on as many channels as are available. Data can be viewed in either **Chart** or **Scope** format with some limitations. Switching between formats is done by changing the acquisition mode in the **Channel Page** of the **Preferences** window. The limitations to viewing data in the **Scope** or **Chart** format are the following:

• If data is recorded in **Scope** mode, each sweep or record segment is treated as a block of data. When viewed in **Chart** mode, sweeps (or blocks) are laid end to end, in the order that they were acquired. Data can be scrolled.

• If data is recorded in **Chart** mode, each block or record segment is treated as a sweep.

Set Up the Software

To program *DataTrax2* to record in **Scope** mode, pull down the **Edit** menu (**Application** menu on Mac) and select **Preferences**. Open the **Channel** page of the **Preferences** dialog window. Select the appropriate **Scope** acquisition mode. The acquisition mode can be set to either **Repetitive**, **Multiple** or **Averaged**. Next, set the **Start** mode that will begin the recording of the sweep. The triggers that are available are: **User**, for manual starting of the sweep that can be linked to the output of the ADC; **External Trigger**, for triggering by an external source; **Trigger by Channel**, for triggering by an input signal, **Timed Trigger** for delayed start.
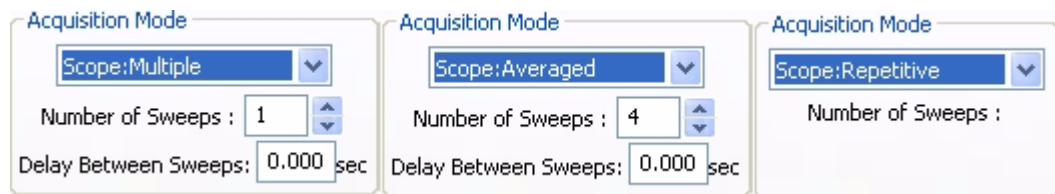

Figure 2-4: The Scope application setup controls.

Repetitive Mode

When the **Record** button is clicked in **Repetitive** mode, each new sweep overwrites the previous sweep. Real oscilloscopes work in this way. When the **Stop** button is clicked, only the last trace is saved.

Multiple Sweep Mode

When the **Record** button is clicked in **Multiple Sweep** mode, *DataTrax2* will acquire and save a predetermined number of sweeps. The number of sweeps in a series can be set in the number of **Sweeps** box on the **Channel** page. The beginning of each recorded sweep is determined by the **Start mode** conditions. A **Delay Between Sweeps** can also be programmed in the **Multiple Sweep** mode. If zero is selected as the **Delay Between Sweeps**, the next sweep in the series is taken as soon as the software is ready.

Averaged Mode

When the **Record** button is clicked, *DataTrax2* will record the user specified number of Sweeps, but will only save and display the averaged sweep.

Sweep Length　　　When operating in **Scope** mode, *DataTrax2* takes "snapshots" of data. Each "snapshot" is called a sweep and has a pre-determined length. The sweep length (or **Stop Time**) is set in the **Stop Mode** section of the **Channel** page from the **Preferences** dialog window. By choosing **Timed** as the **Stop Mode**, the sweep length (in seconds) can be entered into the edit box in that section of the page.

Sampling Rate　　　Determining the optimal sampling frequency in **Scope** mode is the same procedure as used for **Chart** mode. First, find the shortest event in the record. If the compound action potential is used as an example, the spike goes from baseline to about 80% of its full amplitude in about 0.35 ms. If you substitute.00035s for the rise time in the following equation:

$$0.159/\text{Rise Time} = \text{Bandwidth}$$

$$0.159/0.00035 = 454$$

Multiply the bandwidth by five and the minimum sampling frequency is 2270Hz. Doubling the sampling rate to 5kHz ensures that the trace recorded is representative of the signal.

If the entire event of interest is only 0.1 seconds long, a sweep at 5,000 samples per second occupies only 500 data points. Since the *DataTrax2* program can accommodate up to 1,000,000 points per screen while recording, there is substantial room available for a wider display or a greater sampling rate.

Saving Your Data

Every software manual has a section on the importance of backup and saving. *DataTrax2* is no different. **SAVE YOUR DATA!**

Consider that the data acquisition process imposes new and different constraints on the task of saving data. A word processor or spreadsheet document can be saved anytime that the user thinks about it. The user simply stops writing and saves the document.

In a data recording application, data is constantly being added, sometimes at fantastic rates (200,000 bytes/second at top speed). To save data, the recording would have to be stopped. In most recording applications, this is impractical.

When the recording is stopped *DataTrax2* saves the current data to disk. As the *DataTrax2* program acquires data, it buffers the unsaved raw data to a file on the hard disk. In the event of an unexpected loss of power or computer crash, the data is preserved. When the *DataTrax2* software is reopened after such an event, *DataTrax2* will ask the user if they want to recover the data. If the answer is yes, the backup file is recovered and data is preserved. To permanently save the data, use the **Save As** function BEFORE RECORDING ANY NEW DATA.

Chapter 3: Display

Introduction

From a functional standpoint, the *DataTrax2* program is really two programs. It is primarily a **data acquisition application**, whose main task is to record analog experimental data as data points (with X and Y values) and store these data points on the computer. The second task of the *DataTrax2* program is to **graphically display** the collected data in a way that is understandable to the user. Chapter 2 explains the various controls and options relative to the acquisition of data. This chapter will discuss considerations regarding the display of collected data on the various windows in the *DataTrax2* user interface.

User Interface

The *DataTrax2* user interface contains six primary windows; **Main**, **Analysis**, **XY**, **FFT**, **Marks**, and **Journal**. There are also dialog windows, accessible through the **Preferences** selection in the **Edit** menu, which provide controls of the **Channel**, **Stimulator**, **Views**, **Sequences** and **Options**. Most of the interface related features in *DataTrax2* can be found in the **Main** window (Figure 3-1 on page 20). *DataTrax2* can display up to 64 channels in the **Main** window.
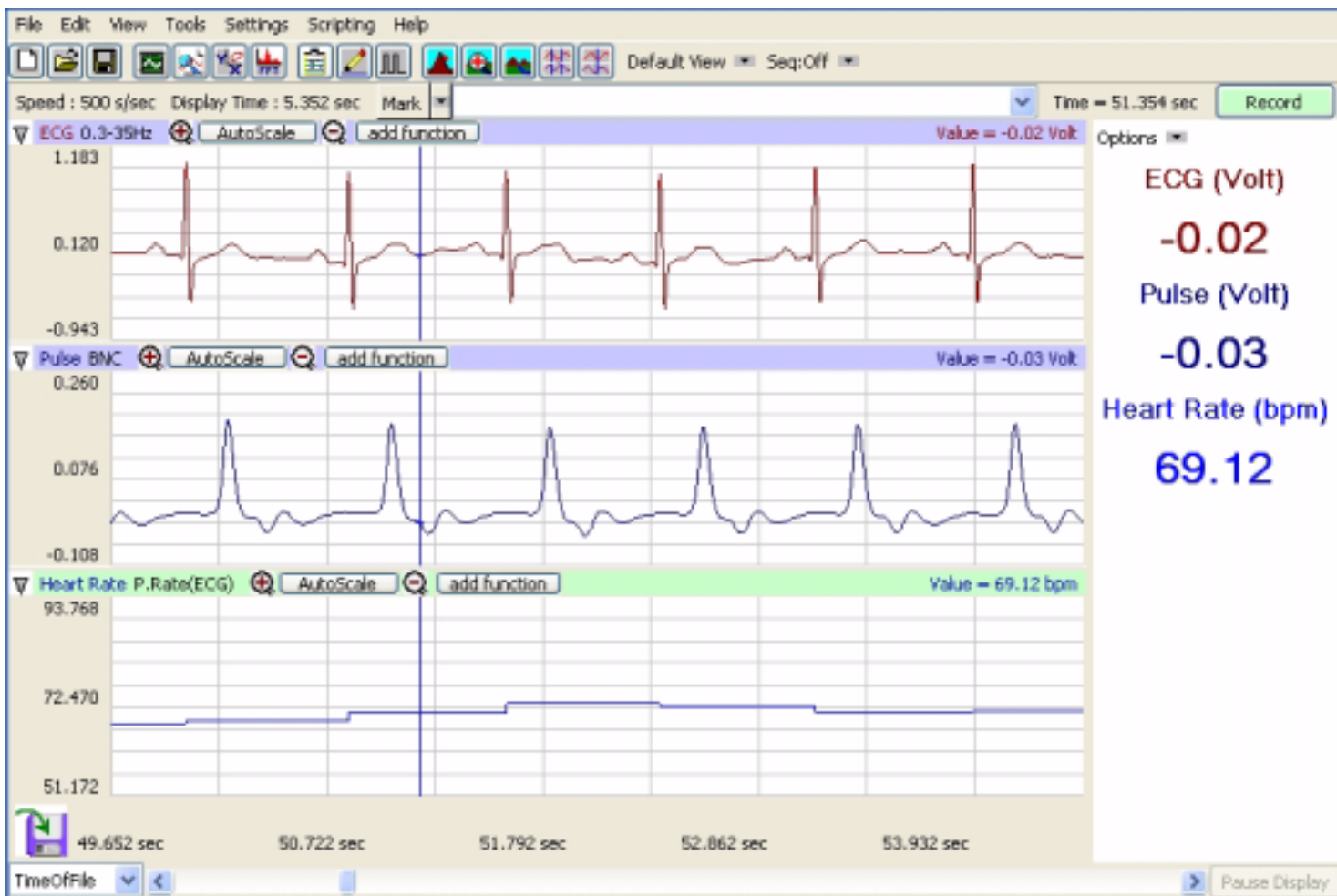
Figure 3-1: The DataTrax2 Main window.

Each channel has its own set of controls located in a **Channel Bar** immediately above each channel's data window (Figure 3-2 on page 21). Y-axis scaling is printed to the left of the data. The **Channel Bar** contains the **Channel Menu**, **Channel Title**, buttons for Y-axis display control (**Zoom-In, AutoScale and Zoom-Out**), **Add Function** button and a **Value Display** area. The **Value Display** box located to the extreme right on the channel bar will display different values depending on the state of the program. While recording, the **Value Display** box shows the value of the last data point collected. Off-line, in **Single Cursor** mode, the **Value Display** box displays the Y-axis value of the data point intersected by the cursor. In **2 Cursor** mode, the **Value Display** box displays the difference between the Y-axis values intersected by the two cursors.
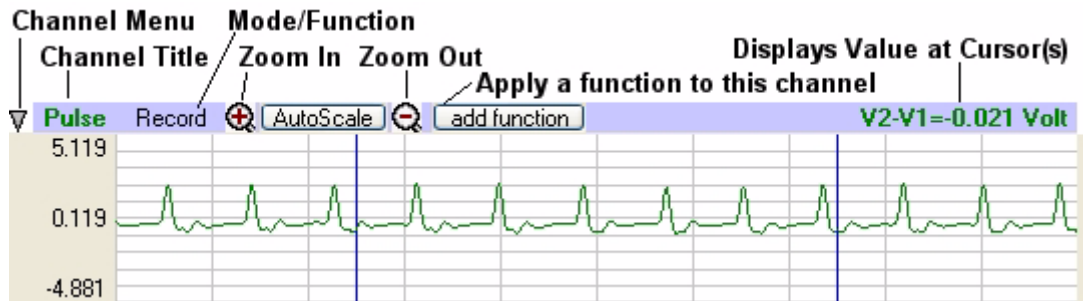
Figure 3-2: The common controls for channels displayed in the DataTrax2 Main window.

Cursors

If someone handed you a paint brush, gestured in a general direction, and said: "Paint that!"; you would probably want to clarify exactly what "that" was. In mathematical terms, the verb "paint" is the **function** and the object "that" is the **argument**. In mathematical operations, it is important to understand exactly the "that" on which you are operating.

This is also true for *DataTrax2* software. Many commands are included in the program that allow the user to zoom, measure and calculate. In order for these commands to work, a mechanism is required for directing the software to focus on a range of data or specific data points of interest. *DataTrax2* uses cursors to identify the data points. The explanation of cursors begins with basic modes of operation and how to apply them in the window where the cursors are used.

Cursor Modes

Cursors are vertical blue lines that pass through all channels. The *DataTrax2* software has two cursors. Icons in the toolbar allow you to choose between using **Single Cursor** or **2 Cursor** modes (Figure 3-3 on page 21). Cursors can also be "garaged," or placed all the way to the left hand side of the data so they measure or report nothing. Cursors in the *DataTrax2* software are always in one of three modes — garaged, single, or 2 cursor.
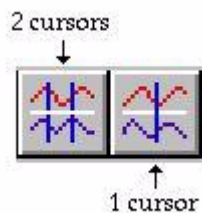

Figure 3-3: Cursor controls on the DataTrax2 Main window toolbar.

To access **Single Cursor** mode, press the **Single Cursor** icon (Figure 3-3 on page 21) in the toolbar. In this mode, the value of the data point on each channel that is intersected by the cursor bar, is reported in the **Value Display** area on the right side of each channel bar. The time

for the position of the cursor is reported in the **Clock Area** in the upper right hand corner of the **Main** window. **Single Cursor** mode is used to determine values and to place marks in the record after recording has stopped.

To access **2 Cursor** mode, press the **2 Cursor** icon (Figure 3-3 on page 21) in the toolbar. In this mode, the **Value Display** area reports the difference in value between **Cursor 1** and **Cursor 2**. In a similar way, the **Clock Area** reports the difference in time between the two cursors. When using **2 Cursor** mode, the cursor farthest to the left is always **Cursor 1** and the other to the right is always **Cursor 2**. **2 Cursor** mode is also used to define the right and left boundaries of a selection of data. This selected area can be used to determine what two values are used to accomplish the **Units Conversion** function.

Moving Cursors   Once cursors have been called, they may be moved by placing the mouse over the cursor bar, clicking, holding and dragging them to the right or left. Cursors may also be moved using the arrow keys on the keyboard. In **2 Cursor** mode, you can change the cursor that is moved by using the UP arrow. Pressing the RIGHT or LEFT arrow key on the keyboard, moves the cursor one data point. Holding the SHIFT key down while using the arrow key causes the cursor to move five data points at a time; and holding the CONTROL key down while pressing the RIGHT or LEFT arrow moves the cursor forward 10 points at a time.

Behavior of   In **Single Cursor** mode, the value displayed represents the value identified on the channel by
Cursors   the one and only cursor. If the data is scrolled so that the cursor goes off screen, the cursor is automatically garaged and the channel display area will display the value of the visible data point furthest to the right.

In **2 Cursor** mode, **Cursor 1** is always the left hand cursor. If **Cursor 2** is moved to the left of **Cursor 1**, it becomes **Cursor 1**. Values reported in the **Value Display** area are always the difference between **Cursor 2** and **Cursor 1** (Cursor 2 minus Cursor 1). If the data is scrolled, so that one of the cursors is no longer on the screen, that cursor is automatically garaged and the display goes to **Single Cursor** mode.

Exercise   Load the data file **Cursor_Exercise.iwxdata** in the Program Files->*WPI*->*DataTrax2*2->tutorial files folder, by selecting the **Open** item in the **File** menu. The data displayed on Channel 1 is a ±2V sine wave. Note that the **Value** in the **Value Display** area corresponds to the voltage of the last data point (farthest to the right) in the window.

• Select **Single Cursor** mode by pressing the **Single Cursor** icon in the toolbar (Figure 3-3 on page 21).

• Record the value that corresponds to the position of the cursor.

• Click, hold and drag the cursor over the highest point in a given cycle of the data. Adjust the position of the cursor bar left or right by using the left or right arrow keys on the keyboard. Adjust the position of the cursor bar so that the **Value** in the **Value Display** area reads +2V.

• Call the **2 Cursor** mode by pressing the **2 Cursor** icon in the toolbar (Figure 3-3 on page 21).

• Position **Cursor 1** so that it is over the maximum value in a given cycle, then position **Cursor 2** over the minimum value as shown (Figure 3-4 on page 23). The **Value** reported will be 4V.

• Now, drag **Cursor 2** to the minimum value before **Cursor 1**. When you release **Cursor 2**, it becomes **Cursor 1** and the new **Value** reported is -4V.

Main Window Display Considerations

Recorded data has two important dimensions: length and height. The length is the **Time** axis and the height is the **Amplitude**. Each of these has its own set of controls

Managing Display Time

The events recorded using *DataTrax2* software may occur over varying time frames. For example, recording the discharge curve of a 9-Volt battery could take hours, while recording the QRS complex in a human electrocardiogram might take only a fraction of a second. *DataTrax2* software allows the recording of both very slow and very fast events while displaying the recorded results in a format that is easily interpreted. To manage the time display in the *DataTrax2* software, we use a parameter called **Display Time**. **Display Time** is the amount of time represented by one full screen of data. When the program opens, the default **Display Time** is set to 10 seconds. **Display Time** can be changed by using the display controls in the toolbar (Figure 3-4 on page 23), or by manually entering a **Display Time** in the place provided at the top of the **Channel** page of the **Preferences** dialog window, which is launched by selecting **Preferences** under the **Edit** menu.

Clicking the **Half Display Time** icon (big mountain) halves the screen time. If you clicked this icon once, a 10-second full-screen display would become a 5-second, full-screen display. This doubles the screen resolution, but cuts the amount of data seen on the screen in half. Clicking the **Double Display Time** icon (little mountain) doubles the screen time. In this case, a 10-second, full-screen display would become 20-seconds wide. This reduces the screen resolution by half, but doubles the amount of data that you see on the screen. Clicking the **Zoom Between Cursors** icon fills the display with the data located between the cursors.

🔺 Half Display Time
➕ Zoom Between Cursors
⛰️ Double Display time
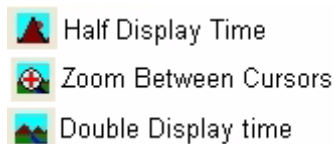
Figure 3-4: Display Time icons in DataTrax2 *Main window toolbar.*

Limits

Using the **Double Display Time** control compresses the data. The display time can be doubled as many times as requested until the limit of the maximum size of the data file or 1,000,000 data points is reached.

Using the **Half Display Time** tool expands the record as many times as requested. But, the *DataTrax2* program will not display less than 10 data points on a screen of data.

Double-clicking the **Display Time** area at the top of the **Main** window resets the X-axis time display to the default value originally entered on the **Channel** page of the **Preferences** dialog window.

Examples

Here are two examples demonstrating the use of the **Display Time** feature. In the first example (Figure 3-5 on page 24), a human electrocardiogram is recorded with a 160-second screen time. Notice that the record is compressed and it is impossible to resolve detail in individual events. By clicking **Half Display Time** five times, the screen time is reduced from 160 seconds to about five seconds, making individual events clearly visible.
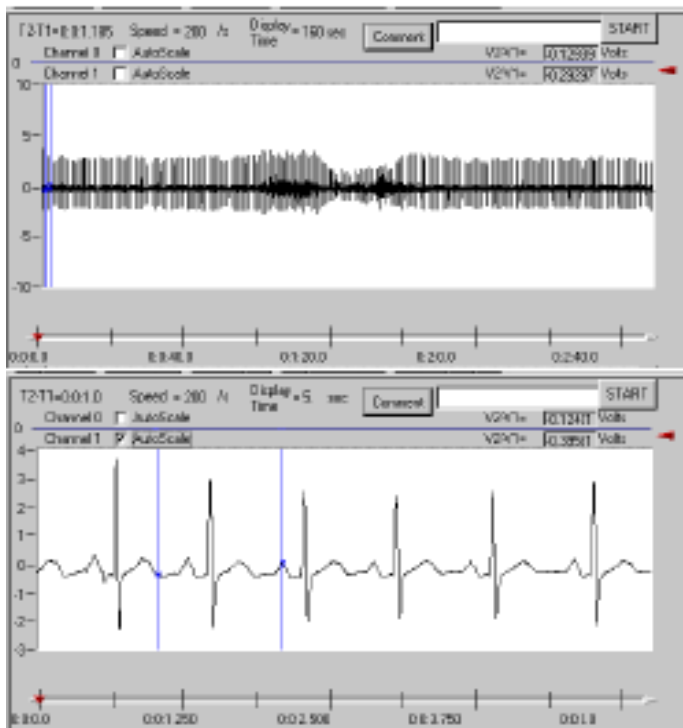
Figure 3-5: ECG viewed with a long screen time (top) and a short screen time (bottom).

In the second example (Figure 3-6 on page 24), a one hour battery discharge curve is shown in two views. The 2-second screen time shows almost a straight line, but clicking **Double Display Time** eight times reveals the entire curve. The **Display Time** parameter, with its two controls, effectively handles the time display.
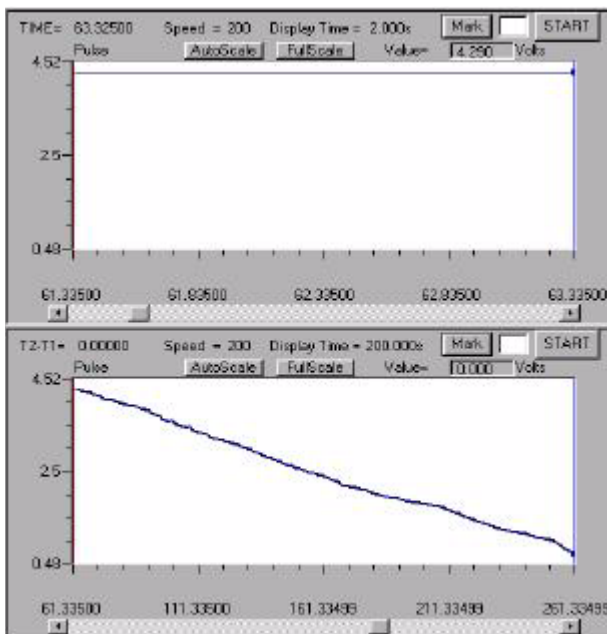

Figure 3-6: Voltage drop of 9-Volt battery viewed with short (top) and long (bottom) screen times.

Managing Amplitude Display

The vertical display of your signal is managed in a number of ways. In each channel toolbar (Figure 3-7 on page 25), there are three icons. They represent the **Zoom-In** (x2), **AutoScale** and **Zoom-Out** (x2) features. Clicking and dragging the waveform can be used to offset the recorded wave up or down. In addition there is a **Scale** menu available from either the **Channel** menu or by clicking on the values in the Y-Axis on the left hand side of each channel. There are five options available from the **Scale** menu: **Set Scale**, **Preferred Scale**, **Zoom In**, **AutoScale** and **Zoom Out**.
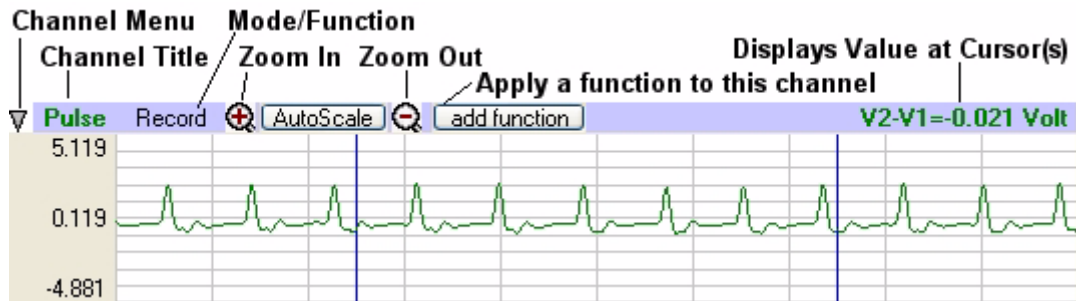
Figure 3-7: The channel toolbar in the DataTrax2 Main window.

AutoScale

AutoScale, as used in the *DataTrax2* software, reviews the current screen of data on the channel of interest and locates the highest and lowest data points in that screen. It then adjusts the Y-scale range to include those points. This optimizes the look of the waveform on the screen giving you the best possible curve.

Zoom-In/Out

From any view of data, pressing the **Zoom-In** button will double the displayed size of the trace. Pressing the **Zoom-Out** button will reduce the displayed size of the trace by a factor of two.

Preferred Scale

For any data, there is a basic range within which the Y values of the signal will occur. For example, if measuring room temperature, the user might select a range of 50°F to 100°F because it is doubtful that the signal will move beyond these limits. If an event of interest occurs during the recording, the area of interest can be expanded using the **Zoom** or **AutoScale** tools. To return to the **Preferred Scale** set by the user, simply select the **Preferred Scale** option from the **Scale** Menu.

Set Preferred Scale

To set the range of the **Preferred Scale**, choose the **Set Scale** option from the Scale menu and enter the upper and lower limits of Y-scale in the **Set Scale** dialog window that appears.

Scroll Up/Down

When the **Zoom** tools are used, the data may drift above or below the center of the display area, or even go out of the range of display. To adjust the position of the trace on a channel, click and drag the waveform to the desired position.

Views

DataTrax2 can acquire data on up to eight channels on the *WPI* 118 (four channels on the *WPI* 214), but it can calculate data on up to 128 channels. The extra channels can be used to display computed functions that are mathematically derived from the raw data. For example, the arterial pressure from eight different animals could be recorded on channels 1-8. Simultaneously, the

Rate function could be used on Channels 9-16 to calculate the heart rate of each animal from its recorded blood pressure on Channels 1-8. As with any multi-channel display, the more channels displayed means there is less space that can be given to each channel. In the case of a 16-channel **Main** window display, it is hard to resolve detail in the trace of recorded data in each channel.

   DataTrax2 solves this problem by allowing the user to create many different arrangements of channels that can be displayed on the screen at one time. Each arrangement of channels that is displayed is known as a **View**. Using the example from the previous paragraph, a **View** could be created that displayed the arterial pressure from Channel 1 and the calculated heart rate for the same animal displayed on Channel 9. The data from Channels 1 and 9 would appear in the first and second data display areas, respectively. The other fourteen raw data and computed heart rate channels would not be displayed. For the data recorded from another animal on Channel 2, another view with Channels 2 and 10 (its matching rate channel) can be created.

Using Views   To create a new **View**, click on the **View Control** in the Toolbar and choose **New View**. Name the view in the **Edit View Name** dialog. You can now edit the view by choosing the **Edit View** option from the **View Control**, by resizing the channels in the **Main** window, or by hiding or minimizing the channel from the **Channel** menu.

Edit View   You can access the **Edit View** dialog from the **View Control** on the Toolbar. For each view you can add or remove the Channels available in the View. You can also organize them vertically. Each view has its own Voltmeter Panel and you can choose the channels in the voltmeter and their position.
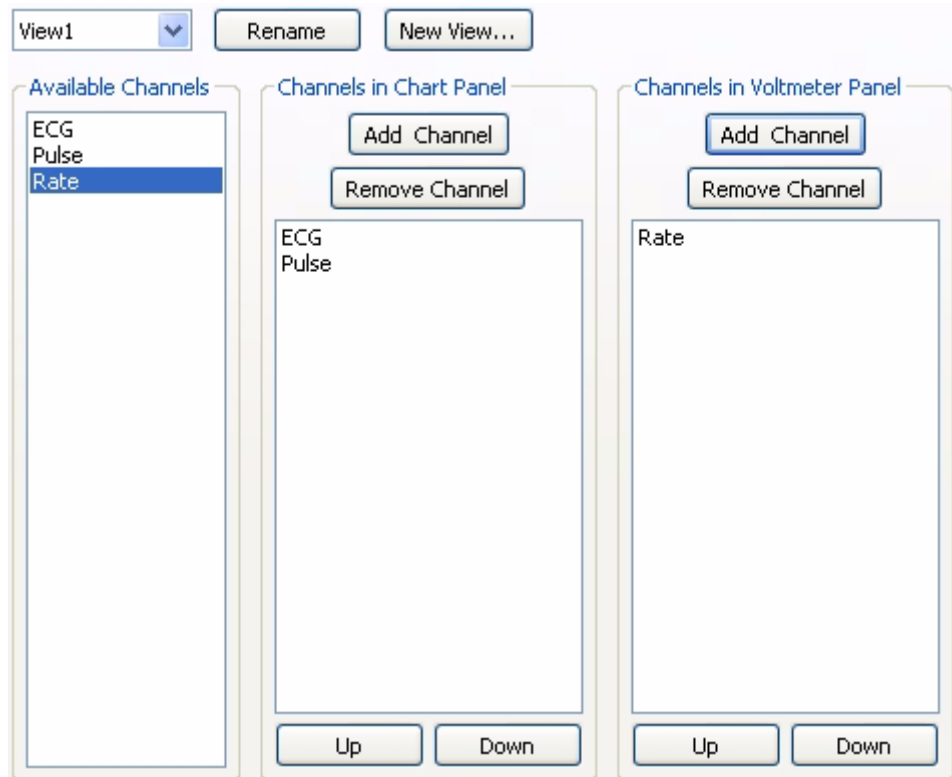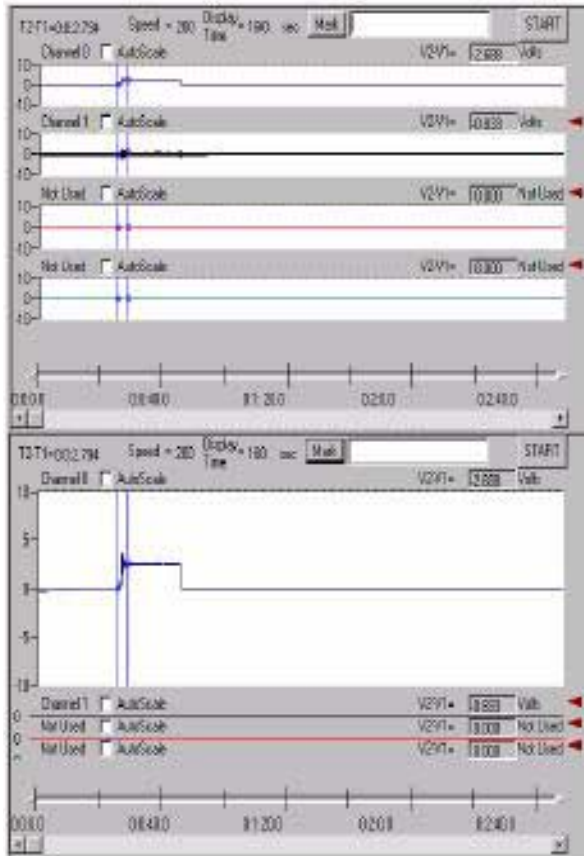
Channel Sizing

Figure 3-8: The height of the data displays for four channels in the Main window as set by the default setting (top), and after the handle for Channel 1 was slid down (bottom).

In addition to scaling the height of the signal to fit in a given channel area, the amount of display area allotted to each channel in the **Main** window can be controlled by clicking and dragging on the top of the channel bar. To change the allocated space for a channel, click on the top of the channel bar and drag it up or down (Figure 3-8 on page 27).

Converting Units

When used with *WPI* or *National Instruments* hardware, the *DataTrax2* software functions as a calibrated voltmeter, which means the software will accurately display the exact voltage that the user presents to the analog-to-digital converter. The displayed (and default) units will always be **Volts**. While this is useful in many cases, it is not always the most appropriate unit.

If the *DataTrax2* software is used to record the output of a transducer designed to measure a physical parameter, such as force or pressure, other units are more appropriate. In these cases, Volts can be changed into milligrams or grams. The *DataTrax2* software can handle these conversions easily, provided that the function (or transducer) that relates voltage to real units is linear.

The first step in converting voltage input to real units is to record a portion of data at two known values. In the case of a force transducer, record output at two known forces such as zero and 100 grams. The recorded trace may look something like Figure 3-9 on page 28.
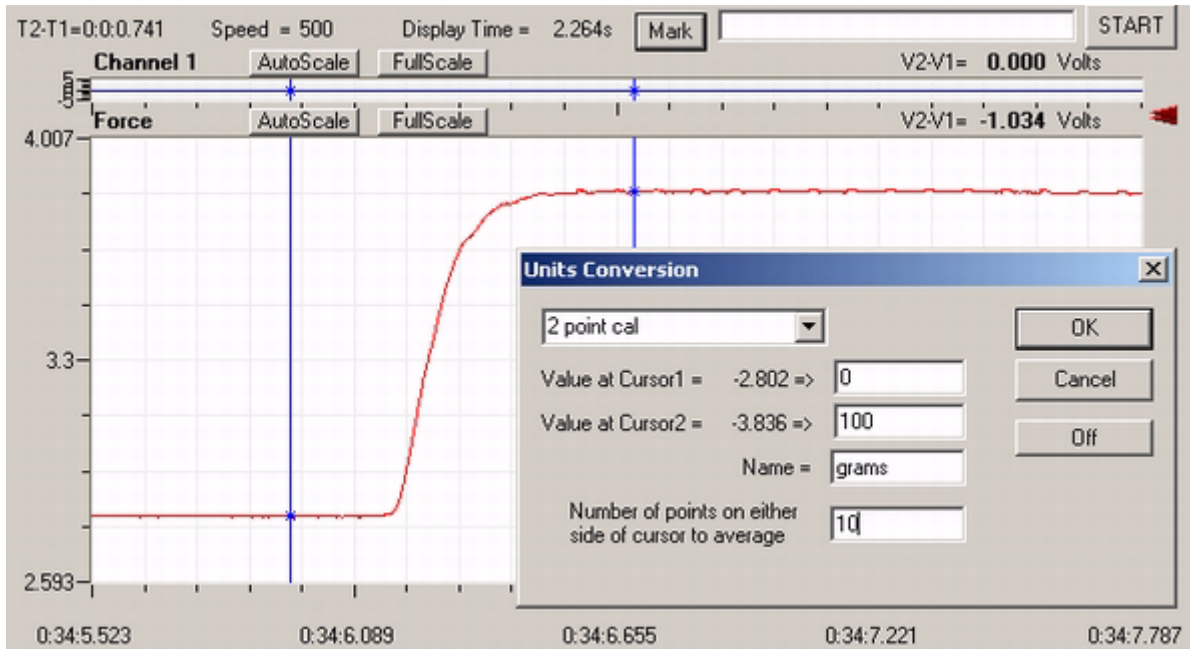
Figure 3-9: Data file of weights hung from a transducer and the units conversion dialog window that sets volts to grams.

