LabScribe2
User's Manual

Contents

Chapter 2: The Menus

Chapter 3: Acquisition

Chapter 4: Creating Your Own Preferences and Settings

Chapter 5: The Stimulator

Chapter 6: Computed Channels

Chapter 7: Analysis

Chapter 8: Advanced Analysis

Chapter 9: Digital Input and Output

Chapter 10: The Journal and Data Export

Introduction

Welcome

Thank you for choosing *iWorx LabScribe2* Data Recording and Analysis Software. 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 (877) 273-7110 (Research products) or (800) 549-5748 (Teaching products). International customers should call (603) 742-2492.

Contact us via email at either: research@iworx.com or teaching@iworx.com.

How to Use This User's Manual

This User's Manual assumes you are familiar with basic Windows and Macintosh OSX 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.

Although it contains some practice exercises, this manual is not designed to be a tutorial, or an introduction to *LabScribe2*. It is a reference manual covering all of the features and capabilities of the software. Refer to the **Quick Start** guide for a practical overview of *LabScribe2's* most commonly used features. A tutorial containing a number of exercises is included in the electronic laboratory manual that came with your software, and there are a number of video tutorials available in the User Area at iworx.com.

System Requirements

LabScribe2 recording software requires a Windows (Windows 98 through Windows 7) or a Macintosh (OSX 10.3 through OSX 10.6) computer.

If you are using a Windows computer, a Pentium 4 processor computer with at least 2 gigabytes of RAM and 500 megabytes of free space on your hard drive is required. A Windows computer with a Dual Core Processor with an AGP graphics card and 2 gigabytes of RAM is preferred.

If you are using a Macintosh, a G4 processor with at least 2 gigabytes of RAM and 500 megabytes of free space on your hard drive is required. An Intel Dual Core Processor and dedicated graphics card are preferred.

Installation

Installation From CD

LabScribe2 software is provided on a hybrid CD-ROM, with software for both Windows and Macintosh Operating Systems. If you don't have a CD drive, the software installer can be downloaded from our User Area. Point your browser to http://www.iworx.com and enter the User Area (register first if necessary). All installers are available in the Software section of the User Area.

*Note: You will need to have administrative privileges to install LabScribe2 on some computers. Contact your IT department for assistance or to confirm your administrator status. Additionally, Windows users will need permissions to write to **C:\Users\USERNAME\AppData\Local\LabScribe2**. Do not connect your iWorx hardware to the computer until **AFTER** the software installation is complete.*

Windows

To install the software using the CD-ROM on a Windows computer:

- Insert the *LabScribe2* installation CD.
- Double-click on the Setup icon.
- Follow the prompts in the Setup Wizard to complete installation.
- Once you have completed the installation process, remove the installation CD from the drive, then connect and turn on your *iWorx* data acquisition unit.

Note: *When your hardware is connected for the first time, Windows will advise you "New Hardware found" and proceed to load the driver automatically. If for some reason Windows cannot locate the driver, locate the appropriate drivers for your operating system in the iWorx/LabScribe2/**drivers** folder in the **Program Files** folder on your C drive.*

Macintosh OSX

To install the software using the CD-ROM on Macintosh OSX 10.3 or later:

- Insert the *LabScribe2* installation CD.
- Copy the files from the CD to the Applications folder of your computer.
- Remove the installation CD from the drive, then connect and turn on your *iWorx* hardware.

Installation Download From User Area

To install using a downloaded installer from our User Area:

- Go the User Area on the iworx.com website. Click on the Software link and select the proper installer for the Windows or Macintosh operating system on your computer. After downloading the *LabScribe2* software installer, double click on the downloaded file. It is a self extracting archive, which will automatically launch the installer after it extracts.
- In Windows, follow the prompts to complete installation. Restart your computer once installation is complete.
- In Macintosh OSX, copy the files to your Applications folder.
- Once you have completed the installation process, connect and turn on your *iWorx* hardware.

iWorx.com User Area

The User Area at iWorx.com contains a wealth of resources, including software files, experiments, archived newsletters, hardware documentation, and a complete online catalog of our research and teaching products. You will need to register and choose a password to access the User Area. Simply click **Register** on the User Area home page for complete registration information. By registering with us at iworx.com, you will be notified of updates and new releases, and you will be able to access the free software upgrades you are entitled to for as long as you use *LabScribe2*. *iWorx Systems* recommends using Firefox as your Web browser when accessing the iworx.com website.

Comments and Suggestions

iWorx Systems understands that user feedback is critical to the improvement of any software. If you have ideas, suggestions or criticisms, please contact us at either: research@iworx.com (Research products) or teaching@iworx.com (Teaching products).

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 phone or email. 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.
- Save a data file that can be emailed to an *iWorx* Technical Support representative.
- Note your computer model and operating system version.
- Note the amount of RAM (Random Access Memory) in your computer.
- Note your *LabScribe2* Software version number.

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

Contacting Us

iWorx Systems, Inc.
One Washington Street, Suite 404
Dover, NH 03820 USA

For Sales

(800) 234-1757 (North America)
(603) 742-2492 (outside North America)
FAX: 603-742-2455
Email: sales@iworx.com
Web: http://www.iworx.com

For Technical Support

Research products:
(877) 273-7110
research@iworx.com

Teaching products:
(800) 549-5748
teaching@iworx.com

International customers should call (603) 742-2492.

Chapter 1: The Display

Overview

LabScribe2 both acquires data and uses an intuitive display that makes it possible for the user to view, interpret, and manipulate the recorded data. This chapter discusses how data are displayed in the various windows of the *LabScribe2* user interface.

User Interface

The *LabScribe2* user interface contains five primary display windows: the **Main** and **Analysis Windows**, the **XY** and **FFT Views**, and the **Journal**. There are also dialog windows and control panels, accessible through **Toolbar** icons and the software **Preferences Dialog** (accessed in the **Edit** menu in Windows, and the **LabScribe2** menu on the Macintosh), which displays the **Channel**, **Stimulator**, **Views**, **Sequences**, **Options** and **Events** configurations. The **Marks** icon in the **Toolbar** opens the **Marks Dialog** and the **Stimulator** icon opens the **Stimulator Control Panel** directly beneath the **Toolbar**. The **Main Window** and **Toolbar** contain most of the controls necessary for data acquisition.

The Main Window

Most of *LabScribe2*'s interface features can be found in the **Main Window**. *LabScribe2* can display up to 128 channels of raw and computed data in the **Main Window**.
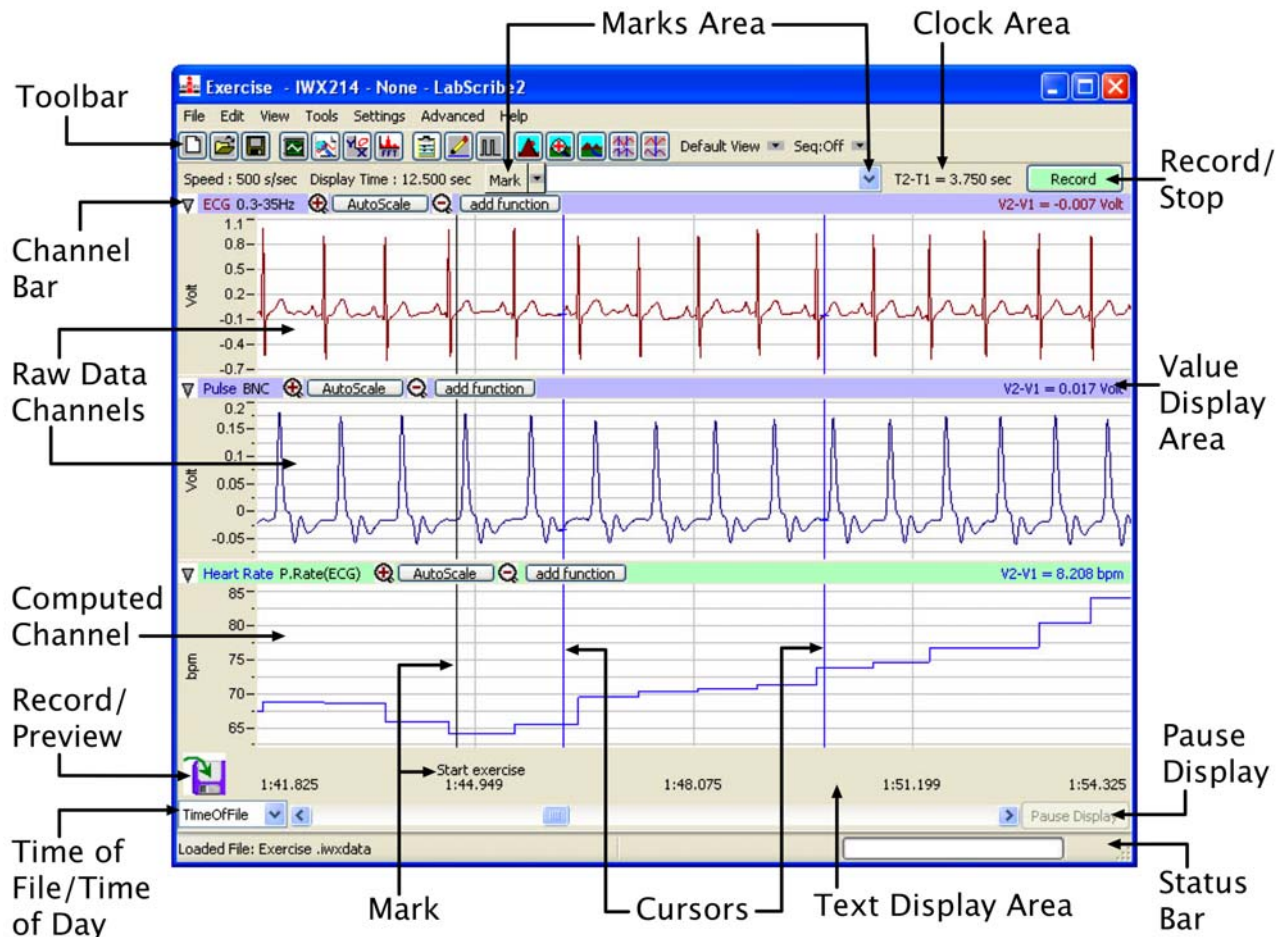
The LabScribe2 Main Window.

The Toolbar   Many of the functions of *LabScribe2* can be accessed via the icons in the **Toolbar**, which is directly beneath the menu headings on the **Main Window** display. These functions are described in more detail in the relevant parts of this manual.
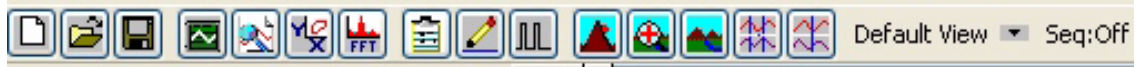
The Main Window Toolbar.

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: Opens the **XY View**.

FFT (Spectrum): Opens the **FFT WIndow**.

Journal: Opens the **Journal** on the right side of the **Main**, **Analysis**, **XY**, and **FFT Windows**.

Marks: Opens the **Marks Dialog**.

Stimulator Preferences: Hides or displays the **Stimulator Control Panel** directly beneath the **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 between the two cursors in **Two Cursor Mode**.

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

Two Cursor Mode: Displays two cursors on the data window. Time and voltage differences between the data points intersected by the cursors are measured and displayed in the **Clock** and **Value Display Areas**. **Two Cursor Mode** is the default condition, and is the only option available in the **Analysis Window**.

Single Cursor Mode: Displays one cursor on the data window. The absolute voltage at the cursor is displayed in the **Value Display Area**. The time from the beginning of the trace to the cursor is displayed in the **Clock Area**.
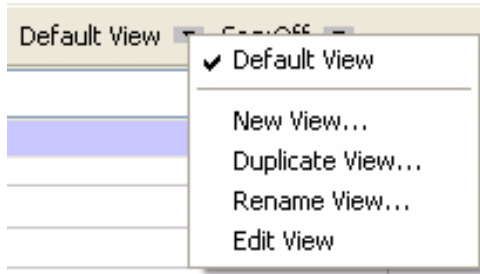
Views Toolbar button and drop-down menu.

Views: Clicking on the arrow next to the **View** name displays the **View** control menu. You can switch the current view from this menu, create a new view from the default view, and duplicate or rename the current view. Selecting **Edit View** opens the **Views** page of the **Preferences Dialog**, where more options are available. A more complete description of **Views** and how to create them can be found starting on page 11 in this chapter.
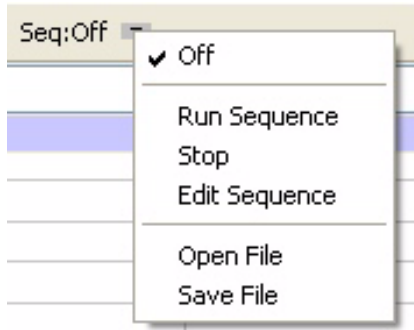
Sequence Toolbar button and drop-down menu.

Sequences: The **Sequence** menu is a drop-down list containing sequences of analog or digital outputs defined in the **Sequences** page of the **Preferences Dialog**. Clicking the **Sequence** button will fire the output sequence currently selected. Selecting a new sequence in the drop-down list also automatically fires that sequence. See page 59 of **Chapter 5: The Stimulator** and page 133 of **Chapter 9: Digital Input and Output** for more complete discussions of **Sequences**.

Channel Bar   Each channel has its own set of controls located in a **Channel Bar** immediately above the channel's data window. The **Channel Bar** will appear slightly different depending on which window is currently being displayed.
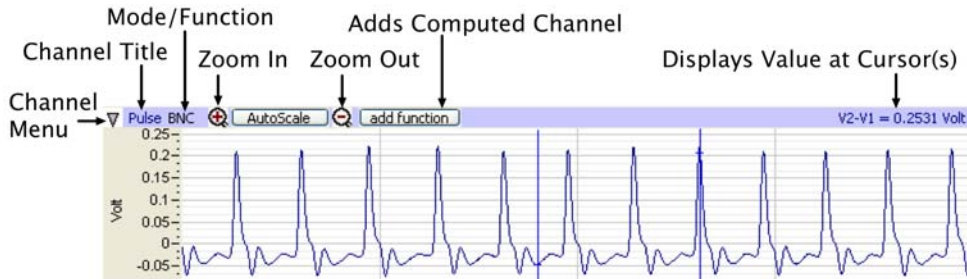
The Main Window Channel Bar.

The **Channel Menu** contains functions specific to the channel, and is accessed by the arrow on the left of the **Channel Bar**. The **Channel Menu** can also be displayed by right-clicking anywhere in the channel. The menu items include:
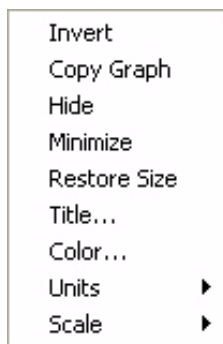
The Main Window Channel Menu.

- **Invert**: Inverts the trace.

- **Copy graph**: Copies the graphical channel data in the current screen and sends it to the clipboard. It can then be pasted into the **Journal** or an external application.

- **Hide**: Hides the channel.

- **Minimize/Restore Size**: Reduces the size of the channel to just the **Channel Bar** or restores the full channel.

- **Delete**: Permanently deletes the channel.

- **Title...**: Opens a dialog that allows the user to change the name of the channel.

- **Color**: Opens a color pallette that allows the user to change the color of the channel's trace.

- **Units**: Choosing this menu item displays the **Units Conversion** options. Clicking in the **Value Display Area** will also display these options. Refer to the **Units Conversion** section beginning on page 13 of this chapter for a complete description of these options.

- **Scale**: Clicking this item displays the options for controlling the vertical scale of the channel. Refer to the discussion beginning on page 30 in **Chapter 3: Acquisition** for a complete description of these options.

When the **Channel Menu** is opened by right-clicking in the channel display area, an additional menu item is added below the line at the bottom of the menu:

- **Normalization**: Choosing **Normalization** opens a submenu allowing the user to access the **Normalization** functions used to calibrate vessel size for the use of wire myographs. **Normalization** is a separately licensed software routine. **Normalization** is discussed in more detail beginning on page 129 in **Chapter 8: Advanced Analysis Routines**.

Refer to the appropriate sections of the manual for descriptions of the **Channel Menu** in the **Analysis Window** (page 83**)**, the **XY View** (page 89), and the **FFT Window (**page 92).

The **Channel Title** is displayed to the right of the **Channel Menu** arrow. The title can be changed with the **Title..**. option in the **Channel Menu**.

The **Mode/Function** position displays which input and hardware filters are being used on a raw data channel. Clicking on **Mode/Function** will allow you to change these options. On a computed function channel, **Mode/Function** displays the computed function. Clicking on **Mode/Function** in a computed channel will display the list of available computed functions, allowing the user to change the computed function of that channel. Refer to **Chapter 6: Computed Channels** for a more complete discussion.

Zoom In, **Autoscale**, and **Zoom Out** set the vertical scale and are described more fully beginning on page 30 of the **Managing Amplitude Display** section in **Chapter 3: Acquisition**.

Clicking **add function** adds a computed channel to the display. The functions are discussed in **Chapter 6**: **Computed Channels**.

The **Value Display Area** 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 Area** shows the value of the last data point collected.

- Offline, in **Single Cursor Mode**, the **Value Display Area** displays the Y-axis value of the data point intersected by the cursor.

- In **Two Cursor Mode**, the **Value Display Area** displays the difference between the Y-axis values intersected by the two cursors.

Channel Sizing   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, position the cursor at the top of the **Channel Bar** until it becomes a double-headed arrow.

- Click and drag this arrow up or down to resize the channel, as illustrated in the figure on the next page.
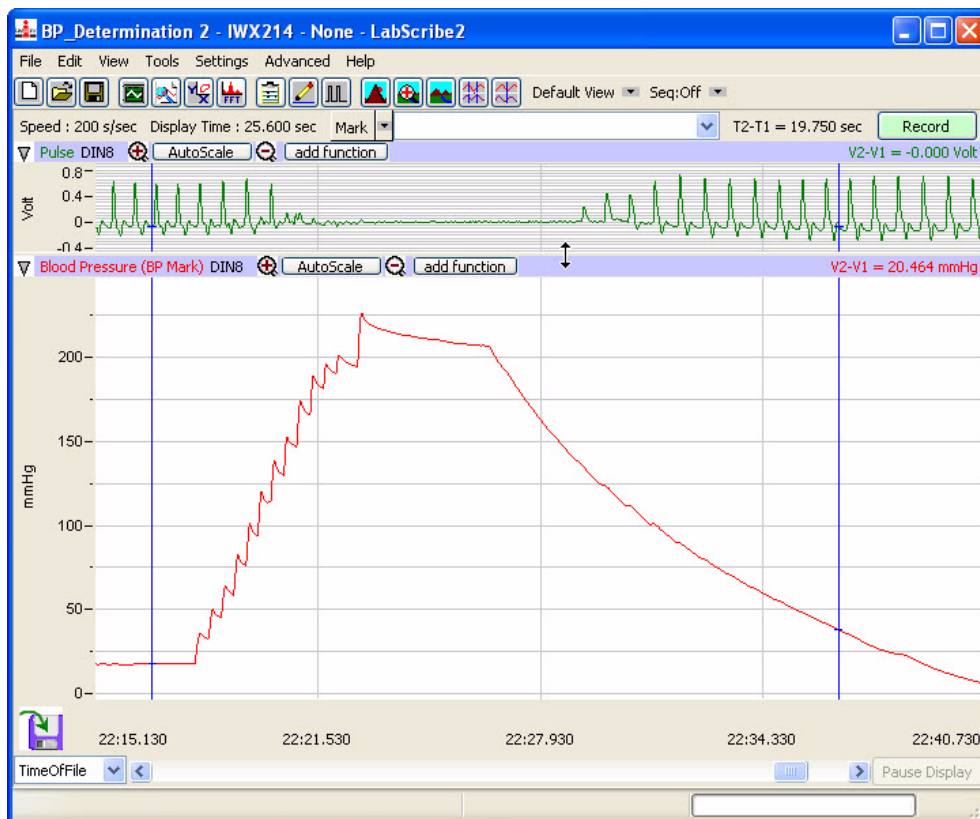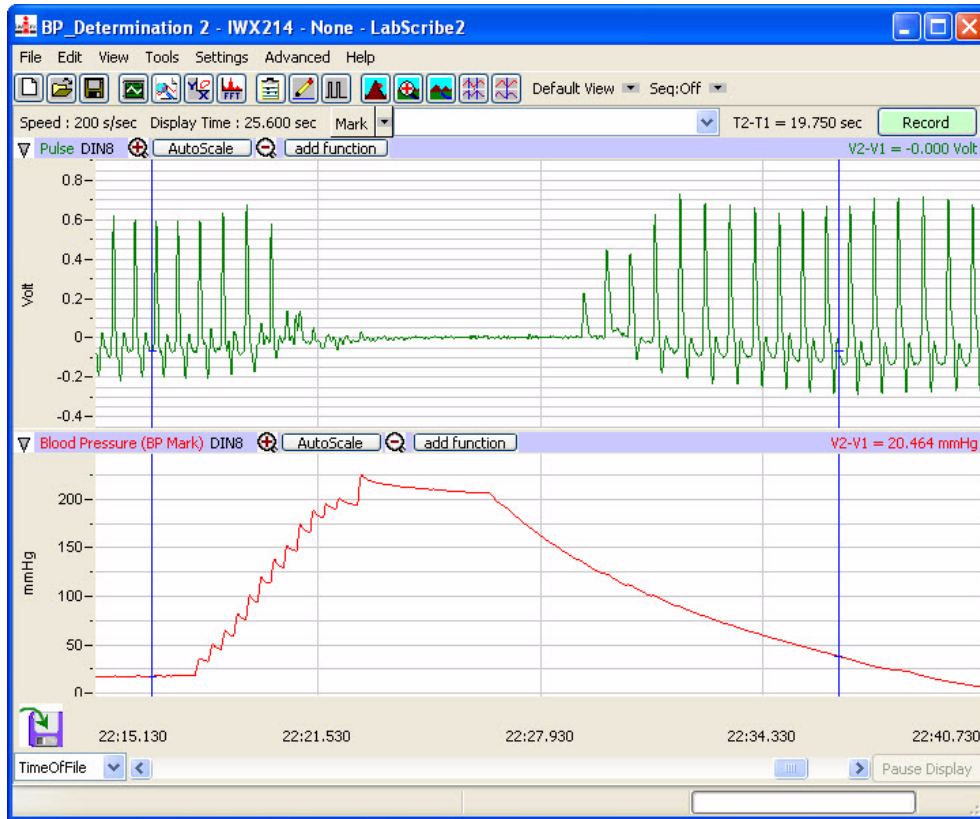
The height of the data displays for two channels in the Main Window as set by the default setting (upper figure), and after the handle for Channel 2 was moved up (lower figure).

Cursors

Many of the features of *LabScribe2* allow the user to label, measure, compare, and analyze specific data points or regions. *LabScribe2* uses **Cursors** to identify and isolate the data points of interest.

Cursor Modes   **Cursors** are the vertical blue lines that pass through all channels. Icons in the **Toolbar** allow you to choose between using **Single Cursor** or **Two Cursor Modes**.
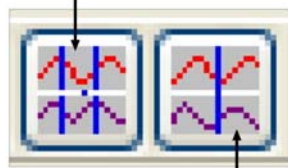
Cursor controls on the Main Window Toolbar.

Single Cursor Mode

- To access **Single Cursor Mode**, click on the **Single Cursor Mode** icon in the **Toolbar**.

- The **Value Display Area** on the right side of each channel bar displays the voltage of the data point that is intersected by the cursor in that channel. The **Clock Area** in the upper right hand corner of the **Main Window** displays the corresponding time.

- **Single Cursor Mode** is used to determine values and to place marks in the record after recording has stopped.

Two Cursor Mode

- To access **Two Cursor Mode**, click on the **Two Cursor Mode** icon in the **Toolbar**.

- When using **Two Cursor Mode**, the cursor on the left is always **Cursor 1** and the one to the right is always **Cursor 2**. If **Cursor 2** is moved to the left of **Cursor 1**, it becomes **Cursor 1**.

- The **Value Display Area** displays the difference in voltage between the data points at **Cursor 1** and **Cursor 2**. The value shown is always the amplitude at **Cursor 2** minus the amplitude at **Cursor 1**, and can therefore be a negative number.

- The **Clock Area** reports the difference in time between the two cursors.

- In **Two Cursor Mode**, the cursors can also be used to define the left and right boundaries of a selection of data.

Moving Cursors   A cursor may be moved by placing the mouse over the cursor, clicking, and dragging it to the right or left.

Cursors may also be moved using the arrow keys on the keyboard.

- Pressing the RIGHT or LEFT arrow key on the keyboard moves the cursor one data point.

- In **Two Cursor Mode**, pressing the keyboard's UP arrow changes the cursor that is moved.

- In the **Analysis Window**, holding the SHIFT key down while pressing the RIGHT or LEFT arrow causes the cursor to move five data points at a time.

- In the **Analysis Window**, holding the CONTROL key down while pressing the RIGHT or LEFT arrow moves the cursor ten points at a time.

Locking Cursor Separation

The cursors may be locked a set duration apart allowing you to look at a consistent amount of data between them. To lock the separation distance, positon the two cursors at the desired separation and choose **Lock Cursor Separation** in the **Tools** menu. To unlock the cursors, click **Lock Cursor Separation** a second time.

CURSOR EXERCISE

1. Record some data using the pulse transducer, as demonstrated in the **Tutorial** exercise that came with your software.

2. Select **Single Cursor Mode** by pressing the **Single Cursor Mode** icon in the **Toolbar**.

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

4. 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 so that the value in the **Value Display Area** reads the maximum value.

5. Enter **Two Cursor Mode** by clicking the **Two Cursor Mode** icon in the **Toolbar**.

6. Position **Cursor 1** so that it is over the minimum value in a given cycle, then position **Cursor 2** over the maximum value of the beat to the right. The value reported will be positive and represents the amplitude of that pulse beat.

7. Now, drag **Cursor 2** to the maximum value of the beat to the left of **Cursor 1**. When you release **Cursor 2**, it becomes **Cursor 1** and the new value reported will be of a similar magnitude, but will be a negative number.

Marks

LabScribe2 can record large amounts of data, so specific data of interest must be easily located and retrieved to be useful. To locate and identify specific sections of data, it is possible to put marks on the data while *LabScribe2* is recording. Marks can also be inserted and edited after the recording has stopped.

Making Marks Online

Marks can be placed on the recording without interrupting data recording.

• As soon as the **Record** button is clicked and data recording begins, *LabScribe2* sets an active text cursor in the **Marks** text box, to the right of the **Mark** button on the **Main Window**. In the **Marks** text box, the user can type a comment describing an upcoming step in the experiment.

• The **Mark** is placed on the recording when the ENTER (or RETURN) key on the keyboard is pressed, or the **Mark** button 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 into the **Marks** text box prior to the event can be seen in the **Text Display Area** at the bottom of the screen.
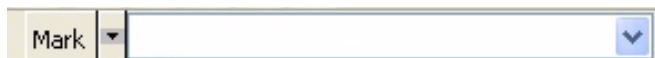
The Mark button and text box.

Preset Marks

If you know the marks that you will be adding to the record prior to recording, you can store the marks text as preset marks.

To create a preset mark:

• Type the mark text in the **Marks** text box, and choose the **Add to Mark Presets** option in the menu that opens by clicking on the arrow to the right of the **Mark** button. This will save the typed text as a preset mark**.**
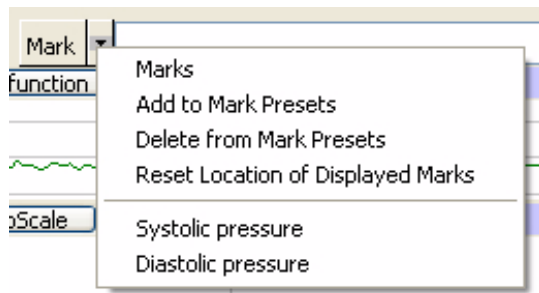
The Mark button and drop-down menu.

- The preset mark can be called by clicking on the arrow to the right of the **Marks** text box and choosing the desired preset mark. This loads the **Marks** text box with the desired **Mark** text. Clicking the **Mark** button or striking ENTER (or RETURN) will add the preset mark to the record.
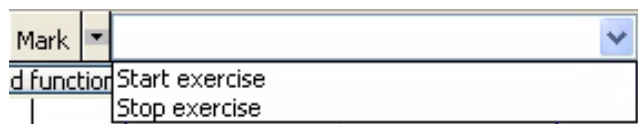
The Preset Marks drop-down menu

- You can delete a preset mark by selecting a preset mark (using the arrow to the right of the **Marks** text box) and choosing the **Delete from Mark Presets** option in the drop-down menu next to the **Mark** button.

Preset marks can also be entered directly into the **Marks Dialog**:

- Click on the **Marks** icon in the **Toolbar** to open the **Marks Dialog**. The **Marks Dialog** can also be opened by choosing **Marks** in the **Marks** sub-menu of the **View** menu.

- Select the **Preset Marks** tab.

- Enter the comments to be associated with the preset marks into the text boxes. The row sizes in the **Marks Dialog** can be adjusted by placing the computer cursor over a dividing line until it changes to a double-headed arrow. Click and drag the line to its desired location.

- Each preset mark is associated with a keyboard function key (or ALT plus a number). Once recording has begun, striking the appropriate function key (or ALT plus a number) enters the associated preset mark into the **Marks** text box. Clicking the **Mark** button or pressing ENTER (or RETURN) on the keyboard will attach the mark to the record.
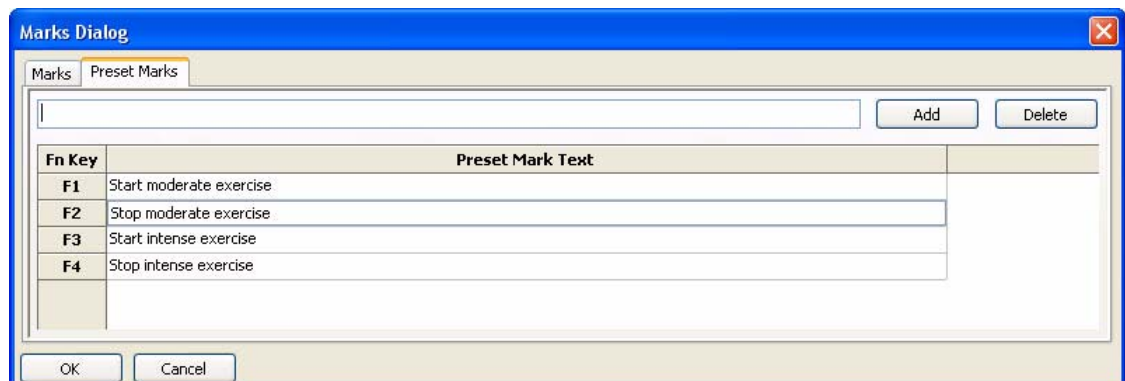
Preset Marks options in the Marks Dialog.

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 that could be added later would be a change in room temperature.

To add this information to the record after recording is completed:

- Click on the **Single Cursor Mode** icon in the **Toolbar**.

- Position the cursor on the record where the mark is to be positioned.

- Type the text (a maximum of 50 characters) associated with the new mark in the **Marks** text box. Click the **Mark** button or press the keyboard's ENTER (or RETURN) key, and the mark and its text comment are inserted at the current position of the cursor.

- You can also add a mark to the record by selecting **Add Mark** from the **Marks** sub-menu of the **View** menu. Type your text into the window that opens and click **OK**. The mark will be added to the record at the current position of the cursor.

Editing Marks

Marks already on the record can be changed or deleted.

To edit a mark:

- Access the **Marks Dialog** from the **View** menu or by clicking the **Marks** icon on the **Toolbar.** The **Marks Dialog** lists all marks in a file.

- Both the column and row sizes in the **Marks Dialog** can be adjusted by placing the cursor over a dividing line until it changes to a double-headed arrow. Click and drag the line to its desired location.

- Highlight the mark by clicking twice on the mark text, and make the desired changes.

- Click **OK** to exit the dialog.

To delete a mark:

- In the **Marks Dialog**, select the mark you'd like to delete by clicking on the number to the left of the row.

- Click on **Delete**.

- Click **OK** to exit the dialog.
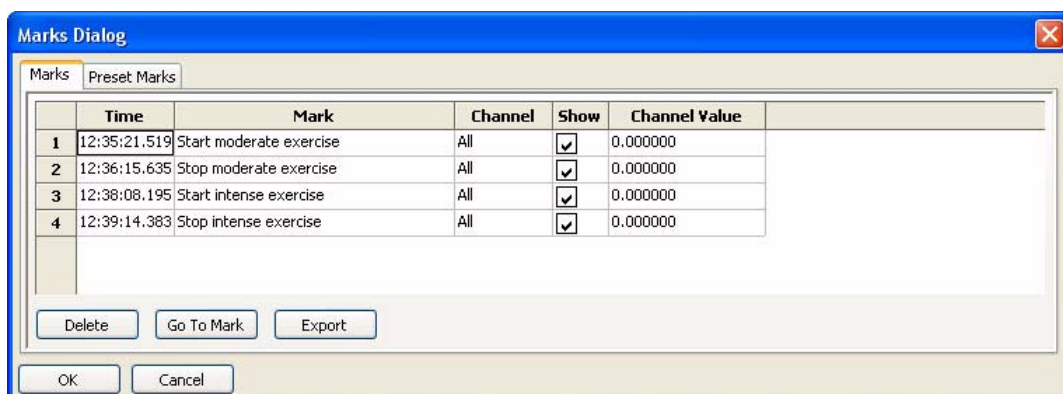
The Marks Dialog.

Navigating By Marks

Marks that are placed on the recording can serve as "sign posts", indicating where important sections of data are located. You can use the marks to navigate between important areas of an experiment without hunting or scrolling for the areas of interest.

To navigate by marks:

- Click on the arrow next to the **Mark** button and choose the mark you want to "go to". *LabScribe2* will find the data point associated with that mark and display that section of data in the **Main Window**.

- Alternatively, select the mark in the **Marks Dialog** and click on **Go To Mark**.

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.

To position marks on the trace:

- After recording has stopped, the mark text associated with the mark immediately preceding the displayed data appears in the **Text Display Area** at the bottom of the **Main Window.** Click on the mark text in the **Text Display Area** and drag it up into the data area of the record, allowing the mark to be read and printed in any window in which the data appear.

- 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 arrow next to the **Mark** button and choose **Reset Location of Displayed Marks**. This command returns only the marks on the data window of the visible screen of data to the bottom of the **Main Window**. All other marks will remain where they were originally positioned.

- You can also reset the location of marks from the **Marks** sub-menu of the **View** menu.

Comments moved to the channel data display in the Main Window.

Sorting and Exporting Marks

The **Marks Dialog** 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.

To export marks to a text file or spreadsheet program, select the mark in the **Marks Dialog** and click the **Export** button.

MARKS EXERCISE

1. Click **Record** and record a few minutes of pulse transducer data as demonstrated in the **Tutorial** exercise that came with your software.

2. As data are being recorded, type "Test 1" on the keyboard and strike the ENTER (or RETURN) key on the keyboard.

3. Wait one minute, type "Test 2" and strike the ENTER (or RETURN) key.

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

5. Click on the arrow next to the **Mark** button and choose "Test 2". Notice that the data 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 comment to a new position on one of the available channels.

7. Release the mouse button and the mark text 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 **Single Cursor Mode** by clicking the **Single Cursor** icon in the **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 ENTER (or RETURN) key. The mark appears at the cursor location and the comment appears at the bottom of the screen.

Views

The number of channels of raw data that *LabScribe2* can acquire and display is limited to the number of inputs on the hardware in use, 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 four different animals could be recorded on Channels 1-4. Simultaneously, the **Rate** function could be used on Channels 5-8 to calculate the heart rate of each animal from its recorded blood pressure on Channels 1-4. Displaying more channels means there is less space on the display for each one. In the case of a 16-channel **Main Window** display, it is hard to resolve detail in the recorded data on each channel.

LabScribe2 solves this problem by allowing the user to create many different arrangements of channels that are 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 only the arterial pressure from Channel 1 and the calculated heart rate for the same animal displayed on Channel 5. The data from Channels 1 and 5 would appear in the first and second data display areas, respectively. The other six 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 6 (its matching rate channel) can be created.

Creating and Editing Views

To create a new view:

- Make any desired changes to the **Main Window** by resizing the channels, or by hiding or minimizing a channel from the **Channel Menu** in the channel's **Channel Bar**.

- Click on the arrow next to **Default View** in the **Toolbar** and choose **New View**.

- Name the view in the **Edit View Name** dialog.

- Views can also be copied, renamed or deleted from the same menu.

It is also possible to create and edit a view from the **Views Preferences Dialog**. This dialog can be accessed by clicking on the down arrow next to **Default View** in the **Toolbar** and choosing **Edit View**. To use the **Views Preferences Dialog** to create and edit a view**,** or edit an existing view:

- Click on **New View...**, or select an existing view from the drop-down menu.

- Use the controls in the dialog to change the amount of time displayed, add or remove the channels available in the view, and change the vertical organization of the channels.

- Each view can display a **Voltmeter** (see page 16 for information about the **Voltmeter**) and you can choose which channels are displayed in the **Voltmeter** and their positions.

- Any of the views can be renamed.

- The **Views Preferences Dialog** can also be accessed by choosing **Preferences** in the **Edit** menu in Windows or in the **LabScrbe2** menu on the Macintosh and clicking the **Views** tab.
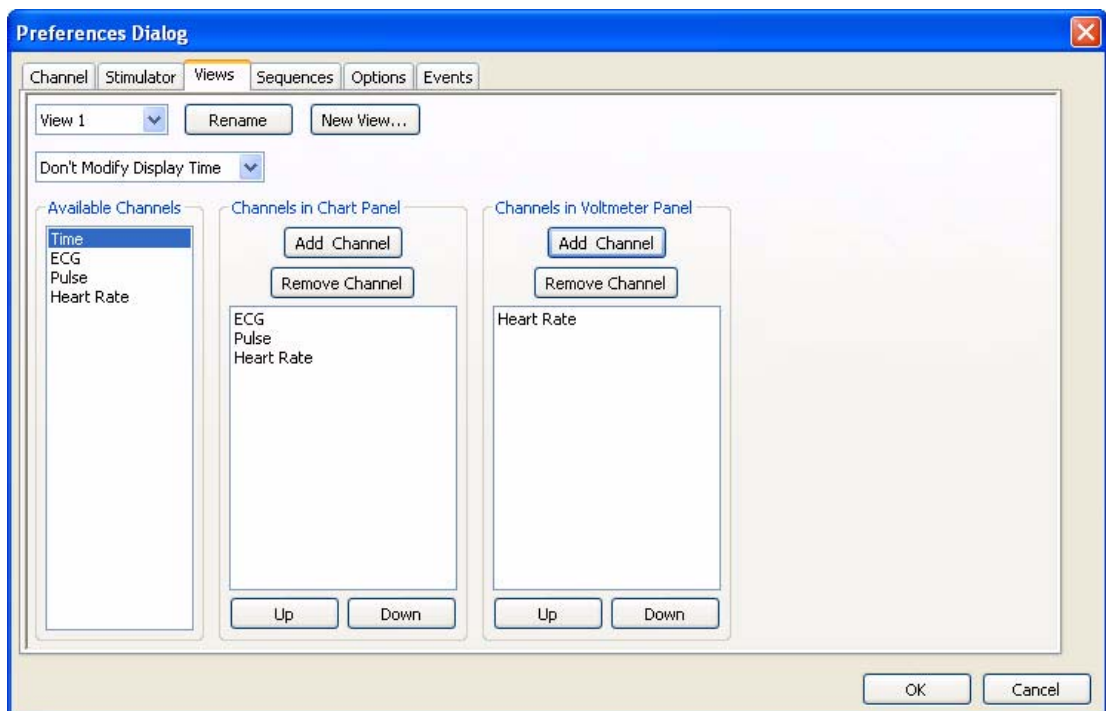
The Views Preferences Dialog.

Note: *Any changes to the default view will also be made to the preferences and vice-versa.*

Units Conversion

When used with *iWorx* hardware, *LabScribe2* 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 appropriate unit for the data being recorded.

If *LabScribe2* 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 converted into milligrams, grams, or any other units. *LabScribe2* can handle these conversions easily, provided that the function that converts voltage into units appropriate to the transducer is linear.

LabScribe2 offers several options for Units Conversion. They are listed in the **Units** submenu of the **Channel Menu**.

Invert
Simple...
Advanced...
Set Offset...
Off (all blocks)

The Units submenu.

- **Invert**: Inverts the trace.

- **Simple...** : Opens the **Simple Units Conversion** dialog.

- **Advanced...**: Opens the **Advanced Units Conversion Dialog**.

- **Set Offset...**: Allows the user to set an offset required by certain transducers.

- **Off (all blocks)**: Turns all units conversions off.

To convert voltage input to real units in the **Simple Units Conversion** dialog using **2 point calibration**:

- Record a portion of data at two known values. In the case of an oxygen probe, record output at two known concentrations such as zero and 100% oxygen saturation of water. The recorded trace may look something like the figure below.
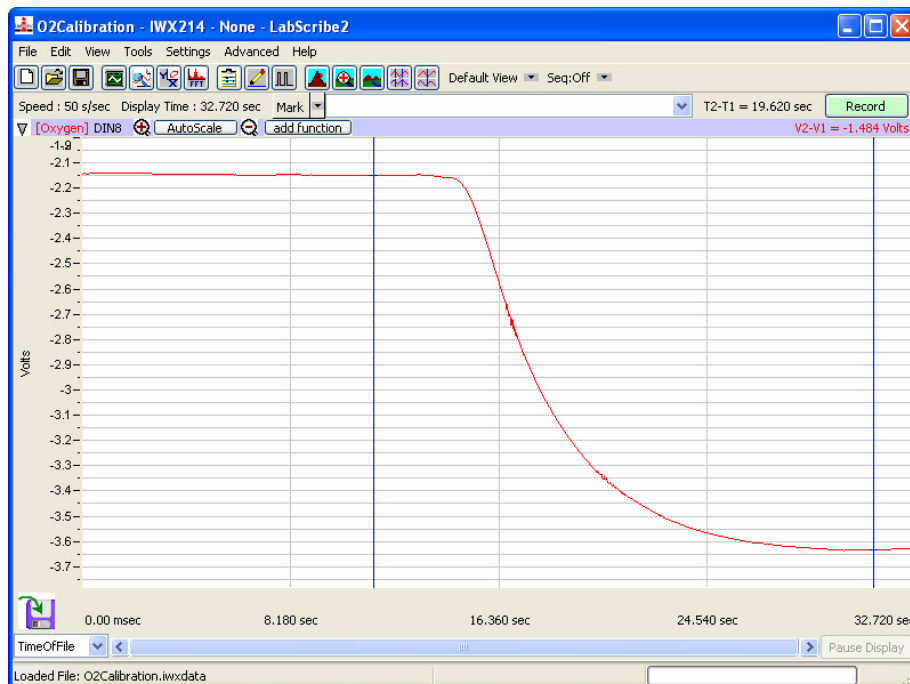
Data file of oxygen concentrations with cursors positioned at two known concentrations.

- Once recording is complete, proceed to **Two Cursor Mode** in the **Main Window** by clicking on the **Two Cursor** icon in the **Toolbar**. The **Units Conversion** dialog window cannot be entered without being in **Two Cursor Mode**.

- Position **Cursor 1** over one of the known values, and **Cursor 2** over the other known value.

- Open the **Channel Menu** by clicking on the arrow on the left end of the **Channel Bar** or right-click anywhere in the data channel and select **Units** from the **Channel Menu**. The **Units** menu can also be accessed by left-clicking on the **Value Display Area** of the **Channel Bar**.

- Select **Simple...**. to open the **Simple Units Conversion** dialog window.

- Select **2-point calibration** from the drop-down menu of the **Simple Units Conversion** dialog.
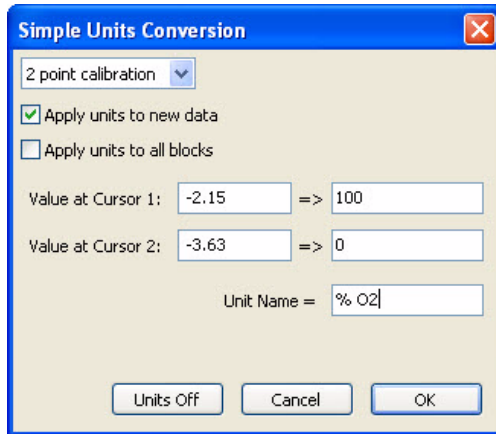
The Units Conversion dialog window showing the 2-point calibration configuration.

- 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 the default name.

- The units are always applied to the selected data 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.

- Click **OK** to exit the dialog.

For some recordings, it may be more accurate to use the mean of a small range of data to set the calibration values, The **Advanced Units Conversion Dialog** makes this possible. To use the **Advanced Units Conversion Dialog** to make a 2 point calibration:
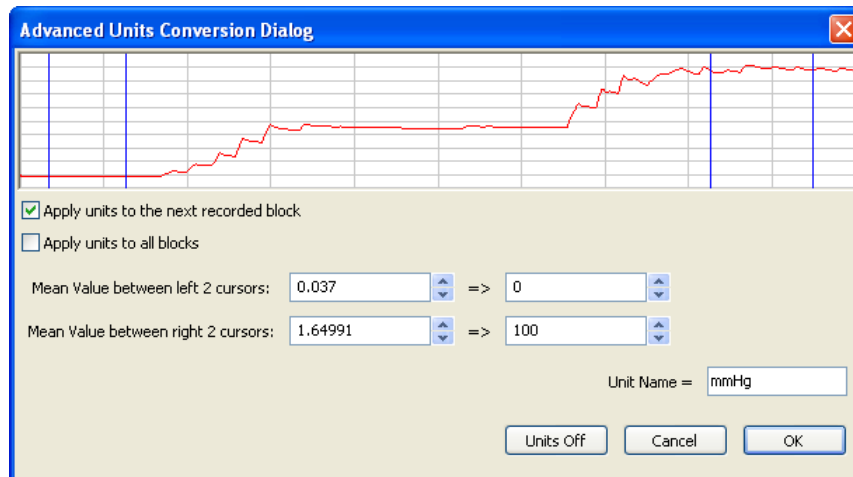
The Advanced Units Conversion Dialog.

- Choose **Advanced...** in the Units submenu.

- Use the two cursors on the left to bracket an area of known average value, and the two on the right to bracket the second area of known average value. *LabScribe2* will display the mean of the bracketed areas. In this way, it is possible to get representative values in recordings with some degree of amplitude fluctuation.

- In the dialog, the mean voltage values between the two cursors on the left and the two cursors on the right are entered into the two text boxes on the left. Enter the corresponding values in real units into the two value boxes on the right.