Once recording is complete, proceed to the **2 Cursor** mode in the **Main** window. (See the Cursors section of this chapter.) The **Units Conversion** dialog window cannot be entered without being in **2 Cursor** mode. Position **Cursor 1** over one of the known values, and **Cursor 2** over the other known value.
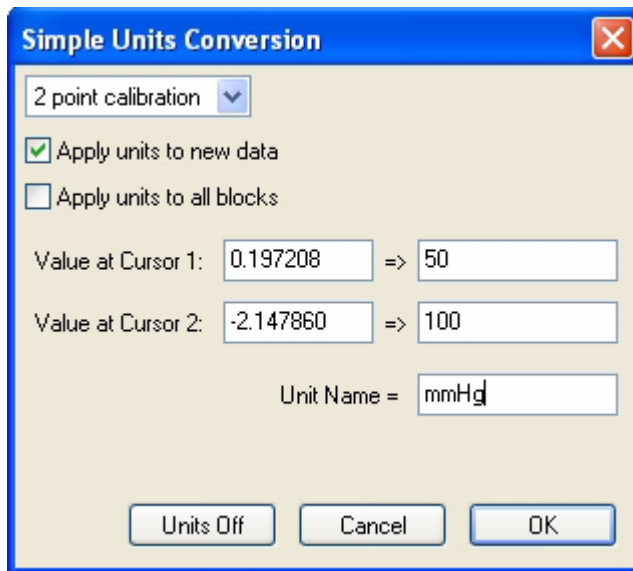
Figure 3-10: The Units Conversion dialog window showing the 2-point calibration boxes.

Select the **Units** menu from the **Channel** menu or by clicking on the Value display area on the channel bar. Select the **Simple Units Option**. The **Units** dialog window will appear. The typical conversion of the output of a transducer is a 2-point calibration, which is the default conversion method (Figure 3-10 on page 28). Below that menu is an area where the values for the positions of the cursors are listed. The values on the left are the voltage values at the positions of

Cursors 1 and **2**. Enter the corresponding values in real units into the two value boxes on the right. In the **Name** area, enter the name of the unit to be displayed on the Y-axis. If a unit name is not entered, Volts will be used as a default name.
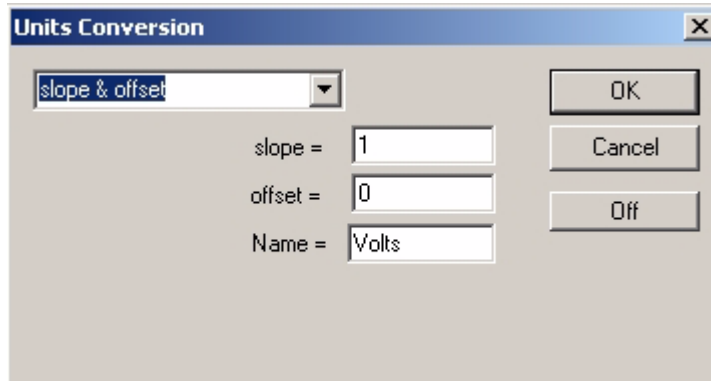
Figure 3-11: The Units Conversion dialog window showing slope and offset boxes.

It is also possible to set the **slope** and **offset** directly, if those values are known (Figure 3-11 on page 29). For example, if a pressure transducer produces 5mV (0.005V) per mmHg, the slope would be 0.005 and the units would be mmHg. To use **slope** in a units conversion, the **slope** must be expressed as **Volts/unit**. Ideally, when a sensor puts out zero volts, the value of the converted units would also be zero. For many sensors this is true. However, there are many sensors that can have their offset changed by ambient conditions, such as changing barometric pressure. To correct for sensor offset, determine the value (in converted units) that *DataTrax2* reports on the screen when the sensor should be reading zero. Enter this value in the offset area of the **Units Conversion** dialog window.
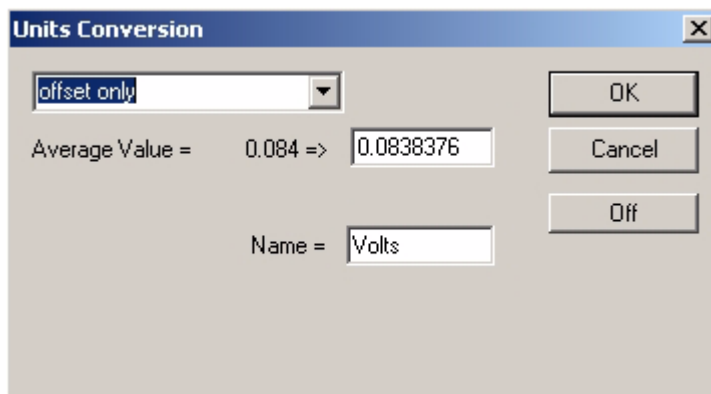
Figure 3-12: The Units Conversion dialog window showing offset only boxes.

Sometimes, it is necessary to keep the units conversion relationship and change the offset, like turning the offset control on an amplifier. For example, if we want to set a particular region to zero, set the two cursors around the desired region of data. Select **Units** from the **Right-Click** menu. Choose **Offset Only** from the drop-down menu (Figure 3-12 on page 29). The average value between the two cursors can now be set to the required offset value (zero, in this example).

The units are always applied to the selected block(s). To apply the units to all blocks, select the **Apply units** to all blocks checkbox. To apply units to new data which will be recorded select the **Apply units to new data** checkbox.

Units can also be set from the **Channel** page of the **Preferences** dialog window. Each channel can be set with the conversion factors provided by the transducer manufacturer (see the section on **Preferences** in this manual).

All of the recorded data, acquired by the hardware as raw data, are displayed as voltages. From time-to-time, it may be desirable to turn off the **Units Conversion** and simply view the raw data. To view raw data, open the **Units Conversion** dialog window (Figure 3-10 on page 28). Notice that the program reports the current values in voltage at each cursor and the representative converted values. Selecting **Units Off** in this dialog window returns the original display to **Volts** as the default unit.

From the **Units** menu you can also **Zero all Blocks**, which takes the value between the cursors and sets that value to zero for all blocks.

From the **Units** menu, you can also turn off the units for all blocks.

Inverting the Trace

When recording physical parameters, such as temperature, pressure or force, it is best if the polarity of the data display matches the real-world behavior of the parameter. For example, if the observed temperature goes up, the trace on the computer screen should go up. Increasing pressure or force should also produce a positive or upward deflection of the trace.

Depending how sensors and amplifiers are wired, this may or may not be the case. In the event that the data display has the wrong sense or polarity, the trace can be inverted by selecting **Invert** from the **Channel** menu, **Units** menu, or the **Right-Click** menu of any data channel. As with **Units Conversions**, the **Invert** function can be switched off at any time.

Marks

The *DataTrax2* software can record large amounts of data, but the data of interest must easily be located and retrieved to be useful. Also, information needs to be extracted from the data to make it useful. To locate specific sections of data and to differentiate parts of the record from one another, it is possible to put **Marks** on the data while *DataTrax2* is recording. **Marks** are one of three navigation tools provided by the *DataTrax2* software. In addition to being made on-line, **Marks** can be inserted and edited after the recording has stopped.

Making Marks On-Line

Marks (and their associated comments) can be placed on the recording without interrupting the flow of data to the computer.

As soon as the **Start** button is clicked and data recording begins, the *DataTrax2* program sets an active text cursor on the **Mark** line, next to the **Mark** button on the **Main** window. On the **Mark** line, the user can type a comment describing an upcoming step in the experiment. The comment will be associated with the next mark that is placed on the recording.

The mark is placed on the recording when the **Enter** (or **Return**) key on the keyboard is pressed, or the **Mark** button on the **Main** window is clicked. The mark will be signified by a vertical black line that is inserted on the data at the moment the **Enter** key is pressed or the **Mark** button is clicked. When recording is halted, the typed comment that was loaded on the **Mark** line prior to the event can be seen in the **Text Display** area at the bottom of the screen.

A mark is time-locked to the record at the moment the **Return** or **Enter** key is pressed. An unlimited number of marks can be attached to the record. The vertical line on the recording that is associated with a mark is provided as visual confirmation that the mark has been placed.

Preset Marks   If you know the marks that you will be adding to the record prior to recording, you can store the mark text as preset marks. To create a preset mark, type in the mark text in the **Mark** window, and choose the **Add to Mark Presets** option by clicking on the down arrow next to the **Mark** button. This will save the typed text as a preset mark, which can be called by clicking on the **Mark** drop-down box and choosing the desired preset mark. This loads the **Mark** window with the desired mark text, now clicking the **Mark** button or hitting **Return** will add the preset mark to the record.

You can delete a preset mark by selecting **Preset Mark** from the **Mark** drop-down box and choosing the **Delete from mark presets** option by clicking on the down arrow next to the **Mark** button.

Making Marks Off-Line   Marking events as they happen is a necessity for events that are time critical, like drug deliveries or experimental interventions. Information about the experiment that is important, but not time critical, can be marked on the recording after the recording is completed. An example of the type of comment to be added to the record would be a change in room temperature.

To add this information to the record after recording is completed, select the **Single Cursor** mode from the *DataTrax2* toolbar. Position the single cursor bar on the record where the mark is to be positioned. Type the text (a maximum of 50 characters), associated with the new mark, on the **Mark** line in the **Main** window. Click the **Mark** button and the mark and its text comment are inserted at the position of the cursor.

Editing Marks   Marks already on the record can be changed by going to the **Marks** window, available from the **Window** menu or the *DataTrax2* toolbar. The **Marks** window is a dialog box containing a list of all marks in a record (Figure 3-13 on page 31).

Time	Mark	Channel	Value...
0:18:55.9...	Mark#0	All	0.000...
0:20:3.959	Mark#1	All	0.000...
0:41:2.364	Mark#2	All	0.000...
0:41:3.324	Mark#3	All	0.000...
0:41:4.159	Mark#4	All	0.000...
0:41:4.925	Mark#5	All	0.000...
0:41:25.4...	Mark#6	All	0.000...
0:41:26.5...	Mark#7	All	0.000...
0:41:27.3...	Mark#8	All	0.000...
0:41:27.9...	Mark#9	All	0.000...
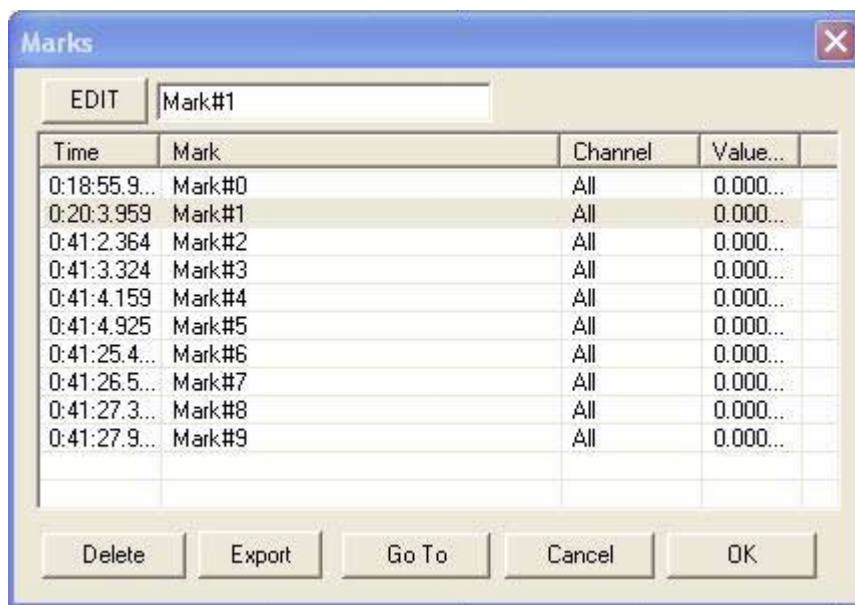
Figure 3-13: The DataTrax2 Marks window.

To edit a mark, highlight the mark, by clicking twice on the mark text.

Marks that are placed on the recording can serve as "sign posts," indicating where important sections of data are located. Click on the down arrow next to the **Mark** button and choose the mark you want to "go-to." *DataTrax2* software will find the data point associated with that mark and display that section of data in the **Main** window. In this manner, the user can move between important areas of an experiment without hunting or scrolling for the areas of interest.

Positioning Mark Comments

On presentations or printed copies of the data record, it is useful to position the text of the comment associated with a mark directly over the data to which it applies. This is particularly useful if more than one channel of data was recorded and the mark does not apply to all channels.
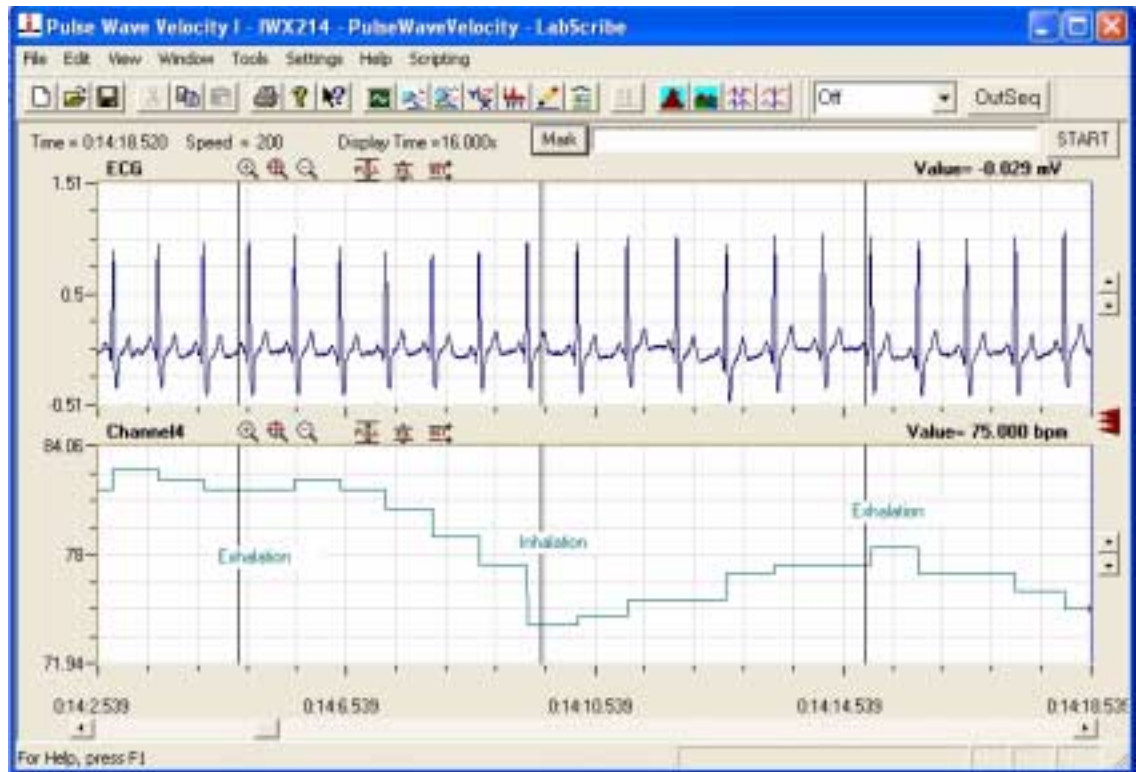
Figure 3-14: Comments moved on to the channel data display in the Main window.

When marks are created, the comments appear in the **Text Display** area at the bottom of the **Main** window above the **Time** axis. After recording has stopped, comments associated with marks can be dragged into the data area of the record. The text of these comments can be read and printed in any window in which the data appears (Figure 3-14 on page 32).

Sometimes, it is necessary to return the marks in a particular view to the **Text Display** area at the bottom of the **Main** window. To do this, click on the down arrow next to the **Mark** button and choose **Reset Location of displayed marks**. This command returns only the mark comments on the data window of the current screen of data to the bottom of the **Main** window. All other marks will remain where they were originally positioned.

Exercise

1 Click **Start** and record a few minutes of data.

2 As data is being recorded, type the word "Test 1" on the keyboard and strike the **Return** or **Enter** key on the keyboard.

3 Wait one minute, type "Test 2" and strike the **Return** or **Enter** key.

4 Click **Stop**. Scroll through the data using the scroll bar on the bottom of the **Main** window until you locate "Test 1" in the **Text Display** area.

5 Click on the down arrow next to the **Mark** button and choose "Test 2". Notice that the record in the **Main** window has moved to the "Test 2" mark.

6 Using the mouse, click and hold on the comment "Test 2" at the bottom of the screen. Continue holding the mouse button down, and drag the mark comment to a new position on one of the available channels.

7 Release the mouse button and the mark is locked on the selected channel. Comments positioned in this way will remain where they are placed and will print exactly as you see them.

8 Try this exercise again with a mark created off-line. To create an off-line mark, open the **Single Curso**r mode by clicking the **Single Cursor** icon in the *DataTrax2* toolbar. Position the cursor where the off-line mark is to be positioned. Type some text on the comment line at the top of the screen, and press the **Return** or **Enter** key. The mark appears at the cursor location and the comment appears at the bottom of the screen.

Sorting and Exporting Marks

The **Marks** window displays the **Time** that a mark was made, the text comment attached to the **Mark**, the **Channel** on which the mark was made and the **Value** of the amplitude at the mark. Marks can be sorted by **Time**, **Channel** or the text comment of the **Mark** by clicking on the column titles. Click once to organize marks in ascending order and a second time to change to descending order.

Using the standard <u>Windows</u> commands, SHIFT-CLICK and CONTROL-CLICK, specific marks can be highlighted. Once selected, use the **Export** button to send selected marks to a text file which can be opened in <u>Excel</u> or another spreadsheet program.

Voltmeter

Large digital readouts of the amplitudes recorded on each channel can be displayed on the right side in the **Main** window. Select the **Voltmeter** function from the **View** menu to display these readouts. While data is being recorded, the most recent value is displayed. Also it displays the amplitude of the trace on each channel at the position of a **Single Cursor**, or can display the difference in amplitudes between the positions of **2 Cursors** on each channel. As discussed in the section on Views, the voltmeter display can be independently organized for each view. You can set the fonts for the title and the value display as well the number of columns displayed from the **Options** menu in the **Voltmeter** panel.

Online XY

In **OnlineXY** mode, the Y-values from one channel in the **Main** window are plotted in real time against the Y-values from another **Main** window channel. The resulting **XY plot** is dramatically different from a linear plot of data against time. Select the **OnlineXY** function from the **View** menu to display this plot. Select the Y and the X channel from the drop-down box, at the bottom of the plot window. The **Main** window display time also applies to the **OnlineXY** plot.
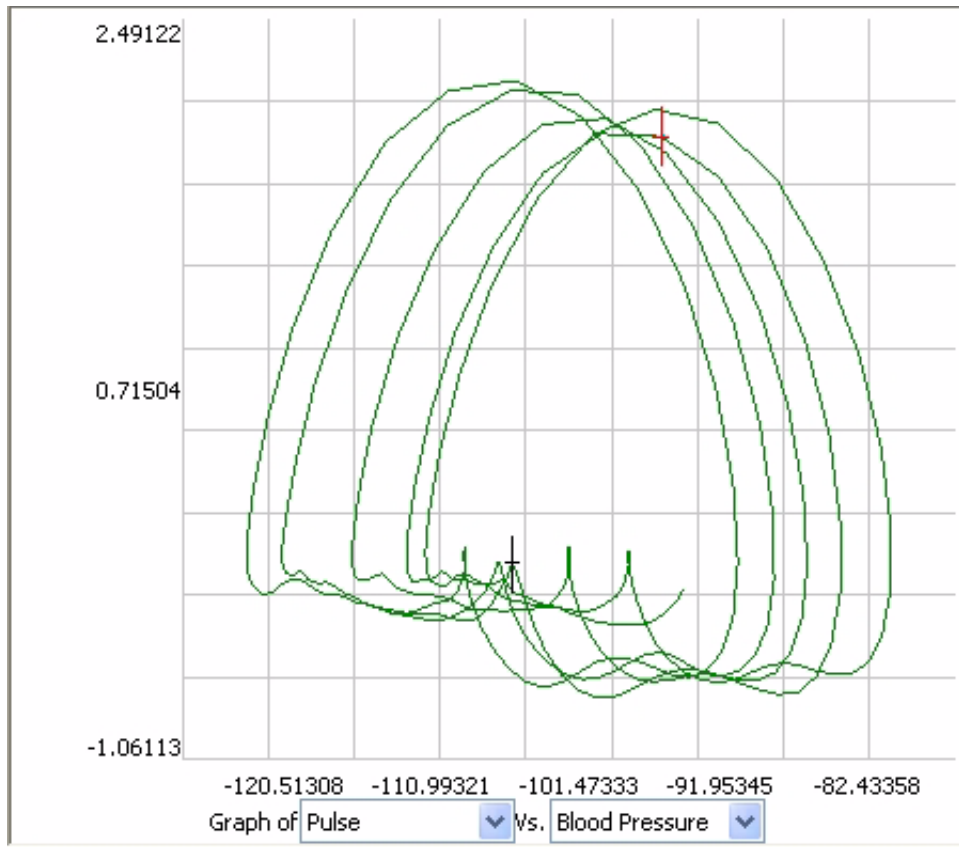
Figure 3-15: Online XY

Display Considerations in Other Windows

Data is only recorded in the **Main** window. However, other windows can receive selected data from the **Main** window for closer examination or display in another format, like an XY plot or FFT. There are several viewing options available in these other windows that are not available in the **Main** window. The actual discussion of analytical functions found in these windows is deferred to the **Analysis** chapter later in this manual.

Chapter 4: Controls

Menus

File

The *DataTrax2* software supports all of the elements found in a standard **File** menu:
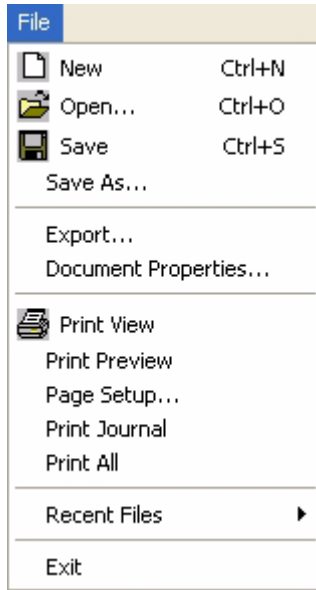
Figure 4-1: The DataTrax2 *File menu.*

New: Opens a new file. Only one acquisition window may be open at a time. This function is also available from the toolbar.

Open: Opens a previously recorded file and is also available from the toolbar.

Save: Saves data to the file currently open and is also available from the toolbar.

Save As: Saves data to a new file with a different name or format.

Export: Allows the user to export the data viewed as text, a picture or in a variety of formats appropriate to external analysis programs.

Doc Properties: Shows sample speed, gain and other information concerning the data in the file.

Print View: Prints the window in the foreground and is also available from the toolbar.

Print Preview: Previews the image to be printed.

Page Setup: Dialog box for control of the printing characteristics.

Print Journal: Prints the Journal.

Print All: Prints the complete data file including the Journal.

Recent Files: Displays the last four files used.

Exit: Quits the program.

Edit

The *DataTrax2* software supports elements found in a standard **Edit** menu:
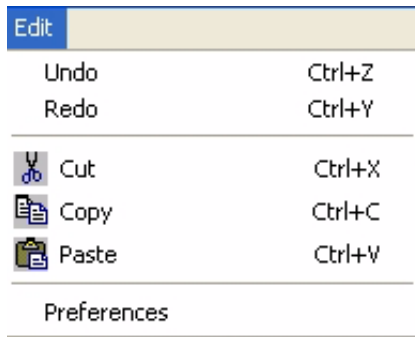
Figure 4-2: The Edit menu.

Undo: Un-does the last command (Journal Only).

Redo: Re-does the last command (Journal Only).

Cut: Cuts the selection (Journal Only).

Copy: Copies the window in the foreground or selection in the journal to the clipboard for pasting. Also available from the toolbar.

Paste: Pastes the contents of the clipboard to the Journal (Journal Only).

Preferences: Calls a multi-page dialog box that controls settings for **Channels**, **Sampling**, **Stimulator**, **Sequences**, **Options** and **Events** items.

View

The **View** menu supports display elements unique to the *DataTrax2* software.
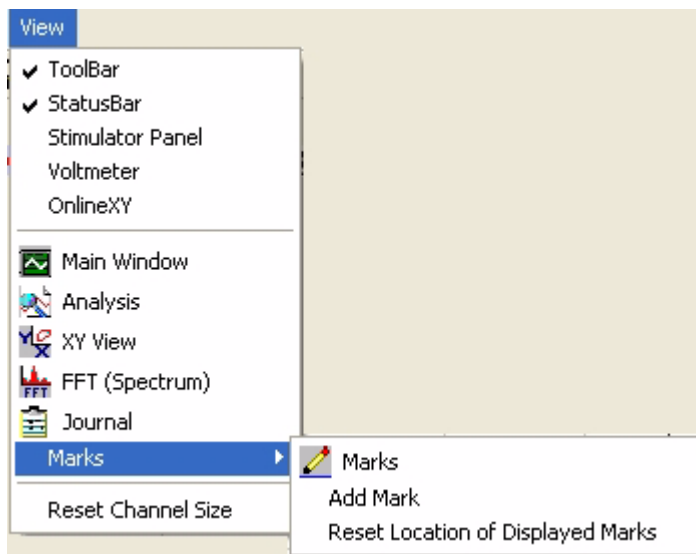
Figure 4-3: The View menu.

ToolBar: Hides or displays the *DataTrax2* toolbar.

StatusBar: Hides or displays the <u>Windows</u> status bar.

Stimulator Panel: Hides or displays the Stimulator toolbar.

Voltmeter: Displays amplitude values in large type and in digital voltmeter fashion.

OnlineXY: Displays realtime XY plot to the right of the **Main** window channels.

Main Window: Brings the **Main** window to the foreground. Also available from the *DataTrax2* toolbar.

Analysis: Brings the **Analysis** window to the foreground. Also available from the *DataTrax2* toolbar.

XY View: Brings the **XY View** window to the foreground. Also available from the *DataTrax2* toolbar.

FFT (Spectrum): Brings the **FFT** window to the foreground. Also available from the *DataTrax2* toolbar.

Journal: Brings the **Journal** to the foreground. Also available from the *DataTrax2* toolbar.

Marks: Opens a submenu with controls for editing, adding, or resetting marks.

Reset Channel Size: Returns all open channels to their default screen spacing.

Tools

The **Tools** menu provides functions that can find and move specific data points to tables created in the **Journal**.
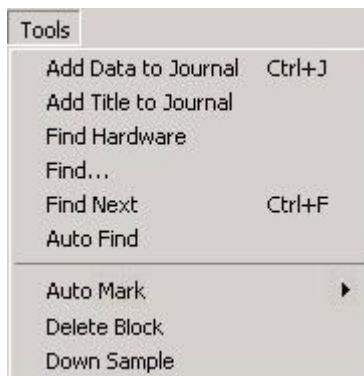
Figure 4-4: The DataTrax2 Tools menu.

Add Data to Journal: Sends the calculated values for the selected **Table Functions** from the **Analysis** or **ScopeView** windows to the **Journal**.

Add Titles to Journal: Sends the titles of the calculated values for the selected **Table Functions** from the **Analysis** or **ScopeView** windows to the **Journal**.

Find Hardware: Identifies and initializes the computer's connection to the *WPI* data acquisition device.

Find: Calls a dialog box for programming the placement of cursors on data displayed in the **Analysis** or **ScopeView** windows.

Find Next: Finds the next data point in the file which meets the criteria set in the **Find** box.

Auto Find: Automatically finds each successive data point in the file that meets the criteria set in the **Find** dialog box. Calculated values from the **Table Functions** selected in the **Analysis** window are automatically added to the **Journal** for each matching data point.

Auto Mark: Calls a list of functions that can automatically locate characteristic data points, like the LVEDP or QRS, in a data file displayed in the **Analysis** or **ScopeView** window.

Delete Block: Can be used in the **Analysis** window to delete the current block. **Warning: This cannot be undone!**

DownSample: Used to downsample the data. This reduces the information in the file and should be used with caution. **Warning: This cannot be undone!**

Settings

The **Settings** menu lets you load and edit files containing preset recording and analysis parameters. Selecting a file from the list programs *DataTrax2* to record data in a specified manner. Settings files have been preprogrammed for each *DataTrax2* experiment and specific settings are associated with each lab exercise.
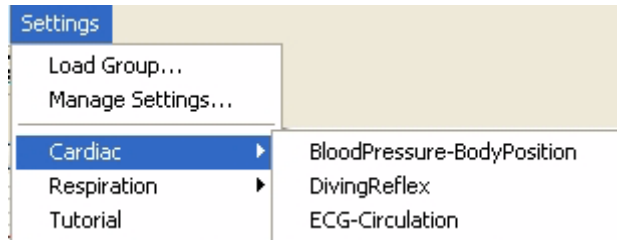

Figure 4-5: The Settings menu.

Load Group: Loads a group of existing setting files. Each file contains the recording and analysis parameters needed in *DataTrax2* to record a specific experiment or type of data.

Edit Group: Allows the user to edit existing settings files in a group, remove them from a group or to add new ones to a group.

Cardiac, Respiration: Examples of the categories within a settings group.

Tutorial, BloodPressure-BodyPosition, Diving Reflex etc.: experimental settings in a group.

Scripting

Not yet implemented.

Help

The **Help** menu provides links to information about the *DataTrax2* software and hardware.
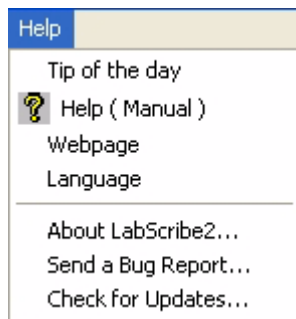
Figure 4-6: The Help menu.

Tip of the Day: Selecting this function displays an important bit of information.

Help (Manual): Sends user to the on-line version of the *DataTrax2* software manual. Also available from the toolbar.

WebPage: Takes the user to the *WPI* home page.

Language: Allows user to change the language.

About *DataTrax2*: Displays version and copyright information pertaining to the *DataTrax2* program.

Send a Bug Report: Sends a bug report. You need to be connected to the internet.

Check for Updates: Checks the *DataTrax2*.com website for the latest version of the software. You need to be connected to the internet.
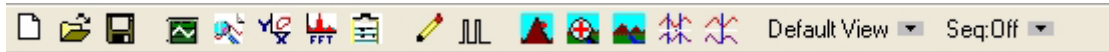
Toolbars

Main Toolbar

Figure 4-7: The toolbar in the DataTrax2 *Main window.*

New File: Opens a new file. Only one acquisition window may be open at a time.

Open File: Opens a previously recorded file.

Save File: Saves data to the file currently open.

Main Window: Brings the **Main** window to the foreground.

Analysis: Brings the **Analysis** window to the foreground.

XY View: Brings the **XY View** window to the foreground.

FFT (Spectrum): Brings the **FFT** window to the foreground.

Journal: Brings the **Journal** to the foreground.

Marks: Brings the marks list to the foreground.

Stimulator Preferences: Hides or displays the **Stimulator** toolbar.

Half Display Time; Reduces the time displayed on the screen by a factor of 1/2, each time the icon is clicked.

Zoom between cursors: Zooms to the area selected by the two cursors.

Double Display Time: Increases the time displayed on the screen by a factor of two each time the icon is clicked.

2 Cursor Mode: Click icon to display two cursors on the data window. Time and voltage differences between the data points intersected by the cursors are measured.

Single Cursor Mode: Click icon to display one cursor on the data window. Absolute time and voltage from the beginning of the trace to the cursor are displayed.

Views: (Shown in the above toolbar as default view).
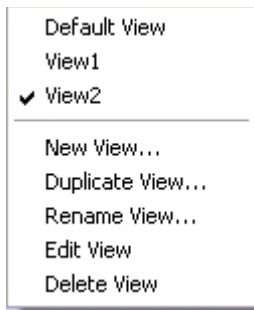
Clicking on the down arrow next to the view name, brings up the **View** control menu. You can switch the current view from this menu, create a new view from the default view, duplicate the current view, rename the current view, edit the current view, which brings up a **View** page of the **Preferences** dialog, or delete the current view.

Sequences:
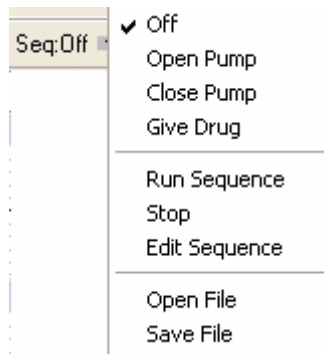
The **Output Sequence Toolba**r is a drop-down list containing sequences of analog or digital outputs defined in the **Sequence** page of the **Preferences** dialog window. Clicking the **Sequence** button will fire the output sequence selected from the list. Selecting a new sequence in the drop-down list also automatically fires that sequence.

Stimulator Toolbar

Pulse Mode

Figure 4-8: The Stimulator Pulse toolbar in the DataTrax2 Main window.

Amp: Amplitude of the stimulus pulse in Volts.

#pulses: Number of Pulses.

Width(ms): Width of the pulse in milliseconds (ms).

F(Hz): Frequency of stimulation in Hertz (Hz) (1Hz=1 cycle/second, in this example, five pulses would be given every second.

HP: Holding Potential

Apply: When clicked applies any changes to the stimulus protocol made through the Stimulator Toolbar. Clicking **Apply** while recording will send a stimulus pulse from the output.
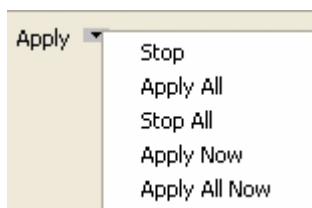
Clicking the down arrow next to the **Apply** button, shows a menu. All options in this menu may not be available for all hardware. On hardware that allows it, **Apply** will wait until a "good" time to apply the changes, i.e., until a pulse is completed, before changing the number of pulses.

•**Stop**: Stop the stimulator.

- **Apply All**: Applies the changes and fires all stimulators. As with **Apply**, on hardware that supports it, the change will happen at the next "best time."

- **Stop All**: Stops all stimulators.

- **Apply Now**: Forces the stimulator to apply the changes as soon as possible.

- **Apply All Now**: Forces all the stimulators to apply the changes as soon as possible.

Constant Mode

Figure 4-9: The Stimulator Pulse toollbar in the DataTrax2 Main window.

Amplitude: Amplitude of the stimulus in Volts.

Apply: Applies any changes to the stimulus protocol made through the **Stimulator Toolbar**. Refer to the pulse mode for additional information.

Train Mode
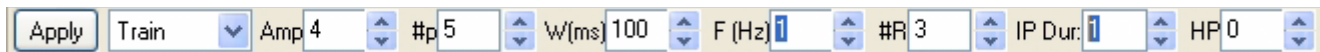
Figure 4-10: The Stimulator Pulse toolbar in the DataTrax2 Main window.

Amp: Amplitude of the stimulus pulse in Volts.

#p: Number of Pulses.

W(ms): Width of the pulse in milliseconds (ms).

F(Hz): Frequency of stimulation in Hertz (Hz).

#R: Number of Trains (or Repeat Count).

IP Dur: InterTrain(or inter protocol) duration.

HP: Holding Potential

Apply: Applies any changes to the stimulus protocol made through the **Stimulator Toolbar**. Clicking **Apply** while recording will send a stimulus pulse from the output. Refer to the **Pulse** mode for additional information.

Step Mode

Figure 4-11: The Stimulator Pulse toolbar in the DataTrax2 Main window.

A1: Starting Amplitude.

A2: Stopping Amplitude.

#steps: Number of Steps.

W(ms): Width of the step in milliseconds (ms).

F(Hz): Frequency of stimulation in Hertz (Hz).

#R: Repeat Count.

IP Dur: Inter protocol duration.

HP: Holding Potential.

Apply: Applies any changes to the stimulus protocol made through the **Stimulator Toolbar**. Clicking **Apply** while recording will send a stimulus pulse from the output. Refer to the **Pulse** mode for additional information.
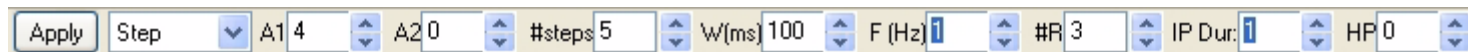
Dialog Windows

Preferences

There are six pages in the **Preferences** dialog box.
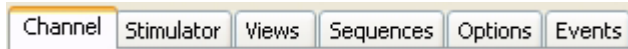
Figure 4-12: The pages of the Preferences dialog window.

Channels Page

The preferences that can be set for all channels include:
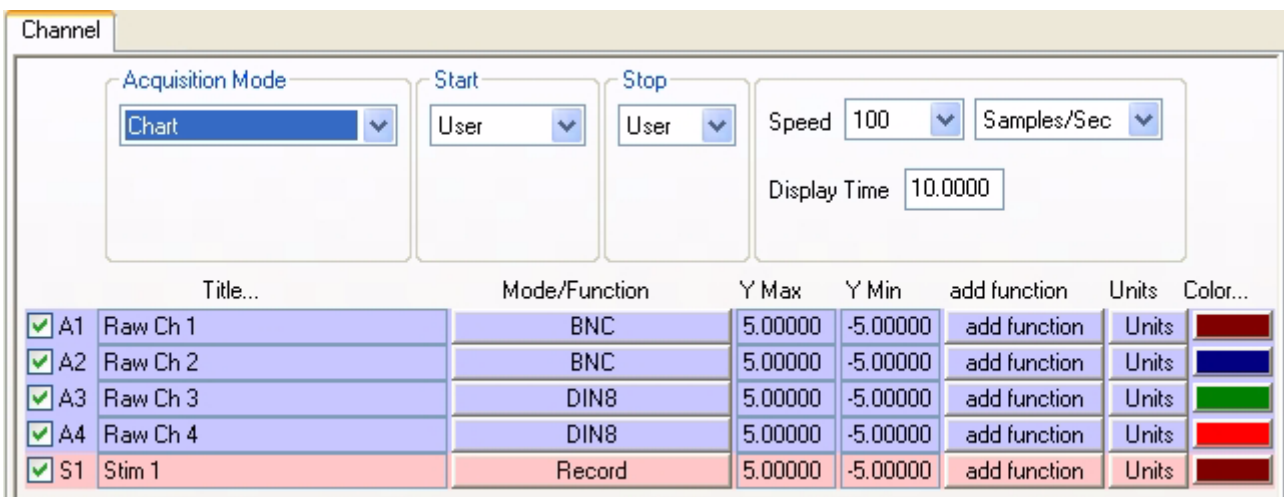
Figure 4-13: The Channels page of the Preferences dialog window.

Acquisition Mode: Sets the type to acquisition.

Start: Sets the parameters to start recording.

Stop: Sets the parameters to stop recording.

Speed: Sets the number of samples taken every second on each channel.

Display Time: Sets the time and the amount of data displayed on the screen

The preferences that can be set for each individual channel include:

Title: Edits the title of each channel to better identify the parameter being recorded.

Mode: Sets the record or the display mode of the channel. Channels 1 and 2 on the *WPI* 214 (Channel 1 only on the *WPI* 114) have record modes which allow either the bioamplifiers to be activated with a bandpass filter (0.03-150Hz, 0.3-35 Hz, or 3-10KHz) or the BNC input to be activated for recording signals from other devices with analog outputs.

Function: Selects either Raw Data or one of the many available calculated values to be displayed on the channel.

Ymax, Ymin: Sets the maximum and minimum Y-axis values.

add function: used to add a computed function to a channel.

Units: Used to convert two raw data values to two corresponding calculated values, using a linear scale. For more information refer to the **Units Conversion** section

Color: Sets the color of the channel.

Stimulator Page   Used to setup the **Stimulator(s)** on the *WPI* 214/118. There are five modes: **Off**, **Pulses**, **Trains**, **Constant** and **Step**. The available parameters in each of these modes can be preset and made part of a settings file.

Refer to the **Stimulator** section of the **Input/Output** chapter for information on the programming of the stimulator.

Views Page   Refer to the **Display** chapter for information on configuring this page.

Sequences Page   Refer to the **Digital Inputs** and **Digital Outputs** sections of the Input/Output chapter for information on configuring this page.

Options Page   These settings are at the application level and will apply across different settings files.
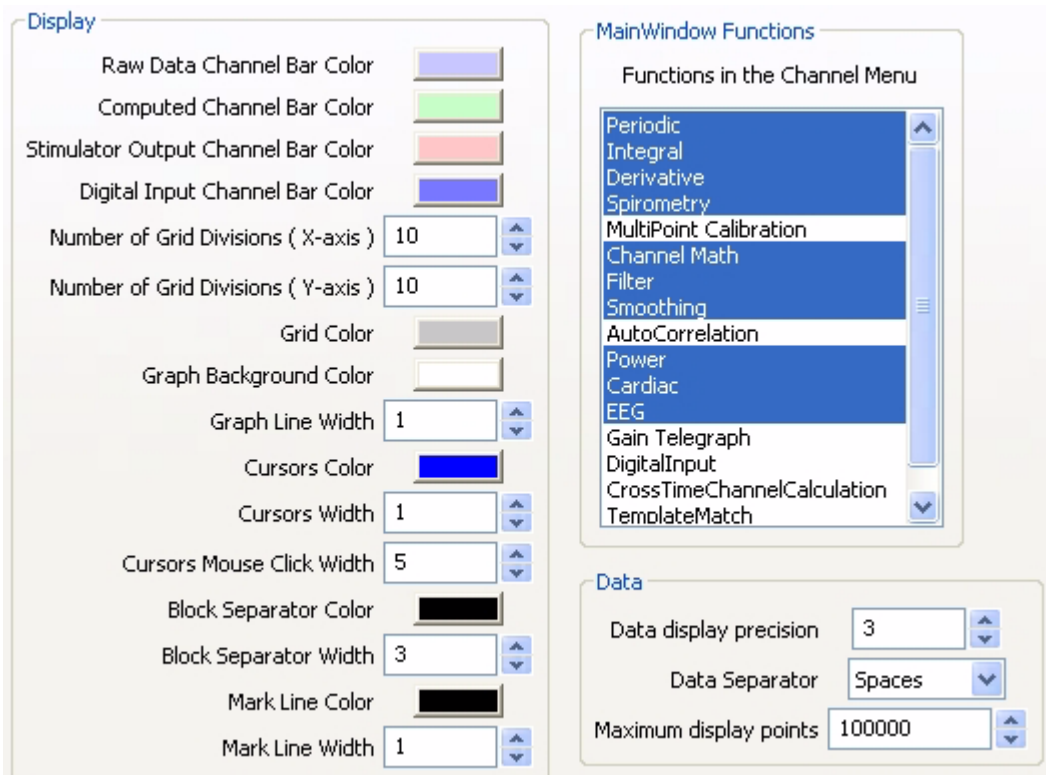
Figure 4-14: Options page of the preferences dialog

Display: The colors and line thickness, etc. can be changed here.

MainWindowFunctions: Allows the user to select the functions available from the **Main** window.

Data display precision: The number of digits of precision to be shown.

Data Separator: The data separator to be used for exporting data to the journal.

Maximum display points: The maximum number of data points that will be displayed.

Chapter 5: Analysis

Overview

Today's analog to digital converters make acquiring data very easy. The challenge to all users is to extract meaning from the mountains of recorded data in order to generate conclusions, reports, and Nobel Prizes. This process is called analysis.

Built into the *DataTrax2* data recording software is a powerful array of data analysis tools. The variety of tools available strike a balance between the straightforward, general operations that everyone uses and the vertical, complex routines that only YOU use. So, *DataTrax2* is a powerful analytical tool that can go to work on data right away, or be customized to execute very specific, complex analysis routines.

Analysis in *DataTrax2* is divided into five kinds. Each kind addresses specific analytical requirements. The following sections of this chapter explain how each kind is used.

Analyses of the First Kind: Wave-to-Wave Transforms

Analytical functions of this kind take ALL of the data in a channel and apply a transform, which converts the entire waveform described by the data points into a completely new wave that is displayed on a different channel. Examples of this kind of analysis would be the first derivative or the integral.

Currently there are more than 50 functions of the **First Kind**, or **Wave-to-Wave Transforms**, included in the *DataTrax2* program. These functions can only be accomplished on the **Main** window. They are called by using the **Add Function** menu and can be used **on-line** or **off-line**. When used **on-line**, the functions can operate at the top acquisition rate of the program, 100,000 samples per second.

In the **Add Function** menu, the transforms are organized into groups. The functions within a group have similar setup requirements and are usually located in a submenu. While the list of available functions is always expanding, *DataTrax2* V2.x includes the following groups: **Periodic**, **Integral**, **Derivatives**, **Spirometry**, **MultiPoint Calibration**, **Channel Math**, **Filter**, **Smoothing**, **Power**, **Cardiac**, **EEG**, **Gain Telegraph** and **Digital Input Functions**. Each group is discussed below.

Before beginning to explain the various **First Kind** functions, it is appropriate to ask: "Where does the raw data for a wave-to-wave analysis come from?" It is a question that is appropriate for all of the **First Kind** functions. To apply a function on a **Raw Data** or a **Computed Data** channel click the **Add Function** button on that channels **Channel** bar. Choose the desired function from the menu. If the function requires user specified parameters, a setup dialog will automatically appear. This setup dialog is also accessible after the **Computed Function** channel has been created, from that computed channel's Mode/Function label in the Channel Bar.

Whether calculated functions are performed **on-line** (in real time) or **off-line** (after recording has stopped), their setup is the same.

Note: Deleting the raw data channel will cause the function channel to have invalid data.

Periodic

Periodic functions operate on cyclic data to produce a graphical representation of how the **Rate**, **Frequency**, **Period**, **Cycle Maximum**, **Cycle Minimum**, **Cycle Mean**, or **RMS** (Root Mean Square) vary with time. The program calculates these parameters with each cycle of the signal and displays the calculation on the screen.

Threshold

To make these types of calculations on each cycle, the software must have a way of defining a cycle of data. As data is collected on-line or processed off-line, the *DataTrax2* program begins the calculation by using the **AutoScale** feature to determine the **Max** and **Min** values in a given screen of data. Then, the program uses a **Threshold** value to find points on the recording where the trace repeatedly crosses the threshold with a positive slope. The control for the **Threshold** level is set in the **Periodic** setup dialog, which appears when a periodic function is selected. The **Threshold** level can be set at **X%** of maximum value in any given screen of data or an absolute value. By default, **X%** is set to **60%**, which is adequate for almost all biological signals. The **Threshold** value can be changed by entering a different number in its box. Finally, the program defines the time between the points, where the recording repeatedly crosses the threshold line with a positive slope, as a **Period** (Figure 5-1 on page 45).
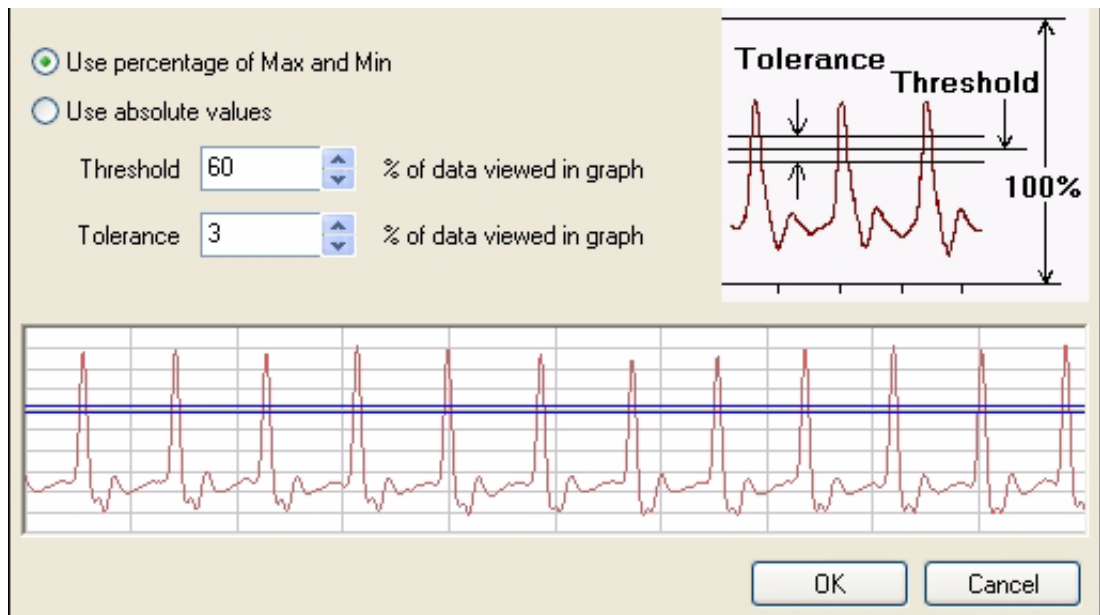
Figure 5-1: The Periodic setup dialog window.

Tolerance

A second **Periodic** control known as **Tolerance**, or hysteresis, is present in the **Periodic** setup dialog window. **Tolerance** is used to reduce false triggering due to noise in the signal. By default the tolerance is set at 3%, if the threshold is set at 60% then the signal has to cross from below 58.5% (60 - 3/2) to above 61.5% (60 + 3/2) for a threshold crossing to be detected. The maximum value between two threshold crossings is taken to be the peak, and adjacent peaks are used to calculate the periodic functions.

To see **Threshold** and **Tolerance** in use, record some pulse data using the **Tutorial** setting from one of the **Settings** groups. Apply the **Rate** function to the pulse channel.

The **Rate** function must be able to ignore the second small wave associated with each larger pulse wave; otherwise, it will report a rate that is twice what it should be. Using the **Rate** function on an ECG would mean that **Threshold** and **Tolerance** should be set so that only one wave of the three separate and distinct waves per beat can be counted.

Activating Periodic Transforms

Rate
Frequency
Period
CyclicMax
CyclicMin
CyclicMean
Max dV/dt
Min dV/dt
Mean dV/dt
RMS

All **Periodic** functions are selected in the same manner. To apply any of these functions to a channel, click on the **Add Function** button in the **Channel** bar. Select **Periodic** from the **Function** menu and a submenu appears. Select one of the functions in the submenu. Click **Start** in the **Main** window to begin recording. **AutoScale** the raw data, then **AutoScale** the calculated channel. No other setup is required.

The functions in the **Periodic** submenu are:

Rate: The program takes the **period** in seconds and divides this value into 60. The result is a **rate**, which is expressed in beats per minute

Frequency: The program takes the **period** in seconds and divides this value into 1. The result is a **frequency**, which is expressed in Hz or cycles per second.

Period: The program takes the **Period** for each cycle.

Cyclic Min: The program examines all of the points in the current period and finds the lowest value.

Cyclic Max: The program examines all of the points in the current period and finds the highest value.

Cyclic Mean: The program examines all of the points in the current period and finds the average of all values.

Max dV/dt: The program examines all of the derivatives in the current period and finds the highest value.

Mean dV/dt: The program examines all of the derivatives in the current period and finds the average of all values.

Min dV/dt: The program examines all of the derivatives in the current period and finds the lowest value.

RMS: The program examines all of the points in the current period and finds the Root Mean Square value for all points.

Integral

The integral is the area under the curve. The **Integral** function, as executed by the *DataTrax2* software, calculates a continuous sum of all the data points on a given channel that satisfy a certain citeria, and plots the running total. There are four types of Integrals:

Standard: Includes all data points in the calculation

Absolute: The Absolute Value of the Integral (Abs. Integral), as the name implies, makes all values of the integral positive and plots the running total. The **Abs. Integral** is used for analysis of cyclic data such as unit action potentials or EMG data.

Positive: Only the positive data points are included in the calculation.

Negative: Only the negative data points are included in the calculation.

Data points with values above zero make the **Integral** larger, those with values less than zero make the **Integral** smaller. To successfully complete the calculation of the **Integral**, the location of the zero-line needs to be known.
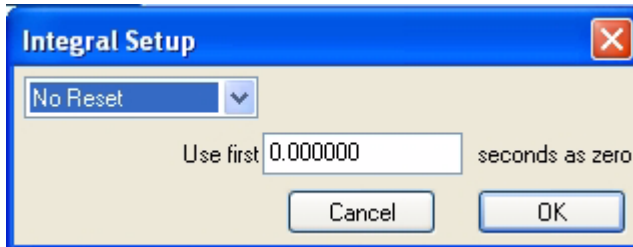
Setting the baseline of the raw data channel to zero is the more important of the two requirements since this function defines the difference between positive and negative areas of the recording. If the baseline of the raw data record is in the positive range of amplitudes, then the **Integral** will have a positive slope, even though no signal is present. Conversely, if the baseline of the raw data is in the negative range, the **Integra**l will have a negative slope.

Zero

As mentioned above, the **Integral** function adds the amplitudes of successive data points and plots the total. The problem is determining where the zero-line is located. *DataTrax2* and most other data programs take the value zero volts as the zero-line. If real units, such as grams or mmHg are used, the *DataTrax2* software will take the zero units value as zero.

DataTrax2 also has a **Use First "Number" seconds as zero** option. The data values for the first **"Number"** of seconds collected are averaged and used by the program as zero. This feature is particularly useful when trying to integrate signals that are difficult to zero manually.

An example of the application of this feature is illustrated in Figure 5-2 on page 47. The output of a respiratory flow sensor is the raw data displayed on the upper channel in each window; the lower channel in each window is the integral of the upper channel, or the volume flowing through the sensor.

In this example, the setting of zero is critical because any offset of the raw data from zero will be taken as a flow and, subsequently, be interpreted as a volume. Two examples of the respiratory integral are shown. In each case the flow sensor has a small, but stable offset. In the first example, the integral shows constant increase in volume, even as the flow is constant. In the second record, where the **Use First "Number" seconds as zero** option is used, the initial flow is constant and is set to zero, so the initial volume is zero. When the flow increases to a level above the "effective" zero-line, then the volume will increase on the integral channel.
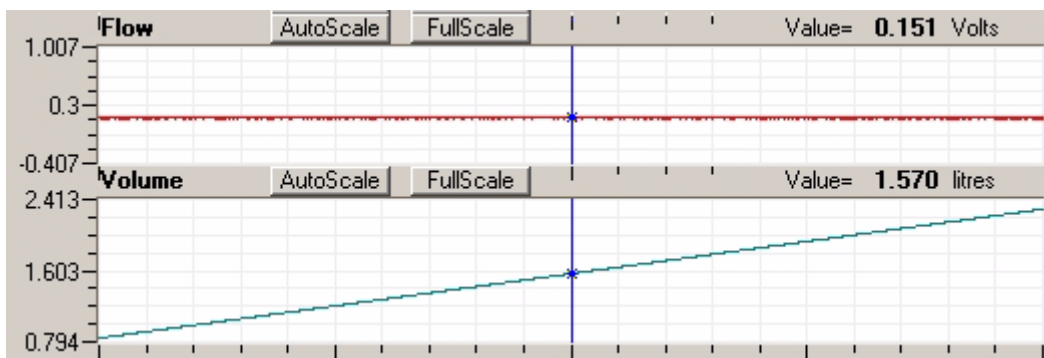

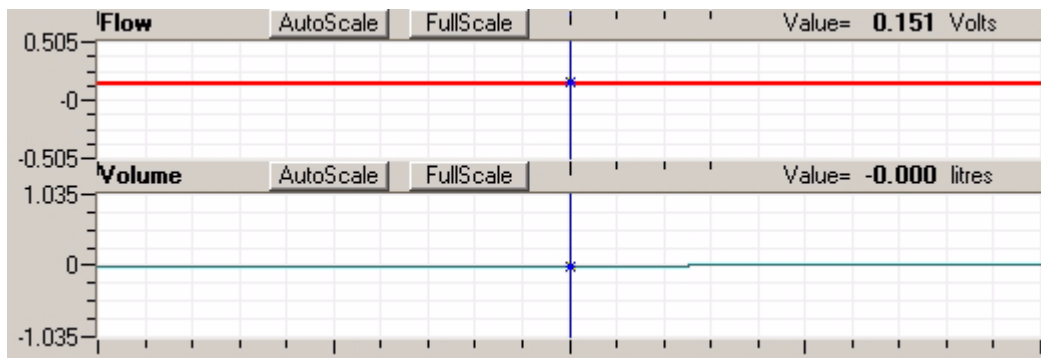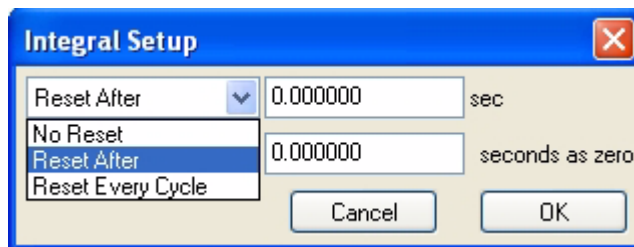
Figure 5-2: Integral without Zero.

Figure 5-3: Integral with Zero

The **Reset** control in the **Integral Setup** dialog window determines when the **Integral** or running count will reset itself to zero. This option allows the **Integra**l to reset itself after a preset time. This is a useful option if the data being integrated contains artifacts that move the **Integral** artificially up or down. The **Reset** function keeps the **Integral** trace in the field of view. In the respiration example, if inhaled air is at 20°C and exhaled air is closer to body temperature, then exhaled air has a larger volume. Since the subject exhales more than he or she inhales, the integral record will have a slow upward drift. Resetting the trace periodically will return it to zero.

Activating Integral Transforms

All **Integral** functions are selected in the same manner. To apply any of these functions to a channel, click on the **Add Function** button in the **Channel** bar. Select **Integral** from the **Function** menu and a submenu appears. Select one of the functions in the submenu. Click **Start** in the **Main** window to begin recording.

Derivative

The **Derivative** function calculates the derivative at each point in the raw data, and then displays it on the calculated channel. On the channel where the **Derivative** is displayed, the units are changed to the units of the raw data channel/second. Higher order derivatives can be calculated by applying a derivative function to a derivative channel.

Activating Derivative Function

To apply the derivative function to a channel, click on the **Add Function** button in the **Channel** bar. Select **Derivative** from the **Function** menu. Click **Start** in the **Main** window to begin recording.

Spirometry

The **Spirometry** function is a specialized version of the integral function for use with the *WPI* spirometer. This transducer, which measures respiratory volumes, is a sensitive air flow sensor. *DataTrax2* integrates the flow values measured by the sensor and displays volume. The settings for this integration are specifically designed for volume calculations.

Each *WPI* spirometer has a calibration value on it. This value can be entered in the **Spirometer Calibration** window. If the user has a calibration syringe and has recorded data where a known volume of air was pushed through the spirometer, this data will be displayed in the graph window. Place the first cursor at Zero Volume and the second cursor at X Volume.

Also enter this X value in the text box below the graph. Clicking the **Calibrate Difference between Cursors** button will calculate the required calibration value and place it in the **Spirometer Calibration** window.

To account for offset in the spirometer, choose a time at the beginning of the record when there is no flow through the spirometer and enter this value in the **Use First N Seconds as Zero** text box. *DataTrax2* will take the first N seconds, average them together and use the result to set the zero-line of the **Volume** channel. This is necessary because the output of the air flow sensor is always offset and any offset causes the **Volume** channel to display a volume change, even though no air is flowing through the sensor.

The **Volume** integral can also be set to reset after a certain time or with every cycle. During spirometry experiments, the volume trace drifts upward because the volume of exhaled air is larger than the volume of inhaled air. Cooler inhaled air at room temperature occupies less volume than warmer exhaled air. The **Volume** integral will drift up and out of view, as it reports correct values, unless a periodic reset is employed.

Activating Spirometry Function

To apply the Spirometry function to a channel, click on the **Add Function** button in the **Channel** bar. Select **Spirometry** from the **Function** menu.

Channel Math

The **Channel Math** function applies a user-defined function to points from up to four data channels and displays the output in a computed channel. There are four Variables **A**, **B**, **C** and **D**, each of which can be associated with a user selectable data channel. For example, if you wanted to divide Channel 1 by Channel 2. Click on the **Add Function** button on either the Channel 1 or Channel 2 channel bar. Select the **Channel Math** option. Select **A** to be Channel 1, and **B** to be Channel 2. Set the channel value to be **A/B**.
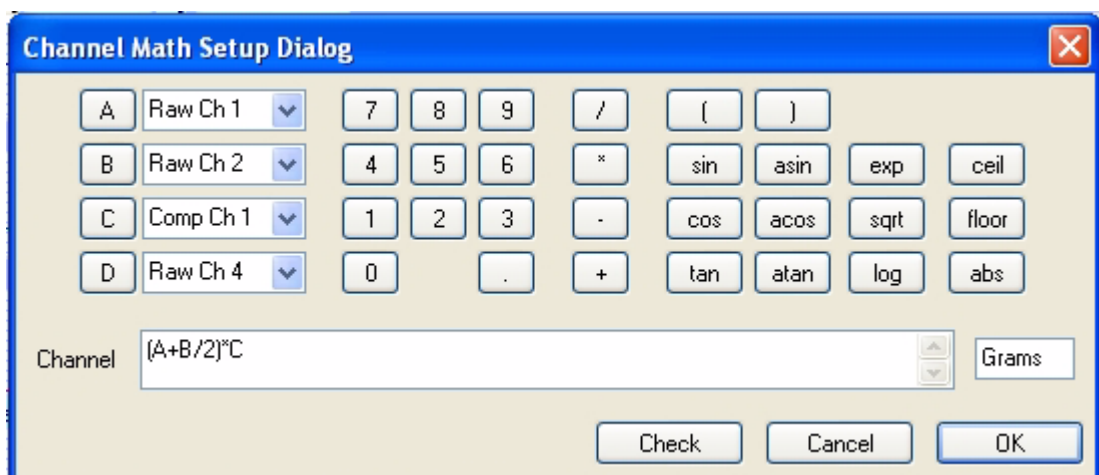
*Figure 5-4: The Channel Math dialog window. eg, output channel is equal to (RawCh 1 plus Raw Ch 2 / 2) * Comp Ch 1 in grams*

The result is every Channel 1 data point divided by the corresponding Channel 2 data point. A full range of **trig** and **log** functions, as well as the common mathematical operators, are available in the **Channel Math** dialog window. The **Unit Name** for the calculated value can be specified in a box. There is also a **Check Expression** feature, which will check for errors like

unclosed parentheses or division by zero. Since division by zero is a possible occurrence even in legitimate expressions, but it cannot be calculated by a computer, the program substitutes the last calculated value for the quotient if division by zero is attempted.

Activating Channel Math

To apply the **Channel Math** function to a channel, click on the **Add Function** button in the **Channel** bar. Select **Channel Math** from the **Function** menu. Select the channels corresponding to **A**, **B**, **C** and **D.** Program the expression desired. Set units if desired, and click OK.

Filter

Each channel has its own digital filter that can be used on-line or off-line. *DataTrax2* uses a **FIR (Finite Impulse Response)** filter. There are various windowing functions that can be used for setting up an FIR filter. The **Hamming** window (default) is appropriate for most applications. In addition to this windowing function *DataTrax2* also provides **Rectangular, Bartlett, Hanning, Blackman** and **Blackman-Harris** windowing functions. The filter order is the number of data points in the raw data required to calculate each point in the filtered data. The strength of the filter is determined by the filter order. The higher the order, the stronger the filter. Note that the higher the order the longer it takes to calculate, which can slow down the display. Also the first and last order/2 data points in the filtered data are invalid. For example, if the filter order is 51, then the first 25 and the last 25 data points in the filtered data are invalid.
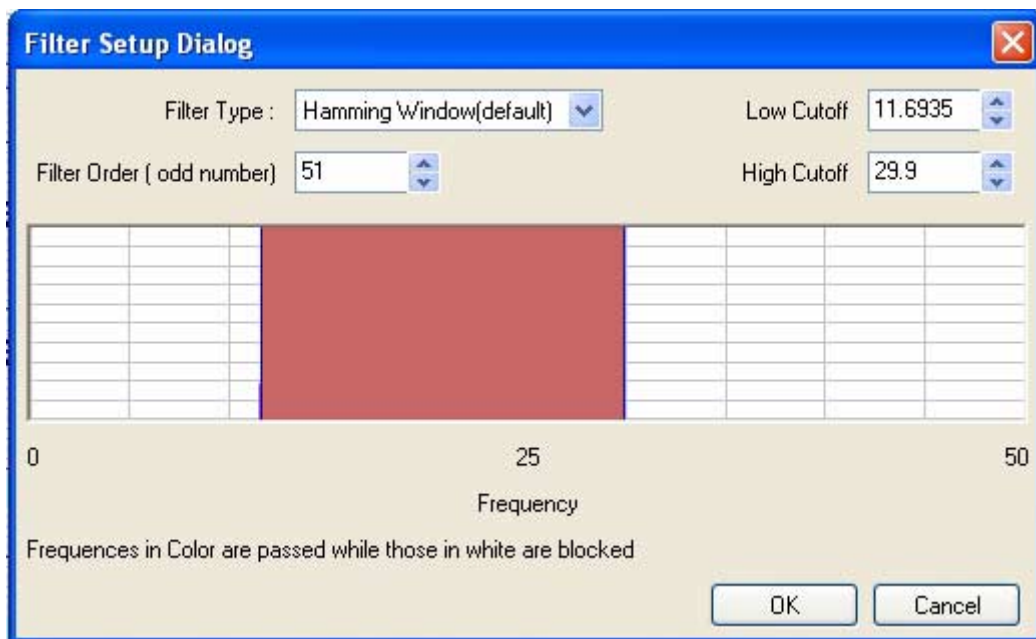

Figure 5-5: The digital (software) Filter dialog window.

In the **Filter Setup** dialog, drag the boundaries of the filter or enter the values in the Low Cutoff and the High Cutoff fields to create the desired filter. The graphic interface in the **Filter Setup** dialog is straight forward. The colored area corresponds to the frequencies that are passed, and the white area corresponds to the frequencies that are rejected. To remove high frequencies from the signal, click on the right boundary of the colored area and drag this boundary to the left. To remove low frequencies from the signal, click on the left boundary of the blue area and drag this boundary to the right. After clicking and dragging a boundary, it can be placed more accurately by entering the values in the Low Cutoff and High Cutoff fields. Boundaries can be placed in configurations that create **High Pass**, **Low Pass**, **Band Pass** (as shown in *Figure 5-5 on page 50*), or **Notch** filters.

Notice that the filter is subject to the Nyquist limitation of frequency. The maximum frequency in the **Filter Setup** dialog is exactly half of the sample frequency.

Smoothing   All experimental data includes noise to varying degrees. Noise can obscure important features like peaks, valleys, or peak widths. *DataTrax2* allows you to smooth noisy signals to reveal significant features of your data. In addition to filters, *DataTrax2* also offers the ability to smooth the data. Smoothing is used to remove noise that is uniform across all frequencies. Each method offers different performance to best show significant aspects of your results. Filtering allows you to eliminate noise above a specified frequency. *DataTrax2* provides two kinds of smoothing; **Moving Average** does wide smoothing, while **Savitzky-Golay** seeks to preserve shapes of peaks.

Moving Average: Applies a user specified number of points on either side of each data point to calculate the mean and replaces the datapoint with the calculated mean. For example, if you set the number of data points on either side to 1, then

new data (N) = {data(N-1) + data(N) + data(N+1)}/ 3

Savitsky-Golay: The Savitzky-Golay method essentially performs a local polynomial regression to determine the smoothed value for each data point. This method is superior to Moving Average because it tends to preserve features of the data such as peak height and width, which are usually 'washed out' by adjacent averaging. To use the Savitsky-Golay smoothing type, the order of the polynomial and the number of points on each side of the data need to be specified.

In addition to smoothing the data, *DataTrax2* also provides the option to calculate the smoothed 1st, 2nd or 3rd derivative of the data.

Power   The **Power** function performs a **Fast Fourier Transform (FFT)** on data in the selected channel and returns the average **Power** in the frequency band selected.
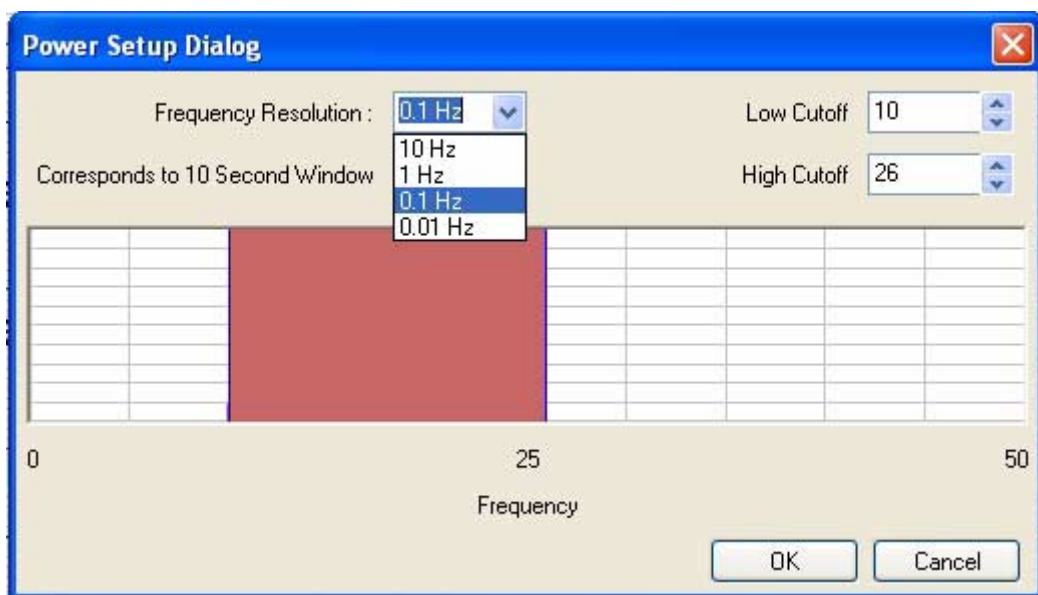

Figure 5-6: The Power dialog window.