- Click **OK** to exit the dialog.

It is also possible to set the slope and offset directly in the **Simple Units Conversion** dialog, if those values are known. 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 set **slope and offset**:

- Right-click anywhere in the data channel, select **Units** from the **Channel Menu** or left-click on the **Value Display Area** of the **Channel Bar** to open the **Units** sub-menu.

- Select **Simple...**. to open the **Simple Units Conversion** dialog window.

- Select **slope and offset** from the drop-down menu of the **Simple Units Conversion** dialog.
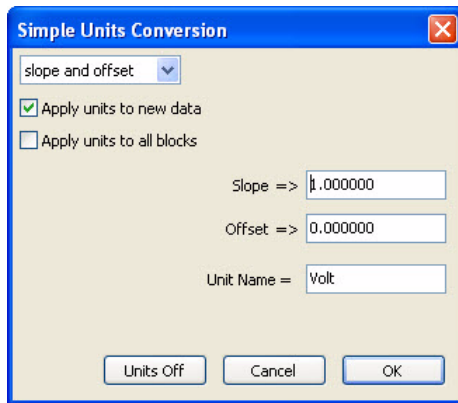
- Enter the slope and unit name in the appropriate boxes.

- 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 *LabScribe2* reports on the screen when the sensor should be reading zero. Enter this value in the **offset** area of the **Units Conversion** dialog window.

The Simple Units Conversion dialog window showing the offset only configuration.

- 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.

- Click **OK** to exit the dialog.

Sometimes, it is necessary to keep the units conversion relationship and change the offset, like turning the offset control on an amplifier. For example, to set a particular region to zero:

- Set the two cursors around the desired region of data.

- Right-click anywhere in the data channel, select **Units** from the **Channel Menu** or left-click on the **Value Display Area** of the **Channel Bar** to open the **Units** sub-menu.

- Select **Simple...** from the **Units** submenu of the **Channel** menu.
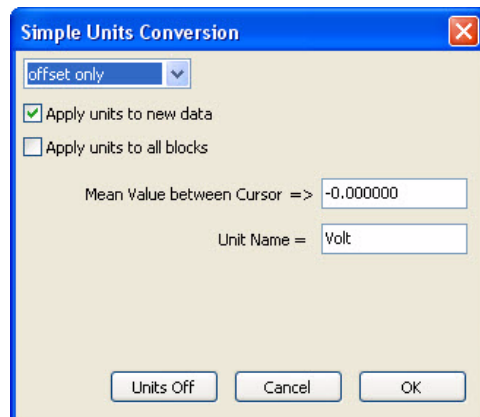
- Choose **offset only** from the drop-down menu in the **Simple Units Conversion** dialog. The average value between the two cursors can now be set to the required offset value (zero, in this example).

- You can alternatively select **Set Offset...** from the **Units** sub-menu and enter the required offset in the **Apply Offset to Data** dialog.

- 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.

The Simple Units Conversion dialog window showing the offset only configuration.

- Click **OK** to exit the dialog.

Unit conversions can also be set from the **Channel** page of the **Preferences Dialog**. Each channel can be set with the conversion factors provided by the transducer manufacturer (refer to page 42 in **Chapter 4: Creating Your Own Settings and Preferences**).

At times, it may be desirable to turn off the **Units Conversion** and simply view the raw data in the default unit, which is volts. You can turn off the units for all blocks directly from the **Units** sub-menu or from the **Simple Units Conversion** dialog box.

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 polarity, the trace can be inverted by selecting **Invert** from the **Channel Menu**, **Units** sub-menu, or the right-click menu in any data channel. The **Invert** function can be switched off at any time by selecting **Invert** a second time.

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 are being recorded, the most recent values are displayed.

- On a recorded trace, the **Voltmeter** displays the amplitude of the trace on each channel at the position of a single cursor, or the difference in amplitudes between the positions of two cursors.
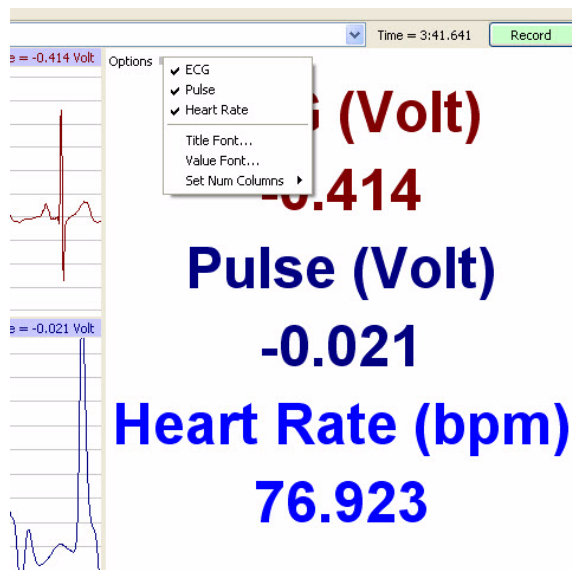
The Voltmeter and Voltmeter Options menu.

- The voltmeter display can be customized in different views. The channels to appear in the **Voltmeter** display can be chosen. The font size and style, the color, and the channel order can be set in the **Voltmeter's Options** menu. It is also possible to organize the **Voltmeter** display in the **Views** page of the **Preferences Dialog**, as discussed on page 11 earlier in this chapter.

Online XY

In **Online XY** 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-axis and the X-axis channel from the drop-down menu at the bottom of the plot window. The **Main Window** display time also applies to the online XY plot. For a complete discussion of XY plots, refer to **XY Plot** on page 89 in the **Redisplayed Data** section of **Chapter 7: Analysis**.
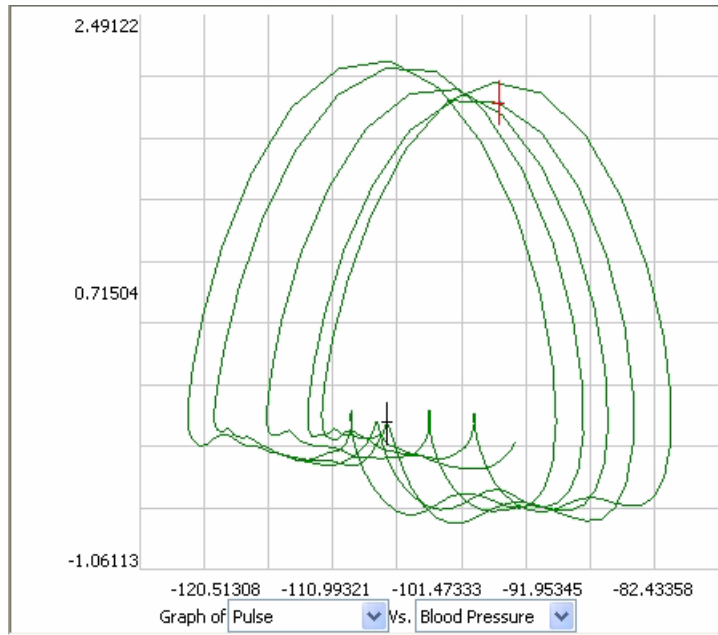
Online XY Plot data.

Other Display Windows

Data recording occurs only in the **Main Window**. However, once recording is complete, other windows can receive selected data from the **Main Window** for closer examination or display in another format, like an XY plot or a FFT analysis. 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 **Redisplayed Data** section beginning on page 89 of **Chapter 7: Analysis**.

Chapter 2: The Menus

Overview

While many of *LabScribe2's* features can be accessed in more than one way, the main menus and the dialogs they open provide a systematic way to access virtually every *LabScribe2* feature.

The Menus

File

LabScribe2 supports all of the elements found in a standard **File** menu.
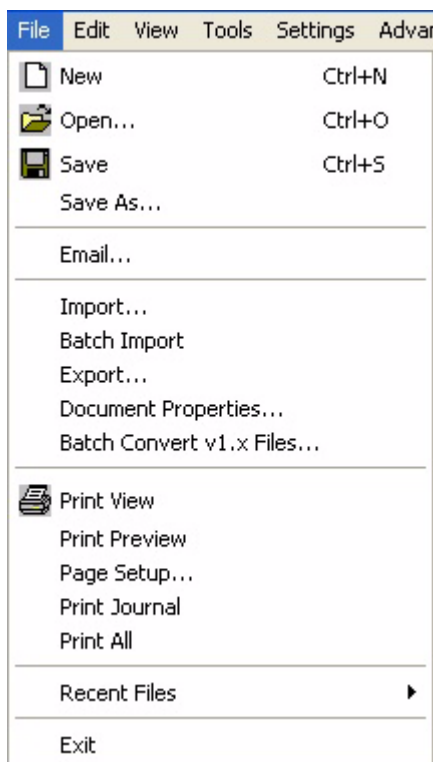
The File menu.

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. This function is also available from the **Toolbar**.

Save: Saves data to the active file. This function is also available from the **Toolbar**.

Save As: Saves data to a new file with a different name or format. It is possible to rename a data file, save the settings as a **Settings File**, generate a **Sequences Output**, or save the **Journal** as an html (*.html) or xml (*.xml) file.

Email...: Opens a dialog that allows the user to email the active file. This function requires an active Internet connection.

Import...: Opens a dialog that allows the user to set the parameters for importing text data files. This is a separately licensed feature of *LabScribe2.*

Batch Import: Once a single text file is imported, a number of additional files can be imported with the same input parameters.

Export...: Allows the user to export the entire data file as text, or in a variety of formats appropriate to external analysis programs. These formats include Matlab (*.mat), DADISP (*.dat), and Excel (*.xls). The current screen can be exported as a JPG (*.jpg) or Portable Network Graphics (*.png) file. The current screen can also be exported as a *LabScribe2* data file.

Document Properties...: Opens the **Document Properties** dialog that allows the user to annotate the file wth notes, or to view the acquisition characteristics of the file, including the date and time of recording, the sampling speed, and the number of input channels.

Batch Convert v1.x Files...: Converts all v1.x files in a folder to v2.x files.

Print View: Prints the active window.

Print Preview: Previews the image to be printed.

Page Setup...: Opens a dialog box that allows the user to set up page formatting features specific to the printer being used.

Print Journal: Prints the **Journal**.

Recent Files: Opens a submenu displaying the last ten files opened. Choosing one closes the current file and opens the selected data file.

Exit: Quits the program. On the Macintosh, use **Quit Labscribe 2** in the **Labscribe 2** menu.

Edit

LabScribe2 supports elements found in a standard **Edit** menu:
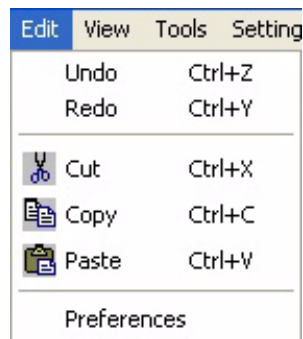
The Edit menu.

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

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

Cut: Cuts the selected information in the **Journal**.

Copy: Copies the window in the foreground or the current selection in the **Journal** to the clipboard for pasting into another application.

Paste: Pastes the contents of the clipboard to the **Journal**.

Preferences: Opens the **Preferences Dialog**, a tabbed multi-pane dialog box that displays the **Channels**, **Stimulator**, **Sequences**, **Options** and **Events** configuration panels. The **Preferences Dialog** is accessed from the **LabScribe2** menu on the Macintosh.

View   The **View** menu supports display elements specific to *LabScribe2*.
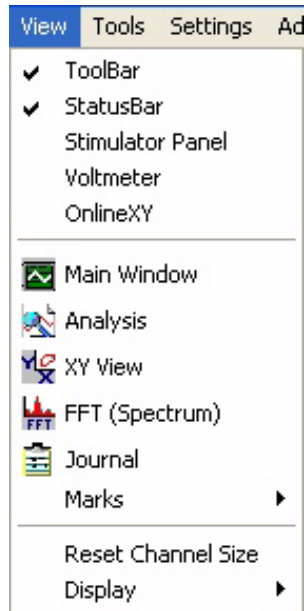
The View menu.

ToolBar: Hides or displays the *LabScribe2* **Toolbar**. The **Toolbar** is part of the default display.

StatusBar: Hides or displays the **Status Bar**. The **Status Bar**, at the bottom of the **Main Window**, displays the progress status when files are being loaded or saved. The **Status Bar** is part of the default display.

Stimulator Control Panel: Hides or displays the **Stimulator Control Panel** directly beneath the **Toolbar**. The **Stimulator Control Panel** is hidden in the default display.

Voltmeter: Hides or displays the **Voltmeter** pane on the right side of the **Main Window**. In the default configuration, the **Voltmeter** displays, in large type and in digital voltmeter fashion, the amplitude values at the cursor (or the difference between the cursor values in **Two Cursor Mode**) from all channels. The **Voltmeter** display configuration can be changed through the **Option**s drop-down menu in the **Voltmeter** panel itself, or from the **Views** page of the **Preferences Dialog**.

OnlineXY: Displays a realtime **XY plot** on the right side of the **Main Window**. The two channels to be plotted can be selected with the drop-down menus at the bottom of the plot.

Main Window: Brings the **Main Window** to the foreground. This function is also available from the **Toolbar**.

Analysis: Brings the **Analysis Window** to the foreground. This function is also available from the **Toolbar**.

XY View: Opens the **XY View**. This function is also available from the **Toolbar**.

FFT (Spectrum): Opens the **FFT Window**. This function is also available from the **Toolbar**.

Journal: Opens the **Journal** on the right side of the **Main**, **Analysis**, **XY**, and **FFT Windows**. This function is also available from the **Toolbar**.

Marks: Opens a submenu with controls that open the **Marks Dialog**, add the mark in the **Marks** text box to the recording, and reset the marks on the current screen to their original positions in the **Text Display Area** at the base of the **Main Window**.
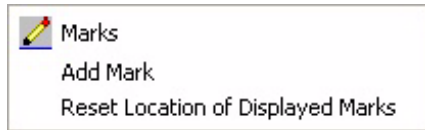
The Marks submenu.

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

Display: Opens a submenu with controls for changing display time parameters. These parameters are discussed starting on page 27 of **Chapter 3: Acquisition**.
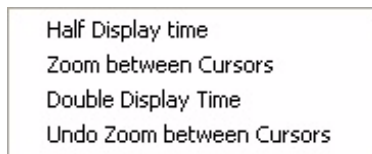
The Display submenu.

Tools

The **Tools** menu controls functions that can find and move specific data to the **Journal,** find specific events or regions in the recording, and edit the data file.
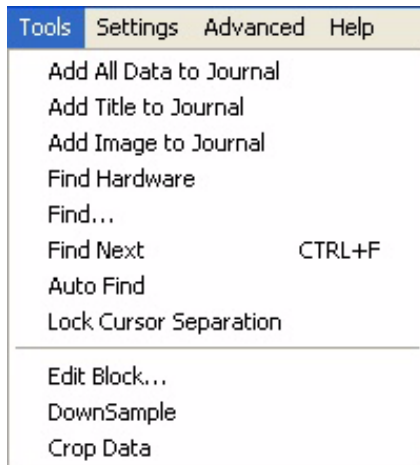
The Tools menu.

Add All Data to Journal: Sends data from the **Main** or **Analysis Windows** to the **Journal**.

Add Titles to Journal: Sends the titles of the selected **Table Functions** from the **Analysis Window** to the **Journal**.

Add Image to Journal: Sends an image of the current screen to the **Journal**. Refer to the section beginning on page 136 of **Chapter 10: The Journal and Data Export** for a detailed account of this and the preceding two functions.

Find Hardware: Identifies the *iWorx* data acquisition device in use and initializes the computer's connection to it.

Find: Calls a dialog box with controls that program *LabScribe2's* cursors to find data that matches user-defined criteria in the **Main** or **Analysis 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 can be automatically added to the **Journal** for each matching data point. Refer to the **Find Functions** section beginning on page 96 of **Chapter 7: Analysis** for a complete discussion of all the **Find Functions**.

Lock Cursor Separation: Locks the cursors at a fixed separation distance. Refer to **Locking Cursor Separation** on page 7 in the **Cursors** section of **Chapter 1: The Display** for a complete discussion.

Edit Block...: Opens a dialog box that offers a number of options to edit the current file. **Warning: These edits are permanent and cannot be undone!**
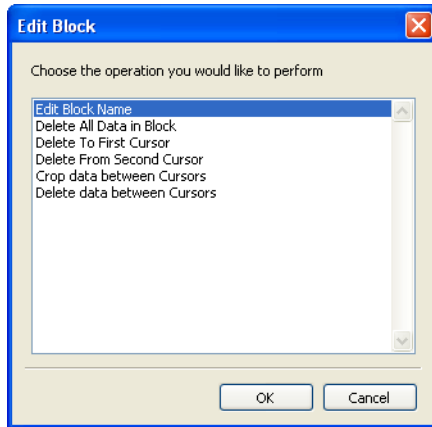
The options in the **Edit Block** dialog are:

- **Edit Block Name**: Allows the user to name or change the name of the data block.

- **Delete** options: All the data in the block, the data in the block before **Cursor 1**, or the data in the block after **Cursor 2,** can be deleted. The data between the two cursors can also be deleted.

- **Crop**: The data between the cursors can be croppped and saved as a data file.

The Edit Block dialog.

DownSample: Used to downsample the data to a sampling speed chosen by the user. This reduces the information in the file and should be used with caution. See the discussion on choosing a sampling speed beginning on page 34 of **Chapter 3: Acquisition**. It is important to be sure that the sampling rate chosen doesn't sample the data inadequately. **Warning: This cannot be undone!**

Crop Data: Crops the data between the cursors. This data section can be saved as a new file.

Settings

The **Settings** menu allows the user to load, create, or edit files containing preset recording and analysis parameters. Selecting a file from the list of experiments beneath the line programs *LabScribe2* to record data in a manner specified by the **settings file** associated with each experiment.
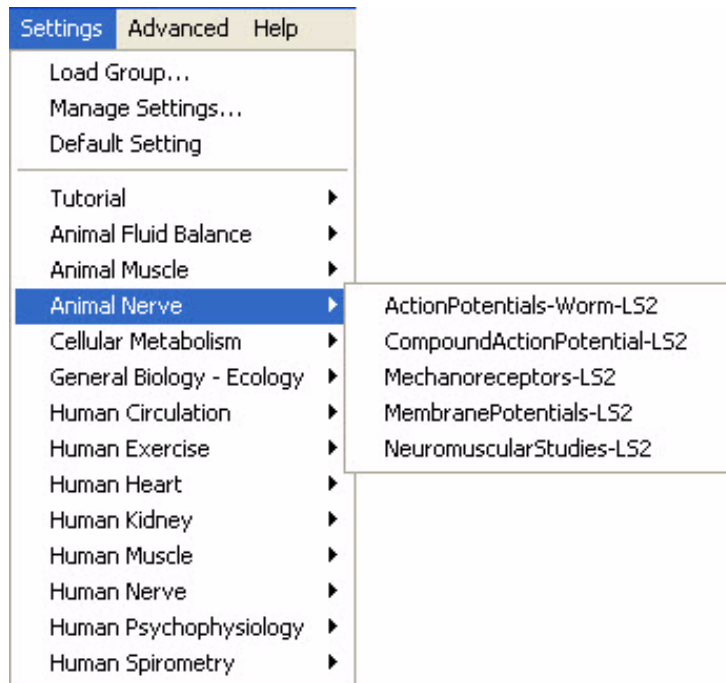
The Settings menu.

Load Group: Loads a group of existing **settings files**. Each file that is part of the group contains the *LabScribe2* recording and analysis parameters necessary to record a specific experiment or type of data. Refer to the section starting on page 26 in **Chapter 3: Acquisition** for detailed instructions.

Manage Settings...: Opens the **Settings Manager** dialog box. The controls in the dialog allow the user to edit existing **settings files** in a group, remove them from a group or to add new ones to a group. Refer to the section beginning on page 45 of **Chapter 4: Creating Your Own Preferences and Settings** for more details.

Default Setting: Restores the **Main Window** to its default view.

The menu items beneath the line refer to categories of experiments that have been loaded into *LabScribe2*. Selecting any of them will open a sub-menu listing the experiments in that category.

Advanced

The **Advanced** menu has options for performing advanced analyses specific to a physiological function or type of data. Through these analyses, *LabScribe2* is able to locate specific data points of physiological interest and perform calculations pertaining to these data. Refer to **Chapter 8: Advanced Analysis Routines** for a complete discussion of these functions.
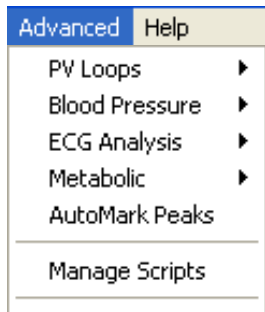
The Advanced menu.

PV Loops: Displays a submenu that opens dialogs allowing the user to provide criteria by which *LabScribe2* can locate and mark specific **Pressure Volume Loop** features, and perform both online and offline mathematical calculations on these parameters. Refer to the **PV Loops** section beginning on page 100 of **Chapter 8: Advanced Analysis Routines** for complete instructions. **PV Loops** is a separately licensed analysis module.
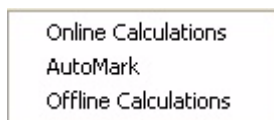
The PV Loops submenu.

Blood Pressure: Dispays a submenu that opens dialogs allowing the user to provide criteria by which *Labscribe2* can locate and mark specific **Blood Pressure** parameters, and perform both online and offline mathematical calculations on these parameters. Refer to the **Blood Pressure** section beginning on page 110 of **Chapter 8: Advanced Analysis Routines** for complete instructions. **Blood Pressure** is a separately licensed analysis module.
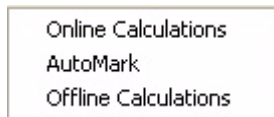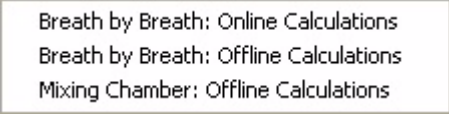
The Blood Pressure submenu.

ECG Analysis: Displays a submenu that opens dialogs allowing the user to provide criteria by which *LabScribe2* can locate and mark specific **ECG** features, and perform offline mathematical calculations on these parameters. See the **ECG Analysis** section beginning on page 116 of **Chapter 8: Advanced Analysis Routines** for details and complete instructions. **ECG Analysis** is a separately licensed analysis module.
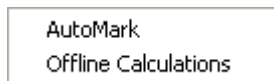
The ECG Submenu.

Metabolic: Displays a submenu that opens dialogs allowing the user to set up the criteria by which LabScribe2 can make both online and offline calculations of **Metabolic** functions. Refer to the **Metabolic** section beginning on page 122 of **Chapter 8: Advanced Analysis Routines** for more detailed information. **Metabolic** is a separately licensed analysis module.

Breath by Breath: Online Calculations
Breath by Breath: Offline Calculations
Mixing Chamber: Offline Calculations

The Metabolic submenu.

AutoMark Peaks: Opens a dialog in which the user can set the criteria for *Labscribe2* to automatically locate peaks in many types of recordings and calculate a number of peak characteristics. See the **AutoMark Peaks** section beginning on page 127 of **Chapter 8: Advanced Analysis Routines** for details and complete instructions.

Manage Scripts: Additional data analysis can be accomplished using external programs. Selecting **Manage Scripts** opens the **Scripting Setup Dialog** in which the user can set up shortcuts to export the relevant data, and then call an external script for further analysis. Scripting functions are described in more detail in the **Managing Scripts** section beginning on page 99 of **Chapter 7: Analysis**.

Help

The **Help** menu provides links to information about the *LabScribe2* software and hardware.
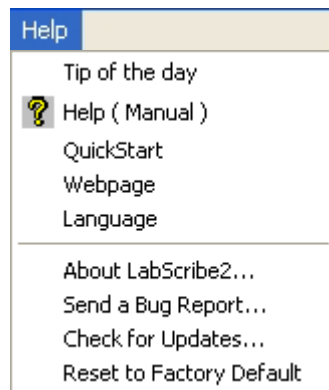
The Help menu.

Tip of the Day: Displays a useful *LabScribe2* tip.

Help (Manual): Opens the online version of the *LabScribe2* software manual.

QuickStart: Opens the **QuickStart** guide.

WebPage: Takes the user to the *iWorx* home page. This function requires an open Internet connection.

Language: Allows the user to change the language *LabScribe2* uses. Changing the language requires a restart to take effect.

About *LabScribe2***...**: Displays the version number and creation date of the copy of *LabScribe2* in use, as well as information about the connected hardware.

Send a Bug Report...: Sends a bug report to *IWorx*. This requires an active Internet connection.

Check for Updates...: Sends the user to the **User Area** login page at iworx.com. The current *LabScribe2* software can be downloaded from the Software page of the **User Area**. This function requires an open Internet connection.

Reset to Factory Default: Resets all **Preferences** to the Factory Default condition.

Chapter 3: Acquisition

Overview

Start Recording

The most basic control in *LabScribe2* 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 unit, click the **Record** button to begin recording.

While data are being recorded the **Record** button changes to **Stop**. Click **Stop** at any time to end data recording.

Settings

In addition to the hardware and software provided in its teaching kits, *iWorx* provides a variety of experiments in electronic lab manuals. To support these experiments, *iWorx* has created **settings groups** that contain links to **settings files** for each experiment in that group. 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
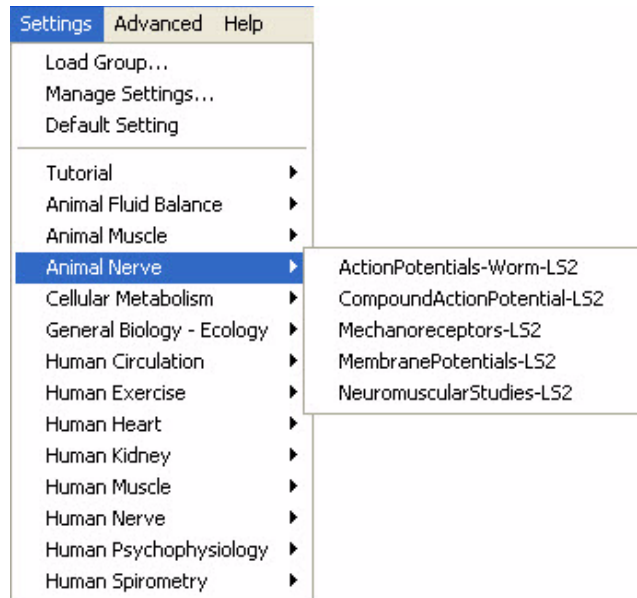
To load a **settings group** and the **settings files** for the experiments:

- Select **Load Group** from the **Settings** menu. This will open the *LabScribe2* folder, which contains the **Settings** folder. From the **Settings** folder, choose the **settings group** file **(settings group** files will be of type *.iwxgrp) that you wish to load, and click **Open.** Generally you will want to load the **Complete settings group**.

The Settings menu.

- The **settings files** are organized into categories. Once a **settings group** is loaded, the experimental categories appear by name in the lower bracket of the **Settings** menu. To open a specific **settings file**, select the appropriate category to display the experiments in that category. Highlight the name of the file and click it. The **settings file** associated with that experiment will load into *LabScribe2* and set the appropriate parameters for recording and displaying data.

- *iWorx* experiments are grouped into one of several electronic laboratory manuals that are available for download from the **User Area** of iworx.com. Each **settings file** opened from the **Settings** menu is associated with a **helper file**, which is a .pdf copy of the laboratory exercise as it appears in one of the *iWorx* laboratory manuals. Opening the exercise's **settings file** will also open this .pdf file.

Main Window Display Considerations

Recorded data have two important dimensions: time and amplitude. Each of these has its own set of controls.

Managing Display Time

The events recorded using *LabScribe2* occur over very different time frames. For example, recording the discharge curve of a 9-Volt battery takes hours, while recording the QRS complex in a human electrocardiogram takes only a fraction of a second. *LabScribe2* allows the recording of both very slow and very fast events while displaying the data in a format that is easily interpreted. To manage the time component of recorded data, a parameter called **Display Time** is used. **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. **Settings files** for specific lab exercises may set a different **Display Time**. **Display Time** can be changed by the user by using the display controls in the **Toolbar**, or by manually entering a **Display Time** on the **Channel** page of the **Preferences Dialog**, which is launched by selecting **Preferences** in the **Edit** menu in Windows, or in the **LabScribe2** menu on the Macintosh.

Using the **Toolbar** icons to control **Display Time**:

- Clicking the **Half Display Time** icon (icon with the big mountain) in the **Toolbar** 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. Using the **Half Display Time** tool expands the record as many times as requested, until there are only 10 data points on a screen of data. By the time this happens, the data will usually appear to "flatline" and events on this scale will not be recognizable.The **Half Display Time** command can also be accessed through the **View** menu's **Display** submenu.
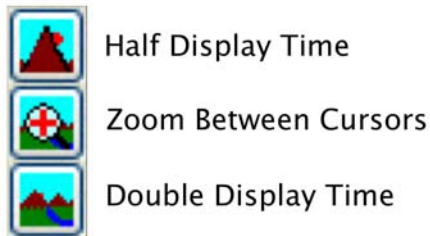
Display Time icons in the Main Window Toolbar.

- Clicking the **Double Display Time** icon (icon with the two smaller mountain peaks) in the **Toolbar** 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. 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 are displayed on one screen. By default, the maximum number of data points that may be displayed on the screen is 100,000. This can be changed on the **Options** page of the **Preferences Dialog**. The **Double Display Time** command can also be accessed through the **View** menu's **Display** submenu.

- In **Two Cursor Mode**, clicking the **Zoom Between Cursors** icon fills the display with the data located between the cursors. You can undo the **Zoom Between Cursors** by choosing **Undo Zoom Between Cursors** in the **Display** submenu in the **View** menu.

There is a scrollbar at the bottom of the window that can be used to scroll through the data. Scrolling can also be achieved by holding down the CRTL key (COMMAND on the Macintosh), and clicking and dragging in the graph window.

Two examples demonstrating the use of the **Display Time** feature are illustrated below.

In the first example, 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.
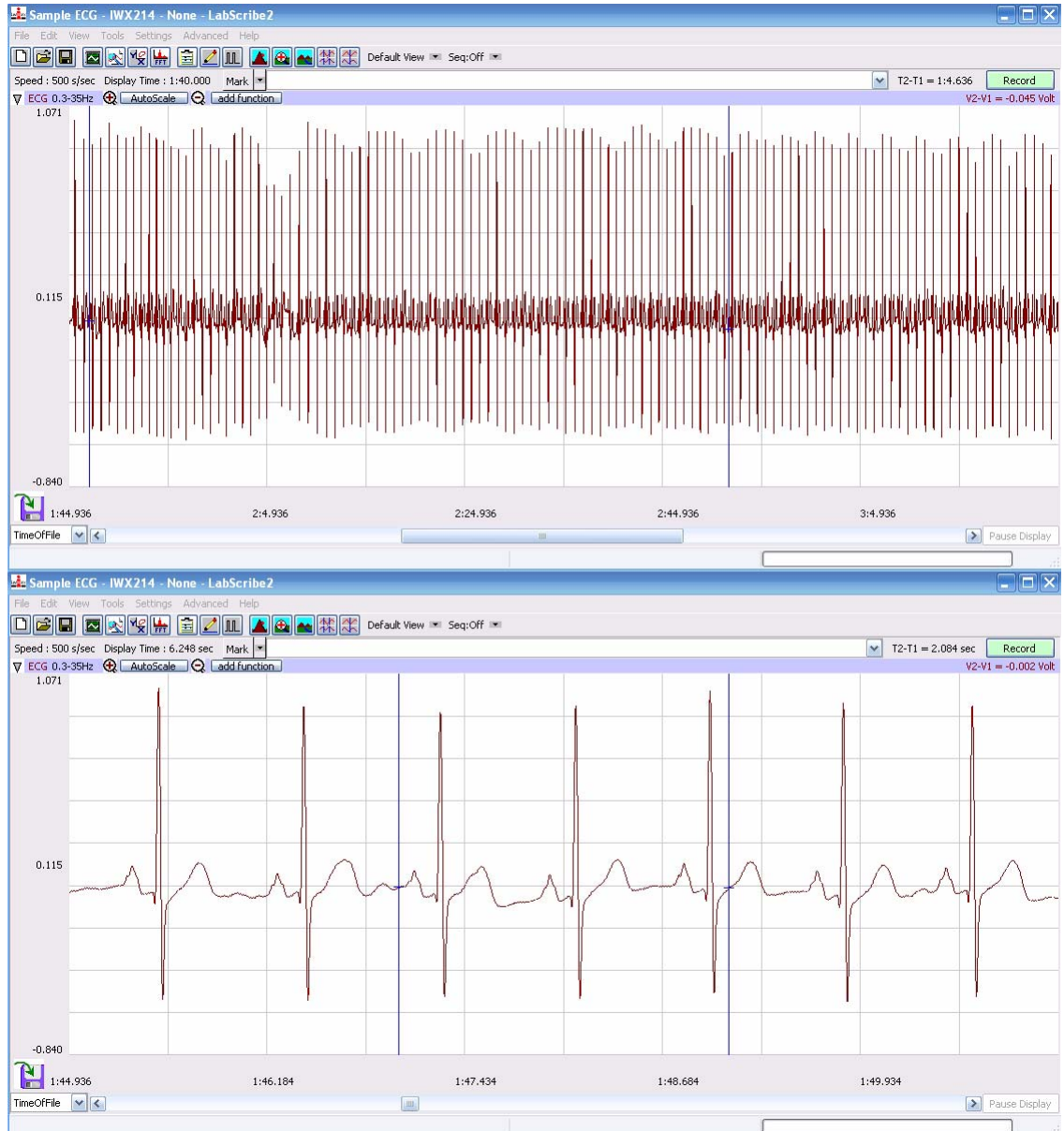
ECG viewed with a long Display Time(top) and a short Display Time (bottom).

In the second example, a one hour battery discharge curve is shown in two views. The two-second screen time shows almost a straight line, but clicking **Double Display Time** eight times reveals the entire curve.
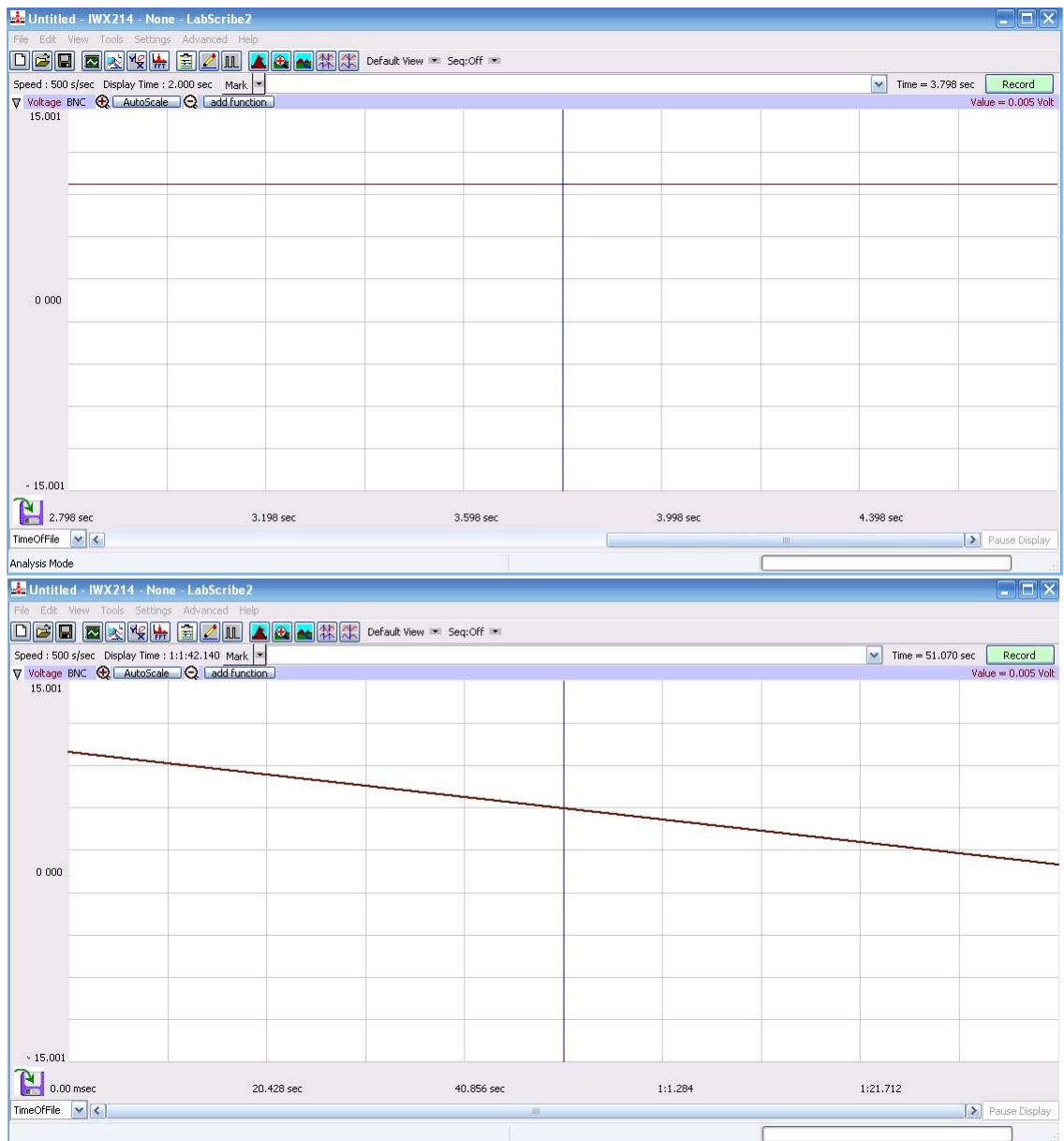
Voltage drop of 9-Volt battery viewed with short (top) and long (bottom) Display Times.
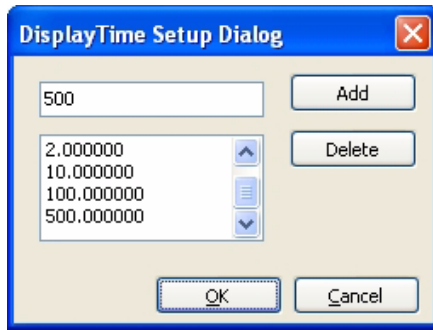
Display Time
Setup Dialog

Setting the display time to a specific value can be done on the **Channel** page of the **Preferences Dialog**. To switch between certain known display time values, using the **Preferences Dialog** can be cumbersome. Specific display time values can be added to the **Display Time** menu directly beneath the **Toolbar** by using *LabScribe2's* **Display Time Setup Dialog**.

The Display Time Setup Dialog.

To access the **Display Time Setup Dialog**:

• Click on the **Display Time** label (just beneath the **Toolbar**) and choose **Manage...**. This brings up the **Display Time Setup Dialog**.

• Enter the value of the desired display time (in seconds) in the text box, and click the **Add** button. This will add the desired display time to the list of display times. Once all the desired display times have been added, click **OK** to exit the dialog.

• When you click on the **Display Time** label just below the **Toolbar** again, you will see the added values in the **Display Time** menu.
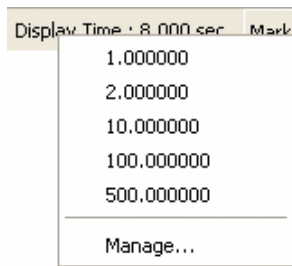
• Choosing any of the preset **display times** will set the **display time** to the chosen value.

Display Time presets menu.

Managing Amplitude Display

The vertical display of the recorded signal is managed in a number of ways. In addition to the **Toolbar** icons, there is a **Scale** submenu available from the **Channel Menu** (accessed by clicking on the arrow on the left side of the **Channel Bar** or right-clicking anywhere in the data channel). Right-clicking on the values in the Y-Axis on the left hand side of each channel will also display the **Scale** submenu.

Zoom In, **AutoScale** and **Zoom Out** can be controlled using the **Toolbar** icons or by using the commands in the **Scale** submenu. The other options in the **Scale** submenu are: **Set Scale** and **Preferred Scale**.

Zoom Tools

The **Zoom In**, **AutoScale** and **Zoom Out** icons in each **Channel Bar** control the vertical display in that channel.
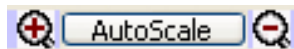
• Clicking the **Zoom In** button will double the displayed amplitude of the trace.

The Zoom Tools

• **AutoScale** locates the highest and lowest data points in the channel. It then adjusts the Y-scale range to include those points, optimizing the amplitude of the channel's data.

• Clicking the **Zoom Out** icon will reduce the displayed amplitude of the trace by a factor of two.

• **Zoom In**, **AutoScale** and **Zoom Out** can also be controlled using the commands in the **Scale** submenu.

Set Scale

Units	▶
Scale	▶

Set Scale...
Prefered Scale
Zoom In
AutoScale
Zoom Out
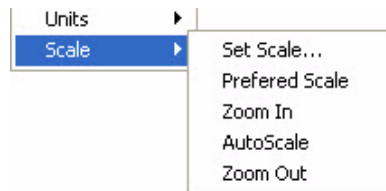
The range of the amplitude scale can be set manually by entering the lower and upper levels in the **Set Scale** dialog. The possible range will be limited by the *iWorx* data acquisition unit in use (±5V or ±10V).

The Scale submenu.

Preferred Scale   For any data, there is a range within which the Y values of the signal are most likely to occur. For example, if measuring room temperature, the user might select a range of 50°F to 100°F because it is unlikely that room temperature will be colder or warmer than these limits.

- To set the range of the **Preferred Scale**, choose the **Set Scale** option from the **Scale** submenu and enter the upper and lower limits of Y-scale in the **Set Scale** dialog window that appears. The range will be limited by the hardware in use (±5V or ±10V).

- To display the **Preferred Scale** you have specified, select the **Preferred Scale** option from the **Scale** submenu.

- If an event outside the **Preferred Scale** occurs during the recording, the area of interest can be expanded using the **Zoom** or **AutoScale** tools. To return to the **Preferred Scale**, select the **Preferred Scale** option from the **Scale** submenu.

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

Signal Conditioning

Gain   There is a minimum voltage that the A/D Converter can display. The specific voltage depends on the *iWorx* data acquisiton unit in use. If the signal being measured is smaller than this minimum voltage, additional gain can be applied to the signal before it is presented to the A/D Converter by using a bioamplifier.

For example, consider an ECG with a total peak to peak amplitude of only 2 mV. Because of its voltage limitations, the A/D Converter is not sensitive enough to record any meaningful changes to a signal this small. The signal is often smaller than the noise level, and it's not possible to distinguish the signal from the noise. If an amplifier is placed between the signal and the A/D Converter, the raw signal can be amplified by a selected gain. If the gain is 100 times (X100), the 2mV ECG signal becomes a 200mV signal. Now, when the amplified signal is presented to the A/D Converter, it is 100 times bigger, and large enough for the A/D Converter to display it accurately and distinguish it from the noise.

Adding gain to the recording system improves the signal to noise ratio of the measuring system, but the A/D Converter is still limited to a total range of ± 5V or ±10V, depending on the A/D Converter in use. In the case where X100 gain is applied to a 0.2V signal, the amplified signal becomes +20V, which is outside the input limit of the A/D Converter and the signal goes out of range. If a gain of X10 is used on an amplifier with a total range of ±10V, the effective input range of the A/D Converter drops to ±1V (±10V range/X10 gain). Any input signal larger than 1V will be out of range. If a gain of X100 is applied to the input signal, the effective input range of the A/D Converter is restricted to ±0.1V.

Bioamplifiers   The bioamplifier channels of the *iWorx* A/D converters apply gain to the input signals coming through them. The gain and filter modes of channels equipped with a bioamplifier are selected from the **Mode/Function** menu on the **Channels** page of the **Preferences** dialog window (accessed from the **Edit** menu in Windows or from the **LabScribe2** menu on the Macintosh), or the **Mode/Function** button drop-down menu in the **Channel Bar** of the raw data channels with a bioamplifier input. The *LabScribe2* **settings files** set the appropriate mode for each experiment that uses channels equipped with bioamplifiers.

DIN 8 Inputs   The DIN 8 inputs on *iWorx* A/D 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 *iWorx* units. Gain programming resistors are already present in *iWorx* transducers. Gain programming resistors can be installed on non-*iWorx* transducers by rewiring the connector. Consult the hardware documentation for the pin configurations and diagrams of the DIN 8 connectors used with *iWorx* A/D Converters.

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. The positioning control permits the the signal to be centered in the data window, making measurements more convenient and lowering the baseline to accommodate the display of a signal that has more gain applied to it. The availability of the **AutoScale** feature in the *LabScribe2* software reduces the need for a positioning knob. In fact, the very low noise of the *iWorx* A/D Converters makes positioning controls unnecessary for many transducers as signals are automatically centered and expanded to fill the recording screen when the **AutoScale** button is clicked. Positioning of the waveform may be also be accomplished by clicking on the waveform and dragging it to the desired position within the channel. Some transducers still require a positioning control in order to position the trace within the data channel, and these *iWorx* transducers come equipped with an offset control.

Filters

Filters can be set to remove certain frequencies from signals. The goal of filtering is to remove noise from the recording while passing those frequencies that make up 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. As an example, a large percentage of ECG signal information is contained in frequencies below 40Hz. A significant noise source in such recordings is the 60Hz line voltage (from 110V AC power, or mains) used to power equipment and lights. A 50Hz **Low Pass** filter allows all frequencies below 50Hz (including most ECG information) to pass to the recorder, but excludes all frequencies above 50Hz, including the 60Hz noise from the mains. Applying the 50Hz Low Pass filter creates a quieter, more readable ECG. In general, the application of a low pass filter "quiets" high frequency noise 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 slow baseline drift or a standing offset voltage, so that the user sees a more stable baseline. The signals of interest (extracellularly recorded action potentials) in neuronal extracellular recordings are of a very high frequency, while there is little of biological interest in the lower frequencies, so **High Pass** filters are used to filter out both 60Hz noise and slower oscillations that would otherwise cause the trace to drift in and out of the data channel.

The simultaneous use of **High Pass** and **Low Pass** filters can create a **Band Pass** filter. Three different analog band pass filters are available for the bioamplifiers on the *iWorx* A/D Converters. These **Band Pass** filters are enabled automatically when they are selected from the **Mode/Function** menu on the **Channels** page of the **Preferences** dialog window, or from the **Mode/Function** button drop-down menu in the **Channel Bar** of channels equipped with bioamplifiers. The appropriate filters are set by the **settings files** of experiments that require these bioamplifier filters. These are active hardware filters with the values in the table below.

Bioamplifier Band Pass Filter Frequencies

Band	Application
0.3-35 Hz	EEG/ECG
0.1-150 Hz	ECG
3Hz-10KHz	EMG

High Pass, **Low Pass** and **Band Pass** filters can also be applied to each channel and displayed in a computed channel by using one of the **Filter** functions 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 A/D Converters running *LabScribe2*. Unlike hardware filters, software filters can be applied after the data are recorded. These filters also work in real time and can be applied to data during recording. More detailed descriptions of these digital filters can be found starting on page 72 in **Chapter 5: Computed Channels**.

Averaging

In addition to filtering, *LabScribe2* can apply another quieting technique to data recorded at sampling rates lower than 200 samples per second. While fast events require fast sampling speeds, when slow events are recorded at fast sampling speeds, more information is collected than is needed to accurately display the signal. At sampling speeds of 100 samples per second or less, the *LabScribe2* 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

The built-in amplifiers and filters in *iWorx* A/D Converters 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 *iWorx* hardware. *iWorx* 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 *iWorx* data acquisition units. Outputs from these and other devices can be connected to the unamplified BNC inputs on the front panel of *iWorx* data acquisition systems. Outputs of external devices can also be connected to the DIN 8 inputs with an adapter cable available from *iWorx*.

Chart Mode

Once the signal source is connected to the *iWorx* 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. *LabScribe2* can start a recording manually, after a predetermined delay or when triggered by a specific event or condition. A recording can be stopped manually or after a predetermined amount of time. Before recording begins, an appropriate **sampling speed** must be determined.

Selecting a Sampling Speed

Temporal resolution in digitally recorded data is determined by the sampling rate (**sampling speed**).

The *iWorx* A/D Converter takes voltage measurements at regular intervals. The voltage measurements (y-axis values) and the times at which the voltages were recorded (x-axis values) are interpreted by the *LabScribe2* software as a pair of (X,Y) coordinates. *LabScribe2* plots these coordinates as data points. The software then connects the data points with a line to create the smooth, graphical appearance of an analog chart recording. Only the data points themselves are hard measurements. The connecting lines are "educated guesses" made by the software and will not accurately model the real data if the **sampling speed**, or the rate at which the data are sampled, is not fast enough for the data being recorded. To avoid creating recordings that may be inaccurate, the sampling interval needs to be short enough to insure that no important events occur between the sampled data points.

In the example below, a sine wave with a frequency of 100Hz has been sent to the A/D Converter. In the graph on the left, the data points recorded by an A/D Converter set to the same **sampling speed** as the frequency of the sine wave, are marked at the top of each cycle. 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. The graph on the right shows that a straight line is the result when the data points are connected. To render a more accurate representation of the data, a faster sampling speed is required. 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.
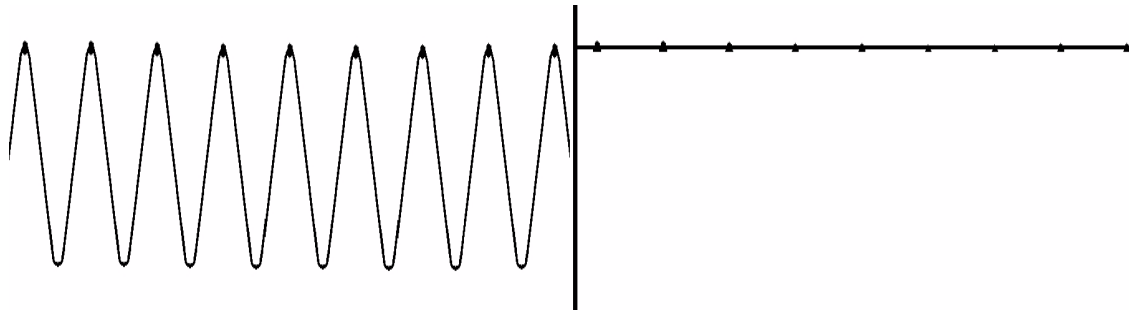
On the left, a 100Hz sine wave appears as displayed on an analog recording device. On the right , the same sine wave appears as a line when recorded on an analog-digital recording device at 100Hz sampling rate, the same frequency as the wave.

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 an ECG and should be used to determine the sampling speed needed to record an accurate representation of the ECG. Next, find the rise time (in seconds) of the R wave. The rise time is the time it takes the event of interest to go from its start to about 2/3 of its full amplitude. 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):

$$0.159/RiseTime = Bandwidth$$

$$(0.159/0.020) = 8$$

Multiply the bandwidth by five to determine the minimum sampling frequency needed, which in this example is 40Hz. Higher sampling rates represent the data more accurately, but this accuracy comes at the expense of larger data files. The optimal **sampling speed** strikes a balance between the accurate portrayal of the data and the unwieldy size of the data file.

The default *LabScribe2* **sampling speed** is 200 samples per second. This is adequate to execute most of the laboratory experiments presented in the online *iWorx* laboratory manuals. Some experiments utilize higher or lower sampling rates appropriate to the data being acquired.

Sampling speed can also be set experimentally. 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 causes degradation.

Sampling rates are set from the **Speed** menu on the **Channel** page of the **Preferences Dialog**, accessed from the **Edit** menu in Windows or the **LabScribe2** menu on the Macintosh.

The speeds displayed in the **Speed** menu are rates per second per channel. The maximum available sampling speed will vary depending on how many raw data channels are open.

Sampling speed menu.

SAMPLING SPEED EXERCISE

1. Using the **Tutorial** exercise as a guide, prepare to record some pulse transducer data.

2. Open the **Channel** page in the **Preferences Dialog** and set the **sampling speed** to 1,000 samples per second and the **Display Time** to 10 seconds. 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.

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

Vertical Resolution

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 allows more precise measurements 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 A/D Converter 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 a 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. In a 16-bit A/D Converter, there are 65,536 different possibilities, while a system using a 24-bit A/D Converter provides 16,777,216 possibilities. The Digital Resolution of the A/D Converter is its voltage input range divided by the number of possible A/D Converter steps. This is the minumum amplitude measurement possible with the A/D Converter.

If the input range of a 16 bit A/D Converter is 10V, dividing by 65,536 gives a minimum measurement of152µV. Individual *iWorx* data acquisition units are capable of different Digital Resolutions.

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 A/D Converter to be placed in the unit. However, the input range can be narrowed easily by applying amplification (gain) to the signal before the signal is presented to the A/D Converter. If a X1000 gain is applied to the incoming signal by an amplifier, the minimum resolution of a 16-bit A/DConverter improves to152nV.

Note: *The actual resolution of the system depends on both the resolution of the analog input device and the digital resolution of the A/D Converter. It is important to consider the resolution of the system as a whole.*

Starting Recording
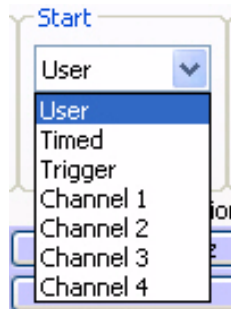
The Start dialog.

The **Start** dialog on the **Channel** page of the **Preferences Dialog** (accessed from the **Edit** menu in Windows or the **LabScribe2** menu on the Macintosh) shows the options for starting the acquisition process. Recording can be started manually (**User**), delayed for a specific amount of time after the **Record** button is clicked (**Timed**), triggered by a specific event from an external TTL device (**Trigger**), or triggered by an event or condition on one of the data channels (**Channels 1-4**).

User

The simplest way to start the recording is to click the **Record** button in the upper right hand corner of the *LabScribe2* **Main Window**. **User** is the default setting in *LabScribe2*. 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** or **Timed** mode.

Timed

If desired, the recording of data can be delayed for a user-specified fixed duration by choosing **Timed** and entering the desired value in the **Wait (sec)** text box of the **Start Dialog**.

External Trigger

It may be necessary to synchronize the beginning of the recording with the beginning of an external event. *LabScribe2* can be configured to start recording when the *iWorx* hardware detects a voltage pulse (+3V to +5V amplitude) through the BNC connector of the **Trigger** input. Many devices have **Trigger** or **TTL** outputs that are capable of starting the *LabScribe2* recording software. These devices include stimulators, relays, pumps, valves and cameras.

Triggered from Channel

An experiment may demand that data recording begin when the data meet certain criteria. For example, it may be necessary to record an animal's body temperature only if body temperature exceeds 100°F. *LabScribe2* can be programmed to begin recording when the data on the temperature channel exceeds 100°F. Recording is triggered when data values enter the window between two threshold values.

To set triggering thresholds:

Options for Triggering from a data channel with Pretriggering enabled.
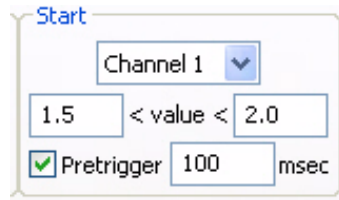
- Open the **Preferences Dialog** (from the **Edit** menu in Windows or the **LabScribe2** menu on the Macintosh).

- Click on the tab for the **Channel** page.

- Pull down the **Start** menu and select the channel that contains the data to be used as a trigger.

- Set the values that the data must meet before recording is triggered. In this example, *LabScribe2* 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 2V.

Pretriggering
When an external trigger or a trigger from a data channel is programmed, 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 *LabScribe2* software can be used to look back in time and display a small piece of data prior to the trigger.

To enable **Pretriggering**:

- In either **Trigger** or **Channel (1-4)** mode, check the box next to **Pretrigger** in the **Start** area on the **Channel** page of the **Preferences Dialog.**

- Enter the desired **Pretrigger** time in the **Pretrigger** text box.

Stopping

The Stop dialog.

Once recording begins, the *LabScribe2* programs offers two different ways to halt the recording: **User** or **Timed.** Each time recording starts and stops represents a data block, indicated on the display by a vertical line separating the blocks.

User
User is the default **Stop** mode for *LabScribe2* and can be reset from the **Stop** box on the **Channel** page in the **Preferences** dialog. In **User** mode, the **Stop** button in the upper right hand corner of the *LabScribe2* **Main Window** will stop the recording when clicked. The **Record** button changes to **Stop** 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, *LabScribe2* will stop recording automatically after a predetermined time (in seconds) that has been entered into the text box.

Pausing Display
It is sometimes desirable to pause the scrolling display of data while at the same time continuing to record data. In *LabScribe2*, this is accomplished by clicking the **Pause Display** button in the lower right hand corner of the **Main Window**.

Preview Mode
It is possible to preview or monitor data acquisition without saving the data to disk. This is accomplished by clicking on the **Preview/Record** icon in the lower left hand corner of the **Main Window**. When *LabScribe2* is operating in **Preview** mode, a red "X" covers the icon and the

acquisition screen is dimmed. Clicking the icon again will allow the recorded data to be saved. It is important to remember to disable **Preview** when you want to save the data to disk. No data acquired in **Preview** mode can be saved.

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. The need to record faster events was solved by the introduction of the oscilloscope. An oscilloscope is able to take a fast, brief "snapshot" of an event, but continuous chart-like recording is sacrificed.

To effectively use **Scope** mode in *LabScribe2*, the length of the snapshot (display time) and the sampling rate need to be set to visualize the event with accuracy. Events that are best captured using **Scope** mode are brief and repetitive. Action potentials and other neurophysiological events 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. If the "snapshot" occurs too soon or too late, it may not show the event of interest. The acquisition of data at the right moment requires a proper trigger. In **Scope** mode either of the previously discussed triggering methods, 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**, which is accessible in the **Edit** menu in Windows and the **LabScribe2** menu on the Macintosh.

Data acquired in **Scope** mode can be viewed in either **Chart** or **Scope** format. Switching between formats is done by changing the acquisition mode in the **Channel Page** of the **Preferences Dialog**. When viewing **Scope** data in **Chart** format, each sweep is treated as a block of data. When viewed in **Chart** mode, sweeps are laid end to end, in the order that they were acquired, divided by block separation lines. Data can be continuously scrolled.

Data recorded in **Chart** mode can be viewed in **Scope** format. Each data block is treated as one sweep.

Set Up the Software

To program *LabScribe2* to record in **Scope** mode:

• Select **Preferences** from the **Edit** menu in Windows, or the **LabScribe2** menu on the Macintosh. Open the **Channel** page of the **Preferences Dialog**.

• Select the appropriate **Scope** acquisition mode. The acquisition mode can be set to either **Repetitive**, **Multiple** or **Averaged**.

• Set the **Start** mode that will begin the recording of the sweep. The available **Start** modes are the same as those in **Chart** mode:

  • **User**: Recording starts when the user clicks **Record**.

  • **Trigger**: Recording starts with a signal from an external TTL source connected to the **Trigger** input of the A/D Converter. Because the **Trigger** input is being monitored for this signal, it is possible to see data occurring before the signal by setting the amount of time in the **Pretrigger** text box.

  • **Channel 1, 2, 3,** or **4**: Recording starts when a predetermined data conditon is met on the selected channel. The user enters the low and high threshold values in the appropriate boxes, and the **Pretrigger** time (if desired) in the **Pretrigger** text box.

• **Timed**: Recording starts after an amount of time programmed by the user in the **Wait (sec)** box.
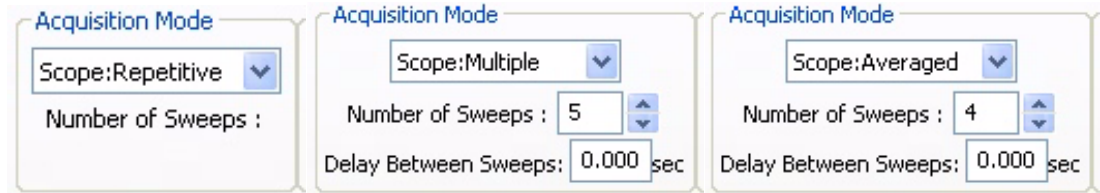
The Scope recording mode setup controls.

Repetitive Mode When the **Record** button is clicked in **Repetitive** mode, each new sweep overwrites the previous sweep. While digital oscilloscopes can save a number of sweeps, analog 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, *LabScribe2* 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 of the **Preferences Dialog**. 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.

After recording, individual sweeps can be selected and viewed by clicking on the corresponding sweep number in the **Sweep Selection Bar** at the bottom of the screen. The RIGHT and LEFT arrow keys on the computer keyboard can be used to cycle through the sweeps on the **Sweep Selection Bar**. Right-clicking on a sweep number opens a drop-down menu allowing that sweep to be titled or deleted.
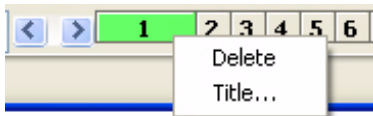
Sweep Selection Bar and drop-down menu.

Averaged Mode When the **Record** button is clicked in **Averaged** mode, *LabScribe2* will record the user-specified **Number of Sweeps**, but will only save and display one sweep representing the average of all the individual recorded sweeps.

Sweep Length When operating in **Scope** mode, *LabScribe2* 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 box of the **Channel** page from the **Preferences Dialog**. By choosing **Timed** as the **Stop Mode**, the sweep length (in seconds) can be entered into the **Stop** mode edit box. Although used less frequently as a way to stop recording in **Scope** mode, each sweep can be ended manually by setting the **Stop** mode to **User** and clicking the **Stop** button when the sweep has acquired the desired data.

Sampling Rate   The optimal sampling frequency in **Scope** mode can be determined by the same procedure used in **Chart** mode. First, find the quickest 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 0.00035s. If you substitute 0.00035s for the rise time in the following equation:

$$0.159/RiseTime = Bandwidth$$

$$(0.159/0.00035) = 454$$

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 uses only 500 data points. Since *LabScribe2* can accommodate up to 1,000,000 points per screen while recording, there is substantial room available for longer individual sweeps or a faster sampling rate.

Saving Your Data

Every software manual has a section on the importance of backup and saving. *LabScribe2* is no different. **SAVE YOUR DATA!**

Consider that the data acquisition process imposes unique demands on the task of saving data. A word processor or spreadsheet document can be saved anytime that the user thinks about it. Most applications can also save data automatically, at regular intervals, and in the background.

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

Whenever it's practical to do so, when recording is stopped, the user should instruct *LabScribe2* to save the current data to disk. As a safeguard, *LabScribe2* 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 are preserved. When the *LabScribe2* software is reopened after such an event, *LabScribe2* will ask the user if they want to recover the data. If the answer is yes, the backup file is recovered and data are preserved. To permanently save the data, a new file should be created by using the **Save As** function before recording any new data.

Chapter 4: Creating Your Own Preferences and Settings

Overview

In additon to using *LabScribe2's* pre-loaded preferences and settings, it is possible to modify or create your own **Preferences** and **Settings**.

The Preferences Dialog

The **Preferences Dialog** can be accessed from the **Edit** menu in Windows and the **LabScribe2** menu on the Macintosh. There are six tabbed pages in the **Preferences Dialog**. Details concerning the setup and use of many **Preferences** can be found in the relevant chapters of this manual.

When setting up **Preferences** and **Settings** for a new lab exercise, starting from the default values will make the process easier, as you will not be constrained by changes that have been made for other exercises. It is also possible to make the process easier by finding a similar experiment from among the *LabScribe2* experiments, one that uses a similar system or technique, and use those settings as a starting point.
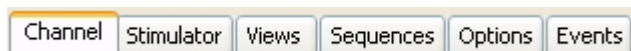
The pages of the Preferences dialog window.

Channel Page Many of the display parameters for each channel, as well as the recording parameters for data collection channels, and the function of each computed channel, can be set on the **Channel** page of the **Preferences Dialog**.
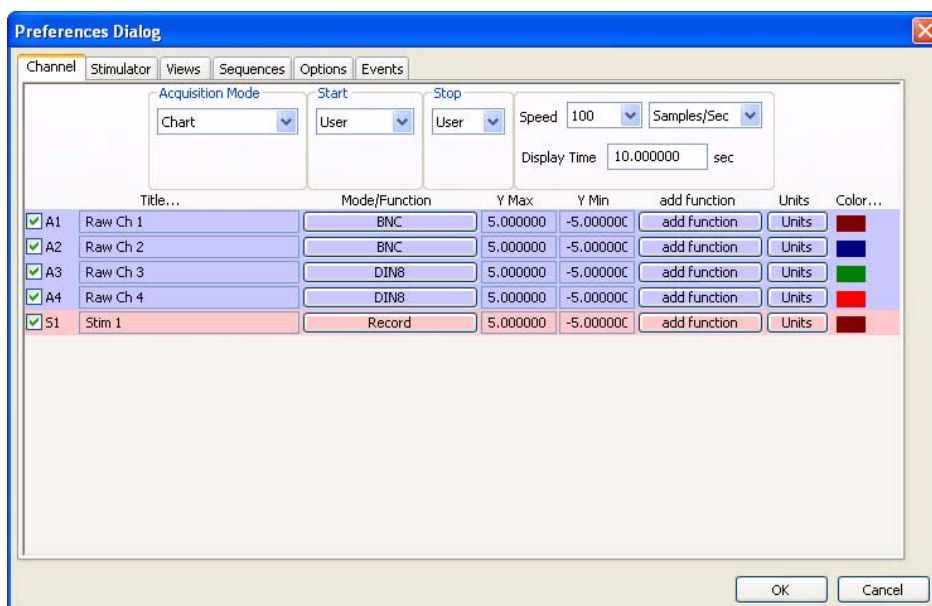
The Channel page of the Preferences Dialog.

Some of the **Preferences** set on the **Channel** page apply to all channels (refer to the discussion beginning on page 36 of **Chapter 3: Acquisition**):

- **Acquisition Mode**: Sets the type of acquisition: **Chart**, or one of the three **Scope** modes.

- **Start**: Sets the parameters that start the recording. The user can start the recording manually, recording can start a pre-determined amount of time after **Record** is clicked, or the start of recording can be triggered by an external event or by an event or condition on another channel.

- **Stop**: Sets the parameters to stop recording. Recording can be stopped manually by the user or after a pre-determined amount of time after recording starts.

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

- **Display Time**: Sets the time and the amount of data displayed on one 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 (titles can also be edited in the individual **Channel Menus** of the **Main Window**).

- **Mode/Function**: For raw data channels, clicking the **Mode/Function** button displays the input choices. These vary depending on the specific *iWorx* data acquisition unit. The filters applied to hardware channels with bioamplifiers can be selected, or the BNC or DIN8 input can be selected. On computed channels, clicking the **Mode/Function** button will display a list of the functions that can be applied to that channel instead of the one currently applied. Despite changing the function, it will still be computed from the same channel as the previous choice unless configured otherwise. Refer to **Chapter 5: Computed Channels** for a discussion of the functions available and how to configure them.

- **Y Max, Y Min:** Sets the maximum and minimum Y-axis values. These values can also be set in the **Scale** sub-menu of the **Channel** menu, or by right-clicking on the Y-axis. It is important to remember that each *iWorx* data acquisition unit is capable of recording a certain range of amplitude values, and not to set **Y Max** and **Y Min** outside these values.

- **add function:** Adds a computed function to a channel. **Add function** opens an additional channel that will display the function computed from the data in the chosen channel. The **add function** control can also be found on the individual **Channel Bars**. Refer to **Chapter 6: Computed Channels** for a discussion of the functions available.

- **Units**: Used to convert two raw data values in volts to the corresponding calculated values in units appropriate to the data being recorded. For more information refer to the **Units Conversion** discussion beginning on page 13 of **Chapter 1: The Display**.

- **Color...**: Sets the color of the channel's trace. This can also be set from each individual **Channel Menu.**

Stimulator Page Some *iWorx* data acquisition units contain a Digital to Analog converter that can function as a stimulator for use in experiments on excitable tissues. The stimulator(s) can be configured on the **Stimulator** page of the **Preferences Dialog**. There are six stimulator protocols: **Pulse**, **Train**, **Constant**, **Step, Ramp** and **Triangle**. The appropriate parameter values for each of these modes can be preset and made part of a settings file. Refer to **Chapter 5: The Stimulator** for information on programming the stimulator.

Views Page *LabScribe2* can display various arrangements of the channels in the **Main** and **Analysis Windows**. Refer to page 11 of **Chapter 1: The Display** for information on configuring **Views**.

Sequences Page   The operation of the internal stimulator and, on *iWorx* data acquisition units with digital outputs, some external devices can be automated by building a sequence of events that is triggered by selecting the **Sequence** button on the **Toolbar** in the **Main Window**. Refer to the **Sequences** sections of **Chapter 5: The Stimulator** (starting on page 59) and **Chapter 9: Digital Input and Output** (starting on page 133) for information on building **Sequences**.

Options Page   Various aspects of the display are configured on the **Options Page**. Preferences configured here are saved and are applied to both the current and future files.
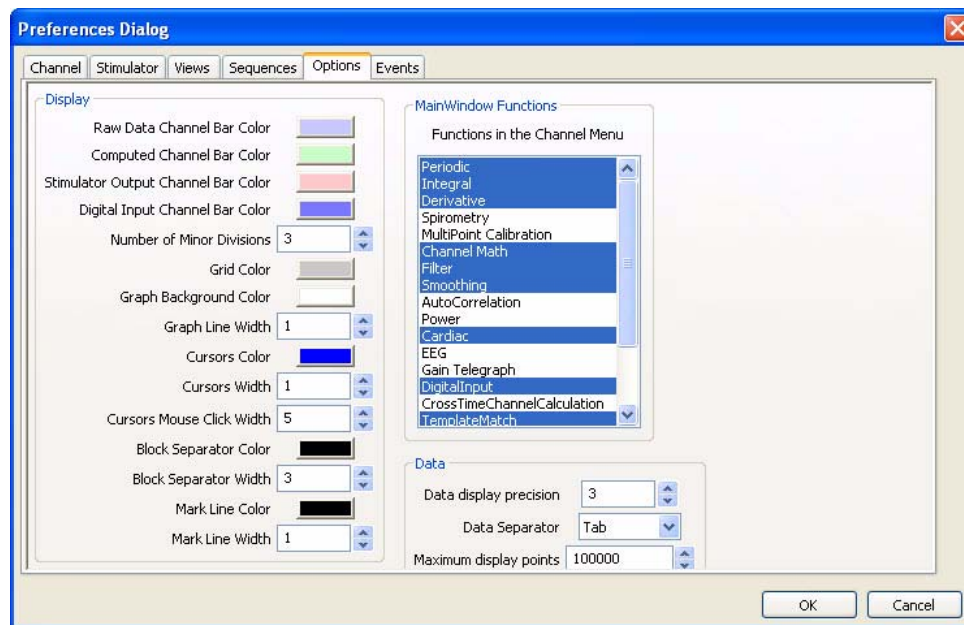
Options page of the Preferences Dialog.

The **Display** options configured on the **Options** page include:

- **Channel Bar colors**: The color of the raw data, computed channel, stimulator output and digital input **Channel Bars** can be chosen.

- **Grid characteristics**: Sets the color of the grid lines in the channel windows, as well as the number of minor grid lines between y-axis numbers.

- **Graph parameters**: It is possible to set the background color of the channel windows and the width of the trace line.

- **Cursor characteristics**: The cursor color and width can be set, as can the cursor mouse click width.

- **Block separators**: The width and color of the line separating recording blocks can be set.

- **Mark characteristics**: The mark line color and width can be changed.

The **Main Window Functions** box allows the user to select the functions that appear in the **add function** list accessed in the **Main Window**.

The following **Data** characteristics can be set on the **Options** page:

- **Data display precision**: Sets the number of digits of precision displayed.

- **Data Separator**: Set the type of data separator to be used for exporting data to the **Journal**.

- **Maximum display points**: The maximum number of data points that can be displayed on a screen of data. The default value is 100,000 points. This can be increased on the **Options** page, but it should be noted that files with more data points, and a faster sampling speed, take up more of the computer's memory.

Events Page   As *LabScribe2* acquires data, it is aware of the value of each data point as it happens. It is possible to instruct the software to watch for values above or below a specified level and have *LabScribe2* advise the user when such conditions are met. In *LabScribe2*, such an occurrence is called an **Event**. These **Events** can trigger output **Sequences.** Refer to **Chapter 9: Digital Input and Output** for a complete discussion of **Sequences**.
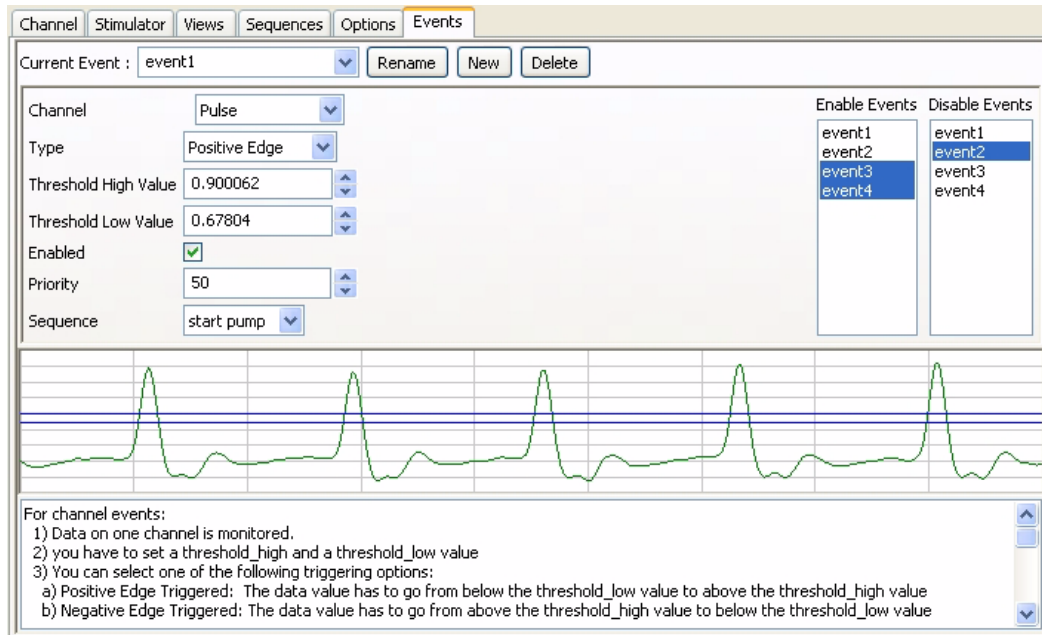
Controls for Events programming.

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

Channel Events   In the detection of **Channel Events**, one channel is monitored for **Events** that meet designated criteria. To set up *LabScribe2's* detection of an **Event**:

- In the **Channel** box, choose the channel to be monitored.

- Choose the **Type** of **Event** detection.

   - **No Triggering**: The detection of the **Event** will not trigger the initiation of a sequence.

   - **Positive Edge Triggered**: The data have to pass from below the **Low Threshold** to above the **High Threshold** in order to be detected as an **Event**.

   - **Negative Edge Trigger**: The data have to pass from above the **High Threshold** to below the **Low Threshold**.

   - **In Window**: An **Event** is detected if the data values enter the window between the **Low** and **High Thresholds** and remain there.

   - **Out of Window**: An **Event** is detected if data previously contained within the **Threshold** window (between **Low** and **High Thresholds**) move outside the window.

- The positions of the **Low** and **High Thresholds** are set depending on the type of data being recorded and the type of triggering that has been set. The thresholds can be chosen by entering data values in the **Low** and **High Threshold** boxes, or setting the values by moving the threshold lines in the graphical data sample.

- **Enable** event detection by checking the checkbox.

Timed Events　　**Timed Events** trigger a sequence after a designated amount of time has passed. To set the criteria for the detection of a **Timed Event**:

- Choose **Timed** as the **Type** of **Event**.

- Set the **Time** (in seconds) that should pass before **Events** are detected, and after a previous **Event** if more than one is programmed.

- Set the **Count**, or the number of events to be detected. Set the **Count** to zero for continuous **Events** occurring at the programmed **Time** interval.

- **Enable** the event detection by checking the checkbox.

Event Priority　　The **Event Priority** is set in relation to other **Events**. **Sequences** triggered by **Events** inherit their priority from the triggering **Event**. A higher priority **Event** can stop a lower priority **Sequence**, but a lower priority **Event** cannot stop a higher priority **Sequence**.

If a **Sequence** is manually triggered by the user, the trigger is considered a **User Event** with a **priority** of 50. Any **Event**s with a higher priority than 50 will interrupt a user initiated **Sequence**, while **Events** with a lower priority than the **User Event** cannot interrupt a user initiated **Sequence**.

An **Event** can start in an enabled or disabled state. A disabled **Event** is ignored, but it can be enabled by other **Events**. The **Enable Events** and **Disable Events** boxes determine which **Events** enable or disable other **Events**. The **Event** being configured will enable **Events** highlighted in the **Enable Events** box, and will disable **Events** in the **Disable Events** box, regardless of relative priorities.

In the example shown on page 133 of **Chapter 9: Digital Input and Output**, the **Pulse** channel is being monitored for a positive threshold crossing from below 0.67804 to above 0.900062. The **Event** has a priority of 50 and is enabled to trigger the **Sequence** "**Start Pump**" when the triggering criteria are satisfied. This event will also enable "**event3**" and "**event4**", while disabling "**event2**".

The Settings Menu

LabScribe2 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 create and use **Settings**, 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 configured by using display controls and the options on the various pages of the **Preferences Dialog**. The settings contained in a file determine the number and titles of channels in the *LabScribe2* windows, the sampling speed, the units conversions, the stimulator settings, and more.

LabScribe2 experiments use preconfigured **settings files**. Users can modify these settings or design their own **settings groups** and **files**. They can also associate their own **settings 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**. To create a **settings file**:

- Configure *LabScribe2* with the settings necessary to do the experiment. For example, go to the **Channel** 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. Stimulator settings, sequence outputs, a specific display arrangement, and even options like the colors of **channel bars** and traces can be set through the pages in the **Preferences Dialog**. Refer to the first part of this chapter for more detailed instructions.

- Each experiment can be assigned an aim and a difficulty level through the **Document Properties** command in the **File** menu.

- Click on the **File** menu and choose **Save As.** A **Save File** 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 in Windows or the **Documents** folder in your home directory on the Macintosh.

- 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. This will launch the **Settings Manager.** The **Available settings** pane lists the **settings files** available for addition to a **settings group**. The **Current Setting Group** lists the **settings files** in the curently loaded **settings group**.
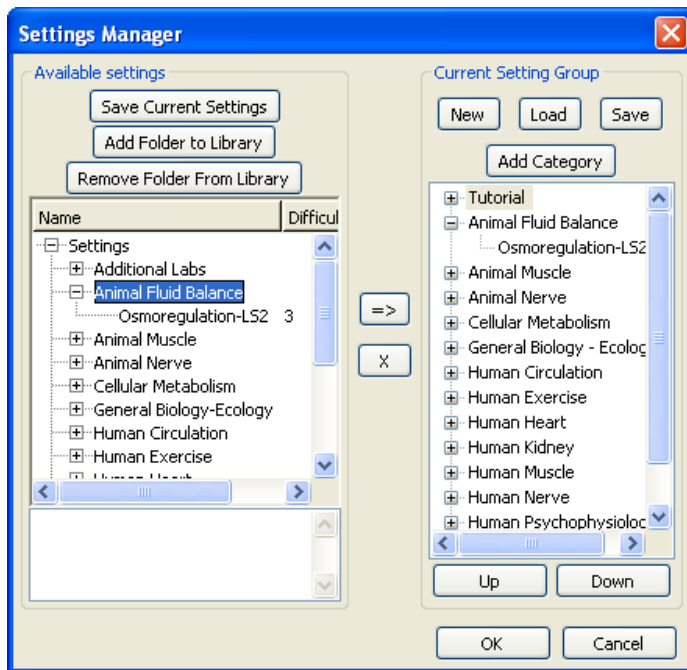
The Settings manager.

- By default *LabScribe2* will load the **Settings** folder in the *LabScribe2* folder into the **Settings library** pane. If you have previously created a folder for your own settings (as described above), 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 experiment can have a a difficulty level and an aim associated with it. These can be attached to the **settings file** in the **Document Settings** dialog accessed from the **File** menu. The difficulty level for each setting is visible in the **Difficulty** column. If you select a **settings file**, the aim of the exercise is visible under the **Available settings** pane.

- To create a new **settings group**, click the **New** button in the **Current Settings Group** area. This will remove the current group and clear the **Current Settings Group** box. Click **Save** to name the group (it will be given a .iwxgrp suffix) and place the **settings group** file in the folder with your settings.

- To load an existing group, click on the **Load** button in the **Current Settings Group** area. This will open a dialog where you can choose the **settings group** to load. **Settings group** files will have a .iwxgrp suffix. The **settings files** in the new **settings group** will now be displayed in the **Current Settings Group** window.

Settings files (*.iwxset) 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 at the root level. Settings added to the root level are listed in the **Current Settings Group** below the designated categories.

- The category order can be changed by selecting a category in the **Current Settings Group** and moving it up or down the list with the **Up** or **Down** buttons.

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. When a preconfigured *iWorx* **settings group** is loaded, a .pdf copy of each individual lab experiment is linked to the corresponding **settings file** in the **settings group**. When you select a **settings file** to do a particular experiment, a .pdf copy of the linked **helper file** (the experimental write-up) opens in Acrobat Reader. You now have a set of instructions, on the computer screen, to follow as you do the experiment. Alternatively, the .pdf file can be printed, edited in a .pdf editor, or uploaded to a content management system such as Blackboard.

Users can attach their **helper files** to their own or pre-existing **settings files**. Although *iWorx Systems* has chosen to attach .pdf documents to its **setting files**, any file (html, Microsoft Word, Open Office, 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**, 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, the html and the .jpg file to the same folder as the Tutorial.iwxset **settings file**. Name the .pdf file "Tutorial.pdf", the html file "Tutorial.html", and the .jpg file "Tutorial.jpg". Now when the Tutorial **settings file** is chosen in *LabScribe2*, the Tutorial.pdf, the Tutorial.html and the Tutorial.jpg files will be opened as well.

Chapter 5: The Stimulator

Overview

Some *iWorx* data acquisition units contain **Stimulators** capable of sending controlled current to the stimulating electrodes used in experiments involving excitable tissues like nerves and muscles. They do this through the use of internal Digital-to-Analog Converters (D/A Converters). The **Stimulator** page in the **Preferences Dialog** box controls the stimulator functions. It can be reached by selecting **Preferences** from the **Edit** menu in Windows or from the **LabScribe2** menu on the Macintosh. Using controls described in this chapter, output protocols can be built that deliver current to the stimulating electrodes.

Basic Stimulator Settings

The **Stimulator Preferences** page sets up the parameters and protocols of the stimulator output. It also sets the parameters of the **Stimulator Control Panel**, which opens directly beneath the **Toolbar** in the **Main Window** when the **Stimulator** icon in the **Toolbar** is clicked. The **Stimulator Control Panel** can also be opened by selecting **Stimulator** in the **View** menu. The **Stimulator Control Panel** allows the user to change many of the **Stimulator** output parameters directly from the **Main Window** during recording.
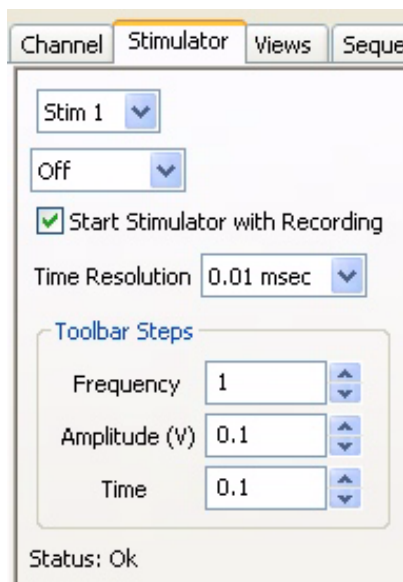
The Stimulator Preferences panel.

Some of the settings in the **Stimulator Preferences** are common to all stimulator output protocols:

• *iWorx Systems* manufactures A/D Converters with different D/A Converter configurations. In units with single or multiple D/A Converters, each D/A Converter can be controlled independently. The **Stimulator** selection drop-down box (set in this example to **Stim1**) allows selection of the **Stimulator** to be configured. For devices with multiple stimulators, an **Import Settings** button (not shown here) allows the user to copy settings from one stimulator to another.

• Six output modes can be selected using the **Mode** drop-down menu (set in this example to **Off**). The available modes are: **Pulse**, **Train**, **Constant** (voltage), **Step, Triangle** and **Ramp. Triangle** and **Ramp** modes are available only on some *iWorx* data acquisition units.

• The **Bipolar** checkbox (not shown here) allows the creation of bipolar pulses.

• By checking the **Start Stimulator with Recording** checkbox, the stimulator can be set to fire when recording commences.

• **Time Resolution**: The finer the **Time Resolution**, the shorter the maximum time duration for each parameter. For example, at a 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. Individual *iWorx* data acquisition units are capable of different **Time Resolutions**. The software configures itself depending on the hardware selected and will limit the available range of values for each parameter accordingly.

• **Toolbar Steps**: This selection determines the minimum step increments that the **Stimulator Control Panel** will use when the up/down buttons on the **Control Panel** are clicked. These are also the increments used by **Sequences** that program the stimulator output.

Pulse Protocols

Current pulses are delivered according to parameters set by the user in the boxes of the **Stimulator Preferences** page of the **Preferences Dialog** window.
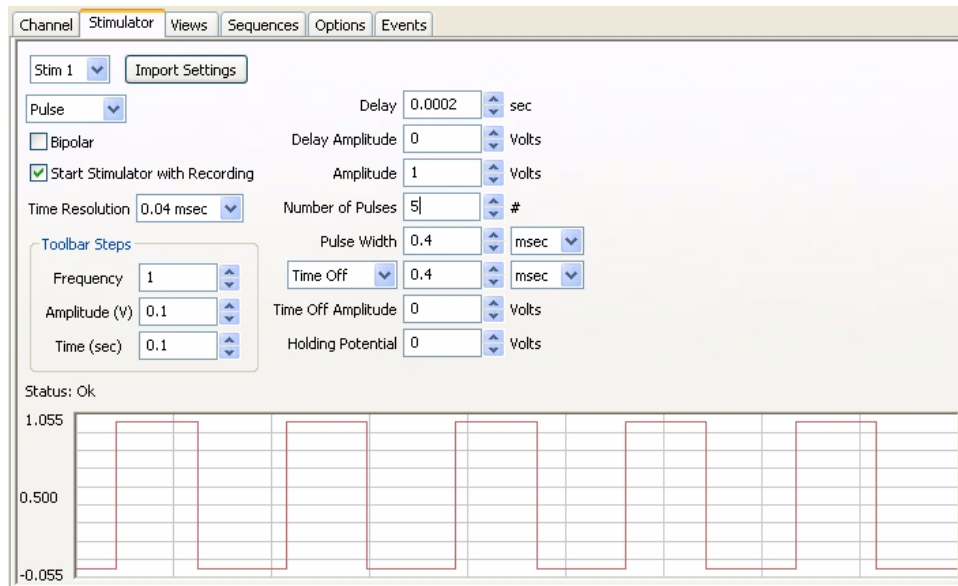
The Stimulator Preferences panel of the Preferences Dialog, configured to set up Pulse protocols.

Building Output Protocols in Pulse Mode

By setting the **Mode** drop-down menu to **Pulse**, the relevant parameters for **Pulse** protocols are displayed in the dialog. In order to understand how the protocols are created, it is necessary to define the terms used on the **Stimulator** page of the **Preferences Dialog**:

- **Delay**: This is the time between the clicking of the **Record** button to start recording and the first current pulse. This **Delay** is adjustable, and the maximum **Delay** varies with individual *iWorx* data acquisition units.

- **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** (described more completely later in the chapter).

- **Number of Pulses**: The total number of pulses that will be sent from the stimulator output.

*Note: In order to produce continuous pulses, the **Number of Pulses** must be set to zero.*

- **Pulse Width**: The pulse is the basic unit of an output protocol and it has two basic dimensions: **Amplitude**, as described above, and duration. The duration of the pulse is also called **Pulse Width**.

- **Pulse Frequency**: This is the number of programmed pulses that are delivered in one second. The maximum frequency that can be set is dependent on the pulse duration. If the pulses are too long for the chosen frequency, they will overlap, and the voltage output will be continuous and not pulsed. The **Pulse Width** cannot be longer than the inverse of the frequency (also called the period).

- **Time Off**: This is the interpulse duration, the amount of time between two consecutive pulses. **Pulse Frequency** is **1/(Pulse Width + Time Off)**.

- **Holding Potential**: This is a voltage that can be programmed to shift the resting membrane voltage of an excitable tissue (like a neuron). It is known as a holding voltage because it can be used to hyperpolarize the membrane potential of an excitable tissue and hold it at a level hyperpolarized enough to prevent "spiking". Individual *iWorx* data acquisition devices have different possible voltage output limitations, and the **Holding Potential** is limited to a value within this range.

To record and display stimulus pulses as in the examples that follow, the **Stimulator** channel should be turned on in the **Channel** page of the **Preferences Dialog** window.

*Note: While it is possible to record and display the stimulator voltage by routing the stimulator output to one of the hardware recording inputs, the **Stimulator** channel makes this unnecessary. Never connect both the positive (red) and the negative (black) banana outputs of a data acquisition 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.*

The following exercise will take you step by step through the construction of two different **Pulse** protocols.

PULSE MODE STIMULATOR EXERCISE

1. To construct and record some examples of stimulus pulses, select **Preferences** from the **Edit** menu. On the **Channel** page of the **Preferences Dialog**:

- Set the sampling **Speed** to 10000 samples/sec, and the **Display Time** to 0.5 sec.
- De-select the **Raw Ch 1-4** by removing any checks in the checkboxes on the left.
- Select the **Stim1** channel, check its checkbox, and set it to **Record** the stimulator.
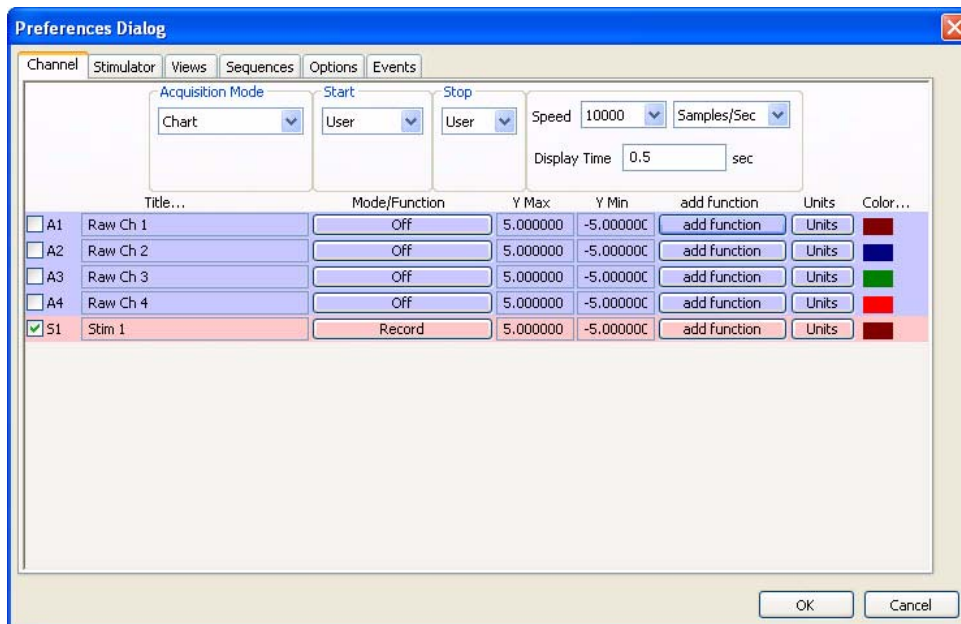
Channels page of the Preferences Dialog set for the pulse mode exercise.

2. Open the **Stimulator** page of the **Preferences Dialog** and select **Pulse** from the mode box in the upper left corner of the page.

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

- **Delay:** 0.1sec (an arbitrary value)
- **Amplitude:** 1V
- **Number of Pulses:** 0 (for continuous pulse output)
- **Pulse Width:** 5ms
- **Pulse Frequency:** 100 Hz
- **Holding Potential:** 0V

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

4. Click the **Record** button. The D/A Converter 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 **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.

5. Next, use the same settings, with one exception, to produce a short burst of pulses. Set the **Number of Pulses** to a number other than zero. If the number 10 is entered in this box, the output from the D/A Converter would wait for 100 milliseconds after the **Record** signal, deliver pulses of the same amplitude and duration as before, but stop when 10 pulses have been delivered. A graphical representation of the two examples above is pictured below.
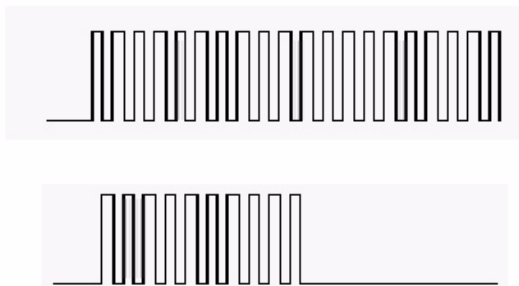
Continuous (top) and burst (bottom) of pulses that are the same amplitude, width, and frequency.