In *DataTrax2*, this function works only off-line. The graphic interface in the **Power** dialog window is similar to the one in the **Filter Setup** dialog. The band of frequencies selected from the **Power** dialog window are not filtered, but they are subjected to an FFT whose output is

known as a **Power Number**. In addition to specifying the range of frequencies transformed, the frequency resolution can be selected. The higher the frequency resolution, the more data points required to compute the FFT. Therefore, at slow sampling rates, higher frequency resolutions may require the processing of more data points from longer recording periods.

Cardiac

The **Cardiac** functions are specifically used for the analysis of electrocardiograms (ECG). Four of the **Cardiac** functions calculate **Leads: III, aVL, aVF** and **aVR** from the recordings of **Lead I** and **Lead II**.
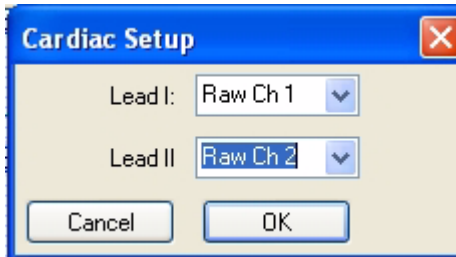
You can specify which channel corresponds to **Lead I** and **Lead II** in the **Cardiac Setup** dialog. *DataTrax2* can be programmed to do these calculations because all the points of view in a 6-lead ECG are in the same plane (frontal) of the body and each lead can be considered as a vector. So if any two of the limb leads are recorded, the other four leads can be calculated from them.
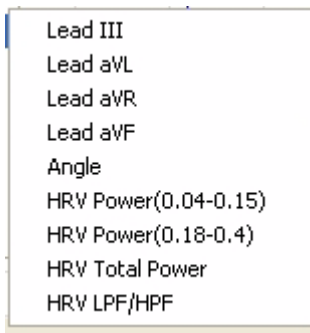
The **Cardiac** submenu also includes other functions. The **Cardiac Angle** function calculates the vector of the cardiac depolarization that passes through the interventricular septum. Three **Power** functions, which are special cases of the general power function described earlier, are also available. These three power functions are useful for heart rate variability (**HRV**) analysis. **Low Frequency** (**LF**) **Power** (0.04-0.15 Hz), **High Frequency** (**HF**) **Power** (0.18-0.4 Hz) and **Total Power** are each calculated from the transformation of a tachogram of an ECG on one of the raw data channels. The **HRV Ratio** which is (**Low Frequency** (**LF**) **Power)/High Frequency** (**HF**) **Power** is also available.

Activating Cardiac Functions   To apply the **Cardiac** function to a channel, click on the **Add Function** button in the **Channel** bar. Select **Cardiac** from the **Function** menu, then select the appropriate submenu. Select the channels corresponding to Lead I and Lead II and click OK
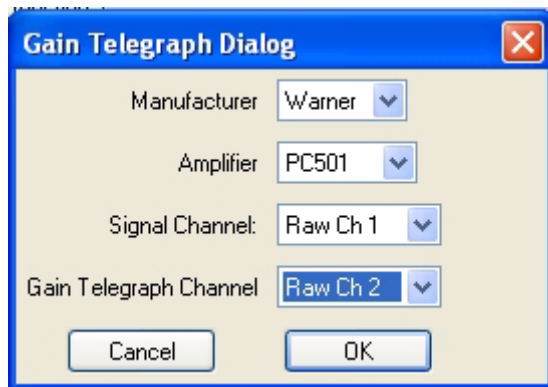
EEG

The **EEG** function is another specialized case.

Alpha
Beta
Theta
Delta
Beta Low
Beta Mid
Beta High
Alpha Power
Beta Power
Theta Power
Delta Power
Beta Low Power
Beta Mid Power
Beta High Power

In this function, a preset frequency band is available for each component of the EEG: **Alpha**, **Beta**, **Theta**, **Delta, Betalow, Beta Mid** and **Beta High**. For each selected band, *DataTrax2* can also calculate the average power in the band and display the value against time. These are beneficial functions to use when teaching students, because they can clearly see the effect that behavior has on components of the EEG.

Activating EEG Functions

To apply the **EEG** function to a channel, click on the **Add Function** button in the **Channel** bar. Select **EEG** from the **Function** menu, then select the appropriate submenu and click OK.

Gain Telegraph

Gain Telegraph Dialog

Manufacturer Warner

Amplifier PC501

Signal Channel: Raw Ch 1

Gain Telegraph Channel Raw Ch 2

Cancel   OK

The **Gain Telegraph** function is slightly different from the other functions. Some amplifiers have an additional output that sends a calibration signal to the data recording unit. This calibration signal relays information about the gain settings of the external amplifier. This information permits the recording program to re-calibrate the amplifier output in the correct units regardless of the gain set on the amplifier.

Using Gain Telegraph

Connect the analog output of the supported amplifier to any input channel. For this example, assume this is Channel 1. Next, connect the gain telegraph output of the amplifier to any input channel. For this example, assume this is Channel 2.

To apply the **Gain Telegraph** function to a channel, click on the **Add Function** button in the **Channel** bar. Select **Gain Telegraph** from the **Function** menu. Select the manufacturer of your amplifier, then select the amplifier you are using. Select the channel that the output from your amplifier is connected to (Channel 1 in this case), then select the **Gain Telegraph Channel** (Channel 2 in this case) and click OK.

Digital Input
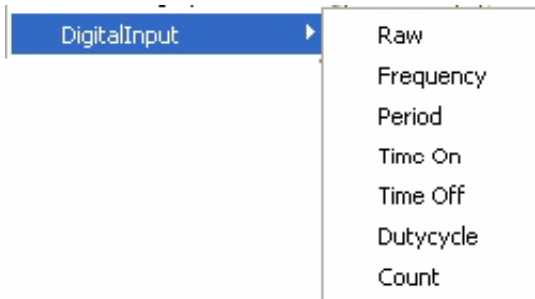
The **Digital Input** function treats the data on the selected channel as a binary number.

The functions in the **Digital Input** submenu are:

Raw: The digital value as a 32 bit word is displayed

Frequency: The program takes the **period** in seconds and divides this value into 1. The result is a **frequency**, which is expressed in Hz or cycles per second.
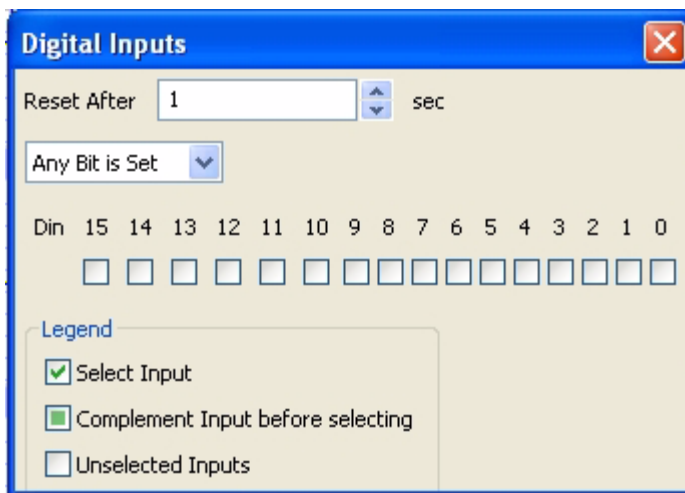
Period: The program takes the period for each cycle.

Time On: The time that the selected channels are high.

Time Off: The time that the selected channels are low.

Duty Cycle: 100* (Time On)/Period.

Count: Number of events.

The channels of interest can be selected using the checkboxes. If you desire to complement the input before it is used then click on the checkbox again, as shown in the legend.

Reset After n sec: This option is only available for the **Count** subfunction, which is used to reset the count after n seconds.

Any Bit is Set/ All Bits are set: Choose if you want an event to be detected if any channel is high or if you only want an event to be detected if all channels are high.

CrossTime Channel Calculation

Initial Delay: The amount of time from the start of recording until the function is initiated.

Selection Duration: Period of the protocol.

To setup **CrosstimeChannelCalculation**, you need to setup the **Initial Delay** before the function is initiated and set the **Selection Duration** to the period of the protocol. Selection can be set to start when the data on a selected channel goes above an operator selected value.

For each selection as defined above, variable **A** can be setup to equal the mean, maximum, or minimum of the data between the selected time range on a selected channel. Variables **B**,**C** and **D** can be setup in similar fashion.

The title and units name of the computed channel can be set. The variables **A,B,C,D** and the calculator can be used to create an equation for the computed channel.

MultiPoint Calibration

MultiPoint Calibration can be used to perform a linear and non-linear calibration of sensors, transducers, amplifiers and other equipment. It can also be used to check if an existing sensor is linear.

Multipoint Units Conversion Dialog

Convert | Mean ▾ | between cursors : [   ] to [   ] pA

	Current Value	Calibrated Valu
1	253.131836	5.000000
2	459.215210	10.000000
3	990.314392	20.000000
4	1961.177856	40.000000

Fit | y = m*x+ c ;eq. ▾ | to data

Equation: y = 0.020283x + 0.171167

R Value = 0.999703

Sum of Squared Errors = 0.427020

Export | Cancel | OK
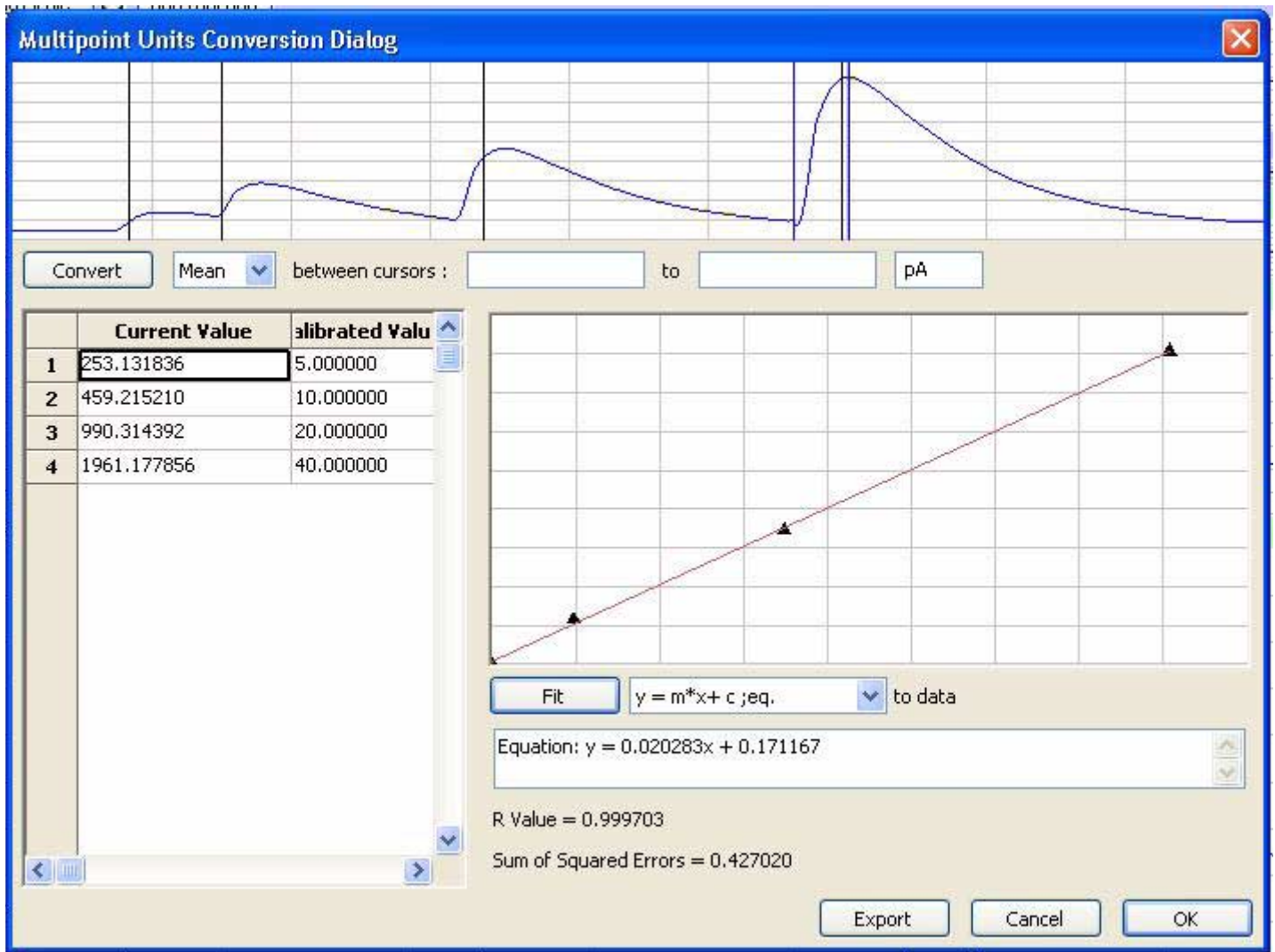
Template Match

Template Match Dialog

Choose a Region of the waveform to create a template.

Create Template from data between cursors

Tolerance   Variable

High Threshold Limits

lock curve   points

Low Threshold Limits

freehand Move using   3   points
lock curve
Block Move curve
freehand Move using
Reset curve

Warning!!! Sampling Speed has to be the same in all blocks for this function to work properly

OK   Cancel
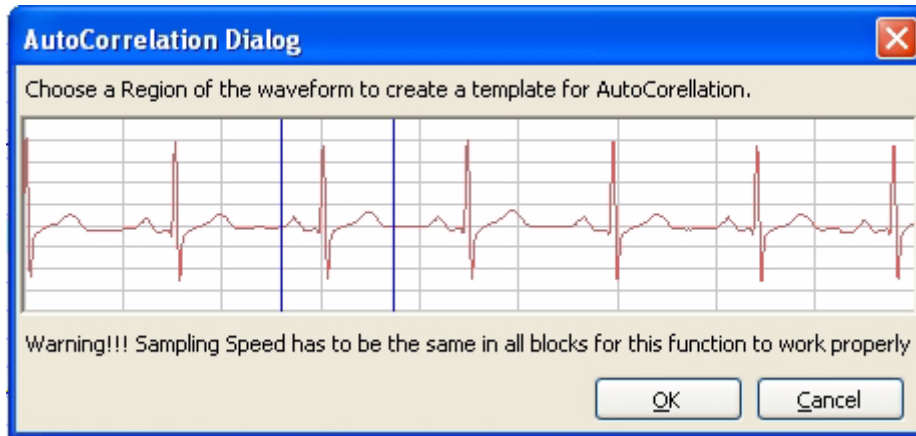
Template Matching is a technique in digital signal processing for finding small parts of a signal which match a template. The idea of template matching is to create a model of a signal of interest (the template, or kernel) and then to search over the recorded data for objects that match the template. To create a template, select a region of interest using the two cursors. Click **Create Template from data between cursors** to create a baseline for the template. For each point in the template, you can set the acceptable tolerance. The tolerance can be fixed for all points or variable. In case of variable tolerance, you set the high and the low limits of the tolerance separately. You can move the complete line up/down using **Block Move Cursor**, or you can move only certain segments of the curve using **freehand Move using**. When using the freehand move, you also have to specify the number of points each move will influence.

Once a template is created, the program will find segments of the data where each datapoint in the segment matches the template within the tolerance for each point.

Auto Correlation

AutoCorrelation Dialog

Choose a Region of the waveform to create a template for AutoCorellation.

Warning!!! Sampling Speed has to be the same in all blocks for this function to work properly

[OK]   [Cancel]

Correlation is the mutual relationship between two or more random variables (Ali). AutoCorrelation is the correlation of a signal with itself (Parr 1999). AutoCorrelation is a mathematical tool used frequently in signal processing for analyzing functions or series of values, such as time domain signals. Informally, it is a measure of how well a signal matches a time-shifted version of itself, as a function of the amount of time shift. More precisely, it is the cross-correlation of a signal with itself. AutoCorrelation is useful for finding repeating patterns in a signal, such as determining the presence of a periodic signal which has been buried under noise, or identifying the missing fundamental frequency in a signal implied by its harmonic frequencies.(Wikipedia 2007).

Analyses of the Second Kind: Digested Data

The functions in this kind of analysis take a group of data points selected by the user and "digests" them into a single value. These values can be saved to the **Journal** within *DataTrax2* or exported to other programs. This kind of analysis can proceed in the **Analysis** window. There are currently 24 functions available in the *DataTrax2* program to do this kind of analysis. Some of the functions are: **Mean**, **Max**, or **Max-Min**.

Analysis Window Setup

The **Analysis** window (see Figure 5-7 on page 60) is used to display and perform calculations on selected pieces of data or sweeps captured from the **Main** window. The data used in the **Analysis** window is defined by the data displayed in the **Main** window. The data is then captured by either clicking the **Analysis** icon on the *DataTrax2* toolbar or selecting **Analysis** from the **Windows** menu. Data from all channels within the selected area are displayed in the **Analysis** window.

Many of the same tools in the **Main** window are also available in the **Analysis** window. These include: **Display Time** controls; **Marks**, which are the same as the ones positioned in the **Main** window; and **2 Cursor** mode (**Single Cursor** mode is unavailable). Scroll bars can be used to fine tune the area of data upon which the selected functions will operate in **Chart** mode. In **Scope** mode the sweeps of interest can be selected Individually or as part of a group.

Data in the **Analysis** window can be operated upon by the functions selected with the **Add Function** button just above the left hand side of the first channel. Functions are organized in three groups: General, Derivative and Integral (Area). To add a function click on the **Add Function** button, choose the **Function Group** and then the desired function from the submenu. The selected function will now appear to the left of the **Add Function** button.

Again, the **2 Cursors** are positioned on the left and right edges of the data to be analyzed. *DataTrax2* will immediately calculate and display the values for the selected parameters in the **Channel Bar** of each channel. The precision of the calculations performed is adjustable using the **Data Display Precision** value in the **Options** page of the **Preferences** window.

There are 24 analytical functions available in the **Functions** list. These functions are described below. The values listed in the **Channel Bar** area can be sent to the **Journal** for formatting and inclusion in reports. To do this, click on the Channel menu (or right-click in the channel area) and select **Add Ch. Data to Journal**, **Add Title to Journal** or **Add All Data to Journal**.

Add Ch. Data to Journal: Adds data from the selected channel to the Journal.

Add All Data to Journal: Adds data from all channels to the Journal.

Add Title to Journal: Adds the function titles to the Journal.

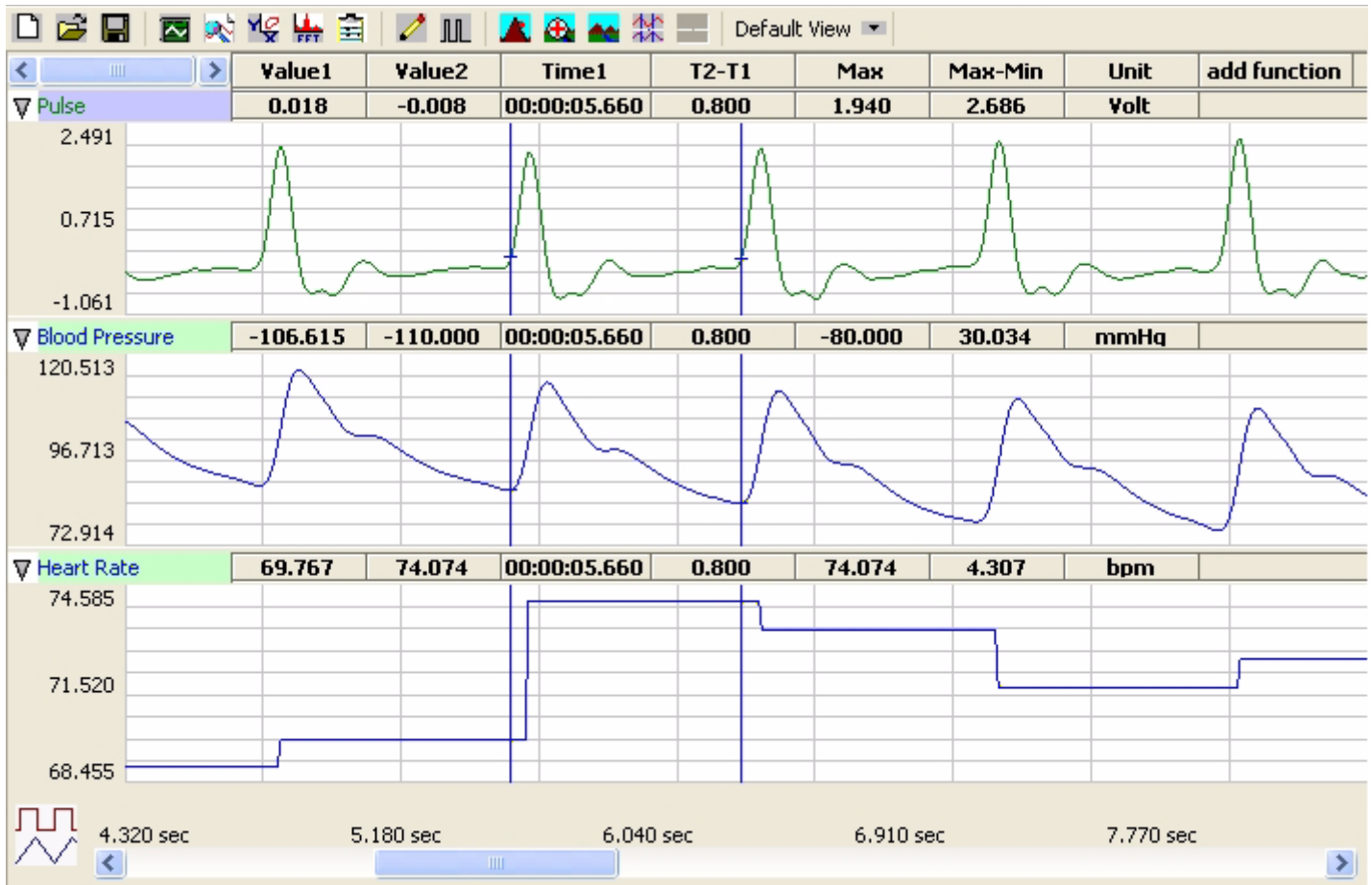The **Analysis** window can also be copied, printed, or exported to ASCII, .png, or .bmp format documents.

*Figure 5-7: The* DataTrax2 *Analysis window.*

Analysis Window Components

Functions List

The functions selected in the **Add Functions** list determine the calculations performed on the data points between the two cursors in the **Analysis** window. The results of the selected calculations are displayed in the **Channel bar** area at the top of the each Channel.
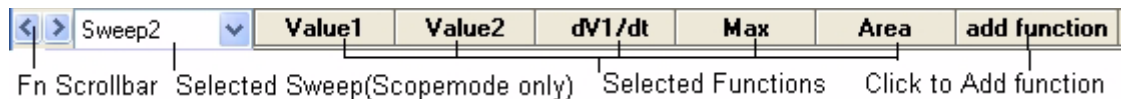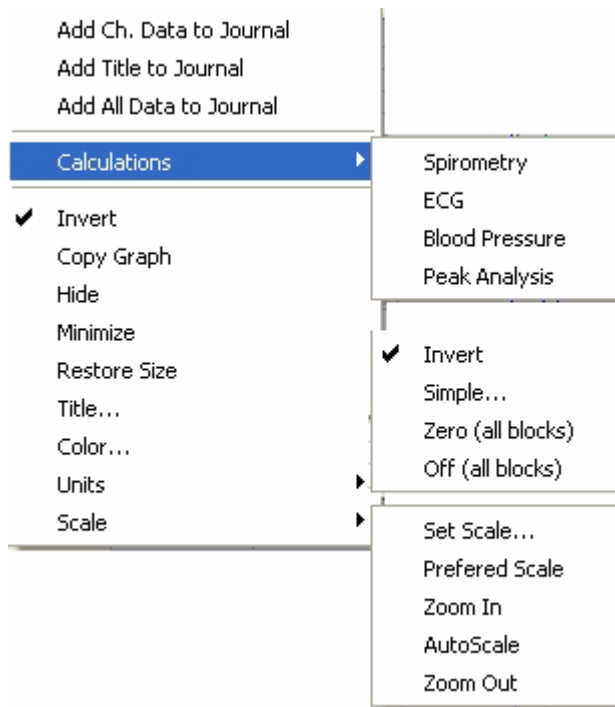

Tiled or Stacked Display
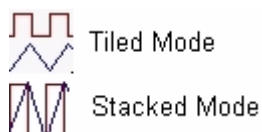
By default, the channels selected for display are presented in **Tiled** mode. Each channel is displayed in its own area, as they are shown in the **Main** window. By clicking the **Stacked/Tiled** icon in the lower left hand corner of the **Analysis** window, selected waveforms can be overlaid on the same set of axes.

Channel Menu in the Analysis Window

| Add Ch. Data to Journal |
| Add Title to Journal |
| Add All Data to Journal |

✔ Calculations ▶	Spirometry
	ECG
	Blood Pressure
	Peak Analysis

| ✔ Invert |
| Copy Graph |
| Hide |
| Minimize |
| Restore Size |
| Title... |
| Color... |
| Units ▶ |
| Scale ▶ |

| ✔ Invert |
| Simple... |
| Zero (all blocks) |
| Off (all blocks) |

| Set Scale... |
| Prefered Scale |
| Zoom In |
| AutoScale |
| Zoom Out |

Clicking on the **Channel Menu** button or right-clicking in the display area of the **Analysis** window brings up the **Channel** menu.

Add Ch. Data to Journal: Adds data from the selected channel to the Journal.

**Add All Data to Journal**: Adds data from all channels to the Journal.

Add Title to Journal: Adds the function titles to the Journal.

Calculations: Brings up the calculations submenu, for performing dedicated calculations in Spirometry, ECG, Blood Pressure and Peak Analysis. This is discussed later in this chapter

Copy Graph: Copies the channel graph to the clipboard, This can be then pasted into the Journal or any other program.

The remaining functions operate in the same manner as in the **Main** window

Scope Mode Display Options

Sweep Menu   Sweep Scrollbar   Available Sweeps
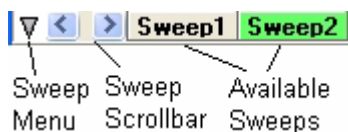
In **Scope** mode, you can view multiple sweeps at the same time. Sweeps can be selected and deselected from the **Sweep Selection** bar. Data can be calculated from only one sweep at a time. The primary sweep, on which the functions operate, is selected in the drop-down box to the left of the **Add Function** bar. The primary sweep cannot be deselected from the **Sweep Selection** bar.

To manage several sweeps, *DataTrax2* provides a **Sweeps Manager**. The **Sweeps Manager** is available from the **Sweep** menu button to the left of the **Sweep Selection** bar. In the **Sweep Manager,** one can organize the sweeps into groups. To create a group, click on the **Create New Group** button and the **Edit Group Name** dialog will appear. Enter a name for the new group and click OK. The group is now listed in the group tree. To add a sweep to a group, select a sweep from the available sweeps list and drag it to the group, or click the **Add** button. To remove a sweep from a group, select the sweep and click the **Remove** button.

To switch the display from **Sweep** mode to **Group** mode, click on the **Sweep/Group** mode button to the left of the scrollbar. This button changes between **Sweep** and **Group** mode.

Table Functions

These functions are available in the **Analysis** window.

General

Value1: Value at Cursor 1

Value2: Value at Cursor 2

Time1: Time at Cursor 1

Time2: Time at Cursor 2

V2-V1: Value2-Value1, i.e. the difference in value between the cursors

T2-T1: Time2-Time1, i.e. the difference in time between the cursors

Max: Maximum value between the cursors

Min: Minimum value between the cursors

Mean: Mean or Average value between the cursors.

Max-Min: Difference between the maximum and minimum values between the cursors.

Mark: Text of a mark between the cursors

Unit: The units of the channel

RMS: Root Mean Square of the values between the cursors

StdDev: Standard Deviation of the values between the cursors

Count of Events: Number of events between the cursors.

Derivative

dV1/dt:Derivative at Cursor 1

dV2/dt:Derivative at Cursor 2

Max dV/dt: Largest dv/dt values for all of the data points between the two cursors.

Min dV/dt: Smallest dv/dt values for all of the data points between the two cursors.

Mean dV/dt: Mean dv/dt for all of the data points between the two cursors. This is also the slope of the line of best fit for all the data points between the cursors.

Integral

All the integral and area functions are calculated as integrals.

Area: The Area function uses the line between **V1** and **V2** as the zero baseline, and then calculates the integral.The Area function gives more control over which segments of a waveform are included in the integral.

Abs. Area: The absolute value of each data point is used to calculate the area as described above. Areas that would have been subtracted in the Area function, are instead added in the calculation of the Absolute Area.

Int: For the Integral calculation, zero volts (or zero calculated Y-value) is used as the zero reference for the integral. Values above zero add to the Integral and values below the zero-line subtract from the Integral.

Abs Int: The Absolute Integral is very much like the Integral, except that the program takes the absolute value of the raw data before performing the integral operation.

Calculations

All calculations have a similar dialog, there is a picture at the top, which shows some sample data and where to place the cursors. You can choose the parameters to be exported to the Journal in the **Parameters to Export** list box. The data table shows the calculations for the current position of the cursors. The values in the data table are exported to the journal on clicking the **Add to Journal** button.
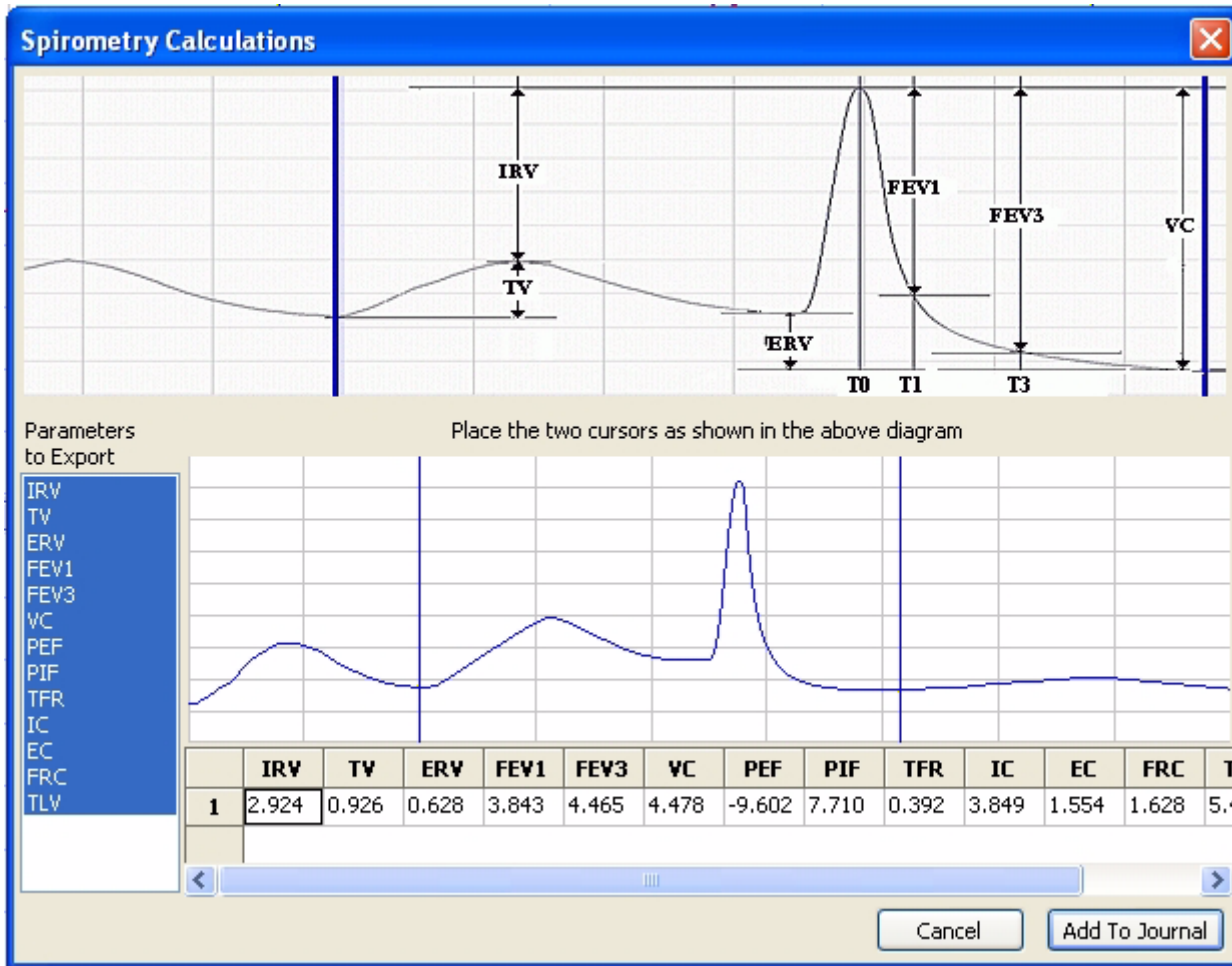
TV: Tidal Volume

IRV: Inspiratory Reserve Volume

ERV: Expiratory Reserve Volume

FEV1: Forced Expiratory Volume in one second. How much air can be exhaled in the first second of expiration

FEV3: Forced Expiratory Volume in three seconds. How much air can be exhaled in the first three seconds of expiration

VC: Vital Capacity

PEF: Peak Expiratory Flow
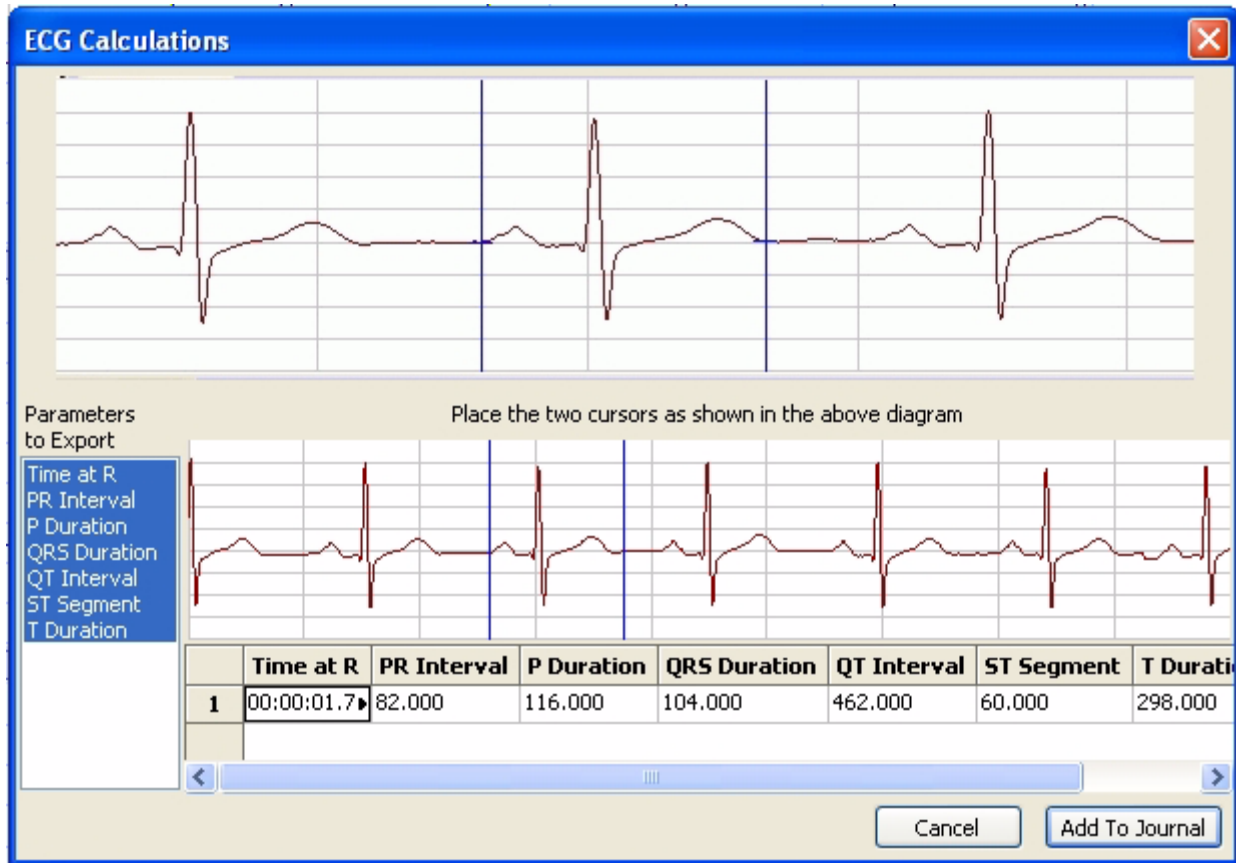
PIF: Peak Inspiratory Flow

TFR:Tidal Flow Rate.