Train Mode

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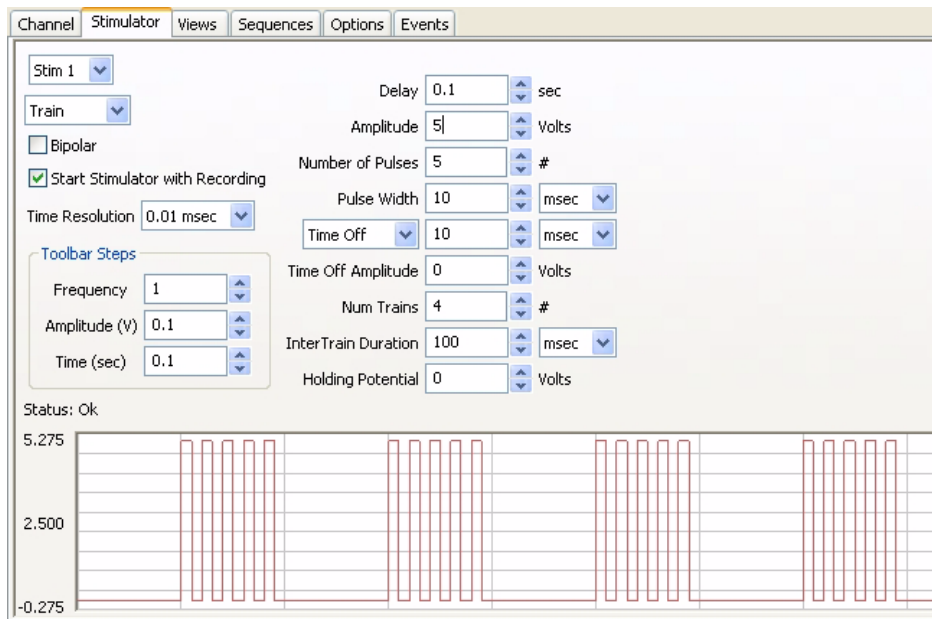
The parameter boxes on the Stimulator Preferences page for Train mode.

Building Output Protocols in Train Mode

In addition to the pulse parameters already described, configuring a pulse train requires that the following parameters also be set:

• **Num Trains**: Number of Trains.

• **InterTrain Duration**: The length of time between successive **Trains** or bursts.

TRAIN PROTOCOL STIMULATOR EXERCISE

In many applications, more than one **Train**, or burst 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.

1. To construct and record examples of the **Train** mode, select **Preferences** from the **Edit** menu. On the **Channel** page of the **Preferences Dialog**:

• Set the sampling **Speed** to 10000 samples/sec, and the **Display Time** to 5 sec.

• De-select the **Raw Ch 1-4** by removing any checks in the checkboxes on the left.

• Select the **Stim1** channel, check its checkbox, and set it to **Record** the stimulator.

2. Go to the **Stimulator** page of the **Preferences Dialog** and select **Train** from the **Mode** 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**: 5
- **InterTrain Duration**: 900ms

These settings will create a protocol that delivers a train of pulses every second. Each train will have 10 pulses with a frequency of 100 Hz. All pulses will have an amplitude of 1V and a **Pulse 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. Click the **Record** button. The D/A Converter 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 5 bursts have occurred.

5. If you only wanted the train or burst to repeat four times, the number 4 should be entered in the **Num Trains** box. The completed output protocol should look something like the figure below.

Burst of pulses separated by InterTrain Durations.

Constant Voltage Mode

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 continuously to the low voltage output of the *iWorx* data acquisition 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 the stimulator output can be increased or decreased in a step-wise manner.

The parameter boxes on the Stimulator page set for Step mode.

Building Output Protocols in Step Mode

In addition to the parameters already described, configuring a **Step Protocol** requires that the following parameters also be set:

Start Amplitude: The starting amplitude of the **Steps**.

Stop Amplitude: The ending amplitude of the **Steps**.

Number of Steps: The number of **Steps** in each "staircase", or protocol.

Step Width: The duration of each individual **Step**.

Time Off (or InterStep Width): Time between individual **Steps**.

Repeat Count: The number of times the **Step** protocol will repeat.

Interprotocol Duration: The duration of time between the termination of one protocol (one "staircase") and the initiation of the next.

The **Amplitude** of each individual **Step** in the protocol is determined by *LabScribe2* with the starting and ending **Amplitudes** and the **Number of Steps** entered on the **Stimulator** page. The equation used to perform this calculation is:

(**Start Amplitude** - **Stop Amplitude**)/**Number of Steps** = **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:

(Start Amplitude - Stop Amplitude)/Voltage Increment = 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 overall length of the protocol is determined by the **Step Width** and the time between steps, the **Time Off**.

VOLTAGE STEP MODE STIMULATOR EXERCISE

1. To construct and record an example of pulses in **Step** mode, select **Preferences** from the **Edit** menu. On the **Channel** page of the **Preferences Dialog**:

• Set the sampling **Speed** to 10000 samples/sec, and the **Display Time** to 2 sec.

• De-select the **Raw Ch 1-4** by removing any checks in the checkboxes on the left.

• Select the **Stim1** channel, check its checkbox, and set it to **Record** the stimulator.

2. Go to the **Stimulator** page of the **Preferences** dialog window and select **Step** from the **Mode** 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**: 100ms

• **Time Off**: 0

• **Repeat Count**: 1

• **Interprotocol duration**: 0

• **Holding Potential:** 0

4. These settings will create a waveform that starts at -1V and climbs to +1V in eight **Steps**. Each **Step** has a voltage increment of 0.25V and is 100ms wide.

5. Press the **Record** button. The resulting wave would be similar to the step-wise elevation of output amplitude seen in the figure below.
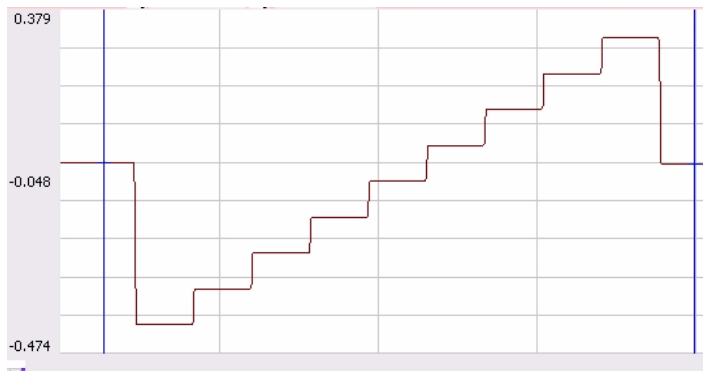
An example of a continuous step protocol.

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 the figure below.
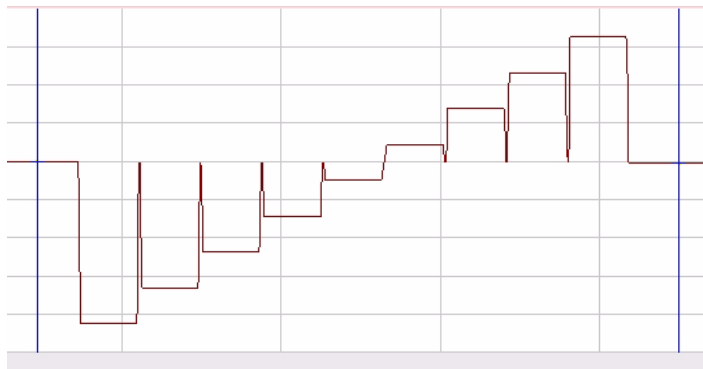
An example of a step protocol punctuated by returns to baseline between steps.

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 Mode

In **Ramp** mode, stimulator output increases or decreases periodically, as it does in **Step** mode, but the voltage change in **Ramp** mode is linear, as opposed to occurring in discrete steps. **Ramp** mode is only available in some *iWorx* data acquisition units.
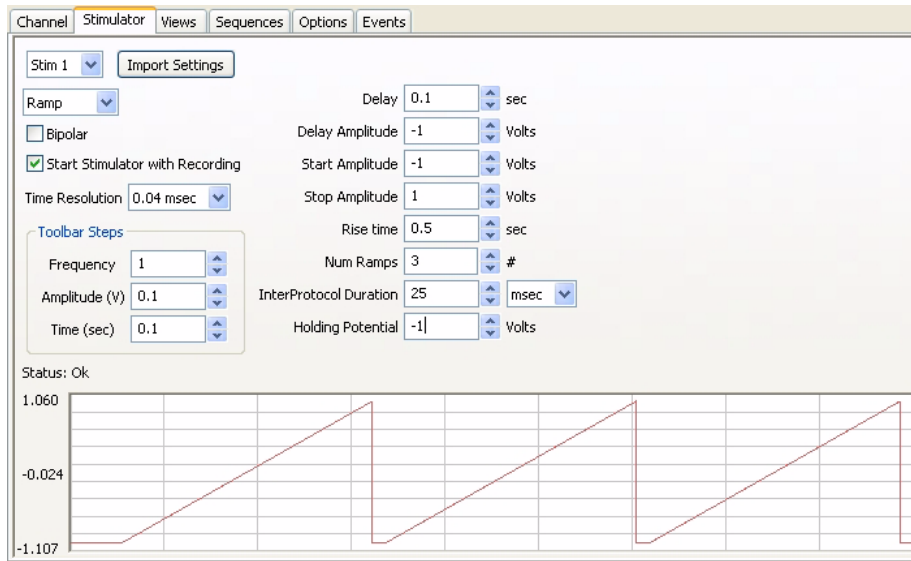
The parameter boxes on the Stimulator page set for Ramp mode

In addition to the **Pulse** parameters already described, configuring a **Ramp** protocol requires that the following parameters also be set:

- **Delay Amplitud**e: The **Amplitude** during the **Delay** period.

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

- **Num Ramps**: Number of **Ramps.**

Triangle Mode

Triangle mode is similar to the **Ramp** mode, in that the voltage changes are linear. In **Triangle** mode, each protocol consists of an ascending and descending **Ramp**. **Triangle** mode is available only on some *iWorx* data acquisition units.
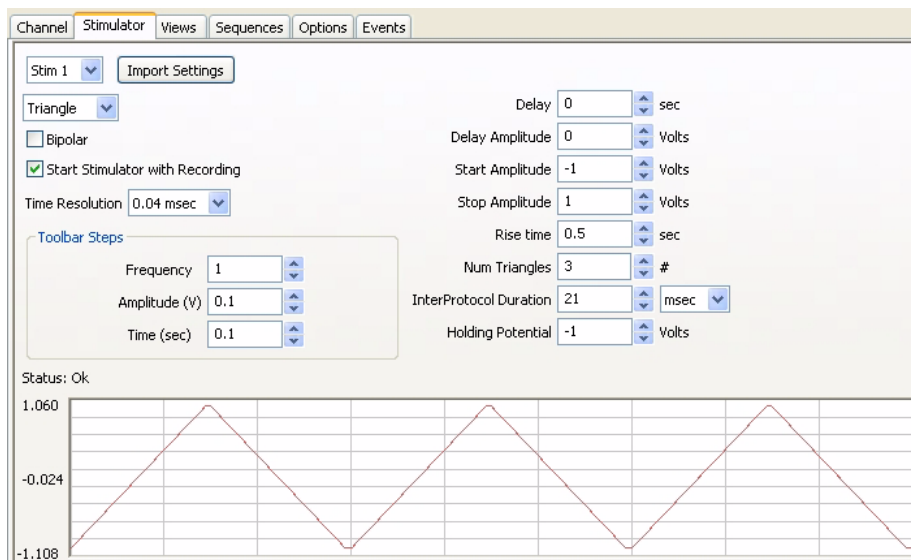
The parameter boxes on the Stimulator page set for Triangle mode.

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.

The Stimulator Control Panel

Clicking the **Stimulator** icon in the **Toolbar**, or selecting the **Stimulator Panel** item in the **View** menu will place a **Stimulator Control Panel** directly beneath the **Toolbar** in the **Main Window**.

The **Stimulator Control Panel** can be used to create a new protocol or change the parameters of a protocol that was created on the **Stimulator Preferences** page.

Each **Stimulator** mode has a unique **Control Panel.** The modes are described in the preceding section. The **Stimulator Control Panel** variations are illustrated and explained below.
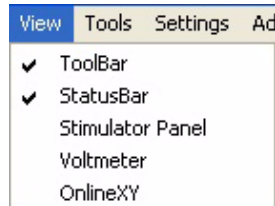
The View menu.

Pulse Mode Control Panel

The Stimulator Pulse Control Panel.

Amp: Amplitude of the stimulus pulse in Volts.

#pulses: Number of pulses (should be set to zero for continuous pulses).

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

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

HP: Holding potential is the voltage that pulses start from and return to.

Apply: Applies any changes to the **Stimulus** protocol made to the **Stimulator Control Panel**. This button changes to **Fire** when recording starts. Clicking **Fire** while recording will send the stimulus defined by the parameters in the **Control Panel**. It is necessary to click on **Apply** when the parameters are changed. Otherwise, clicking on **Fire** will send an unchanged stimulus.

Train Mode Control Panel
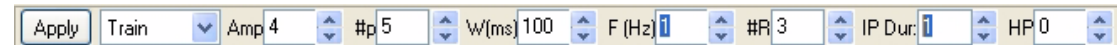
The Stimulator Train Control Panel.

Amp: Amplitude of the stimulus pulse in Volts.

#p: Number of pulses in each **Train**.

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

F(Hz): Frequency of the pulses in each train in Hertz (Hz).

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

IP Dur: InterTrain (or inter-protocol) duration.

HP: Holding potential is the voltage that pulses start from and return to.

Apply: Applies any changes to the **Stimulus** protocol made to the **Stimulator Control Panel**. This button changes to **Fire** when recording starts. Clicking **Fire** while recording will send the stimulus defined by the parameters in the **Control Panel**. It is necessary to click on **Apply** when the parameters are changed. Otherwise, clicking on **Fire** will send an unchanged stimulus.

Step Control Panel
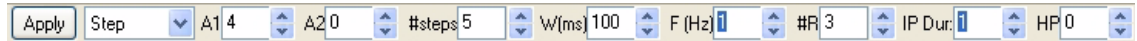
The Stimulator Step Control Panel.

A1: The **Starting Amplitude** of each protocol.

A2: The **Stopping Amplitude** of each protocol.

#steps: Number of **Steps** in each protocol.

W(ms): Width of each **Step** in milliseconds (ms).

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

#R: Number of **Step** protocols (or Repeat Count).

IP Dur: Inter-protocol duration (time between "staircases").

HP: Holding potential is the voltage that **Steps** start from and return to.

Apply: Applies any changes to the **Stimulus** protocol made to the **Stimulator Control Panel**. This button changes to **Fire** when recording starts. Clicking **Fire** while recording will send the stimulus defined by the parameters in the **Control Panel**. It is necessary to click on **Apply** when the parameters are changed. Otherwise, clicking on **Fire** will send an unchanged stimulus.

Constant Voltage Mode Control Panel

The Stimulator Constant Voltage Control Panel.

Amplitude: Amplitude of the constant voltage.

Apply: Applies any changes to the **Stimulus** protocol made to the **Stimulator Control Panel**. This button changes to **Fire** when recording starts. Clicking **Fire** while recording will send the stimulus defined by the parameters in the **Control Panel**. It is necessary to click on **Apply** when the parameters are changed. Otherwise, clicking on **Fire** will send an unchanged stimulus.

Ramp Mode Control Panel
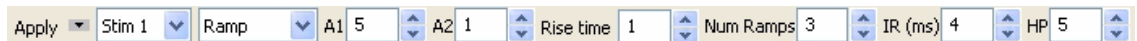
The Stimulator Ramp Control Panel.

Ramp mode is avalable only on some *iWorx* data acquisition units.

A1: The **Starting Amplitude** of each protocol.

A2: The **Stopping Amplitude** of each protocol.

Rise time: The time taken to go from the **Starting Amplitude** to the **Stopping Amplitude**.

Num Ramps: Number of **Ramps**.

IR: Time between **Ramps** (ms).

HP: Holding potential is the voltage that **Ramps** start from and return to.

Apply: Applies any changes to the **Stimulus** protocol made to the **Stimulator Control Panel**. This button changes to **Fire** when recording starts. Clicking **Fire** while recording will send the stimulus defined by the parameters in the **Control Panel**. It is necessary to click on **Apply** when the parameters are changed. Otherwise, clicking on **Fire** will send an unchanged stimulus.

Apply ▾ | Stim 1 ▾ | Triangle ▾ | A1 5 ▴▾ | A2 1 ▴▾ | Rise time 1 ▴▾ | # Trains 3 ▴▾ | IT (ms) 4 ▴▾ | HP 5 ▴▾

The Stimulator Triangle Control Panel.

Triangle mode is available only in some iWorx data acquisition units.

A1: The **Starting Amplitude** of each protocol.

A2: The **Stopping Amplitude** of each protocol.

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

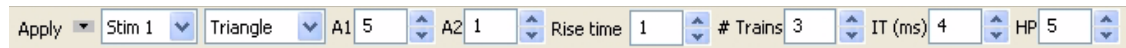**# Triangles**: Number of **Triangles**.

IR: Time between **Triangles**.

HP: Holding potential is the voltage that the **Triangles** start from and return to.

Apply: Applies any changes to the **Stimulus** protocol made to the **Stimulator Control Panel**. This button changes to **Fire** when recording starts. Clicking **Fire** while recording will send the stimulus defined by the parameters in the **Control Panel**. It is necessary to click on **Apply** when the parameters are changed. Otherwise, clicking on **Fire** will send an unchanged stimulus.

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 **Sequences Preferences**, a tabbed page in the **Preferences** dialog window, can be used to automatically change parameters of the D/A Converter stimuli.

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 stimulus 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.

The Sequences Preferences Page

The **Sequences Preferences** page is used to build **Sequences** that instruct *LabScribe2* to automate **Sequences** of commands. The user can name a **Sequence** and build a **Sequence** of pre-defined **States** that is triggered by selecting the **Sequence** from the **Toolbar** in the **Main Window**. Refer to the **Digital Input and Output** chapter for a more detailed discussion of building **Sequences** in *LabScribe2*. Included in the pre-defined **States** are a number of stimulator functions, which are used here to build a **Sequence** that will instruct the stimulator to produce increasingly larger pulses.

Building Output Protocols Using Sequences

In **Scope** mode, each sweep can record the response of the excitable tissue to a stimulus and the **Sequence** automatically increases the amplitude of the stimulus between each sweep. A step by step exercise follows.

SCOPE MODE SEQUENCE BUILDING EXERCISE

1. To construct an example of a **Sequence** designed to control the D/A Converter in **Scope** mode, select **Preferences** from the **Edit** menu in Windows or the **LabScribe2** menu on the Macintosh. On the **Channels** page of the **Preferences Dialog**:

• Set the **Acquisition Mode** to **Scope: Multiple**, and the **Number of Sweeps** to 10.

• Set the **Delay Between Sweeps** to 2 seconds.

- Set the **Start** mode to **User**.
- Set the **Stop** mode to **Timed** and the time to 0.03 seconds.
- Set the **Sampling Speed** to 20000 samples/second and the **Display Time** to 0.03 seconds.
- De-select the **Raw Ch 1-4** by removing any checks in the checkboxes on the left.
- Select the **Stim1** channel, check its checkbox, and set it to **Record** the stimulator.
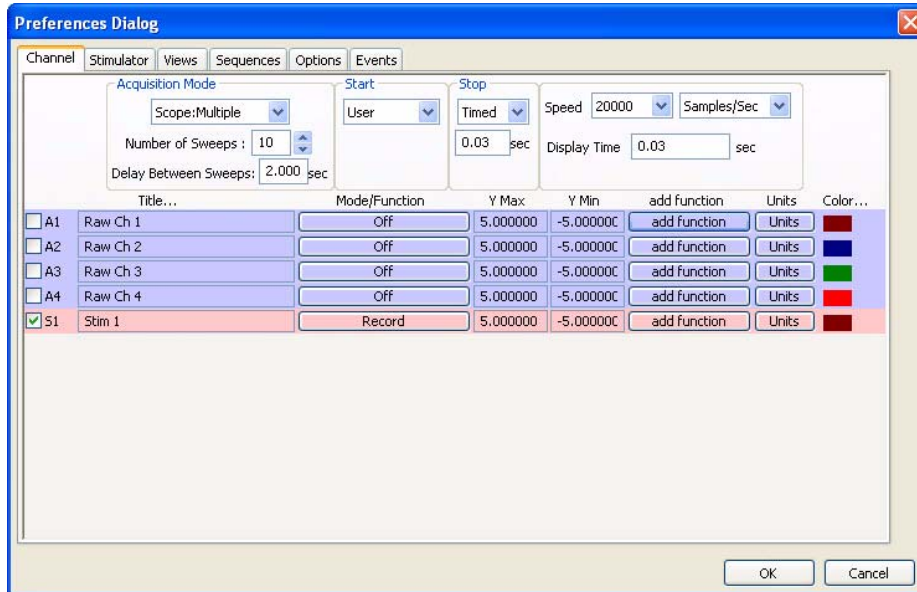
Channels page of the Preferences dialog window set for testing an Output Sequence.

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

Toolbar Steps:

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

Initial Parameters:

- **Delay**: 0.6s
- **Amplitude**: 0.25V
- **Number of Pulses**: 1
- **Pulse Width**: 0.1s
- **Time Off**: 0.1s
- **Holding Voltage:** 0V
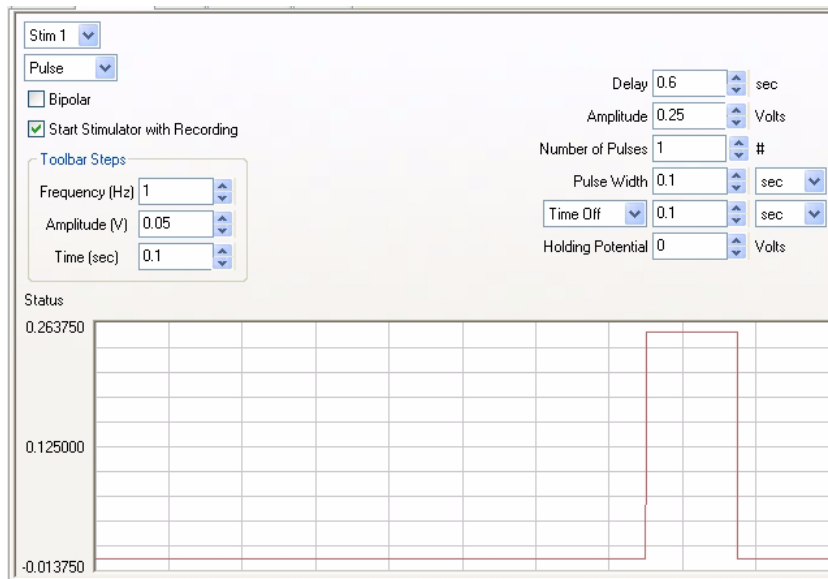
Stimulator page of the Preferences dialog window set for testing an Output Sequence.

3. On the **Sequences** page, add a **CAPRecruit** (for Compound Action Potential Recruitment) sequence to the list of current **Sequences**. Type the name **CAPRecruit** in the text box next to the **Current Sequence Label**, and click the **New** button.

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

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

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

7. Return to the **Main** window; select the **CAPRecruit** sequence from **Sequences** control on the **Toolbar**.

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

9. Click on the **Analysis** icon in the toolbar to view the sweeps recorded for each stimulus amplitude.
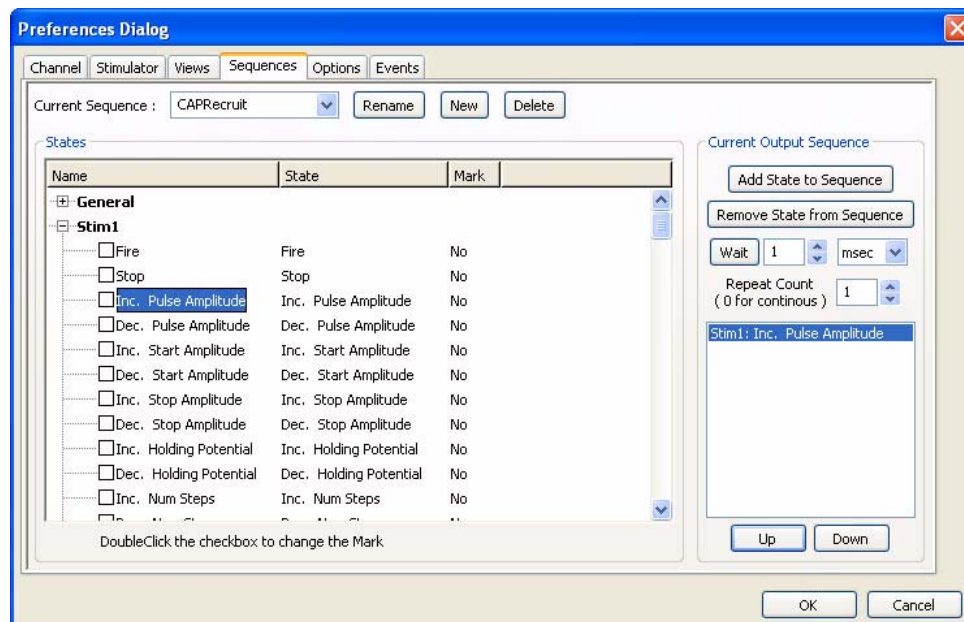
Output sequences page of the Preferences dialog window.

Chapter 6: Computed Channels

Overview

LabScribe2 makes it possible to 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.

Currently there are more than 50 computed functions included in *LabScribe2*. These functions can only be accessed in the **Main Window** or from the **Channel** page of the **Preferences Dialog**. They are called by using the **add function** menu and most can be used **online** or **offline**. When used **online**, the functions can operate at the top acquisition rate of the program, 100,000 samples per second.

Adding a Computed Function Channel

In the **add function** menu, the transforms are organized into categories. The functions within a category have similar setup requirements and are usually located in a submenu. Many of these functions are included in the **add function** menu by default, but some more specialized functions need to be added to the **add function** menu by modifying the **MainWindow Functions** box on the **Options** page of the **Preferences Dialog**, as illustrated and explained on page 43 in **Chapter 4: Creating Your Own Preferences and Settings**. All of the available functions are discussed below.

Periodic	▶
Integral	▶
Derivative	
Spirometry	▶
MultiPoint Calibration	
Channel Math	
Filter	
Smoothing	
AutoCorrelation	
Power	
Cardiac	▶
EEG	▶
Gain Telegraph	
DigitalInput	▶
CrossTimeChannelCalculation	
TemplateMatch	
PVLoop	▶
TimeShift Channel	
Event Marker	▶
Sonomicrometry	▶

The complete list of function categories.

To apply a function to a **Raw Data** or a **Computed Data** channel:

- Click the **add function** button on that channel's **Channel Bar**, or on the **Channel** page of the **Preferences Dialog**, accessed through the **Edit** menu in Windows or from the **LabScribe2** menu on the Macintosh. Choose the desired function category from the menu. Creation of a function channel through either method will add the channel to the list of channels on the **Channel** page of the **Preferences Dialog**.

- If the function requires user specified parameters, a setup dialog will open. This setup dialog is also accessible by clicking on the computed channel's **Mode/Function** label after the **Computed Function** channel has been created. **Setup Function** is at the top of the menu.

- The computed channel's function can also be changed by clicking on the **Mode/Function** button in the **Channel Bar** and choosing a new function from the **add function** list. The channel to which the function is applied remains the same, unless it is changed in the **Setup Function** dialog of the new function.

- Whether computed functions are performed **online** (in real time) or **offline** (after recording has stopped), their setup is the same.

Note: *Deleting the channel to which the function is applied will cause the function channel to have invalid data.*

The Functions

Periodic

Periodic functions operate on cyclic data to produce a graphical representation of how the functions vary with time. *LabScribe2* calculates these parameters with each cycle of the signal and displays the calculation graphically in a new data channel.

The Periodic Setup Dialog

Choosing any of the **Periodic** functions opens the **Periodic** data setup dialog. To make **Periodic** calculations on each cycle, the software must have a way of defining a cycle of data, and the cyclic criteria are defined in this dialog. As data are collected online or processed offline, *LabScribe2* begins the calculation by determining the maximum (**Max**) and minimum (**Min)** values in a given screen of data.

The program then finds the points on the recording where the trace repeatedly moves across a **Threshold** in a positive direction.The control for the **Threshold** level is set in the **Periodic** setup dialog. The **Threshold** level can be set at **X%** of the maximum value in any given screen of data or at an absolute value. By default, the **Threshold** 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, or by adjusting the horizontal threshold line in the dialog's data graph. *LabScribe2* can now define the time between these points as a **Period**. If the recorded data are very stable, it is possible instead to enter an absolute **Threshold** value.
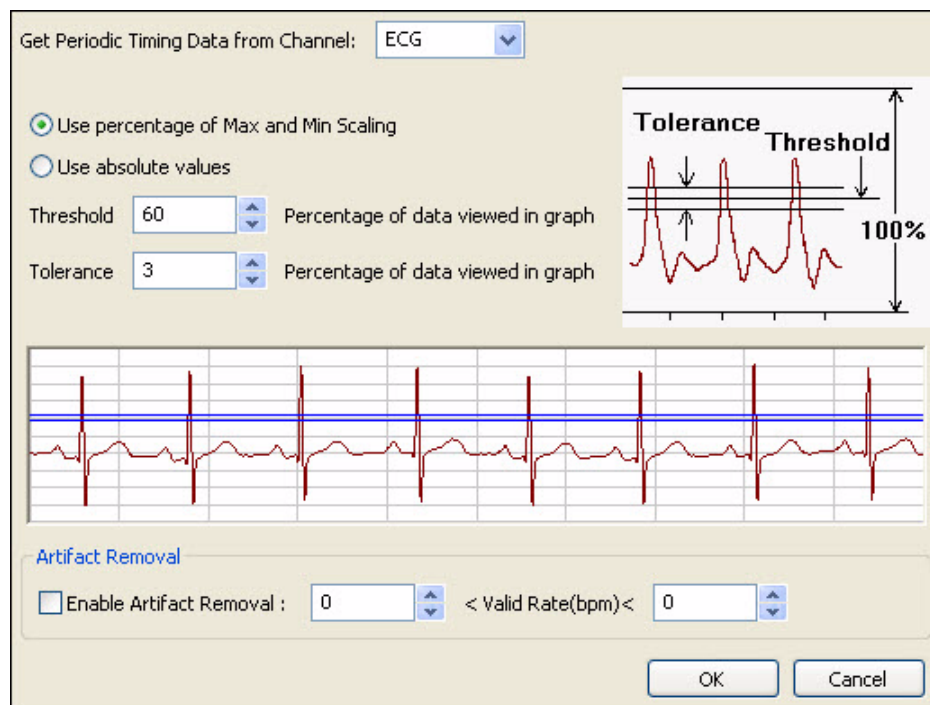
The Periodic Setup Dialog window.

Whether a relative or absolute **Threshold** criterion has been set, a second **Periodic** control known as **Tolerance**, or hysteresis, is also set 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 to 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.

LabScribe2 makes it possible to remove the effects of artifacts from the cyclic data. By enabling **Artifact Removal**, and entering the minimum possible rate and the maximum possible rate for the type of data being recorded into the two edit boxes, the software will ignore areas of the trace where artifacts of some sort create artificially high or low rates. For example, if ECG data are being taken with wrist electrodes, there may be places in the trace where the subject moved their fingers and the movement artifact temporarily overwhelms the ECG data, creating an artificially high heart rate to be recorded. If **Artifact Removal** is enabled, and the minimum and maximum heart rates are set at 50 and 180, values less than and greater than heart rates are likely to be, even during exercise, any sections of the recording registering higher or lower than these values will be ignored, and this section of the recording won't display the artificially high heart rate.

PERIODIC FUNCTIONS SETUP EXERCISE

1. To see **Threshold**, **Tolerance,** and **Artifact Removal** in use, record some pulse data, following the procedure in the pulse monitor **Tutorial** exercise. Flex your fingers vigorously for four or five seconds at some point in the recording. Stop the recording and **Autoscale** the data in a section of the record that doesn't include the finger flexing.

2. Apply the **Rate** function to the pulse channel by clicking on **add function** in the Pulse Channel Bar and choosing **Periodic** from the Function menu. Choose **Rate** from the **Periodic** submenu to open the **Periodic** functions setup dialog.

3. 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. Look at the sample of data in the dialog window and adjust the **Threshold** and **Tolerance** such that the **Threshold** and **Tolerance** lines clear the smaller waves. Do not enable **Artifact Removal**, and click **OK**.

4. Enter **Single Cursor Mode** and move the cursor to various regions of the recording and read the rate in the **Value Display** area of the **Rate Channel Bar**. The rates should be accurate in most areas of the trace. Move the **Cursor** into the section where you flexed your fingers, and read the rate in this section. It will probably not be accurate because the movement artifact will prevent an accurate assessment of rate in this location.

5. Open the **Periodic** setup dialog again by clicking on the **Mode/Function** button on the **Rate** channel, opening the **Functions** menu. Choose **Setup Function** from the top of the list. This will open the setup dialog. Enable **Artifact Removal** and enter minimum and maximum rates of 50 and 150 respectively. Click **OK**.

6. Move the cursor so that it is in the section where you flexed your fingers. The rate here should now read the same as the rate to either side of the flexing, and is a much more accurate appraisal of your heart rate.

7. Open the **Periodic** setup dialog again by clicking on the **Mode/Function** button on the **Rate** channel, opening the **Functions** menu. Choose **Setup Function** from the top of the list, opening the setup dialog.

8. Adjust the **Threshold** and **Tolerance** so that the **Threshold** and **Tolerance** lines pass through both the larger and smaller waves. Click **OK**.

9. Move the cursor around the **Rate** channel again, checking the rate in the **Value Display** area of the **Rate Channel Bar**. The rate should now be indicating approximately twice the actual heart rate. You can see why it is necessary to adjust the **Threshold** so it is greater than the maximum peaks of the smaller waves. If you had been determining rate from an ECG, it would have been necessary to adjust the **Threshold** so that it cleared the P and T waves in the recording, and passed through just the QRS complex.

Using the Periodic Functions

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 **add function** menu and a submenu appears. Select one of the functions in the submenu, opening the **Periodic** function setup dialog.

- Adjust and/or activate the parameters in the setup dialog as neccessary, using the information in the previous section as a guide.

- Click **Record** in the **Main Window** to begin recording.

- **AutoScale** the raw data, then **AutoScale** the calculated channel

- Click **Stop** when you have recorded the desired data.

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

```
Rate
Frequency
Period
CyclicMax
CyclicMin
CyclicMean
Max dV/dt
Min dV/dt
Mean dV/dt
RMS
Max-Min
Count
Delta P-P
```

The Periodic submenu.

- **Rate**: *LabScribe2* takes the **Period** in seconds and divides this value into 60. The result is a **Rate**, which is expressed in events 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 (cycles per second).

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

- **CyclicMax**: The program examines all of the data points in the current **Period** and finds the highest value.

- **CyclicMin**: The program examines all of the data points in the current **Period** and finds the lowest value.

- **CyclicMean**: The program examines all of the data 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. This will correspond to the steepest slope in the cycle.

- **Min dV/dt**: The program examines all of the derivatives in the current **Period** and finds the lowest value. This will correspond to the flattest area in the cycle.

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

- **RMS**: The program examines all of the data points in the current **Period** and finds the **Root Mean Square** value for all points.

- **Max-Min**: **Cyclic Max** - **Cyclic Min**.

- **Count**: Choosing **Count** opens a dialog in which the user can set a threshold over which events are counted and tallied from the beginning of the recording block.

- **Delta P-P**: Change in **Period** from the preceding cycle.

Integral　An integral is the area under a curve. The **Integral** function, as executed by *LabScribe2*, calculates a continuous sum of all the data points on a given channel that satisfy certain criteria, 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 established.
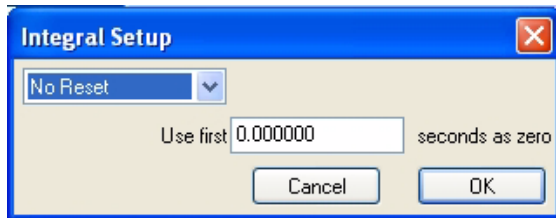
Integral Setup dialog.

Setting the baseline of the raw data channel to zero is important because 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.
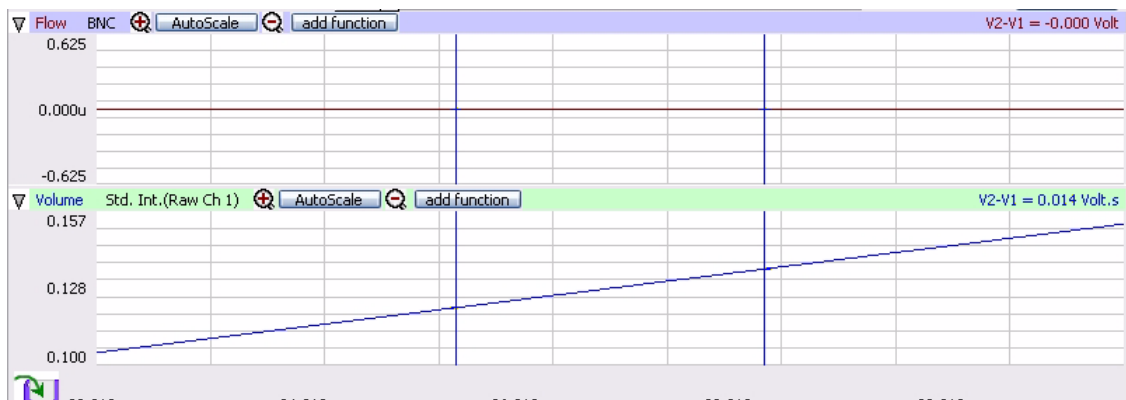
Zero

To determine where the zero-line is located, *LabScribe2* takes a value of zero volts as the zero-line. If real units, such as grams or mmHg are used, *LabScribe2* will take the zero units value as zero.

LabScribe2 also has a **Use First "N" seconds as zero** option. The data values for the first "N" 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 the figure below. The output of a respiratory flow sensor is the value displayed on the upper channel; the lower channel 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.
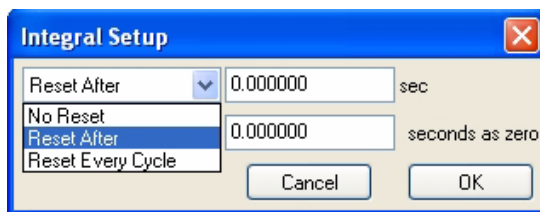

Integral without Zero.

In the second example, where the **Use First "N" 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.
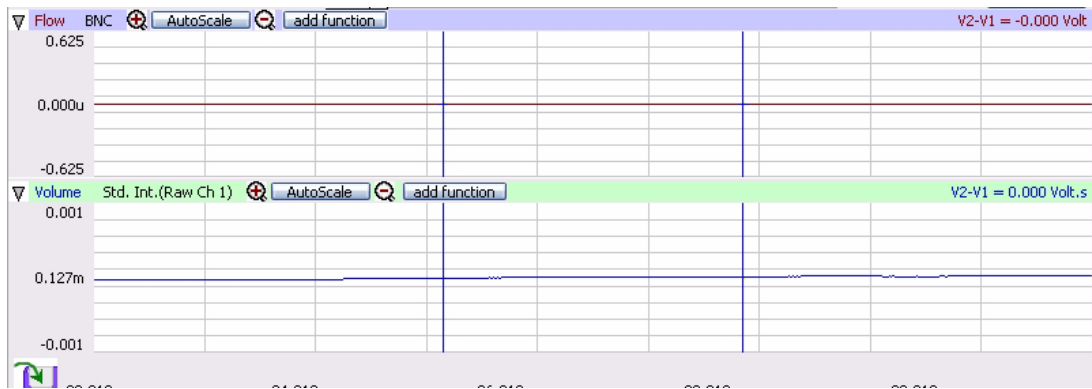
Integral with Zero.

Integral Setup dialog.

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.

Using the Integral Functions

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 and configure the **Integral Setup** dialog.

• Click **Record** in the **Main Window** to begin recording.

Derivative

The **Derivative** function calculates the derivative (slope) around 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.
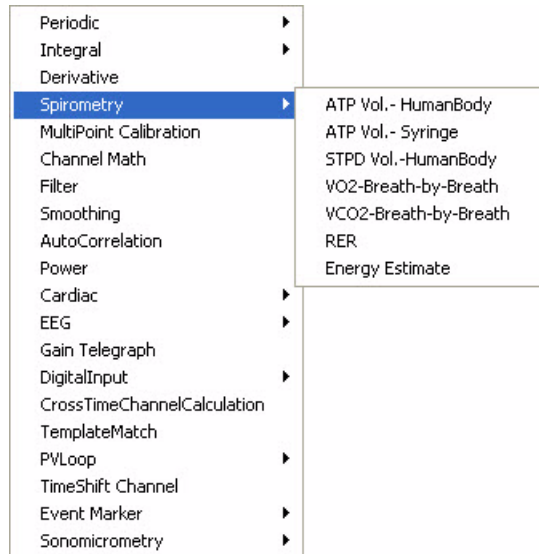
Using the Derivative Function

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 **Record** in the **Main Window** to begin recording.

Spirometry   The **Spirometry** volume functions use a specialized version of the integral function for use with *iWorx* spirometers. Spirometers, which measure respiratory volumes, are sensitive air flow sensors. *LabScribe2* integrates the sequential flow values measured by the sensor as the subject breathes and displays the air flow volume. If a gas analyzer is used, **VO$_2$**, **VCO$_2$**, and the **RER** can also be calculated. The advanced *LabScribe2* **Metabolic** module depends on the accurate recording of basic **Spirometry** functions.

Periodic	▶
Integral	▶
Derivative	
Spirometry	▶
MultiPoint Calibration	
Channel Math	
Filter	
Smoothing	
AutoCorrelation	
Power	
Cardiac	▶
EEG	▶
Gain Telegraph	
DigitalInput	▶
CrossTimeChannelCalculation	
TemplateMatch	
PVLoop	▶
TimeShift Channel	
Event Marker	▶
Sonomicrometry	▶

Spirometry submenu:
- ATP Vol.- HumanBody
- ATP Vol.- Syringe
- STPD Vol.-HumanBody
- VO2-Breath-by-Breath
- VCO2-Breath-by-Breath
- RER
- Energy Estimate

The Spirometry submenu.

The **Spirometry** functions are:

- **ATP Vol.-Human Body**: A measure of air flow from human respiration at ambient temperature and atmospheric air pressure.

- **ATP Vol.-Syringe**: An air volume measurement used in the calibration of the gas analyzer.

- **STPD Vol.-Human Body**: Respiratory air flow normalized to standard temperature.

- **VO$_2$-Breath-by-Breath**: Online continuous calcualtion of the rate of oxygen consumption.

- **VCO$_2$-Breath-by-Breath**: Online continuous calculation of the rate of carbon dioxide production.

- **RER** (Respiratory Exchange Ratio): **VCO$_2$/VO$_2$**.

- **Energy Estimate**: Mathematical estimate of energy expenditure based on rates of oxygen consumption and carbon dioxide production.

Using the Spirometry Functions

To apply a **Spirometry** function to a channel, click on the **add function** button in the **Channel Bar**, or on the **Channel** page in the **Preferences Dialog**. Select **Spirometry** from the **add function** menu.

Each of the **Spirometry** functions requires configuration in the appropriate dialog window.

The calibration dialogs for the volume functions (**ATP Vol.-Human Body**, **ATP Vol.-Syringe**, and **STPD Vol.-Human Body**) are quite similar.
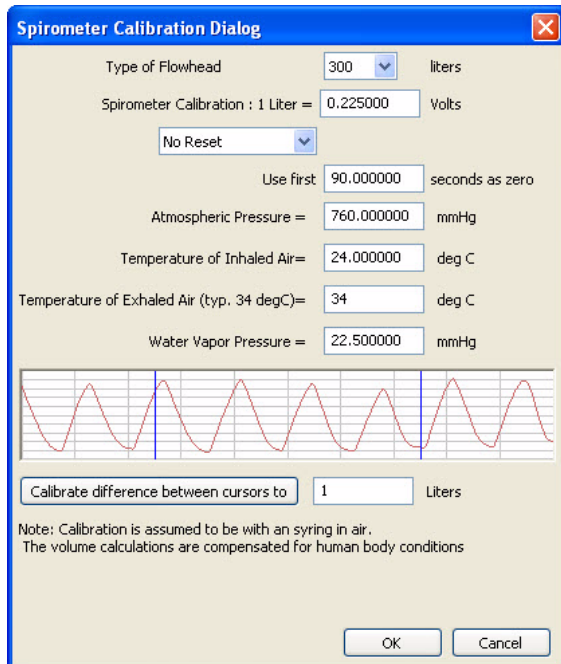
The Spirometry Calibration Dialog.

To configure the **Spirometry Calibration Dialogs**:

- Choose the type of flowhead being used.

- Each *iWorx* spirometer has a calibration value on it. This value can be entered in the **Spirometer Calibration** text box.

- If the specific **Spirometry Calibration Dialog** has entries for **Atmospheric Pressure**, **Temperature of Inhaled Air**, **Temperature of Exhaled Air**, **Relative Humidity**, and **Water Vapor Pressure**, enter these values in the appropriate text boxes.

- If a calibration syringe is being used and the user has recorded data where a known volume of air was pushed through the spirometer, these 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. *LabScribe2* 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 (which is based on an integral function) to display a volume change, even though no air is flowing through the sensor.

- The **Volume** integral can also be programmed 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 can drift up and out of view, as it reports correct values, unless a periodic reset is employed. Unless this drift is occurring, it is best not to reset the **Volume** calculation.

The calculation of **VO$_2$-Breath-by-Breath** or **VCO$_2$-Breath-by-Breath** functions is a very similar process. To set up the **VO2-Breath-by-Breath**, or the **VCO2-Breath-by-Breath** functions:

- Choose **VO$_2$-Breath-by-Breath** or **VCO$_2$-Breath-by-Breath** from the **Spirometry** submenu. This opens the appropriate **Metabolic Calculation Output Dialog**.
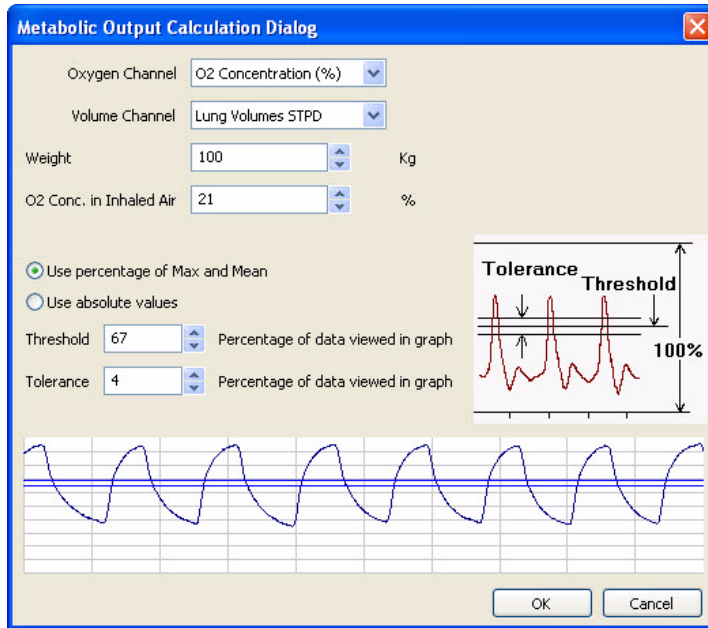
The Metabolic Output Calculation Dialog.

- Select the **Output Channel** (**O₂** or **CO₂**, depending on whether the calculation of **VO₂** or **VCO₂** is being configured) and a calibrated air flow volume channel. Enter the subject's weight and the percentage of O_2 or CO_2 in inspired air.

- Set the **Threshold** such that each data cycle passes through the **Threshold** lines.

- Click **OK**.

To set up the **RER** (Respiratory Exchange Ratio) function:
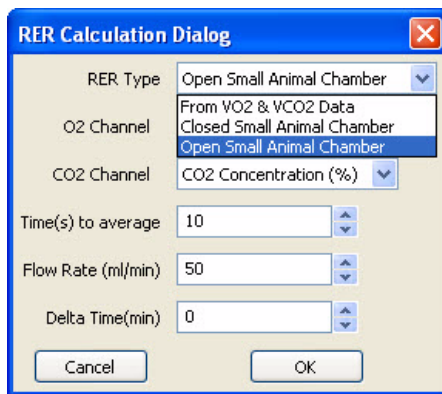

The RER Calculation Dialog.

- Select RER from the **Spirometry** submenu, opening the **RER Calculation Dialog**.

- Choose the source of the data: **From VO2 and VCO2 Data**, **Closed Small Animal Chamber**, or **Open Small Animal Chamber**. The **RER** is calculated from the changing percentages of oxygen and carbon dioxide in expired air. The setup menu also asks for the time intervals over which the values are averaged, and the flow rate through the mixing chamber, if appropriate.

- Select the **O₂** and **CO₂** channels.

- Choose the duration to average the values over.

- If appropriate, enter the flow rate.

- **Delta time**: Time taken for a 1% change in CO2 or O2 concentration. Set the cursors so that there is a 1% change between them in the relevant concentration, and use T2-T1 as **Delta time**.

- Click **OK**.

MultiPoint Calibration

MultiPoint Calibration can be used to perform both linear and non-linear calibration of sensors, transducers, amplifiers and other equipment. It can also be used to check if an existing sensor's output is linear.
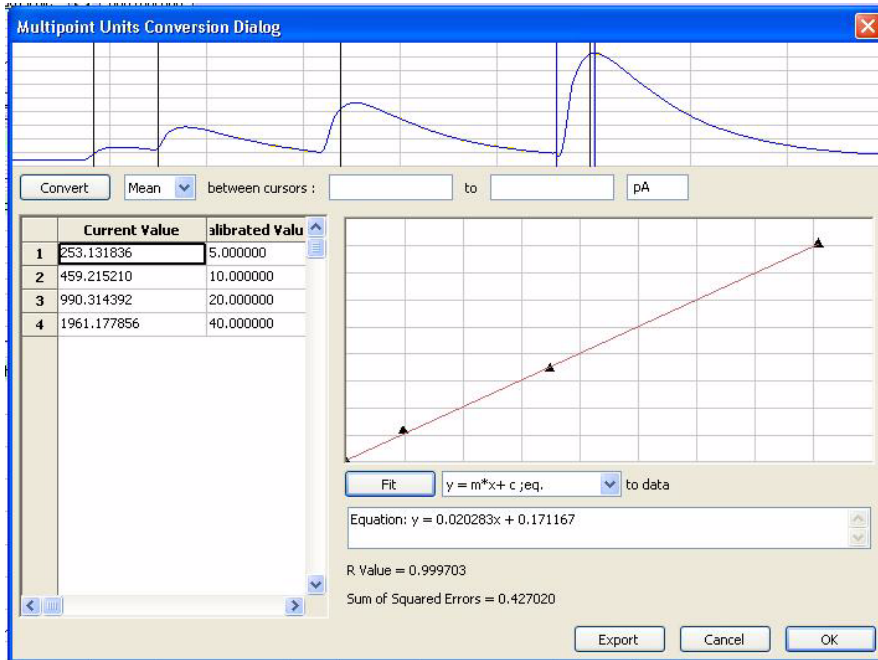
The Multipoint Units Conversion Dialog.

### Using the MultiPoint Calibration Function

To use the **MultiPoint Calibration** function:

- Place the cursors in the raw data graph, such that the **Mean**, **Max**, or **Min** value between the cursors, or the difference between the cursor values, is a calibrated value. The **Current Value** will be displayed in the text box following "between cursors".

- Enter the **Calibrated Value** in the next text area. Set the units in the last text area.

- Click the **Convert** button to add the **Current Value** and the **Calibrated Value** to the table. Repeat this procedure for all values that you want to calibrate.

- Once the table has been populated, you can select the equation you want to fit to the raw data and then click on the **Fit** button. *LabScribe2* will try to fit the selected equation to the values in the table. The fitted equation as well as the error is shown. If **MultiPoint Calibration** is being used to test the linearity of the sensor, the data from this window serves as confirmation.

- Selecting **OK** will apply the fitted equation to the raw data to get a calibrated output.

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. One of four variables (**A**, **B**, **C** and **D)** can be associated with a user-selected data channel.

As an example, to divide Channel 1 by Channel 2, click the **add function** button on either the Channel 1 or Channel 2 **Channel Bar**. Select the **Channel Math** option to open the **Channel Math Setup Dialog**. Select **A** to be Channel 1, and **B** to be Channel 2. In the **Channel** edit box, name the computed channel **A/B.** Every Channel 1 data point is divided by the corresponding Channel 2 data point and the resulting waveform is displayed in the **A/B** computed channel.
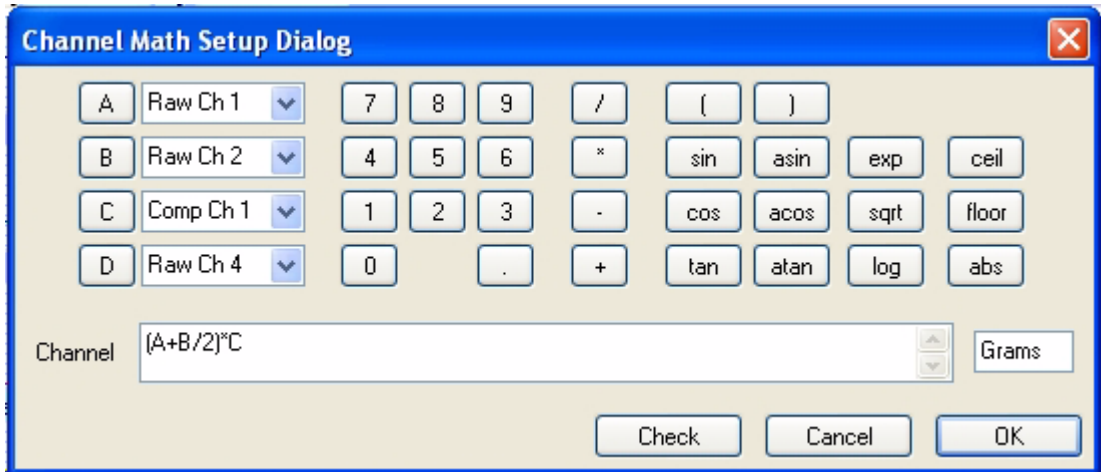
*In this Channel Math Setup Dialog, the output channel is equal to (RawCh 1 plus Raw Ch 2 / 2) * Comp Ch 1 in grams.*

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. There is also a **Check Expression** feature, which checks 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.

Using the Channel Math Function

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 list.

• Select the channels corresponding to **A**, **B**, **C** and **D.**

• Program the desired expression. Set the units if applicable, and click **OK**.

Filter

A digital filter can be applied to any channel in real time or to previously recorded data. *LabScribe2* 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 the **Hamming** function, *LabScribe2* 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, and the longer *LabScribe2* takes to calculate the function, which can slow down the display. Data points at the beginning and end of the filtered data (the first **Filter Order/ 2** and the last **Filter Order/2** data points) 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.
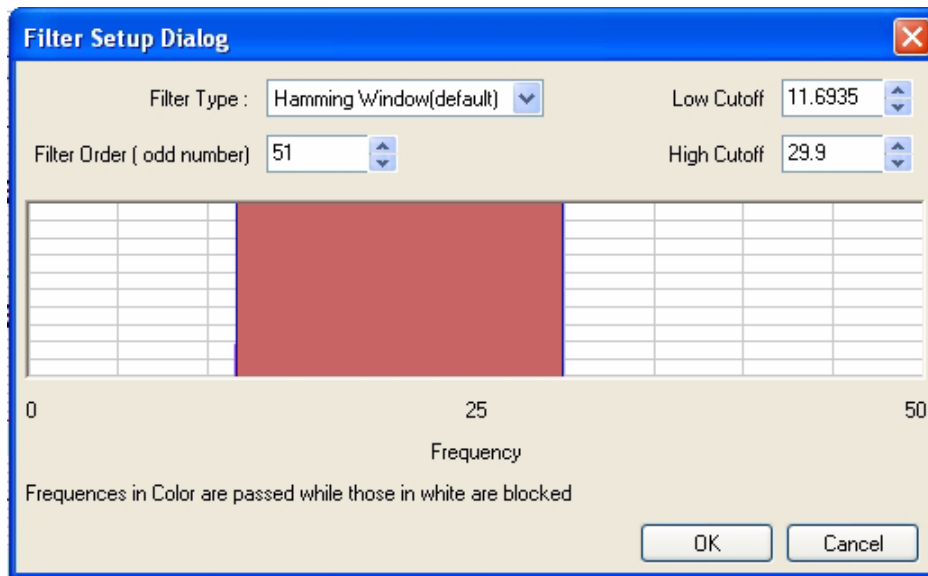
The digital (software) Filter Setup Dialog.

The graphic interface in the **Filter Setup Dialog** is straightforward. The colored area corresponds to the frequencies that are passed or allowed, 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 colored 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 illustrated in the figure above), 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 sampling speed.

Additional information about the digital filter functions can be found on page 32 of **Chapter 3: Acquisition**.

Using the Filter Functions

To apply a digital filter to a channel:

- Click **add function** in the **Channel Bar**.
- Choose **Filter** from the function list, opening the **Filter Setup Dialog**.
- Select a **Filter Type** from the drop-down menu.
- Choose a **Filter Order**.
- Adjust the boundaries of the colored area of the graph, or enter the desired values into the **Low Cutoff** and **High Cutoff** text boxes.
- Click **OK** to add the **Filter** channel to the display.

Smoothing

All experimental data recordings include varying degrees of noise that can obscure important data. Hardware and software filters can increase the signal to noise ratio by removing certain frequencies from the data. In addition to the use of filters, *LabScribe2* has the ability to smooth the data in order to reveal significant features of your data. **Smoothing** is used to remove noise that is uniform across all frequencies. *LabScribe2* uses two different algorithms to smooth the data; the **Moving Average** function does wide smoothing, while the **Savitzky-Golay** function 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 often diminished by adjacent averaging. To use the **Savitsky-Golay** smoothing function, 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, *LabScribe2* also provides the option to calculate the smoothed 1st, 2nd or 3rd derivative of the data.

Using the Smoothing Function
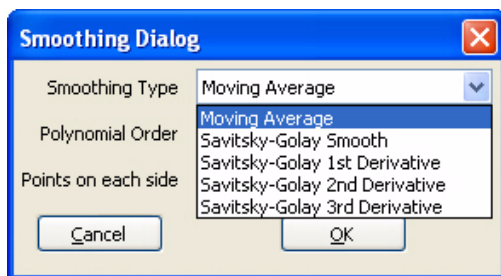
The Smoothing Dialog.

To apply smoothing to a raw data or derivative channel:

- Click **add function** in the **Channel Bar**.
- Choose **Smoothing** from the function list, opening the **Smoothing Dialog**.
- Choose one of the **Smoothing** functions from the drop-down menu.

- For the **Moving Average** function, enter into the text box the number of points that should be averaged on either side of each data point.

- For the **Savitsky-Golay** function or any of its derivatives, enter into the appropriate text boxes both the number of points on either side of each data point to which the polynomial regression should be applied, and the order of the polynomial regression.

Auto Correlation

AutoCorrelation is a mathematical tool used frequently in signal processing for analyzing functions or series of values, such as time domain signals. Practically, 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.
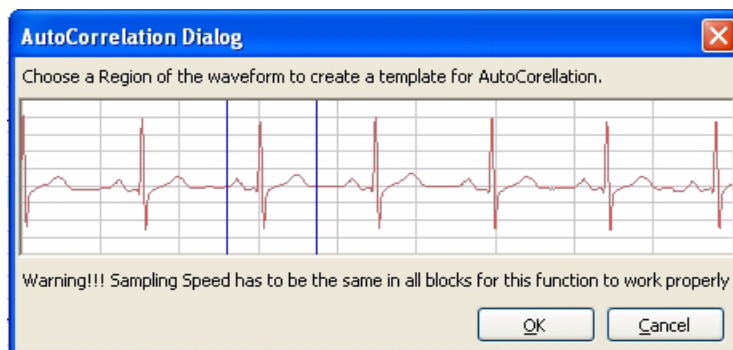

The AutoCorrelation Dialog.

Using the Autocorrelation Function

To apply the **AutoCorrelation** function to a channel:

- Click **add function** in the **Channel Bar**.

- Choose **AutoCorrelation** from the function list, opening the **AutoCorrelation Dialog**.

- Choose a region of the dialog waveform that you wish to check for **AutoCorrelation** and click **OK**.

- A high degree of **AutoCorrelation** will be displayed as a very regular pattern in the **AutoCorrelation** computed channel.

Power

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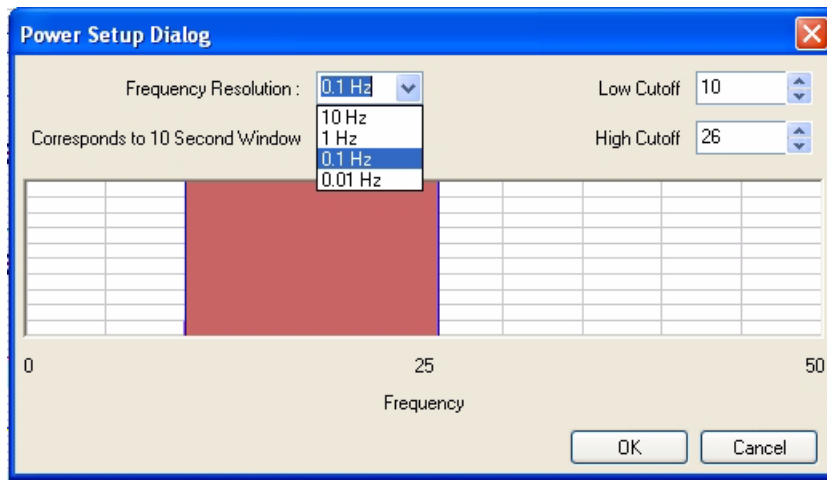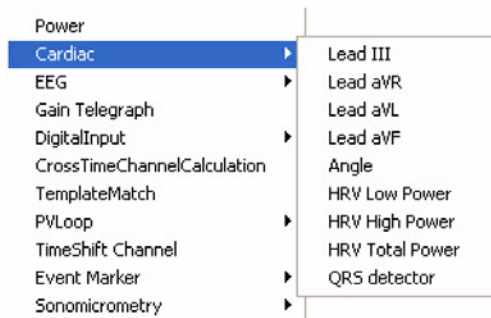

The Power Setup Dialog.

In *LabScribe2*, this function works only off-line. The graphic interface in the **Power Setup Dialog** is similar to the one in the **Filter Setup Dialog**. The band of frequencies selected from the **Power Setup Dialog** are not filtered, but they are subjected to an FFT with an output known as a **Power Number**. In addition to specifying the range of frequencies transformed, the F**requency Resolution** can be selected. The higher the frequency resolution is, the more data points are 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.

Using the Power Function

The **Power** function can only be applied offline. To apply the **Power** function to a previously recorded channel:

- Click **add function** in the **Channel Bar**.

- Choose **Power** from the function list, opening the **Power Setup Dialog**.

- Choose a **Frequency Resolution**. Depending on the **Frequency Resolution** chosen, the **Power** function will be applied to data samples of different durations.

- Choose a range of frequencies for which the **Power** function will determine the average **Power** (amplitude) frequency. This can be done by adjusting the position of the colored area of the graph, or entering the **Low** and **High Cutoffs** in the appropriate text boxes. Click **OK**.

- The **Power Number** for each section of data will be graphed in the **Power** channel. The duration of the sections is determined by the **Frequency Resolution**.

Cardiac

Power
Cardiac ▶
EEG ▶
Gain Telegraph
DigitalInput ▶
CrossTimeChannelCalculation
TemplateMatch
PVLoop ▶
TimeShift Channel
Event Marker ▶
Sonomicrometry ▶

Lead III
Lead aVR
Lead aVL
Lead aVF
Angle
HRV Low Power
HRV High Power
HRV Total Power
QRS detector

The **Cardiac** functions are specifically used for the analysis of electrocardiograms (ECGs). Four of the **Cardiac** functions calculate **Leads: III, aVR, aVL** and **aVF** 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.

The Cardiac submenu.

LabScribe2 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, and can indicate abnormalities in electrical conduction, or the actual anatomical orientation of the heart. Three **Power** functions, which are special cases of the general **Power** function described previously, are also available. These three power functions are useful for heart rate variability (**HRV**) analysis. **HRV Low Power** (0.04-0.15 Hz), **HRV High Power** (0.18-0.4 Hz) and **HRV Total Power** are each calculated from a tachogram transformation of one of the ECG raw data channels, or from the **QRS detector** channel. The **QRS detector** displays a trace with only peaks representing the QRS complexes with amplitudes reflecting the amplitudes of the individual QRS complexes.

The **ECG Analysis** module in the **Advanced** menu can perform more extensive ECG analysis. Refer to the discussion beginning on page 116 for complete details and instruction.

Using the 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, which will display the **Cardiac** submenu.

• Choosing **Lead: III, aVL, aVF** or **aVR,** or cardiac **Angle,** will open a dialog in which the channels corresponding to Lead I and Lead II can be chosen. Click **OK** to display the chosen lead or the cardiac **Angle** in the computed channel.

• Choosing **HRV Low Power**, **HRV High Power**, or **HRV Total Power** will open a dialog in which the threshold lines should be adjusted to pass through just the QRS complexes. **Artifact Removal** is also possible from this dialog. Click **OK** to display the chosen function in the computed channel.
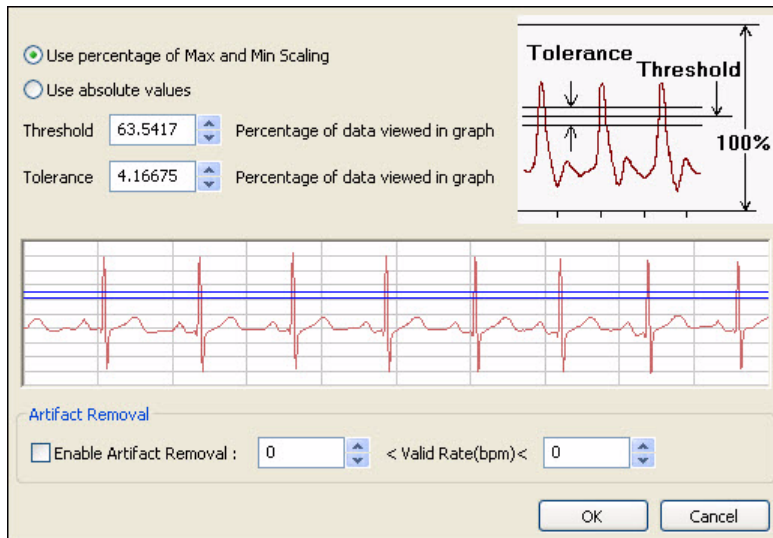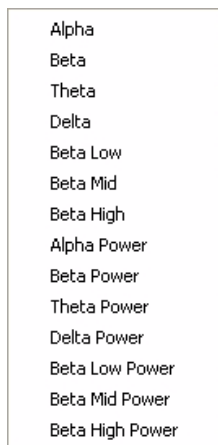
The HRV Power dialog.

- Choosing **QRS Detector** will open a channel showing only the QRS complexes.

Many more ECG parameters can be determined by choosing **ECG Analysis** in the **Advanced** menu. Refer to the **ECG Analysis** section beginning on page 116 of **Chapter 8: Advanced Analysis** for a detailed discussion. **ECG Analysis** is a separately licensed *LabScribe2* module.

EEG

The EEG submenu.

In the **EEG** function submenu, a frequency band is chosen representing a component of the electroencephalograph (**EEG**): **Alpha**, **Beta**, **Theta**, **Delta, Beta Low, Beta Mid** or **Beta High**. For each selected band, *LabScribe2* calculates the average power represented in the band and displays the power value against time. Both the individual band and the band's Power functions can be displayed through the use of the **EEG** function. By displaying separate channels representing the bands, it is possible to see the effect that behavior has on the separate components of the **EEG**.

Using the 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 list, opening the **EEG** submenu.

- Choose one of the **EEG** bands or a Power function. Repeat for other bands you would like displayed.

Gain Telegraph
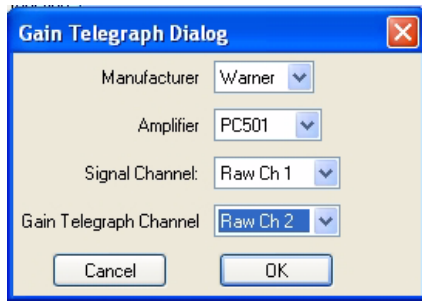
The Gain Telegraph Dialog.

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 the Gain Telegraph Function

To configure the **Gain Telegraph** function:

- 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** and select **Gain Telegraph** from the function list. Select the manufacturer of your amplifier, then select the amplifier you are using. Select the **Signal 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

Some *iWorx* data acquisition units have **Digital Input** connectors that allow external TTL devices to be monitored for state changes and have the changes indicated on the *LabScribe2* recording. Because these functions are closely related to *LabScribe2's* **Digital Output** functions, the details of configuring **Digital Input** computed channels are covered in **Chapter 9: Digital Input and Output**, beginning on page 131.

CrossTime Channel Calculation

CrossTime Channel Calculation can be used to demultiplex data from different sources that have been recorded on one channel. For example, in flourescence studies, the values of absorption as the filter wheel is changed from one filter to another may be recorded on one channel. It is then necessary to mathematically separate the absorption values corresponding to each filter value for further analysis.
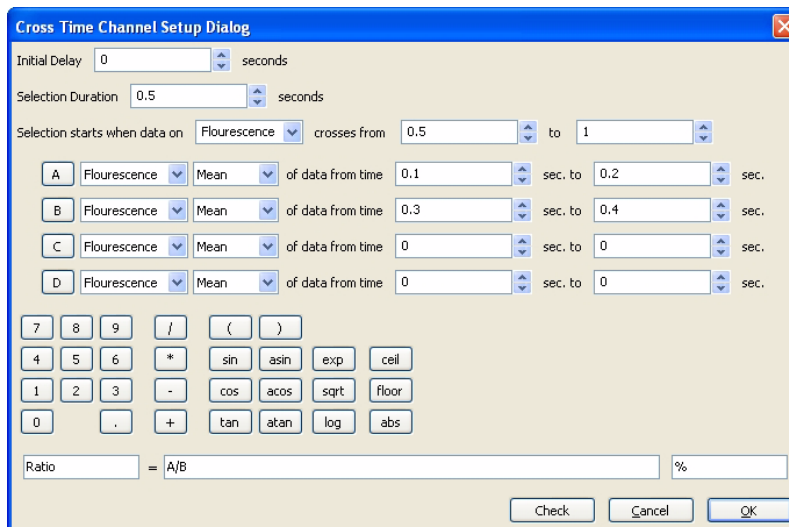

The Cross Time Channel Setup Dialog.

To use the **CrossTime Channel Calculations** function:

- Click on **add function** in the **Channel Bar** of the raw data channel, and choose **CrossTime Channel Calculation**, This will open the **Cross Time Channel Setup Dialog**.

- In the **Initial Delay** text box, enter the amount of time from the start of recording until the function is initiated.

- In the **Selection Duration** text box, enter the duration of the protocol.

- The selection can be set to start when the data on a selected channel crosses from one operator selected value to another.

- For each selection as defined above, variable **A** can be set up to equal the mean, maximum, or minimum of the data during the selected time range on a selected channel. Variables **B**, **C** and **D** can be set up 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.

Template Match

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. One common use of this function is to sort extracellularly recorded action potentials by amplitude and width.
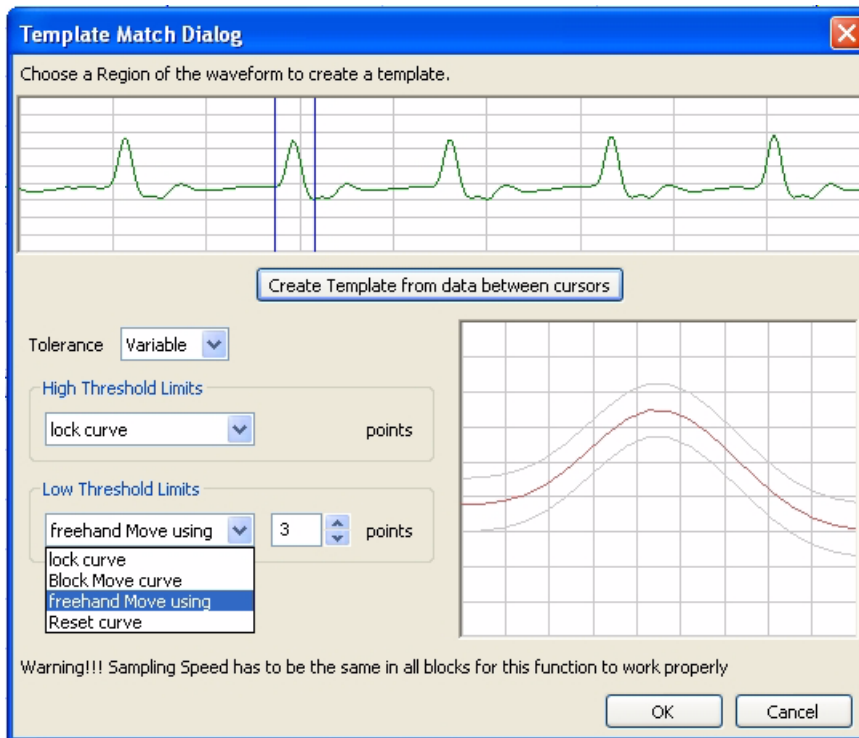
The Template Match dialog.

Using Template Matching

To create a template:

- Select a region of interest using the two cursors. This will be a feature that you want *LabScribe2* to search for matching waveforms throughout the recording.

- 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 data points or the tolerance can vary for different parts of the waveform. In case of variable tolerance, the high and the low limits of the tolerance are set separately.

- Each tolerance line can be moved up or down using **Block Move Curve**. It is also possible to move only certain segments of the curve using **freehand Move using**. When using the freehand move, it is necessary 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.

PVLoop

The **PV Loop** function is used to calibrate the contribution of parallel conductance and volume to ventricular volume estimates based on conductance catheter measurements. Details on integrating this function into **PV Loop** analysis can be found in the **PV Loops** section of **Chapter 8: Advanced Analysis Modules**, which begins on page 100.

TimeShift Channel

Occasionally there is a known delay between two sensors and it makes sense to synchronize them. For example, there may be a latency between the O_2 and CO_2 sensors, and the air volume channel in metabolic recordings. The ability to time shift individual channels allows the user to compensate for these latencies.
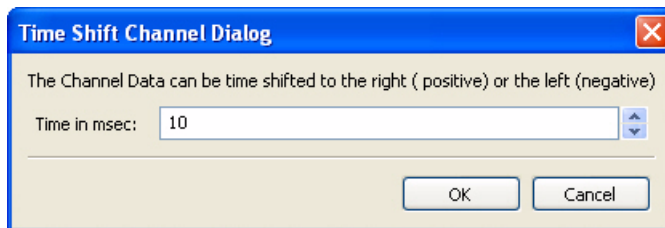

Time Shift Channel Dialog.

Using Channel TimeShifting

To add a computed channel that is shifted to the right by a distance determined by the user:

- Choose the channel you wish to shift, and click on **add function** in the **Channel Bar**.

- The **Time Shift Channel Dialog** will open.

- Enter the desired time shift in milliseconds. Negative values will shift the channel to the left, and positive values will shift it to the right. The up and down arrows to the right of the edit box change the time shift by a millisecond with each mouse click.

Event Marker

The *iWorx* 4-channel Event Marker allows event markers of four different amplitudes to be placed on one raw data channel. The **Event Marker** function separates these markers and puts one, two, three, or all four on individual data channels. A **Count** function can be added as a computed channel for each of the individual event marker channels.
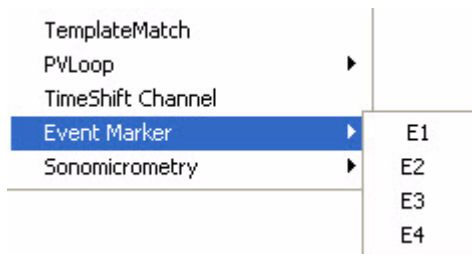
The Event Marker Submenu.

Using Event Markers

To use the **Event Marker** function:

- In the **Event Marker** raw data **Channel Bar**, click on **add function**. Choose **Event Marker** from the list of functions.

- Select **E1**, **E2**, **E3**, or **E4** from the **Event Marker** sub-menu to add one of the four sizes of markers to a computed channel. Repeat for as many of the other size markers as desired.

- If you would like a count of markers from the beginning of the recording block, click **add function** on the appropriate computed channel. From the **Periodic** functions, choose **Count**. A dialog will open in which you can set the correct threshold for the marker amplitude. A new computed channel will be added that will display the count for that size marker from the start of the recording block.

Sonomicrometry

Sonomicrometry is a technique by which changes in length or volume of physiological tissue can be measured through the use of piezoelectric crystals that send sound waves through the tissue. Distance can be estimated by the amount of time it takes for these waves to reach another crystal. Two or three pair of crystals can be inserted in small animal epicardial tissue, measuring short axis and long axis distance measurements as the heart beats. Changes in ventricular volume can be estimated using the data from these measurements. The volume measurements can be used in the creation of Pressure Volume Loops.

Using the Sonomicrometry Functions

Selecting the **Sonomicrometry** function opens a submenu with two choices:

Volume: Opens a dialog in which three sonometric axes can be specified. Volume changes based on changes in these sonometric axes are computed and displayed in a new computed channel.

Remove Outliers: Data values outside the range of possible axis measurements (based on the known distances between the crystals) are removed.

Chapter 7: Analysis

Overview

Built into *LabScribe2* is a powerful array of data analysis tools. The variety of tools available span the range from the most frequently used straightforward operations to much more complex, specialized routines. *LabScribe2* is a powerful analytical tool that can get to work on analysis immediately, or be customized to execute very specific, complex analysis routines.

Analysis in *LabScribe2* is accomplished in several different ways. Waveform manipulations are discussed in **Chapter 6: Computed Channels**. In this chapter, the **Analysis Window** is introduced. In the **Analysis Window**, regions of data are characterized by mathematical calculations resulting in descriptive quantitative values.

Ways to redisplay the data, such as an **XY View** or a graphical **FFT** analysis, are discussed, as are *LabScribe2's* sophisticated **Find** functions. Finally, **Scripting** options that incorporate other applications into the analysis of recorded data are outlined.

The Analysis Window

Functions specific to the **Analysis Window** take a group of data points selected by the user and reduce them into a single, mathematical value. These values can be saved to the **Journal** within *LabScribe2* or exported to other programs. There are currently more than 20 functions available in the *LabScribe2* program that perform this kind of analysis.

Analysis Window Components

The **Analysis** window is used to display and perform calculations on selected regions of **Chart** data or **Scope** sweeps captured from the **Main Window**. The data in the **Analysis Window** are defined by the data displayed in the **Main Window**. The **Analysis Window** and its analysis functions are activated by clicking the **Analysis** icon on the *LabScribe2* **Toolbar** or selecting **Analysis** from the **View** menu. Data from all channels within the selected area are displayed simultaneously 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**, and **Two Cursor Mode** (**Single Cursor Mode** is unavailable). The **Toolbar** icons have the same functions. Refer to **Chapter 1: The Display** for a complete description of these tools.

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)**.

By positioning the two cursors on the left and right edges of the data to be analyzed, *LabScribe2* will immediately calculate and display the values for the selected parameters in the **Table Function** data boxes of each channel's **Channel Bar**.

The primary features of the **Analysis Window,** including those that differ from the **Main Window**, are labeled in the figure on the next page.
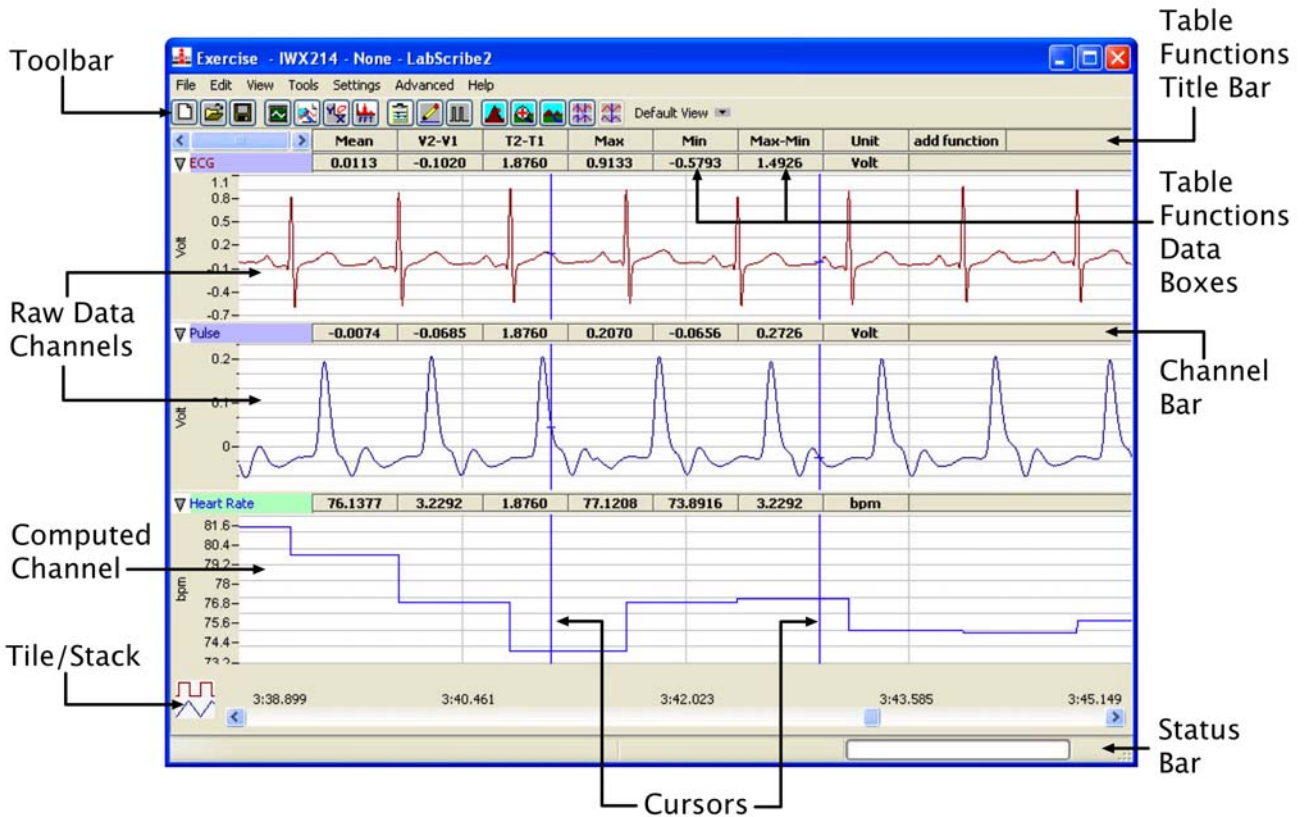
The Analysis Window.

Channel Menus in the Analysis Window
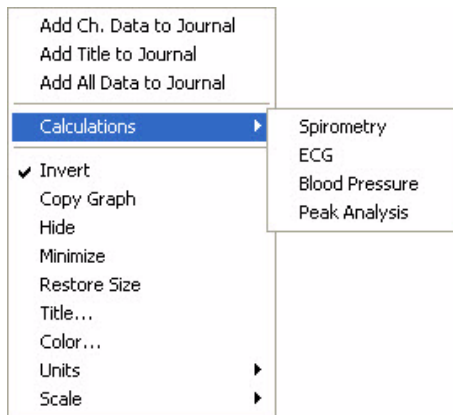
The Analysis Window Channel Menu.

Clicking on the **Channel Menu** arrow on the left side of the **Channel Bar** or right-clicking in any channel of the **Analysis Window** opens the **Channel Menu**.

The items in the lower section of the **Analysis Window Channel Menu** have the same functions as the similar items in the **Main Window Channel Menu**. Refer to page 3 in **Chapter 1: The Display** for a discussion of these menu items.

The first three menu items send the values in the **Channel Bar Table Functions** data boxes to the **Journal** for formatting and inclusion in reports:

• **Add Ch. Data to Journal**: Adds the channel name and data from the selected channel to the **Journal**.

• **Add All Data to Journal**: Adds the channel titles and the data from all channels to the **Journal**.

• **Add Title to Journal**: Adds the **Table Functions** titles to the **Journal**.

The choices in the **Calculations** submenu display calculated data values specific to individual functions:

• **Spirometry**: Opens a dialog that allows the user to choose and display calculations of **Spirometry** functions based on the raw data in a calibrated air flow channel. By placing the cursors in the data sample to the edges of one breath cycle as in the demonstration trace in the top part of the dialog, a number of spirometry parameters will be calculated automatically for that cycle of data. Clicking the **Save** button will add the data from that cycle and open up a new blank data row. The process can be repeated for other breaths in the recording. Clicking **Add to Journal** will add the data table to the **Journal**.
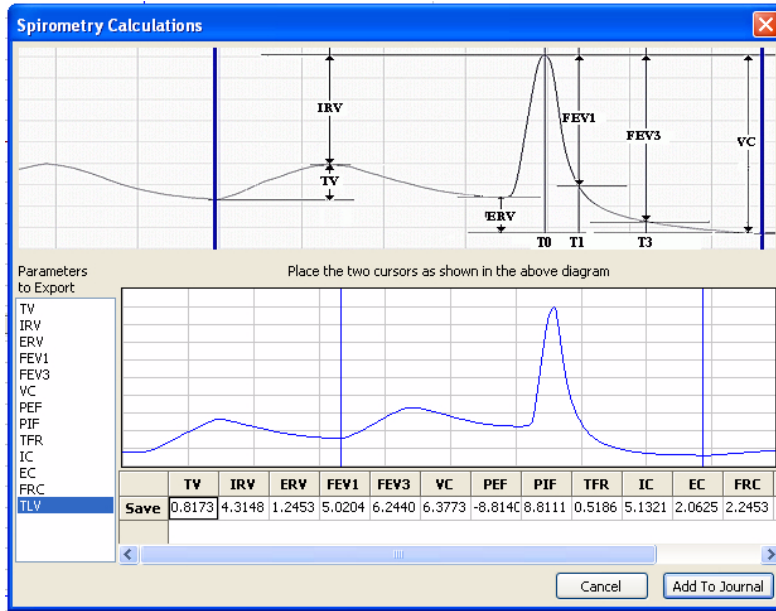
Spirometry Calculations.

• **ECG**: Opens a dialog that allows the user to choose and display calculations of ECG parameters based on the raw data in an **ECG** channel. By placing the cursors in the data sample to the edges of one heart beat cycle as in the demonstration trace in the top part of the dialog, a number of ECG parameters will be calculated automatically for that cycle of data. Clicking the **Save** button will add the data from that cycle and open up a new blank data row. The process can be repeated for other beat cycles in the recording. Clicking **Add to Journal** will add the data table to the **Journal**.
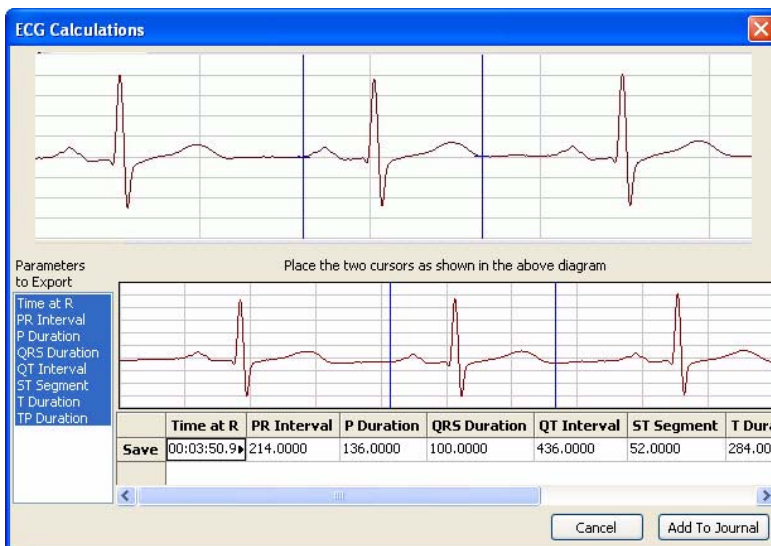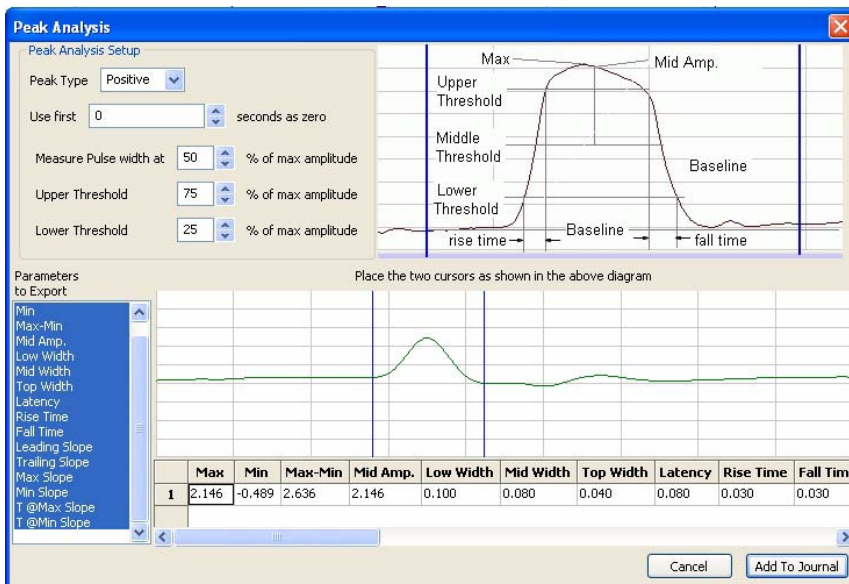

ECG Calculations.

• **Blood Pressure**: Opens a dialog that allows the user to choose and display calculations of blood pressure parameters based on the raw data in a **Blood Pressure** channel. By placing the cursors in the data sample to the edges of one heart beat cycle as in the demonstration trace in the top part of the dialog, a number of blood pressure parameters will be calculated automatically for that cycle of data. Clicking the **Save** button will add the data from that cycle and open up a new blank data row. The process can be repeated for other beat cycles in the recording. Clicking **Add to Journal** will add the data table to the **Journal**.
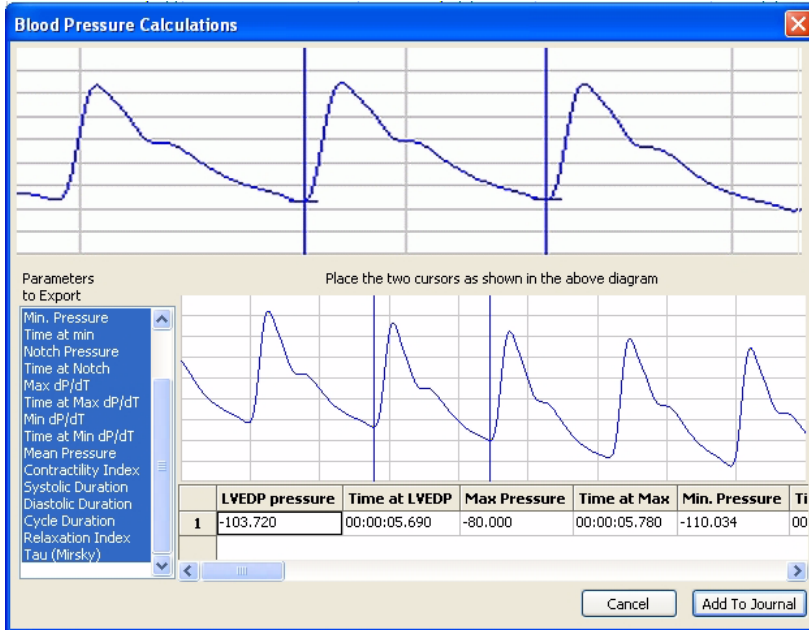
Blood Pressure Calculations.

• **Peak Analysis**: Opens a dialog that allows the user to choose and display calculations of peak parameters based on the raw data in a channel with peak data. By placing the cursors in the data sample to the edges of one peak as in the demonstration trace in the top part of the dialog, a number of peak parameters will be calculated automatically for that peak. Clicking the **Save** button will add the data from that peak and open up a new blank data row. The process can be repeated for other peaks in the recording. Clicking **Add to Journal** will add the data table to the **Journal**.

Peaks Calculations.

The Functions

The functions selected in the **add function** list determine the calculations performed on the data points between the two cursors in the **Analysis** window .
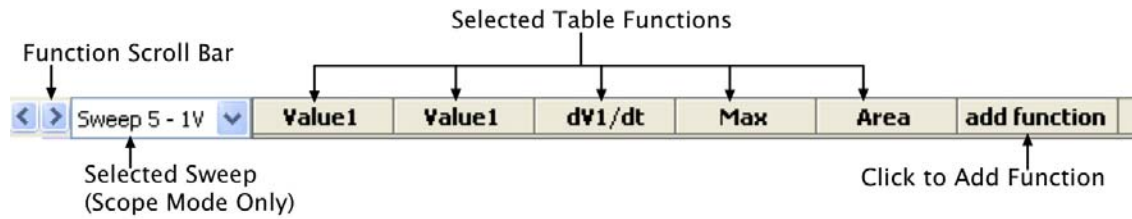
The Analysis Window Table Functions Title Bar.