IC: Inspiratory Capacity (IC) = TV + IRV

EC: Expiratory Capacity (EC) = TV + ERV

FRC: Functional Residual Capacity (FRC) = ERV + RV

TLV: Total Lung Capacity (TLC) = TV + RV + IRV + ERV

Time at R: Time at the peak of the R-wave.

PR Interval: The time from the beginning of the P wave to the peak of the R wave.

**P Duration**: Time from the beginning to the end of the P-wave.

QRS Duration: Time from the beginning of the Q wave to the end of the S wave.

QT Interval: Time from the beginning of the Q wave to the end of the T wave.

ST Segment: Time from the end of the S wave to the beginning of the T wave.

T Duration: Time from the beginning to the end of the T wave.

TP Duration: Time from the beginning of the P wave to the end of the T wave.

Blood Pressure
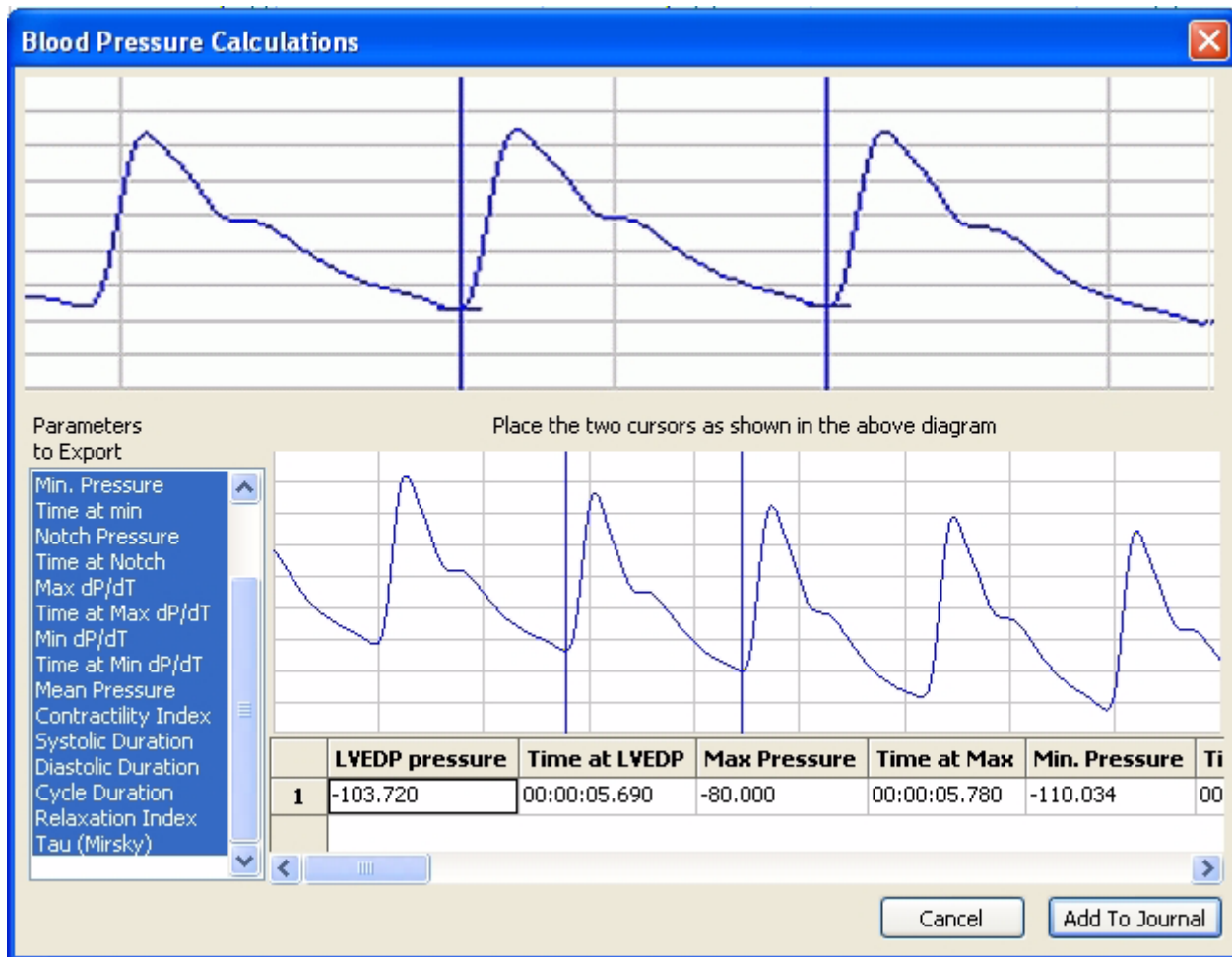
LVEDP pressure: Left Ventricular End Diastolic Pressure

Time at LVEDP: Time at LVEDP

Max Pressure: Maximum of the pressure, also the systolic value

Time at Max: Time at maximum pressure

Min. Pressure: Minimum pressure, also the diastolic value

Time at min: Time at minimum pressure

Notch Pressure: Pressure at the dichrotic notch

Time at Notch: Time at the dichrotic notch

Max dP/dT: Max slope

Time at Max dP/dT: Time at max slope

Min dP/dT: Minimum slope

Time at Min dP/dT: Time at minimum slope

Mean Pressure: Mean pressure

Contractility Index: (Maximum Slope)/(Pressure at Max Slope)

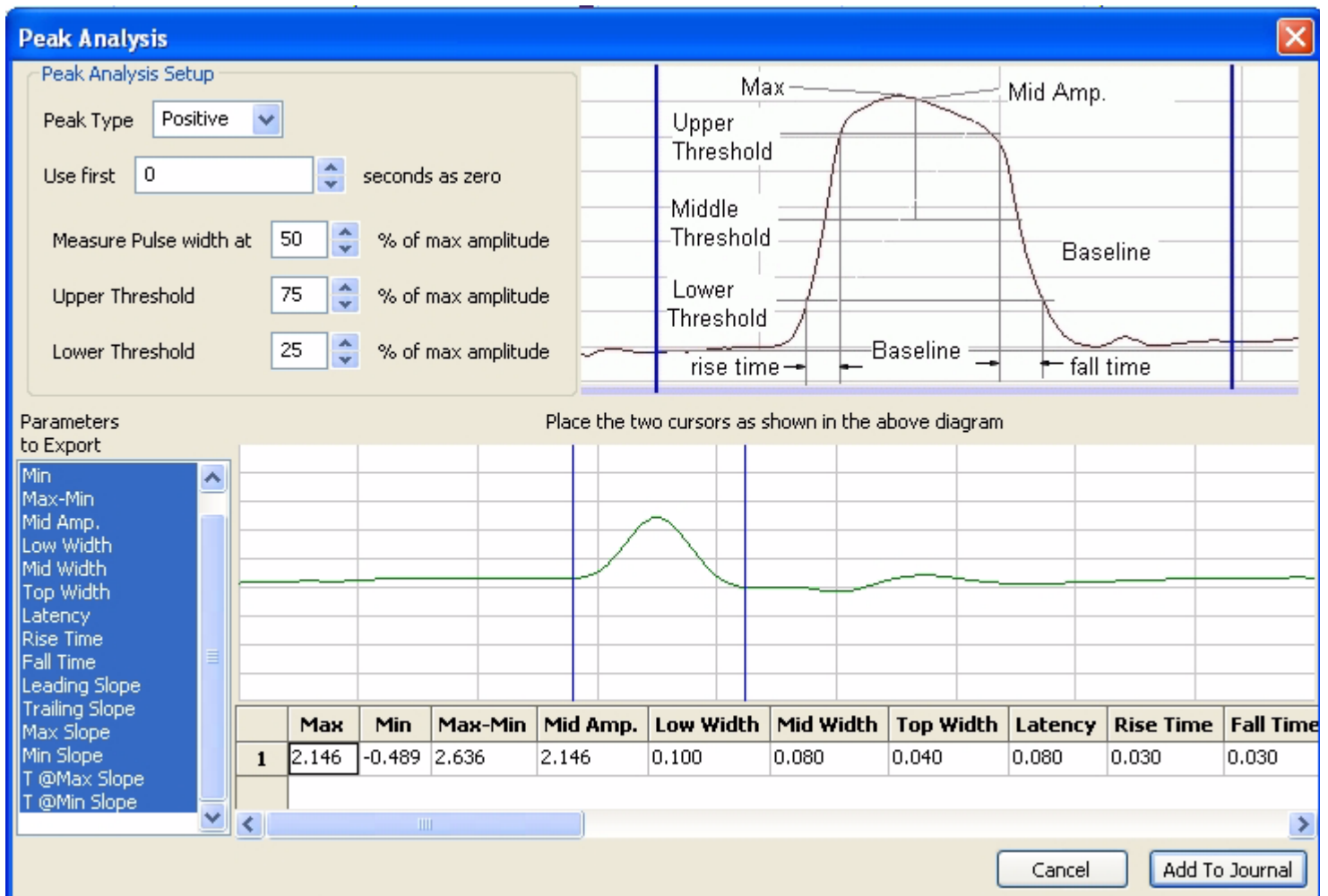**Systolic Duration**: Time from LVEDP to minimum slope

Diastolic Duration: Time from the minimum slope to the next LVEDP

Cycle Duration: Time between LVEDPs

Relaxation Index: (Minimum Slope) / (Pressure at Minimum Slope)

Tau (Mirsky): Time for pressure to drop to half its value from the value at minimum slope

Peak Analysis

Max: Maximum amplitude

Min: Minimum amplitude

Max-Min: Maximum amplitude - Minimum amplitude

Mid Amp.: Amplitude at the midpoint of the middle threshold

Low Width: Width of the peak as measures at the lower threshold

Mid Width: Width of the peak as measures at the middle threshold

Top Width: Width of the peak as measures at the upper threshold

Latency: Time from when the peak leaves the baseline to the peak

Rise Time: Time for the signal to travel from the lower threshold to the upper threshold

Fall Time: Time for the signal to travel from the upper threshold to the lower threshold

Leading Slope: Average slope of the signal as it travels from the lower threshold to the upper threshold

Trailing Slope: Average slope of the signal as it travels from the upper threshold to the lower threshold

Max Slope: Maximum slope

Min Slope: Minimum slope

T @Max Slope: Time at maximum slope

T @Min Slope: Time at minimum slope

Copy, Export, and Print Analysis Window

To copy the view of data displayed in the **Analysis** window, use the **Copy** command in the **Edit** menu. The image can be pasted into any program (including the **Journal**) that supports the clipboard. To copy a Channel, click the **Channel** menu and choose the **Copy Graph** menu.

To export the data viewed, use the **Export** command in the **File** menu. Select the format of the file from the list at the bottom of the **Export File** window.

To print the data viewed, use the **Print** command in the **File** menu.

# Analyses of the Third Kind: Redisplayed Data

As covered in the display section of this manual, data is recorded only into the **Main** window. **Main** window displays are *linear* or *in series*, meaning that Y-value parameters are recorded with respect to time.

Data recorded in a linear manner can be redisplayed in a format that is different than the standard Y-T plot. Currently, *DataTrax2* supports **XY plots** and **FFT**. A host of measurements, like the ones available in the **Second Kind Analyses**, can be made from each type of redisplayed data window.

XY Plot

In an **XY plot**, the Y-values from one channel in the **Main** window are plotted against the Y-values from another **Main** window channel. The resulting **XY plot** is dramatically different from a linear plot of data against time.

XY View Window Components
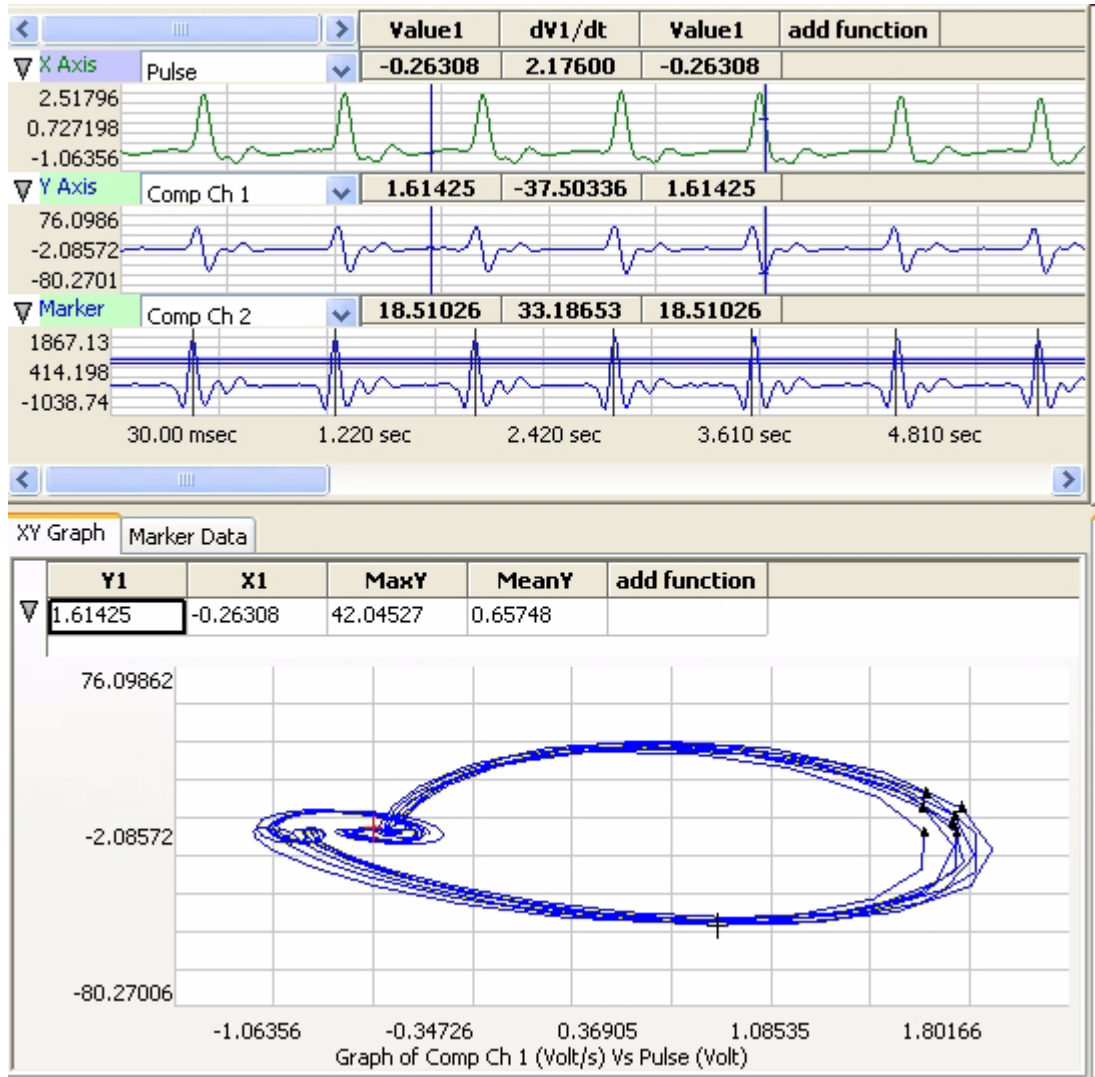
Selecting Channels   To plot an XY graph, select the X-axis channel from the drop-down box in the top most plot. Then select the Y-axis channel from the drop-down box in the second plot. If required you can also select a marker channel, to highlight certain points of interest.

Once the channels are selected, the XY plot will be displayed. All the data visible in the X and Y axis plots will be included in the XY plot. To select the region of data to be included in the XY-Plot, select **2 Cursors** from the toolbar and bracket the region of interest between the two cursors in the X-axis or the Y-axis channel. Next, select **Zoom Between Cursors** from the toolbar.

Functions List   The functions selected in the **Add Function** list determine the calculations performed on the data points between the two cursors in the **XY-PLot**. The results of the selected calculations are displayed above the XY PLot.
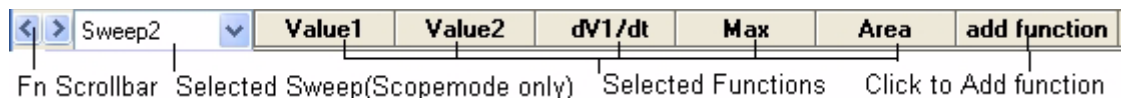
| **Table Functions** | These functions are available in the **XY** window. |

General

Y1: Value of Y-Axis Channel at Cursor 1.

Y2: Value of Y-Axis Channel at Cursor 2.

X1: Value of X-Axis Channel at Cursor 1.

X2: Value of X-Axis Channel at Cursor 2.

Y2-Y1: The difference in value of Y-Axis channel between the cursors.

X2-X1: The difference in value of X-Axis channel between the cursors.

MaxY: Maximum value of Y-Axis channel between the cursors.

MinY: Minimum value of Y-Axis channel between the cursors.

MeanY: Mean or Average value of Y-Axis channel between the cursors.

MaxY-MinY: Difference between the maximum and minimum values of Y-Axis channel between the cursors

MaxX: Maximum value of X-Axis channel between the cursors.

MinX: Minimum value of X-Axis channel between the cursors.

MeanX: Mean or Average value of X-Axis channel between the cursors.

MaxX-MinX: Difference between the maximum and minimum values of X-Axis channel between the cursors.

Length: Length of the line segment in the XY-plot between the two cursors.

Derivative

dY1/dx: Slope of the XY-Plot at Cursor 1.

dY2/dx: Slope of the XY-Plot Cursor2.

Max dV/dt: Maximum slope of the XY-Plot between the two cursors.

Min dV/dt: Minimum slope of the XY-Plot between the two cursors.

Mean dV/dt: Mean slope of the XY-Plot between the two cursors. This is also the slope of the line of best fit for all the data points between the cursors.

Integral (Area)

All the integral and area functions are calculated as integrals.

Area wrt Ymin: The minimum value of the Y channel is taken as baseline and the area of all points between the two cursors is calculated with respect to this baseline.

Area wrt Zero: Zero is taken as baseline and the area of all points between the two cursors is calculated with respect to this baseline.

Area wrt Y1: The value of the Y channel at cursor1 is taken as baseline and the area of all points between the two cursors is calculated with respect to this baseline.

Area Loop: The area of the loop as defined by the position of two cursors is calculated.

Channel Menu in the XY Window

Add Ch. Data to Journal
Add Title to Journal
Add All Data to Journal

Calculations

✔ Invert
Copy Graph
Hide
Minimize

Clicking on the **Channel Menu** button or right-clicking in the display area brings up the **Channel** menu.

Add Ch. Data to Journal: Adds data from the selected channel to the Journal.

Add Title to Journal: Adds the function titles to the Journal.

Invert: Inverts the channel data.

Copy Graph: Copies the channel graph to the clipboard. This can be then pasted into the Journal or any other program.

Marker Channel   The marker channel is used to mark specific points of interest in the XY-Plot. The marker channel is independent of the X and the Y channels, and should be setup in the **Main** window such that the peaks in the marker channel correspond to the points of interest.

When a marker channel is used, data from the XY plot is extracted at the specified points of interest and displayed in the **Marker Data** window. This data can be copied or exported by clicking on **Options**.

Copy, Export, and Print XY View Window   To copy the view of the data displayed in the **XY** window, use the **Copy** command in the **Edit** menu. The image can be pasted into any program (including the **Journal**) that supports the clipboard.

To export the data viewed, use the **Export** command in the **File** menu. Select the format of the file from the list at the bottom of the **Export File** window.

To print the data viewed, use the **Print** command in the **File** menu. The print range, which is either the selection in the **XY** window or all the data, can be selected in the **Print** window

FFT   **FFT** is short for *Fast Fourier Transform*, a mathematical operation that displays recorded data as the relative amplitudes of the frequency components that make up the recorded signal. FFT plots have **Frequency** on the X-axis and the **Power** (amplitude) contained in each frequency on the Y-axis. FFT is used to determine the relative strengths of frequency components in a raw signal. For example, in the EEG studies, the *Alpha* band is frequently mentioned. *Alpha* waves are EEG signals with frequencies between 8 and 13 Hz. If a FFT is performed on an EEG recording that has a high number of *Alpha* waves, the FFT will show a spike or a higher amplitude (**Power**) at the frequencies in the *Alpha* band.

The *DataTrax2* software can perform *Fast Fourier Transforms* on selected pieces of recorded data in the **FFT** window. The **FFT** window is called either from the **Windows** menu or by clicking the **FFT** icon on the toolbar at the top of the **Main** window. The selected data is moved to the **Linear Display** area in the **FFT** window where adjustments can be made which optimize the size of the **FFT plot**.

To ensure problem-free operation of the **FFT function**, there are a couple of rules to keep in mind:

1 1. The mathematical underpinnings of digital sampling, which makes the *DataTrax2* software work, begin with a criteria known as the Nyquist Sampling Theorem. Harry Nyquist showed that the *sampling rate must be at least twice the highest frequency in the sample* to reconstruct the original signal and capture its fundamental frequency. The converse of this rule is that the *fastest frequency that can be reliably recorded is a half of the sampling rate*. If a recording was made at 1000 samples per second,

the maximum frequency that could be recorded reliably would be 500Hz. If a recording was made at 100 samples per second, the maximum frequency that could be recorded reliably would be 50Hz. When a FFT is performed on data recorded with *DataTrax2*, the program sets the X-axis to a scale from 0Hz (DC) to a frequency that is a half of the sampling rate. A region of the scale can be expanded by using the cursors and the functions of the **Channel** menu, but frequencies greater than half the sampling rate cannot be viewed.

2   To make an FFT work, the transform must operate on a *specific number of data points*. The number of data points used in the FFT is algorithm dependent. By default **The Snap to Grid** option is selected, which sets the second cursor to the location of the best selection of FFT data on the linear display section of the **FFT** window. If you have to select a different selection, *DataTrax2* will fill in the remaining data points with zeros so the frequency content of the data that you did select will not be affected.

As stated in the first rule that governs FFT functions, *DataTrax2* sets the X-axis limits in the **FFT** window according to sample rate. Likewise, the number of data points used in the FFT calculation sets the resolution of the X-axis range. If more data points are used, the available **Frequency Resolution** will be greater. **Frequency Resolution** is set from the drop-down menu in the left hand margin of the **FFT** window. Resolution choices are limited to 100 Hz, 10 Hz, 1 Hz and 0.1 Hz. If the FFT cannot be displayed with the required frequency resolution, more data points need to be used to make the calculation.

Moving and Manipulating Data in the FFT Window

As with other analysis functions handled by the *DataTrax2* program, it is necessary to select the data to be operated upon by the program. The **Display Time** controls are used to get the data of interest onto a single screen in the **Main** window. The selected data is moved to the **FFT** window by clicking the **FFT** icon on the *DataTrax2* toolbar or by selecting **FFT** from the **Windows** menu.

Once the primary data set is moved to the **FFT** window, the **Dual Cursors** on the linear graph of the raw data (above the actual FFT display area) can be used to fine-tune the data selection. Only the data between the cursors in the linear graph is actually used in the calculation. The data displayed on the linear graph and available for the transform can be changed using the **Display Time** icons on the **FFT** window or the scroll bar under the linear graph. As mentioned, if **Snap To Grid** is selected, the *DataTrax2* program selects the end point for the data selection.

The **FFT plot** of the selected data is displayed in the area below the linear graph of the raw data. The FFT display area has its own **Display** controls with the **Zoom In, AutoScale** and **Zoom Out** functions and set of **2 Cursors**. These features can be used to expand or contract the area of the FFT graph between the cursors. This set of **2 Cursors** also reports the values for the selected **Table Functions** listed above the **FFT** window. The values and the titles, which are listed in the **Calculated Value Display** area at the top of the window, can be copied to the **Journal** by using the functions in the channel menu for the FFT display area.
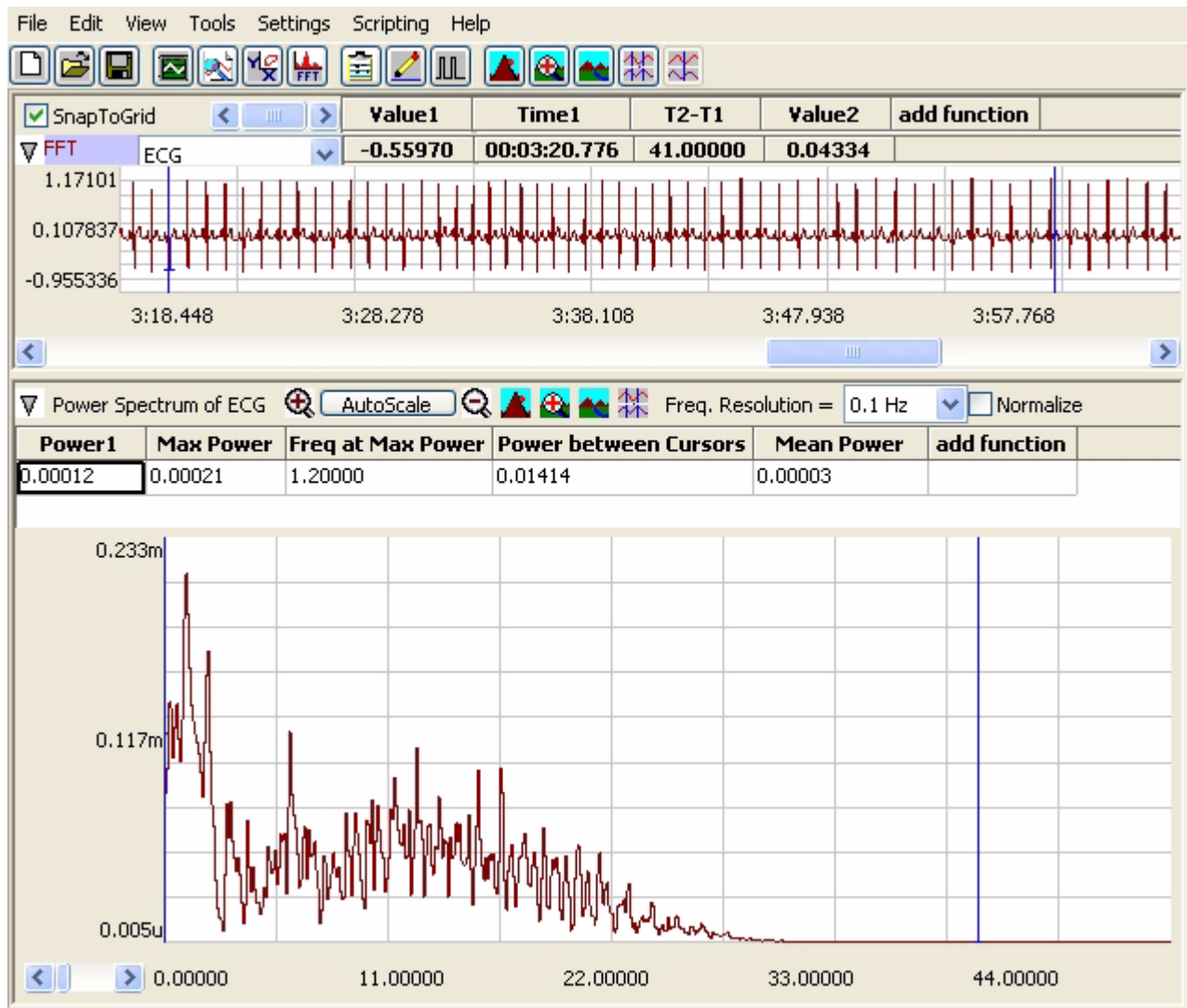
FFT Window Components

Figure 5-8:FFT Window

Display Channel   Located in the upper left of the channel display area, this drop-down box allows the user to select the channel to be used in the FFT plot.

FFT Functions
List
The calculations to be performed on the region of the FFT plot between the **2 Cursors** are selected from the **Add Functions** button on the top left of the **FFT** display area. The results of the selected calculations are displayed in the **Calculated Value Display** area at the top of the **FFT** window.

Display Controls   The FFT plot has display controls similar to the **Main** window such as **Half Display Time, Zoom between Cursors** and **Double Display Time**. The Y-axis can be scaled using the **ZoomIn, AutoScale** and **ZoomOut** buttons in the FFT area toolbar. The frequency resolution drop-down box allows setting the frequency resolution of the FFT from 10Hz to 0.01Hz.

Table Functions

These functions are available in the **FFT** window.

General

Power1: Power at Cursor 1

Power2: Power at Cursor 2

Freq1: Frequency at Cursor 1

Freq2: Frequency at Cursor 2

P2-P1: Difference in power between the cursors

F2-F1: Difference in frequency between the cursors

Max power: Maximum power between the cursors

Min power: Minimum power between the cursors

Mean power: Mean power between the cursors

MaxP-MinP: (Maximum power - Minimum Power) between the cursors

Freq at Max power: Frequency at maximum power between the cursors

Freq at Min power: Frequency at minimum power between the cursors

Integral

Power between Cursors: Total power between the cursors

Channel Menu in the FFT Plot

Click the **Channel** menu to bring up a menu like the one in Figure 5-9 on page 73.
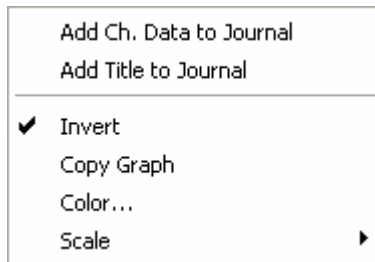
Selecting **Add Title to Journal** sends the names of the analyses (calculations) being performed and displayed in the **Calculated Value Display** area at the top of the **FFT** window, to the **Journal**. Selecting **Add Data to Journal** sends the values for these calculations to the **Journal**. Selecting **Copy Graph** sends an image of the FFT window to the clipboard.

Figure 5-9: FFT channel menu

Copy, Export, and Print FFT Windows

Use the **Copy** command in the **Edit** menu to copy the view of data displayed in the **FFT** window. This image can be pasted into any program, (including the **Journal**) that supports the clipboard.

To export the data viewed, use the **Export** command in the **File** menu. Select the format of the exported file from the list at the bottom of the **Export File** window.

To print the data viewed, use the **Print** command in the **File** menu. The print range, which is either the selection in the **FFT** window or all the data, can be selected in the **Print** window.

Analyses of the Fourth Kind: Detected Data Conditions

DataTrax2 software can be used to identify **Regions Of Interest** (**ROI**) in recorded data. The identified data points can be sent to either the **Journal** or the **Marks** window, where they can be used to either build a report or be exported. While identifying data does not constitute an analysis *per se*, the detection of specific data *always* precedes analysis. For example, to measure changes in **Left Ventricular End Diastolic Pressure (LVEDP)** over time, the LVEDP points need to be located in the field of blood pressure data and the corresponding values need to be recorded. The statistical manipulation of the recorded LVEDP values is classified as analysis; but, the first critical step in the "analysis" was the extraction of the relevant data points or regions of interest from the complete data. The events (data points or regions of interest) that can be detected by the *DataTrax2* program are classified into two categories: **General** and **Specific**.

General Events

General events can be located easily through the **Find** or the **Auto Find** dialog windows.

Find Dialog Window

The **Find** dialog window is accessed by selecting **Find** from the **Tools** menu. Data points of interest can be identified using stock commands from the **Find** dialog window. These commands can move cursors to: the **Previous** or **Next Maximum**; the **Previous** or **Next Minimum**; the **Previous** or **Next, Pos or Neg threshold crossing**; **Data Above** or **Below** a certain amplitude; data at a specific **Mark** with a particular *string* in a comment; data at an **Old** or **New Cursor** position; and more. The calculations to be performed on the selected data points or on the data between the points are chosen from the **Table Functions** list before the **Find** dialog window is opened.

The first step in the **Find** function is the selection of the **Cursor 1 Position** from the box to the right of this label. The next step in the **Find** function is the selection of the **Cursor 2 Position** from the box to the right of this label. Once the source of the data is selected and the **Find** routine is completed, it can be saved by clicking the **Save** button on the **Find** dialog window. The named routine is saved in the **Auto Find** folder as an **.iwxfind** file. When the **Find** dialog window is open, saved routines can be recalled by clicking on the **Load** button.

Find Example

As an example, the **Find** function could be used to determine the time (**T2-T1**) between two events. Occasionally, *R* waves in an ECG have significantly higher amplitudes than the other *R* waves in the same recording. The **Find** function could be used to measure the time between adjacent supranormal *R* waves. The process would include:

- Transferring the data of interest from the **Main** window to the **Analysis** window.

- Selecting the calculations (**V1**, **V2**, **V2-V1**) to be performed on the data from the **Table Functions** list.

- Placing one of the **2 Cursors** on a supranormal *R* wave on the left side of the **Analysis** window.

- Pulling down the **Tools** menu and selecting the **Find** function.

- Programming the cursor positions. To find the next supranormal *R* wave in the data, the **Cursor 1 Position** is set equal to **Old Cursor 1** (which is already on a supranormal *R* wave). The **Cursor 2 Position** is set to **Data Above** and the data channel is selected. The threshold amplitude, above which the **Find** function will stop, is entered into the box below the label **Cursor 2 Position**. Then the **Find** routine is saved.

- Clicking the **Find** button to place the cursors and to display the values for the selected **Table Functions** at the top of the **Analysis** window.

- Copying values and their headings to the **Journal** via the **Add Title to Jrnl** and **Add Data to Jrnl** functions in the **Right-Click** menu of the **Analysis** window.

Auto Find Dialog Window   The **Auto Find** dialog window (Figure 5-10 on page 75) is like the **Find** window. It is also accessible from the **Tools** menu and works only from the **Analysis** or **ScopeView** window. However, the **Auto Find** function can be programmed to find multiple data points with the same parameters within a data selection. The values at those points can be used for **Second Kind Analyses** and the results of these analyses can be written to the **Journal** automatically.

The **Auto Find** function can find the same types of data points found with the **Find** function, like: **Next Maximum**, **Previous Minimum**, **Data Above** a given value, the next **Mark** containing a particular alpha-numeric *string*, and more. Like **Find** routines, **Auto Find** routines may be constructed from the various parameters available and saved in the **Auto Find** folder. For periodic functions, maximum values, and minimum values, *DataTrax2* uses the threshold values set for **Periodic** data on the **Functions** page in the **Preferences** dialog window for the **Main** window. Just like other measurements of periodic data, the data must be scaled properly, which can be accomplished in most cases by clicking the **AutoScale** button in the **Main** window.

Auto Find Example   As an example, the **Auto Find** routine can be used to place cursors on all the *R* waves in an ECG record and to measure the period (**T2-T1**) of each beat. The easiest way to begin the **Auto Find** routine is to manually place the cursors on the two successive *R* waves at the beginning of the record. When selecting the commands to use in this routine, the user should ask themselves: "Where are the next two data points that are needed for measuring the next period (**T2-T1**)?" The answer is that the first data point in the next period is the second *R* wave from the preceding period, and the second R wave in the next period is the next maximum value or *R* wave in succession. Therefore, to make accurate measurements on the next period, the **Auto Find** window is programmed to set the **Cursor 1 Position** to be **Old Cursor 2** and the **Cursor 2 Position** to be the **Next Max**.
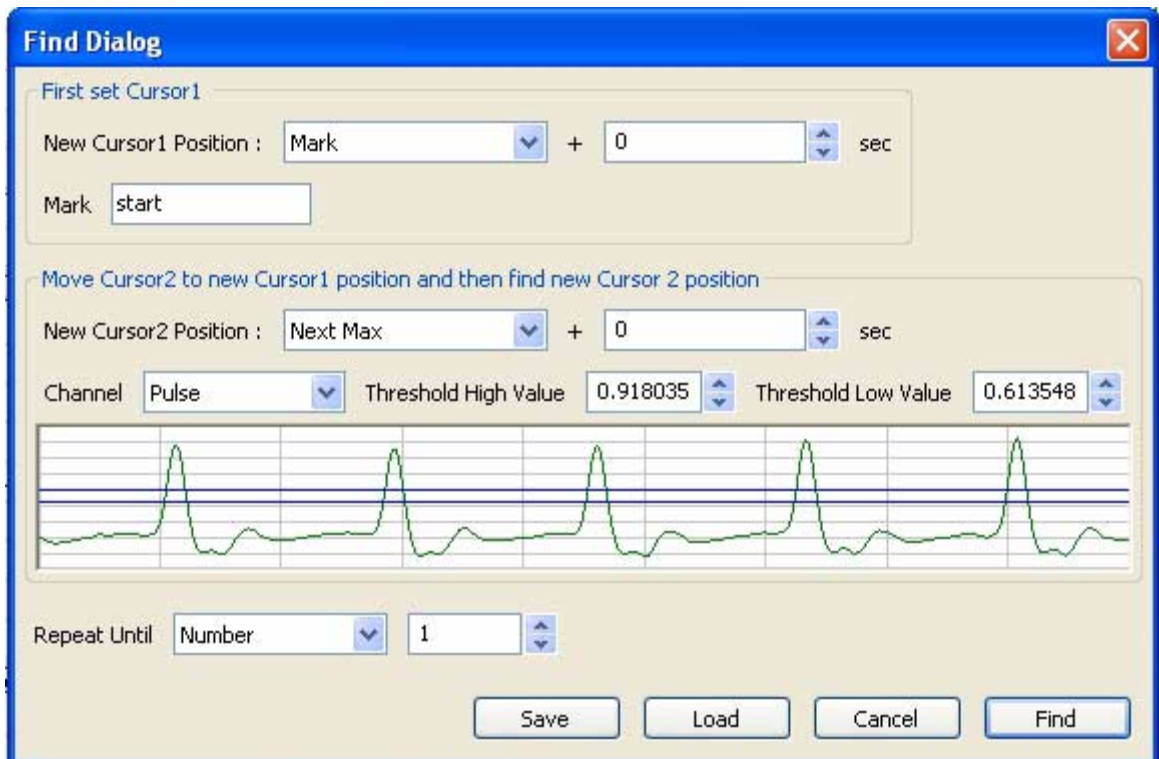

Figure 5-10: The Auto Find dialog window.

After the number of repetitions specified or the specified end of the **Auto Find** routine is reached, the **Auto Find** routine can be saved by clicking **Save** on the dialog window. The named **Auto Find** routines are saved as **.iwxfind** files and can be called by clicking on the **Load** button in the dialog window.

Specific Events

The detection of **Specific** events within a recording requires the use of an algorithm to find the point or **region of interest** (**ROI**). The *LVEDP*, the *R* in an ECG, and the *P* wave in an ECG are good examples of **Specific** events. Detection of these data points requires a defined software routine that resides in the *DataTrax2* program. The routines are accessible by selecting **Auto Mark** from the **Tools** menu and the name of the specific event from the **Auto Mark** drop-down menu. When the detection routine is run on the selected data, the software puts **Marks** on the data at the detected points. The automatic placement of **Marks** on the data accomplishes two things; first, it identifies the data points that need to be manipulated; second, it provides the user with visual cues to confirm that the software did the job of detecting the correct points. Based on the instructions given to *DataTrax2*, the detected data can be exported as text or loaded into a script.

In *DataTrax2*, there are **Auto Mark** functions available to detect: the components of the **ECG**, the **Blood Pressure** and **Peak Analysis**.

ECG

For example, if the **Auto Mark** detector is used to locate the components in an ECG, the **ECG** function is selected from the **Auto Mark** submenu on the **Tools** menu. A window known as the **AutoMark ECG Dialog** appears on the screen. It contains a display of the data to be used with the marking routine. In this window, selections must be made for the **Data Channel** that is the source of the recording being marked, the **Maximum Heart Rate** (BPM) in the data of interest, and the **Start** and **Stop** points for marking the data of interest.

The **Start** and **Stop** points could be the beginning and the end of the data file (Doc.), respectively. They could also be any **Mark** on the data file. Since **Marks** can be placed on the data after the recording, it is possible to place and use **Marks** that are on the margins of the data of interest. To assist in *R* wave detection, a threshold level must be set on the data.

Finally, press **Preview** and close the **AutoMark QRS Dialog** window by clicking **OK**. Each *R* wave in the selected section of data will be marked with a **Text Mark**. These marks can be viewed and edited in the **Marks** window. The **Marks** window is organized so that the marks can be sorted by **Time** (ascending or descending), **Channel**, or by **Content** by clicking on the title of the column in the **Marks** table. Individual comments, or ranges of comments, may be selected or deselected using the SHIFT and CONTROL keys. Appropriate collections of selected comments can be exported or deleted.
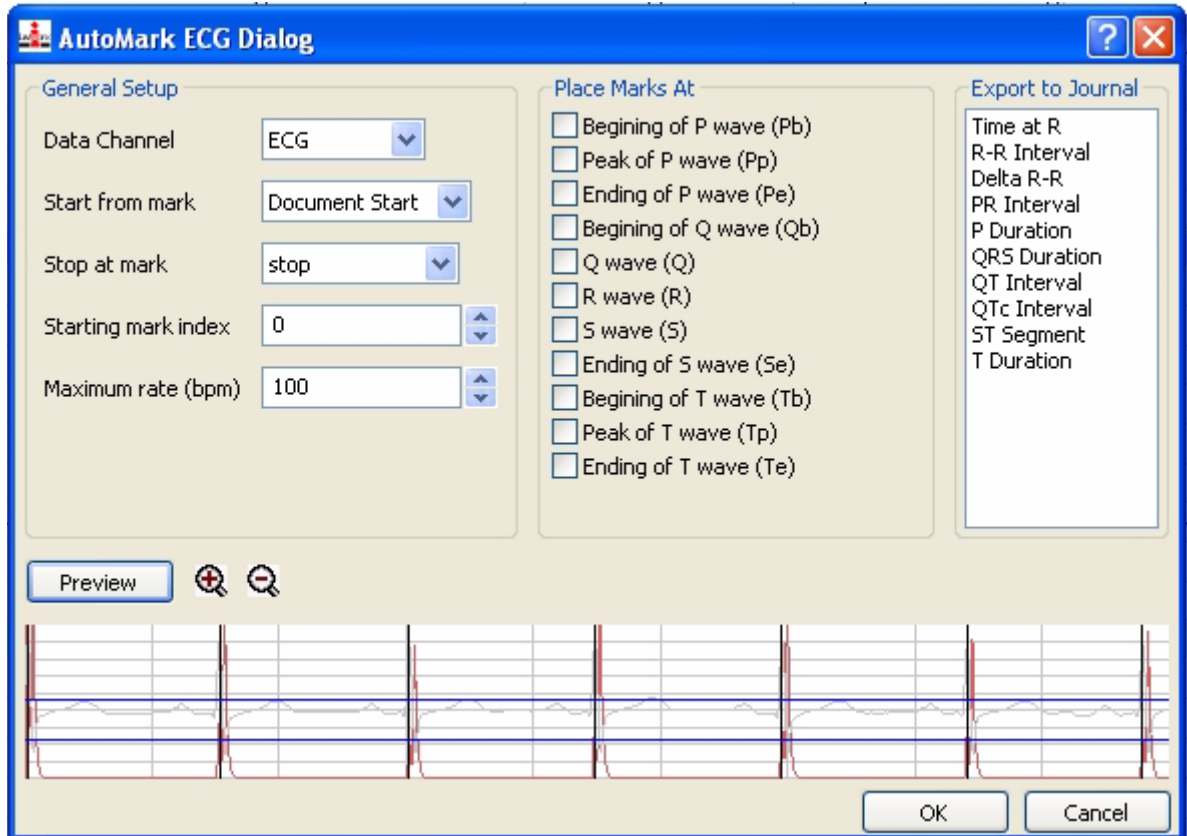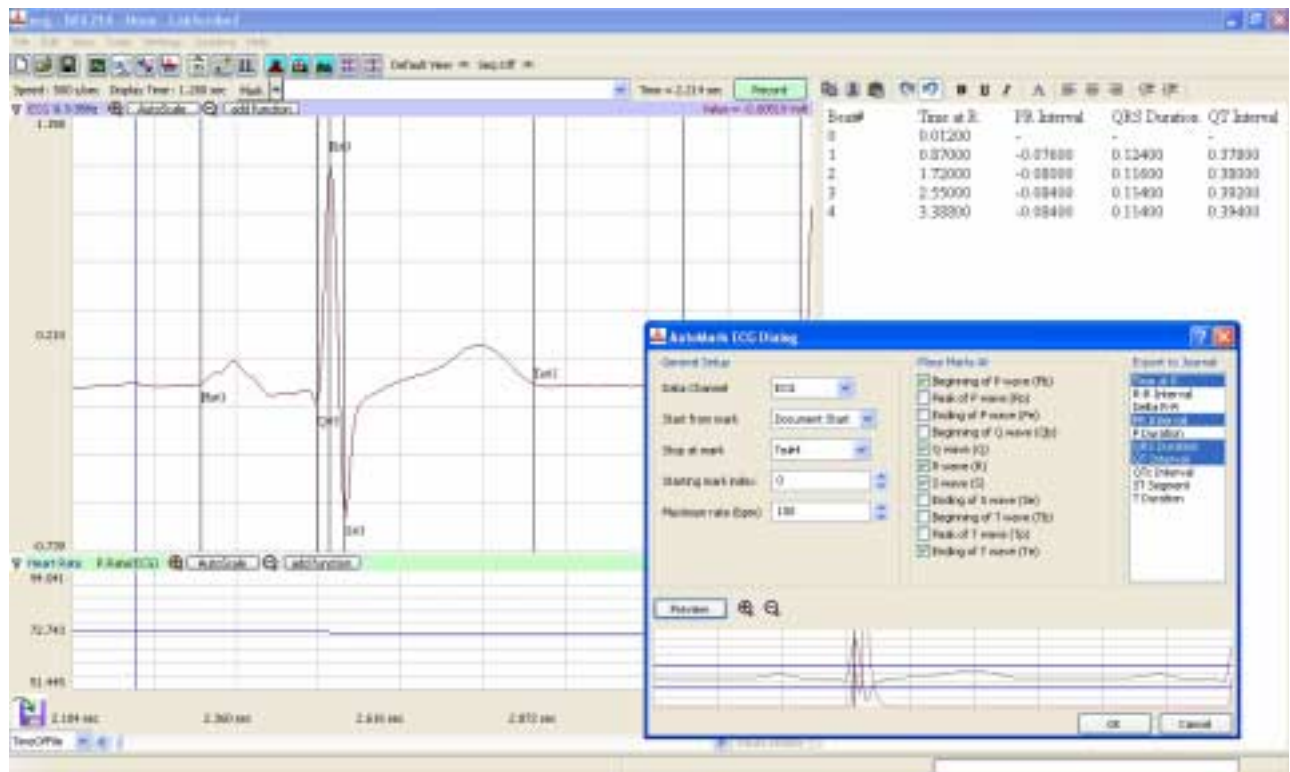
Figure 5-11: ECG AutoMark Dialog

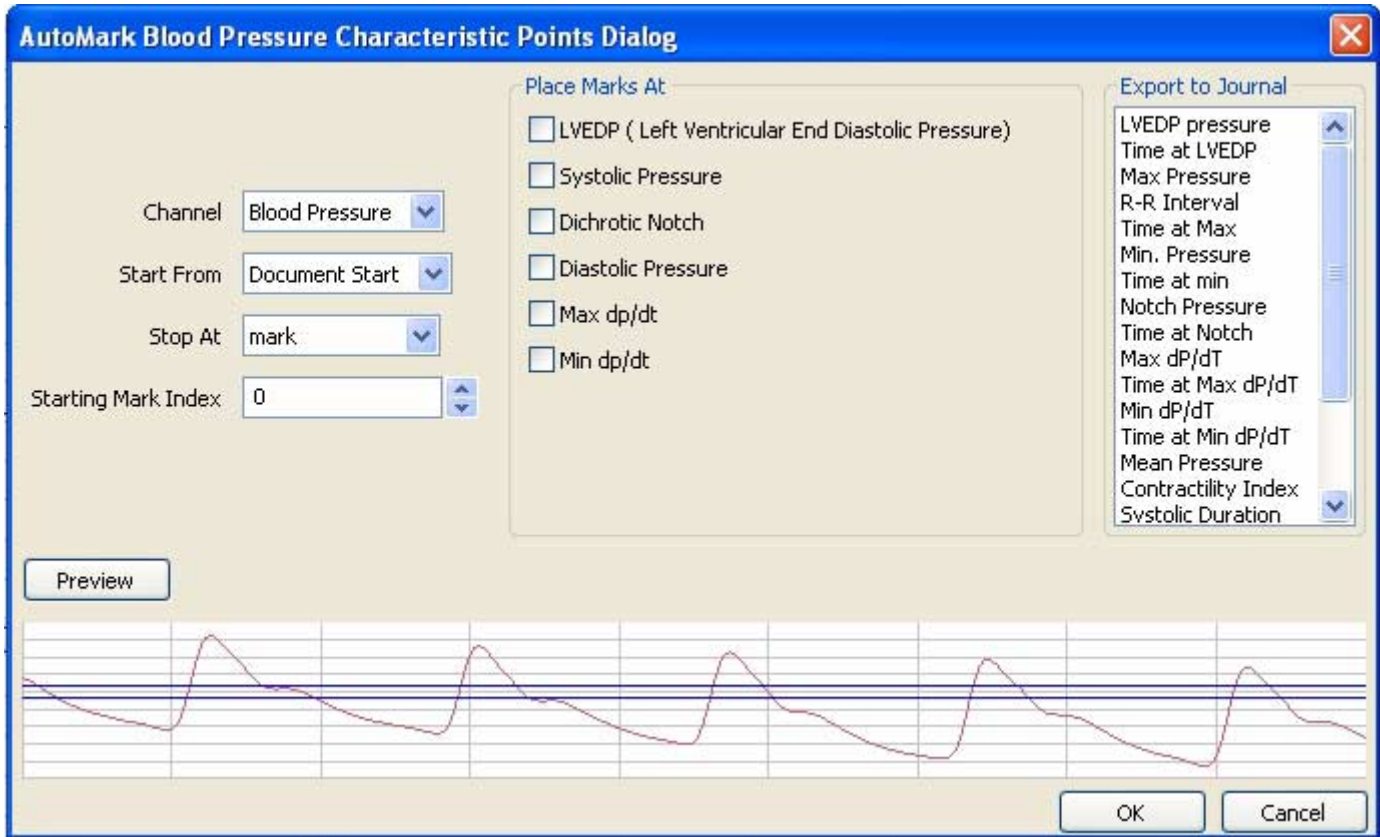*Figure 5-12: ECG AutoMark Example*
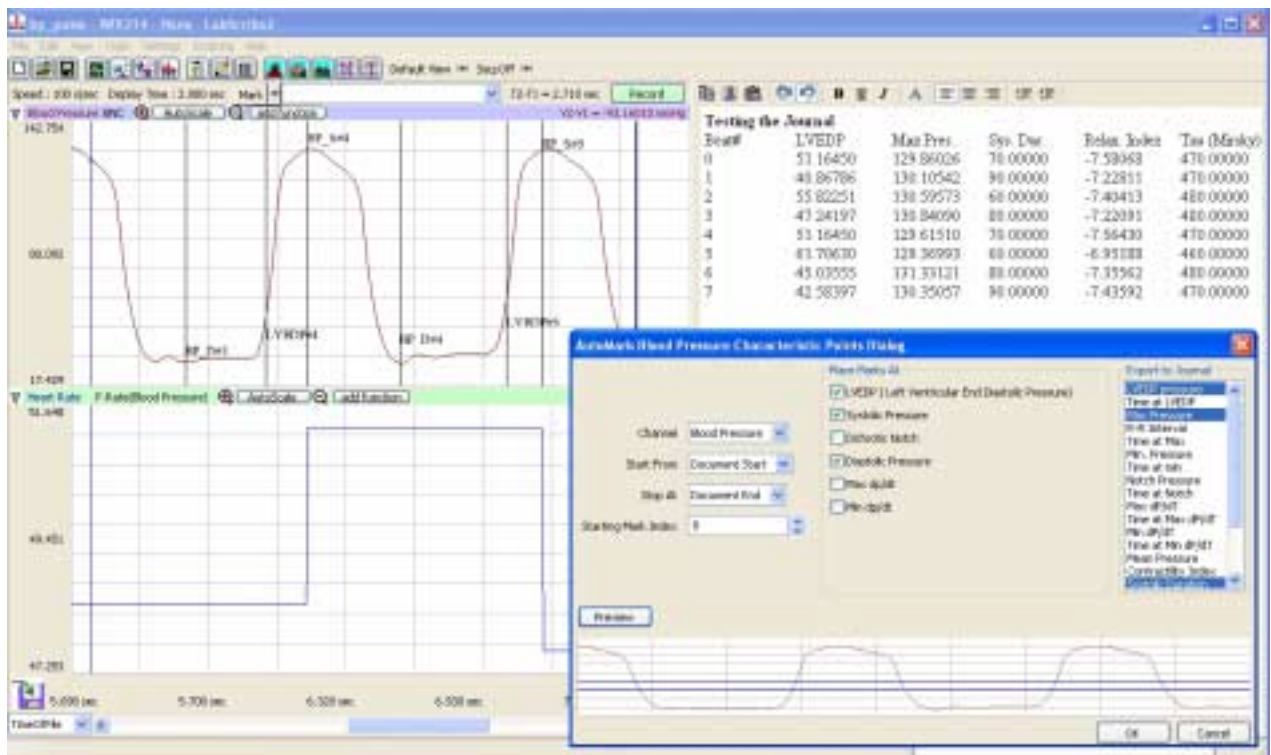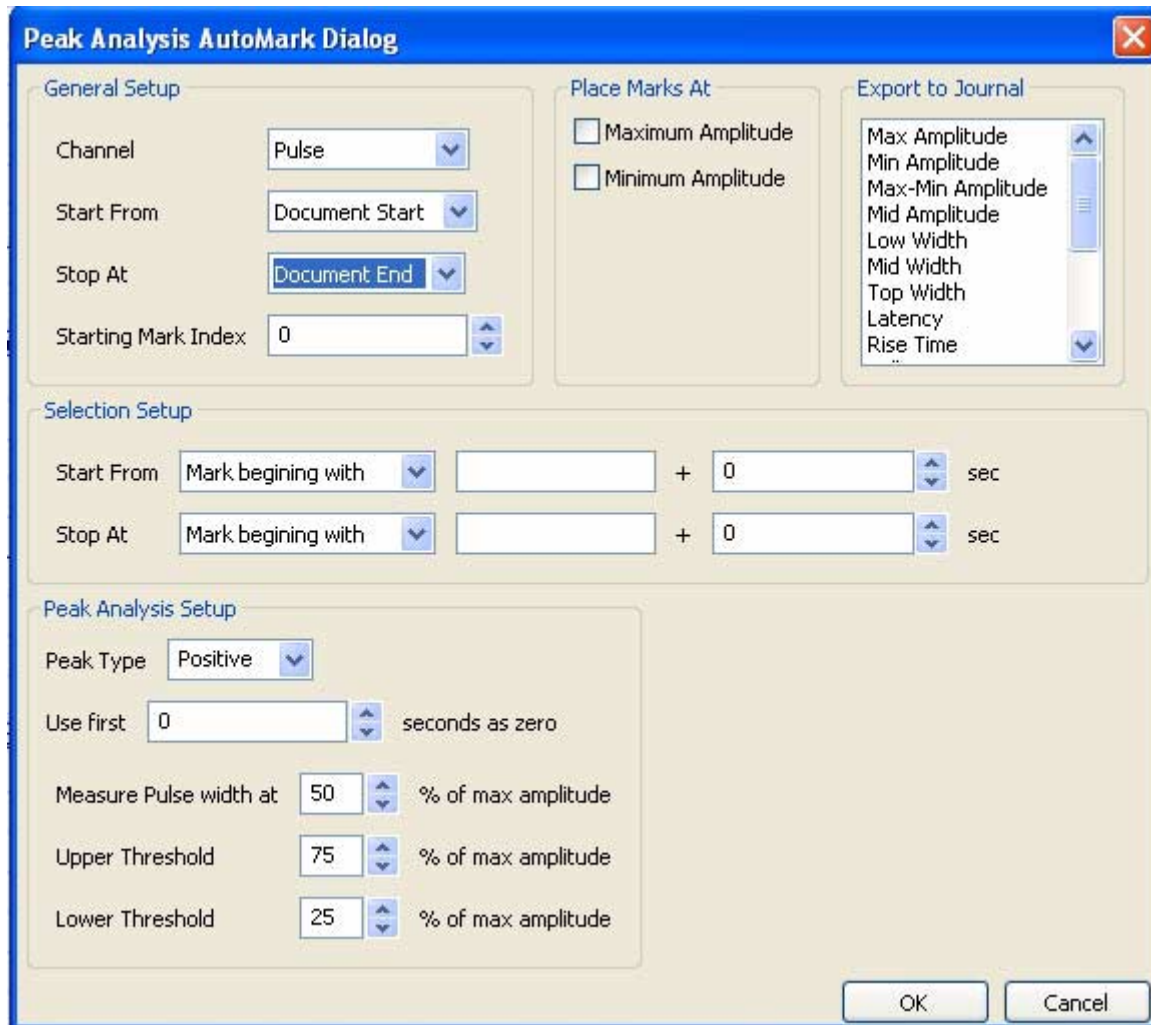
Blood Pressure

Figure 5-13:Blood Pressure Automark Dialog

Figure 5-14: Blood Pressure Automark Example

Analysis of the Fifth Kind: Things Not Possible with 1st, 2nd, 3rd, or 4th Kinds of Analyses

Not Yet Implemented in *DataTrax2*:

It is impossible to include all of the possible analyses and plotting capabilities required for biological research in a single program. We provide, what we believe to be, useful tools for completing the most common kinds of analyses. For circumstances where the built-in analysis tools fall short of the user's requirements, *DataTrax2* has a **Script** function. A **script** is a program, written in any of the languages supported by *DataTrax2*, that can perform a series of calculations on selected data. Once created, a script can be placed in the **Scripts** folder located in the *DataTrax2* folder. The script will then appear as a menu item in the *DataTrax2* **Scripting** menu. When selected, the **script** takes the selected range of recorded data, saves the data in a format compatible with the scripting language, loads the scripting program, and then executes the script on the saved data. The scripting programs supported by *DataTrax2* are listed below.

For example, we have created a **script** in <u>Python</u>, which operates on the *R-R* intervals of the ECG, which are detected by an **Auto Mark** function in the **Tools** menu. Then the *R-R* script computes and displays a tachogram, plots Delta R-R histograms and determines values for statistics, such as mean, mode, and standard deviation.

Writing Scripts

DataTrax2 has a special folder called **Scripts** located in the *DataTrax2* folder. Any scripting files or executables placed in this folder will be included in the **Scripting** menu when *DataTrax2* is launched.

Scripts can be run from the **Main**, **Analysis**, or **ScopeView** windows in the *DataTrax2* program. Different data is presented to the script depending on which window is used to call the script.

The **Default Export File Format**, for exporting the data that the script will use, is set on the **Advanced** page of the **Preferences** dialog window. The default file format is **.mat**, or the **Matlab File** format. For information about the <u>MatLab</u> and <u>DADiSP</u> file formats, refer to the **Export** section of this manual.

Scripts in the Main Window

When a script is run from the **Main** window, *DataTrax2* saves the complete data file in the **Scripts** folder as either **input.mat** (Matlab) or **inputXX.dat** (<u>DADiSP</u>). In <u>DADiSP</u>, each block is numbered as it is saved XX, starting with 00. When *DataTrax2* calls the script file, the <u>Windows</u> operating system opens the script file in the program associated with it. For example, a <u>Python</u> script should open in the <u>Python</u> program and a <u>MatLab</u> script will open in the <u>MatLab</u> program and so on. Scripts should be set up to open and analyze data files with the names as described.

Scripts in the Analysis Window

When a script is run from the **Analysis** window, *DataTrax2* saves the data in the **Analysis** window display in the **Scripts** folder as either **input.mat** (<u>Matlab</u>) or **input.dat** (<u>DADiSP</u>). When *DataTrax2* calls the script file, the <u>Windows</u> operating system opens the script file in the program associated with it. For example, a <u>Python</u> script should open in the <u>Python</u> program, and a <u>MatLab</u> script will open in the <u>MatLab</u> program and so on. Scripts should be set up to open and analyze data files with the names as described.

Scripts in the ScopeView Window

When a **script** is run from the **ScopeView** window, *DataTrax2* saves the data in the **ScopeView** window display in the **Scripts** folder as either **input.mat** (<u>Matlab</u>) or **input.dat** (<u>DADiSP</u>). When *DataTrax2* calls the script file, the <u>Windows</u> operating system opens the script file in the program associated with it. For example, a <u>Python</u> script should open in the <u>Python</u> program and a <u>MatLab</u> script will open in the <u>MatLab</u> program and so on. Scripts should be set up to open and analyze data files with the names as described.

Chapter 6: Input and Output

Stimulator

The **Stimulator** page in the **Preferences** dialog box controls the Digital-to-Analog Converter or DAC of the *WPI* hardware. It can be reached by selecting **Preferences** from the **Edit** menu. The stimulator feature sends the DAC output to the outside world. Using controls described below, output protocols can be built from "square wave" components that are listed on the **Simulator** page of the **Preferences** dialog window. The available output range of the stimulator is ±5V for the *WPI* 214 and ±10V for *WPI* 118 and 3xx.

The *WPI* 118 has two DACs, while the 3xx has a DAC on each channel. Each DAC can be controlled independently. The stimulator drop-down box (currently set to Stim1) allows selection of the stimulator to be setup. For devices with multiple stimulators, the import **Settings** button allows the user to copy settings from one stimulator to another.

Six modes of output, **Pulse**, **Trains**, **Constant** (voltage), **Step, Triangle** and **Ramp,** can be selected using the mode drop-down box (currently set to Off). **Triangle** and **Ramp** protocols are available only on the *WPI* 3xx.

The Bipolar checkbox allows the creation of bipolar pulses.

The stimulator can be set to fire when the recording is started by checking the **Start Stimulator with Recording** checkbox.

Time Resolution: The *WPI* 3xx has three time resolutions, 0.04ms, 0.4ms and 4 ms. The finer the time resolution the shorter the maximum time duration for each parameter. For example, at 0.04msec Time Resolution the maximum delay is 600 ms. At 0.4ms this increases to six seconds, while at the 4msec resolution the maximum delay is 60 seconds.

The software configures itself depending on the hardware selected and will limit the available range of values for each parameter accordingly.

Toolbar Steps: This determines the minimum step increments that the stimulator toolbar will use when the up/down buttons on the toolbar are clicked. These are also the increments used by sequences for programming the stimulator.

Pulse Protocols
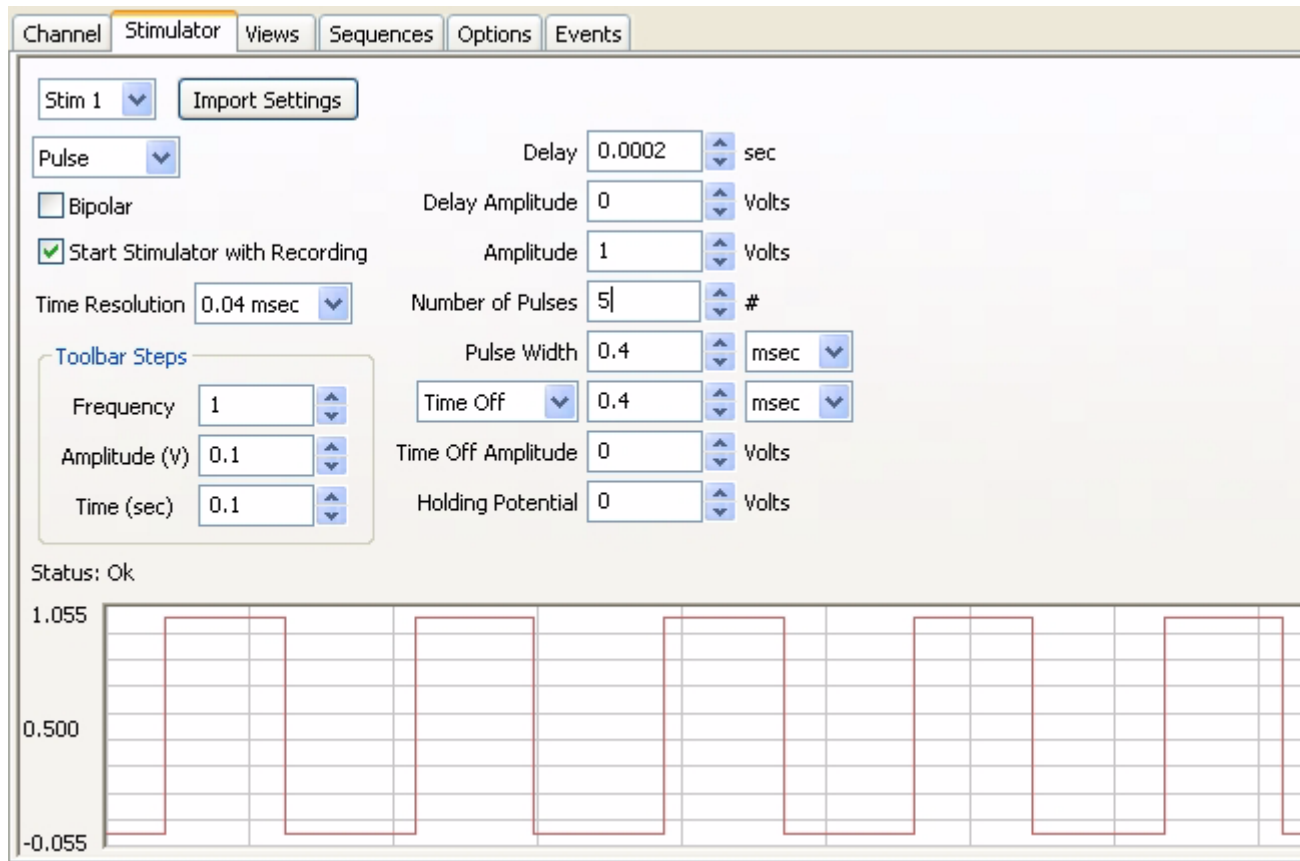
Figure 6-1: The parameter boxes on the Stimulator page of the Preferences dialog window.

In order to understand how protocols are created, it is necessary to define the terms used on the **Stimulator** page of the **Preferences** dialog window:

Delay: This is the time between the beginning of the recording, at the clicking of the **Record** button, and the first pulse. Delay is adjustable as follows:

WPI 118 (6500 ms), *WPI* Latrax 24 series:(600 ms, 6s or 60 s).

Amplitude: This is the height or voltage of the pulse or wave being generated. The amplitude programmed from the **Preferences** window will be the same for all pulses leaving the stimulus output unless the amplitude is changed manually from the **Preferences** window or the **Stimulator Control Panel**, or automatically from a programmed sequence (more later).

Number of Pulses: The total number of pulses that can be sent from the stimulator output

Pulse Width: The pulse is the basic unit of an output protocol and it has two basic dimensions, amplitude, as described above, and length. The length of the pulse is also called pulse width or pulse duration .

Pulse Frequency: This is the number of programmed pulses that are delivered in one second period of time. The maximum frequency that can be set is dependent on the pulse duration. Remember that the period (P) is the inverse of the frequency (F), or P = 1/F.

Time Off: This is the interpulse duration, i.e. the amount of time between two consecutive pulses. Pulse Frequency is 1/(pulse width + Time Off).

Holding Potential: This is a voltage between ±10V that can be programmed to shift the resting voltage of an excitable tissue (like a nerve). It is also known as a holding voltage because it can be used to hyperpolarize the membrane potential of an excitable tissue and prevent it from depolarizing or "firing."