The functions available in the **Analysis** window are divided into three categories: **General**, **Derivative**, and **Integral**.

General

The functions in the **General** category are:

- **Value1**: Amplitude at **Cursor 1**.

- **Value2**: Amplitude at **Cursor 2**.

- **Time1**: Time at **Cursor 1**.

- **Time2**: Time at **Cursor 2**.

- **V2-V1**: The amplitude at **Cursor 1** (the **Cursor** on the left) subtracted from the amplitude at **Cursor 2** (the **Cursor** on the right). This may be a negative number.

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

- **Max**: Maximum amplitude between the cursors.

- **Min**: Minimum amplitude between the cursors.

- **Mean**: Mean or average amplitude between the cursors.

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

- **Mark**: Text of any mark between the cursors.

- **Unit**: The units of the channel data.

- **RMS**: Root Mean Square of the amplitude values between the cursors.

- **StdDev**: Standard Deviation of the amplitude values between the cursors.

- **Events-# Zero Crossings**: Number of times the data cross zero (with a positive slope) in the region between the cursors.

Derivative

The functions in the **Derivative** category are:

- **dV1/dt**: Derivative at **Cursor 1**.

- **dV2/dt**: Derivative at **Cursor 2**.

- **Max dV/dt**: Largest dV/dt values (steepest slope) for all of the data points between the two cursors.

- **Min dV/dt**: Minimum (often negative with a steep slope; not necessarily a flatter region) 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. The functions in the **Integral** category are:

- **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 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.

Adding Functions to the Analysis Window   To add a function to the data in the **Analysis Window**:

- Positon the two cursors in the **Main Window** on either side of the section of data you wish to analyze.

- Click **AutoScale** in each of the **Main Window Channel Bars**.

- Click the **Analysis Window** icon in the **Toolbar**. The **Analysis Window** will now be displayed.

- Click the **add function** button in the **Table Functions Title Bar** just beneath the **Toolbar**.

- Select a function from one of the three function categories.

- That function will be applied to the data between the two cursors on all channels and be displayed in the **Table Functions** data boxes in each **Channel Bar** in the position vertically beneath the function title. While the chosen function will be calculated for all data channels, it may be more meaningful and relevant on some channels than others.

- Additional functions can be added in the same way. If there are more function boxes than space on the screen, the function boxes can be scrolled through using the scroll bar on the left end of the **Table Functions Title Bar**.

- The precision of the calculations performed is adjustable using the **Data Display Precision** value in the **Options** page of the **Preferences Dialog**.

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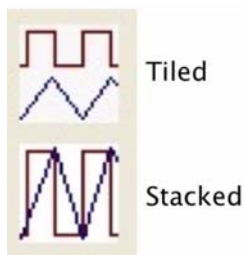
By default, the channels selected for display are presented in **Tiled** mode. Each channel is displayed in its own area, as they are shown in the **Main** window. By clicking the **Tile/Stack** icon in the lower left hand corner of the **Analysis** window, each channel's waveforms are overlaid on the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

Tiled/Stacked icon.

## Scope Mode Analysis Window Options

In **Scope** mode, you can view either a single sweep or multiple sweeps at the same time. Data functions can only be calculated on one sweep at a time.
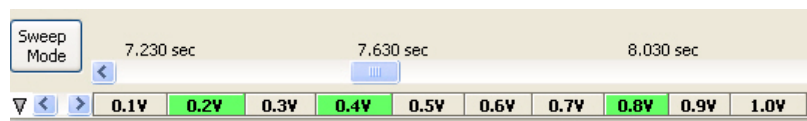
*Sweep Selection bar in Sweep mode with multiple sweeps selected.*

To view and calculate mathematical functions on **Scope** data:

- The displayed sweeps can be chosen by clicking on the desired sweeps in the **Sweep Selection Bar** at the bottom of the **Analysis Window**. To choose multiple sweeps, click on each sweep while pressing CONTROL on the computer keyboard (COMMAND on the Macintosh). Multiple sweeps will be displayed superimposed on one another. Clicking a second time on any sweep will remove it from the display.

- As in the **Main Window**, individual sweeps can be titled (or re-titled) or deleted by right-clicking on that sweep in the **Sweep Selection Bar** and opening the drop-down menu.

- 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 **Table Functions Title Bar**. The primary sweep cannot be deselected from the **Sweep Selection Bar**.

- Functions are added in the same manner as in the **Chart** mode **Analysis Window**.

- Individual sweeps can be organized into groups by using the **Sweep Manager**. The **Sweep Manager** is available from the **Sweep** menu arrow to the left of the **Sweep Selection Bar**. 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.
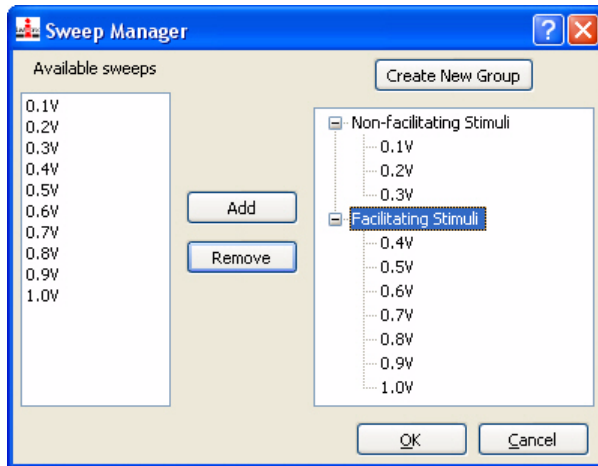
The Sweep Manager.

- 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 from **Sweep** to **Group**. In **Group** mode, it's possible to display any of the named groups. Selecting the **Group** name will display just the sweeps in that **Group**.
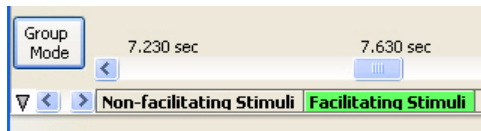
The Sweep Selection Bar in Group mode.

Copy, Export, and Print Analysis Window Data

To copy the graphical 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 an individual channel's trace, open the **Channel Menu** and choose **Copy Graph.**

The data in the **Analysis** window can be exported in Matlab (*.mat), DADiSP (*.dat), Excel (*.xls), or Text (*.txt) formats. An image of the data can be exported in Portable Network Graphics (*.png) or JPEG (*.jpg) formats. 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** dialog.

To print the visible screen (excluding the **Journal**), choose **Print View** in the **File** menu.

Redisplayed Data

Data are 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. *LabScribe2* supports **XY** and **FFT plots**. A host of measurements can be made from each type of redisplayed data window.

The XY View

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. The **XY View** can be chosen by clicking on the **XY View** icon in the **Toolbar** or choosing **XY View** in the **View** menu.

XY View Components
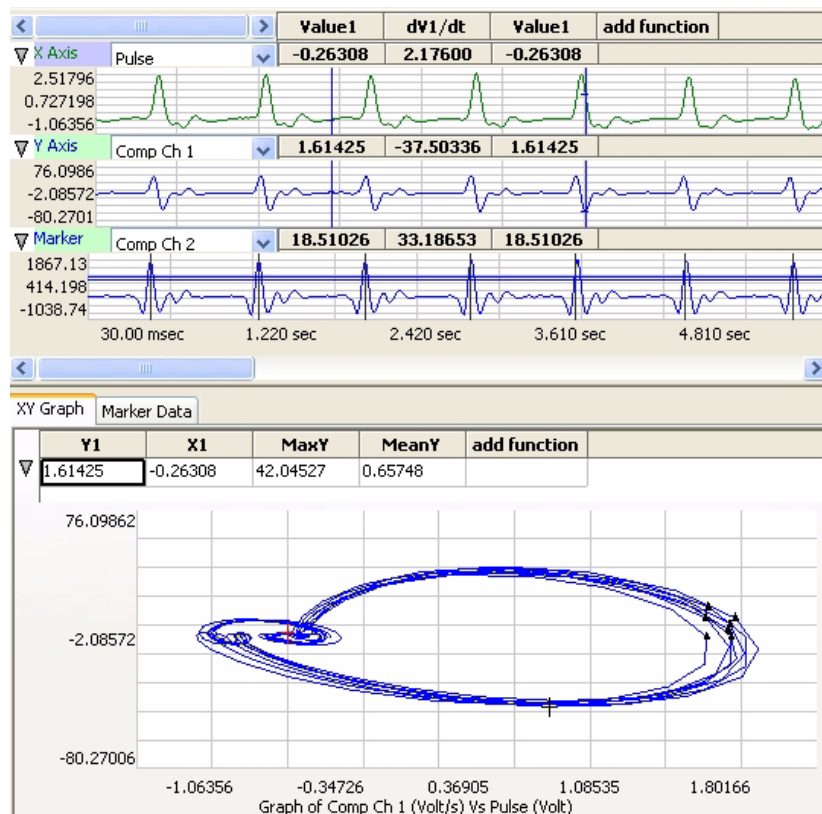
The XY View.

Three data channels are displayed in the top panel of the **XY View**. The raw data channels that are plotted in the **XY Graph** are chosen here, as well as which of the channels will be the X axis and which will be the Y axis. The **XY Graph** is displayed on the **XY Graph** tabbed page in the lower pane of the **XY View**. By selecting the **Marker Data** tab, a table of information about the individual markers opens. (Markers are described below.) There is an **add function** button for the raw data channels and one for the **XY Graph**. These open lists of functions appropriate to the two types of plots.

Selecting the Displayed Channels

To plot an **XY Graph**:

- Select the **X-axis** channel from the drop-down box in the upper trace.

- Select the **Y-axis** channel from the drop-down box in the second plot.

- If desired, a **Marker** channel can be selected in the third window. The **Marker** channel is used to mark specific points of interest in the **XY Graph**. When a **Marker** channel is used, data from the **XY Graph** is extracted at the specified points of interest and displayed as triangular markers on the **XY Graph**. The **Marker** data points are determined by the placement of the threshold lines on the **Marker Data** channel. Each time the data values pass across the threshold lines in a positive direction, a marker is generated. **Marker Data** can be viewed by selecting the **Marker Data** tab in the **XY Graph** window.

Once the channels are selected, the **XY Graph** is displayed. All the data visible on the active screen in the X and Y axis traces will be included in the **XY Graph**. To more finely tune the region of data to be included in the **XY Graph**, select **Two Cursor Mode** 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**.

Channel Menus in the XY View

The raw data Channel Bar menus in XY View.
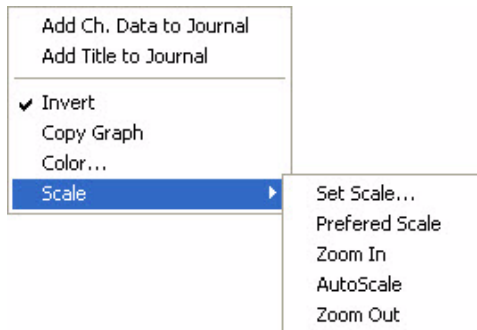
Clicking on the **Channel Menu** button to the far left of a **Channel Bar** or right-clicking in the channel's display area opens the **Channel Menu**.

The following items are included in some or all of the **Channel Menus** of the three raw data channels:

- **Add Ch Data to Journal**: Adds the channel title and **Table Functions** data values from the selected channel to the **Journal**.

- **Add Title to Journal**: Adds the **Table Functions** 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 an external application.

- **Color...**: Allows the user to choose the color of the trace for the channel.

- **Scale**: The items in the **Scale** submenu have the same functions as in the **Channel** menus of the **Main Window**. Refer to the discussion starting on page 30 in **Chapter 3: Acquisition** for details.

The functions available by clicking **add function** in any of the individual **Channel Bars** are the same functions available in the **Analysis Window**.

The XY Graph Window Menu Clicking the arrow to the left of **add function** in the **XY Graph** window opens the **XY Graph** window menu with the following menu items:

- **Add All Data to Journal**: Adds the data from the **XY Graph Table Functions** boxes to the Journal.

- **Add Title to Journal**: Adds the **Table Functions** titles to the **Journal**.

- **Copy graph**: Copies the **XY Graph** to the **Journal**.

XY Graph Table Functions The functions selected in the **add function** list of the **XY Graph** window determine the calculations performed on the data points in the **XY Graph**. The results of the selected calculations are displayed in the **Table Functions** data boxes above the **XY Graph**.

The following functions are available in the **XY Graph** window:

General

- **Y1**: Value of the Y-Axis channel at **Cursor 1**.

- **Y2**: Value of the Y-Axis channel at **Cursor 2**.

- **X1**: Value of the X-Axis channel at **Cursor 1**.

- **X2**: Value of the X-Axis channel at **Cursor 2**.

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

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

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

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

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

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

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

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

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

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

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

Derivative

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

- **dY2/dx**: Slope of the **XY Graph** at **Cursor 2**.

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

- **Min dV/dt**: Minimum slope of the **XY Graph** between the two cursors. This can be a steep negative slope.

- **Mean dV/dt**: Mean slope of the **XY Graph** 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 beneath all points between the two cursors is calculated with respect to this baseline.

- **Area wrt Zero**: Zero is taken as baseline and the area between all points and zero is calculated for the points between the cursors.

- **Area wrt Y1**: The value of the **Y-axis** channel at **Cursor 1** is taken as baseline and the area between all points and this value is calculated for the points between the cursors.

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

Marker Data

By selecting the **Marker Data** tab next to the **XY Graph** tab in the **XY Graph** window, a table displaying information about the markers opens. The time coordinate, the X value, the Y value, and the **Marker** channel value of each marker are listed.

Copy, Export, and Print XY View Window

To copy the graphical data displayed in the **XY View**, use the **Copy** command in the **Edit** menu. The image can be pasted into any application (including the **Journal**) that supports the clipboard.

To export the data from the channels used to construct the **XY Graph**, as well as the **Marker Channel** data, 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 graphs in the current screen, choose **Print View** in the **File** menu.

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** analysis is used to determine the relative contributions of frequency components in a raw signal. For example, in EEG studies, Alpha waves are EEG signals with frequencies between 8 and 13 Hz. If an **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.

LabScribe2 can perform Fast Fourier Transforms on selected pieces of recorded data in the **FFT** window. The **FFT** window is opened from the **View** menu or by clicking the **FFT** icon on the **Toolbar** at the top of the **Main Window**. The selected data are moved to the **Linear Display** area in the **FFT** window where adjustments can be made which optimize the size of the **FFT** plot.
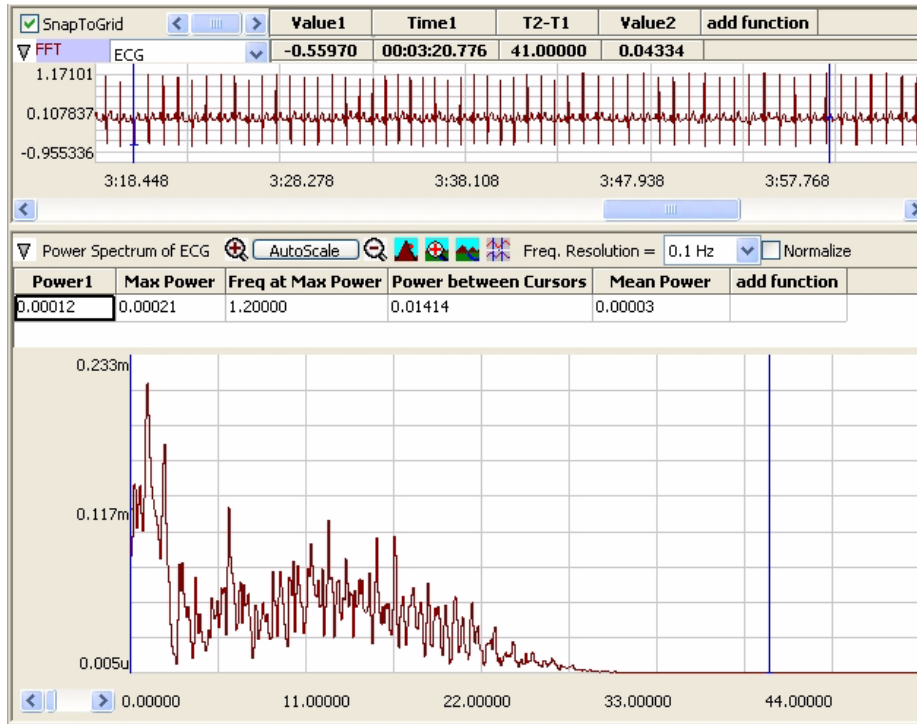
FFT Window.

Located in the upper left of the data channel display area, the **FFT** channel selection drop-down box allows the user to select the data channel to be used in the **FFT** plot. The same functions available in the **Analysis Window** can be applied to the data from this channel by clicking on its **add function** button. The **FFT** plot is displayed in the lower pane. A different set of functions, appropriate to the **FFT** plot, can be accessed by clicking the **add function** button in this lower pane.

The **FFT** plot toolbar has display controls similar to the **Main Window**:

• **Half Display Time, Zoom between Cursors** and **Double Display Time** function as they do in the **Main Window**.

• The Y-axis can be scaled using the **Zoom In, AutoScale** and **Zoom Out** buttons.

• The **Frequency Resolution** drop-down box allows setting the **Frequency Resolution** of the **FFT** from 10Hz to 0.01Hz.

• Checking the **Normalize** checkbox adjusts the vertical scale of the **FFT Plot** to a zero to one scale. This standardized scale allows for direct comparisons with other **FFT Plots** that may have a different Y-axis range.

To configure the **FFT View**:

• 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 are moved to the **FFT** window by clicking the **FFT** icon on the *LabScribe2* toolbar or by selecting **FFT** from the **View** menu.

• Once the primary data set is moved to the **FFT** window, the two cursors on the linear graph of the data channel (above the actual **FFT** plot) can be used to fine-tune the data selection. Only the data between the cursors in the linear graph are 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. If **Snap To Grid** is selected, *LabScribe2* selects the end point for the data selection.

- The cursors also delineate the values for the selected **Table Functions** displayed above the linear data channel graph. The values and the titles can be copied to the **Journal** by using the functions in the **Channel Menu** for the data channel, as described below.

Channel Menu in the FFT View Clicking the **Channel Menu** arrow at the left of the data channel opens the **Channel Menu**. It contains the following commands:

Add Ch. Data to Journal
Add Title to Journal
✔ Invert
Copy Graph
Color...
Scale   ▶
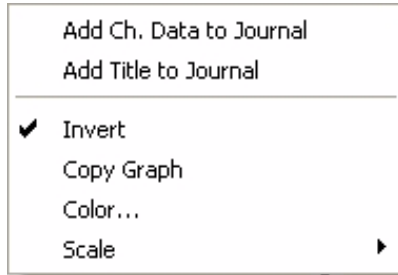
The Data Channel Menu.

- **Add Ch Data to Journal**: Adds the channel title and **Table Functions** data to the **Journal**.

- **Add Title to Journal**: Adds the **Table Functions** 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 an external application.

- **Color...**: Allows the user to choose the color of the trace for the channel.

- **Scale**: The items in the **Scale** submenu have the same functions as in the **Channel** menus of the **Main Window**. Refer to the discussion starting on page 30 in **Chapter 3: Acquisition** for details.

The functions available by clicking **add function** in the data **Channel Bar** are the same functions available in the **Analysis Window**.

The FFT Plot Menu Clicking the arrow to the left of **add function** in the **FFT Plot** window opens the **FFT Plot** window menu with the following menu items:

- **Add All Data to Journal**: Adds the data from the **FFT Plot Table Functions** data boxes to the **Journal**.

- **Add Title to Journal**: Adds the **Table Functions** titles to the **Journal**.

- **Copy graph**: Copies the **FFT** plot to the clipboard. It can then be pasted into the **Journal** or an external application.

FFT Table Functions The following functions are available in the **FFT Plot** 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 values at the two cursors.

- **F2-F1**: Difference in frequency between the values at the two cursors.

- **Max power**: Maximum power between the cursors.

- **Min power**: Minimum power between the cursors.

- **Mean power**: Mean power between the cursors.

- **MaxP-MinP**: The Maximum power value minus the Minimum power value.

- **Freq at Max power**: Frequency at Maximum power between the cursors.

- **Freq at Min power**: Frequency at Minimum power between the cursors.

- **Mean Frequency**: Mean frequency of the values between the cursors.

- **Median Fequency**: Median frequency of the values bewtween the cursors.

Integral

- **Power between Cursors**: Summed power data values between the cursors.

Theoretical Considerations

To ensure problem-free operation of the **FFT** function, certain premises need to be considered.

- The mathematical underpinnings of digital sampling, which makes the *LabScribe2* software work, begin with a foundation 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 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 an **FFT** is performed on data recorded with *LabScribe2*, the program sets the X-axis to a scale from 0Hz (DC) to a frequency that is 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.

- 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 View**. If you make a different selection, *LabScribe2* 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, *LabScribe2* sets the X-axis limits in the **FFT** window according to the sampling 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.

Copy, Export, and Print FFT Windows

Use the **Copy** command in the **Edit** menu to copy the view of data displayed in the **FFT View**. 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.

Choosing **Print View** will print the current window.

Find Functions

LabScribe2 can be used to identify **Regions Of Interest** (**ROI**) in recorded data. The identified data points or marks can be sent to either the **Journal** or the **Marks** window, where they can be used to build a report or to 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 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 is the extraction of the relevant data points or regions of interest from the raw data.

Find Dialog Window

The **Find** dialog window is accessed by selecting **Find** from the **Tools** menu. Data points of interest can be identified using commands from the **Find Dialog** window.
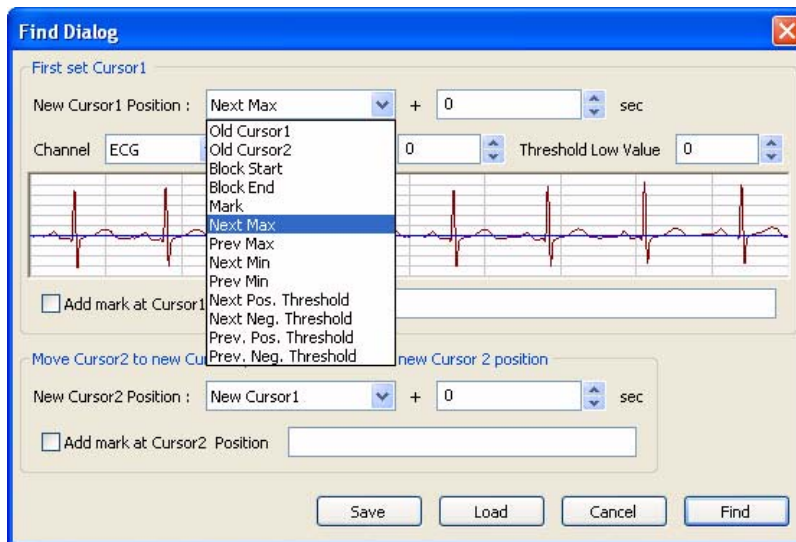
The Find Dialog.

In the **Find Dialog**, each cursor (in **Two Curso**r **Mode**) is programmed individually to move to a location with designated data criteria with the **Find Next** command in the **Tools** menu, or by its keyboard shortcut (CONTROL + F in Windows, or COMMAND + F on the Macintosh). To configure the **Find Dialog**:

- In the top part of the dialog, set the New **Cursor 1** positon to the type of data point or **Mark** you want **Cursor 1** to locate.

- In the lower part of the dialog, set the New **Cursor 2** position to the type of data point or **Mark** you want **Cursor 2** to locate.

- Setting the New **Cursor 2** location to the New **Cursor 1** position enables the user to repeatedly find specific data points over and over, as the cursors will move as one to the positions designated for **Cursor 1**. Setting the New **Cursor 2** positon at the next incidence of the type of data you are looking for (which is not necessarily the same type of data point) will keep the two cursors separated, allowing you to determine both amplitude values and the distance between the two cursors.

- Indicate the channel you want to be used, and use the sample trace in the **Find Dialog** to specify the threshold window. For example, if the threshold lines are set above the P and T waves in an ECG, and **Next Max** is chosen for the desired New Cursor Position location, the cursors will move from R to R, as *Labscribe2* will look for maximum values over the threshold lines.

- The cursors can be instructed to move a set number of seconds past (or before if a negative value is entered) the data criterion set.

- Once the criteria are set, choosing **Find Next**, or using its keyboard shortcut (CONTROL + F in WIndows, COMMAND + F on the Macintosh), will move the cursors to their next positions.

- Although the **Find** commands work in the **Main Window**, using them in the **Analysis Window** will report specified values to the **Table Functions** data boxes, allowing the values to be sent to the **Journal**.

- Once the **Find** routine is defined, 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.

- In the example in the figure, using this **Find** routine on the ECG trace will cause both cursors to move as one from R to R to R, allowing you to locate the position and determine the amplitude of each R. If the New **Cursor 2** were to be set instead at the next example of the type of data you were looking for, each time the **Find** command was called, the cursors would move individually and two R points would be located at each **Find** command, allowing the distance between Rs to be determined as well.

The locations that the cursors can be programmed to move to include:

- **Old Cursor 1** and **Old Cursor 2**: The positions from which the **Cursors** will be moving with each **Find Next** command.

- **New Cursor 1**: The position to which Cursor 1 moves with each **Find Next** command.

- **Block Start** and **Block End**: The cursors can be instructed to move to the start or the end of the current block of data.

- **Mark**: The cursors can be instructed to move to a specifed mark.

- **Next Max** or **Previous Max**: The cursors can be instructed to move to the Next Maximum value or the Previous Maximum value. These **Max** points will be over the threshold set in the sample data.

- **Next Min** or **Previous Min**: The cursors can be instructed to move to the Next or Previous Minimum values, as determined by the **Threshold** position.

- **Next** or **Previous Positive** or **Negative Threshold**: Instructs the cursors to move to the Next or Previous Threshold line the data cross in either a positive or negative direction.

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.

FIND EXERCISE

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. To find these durations:

1. Use the *LabScribe2* ECG **Tutorial** to create an ECG recording.

2. **AutoScale** the ECG recording and transfer the data of interest from the ECG recording from the **Main Window** to the **Analysis Window**.

3. Select the calculations (**T1**, **T2**, **T2-T1**) to be performed on the data from the **Table Functions** list.

4. Pull down the **Tools** menu and select the **Find** function.

5. Program the cursor positions. To find the next supranormal *R* wave in the data, the **Cursor 1** position is set equal to **Next Max**. The New **Cursor 2** position is also set to **Next Max** and the data channel is selected.

6. Adjust the threshold amplitudes for both cursors, above which the **Find** function will **Find** the **Next Max**, so that only the supranormal R waves are above the threshold. Name and save the **Find** routine.

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

8. Copy values and their headings to the **Journal** via the **Add Title to Journal** and **Add Data to Journal** functions in the right-click menu of the **Analysis** window.

9. The **Find Next** command will shift both cursors one supranormal R wave to the right, allowing the next interval to be determined and recorded.
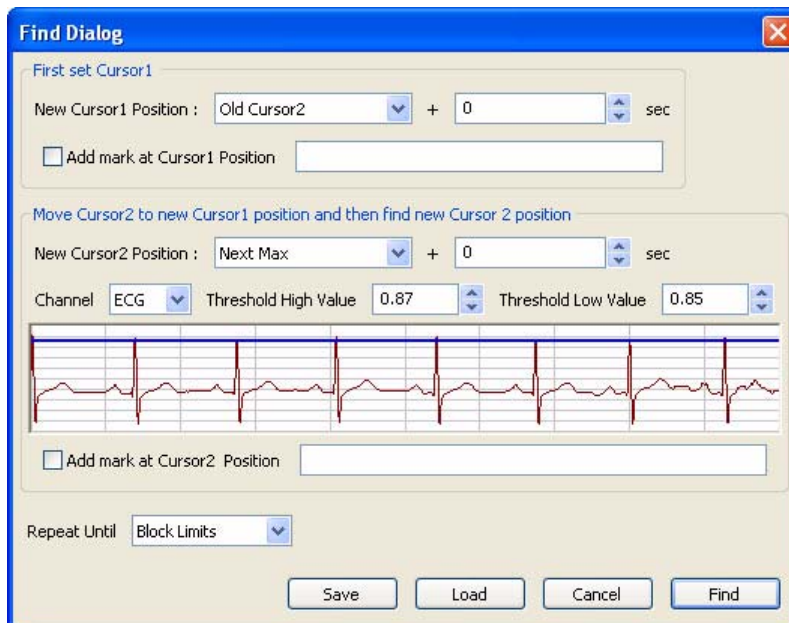
Auto Find Dialog Window

The **Auto Find** dialog window is similar to the **Find** window. It is also accessible from the **Tools** menu and works only from the **Analysis 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 added to the **Table Functions** data boxes 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 **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, *LabScribe2* uses the threshold values set for periodic data on the **add function** menu in the **Channel Bar**. 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**.

The Auto Find dialog window.

AUTO FIND EXERCISE

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.

1. Manually place the cursors on the two successive *R* waves at the beginning of the record.

2. The next two data points that are needed for measuring the next period (**T2-T1**) are the first data point in the next period (the second *R* wave from the preceding period), and the next maximum value or *R* wave in succession (the second R wave in the next period).

3. 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**.

4. 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.

Other Analysis Tools

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, *LabScribe2* has a **Scripting** function. **Scripts** can be written in any language, or they can be any program already installed on your computer. The **Scripting Setup Dialog** allows the user to manage the export of data being displayed on the screen, and to perform an operation on that data. *LabScribe2* can be set up to provide easy access to these scripts in the **Scripting Setup Dialog**.

Managing Scripts

The **Scripting Setup Dialog** is launched by choosing **Manage Scripts** in the **Advanced** menu.
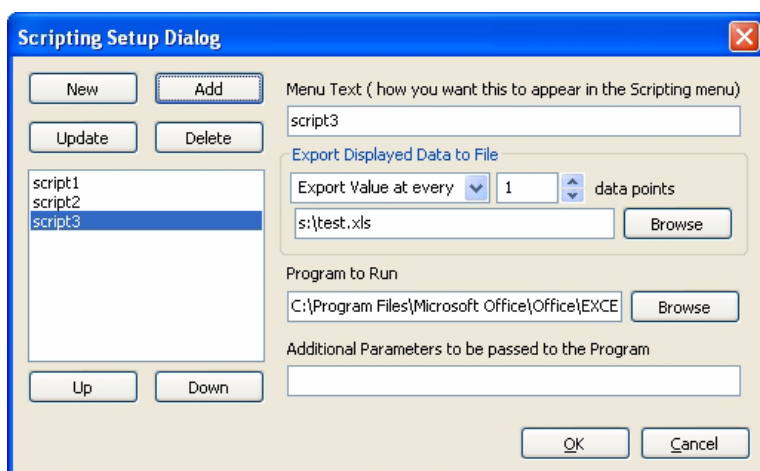
The Scripting Setup Dialog.

Choosing the **Manage Scripts** menu option launches the **Scripting Setup Dialog**. Using the **Scripting Setup Dialog**, you can create various scripting shortcuts that will then be available from the list of shortcuts in the **Scripting Setup Dialog**. Selecting the script from the menu and clicking **OK** will launch the associated script.

To set up a new scripting shortcut:

- Click the **New** button. Give the shortcut an appropriate name.

- You can export every "Nth" datapoint, or the **Mean** or **Maximum** of every "N" datapoints.

- Give the exported datafile a name, and choose the kind of file you want to export. We have chosen to export an Excel file in this example.

- Choose the program to launch (in this case, Microsoft Excel). Browse on your computer until you find the program file. The path of the chosen file will be shown next to the browse button.

- Specify any additional parameters you want applied to this program, such as the exprted data file.

Note: *Exporting in Excel, text and DADiSP format creates multiple files (one for each block).*

- Set the working directory for the program if needed.

- Click the **Add** button to add this scripting shortcut to the menu. You can add many such shortcuts.

- You can change the order in which the shortcuts appear in the menu, using the keyboard UP and DOWN buttons.

Chapter 8: Advanced Analysis

Overview

LabScribe2 includes routines designed to analyze data related to specific physiological processes. Separately licensed routines analyze **PV Loops** (Pressure-Volume Loops), intravascular **Blood Pressure**, **ECG** data, and **Metabolic** recordings. The **Normalization** module (accessed by right-clicking in a data channel and selecting **Normalization**) calibrates and standardizes the diameter of small vessels for experiments using wire myographs. **AutoMark Peaks** analyzes recordings with peak data. Except for the **Normalization** module, these **Advanced Analysis** routines can be accessed from the **Advanced** menu.

AutoMarks These analyses often require the detection of specific events within a recording. *LabScribe2* applies algorithms to find the points or regions of interest. LVEDP, a blood pressure parameter, and the R and P waves in an ECG are examples of specific events. The user can choose which specific events should be located and sets the criteria for their detection. 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 serve as the basis for calculations; second, it provides the user with visual cues to confirm that the software did the job of detecting the correct points.

Calculations Calculations specific to each routine are computed from the automarked data points and the results are displayed in the appropriate analysis windows.

PV Loops

Ventricular pressure-volume loops are used to study many quantitative aspects of cardiac contractility. The **PV Loops** software module integrates the variables and measurements necessary for the recording, analysis, and interpretation of ventricular pressure-volume loops. The **PV Loops** module requires a separate license. The first time you select **PV Loops**, you will be asked for a username and a serial number. Contact *iWorx Systems* for more information.
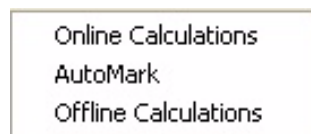
When **PV Loops** is chosen from the **Advanced** menu, a submenu opens, displaying three choices: **Online Calculations**, **Automark**, and **Offline Calculations**.

PV Loops submenu.

Online Calculations *LabScribe2* displays ventricular pressure and ventricular volume data in real time as individual waveforms as well as on a **PV Loops** XY graph. Beat-to-beat summary data from Left Ventricular Pressure **(LVP)** and Left Ventricular Volume **(LVV)** signals are displayed in real-time and can be saved, during acquisition, to the online **Journal**. The summary data contained in the **Journal** can be saved, copied into a spreadsheet, or exported to an external application for further analysis.

While recording data, it is possible to measure certain beat-by-beat parameters using **Online Calculations**. Choosing **Online Calculations** from the **PV Loops** submenu displays the **PV Loop** toolbar above the uppermost channel.

▽ PVLoop:	#	HR	ESP	EDP	Pmax	Pmin	dPmax	dPmin	Vmax	Vmin	ESV	EDV	SV	CO	EF	SW
◄ ▭ ▭ ▭ ▭ ▭ ▭ ▶	0.000	0.000	0.000	0.000	0.000	0.000	0.000	0.000	0.000	0.000	0.000	0.000	0.000	0.000	0.000	0.000

The PV Loops real time toolbar.

Online PV calculations include: Sequential beat number (**#**), Heart rate (**HR**), End-systolic pressure **(ESP)**, End-diastolic pressure **(EDP)**, Maximum pressure **(Pmax)**, Minimum pressure **(Pmin)**, Maximum dP/dt **(dPmax)**, Minimum dP/dt **(dPmin)**, Maximum volume **(Vmax)**, Minimum volume **(Vmin)**, End-systolic volume **(ESV)**, End-diastolic volume **(EDV)**, Stroke volume **(SV)**, Cardiac outpu**t (CO)**, Ejection fraction **(EF)**, and Stroke work **(SW)**.

Clicking on the down arrow on the left side of the **PV Loops** toolbar will display a submenu with three choices:

- **Setup**: Opens the **Online PVLoop Setup** dialog.

- **AutoSize**: Adjusts the size of the **PV Loop** toolbar title and data boxes.

- **Set Font Size**: Changes the size of the font in the **PV Loop** toolbar data boxes.

The criteria for setting up the **PV Loops** online calculations are entered into the **Online Calculation Setup** dialog.
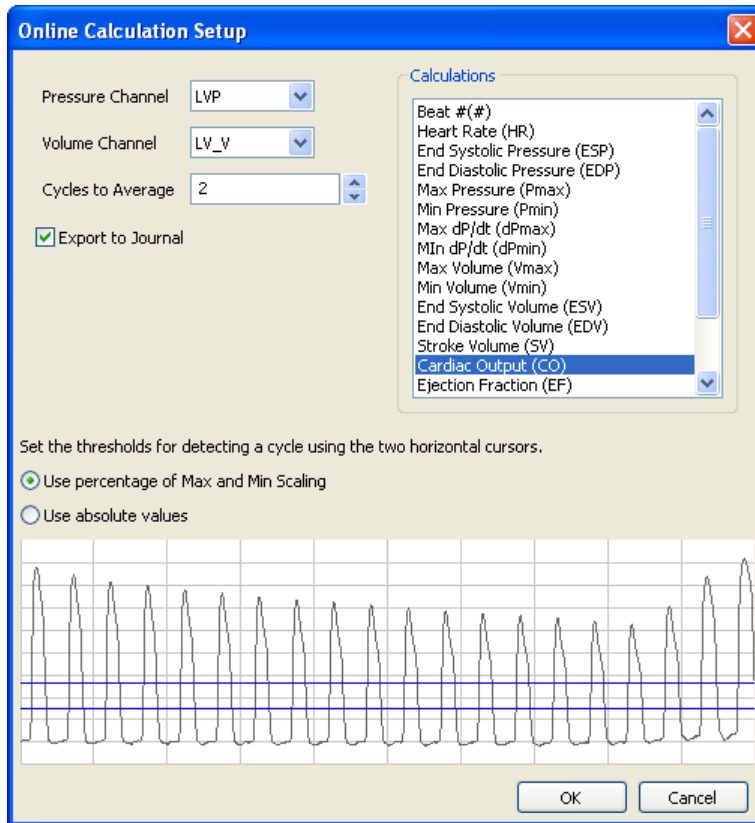
The Online Calculation Setup dialog.

To configure the **Online Calculation Setup** dialog:

- Choose the **Pressure** and the **Volume** channels, the calculations to be performed online, and whether these calculations should be exported to the **Journal**.

- The **Volume** channel may be a recording from a conductance catheter, or a recording in which ventricular volume has been determined in some other way, for example, from **Sonomicrometry** measurements (refer to page 81 in **Chapter 6: Computed Channels**).

- When the **Pressure** channel is selected, its graph will appear in the dialog. Adjust the two horizontal threshold lines so that they are between each cycle's maximum and minimum. *LabScribe2* uses the positive threshold crossing from below the lower threshold to above the upper threshold to determine the cycle.

- In order to compensate for variation from cycle to cycle, it is possible for *LabScribe2* to average a user-selected number of sequential cycles. This number should be entered in the **Cycles to Average** text box.

- Click **OK**. The dialog will close and the selected calculations will be displayed in the data boxes of the **PV Loops** toolbar as data are recorded.

PVLoop Automarks

Choosing **AutoMark** from the **PV Loops** submenu opens the **PV Loop AutoMark Dialog**, in which the **PV Loop** parameters to be calculated, marked, and exported to the **Journal** for further analysis can be configured.
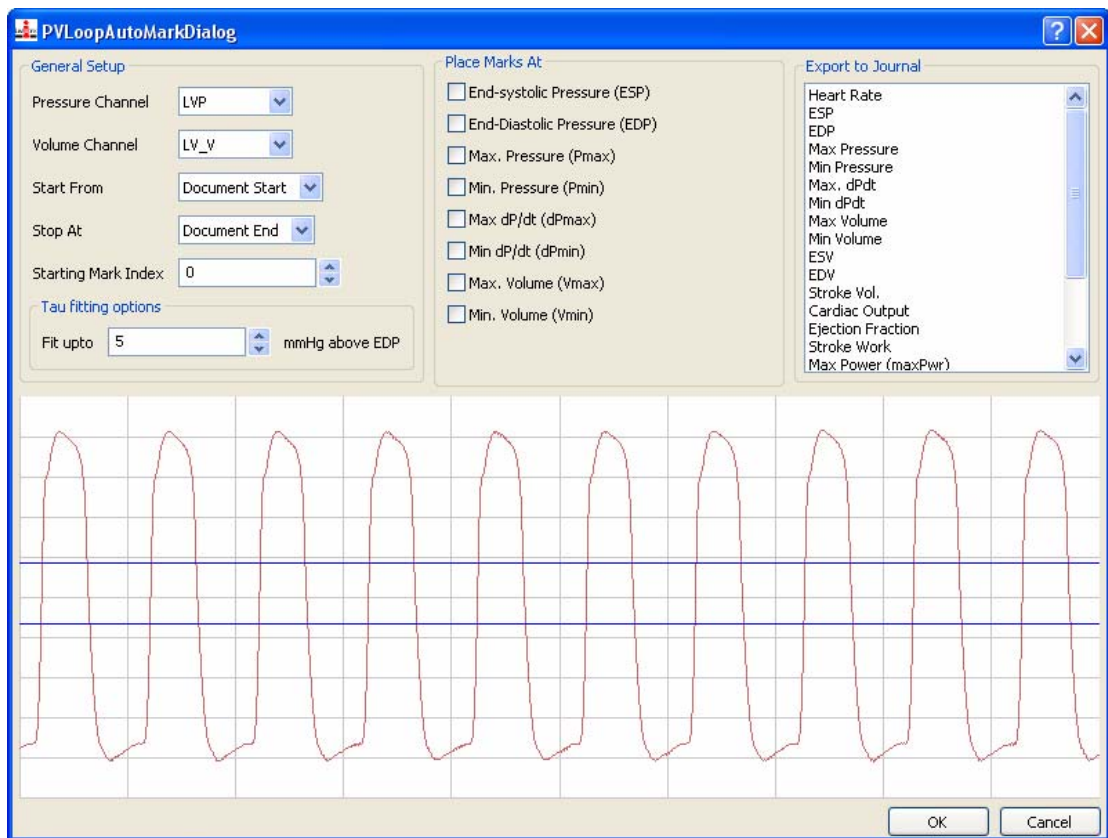
The PVLoop Automark Dialog.

To configure the **PV Loop Automark Dialog**, enter the following information:

- **Pressure Channel**: Channel with pressure data.

- **Volume Channel**: Channel with volume or conductance data.

- **Start From**: Location in the data file to start the analysis. The start of the document or an existing mark can be used to set the start location.

- **Stop At**: Location in the data file to end the analysis. The end of the document or an existing mark can be used to set the stop location.

- **Starting Mark Index**: Marks placed in the record have a number which corresponds to the cycle number. The default cycle number starts at zero; a change from the default starting cycle number can be made here.

- **Tau** fitting options: Enter the value above end-diastolic pressure (**EDP**) that should be used as the **Tau stop** point in the calculation of the best fit line to be used for the calculation of **Tau**. **Tau** is the isovolumic time constant and is calculated using the line from **dPmin** to this user-defined point.

- When the pressure channel is selected, it will appear in the graph area of the dialog. Adjust the two threshold lines so that they pass between the minimum and maximum cyclic values. *LabScribe2* uses the positive threshold crossing from below the lower threshold to above the upper threshold to determine the cycle.

- **Place Marks At**: Choose the points of interest the program should mark in the record. These **AutoMarks** can be used to verify that the program has detected the correct points of interest.

- **Export to Journal**: Choose the values to be exported to the **Journal** for each cycle. In addition to the online functions, several additional parameters can be calculated and sent to the **Journal**. These are: Maximum Power (**maxPwr**), Preload-adjusted Maximum Power (**plPwr**), Arterial Elastance (**Ea**), and three derivations of **Tau**, the isovolumic relaxation time constant: **Weiss**, **Glantz**, and **Mirsky**.

Offline Calculations

Offline, *LabScribe2* can calculate virtually every **PV Loop** derived parameter of cardiac function. XY graphs displaying relationships among the parameters can also be displayed. All calculations can be exported or copied to Excel or other spreadsheets for further analysis. The graphs can all be copied as images to include in presentations or manuscripts.

To perform **Offline Calculations**:

- Choose **Offline Calculations** from the **PV Loops** submenu to open the **PV Loop Calculations Dialog**. The panels of this dialog can be resized by moving the mouse cursor over the boundaries until a double-headed arrow appears, and dragging the boundaries to resize the panels.
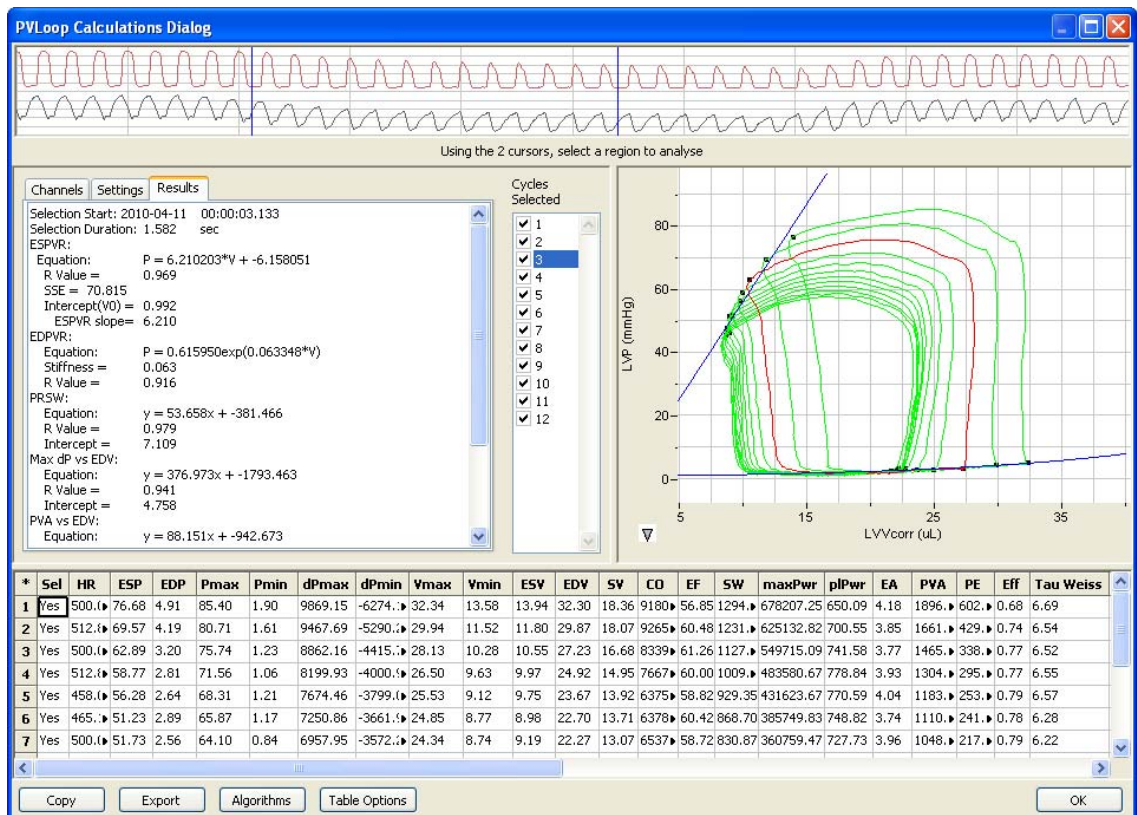
The PV Loops Calculations Dialog.