Building Output Protocols in Pulse Mode

To record stimulus pulses like the examples that follow, the stimulator output of an *WPI* unit can be connected to one of its own BNC inputs. On the *WPI* 118, this is easily accomplished by using a BNC-BNC cable. However, the stimulator outputs of the *WPI* 214 unit consists of three color-coded banana jacks, so a Double Banana-BNC adapter or cable is needed to connect two of the banana jacks to one of the BNC inputs. To record positive pulses, connect the adapter to the positive (red) and ground (green) banana jacks; to record negative pulses, connect the adapter to the negative (black) and ground (green) banana jacks. Double-Banana adapters have a flag or bump on one side that indicates which side should be attached to the ground jack.

Note: Never connect both the positive (red) and the negative (black) banana outputs of a 214 unit to its own inputs, as this causes a short circuit that could damage the amplifier. These red and black outputs can be connected to other devices (nerve chambers, stimulating electrodes, and more), but not to its own inputs.

An Example of Continuous Pulses

1   To construct and record some examples of stimulus pulses, select **Preferences** from the **Edit** menu. On the **Channel** page of the **Preferences** dialog window set a channel that has a BNC input to record at 10,000 samples/second and display 0.5 seconds on the screen. Also select the Stim1 channel and set it to record the stimulator. (Figure 6-2 on page 83).
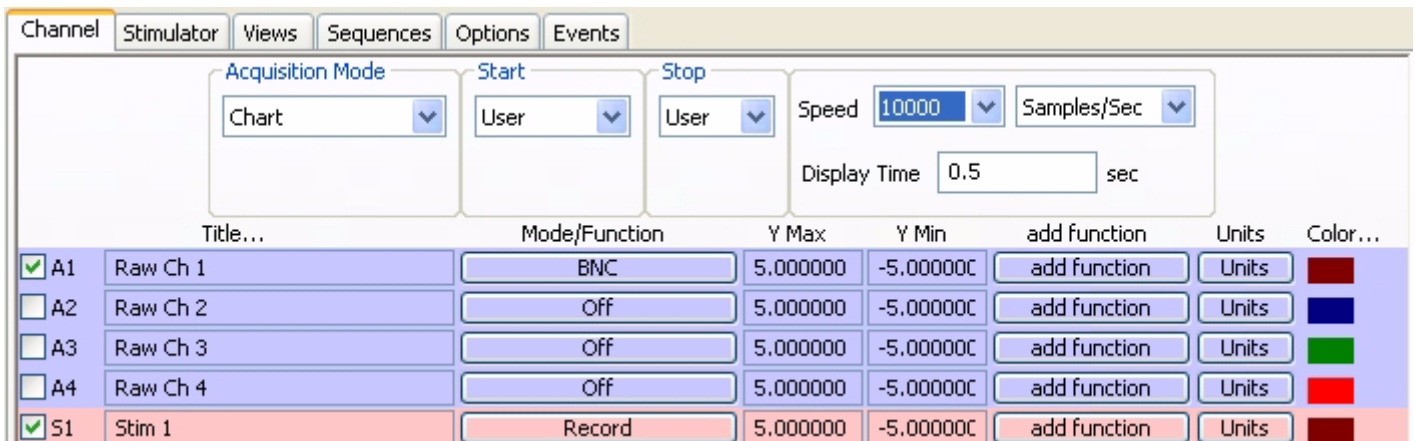
Figure 6-2: Channels page of the Preferences dialog window set for the pulse mode example.

2   Go to the **Stimulator** page of the **Preferences** dialog window; select **Pulses** from the box in the upper left corner of the page.

3   Enter the following values into the appropriate boxes on the **Stimulator** page:

Delay	100msec (an arbitrary value)
Amplitude	1V
Number of Pulses	0 (for continuous)
Pulse Width	5ms
Pulse Frequency	100 Hz

Holding Potential   0 V

These settings will create a protocol that delivers continuous square waves with 1V amplitude and 100Hz frequency.

4 Press the **Record** button. The DAC will wait 100 milliseconds (the **Delay** value entered) and begin to deliver 5ms pulses at the rate of 100 pulses per second. Each pulse will be 1V high. These pulse parameters can be adjusted in any way with one exception, the pulse width cannot be longer than the inverse of the frequency (or the period). In this example, the 100Hz square wave delivers 100 events or pulses every second. Therefore, each single event or pulse must occur in 1/100th of a second, which is a 10ms period. In this example, the pulse width cannot be longer than the period of 10ms or the pulse will overlap the next pulse. The percentage of the period occupied by the pulse width is known as the duty cycle and refers to the amount of time the output amplifiers are active.

An Example of a Burst of Pulses   Use the settings listed for the example of continuous pulses to produce a short burst of pulses, with one exception. Set the **Number of Pulses** to a number other than zero. If the number 10 is entered in this box, the output from the DAC would wait the 100 millisecond delay from the **Start** signal, deliver its 100Hz square wave of 5ms by 1V pulses as before, but stop when 10 pulses had been delivered. A graphical representation of the two examples above is pictured in Figure 6-3 on page 84.

Continuous output

Burst or Train output
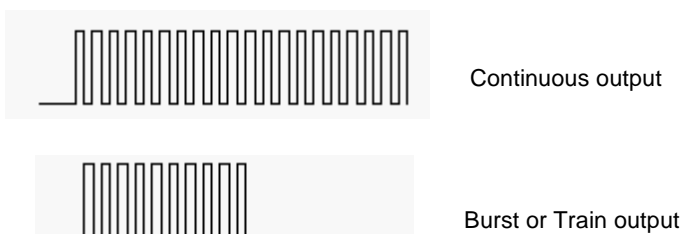
Figure 6-3: Continuous (top) and burst (bottom) of pulses that are the same amplitude, width, and frequency.

Train Protocol   A train of pulses is more properly called a burst of pulses. Trains contain a specified **Number of Pulses**, which occur a specified number of times (**Number of Trains)** at regular intervals (**InterTrain Duration)**. The train protocol is an extension of the pulse protocol so it is essential to be familiar with the pulse protocol to use the train protocol.
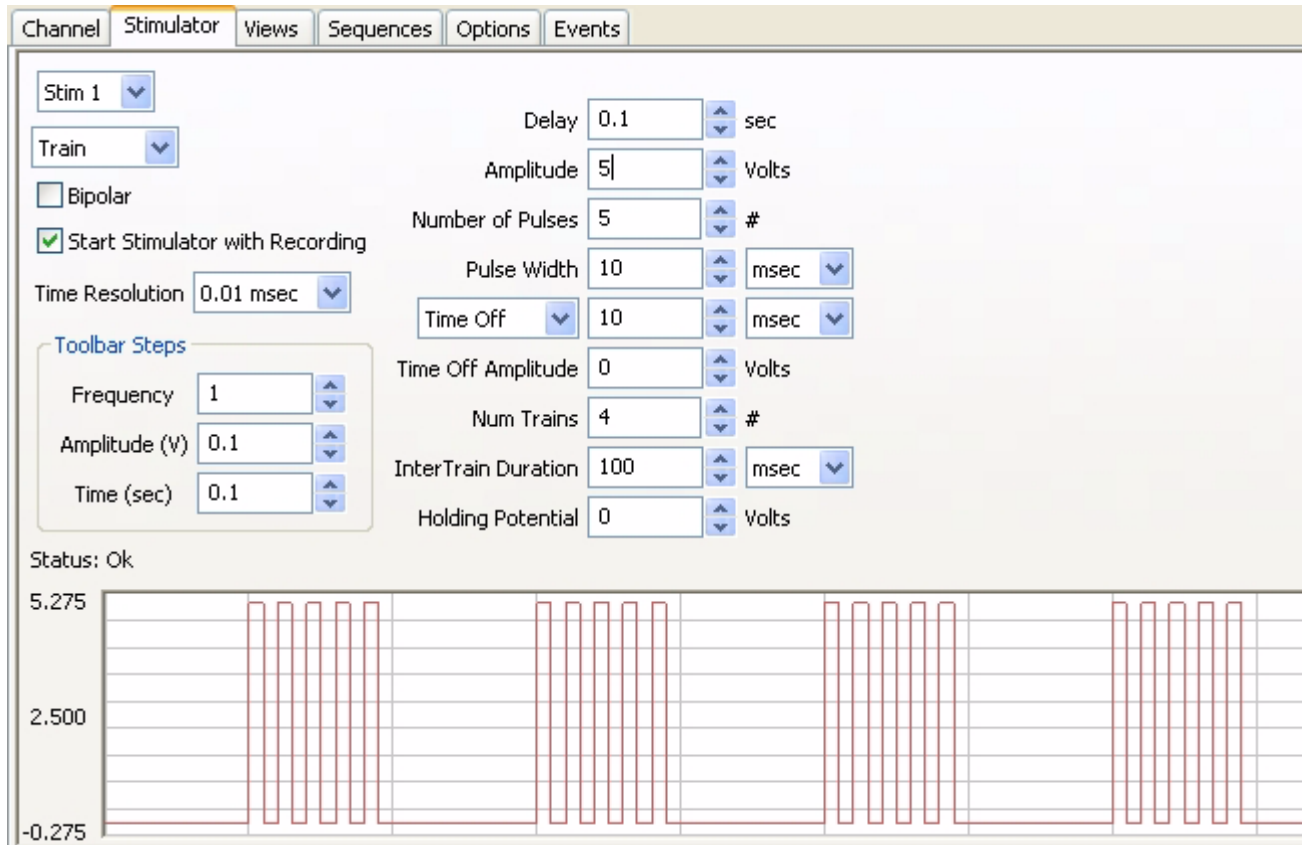
Figure 6-4: The parameter boxes on the Stimulator page for Train mode.

The terms used on the **Stimulator** page of the **Preferences** dialog window for configuring trains of pulses, include:

Delay: This is the time between the beginning of the recording, at the clicking of the **Record** button and the first pulse. Delay is adjustable as follows:

WPI 214 (650ms), *WPI* 118 (6500 ms), *WPI* 3xx:(600 ms, 6s or 60 s)

Amplitude: This is the height or voltage of the pulse or wave being generated. The amplitude programmed from the **Preferences** window will be the same for all pulses leaving the stimulus output unless the amplitude is changed manually from the **Preferences** window or the **Stimulator Control Panel**, or automatically from a programmed sequence (more later)

Number of Pulses: The total number of pulses that can be sent from the stimulator output

Pulse Width: The pulse is the basic unit of an output protocol and it has two basic dimensions, amplitude, as described above, and length. The length of the pulse is also called pulse width or pulse duration

Pulse Frequency: This is the number of programmed pulses that are delivered in a one second period of time. The maximum frequency that can be set is dependent on the pulse duration. Remember that the period (P) is the inverse of the frequency (F), or P = 1/F

Time Off: This is the interpulse duration, i.e. the amount of time between two consecutive pulses. Pulse Frequency is 1/(pulse width + Time Off)

Number of Trains: Number of Trains

InterTrain Duration: The length of time between successive trains or bursts

Holding Potential: This is a voltage between ±10V that can be programmed to shift the resting voltage of an excitable tissue (like a nerve). It is also known as a holding voltage because it can be used to hyper polarize the membrane potential of an excitable tissue and prevent it from depolarizing or "firing."

Building Output Protocols in Train Mode

Connect the stimulus outputs to one of its own BNC inputs as described in the **Building Output Protocols in Pulse Mode** section of this chapter.

Note: Never connect both the positive (red) and the negative (black) banana outputs of a 214 unit to its own inputs, as this causes a short circuit that could damage the amplifier. These red and black outputs can be connected to other devices (nerve chambers, stimulating electrodes, and more), but not to its own inputs.

In many applications, more than one burst or train of pulses must be delivered. Before the parameters of the trains or bursts are specified, the dimensions of the pulses that will fill the bursts must be selected. After the pulse is designed, the frequency, duration, and number of trains can be programmed.

An Example of Pulse Trains

1 To construct and record an example of pulse trains, select **Preferences** from the **Edit** menu. On the **Channels** page of the **Preferences** dialog window set a channel that has a BNC input to record at 10,000 samples/second and display 0.5 second on the screen (Figure 6-2 on page 83).

2 Go to the **Stimulator** page of the **Preferences** dialog window; select **Trains** from the box in the upper left corner of the page.

3 Enter the following values into the appropriate boxes on the **Stimulator** page:

Delay	100msec (an arbitrary value)
Amplitude	1V
Number of Pulses	10 (10 pulses at 100 Hz = 0.1 sec)
Pulse Width	5ms
Pulse Frequency	100 Hz
Number of Trains	25
InterTrain Duration	900ms

These settings will create a protocol that delivers: a train of pulses every second; 10 pulses in each train with a frequency of 100 Hz; and, all pulses with an amplitude of 1V and a width of 5msec. Since a train occurs every second and is 100msec long, the time between trains, the **InterTrain Duration,** needs to be 900msec.

4 Press the **Record** button. The **DAC** will wait 100 milliseconds (the delay value entered) and begin to deliver ten 5ms pulses in a tenth of second. Each pulse will be 1V high. After 900 milliseconds, a second burst of ten pulses with the same parameters will occur. These bursts will appear in this manner until a total of 25 bursts have occurred.

5 If you only wanted the train or burst to repeat four times, the number four should be entered in the **Number of Trains** box. The completed output protocol should look something like Figure 6-5 on page 87.

Figure 6-5: Burst of pulses separated by Inter Train Durations.

Constant Voltage Protocol

Selecting the **Constant** (voltage) option on the **Stimulator** page of the **Preferences** dialog window disables the entry boxes for all stimulus parameters except **Amplitude** and **Delay**. When the **Record** button is clicked, the voltage set on the **Stimulator** page is delivered to the low voltage output of the *WPI* unit. The voltage output terminates when the recording is stopped.

Step Mode

Step mode is used almost exclusively for voltage clamp protocols. In **Step** mode, the amplitude of stimulator output can be increased or decreased in a step-wise manner, and the user can specify the following parameters: **Delay**, **Start Amplitude**, **Stop Amplitude**, **Number of Steps**, **Step Width**, **Time off** (or Inter Step Width**)**, **Repeat Count**, **Interprotocol Duration** and **Holding Potential**
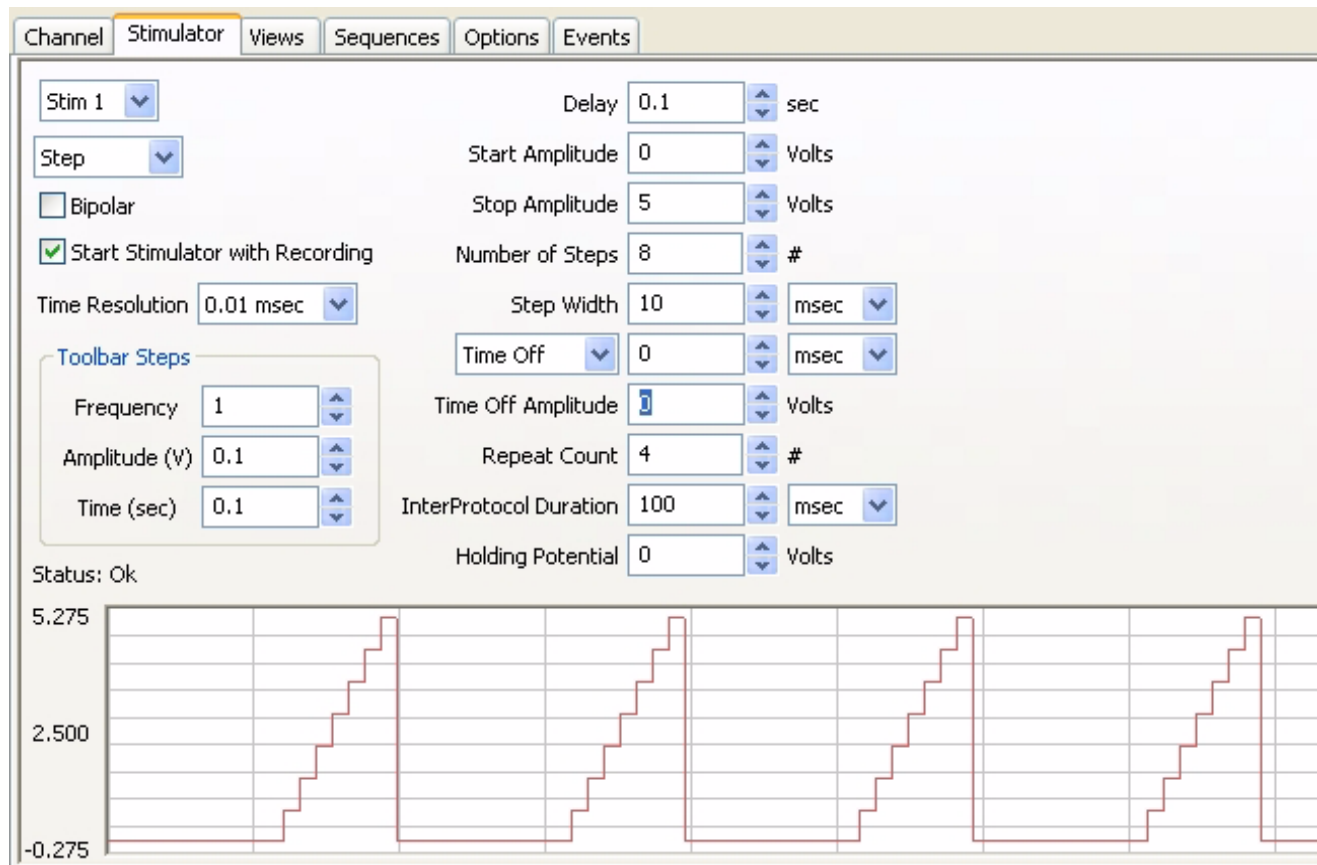
Figure 6-6: The parameter boxes on the Stimulator page set for Step mode.

Building Output Protocols in Step Mode

The amplitude of each individual step in the protocol is determined by the *DataTrax2* program with the **Starting** and **Ending Amplitudes** and the **Number of Steps** entered on the **Stimulator** page. The equation used to do this calculation is:

$$\frac{(\text{\underline{Start Amplitude - End Amplitude}})}{\text{Number of Steps}} = \text{Voltage Increment}$$

If the user knows the voltage increment needed at each step, the equation can be transposed to solve for the **Number of Steps** required in the protocol and this value can be entered on the **Stimulator** page:

$$\frac{\text{(Start Amplitude - End Amplitude)}}{\text{Voltage Increment}} = \text{Number of Steps}.$$

Likewise, if the user knows the previous amplitude and the voltage increment, the succeeding amplitude can be calculated:

Previous Amplitude + Voltage Increment = Succeeding Amplitude

With the starting, ending and incremental voltages set through **Preferences**, the voltage will change in a step wise manner until the ending voltage is reached. The length of the protocol is determined by the **Step Width** and the time between steps, the **Time Off**.

Repeat Count: The number of times the above step protocol will repeat

Interprotocol duration: The duration of time between the termination of one protocol and the initiation of the next protocol

An Example of Voltage Steps

1   To construct and record an example of pulses in **Step** mode, select **Preferences** from the **Edit** menu. On the **Channels** page of the **Preferences** dialog window set a channel that has a BNC input to record at 10,000 samples/second and display 0.5 second on the screen (Figure 6-2 on page 83).

2   Go to the **Stimulator** page of the **Preferences** dialog window; select **Step** from the box in the upper left corner of the page.

3   Enter the following values into the appropriate boxes on the **Stimulator** page:

Delay	0
Starting Amplitude	-1V
Ending Amplitude	+1V
Number of Steps	8
Step Width	100 ms
Time Off	0
Repeat Count	1
Interprotocol duration	0
Holding Potential	0

4   These settings will create a wave form that starts at -1 Volt and climbs to +1 Volt in eight steps, each step has a voltage increment of 0.25V and 100ms wide.

5   Press the **Start** button. The resulting wave would be similar to the step-wise elevation of output amplitude seen in Figure 6-7 on page 88.
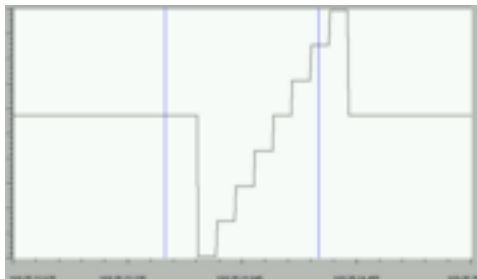
Figure 6-7: An example of a continuous step protocol.

6 To create a step protocol where the voltage returns to a baseline value between steps, alter the **Time Off** to a number greater than zero. The resulting wave would be similar to that seen in Figure 6-8 on page 89.
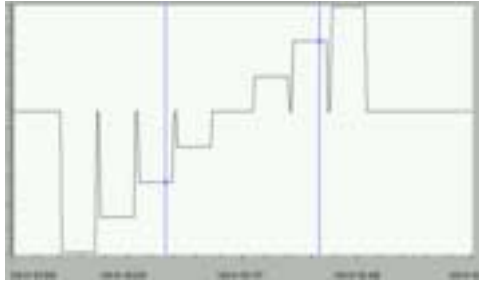
Figure 6-8: An example of a step protocol punctuated by returns to baseline between steps.

7 A **Constant** voltage protocol can be combined with the **Step** protocol to have the voltage return to a different baseline after the completion of the step sequence. For example, setting the **Holding Potential** in the sample step protocol to 500mV would cause the output amplitude of the stimulator to return to 500mV at the end of the step sequence. Subsequent firings of the protocol would begin from the new baseline of 500mV, drop to the **Start Amplitude** in the example (-1V), step to the **End Amplitude** (+1V), and then return to the **Holding Potential** of 500mV.

Ramp Protocol

Figure 6-9: The parameter boxes on the Stimulator page set for Ramp mode

Delay: This is the time between the beginning of the recording, at the clicking of the **Record** button, and the beginning of the ramp

Delay Amplitude: The amplitude during the delay period. Only available on the *WPI* 3xx

Start Amplitude: The starting amplitude of the ramp

Stop Amplitude: The ending amplitude of the ramp

**Rise Time**: The time taken to go from the Start Amplitude to the Stop Amplitude

Num Ramps: Number of Ramps

Interprotocol Duration: Time between ramps

Holding potential: The amplitude during the interprotocol duration

Triangle Protocol
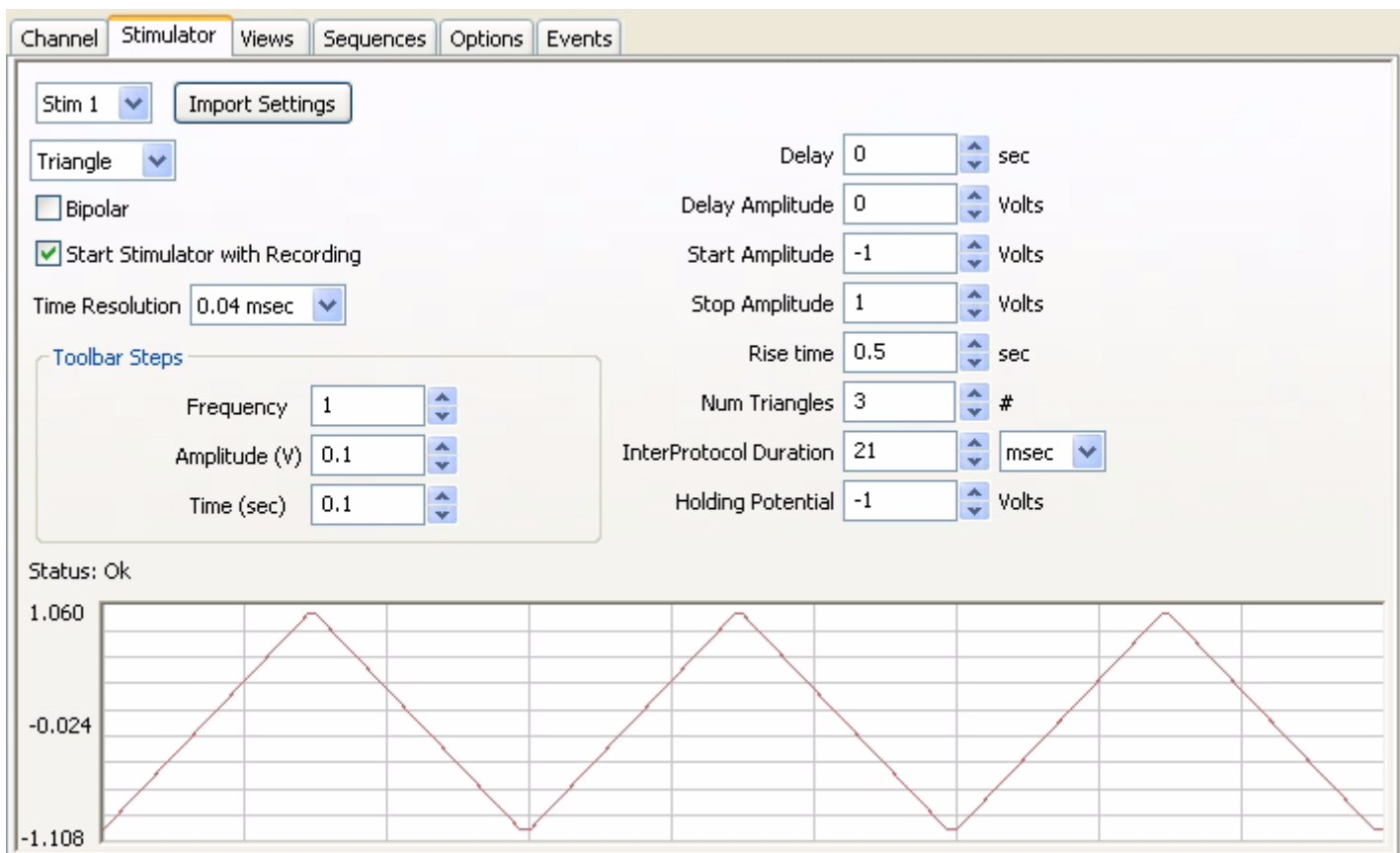
Figure 6-10: The parameter boxes on the Stimulator page set for Triangle mode

 Delay: This is the time between the beginning of the recording, at the clicking of the **Record** button, and the beginning of the triangles

 Delay Amplitude: The amplitude during the delay period. Only available on the *WPI* 3xx

 Start Amplitude: The starting amplitude of the triangle

 Stop Amplitude: The ending amplitude of the triangle

 Rise Time: The time taken to go from the Start Amplitude to the Stop Amplitude

 Num Triangles: Number of Triangles

 Interprotocol Duration: Time between triangles

Holding potential: The amplitude during the interprotocol duration. Should be set to Start Amplitude if not being used

The Stimulator Panel

Selecting the **Stimulator Panel** item in the **View** menu (Figure 6-11 on page 91) will place a **Stimulator Control Panel** (Figure 6-12 on page 91) on the **Main** window.
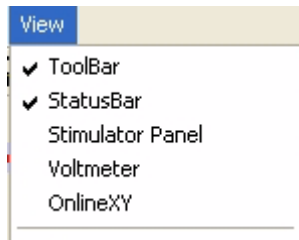
The Stimulator Panel can be used to create a new protocol or change the parameters for an existing protocol

Each stimulator mode has a unique toolbar as shown below.

Figure 6-11: The View menu.

Pulse Mode

Figure 6-12: The Stimulator Pulse Control Panel.

Amp: Amplitude of the stimulus pulse in Volts

#pulses: Number of Pulses

Width(ms): Width of the pulse in milliseconds (ms)

F(Hz): Frequency of stimulation in Hertz (Hz)

HP: Holding potential

Apply: applies any changes to the stimulus protocol made through the **Stimulator Toolbar**. Clicking **Apply** while recording will send a stimulus pulse from the output

Train Mode
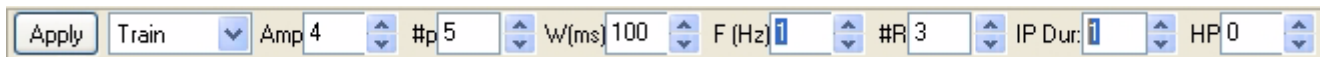
Figure 6-13: The Stimulator Train Control Panel.

Amp: Amplitude of the stimulus pulse in Volts

#p: Number of Pulses

W(ms): Width of the pulse in milliseconds (ms)

F(Hz): Frequency of stimulation in Hertz (Hz)

#R: Number of Trains (or Repeat Count)

IP Dur: InterTrain(or inter protocol) duration

HP: Holding Potential

Apply: Applies any changes to the stimulus protocol made through the **Stimulator Toolbar**. Clicking **Apply** while recording will send a stimulus pulse from the output

Step Mode

Figure 6-14: The Stimulator Step Control Panel.

A1: Starting Amplitude

A2: Stopping Amplitude.

#steps: Number of Steps

W(ms): Width of the step in milliseconds (ms)

F(Hz): Frequency of stimulation in Hertz (Hz)

#R: Repeat Count

IP Dur: Inter protocol duration

HP: Holding Potential

Apply: Applies any changes to the stimulus protocol made through the **Stimulator Toolbar**. Clicking **Apply** while recording will send a stimulus pulse from the output

Constant Mode

Figure 6-15: The Stimulator Constant Control Panel.

Amplitude: Amplitude.

Apply: Applies any changes to the stimulus protocol made through the **Stimulator Toolbar**. Clicking **Apply** while recording will set the amplitude at the output.

Ramp Mode
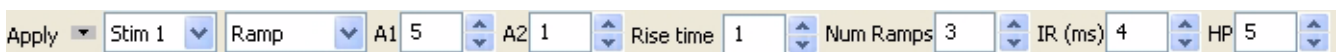
Figure 6-16: The Stimulator Step Control Panel.

A1: Starting Amplitude

A2: Stopping Amplitude

Rise time: The time taken to go from the Start Amplitude to the Stop Amplitude

Num Ramps: Number of Ramps

IR: Time between Ramps

HP: Holding potential

Apply: Applies any changes to the stimulus protocol made through the **Stimulator Toolbar**. Clicking **Apply** while recording will send a stimulus pulse from the output

Triangle Mode
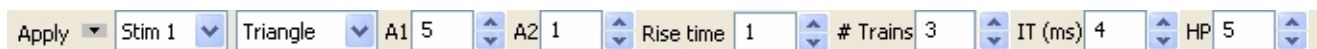
Figure 6-17: The Stimulator Step Control Panel.

A1: Starting Amplitude

A2: Stopping Amplitude

Rise time: The time taken to go from the Start Amplitude to the Stop Amplitude

 # Triangles: Number of Triangles

IR: Time between Triangles

HP: Holding potential

Apply: Applies any changes to the stimulus protocol made through the **Stimulator Toolbar**. Clicking **Apply** while recording will send a stimulus pulse from the output

---

Note: *The Stimulator preferences dialog will draw whatever protocol you specify. Be sure to examine the output representation carefully before closing the dialog to confirm that this is in fact the output that you want.*

Stimulator Display

On *WPI* 214/118/3xx, the stimulator output can be displayed as a channel in the **Main** window.

Stimulus Protocols Built with the Sequence Builder

Experiments are often designed to record the response of a cell (or a tissue) to progressively larger or more frequent stimuli. In these cases, parameters of the stimulus are changed before each recording of the cell's response to the next stimulus. The **Sequence Builder**, a page in the **Preferences** dialog window, can be used to automatically change parameters of the stimuli released from the **DAC**.

For example, excitable tissues like nerves and muscles are composed of multiple fibers, each with a different diameter, conduction velocity and threshold. Fibers with higher thresholds require a larger stimuli to evoke their action potentials. Increasing the amplitude of the stimulus sent to the excitable tissue will cause more fibers in the tissue to fire. This is known as recruitment and is measured as an increase in the amplitude of the tissue's compound action potential.

In **Scope** mode, the **Sequence Builder** and other pages (**Channels**, **Stimulator**) in the **Preferences** dialog window can be used to record each response of the excitable tissue to a stimulus and automatically increase the amplitude of the stimulus between each trial.

Channels Page

To construct an example of an **Sequence** designed to control the DAC and to record in **Scope** mode, select **Preferences** from the **Edit** menu. On the **Channel** page of the **Preferences** dialog window set:

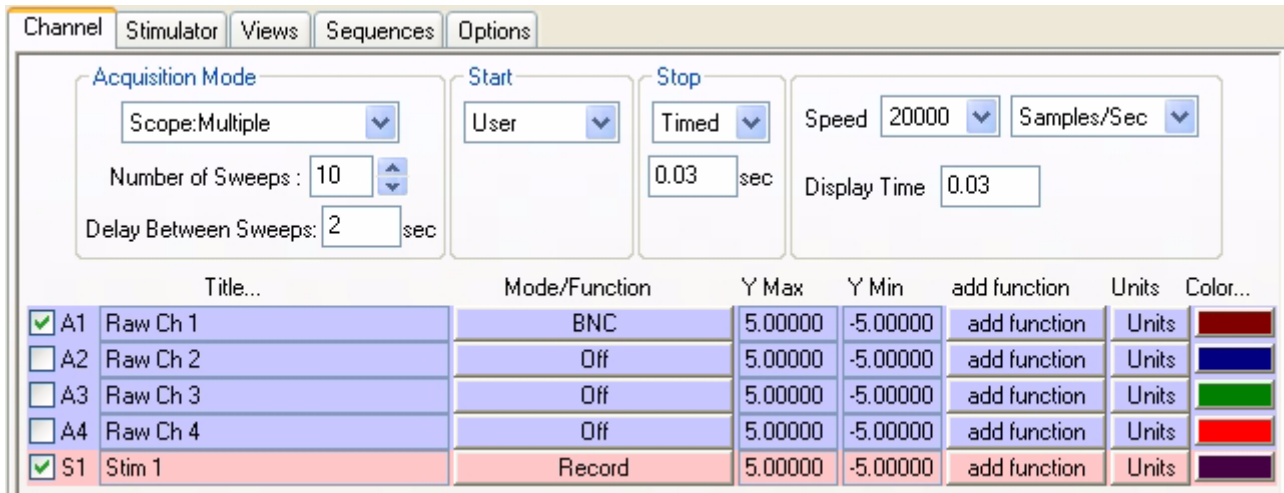|	
Acquisition Mode	Scope: Multiple
Number of Sweeps	10
Delay Between Sweeps	2
Start	User
Stop	Timed: 0.03 sec
Sampling Speed	20000
Display Time	0.030 sec
Select A1	Raw Channel 1: set Mode to BNC
Deselect A2-A4	i.e. raw channels 2, 3 and 4
Select S1	stim 1: Set Mode to Record.