There are three tabbed panels used in setting up the dialog: **Channels**, **Settings**, and **Results**.

On the **Channels** panel:
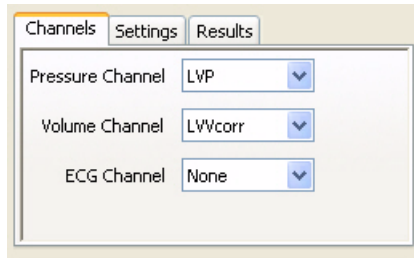
- Select the pressure and the volume channels. Optionally, an **ECG** channel can also be chosen.

PV Loops Channels dialog..

Define the region of interest by adjusting the two cursors in the channel display area at the top of the **PV Loops Calculations Dialog**.

Once the **Pressure** and **Volume** channels are designated, the **PV Loops** will be displayed in the XY graph area for all the cycles in the selection. The highlighted cycle in the **Cycles Selected** window is shown in red, while all other selected cycles are displayed in green. This makes it easy to identify individual cycles for inclusion or exclusion from the analysis. Cycles can be deselected (or selected) by clicking on the check box to the left of the cycle number. The UP and DOWN arrows on the computer keyboard can be used to move quickly through the individual cycles.

On the tabbed **Settings** dialog:
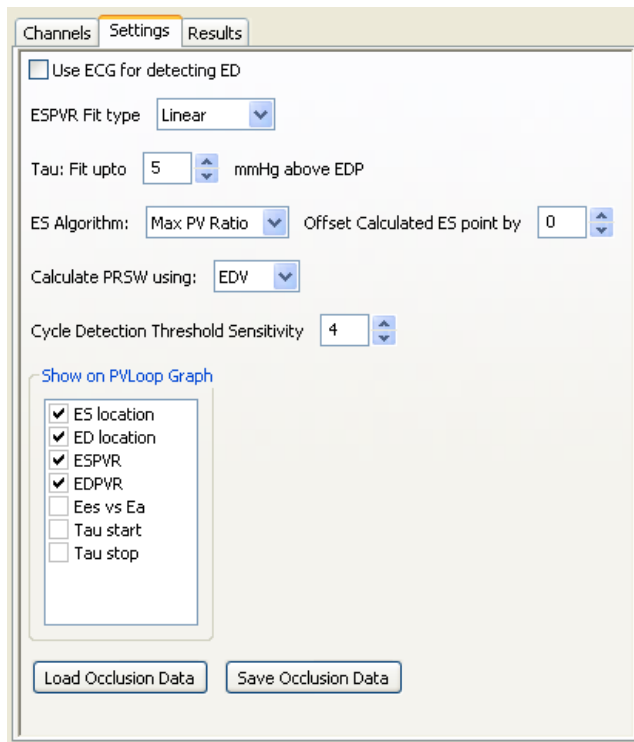
- Check whether you wish to use an **ECG** recording for the detection of end-diastole (**ED**).

- Choose whether a linear or an exponential fit should be applied to the End-Systolic Pressure-Volume Relationship (**ESPVR**) data, using the **ESPVR Fit Type** drop-down box.

- Enter the stop value to be included in the best fit line used for the determination of **Tau**. **Tau** is the isovolumic time constant and is calculated using the line from **dPmin** to this user-defined point.

- Choose the value in each cycle that should be used to compute end-systole (**ES**), and how far to offset **ES** from the specified value. By default, the maximum Pressure to Volume ratio is used to identify **ES**.

PV Loops Settings dialog.

- Specify the value (**EDV** or **Vmax**) that should be used to calculate the preload recruitable stroke work, **PRSW**.

- Enter the number of cycles that should be used to determine the **Cycle Detection Threshold Sensitivity**. By default, a threshold sensitivity of 4 is used to detect cycles (from the **dP/dt** trace). If each cycle is not being detected properly, the sensitivity can be adjusted.

- Enter into the **Show on PV Loop Graph** box those parameters you wish to see displayed on the XY Graph. This feature allows a visual check that you have configured these parameters as desired, and how changes to them affect relevant relationships.

    - The **ES** (End-systole) location for each cycle is based on the **ES algorithm** entered in the **Settings**.

    - The **ED** (End-diastole) location is either the **EDV** derived from the **LVP** signal or the **ECG** signal (if selected).

    - The line of best fit for the **ES** (End-systolic) values is shown as the end-systolic pressure-volume relationship (**ESPVR**) curve. This will be a linear or exponential fit, depending on what has been specified in the **Settings**.

    - The line of best fit for the **ED** (End-diastole) values is shown as the end-diastolic pressure-volume relationship (**EDPVR**) curve. This relationship will always be curvilinear. If pressures fall below zero, a message will appear requesting that the data be offset in the positive direction in order to perform the calculation.

    - The **Ees** (End-systolic elastance) vs. **Ea** (Arterial elastance) relationship is shown for the currently selected (red) cycle.

    - **Tau start** and **stop** are the start and stop points used to fit the line from which **Tau** is calculated. **Tau stop** is defined in the **Settings**.

- Use the **Load Occlusion Data** and **Save Occlusion Data** buttons to load and save the data from the section of the trace where occlusion occurs. This allows the user to perform one occlusion and use those results for analysis of another portion of the recording. This is useful when calculating **PVA**, **PE**, and **Eff** from the steady state data in other locations in the recording.

On the tabbed **Results** panel:

- The equations defining **ESPVR**, **EDPVR**, **PRSW**, **Max dP** vs. **EDV**, **PVA** vs. **EDV**, and **PVA** vs. **ESP** are displayed, as is **Emax.**

- Additional text can be entered into this panel.

Clicking the arrow to the lower left of the XY graph window will open a menu with options for the XY graph. The options are:

Copy Graph
View PVLoops
View PRSW
View Max dp vs EDV
View PVA vs EDV
View PVA vs ESP
View E(t) vs Time
View Markers
Export E(t) vs Time data
Set X axis Scale
Set Y axis Scale
AutoScale X axis
AutoScale Y axis

- **Copy graph**: Copies the current XY graph to the clipboard. It can then be pasted into the **Journal** or an external application.

- **View PV Loops**: Displays the **PV Loops** from the checked cycles in the XY graph window.

- **View PRSW**: Displays the preload-recruitable stroke work (**PRSW**) line, as defined in the **Settings**. By default, **PRSW** is calculated by plotting **SW** on the y-axis and **EDV** on the x-axis. **Vmax** can be used as an alternative to **EDV** if desired.

- **View Max dp vs. EDV**: Displays the XY graph of maximum dP/dt (**Max dP**) vs. End-diastolic volume (**EDV**).

- **View PVA vs. EDV**: Displays the XY graph of the pressure-volume area (**PVA**) vs. end-diastolic volume (**EDV**).

XY graph window menu.

- **View PVA vs. ESP**: Displays the XY graph of the pressure-volume area (**PVA**) vs. end-systolic pressure (**ESP**).

- **View E(t) vs. Time**: Displays time-varying elastance (**E(t)**) vs. **Time**.

- **View Markers**: Displays a graph of time (x-axis) vs. pressure and volume (y-axis) with markers positoned at **ED**, **Max dP**, **ES**, and **Min dP**. The specific cycle displayed is determined by the position of the left cursor in the data recording at the top of the dialog.

- Export **E(t)** vs. **Time** data: Exports the **E(t)** vs. **Time** data as a tab (*.txt) or comma (*.csv) separated text file.

- **Set X-axis Scale**, **Set Y-axis scale**: Allows the user to set the x-axis and y-axis scales.

- **AutoScale X-axis**, **AutoScale Y-axis**: Optimizes the display scale of the x-axis or y-axis of the XY graph.

Offline PV Calculations displayed in the **Data Table** include: Heart rate (**HR**), End-systolic pressure (**ESP**), End-diastolic pressure (**EDP**), Maximum pressure (**Pmax**), Minimum pressure (**Pmin**)**,** Maximum dP/dt (**dPmax**), Minimum dP/dt (**dPmin**), Maximum volume (**Vmax**), Minimum volume (**Vmin**), End-systolic volume (**ESV**), End-diastolic volume (**EDV**), Stroke volume (**SV=EDV-ESV**)**,** Cardiac output (**CO=SV*HR**), Ejection fraction (**EF=100*SV/EDV**), Stroke work (**SW**=area within the **PV Loop** for each cardiac cycle), Maximum power (**maxPwr**), Preload-adjusted maximum power (**plPwr**), Arterial elastance (**Ea**), Pressure-volume area (**PVA**), Potential energy (**PE**), Efficiency (**Eff**), and four derivations of **Tau**, the Isovolumic relaxation time constant: **Weiss**, **Logistic**, **Glantz**, and **Mirsky**.

There are four buttons across the bottom of the **PVLoops Calculations Dialog**: **Copy**, **Export**, **Algorithms** and **Table Options**:
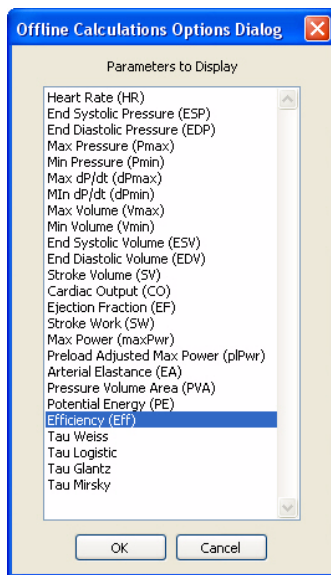
Complete list of Data Table parameters.

- All the calculated data in the data table can be copied to the clipboard by clicking the **Copy** button, or exported by clicking the **Export** button. The data are exported as a tab (*.txt) or comma separated (*.csv) text file, and the XY graph can be exported as a Portable Network Graphics (*.png) or JPEG (*.jpg) image.

- Clicking **Table Options** opens the **Offline Calculations Options Dialog**, which lists the functions from which the **Data Table** parameters can be chosen.

  - Clicking the asterisk at the upper left of the **Data Table** displays the **autosize** option. Clicking **autosize** will optimize the size of the **Data Table** boxes.

  - The **Data Table** displays the chosen calculated values for each of the cycles checked in the **Cycles Selected** window, as well as the mean, standard deviation, and range of each of the chosen parameters averaged over all the selected segments.

  - Clicking **OK** saves the settings for future **PV Loops** analyses.

- Clicking **Algorithms** opens an information window describing the mathematical equations used to compute the offline parameters.

Calibration of Conductance Volume Data

Conductance catheters make possible real time pressure-volume ventricular measurements in even very small animals. However, the raw conductance measurements need to be converted to volumes, and it is necessary to correct for the conductance contributions (parallel conductance) of surrounding heart structures. One method of calibration uses cuvettes of known volumes. When conductance catheters are inserted in cuvettes of known volumes, it is possible to convert the conductances to calibrated volumes. It is also possible to get calibrated volumes from conductance measurements through the use of Baan's equation, which uses variables including the distance between the electrodes, the resistivity of blood, and the conductance measurements to calculate calibrated volumes. A slope factor **Alpha** is also necessary to adjust for the physical shape of a heart and the nature of the electric field itself.

To correct for the parallel conductance, a bolus of hypertonic saline is injected into the blood before it reaches the ventricle and the change in conductance as it passes through the ventricle is measured and used to determine the parallel conductance (or volume), as the parallel conductance won't change while the blood conductance will change as a result of the saline passing through the ventricle.
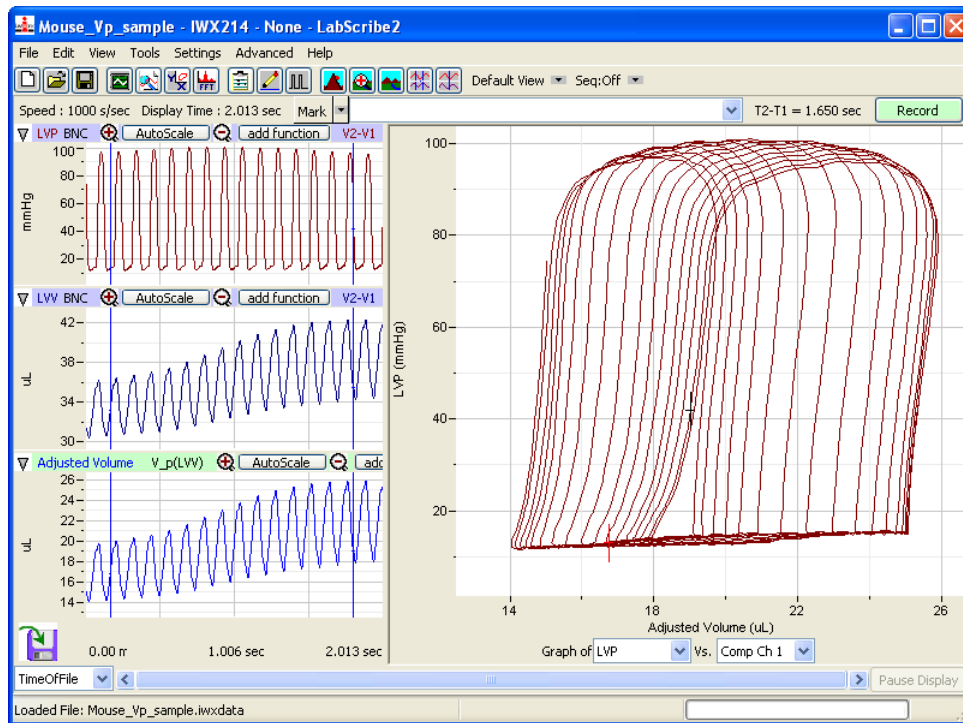
Pressure-volume loops shift to the right as a bolus of hypertonic saline passes through the left ventricle. Data from these loops are used to calculate parallel volume or conductance and compute a corrected, calibrated, ventricular volume.

Selecting the **PV Loops** submenu from the **add functions** list opens a dialog in which **Parallel Volume** or **Vol.-Baan's Equation** can be chosen. To use the **Parallel Volume** function to determine the parallel volume, it is necessary to use a calibrated volume channel. To use the **Vol.-Baan's Equation** function, it is possible to use a raw conductance channel. The resulting function channels will be volume-calibrated and corrected for parallel conductance.

To use the **Parallel Volume** function:
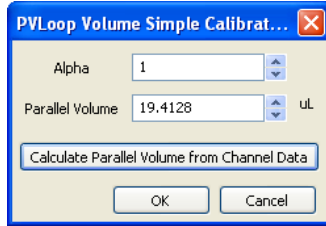
The PVLoop Volume Simple Calibration dialog.

- Click on **add function** on the **Volume** channel.

- In the dialog that opens, select **PV Loops**, and then **Parallel Volume** from the submenu.

- Enter the **Alpha** slope correction value, and click on **Calculate Parallel Volume from Channel Data**, opening the **PVLoop Volume Calibration Dialog**.

- In the **PVLoop Volume Calibration Dialog**:

  - Choose the **Pressure** and calibrated **Volume** channel.

  - Using the cursors in the **Pressure** and **Volume** trace window at the top of the dialog, select the region corresponding to the saline bolus injection.
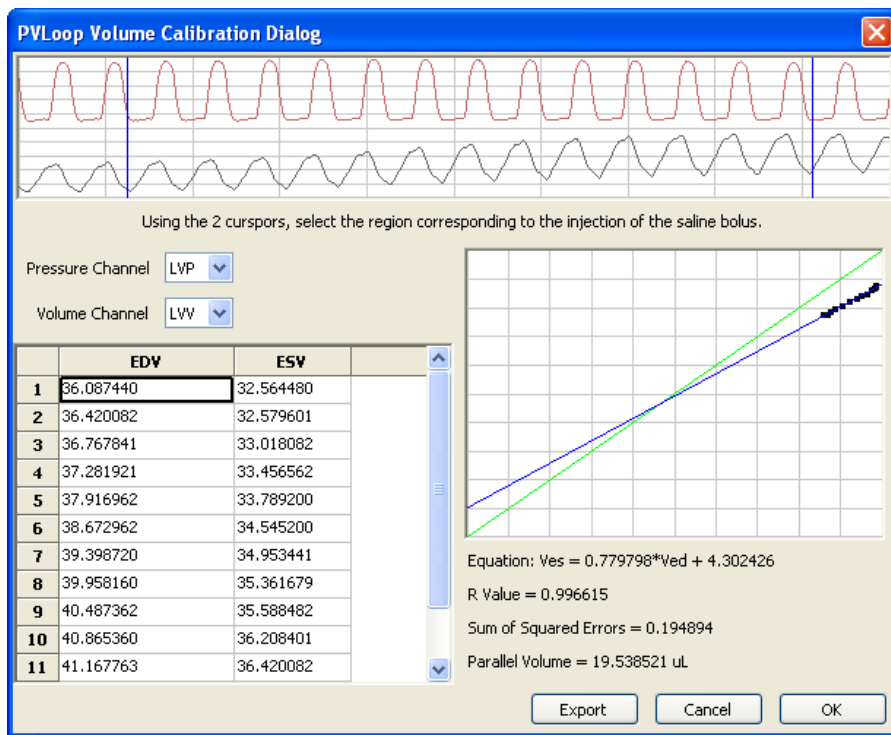
The calculation of parallel volume by selecting a sequence of PV Loops that show the injection of a saline bolus.

  - In the **EDV** vs. **ESV** XY graph, the line created by the shifting conductance values caused by the saline bolus will cross the identity line at the parallel volume. The equation of the data line, its goodness of fit, and the parallel volume are displayed below the XY graph window. Click **OK**.

  - The parallel volume will now be displayed in the **PVLoop Volume Simple Calibration** dialog. Click **OK** and a corrected volume channel is added to the recording.

To use the **Vol.-Baan's Equation** function:

- Click on **add function** on a conductance (**Volume**) channel.

PVLoop Volume Calibration dialog: Baan's Equation.

- In the dialog that opens, select **PV Loops**, and then **Vol.-Baan's Equation** from the submenu, opening the **Baan's Equation** version of the **PV Loop Volume Simple Calibration** dialog.

- Enter the **Alpha** slope correction factor, the resistivity of the blood (**Rho**) and the inter-electrode distance (**Length**).

- Click on **Calculate Parallel Conductance from Channel Data**, opening the **PVLoop Volume Calibration Dialog**.

- Choose the **Pressure** and conductance (**Volume**) channel.

- Using the cursors in the **Pressure** and **Volume** trace window at the top of the dialog, select the region corresponding to the saline bolus injection.

- In the **EDV** vs. **ESV** XY graph, the line created by the shifting conductance values caused by the saline bolus will cross the identity line at the parallel conductance. The equation of the data line, its goodness of fit, and the **Parallel Conductance** are displayed below the XY graph window. Click **OK**.

- The parallel conductance will now be displayed in the **PVLoop Volume Simple Calibration** dialog. Click **OK** and a corrected volume channel is added to the recording.

Blood Pressure

Blood pressure parameters are detected and functions derived from those parameters are computed in the **Blood Pressure** analysis routine. The **Blood Pressure** module requires a separate license. The first time you select **Blood Pressure**, you will be asked for a username and a serial number. Contact *iWorx Systems* for more information.
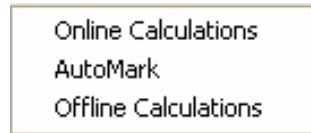
When **Blood Pressure** is chosen from the **Advanced** menu, a submenu opens, displaying three choices: **Online Calculations**, **Automark**, and **Offline Calculations**.

Blood Pressure submenu.

Online Blood Pressure Calculations

While recording data, some beat-by-beat parameters can be calculated and displayed online using **Online Calculations**. Choosing **Online Calculations** from the **Blood Pressure** submenu opens the **Blood Pressure** toolbar above the uppermost channel. The function titles and the corresponding data values are displayed in the toolbar's data boxes.

The Blood Pressure online toolbar.

The online parameters are:

- **Heart Rate**: 60/period of each cycle.

- **Max Pressure**: Maximum pressure, also the systolic value.

- **Min Pressure**: Minimum pressure, also the diastolic value.

- **Time at Notch**: Time at the dicrotic notch.

- **Max dP/dT**: Maximum slope.

- **Min dP/dT**: Minimum slope.

- **Mean Pressure**: Mean pressure.

- **MAP**: Mean arterial pressure.

- **Contractility Index**: (**Max dP/dT**)/(**Pressure** at **Max dP/dT**).

- **Relaxation Index**: (**Min dP/dT**)/(**Pressure** at **Min dP/dT**).

- **Pulse Height**: Amplitude of each beat.

Clicking on the down arrow on the left side of the **Blood Pressure** toolbar will display a submenu with three choices:

- **Setup**: Opens the **Online Blood Pressure Setup** dialog.

- **AutoSize**: Adjusts the size of the **Blood Pressure** toolbar title and data boxes.

- **Set Font Size**: The user can change the size of the font in the **Blood Pressure** toolbar data boxes.

The criteria for setting up the **Blood Pressure** online calculations are entered into the **Online Calculations Setup** dialog.
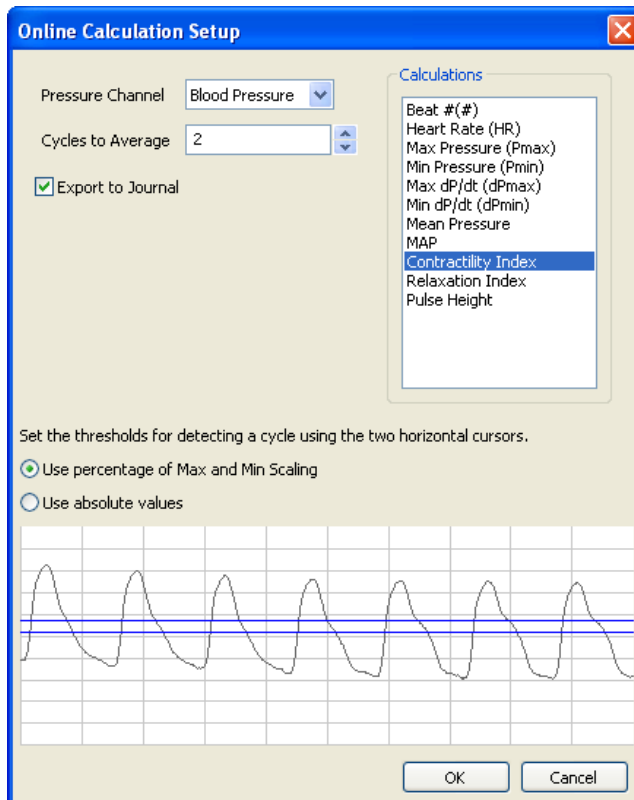
The Online Blood Pressure Setup dialog.

To configure the **Online Blood Pressure Setup** dialog:

- Choose the **Pressure** channel to be used.

- Choose the calculations to be performed online from the **Calculations** box, and whether these calculations should be exported to the **Journal**.

- When the **Pressure** channel is selected, it's graph will appear in the dialog. Set the threshold lines so that they are between the cyclic minimum and maximum data values. *LabScribe2* uses the positive threshold crossing from below the bottom cursor to above the top cursor to determine the cycle.

- In order to compensate for variation from cycle to cycle, it is possible for *LabScribe2* to average a user-selected number of sequential cycles. This number should be entered in the **Cycles to Average** text box.

- Click **OK**. The dialog will close and the selected calculations will be displayed in the data boxes of the **Blood Pressure** toolbar as data are recorded.

Blood Pressure Automarks

The **AutoMark Blood Pressure Characteristic Points Dialog** can be opened by selecting **AutoMark** from the **Blood Pressure** sub-menu in the **Advanced** menu.
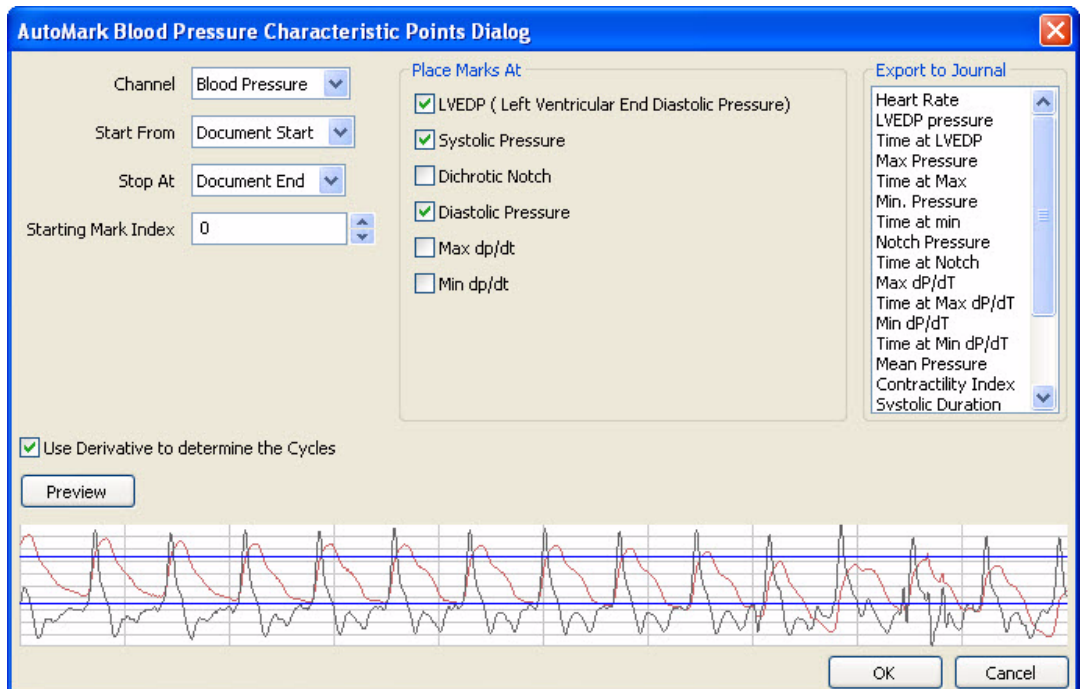
The Automark Blood Pressure Characteristic Points Dialog, set up for the display of LVEDP, Systolic Pressure and Diastolic Pressure.

To configure **Blood Pressure AutoMarks**:

- Select **AutoMark** from the **Blood Pressure** sub-menu, opening the **AutoMark Blood Pressure Characteristics Points Dialog.** In this dialog, enter:

  - **Channel**: Choose a channel with blood pressure data.

  - **Start From**: Location in the data file to start the analysis. The start of the document or an existing mark can be used to set the start location.

  - **Stop At**: Location in the data file to end the analysis. The end of the document or an existing mark can be used to set the stop location.

  - **Starting Mark Index**: Marks placed in the record have a number which corresponds to the cycle number. The default cycle number starts at zero; the number can be changed here.

  - When the **Pressure** channel is selected, it will show up in the graph. Set the threshold lines so that they are between the cyclic minimum and maximum data values. Data crossing from below the lower threshold to above the upper threshold is used to determine the cycle. In some traces, using the derivative of the blood pressure makes it possible to set the threshold values more easily.

  - **Place Marks At**: Allows the user to choose the points of interest the program should mark in the record. This can be used to verify that the program has detected the correct points of interest.

  - **Export to Journal**: Allows the user to choose the values to be exported to the **Journal** for each cycle. The choices consist of all online and offline functions.

Offline Blood Pressure Calculations

In addition to the online **Blood Pressure** calculations, *LabScribe2* can perform additional offline **Blood Pressure** calculations, and display an XY graph of **Blood Pressure** as a function of its derivative.
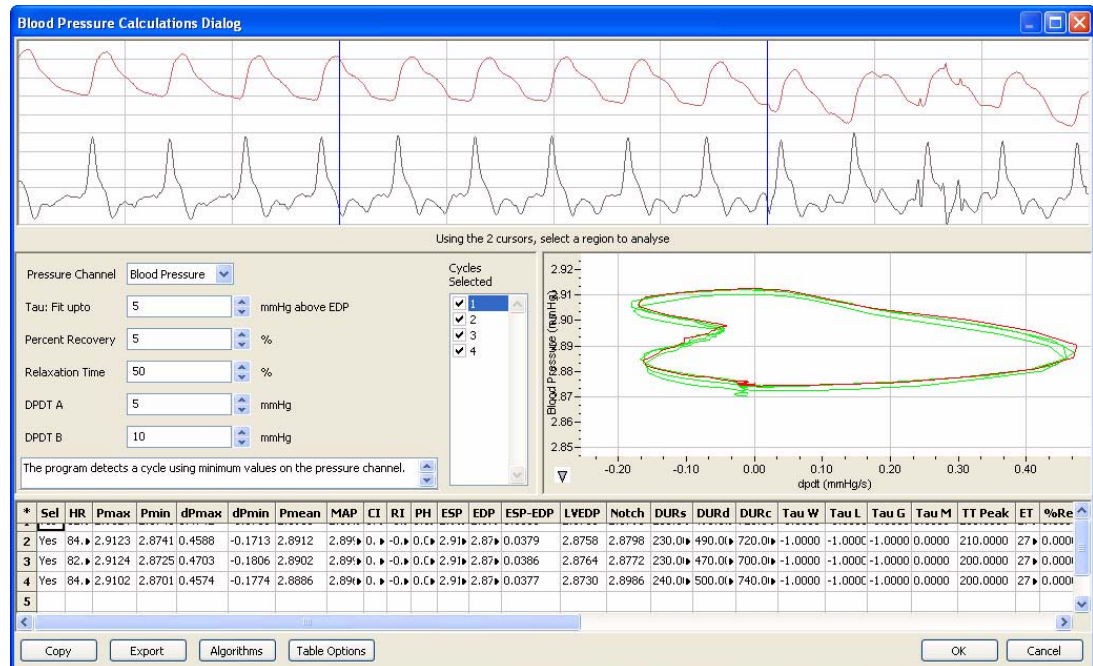
The Blood Pressure Calculations Dialog.

To perform **Offline Calculations**:

- Choose **Offline Calculations** from the **Blood Pressure** submenu, opening the **Blood Pressure Calculations Dialog**. The panels of this dialog can all be resized by moving the mouse cursor over the boundaries until a double-headed arrow appears, and dragging the boundaries to resize the panels.

- Select the **Pressure** channel. The region of interest is defined by the data between the two cursors in the channel display window.

- **Tau Fit**: Allows the user to adjust the uppermost point that should be used to create the best fit line from which **Tau** (the time constant of blood pressure decrease during diastole) is computed.

- **Percent Recovery:** Allows the user to determine at what time point in the cycle a specified **Percent Recovery** occurs. This value is displayed in the data table.

- **Relaxation Time**: Allows the user to determine at what time point in the cycle a specified percentage of **Relaxation Time** occurs. This value is displayed in the data table.

- **dPdt A, dPdt B**: Allows the user to determine the derivative at up to two different specified blood pressures.

The XY graph plots blood pressure as a function of its derivative (the rate of change in blood pressure). Clicking on the arrow in the lower left corner of the XY graph pane opens a menu that offers options for scaling both the x-axis and y-axis, as well as an option to copy the graph to the clipboard. The menu items are:

- **Copy graph**: Copies the current XY graph to the clipboard. It can then be pasted into the journal or an external application.

- **Set X-axis Scale**, **Set Y-axis scale**: Allows the user to set the x-axis and y-axis scales.

- **AutoScale X-axis**, **AutoScale Y-axis**: Optimizes the display scale of the x-axis or y-axis of the XY graph.

- The axes can also be re-scaled by left-clicking and dragging either of the axes.

In the graph, **Blood Pressure XY Loops** are displayed for the cycles defined by the cursors in the raw data graph and selected in the **Cycles Selected** window. The highlighted cycle in the **Cycles Selected** window is shown in red, while all other selected cycles are displayed in green. This makes it easy to identify individual cycles for inclusion or exclusion from the analysis. Cycles can be deselected (or selected) by clicking on the check box to the left of the cycle number in the **Cycles Selected** box. The UP and DOWN arrows on the computer keyboard can be used to move quickly through the individual cycles.

There are four buttons across the bottom of the **Blood Pressure Calculations Dialog**: **Copy**, **Export**, **Algorithms** and **Table Options**:

- All the calculated data in the **Data Table** can be copied to the clipboard by clicking the **Copy** button, or exported by clicking the **Export** button. The data are exported in a tab (*.txt) or comma separated (*.csv) text file, and the XY graph can be exported as a Portable Network Graphics (*.png) or JPEG (*.jpg) image.

- *LabScribe2* is able to calculate a large number of blood pressure calculations for each cycle. By clicking **Table Options** at the bottom of the **Blood Pressure Calculations Dialog**, the **Offline Calculations Options Dialog** opens, and calculations to be displayed in the dialog data table can be chosen from the list of all possible calculations. The **Data Table** displays the chosen calculated values for each cycle.

  - Clicking the asterisk at the upper left of the **Data Table** displays the **autosize** option. Clicking **autosize** will optimize the size of the **Data Table** boxes.

  - The **Data Table** displays the chosen calculated values for each of the cycles checked in the **Cycles Selected** window, as well as the mean, standard deviation, and range of each of the chosen parameters averaged over all the selected cycles.

  - Clicking **OK** in the **Offline Calculations Options Dialog** saves the settings for future **Blood Pressure** analyses.

- Clicking **Algorithms** opens an information window describing the mathematical equations used to compute a number of the offline parameters.
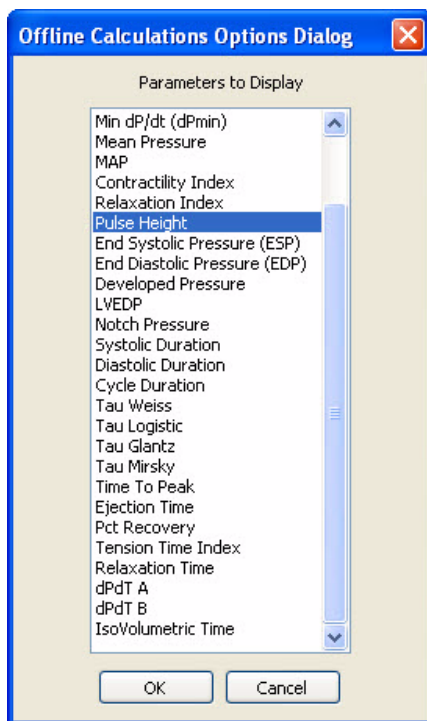
Offline Calculations Options DIalog.

Offline calculations:

- **Heart Rate**: 60/period of each cycle.

- **Max Pressure**: Maximum value of the pressure channel for that cycle.

- **Min Pressure**: Minimum value of the pressure channel for that cycle.

- **Time at Notch**: Time at the dicrotic notch.

- **Max dP/dt**, **Max dP/dt**: The smoothed derivative of the pressure channel is calculated, using 2 points on either side of any given point. **Max dP/dt** is the steepest slope as pressure increases, and **Min dP/dt** is the steepest slope as pressure decreases.

- **Mean Pressure**: Mean pressure in the cycle.

- **MAP**: Mean arterial pressure.

- **Contractility Index**: (**Max dP/dt**)/(**Pressure** at **Max dP/dt**).

- **Relaxation Index**: (**Min dP/dt**) / (**Pressure** at **Min dP/dt**).

- **Pulse Height**: Amplitude of each beat.

- **End Systolic Pressure (ESP)**: Value of the Pressure channel at the end of systole.

- **End Diastolic Pressure (EDP)**: Value of the Pressure channel at end of diastole.

- **Developed Pressure:** Difference between the systolic pressure and the left ventricular end diastolic pressure (SYS-LVEDP).

- **LVEDP**: Left Ventricular End Diastolic Pressure.

- **Notch Pressure**: Pressure at the dicrotic notch.

- **Diastolic Duration**: Time between **Min dP/dt** and the end of the cycle.

- **Cycle Duration**: Time between any two analogous points in two cycles (often the end of diastole).

- **Tau Weiss**: P(t) = Aexp(-t/**Tau**).

- **Tau Logistic**: P(t)= Aexp(-t/**Tau**) + B.

- **Tau Glantz**: Regression of **dP/dt** vs. **Pressure**.

- **Tau Mirsky**: Time required for **LV pressure** to fall to one-half of its value at end-systolic pressure (**ESP**).

- **Time to Peak**: Time from the start of the cycle to the beat's maximum value.

- **Ejection TIme**: Time between the start of the cycle and the dichrotic notch.

- **Percent Recovery**: Designated percentage of the time it takes for the pressure to recover.

- **Tension Time Index**: The average pressure during systole multiplied by systolic duration.

- **Relaxation Time**: The time between **Min dP/dt** and the end of the cycle.

- **dPdT A**: Rate of pressure change as a function of time at user designated point A.

- **dPdT B**: Rate of pressure change as a function of time at user designated point B.

- **IsoVolumetric Time**: Duration of the isovolumetric contraction period.

ECG Analysis

The **ECG Analysis** module detects electrocardiogram parameters and computes functions based on those features. The **ECG Analysis** module requires a separate license. The first time you select **ECG Analysis**, you will be asked for a username and a serial number. Contact *iWorx Systems* for more information.
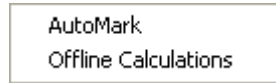
When **ECG Analysis** is chosen from the **Advanced** menu, a submenu opens, displaying two choices: **Automark** and **Offline Calculations**.

ECG Submenu.

Automark

The **AutoMark** detector is used to detect and identify the components of an **ECG**. Selecting **AutoMark** in the **ECG Analysis** submenu of the **Advanced** menu will open the **AutoMark ECG Dialog**.
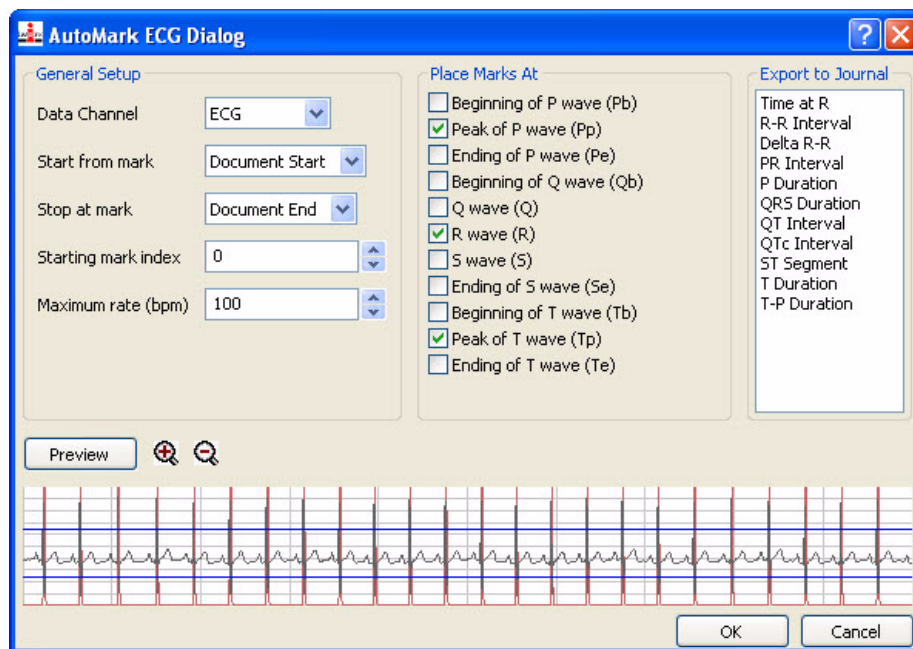
The AutoMark ECG Dialog set up to locate the R waves and the peaks of the P and T waves.

Configuring the AutoMark ECG Dialog

To configure the **AutoMark ECG Dialog**:

- Choose **AutoMark ECG** from the **Advanced** menu, opening the **AutoMark ECG Dialog**.

- In the **AutoMark ECG Dialog**, select the **ECG** raw data channel you wish to analyze.

- Choose the **Marks** where you want the analysis to **Start** and **Stop**. The **Document Start** and **End** are other options.

- In the **AutoMark ECG Dialog**, move the **Threshold** lines on the graph so that they pass through all the R waves above the R wave minimum and below the R wave maximum. Clicking **Preview** indicates where the R wave **Marks** will be set.

- In the **Starting Mark index** box, enter a number which corresponds to the starting cycle number. The default cycle number starts at zero; a change from the default starting cycle number can be made here.

- In the **Maximum rate (bpm)** box, enter the maximum heart rate likely to be registered in the recording.

- In the **Place Marks at:** section, select the ECG landmarks you wish to locate and mark by checking the appropriate checkboxes. These marks can be viewed and edited in the **Marks** window (accessed in the main **Toolbar**). The **Marks** window is organized so that the marks can be sorted by **Time** (ascending or descending), **Channel**, or **Content,** by clicking on the title of the appropriate column in the **Marks** table. Individual comments, or ranges of comments, may be selected or deselected using the SHIFT and CONTROL keys. Collections of selected comments can be exported or deleted.

- Select the calculations you wish to send to the **Journal** from the **Export to Journal** list.

- Click **OK** to place the **Marks** at the desired landmarks on the raw data channel and send the calculation results to the **Journal**.

The calculations that *LabScribe2* can perform based on the **AutoMark** locations are:

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

R-R Interval: Interval between consecutive R waves.

Delta R-R: Change in the **R-R Interval** from the preceding interval.

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.

QTc Interval: Corrected, normalized, **QT Interval: QTc=QT Interval/Square root of preceding R-R Interval.**

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 T wave to the end of the P wave.

These calculations can be exported to the **Journal**. From there, they can be exported or copied and pasted into other applications.

Beat-by-beat ECG analysis can also be performed from the **Channel menu** of an ECG raw data channel in the **Analysis Window**. Refer to the **ECG Calculations** section on page 84 of the **Analysis** chapter.

Offline Calculations

More sophisticated ECG analysis can be accomplished by selecting **Offline Calculations** from the **ECG Analysis** submenu and opening the **ECG Calculations Dialog**. The panels of this dialog can be resized by moving the mouse cursor over the boundaries until a double-headed arrow appears, and dragging the boundaries to resize the panels.
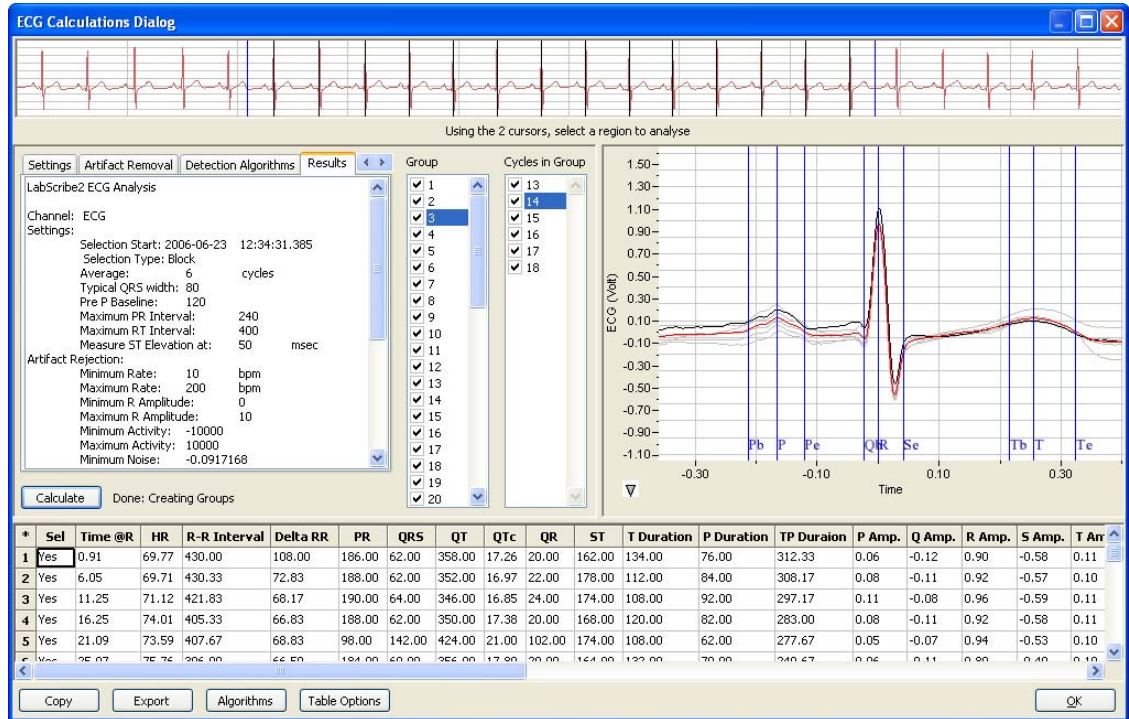
ECG Calculations Dialog.

There are five tabbed panels used in setting up the dialog: **Channels**, **Settings**, **Artifact Removal**, **Detection Algorithms**, and **Results**.

On the tabbed **Channels** dialog:
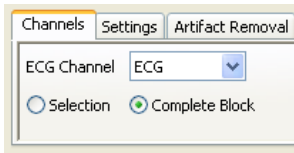
• Select the ECG channel you wish to analyze, and whether you want to analyze the entire block or the sample between the cursor.

ECG Calculations Channels Dialog.

On the tabbed **Settings** dialog:
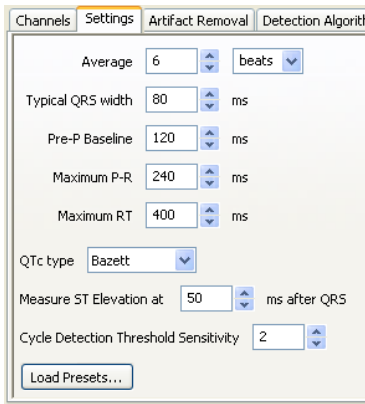
• Click **Load Presets...** to load default values appropriate to the selected ECG into the text boxes of the dialog.

• In order to compensate for variation from cycle to cycle, it is possible for *LabScribe2* to average a user-selected number of sequential cycles. This number should be entered in the **Average** text box.

• Enter the number of cycles that should be used to determine the **Cycle Detection Threshold Sensitivity**. If each cycle is not being detected properly, the sensitivity can be adjusted.

• Changes to the default values can be entered manually into the appropriate boxes.

ECG Calculations Settings Dialog.