Figure 6-18: Channels page of the Preferences dialog window set for testing an Output Sequence.
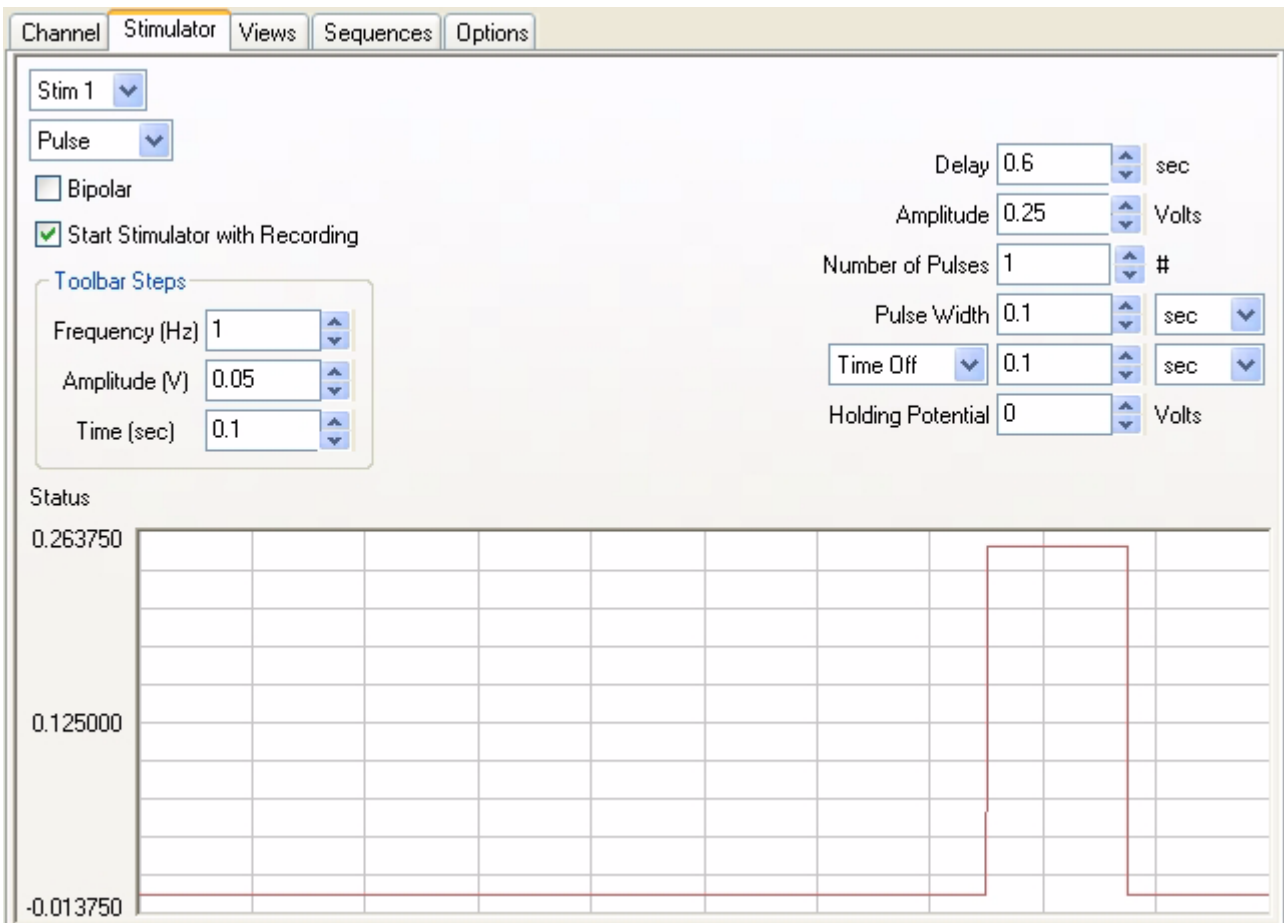
Stimulator Page

Figure 6-19: Stimulator page of the Preferences dialog window set for testing an Output Sequence.

On the **Stimulator** page, select **Pulses** as the stimulating mode and set the parameters.

- Toolbar Steps

  Freq (Hz) = 1; **Amp (V)** = 0.05; **Time (ms)** = 0.1.

- Initial Parameters

Delay	5 ms
Amplitude	0.25 V
Number of Pulses	1
Pulse Width	0.1 s
Time Off	0.1 s
Holding Voltage	0 V

Sequence Page
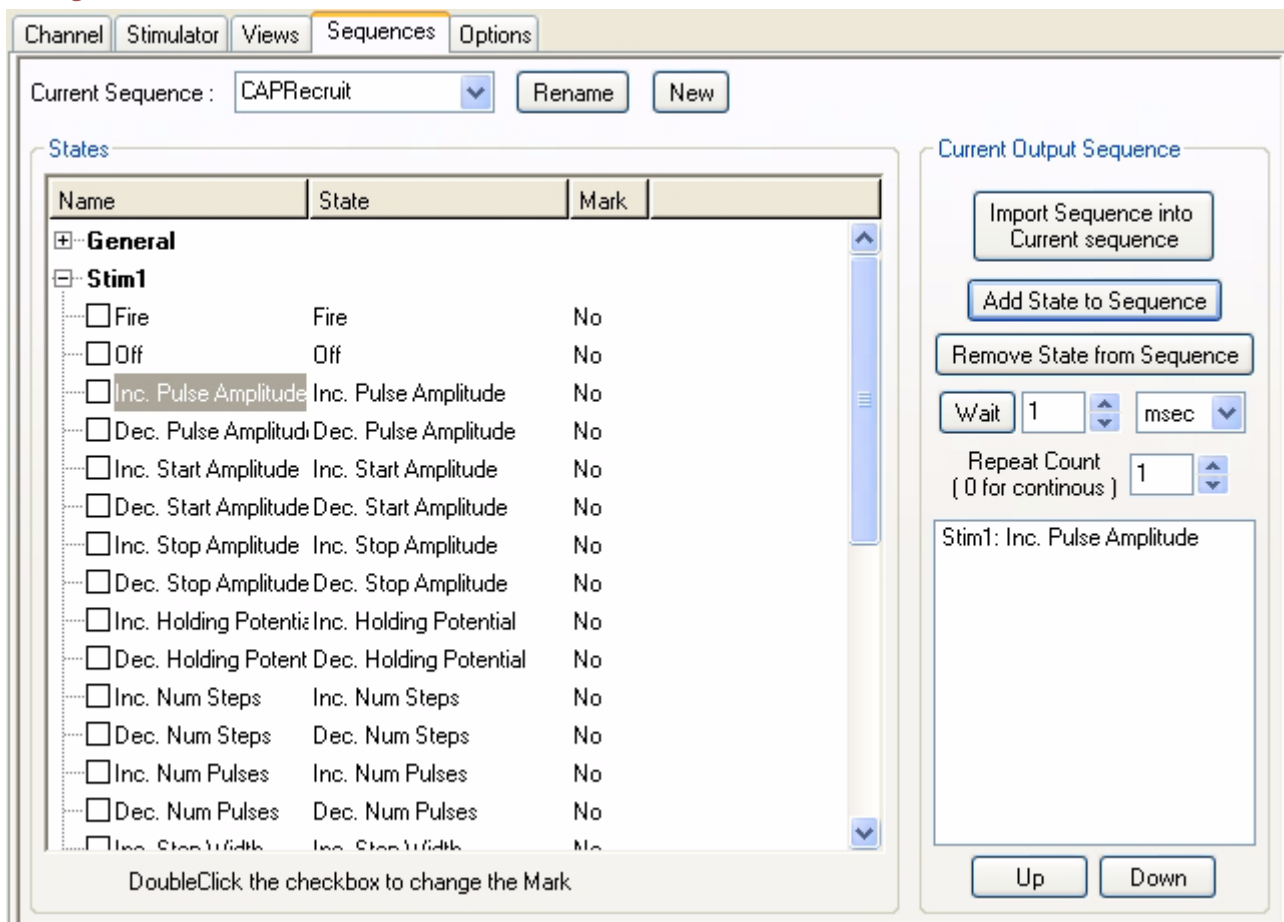
Figure 6-20: Output sequences page of the Preferences dialog window.

1 On the **Sequences** page, add a **CAPRecruit** sequence to the list of current sequences. Type the name CAPRecruit in the **Edit** box next to the **Current Sequence Label**, and click the **New** button.

2 Select the state titled **Inc Pulse Amplitude** under **Stim 1** from the list of states on the left side of the page.

3 Click on the button **Add State to Sequence** to add the state to the **CAPRecruit** sequence.

4 Click **OK** at the bottom of the page.

5 Return to the **Main** window; select the **CAPRecruit** sequence from **Sequences** control on the toolbar.

6 Click the **Record** button. The program records, displays and saves the ten progressively larger stimulus pulses on ten successive **Scope** sweeps.

7 Click on the **Analysis** icon in the toolbar to view the sweeps recorded for each stimulus amplitude.

Outputs States

States

Output states are used in the construction of sequences. There are six categories of output states:
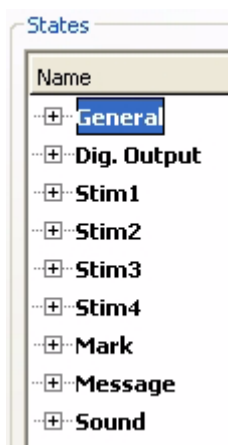
1 General: The general category includes *Start Recording, Stop Recording* and *Update*.

2 Dig. Output: Controls the digital outputs. For each digital output there is an on state and an off state. You can also turn all digital outputs on or off simultaneously.

3 Stim#: Controls the corresponding stimulator parameters. This can be used to change the stimulator parameters on the fly.

4 Mark: Set Marks in the record.

5 Message: Display User Messages on the screen.

6 Sound: Plays an existing audio file, located in the same folder as the current settings file.

The *WPI* 118 has up to eight **Digital Outputs** in two banks. BNC's 5 -8 can be configured as **Digital Inputs** or as **Digital Outputs**. Each digital output can be controlled independently.For each digital output there are two states, **On** and **Off**. So for eight digital outputs we have 16 states.

Naming States

Each output state can be named. To name an output state, click once to select the state, then click again to edit the name. Type the new name of the state in the **Edit** box that appears. If you want a mark to be placed on the record when the digital state is fired, double-click the checkbox, next to the label, so that a checkmark appears. For example, in the figure below, **DO2 On** has been labeled as **Pump On** and set to place a mark in the record when it is fired.
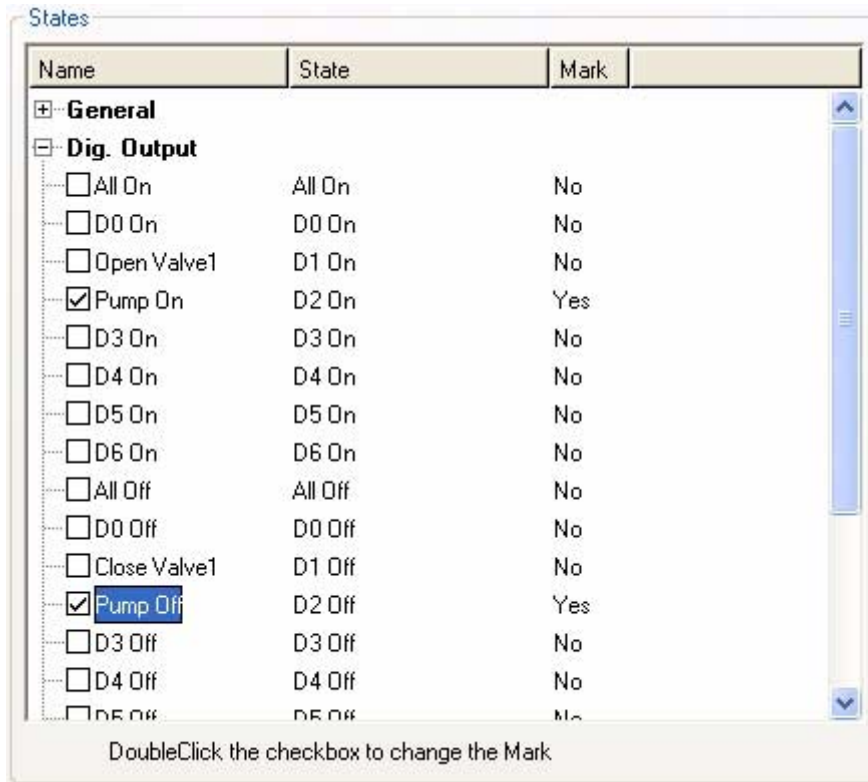
Figure 6-21: Output States

Sequences

Digital states can be grouped together in **Sequences**. A **Sequence** allows timed execution of digital states.
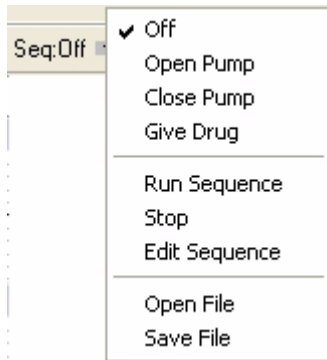
Creating Sequences

To create a **Sequence**, type the name of the sequence in the **Edit** box next to the **Current Sequence Label** and click **New**.

Editing a Sequence

Select the sequence to be edited in the **Current Sequence** drop-down box. To add a digital state to the sequence, select the digital state from the list on the left side of the window and click **Add State to Sequence.** To remove a state from the sequence, select the state in the sequence and click **Remove State from Sequence**. To have the *WPI* device wait for a specified period of time between different digital states, type a time (in seconds) in the **Edit** box next to the **Wait** button and click **Wait** to insert the waiting period into the sequence**.**

The timing for the digital inputs is handled by the computer and since Windows is not a real-time operating system, there is an inherent lag in the processing of events. The digital output timing should be within one second. To set more than one state at the same time, add all the states you want to set simultaneously to the sequence before setting Wait or Update (under General). The *DataTrax2* program will "OR" all the states and execute them when it reaches the Wait statement, the Update statement or the end of the sequence. To repeat the sequence, set the **repeat count**. The order of the states in the sequence can be changed by using the up and the down buttons.

Firing a Sequence

The **Sequence Control** on the **Main** toolbar is a menu containing sequences of analog or digital outputs defined in the **Sequence** page of the **Preferences** dialog window. Clicking the **Sequence** button will fire the output sequence selected from the list. Selecting a new sequence in the list also automatically fires that sequence.

Digital Input

The **Digital Input** function treats the data on the selected channel as a binary number.

The functions in the **Digital Input** submenu are:

Rate: The program takes the **period** in seconds and divides this value into 60. The result is a **rate**, which is expressed in beats per minute

Frequency: The program takes the **period** in seconds and divides this value into 1. The result is a **frequency**, which is expressed in Hz or cycles per second

Period: The program takes the **Period** for each cycle

Time On: The time that the selected channels are high

Time Off: The time that the selected channels are low

Duty Cycle: 100* (**Time On**)/**Period**

Count: Number of events

The channels of interest can be selected using the checkboxes. If you desire to complement the input before it is used then click on the checkbox again, as shown in the legend.

Reset After n sec: This option is only available for the Count subfunction, which is used to reset the count after n seconds

Any Bit is Set/ All Bits are set: Choose if you want an event to be detected if any channel is high or if you only want an event to be detected if all channels are high

Events

As *DataTrax2* acquires data, it is aware of the value of each data point as it happens. It should then be possible to "train" the software to watch for values above or below a specified level and have *DataTrax2* advise the user when such conditions are met. This feature in *DataTrax2* is called an **Event**. Events are set up in the events page of the **Preferences** menu.
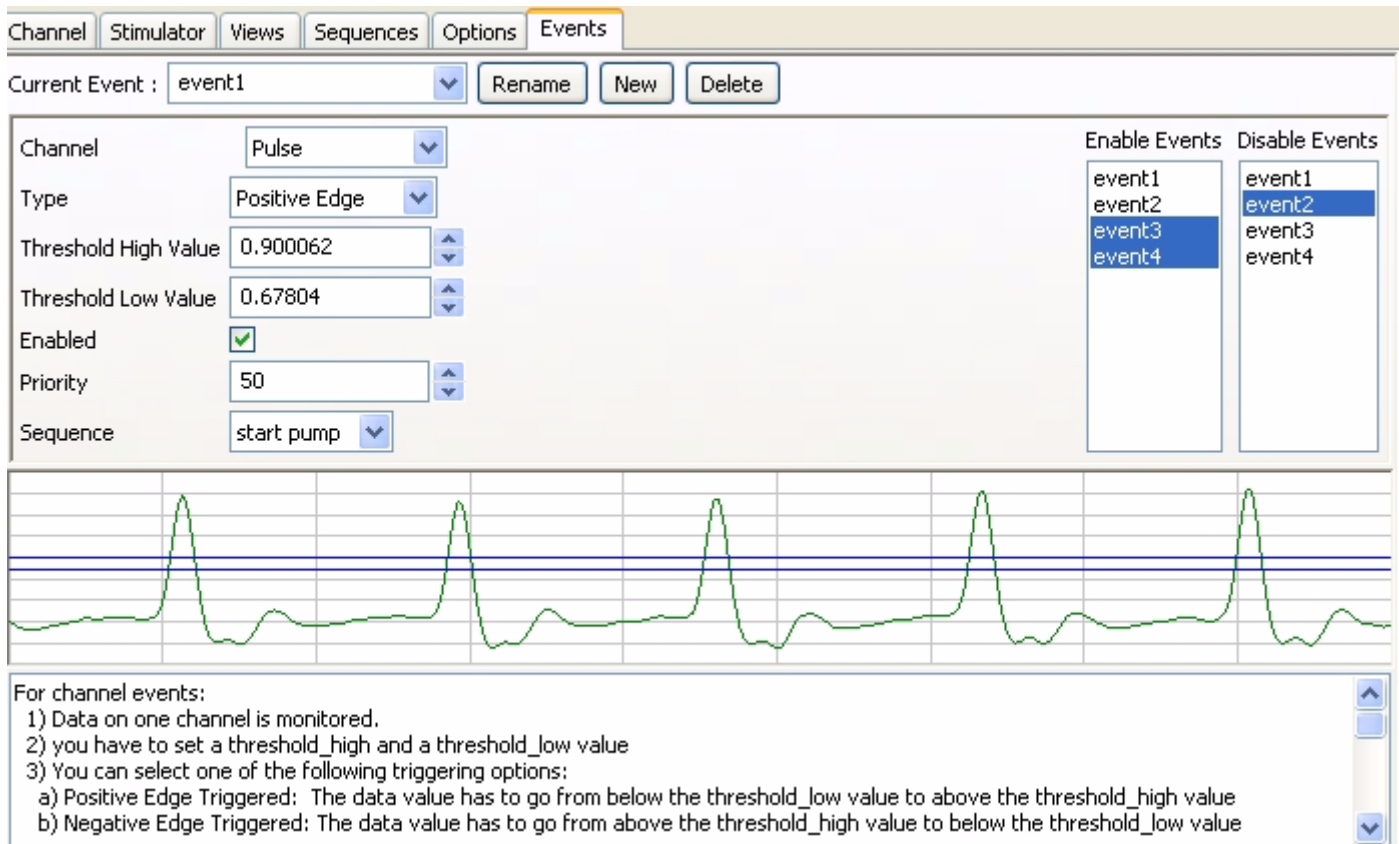

Figure 6-22: Controls for the Event programming

There are two types of events: **Channel** events and **Timed** Events.

Channel Events

1 Data on one channel is monitored.

2 You have to set a threshold_high and a threshold_low value. The threshold values can be set either by entering a value or by dragging the cursors in the graph.

3 You can select one of the following triggering options:

•**Positive Edge Triggered**: The data value has to go from below the threshold_low value to above the threshold_high value

•**Negative Edge Triggered**: The data value has to go from above the threshold_high value to below the threshold_low value

•**In window**: If the data value is between threshold_low and threshold_high value;

•**Out of Window**: if the data value is outside the window defined by threshold_low and threshold_high value

Timed Events   Timed events are triggered after a certain amount of time has passed. Timed events can be repeated, set N = 0 for continuos.

Event Priority   Events have priority. Sequences launched by events inherit the priority from that **Event**. A higher priority event can stop a lower priority sequence, but a lower priority event cannot stop a higher priority sequence.

User Events (i.e. sequences triggered by user) are also treated as events having a priority of 50. For example, Events 1 and 2 could have a higher priority than the user, thereby allowing them to interrupt a user initiated sequence, while Event 3 with a lower priority than the user, cannot interrupt a user initiated sequence.

An event can start in an enabled or disabled state. A disabled event is ignored. Events can enable or disable other events. The enabling or disabling function works independent of the sequences, so lower priority events can enable or disable events, even when there is a higher priority sequence running.

In the example shown Figure 6-22 on page 99, The **pulse** channel is being monitored for a positive threshold crossing from below 0.67804 to 0.900062. The event is enabled, with a priority of 50 and set to fire the sequence "Start Pump." This event will also enable "event3" and "event4", while disabling "event2."

Chapter 7: Using Settings

The Settings Menu

DataTrax2 offers users several choices for recording and displaying data. When certain choices are used repeatedly, a template known as a **settings file**, can be created to reduce the time required to program the recording software and the A/D converter.

To understand how to use **settings files**, two terms need to be defined: **settings group** and **settings file**. A **settings group** is actually a simple text document that can contain links to individual **settings files**. Each **settings file** is a collection of settings for performing an experiment and is programmed by users through the **Preferences** dialog window from the **Edit** menu. The settings contained in a file include items like: the number and titles of channels in the *DataTrax2* windows; the sampling speed; the units conversions; and the stimulator settings.

In addition to the hardware and software provided in its teaching kits, *WPI* provides laboratory manuals with a variety of experiments. To support these experiments, *WPI* has created a **settings group** for each laboratory manual. Each settings group contains links to **settings files** for each experiment. Settings groups are loaded on the computer during the installation of the *DataTrax2* recording software. Once a settings group is loaded, the settings files within the group can be called from a list in the **Settings** menu.

Calling a Settings File
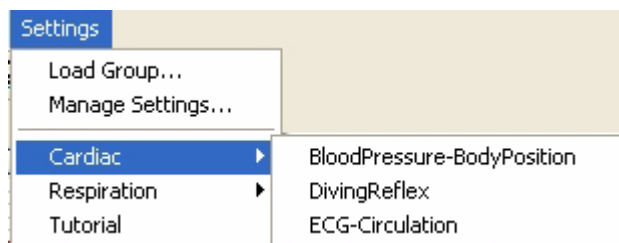
Before a **settings file** can be selected and activated, the **settings group** that contains the file must be loaded. To do this, select **Load Group** from the **Settings** menu. Choose the settings group file, that you wish to load. Settings groups associated with *WPI* manuals are saved in the *DataTrax2* folder when the *WPI* Laboratory manuals are installed.

Once a **settings group** is loaded, all of the **settings files** contained in the group appear by name in the lower bracket of the **Settings** menu. The settings files in the menu are organized by categories. To select a specific **settings file**, highlight the name of the file and click it. The settings associated with the selected file will load into the *DataTrax2* program and set parameters for recording and displaying data.

For a settings group associated with an *WPI* teaching kit, the names of the settings files on the list correspond to the names of the experiments in the lab manual.

If the *WPI* laboratory manuals are installed from the installation CD as prescribed, each settings file in the settings group for a teaching kit is associated with a **helper file**, which is a .pdf copy of the laboratory exercise as it appears in the *WPI* laboratory manual.

Users can design their own settings groups and files. They can also associate their own setting files with documentation (experimental procedures, protocols, instructions, lab exercises) of their own design.

Creating a New Settings File

The first step in creating a settings group is to create a settings file. Configure the *DataTrax2* program to do the experiment you want to do. For example, go to the **Channels** page in the **Preferences** dialog window and select the number of channels needed, their titles and any **Functions** that are needed to display or interpret the data.

You can also assign an aim and a difficulty level to each settings file by using the **Document Properties** command under the **File** Menu.

Click on the **File** menu and choose **Save As.** A **File Save** dialog will appear. In the **Save As Type** drop-down box at the bottom of the dialog, choose **Settings (*.iwxset)**. Using the file browser, choose a location on your computer where you want to save the settings file. We recommend that you create a "My Settings" folder in a location that you have write permissions to, such as the **My Documents** folder. Name the file and click **Save**.

Creating a New Settings Group

To create a new settings group. Choose the **Manage Settings** command from the **Settings** menu, which will launch the **Settings Manager**.
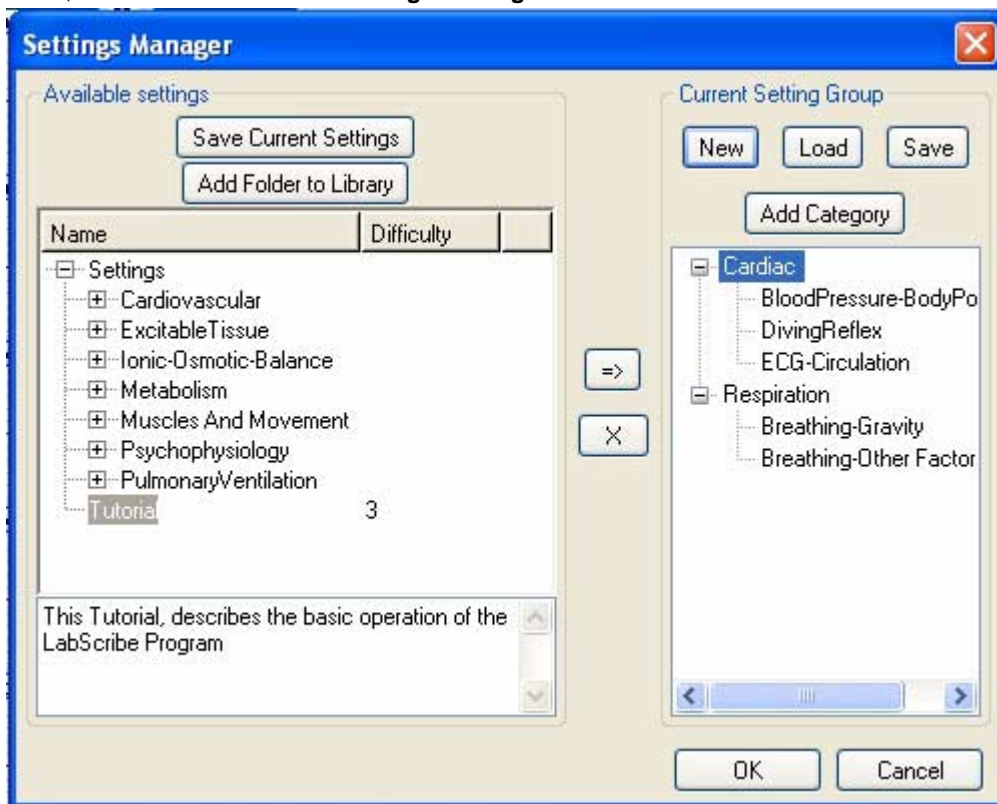
The available settings are listed in a tree in the left hand settings library window. By default *DataTrax2* will try to load the **Settings** folder in the *DataTrax2* folder. If you have created your own settings, you can add your settings folder to this settings library by clicking on the **Add Folder to Library** button and navigating to the desired folder location. Each setting can have a a difficulty level and an aim associated with it. The difficulty level for each setting is visible in the **Difficulty** column. If you select a settings file, the aim of the setting is visible under the settings library window.

To create a new group, click the **New** button in the **Current Settings Group** area. This will remove the current group and clear the **Group Display** window.

To load an existing group, click on the **Load** button in the **Current Settings Group** area. This will remove the current group and replace it with the loaded group in the **Group Display** window.

Categories   Settings files can be organized into categories within a group. To create a new category in the settings group, click on the **Add Category** button and name the category. You can add settings files from the list of available settings to any category or root category. Settings added to the root category are listed in the **Settings** menu below the other categories.

Helper Files   **Helper files** are documents with experimental instructions, diagrams and illustrations that can be linked to **settings files** and displayed on the computer screen when a settings file is selected. On the *DataTrax2* installation CD, *WPI* provides a laboratory manual and a **settings group** for each of its kits. In the installation process, a .pdf copy of each individual lab experiment in a manual is linked to the corresponding settings file in the settings group for that manual. When the user selects a settings file to do a particular experiment, a .pdf copy of the linked helper file (the experimental write-up) opens in <u>Acrobat</u> <u>Reader</u>. The user now has a set of instructions, on the computer screen, to follow as they do the experiment.

Users can associate their own helper files to their own settings files. Although, *WPI* has chosen to attach .pdf documents to its setting files, any file (HTML, Word Perfect, MS Word, etc.) can be linked to a settings file. To associate helper files with any settings file, place your helper files in the same location as the settings file and give the helper file the same name as the settings file name, but with the appropriate extension. For example, to associate a pdf, an html and a jpg file with the Tutorial.iwxset settings file, copy the pdf, html and jpg file to the same folder as the Tutorial.iwxset settings file. Name the pdf file Tutorial.pdf, the html file as Tutorial.html, and the jpg file as Tutorial.jpg. Now when the Tutorial settings file is chosen in *DataTrax2*, the Tutorial.pdf, the Tutorial.html and the Tutorial.jpg files will be opened.

Chapter 8: Export

Printing

Once you have recorded and analyzed your data, you'll probably want to create a report or other presentation. This can be done in a variety of ways, the easiest of which is to simply print the recorded data. The *DataTrax2* software uses a WYSIWYG (What You See Is What You Get) printing scheme. That is, the **Print** command will print the forward window exactly as it appears. This is particularly important for the **Journal** and the **Analysis** window as they require that the correct segments of data be displayed exactly as you want to print them before they can be printed.

Cutting, Copying and Pasting

DataTrax2 supports cutting and copying from all windows. Copying is done from any window using the **Copy** command in the **Edit** menu. Within the *DataTrax2* program, pasting is supported only in the **Journal**. To paste into the **Journal**, first copy data from either the **Main** or **Analysis** windows. Open the **Journal**, go to the **Edit** menu and select **Paste**.

Saving and Saving As

Once recorded, the *DataTrax2* program can save data in its own binary format using the **Save** or **Save As** command found in the **File** menu. These commands work the same as they do in every other program written for <u>Windows</u> or Apple computers. Selecting **Save As** will create a copy of the file on the disk with a new name. *DataTrax2* can save three types of documents:

1 The data file with the.iwxdata extension (*WPI* Document)

2 The **Journal** as an html file

3 The settings with the extension .iwxset (*WPI* Settings)

Journal

The **Journal** is actually very similar to <u>WordPad</u>. It can be exported to the disk in .htm format allowing it to be opened and edited in any program that supports that format. A formatted and edited .htm file may then be printed in whatever program you are using to read it.

In addition to images, calculated values from the **Analysis** window or the **ScopeView** window can be moved to the **Journal** by calling the **Add Data to Journal** command in the **Tools** menu. Prior to moving calculated values, it is recommended that the title line from the **Analysis** window be moved by using the **Add Titles to Journal** command in the **Tools** menu. Titles and data are tab delimited so they can be easily formatted in any word processing program.

By moving pictures and calculated values to the **Journal** and adding typed comments from the keyboard, an entire lab report can be created without ever leaving the *DataTrax2* program.
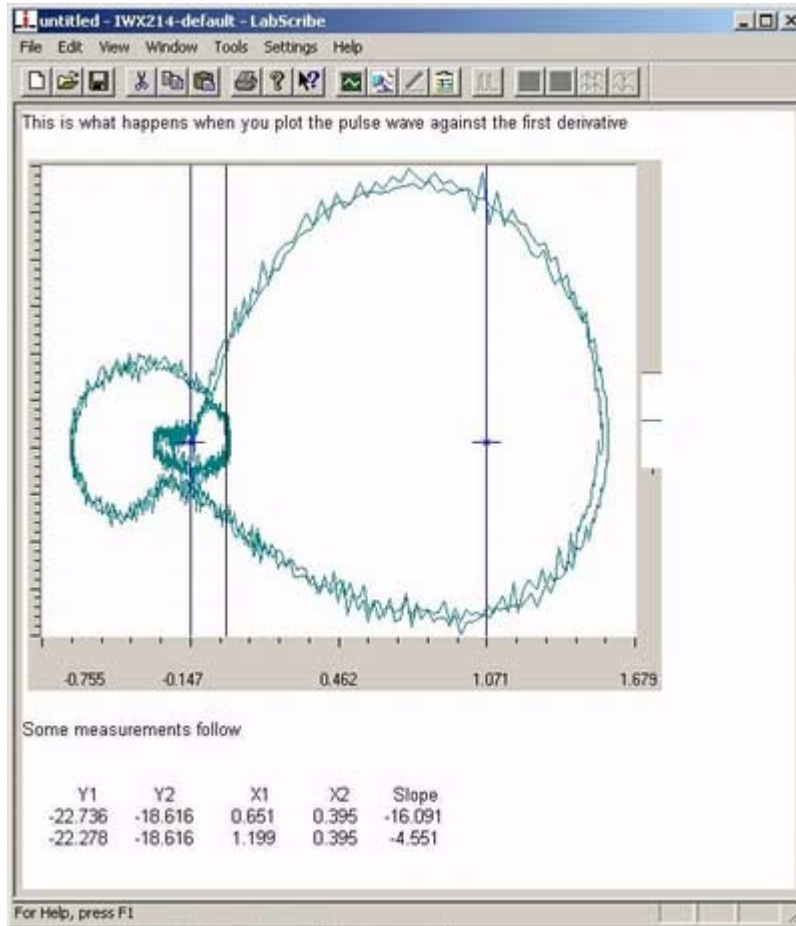
*Figure 8-1: Text and image as displayed in the **Journal** window.*

Exporting Data

Data can be exported from *DataTrax2* as either numbers or pictures from any window. To export data, select **Export** from the **File** menu and choose the file's format, location and name in the dialog window.

Exporting Pictures

The current display can be exported as a picture in bitmap (.bmp) format or in a portable network graphics (.png) format. **Portable Network Graphics** is a high resolution format for images.

Exporting Numerical Data

If data is exported from the **Main** window, the complete data file is exported in either Matlab or DADiSP formats.
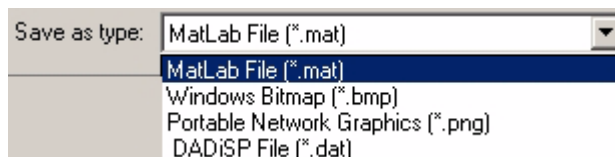
If data is exported from the **Analysis** or **ScopeView** windows, then only the data displayed in the window is exported.
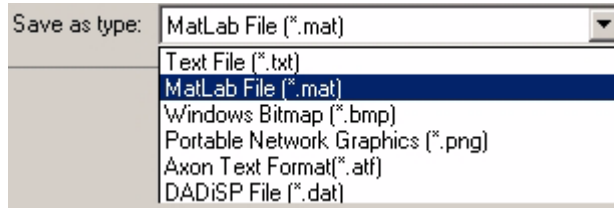
Data from the **Analysis** or **ScopeView** windows can be saved in Text (.txt), Matlab (.mat), Axon Text Format (.atf), or DADiSP (.dat) format.