On the tabbed **Artifact Removal** dialog:
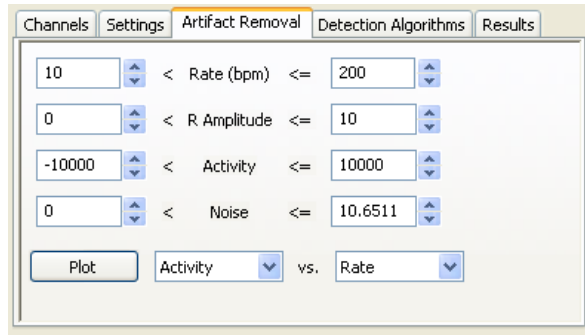
LabScribe2 can remove cycles that are likely to be misinterpreted due to artifacts in the recording. Individual cycles that fall outside expected values for **Rate**, **R Amplitude**, **Activity** and **Noise** will not be displayed or included in the averaged values. **Activity** and **Noise** are both measures of fluctuation from a continuous trace.

ECG Calculations Artifact Removal dialog.

Default values are entered in the **Artifact Removal** dialog. These values can be changed by entering new values in the text boxes manually, or by clicking on the **Plot** button to display the **Artifact Graph** in the graph area of the **ECG Calculations Dialog**. The parameters that are graphed are those entered in the **Plot** boxes of the **Artifact Removal** dialog. The red box in the **Artifact Graph** can be re-sized to exclude cycles represented by outlying data points.
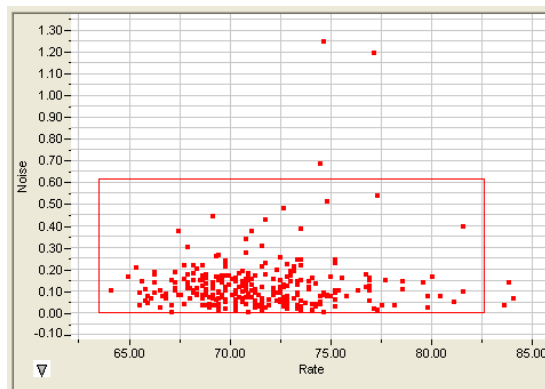
Artifact Graph.

On the tabbed **Detection Algorithms** dialog:
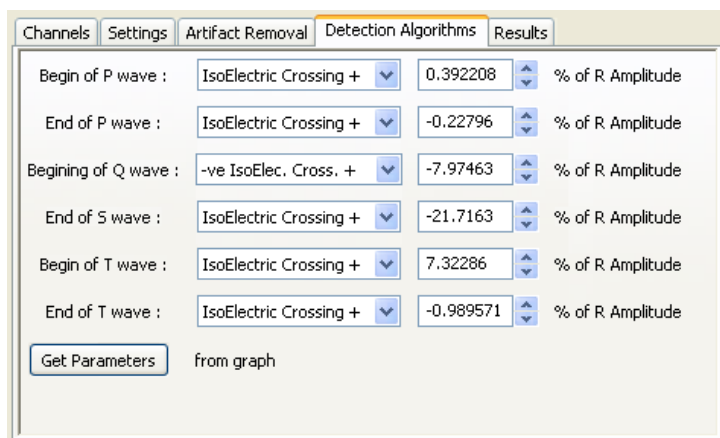
- The default algorithms used to place markers at specific locations in the ECG are displayed.

- The positions of the markers can be set manually by adjusting the markers in the ECG graph and clicking **Get Parameters from graph**.

ECG Calculations Detection Algorithms dialog.

On the tabbed **Results** panel:

- The information from the other tabbed dialogs is summarized.

- Additional text can be entered in this panel.

Clicking **Calculate** on the **ECG Calculations Dialog** will display the electrocardiogram in the graph window. A group of cycles is displayed at a time. The number of cycles displayed is determined by the number entered in the **Average** box of the **Settings** dialog. The group being displayed is indicated in the **Group** window, and the individual cycles in that group are indicated in the **Cycles in Group** window. The UP and DOWN arrows on the computer keyboard can be used to move quickly through the individual groups or cycles. In the graph, the currently selected cycle is in black, and the average of the group is in red. Any others are in gray. The calculated values displayed in the **Data Table** are the averages of each group.

Clicking the arrow to the lower left of the graph window displays the graph menu.
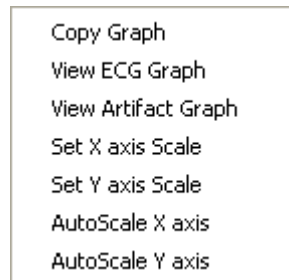
- **Copy graph**: Copies the current XY graph to the clipboard. It can then be pasted into the **Journal** or an external application.

- **View ECG Graph**: Displays the **ECGs** determined by the checked **Group** and **Cycles in Group**. The group's signal averaged ECG is also displayed in red.

- **View Artifact Graph**: Displays the graph configured in the **Artifact Removal** tabbed dialog.

- **Set X-axis Scale**, **Set Y-axis scale**: Allows the user to set the x-axis and y-axis scales.

- **AutoScale X-axis**, **AutoScale Y-axis**: Optimizes the display scale of the x-axis or y-axis of the XY graph.

- The axes can also be re-scaled by clicking and dragging either the x-axis or y-axis.

There are four buttons across the bottom of the **ECG Calculations Dialog**: **Copy**, **Export**, **Algorithms**, and **Table Options**.

All the calculated data in the **Data Table** can be copied to the clipboard by clicking the **Copy** button, or exported by clicking the **Export** button. The data are exported in a tab (*.txt) or comma separated (*.csv) text file, and the XY graph can be exported as a Portable Network Graphics (*.png) or JPEG (*.jpg) image.

- *LabScribe2* is able to calculate a large number of ECG calculations for each group of cycles. By clicking **Table Options** at the bottom of the **ECG Calculations Dialog**, the **Offline Calculations Options Dialog** opens and calculations to be displayed in the dialog data table can be chosen from the list of all possible calculations. The **Data Table** displays the chosen calculated values for each group of cycles.

  - Clicking the asterisk at the upper left of the **Data Table** displays the **autosize** option. Clicking **autosize** will optimize the size of the **Data Table** boxes.

  - Clicking **OK** in the **Offline Calculations Options Dialog** saves the settings for future **ECG** analyses.

- Clicking **Algorithms** opens an information window describing the mathematical equations used to compute a number of the offline parameters.

Data Table parameter options.

The **Offline Calculations** (averaged over the cycles in the group unless indicated otherwise) include:

Time @ R: The time at the peak of the first R wave in the group.

Heart Rate: 60/period of each cycle averaged over the cycles in the group.

R-R Interval: Average of the R-R intervals (peak to peak) of the cycles in the group.

Delta RR: Change of the current R-R interval from the R-R interval of the preceding group.

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

QRS: Averaged time from the beginning of the Q wave to the end of the S wave.

QT: Averaged time from the beginning of the Q wave to the end of the T wave.

QTc: Averaged corrected **QT Interval (QTc=QT Interval/Square root of preceding R-R interval).**

QR: Time from the beginning of the Q wave to the peak of the R wave.

ST: Averaged time from the end of the S wave to the start of the T wave.

P Duration: Averaged time from the beginning to the end of the P waves.

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

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

P Amplitude: Averaged amplitude of the P waves (from the baseline).

Q Amplitude: Averaged amplitude of the Q waves (from the baseline).

R Amplitude: Averaged amplitude of the R waves (from the baseline).

S Amplitude: Averaged amplitude of the S waves (from the baseline).

T Amplitude: Averaged amplitude of the T waves (from the baseline).

ST Height: The height of the point at the beginning of the ST segment (from the baseline).

QR Amplitude: Lowest point of the Q wave to the peak of the R wave.

T Area: Area between the T wave and the baseline from the start to the end of the T wave.

QAT (Q alpha T): The time from the Q wave to the peak of the T wave.

QATN (Q alpha T normalized): Time between the Q wave and the lowest point between the end of the S wave and the end of the T wave.

Metabolic

The Metabolic module uses breath-to-breath or mixing chamber oxygen content, carbon dioxide content, and air flow volume to calculate many metabolic parameters. Many of these parameters can be calculated online, and all the parameters can be viewed in a table. XY graphs can be computed to see how one parameter changes as a function of another. A gas analyzer is required to use the **Metabolic** module. The **Metabolic** module is licensed separately. Contact *iWorx Systems* for licensing information.

Breath by Breath: Online Calculations
Breath by Breath: Offline Calculations
Mixing Chamber: Offline Calculations

The Metabolic submenu.

The **Metabolic** functions can be accessed by selecting **Metabolic** from the **Advanced** menu, opening the **Metabolic** sub-menu.

The **Metabolic** sub-menu options are:

- **Breath by Breath: Online Calculations**: Calculations of several **Metabolic** parameters are made in real time based on the O_2, CO_2, and Air Flow raw data channels.

- **Breath by Breath: Offline Calculations**: Once recording has been completed, the O_2, CO_2, and air flow raw data channels are used to calculate many metabolic parameters. These calculated values can be displayed in a table, and XY plots can be created to see how one parameter changes as a function of another.

- **Mixing Chamber: Offline Calculations**: Samples from a mixing chamber can be taken periodically to examine samples of expired air averaged over specified durations. These raw data values are used to calculate metabolic parameters. These parameters can be displayed graphically, and graphs can be created looking at how one parameter changes as a function of a second.

Using Breath by Breath: Online Calculations

By selecting **Breath by Breath: Online Calculations** from the **Metabolic** submenu, the **Metabolic** toolbar with a number of metabolic parameters opens above the uppermost raw data channel. These values are continually updated during recording.

▽ Metabolic:	#	Time	Abs.VO2 (L)	Abs.VCO2(L)	Rel.VO2(ml/kg/min)	Rel.VCO2(ml/kg/min)	RER	REE	METS	FeO2Min (%)

The Metabolic Online Toolbar

Clicking on the down arrow on the left side of the **Metabolic** toolbar will display a submenu with three choices:

- **Setup**: Opens the **Online Metabolic Calculations Setup** dialog.

- **AutoSize**: Adjusts the size of the **Metabolic** toolbar title and data boxes.

- **Set Font Size**: The user can change the size of the font in the **Metabolic** toolbar data boxes.

To use **Breath by Breath: Online Calculations**:

- Set up the channels to be displayed in the **Main Window** through the **Spirometry** computed channels process. Refer to the **Spirometry** instructions beginning on page 68 in the **Chapter 6: Computed Channels**.

- While in the **Main Window**, select **Metabolic** from the **Advanced** menu. Choose **Breath by Breath: Online Calculations** from the **Metabolic** submenu. The **Metabolic** toolbar will open above the uppermost data channel.

- Click on the down arrow at the left of the data bar and choose **Setup** from the menu. This will open the **Online Metabolic Calculations Setup** dialog.
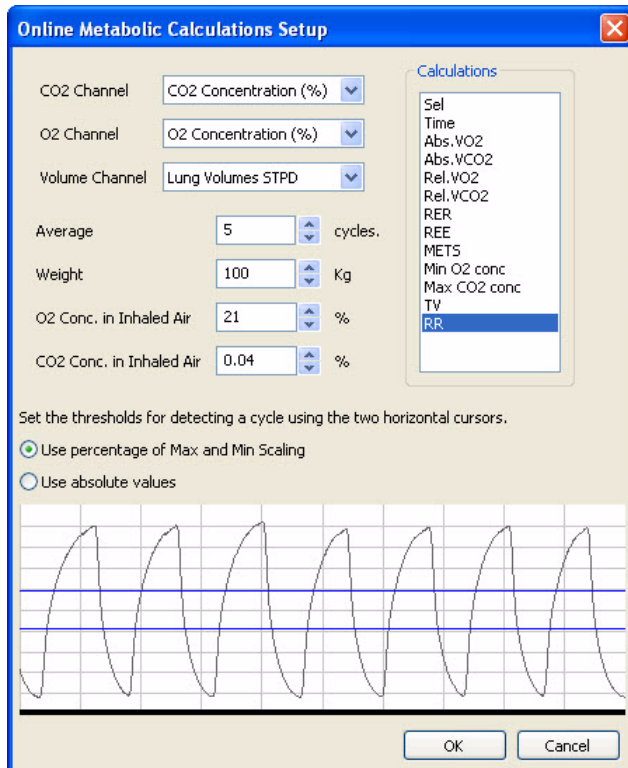
The Online Metabolic Calculations Setup.

- Select the O_2 and CO_2 channels and enter them in the dropdown menu boxes.

- Select a calibrated **Lung Volume** channel from the dropdown channel menu. For instructions on calibrating the spirometer in use, refer to the **Spirometry** function section beginning on page 68 of **Chapter 6: Computed Channels**.

- Next to **Average**, enter the number of breath cycles to be averaged for each reported value.

- Enter the subject's weight, and the estimated O_2 and CO_2 content of inhaled air in the appropriate text boxes.

- Under **Calculations**, select the **Metabolic** parameters to be included in the data boxes. Descriptions of each of these variables can be found at the end of this section.

- Set the **Thresholds** for cycle detection on the data graph. Be sure both threshold lines stay between the minimum and maximum data values. If the data trace is very stable, absolute values can be used instead.

- Click **OK** to exit the dialog.

- Once the subject and the hardware are prepared, click **Record** and the **Metabolic** parameters will be averaged over the pre-set number of cycles and displayed in the data boxes as data are recorded.

Using Breath by Breath: Offline Calculations

Once breath-by-breath data have been recorded, **Metabolic** parameters over designated durations can be displayed in a table and graphs can be created showing one parameter as a function of a second.

To use **Breath by Breath: Offline Calculations**:

• Open a **Metabolic** recording with CO_2, O_2, and calibrated air flow data.

• Open **Breath by Breath: Offline Calculations** from the **Metabolic** sub-menu, opening the **Metabolic Calculations Dialog**.
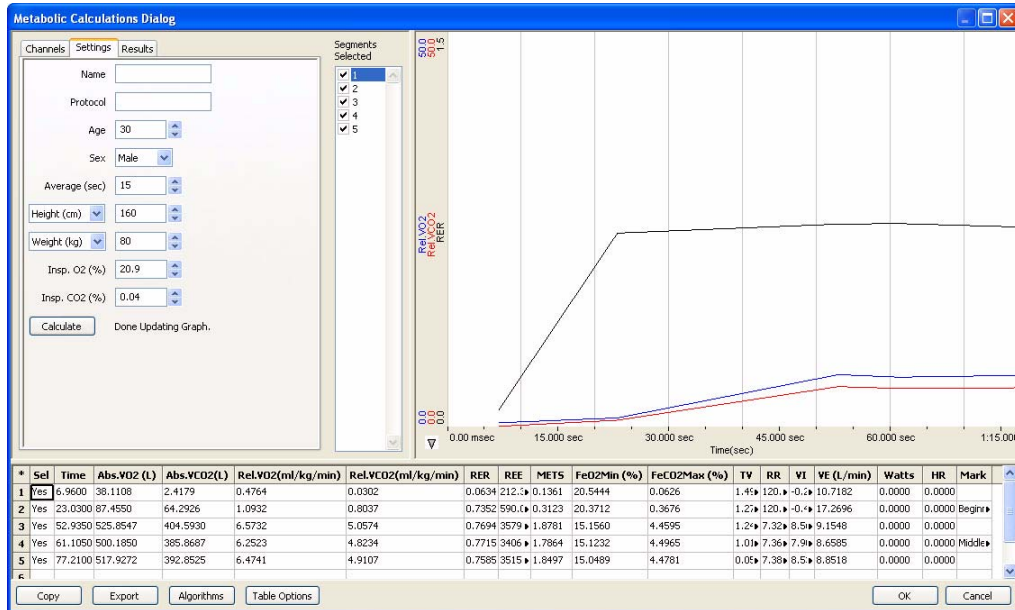
The Metabolic Calculations Dialog.

There are three tabbed pages used in setting up the dialog: **Channels**, **Settings**, and **Results**.

The Metabolic Channels dialog.

On the tabbed **Channels** panel:

• Select the **Channels** to be used. At a minimum, a CO_2, an O_2, and a calibrated **Volume Channel** must be selected. A channel showing heart rate and one showing the actual work done on an ergometer of some sort are required to calculate some parameters.

The Metabolic Settings dialog.

On the tabbed **Settings** panel:

• Enter in the appropriate text boxes the subject's name or code, age, height, weight, and any specific exercise protocol being used. Enter the duration of the segments to average for calculations, and the inspired O_2 and CO_2 levels.

• Clicking **Calculate** on the **Settings** tabbed page displays the chosen parameters in the **Data Table**. A basic **Metabolic** graph showing **Relative VO$_2$**, **Relative VCO$_2$**, and **RER** as a function of time is displayed in the graph area of the dialog.

The Metabolic Results panel.

On the tabbed **Results** panel:

- The **Settings** information will already have been entered.

- Additional information about the experiment can be added to this panel.
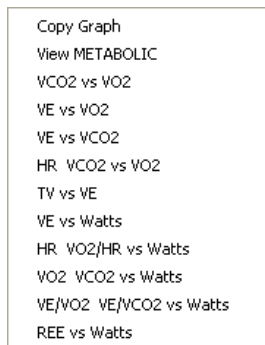
The Metabolic graph menu.

To create a graph of selected data:

- Click the down arrow at the lower left corner of the graph area of the dialog to open the graph menu.

- Choose the parameters to be plotted from the list. Descriptions of the variables can be found at the end of this section.

- Choose **Metabolic** to display a graph of basic metabolic parameters as a function of time.

- The recording is divided into segments based on the segment duration entered into the **Settings** dialog. They are listed in the **Selected Segments** window, where they can be selected or de-selected. Segments selected with a check will be displayed on the graph and the calculated parameters will be included in the **Data Table**.

- Any graph can be copied to the clipboard by selecting **Copy Graph**.

There are four buttons across the bottom of the **Metabolic Calculations Dialog**: **Copy**, **Export**, **Algorithms** and **Table Options**:

- All the calculated data in the **Data Table** can be copied to the clipboard by clicking the **Copy** button, or exported by clicking the **Export** button. The data are exported in a tab (*.txt) or comma separated (*.csv) text file, and the XY graph can be exported as a Portable Network Graphics (*.png) or JPEG (*.jpg) image.

- Clicking **Algorithms** opens an information window describing the mathematical equations used to compute a number of the offline parameters.

Metabolic offline functions list.
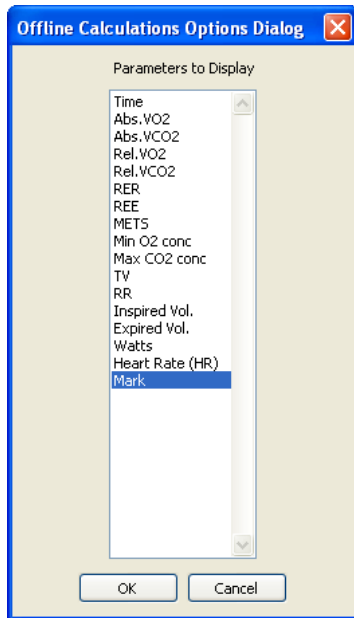
- *LabScribe2* is able to calculate a large number of metabolic calculations. By clicking **Table Options** at the bottom of the **Metabolic Calculations Dialog**, the **Offline Calculations Options Dialog** opens and the calculations to be displayed in the **Data Table** can be chosen from the list of all possible calculations. The **Data Table** displays the chosen calculated values for each of the segments checked in the **Selected Segments** window, as well as the mean, standard deviation, and range of each of the parameters averaged over all the selected segments.

- Clicking the asterisk at the upper left of the **Data Table** displays the **autosize** option. Clicking **autosize** will optimize the size of the **Data Table** boxes.

- Clicking **OK** in the **Offline Calculations Options Dialog** saves the settings for future **Metabolic** analyses.

The **Data Table** parameters are:

- **Abs. VO2**: Volume of oxygen consumed in liters per minute.

- **Abs. VCO2**: Volume of carbon dioxide produced in liters per minute.

- **Rel. VO2**: **Abs. VO2** per kg of body weight.

- **Rel. VCO2**: **Abs. VCO2** per kg of body weight.

- **RER**: Respiratory Exchange Ratio: **VCO2/VO2**.

- **REE (kcal/24 hr)**: Resting Energy Expenditure: 5.46 VO2 + 1.75 VCO2.

- **METS**: Metabolic Equivalents: One MET = 3.5 ml of VO2 per kg of body weight.

- **Min O2 conc**: Minimum percentage of O2 in expired air.

- **Max CO2 conc**: Maximum percentage of CO2 in expired air.

- **TV**: Tidal Volume.

- **RR**: Respiratory Rate.

- **Inspired Vol.**: Inspired air volume.

- **Expired Vol.**: Expired air volume.

- **Watts**: Energy expended.

- **HR** (Heart Rate): 60/period of each cycle.

- **Mark**: Displays any marks in the selected segment.

Using Mixing Chamber: Offline Calculations

Metabolic parameters can be calculated from the gas contents of the mixing chamber, averaged over designated durations indicated in the **Settings** page of the **Offline Calculations** dialog. A small animal chamber can be used instead.

To use **Mixing Chamber: Offline Calculations**:

- Follow the procedure for **Breath-by-Breath: Offline Calculations**.

AutoMark Peaks

The regular occurrence of peaks is a common feature of many types of data files. The **AutoMark Peaks Advanced** function can locate and analyze the peaks in a *LabScribe2* recording.
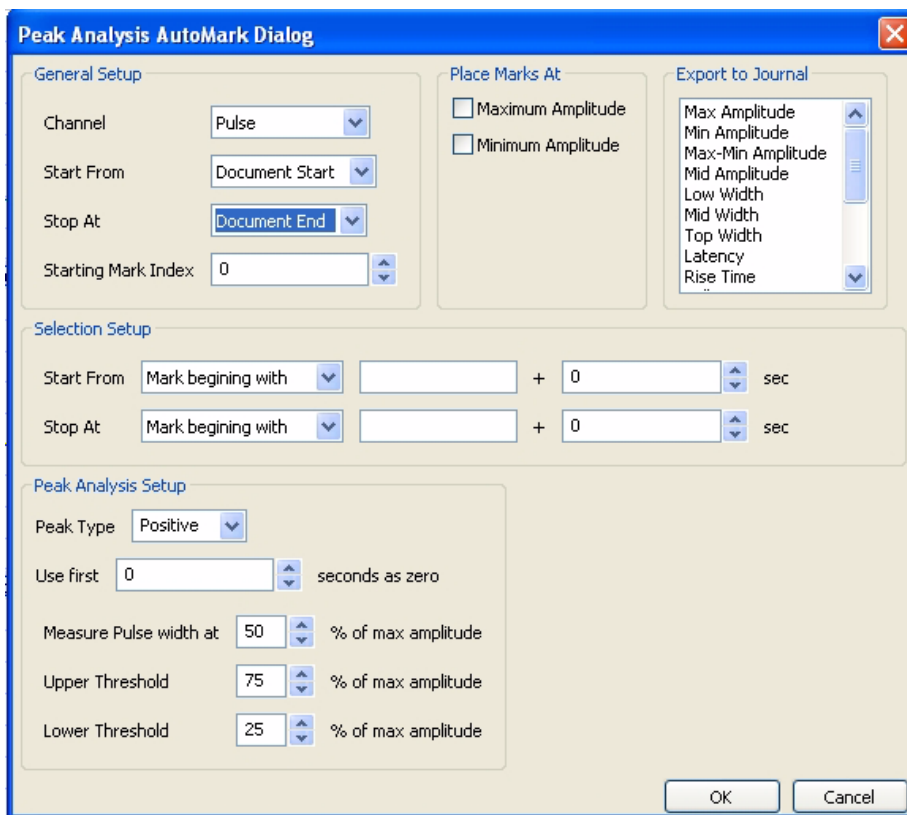
The Peak Analysis Automark Dialog.

Peak Analysis AutoMarks

The **Peak Analysis AutoMark Dialog** can be opened by selecting **AutoMark Peaks** from the **Advanced** menu. The options for configuring the dialog follow.

General Setup:

- **Channel**: Channel with peaks data.

- **Start From**: Location in the data file to start the analysis. The start of the document or an existing mark can be used to set the start location.

- **Stop At**: Location in the data file to end the analysis. The end of the document or an existing mark can be used to set the stop location.

- **Starting Mark Index**: Marks placed in the record have a number which corresponds to the cycle number. The default cycle number starts at zero, but the starting cycle number can be changed here.

- When the data channel is selected, it will appear in the graph. Set the threshold using the two horizontal cursors, such that the positive threshold crossing from below the lower threshold to above the upper threshold can be used to determine the cycle.

Place Marks At:

- Choose the points of interest the program should mark in the record. This can be used to verify that the program has detected the correct points of interest.

Export to Journal:

- Choose the calculated values you want exported to the journal for each peak.

Selection Setup:

- The start and the stop of the selection is set in this drop-down menu. To define a time location one can choose a mark identified by its starting text, the block start, or positive or negative crossing of the data. A time offset can be added to this location if desired.

Peak Analysis Setup:

- **Peak type**: The software will look for positive or negative peaks.

- **Use first "N" seconds as zero**: Used to set the offset, or the baseline.

- **Measure Pulse width at**: A percentage of the amplitude can be set as the location at which peak width is measured.

- **Upper and Lower Thresholds**: Upper and lower detection thresholds can be set in the edit boxes.

The calculated peak functions that can be exported to the **Journal**:

- **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 at Max Slope**: Time at maximum slope.

- **T at Min Slope**: Time at minimum slope.

Normalization

Normalization is a separately licensed module designed specifically to calibrate small vessel size for the use of wire myographs in vessel contractility experiments. In order to make comparisons between different vessels and experimental conditions, an internal circumference that exerts a **Target Pressure** needs to be determined to make comparisons meaningful. The relaxed vessel wall is stretched to a number of micrometer settings for a specific size myograph. The vessel wall force in response to each stretch is recorded by using the **Add Point** command. After a curve is generated from these forces, the **Calculate** command is used to determine the internal circumference at a **Target Pressure**, usually 13.3kPa. The normalized internal circumference is usually determined to be 90% of this value.
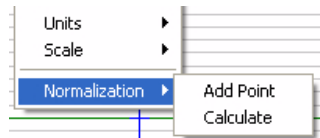
The **Normalization** module is accessed by right-clicking in the appropriate wire myograph channel display area. **Normalization** will be at the bottom of the menu.

The Normalization submenu.

To use the **Normalization** module:
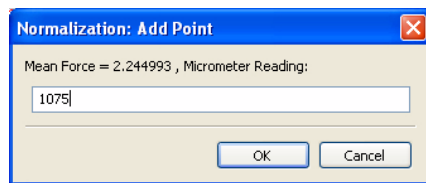
The Normalization: Add Point dialog.

- Calibrate the myograph(s) to be used in the experiment, according to procedures outlined in the myograph documentation.

- With the wires positioned and exerting no stretch, click **Record**.

- If the wires have been properly calibrated, the pressure should be zero. Click **Stop**.

- In **Two Cursor Mode**, position the cursors on either side of a representative section of the trace and select **Add Point** from the **Normalization** menu. The mean value between the cursors will be recorded in the **Normalization Dialog**.

- Increase the micrometer setting to stretch the vessel wall minimally. Click **Record**.

- After the pressure has stabilized, click **Stop**.

- Position the cursors on either side of a stabilized section of the trace and select **Add Point** from the **Normalization** menu.

- Repeat this procedure for a number of micrometer settings. Once enough points have been recorded, select **Calculate** from the **Normalization** menu, opening the **Normalization Dialog**. The micrometer setting, the force registered by the wires, and the vessel wall pressure are displayed for each data point recorded.

- In the **Normalization Dialog**, enter the **Target Pressure**, the **Wire Diameter**, the **IC Ratio**, and the **Vessel Length**.

- Click **Fit Exponential Equation to Data**. A graph is generated and the **Internal Circumference** at the **Target Pressure**, the normalized **IC**, and the correct micrometer setting for that normalized internal circumference are displayed.

• Outlying data points can be deleted by selecting the point and clicking **Delete Point.**
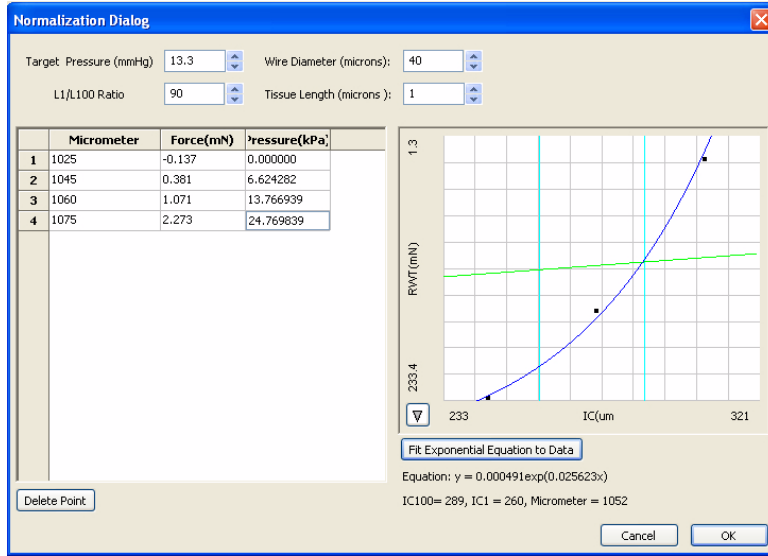
The Normalization Dialog.

Chapter 9: Digital Input and Output

Overview

Some *iWorx* data acquisition units are equipped with **Digital Input** and **Output** connectors. A dedicated **Trigger** input allows external devices with a TTL output to trigger *LabScribe2* recording when the device turns on or off. Other **Digital Input** connectors allow external TTL devices to be monitored for changes and have the changes indicated on the *LabScribe2* recording. *iWorx* units with **Digital Output** connectors can send information to external devices with a TTL input, instructing the devices to turn on or off, based on programmed criteria. This information takes the form of an **Output Sequence** which is programmed to include the instructions as part of a *LabScribe2* operational sequence.

Digital Input
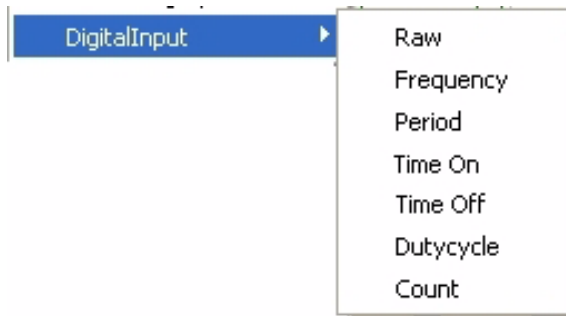
The Digital Input menu.

The **Digital Input** function may have to be added to the list of computed functions available through the **add function** command by adding it to the active function list on the **Options** page of the **Preferences Dialog**. Once added to the list, it can be accessed on the **Channel** page of the **Preferences Dialog**, or through the **add function** command in the **Channel Bar** of each channel in the **Main Window**.

For instructions on using the **add function** command in either of these locations, refer to page 62 in **Chapter 6: Computed Functions**. Unlike other computed functions, **Digital Input** is not usually associated with a certain raw data channel. The **Digital Input** channel can be added directly from the **Channel** page of the **Preferences Dialog**, by activating a computed channel and selecting **Digital Input** from the **add function** list.

Selecting any of the **Digital Input** sub-menu functions will open the **Digital Inputs** dialog. The hardware channels that are to be monitored for TTL changes can be specifed in this dialog. The **Digital Input** function treats the data received on a selected hardware channel as a binary number, on or off, which indicates a change in the state of external devices connected through a TTL Output. These changes are indicated on the added **Computed Channel**.

The changes that can be monitored are:

- **Raw**: The digital value at any given time, as a 32 bit word, is displayed (16 lines, each either on or off).

- **Frequency**: The program takes the **period** of the **Digital Input** data 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 receive high TTL signals from the external device.

- **Time Off**: The time that the selected channels receive low TTL signals from the external device.

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

- **Count**: Cumulative number of events since the last reset.

The options on the **Digital Inputs** setup dialog are:
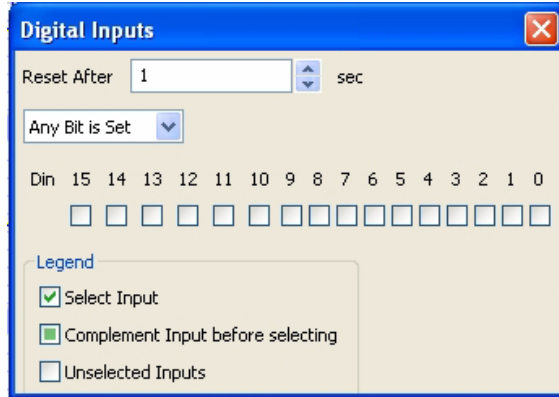
The Digital Inputs dialog.

• **Channel checkboxes**: The channels of interest can be selected using the checkboxes in the **Digital Inputs** dialog. If you desire to complement the input (change the input data from a 1 to a zero or a zero to a 1) before it is selected then click on the checkbox again, which will place a solid square in the checkbox, as shown in the legend.

• **Reset After N sec**: This option is only available for the **Count** function, which is used to reset the event **Count** after a user-determined number of seconds.

• **Any Bit is Set/ All Bits are set**: If **Any Bit is Set**, an event is detected if any channel receives a high TTL signal. If **All Bits are Set**, an event will be detected only if all channels receive a high TTL signal.

Digital Output

iWorx data acquisition units with digital output capabilities are able to control external TTL devices like pumps, heaters, cameras, and relays. Each external device receives a digital signal that instructs the device to change its status from one condition to another. For example, a pump may be turned on or off. The internal stimulators in *iWorx* data acquisition units can also be controlled with digital instructions. Refer to the **Stimulator** chapter for instructions and an example of building an output sequence that controls the stimulator operation.

Output States

Output states are individual instructions used in the construction of **output sequences**. There are six categories of **output states** that can be combined and built into a sequence that controls an external TTL device, and also controls related *LabScribe2* functions. The **output state** options can be found on the **Sequences** page of the **Preferences Dialog**.
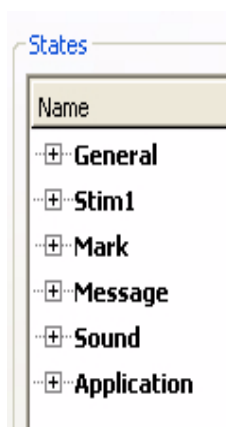
Output states categories.

• **General**: The general category includes instructions for *LabScribe2* to **Start Recording**, **Stop Recording** and **Update**.

• **Stim#**: Controls the internal stimulator operation. This section can be used to change the stimulator parameters on the fly.

• **Marks**: Puts **Marks** in the record indicating when events in the sequence occur.

• **Message**: *LabScribe2* displays user **Messages** on the screen when instructed.

• **Sound**: Plays an existing audio file, located in the same folder as the current settings file, when instructed.

• **Application**: Sends instructions to external devices.

Only some *iWorx* data acquisition units have **Digital Output** connectors. Each digital output can be controlled independently. For each digital output there are two states, **On** and **Off**. For eight digital outputs there are 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 text box that appears. To place a **Mark** 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, **Application 3** has been labeled as **Turn Pump On** and has been set to place a **Mark** in the record when it is fired.
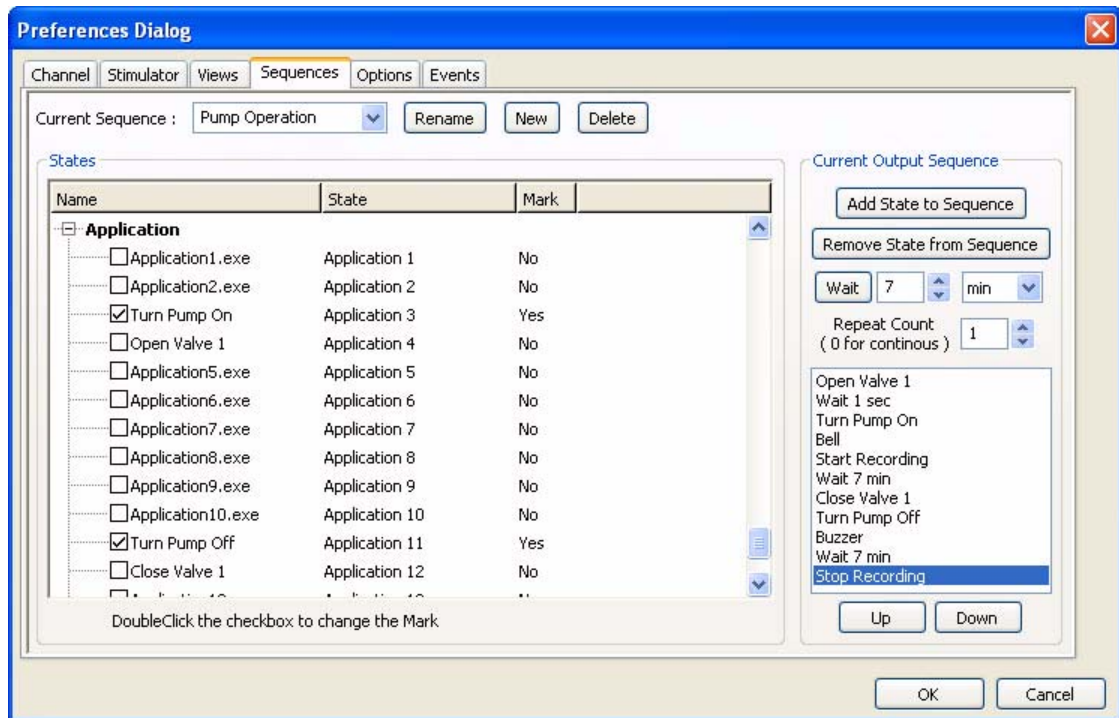
Sequences Page of Preferences Dialog.

Sequences Digital states can be combined to form **Sequences**. A **Sequence** allows the timed execution of digital states.

Creating a Sequence To create and edit a **Sequence:**

- Type the name of the sequence in the text box next to **Current Sequence** and click **New**.

- 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 *iWorx* device wait for a specified period of time between different digital states, type a time in the edit boxes next to the **Wait** button and click **Wait** to insert the waiting period into the sequence**.**

- To activate more than one state at the same time, add all the states you want to activate simultaneously to the sequence before configuring **Wait** or adding **Update** (in the **General** category). *LabScribe2* will execute them simultaneously when it reaches the **Wait** statement, the **Update** statement, or the end of the sequence. To repeat the sequence, set the **Repeat Count** to the desired number of times, or set it to zero to cause the sequence to repeat until manually stopped.

- The order of the states in the sequence can be changed by using the **Up** and **Down** buttons.

In the sequence illustrated on the previous page, **Valve 1** will open first. After one second, the **Pump** will start, a **Bell** audio file will play, and recording will commence. After seven minutes of recording, **Valve 1** will close, the **Pump** will turn off, and a **Buzzer** sound file will play. After another seven minutes, recording will stop and the sequence ends.

Triggering a Sequence
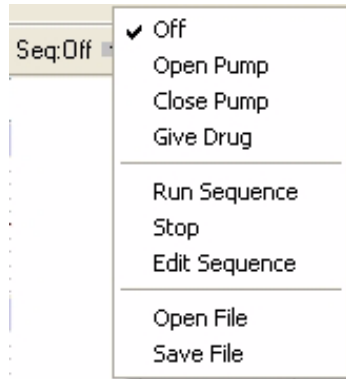
The Main Window Sequences menu.

Sequences of analog or digital outputs defined in the **Sequence** page of the **Preferences** dialog window are added to the **Sequences** menu on the Toolbar. Clicking the **Run Sequence** option will fire the output sequence selected from the list.

Sequences can also be triggered automatically by *LabScribe2's* detection of timed **Events** or **Events** in the recording. These **Events** are defined and given priorities in the **Events** page of the **Preferences Dialog**. Refer to page 44 in the **Events** section of the **Creating Your Own Preferences and Settings** chapter for a detailed discussion of this process.

Chapter 10: The Journal and Data Export

The Journal

The **Journal** is very similar to a text editor. Text can be entered and edited in the **Journal** area and images of the traces and graphs from the raw data and computed channels can be copied to the **Journal**. In addition to images, data values from the **Main** and **Analysis Windows** can be added to the **Journal**. The **Journal** can be printed, and its contents can be copied and pasted into other applications.
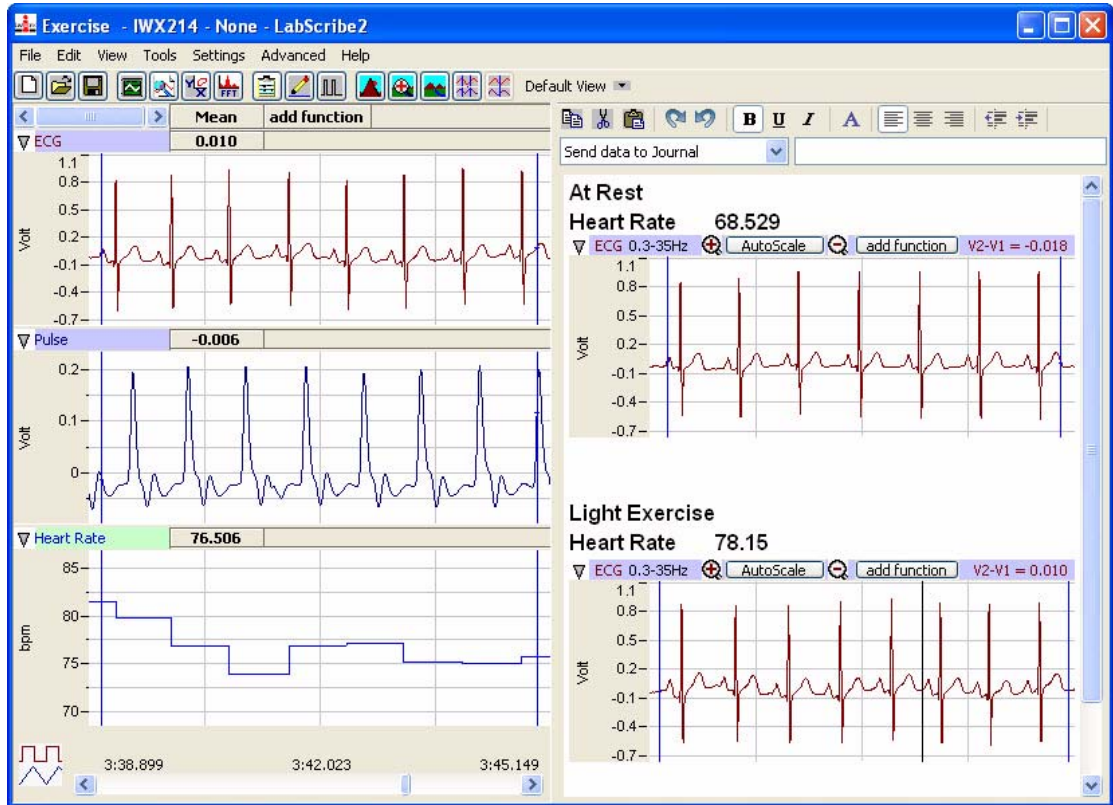
Analysis Window with Journal including both text and graphics.

The **Journal** is not visible in *LabScribe2*'s default **View**. Clicking on the **Journal** icon in the **Toolbar,** or selecting **Journal** in the **View** menu, opens the **Journa**l on the right side of the **Main**, **Analysis**, **XY Tools**, and **FFT Windows**. The **Journal** toolbar runs across the top of the **Journal** area of the screen.

The Journal Toolbar
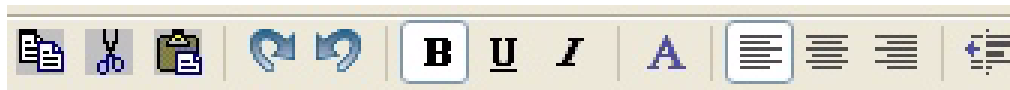
The Journal toolbar.

The **Journal** toolbar functions are:

Copy: Copies selected text and images from the **Journal** to the clipboard.

Cut: Cuts selected text and images from the **Journal** and copies it to the clipboard.

Paste: Pastes the clipboard contents into the **Journal**.

Redo: After an **Undo**, this command restores the **Journal** to its pre-**Undo** state.

Undo: Undoes the most recent **Journal** operation.

Bold: Makes the selected **Journal** text bold.

Underline: Underlines the selected **Journal** text.

Italic: Italicizes the selected **Journal** text.

Font: Opens a dialog in which the user can choose a font, font style, font size, and font color, and apply them to the selected text.

Align Left: Aligns selected text or graphics to the left border of the **Journal**.

Align Center: Centers selected text or graphics on the **Journal** page.

Align Right: Aligns selected text or graphics to the right border of the **Journal**.

Indents Less: Moves selected indented text or graphics to the left.

Indents More: Indents selected text or graphics.

Adding Text, Images, and Data to the Journal

Beneath the **Journal** toolbar is a drop-down menu that allows the user to instruct *LabScribe2* where to send data from the **Main** and **Analysis Windows**.
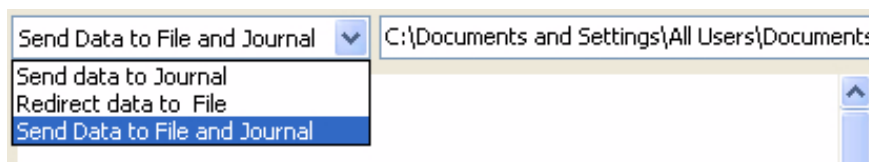
Journal submenu.

Choosing **Send Data to Journal** instructs *LabScribe2* to send data values from the **Main** or **Analysis Window**s to the **Journal**. Once the data file has been saved, choosing **Redirect data to File** opens a dialog from which the user can instruct *LabScribe2* to send data values from the **Main** or **Analysis Windows** to a tab (*.txt) or comma (*.csv) separated text file chosen and named by the user. Choosing **Send Data to File and Journal** opens the same dialog and sends data to both the **Journal** and the chosen file. The specific data values that are sent depends on whether the data are being sent from the **Main** or **Analysis Window**.

From the **Main Window**:

- Choosing **Add All Data to Journal** from the **Tools** menu sends the titles of each channel and each channel's data values to the specified location(s). In **Single Cursor** mode, the ampitude values at the **Cursor** from all channels are sent. In **Two Cursor Mode**, the difference in amplitude values between the two cursors (**Cursor 2** - **Cursor 1**) from all channels are sent.

- Choosing **Add Title to Journal** from the **Tools** menu sends the titles of all channels to the specified location(s).

From the **Analysis Window**:

- Choosing **Add All Data to Journal** from the **Tools** menu or from any of the **Channel Menus** (accessed by clicking on the arrow on the left side of any of the **Channel Bars** or right-clicking in any of the channels) sends the channel titles and the values from the **Table Functions** data boxes in the **Channel Bars** of all channels to the specified location(s).

- Choosing **Add Title to Journal** from the **Tools** menu or from any of the **Channel Menus** sends the **Title(s)** from the **Table Functions** data box title bar to the specifed location(s).

- Choosing **Add Ch. Data to Journal** from a channel menu sends that channel's title and **Table Functions** data box values to the specified location(s).

- These three **Journal** commands can also be displayed by clicking on any of the **Table Functions** data boxes in a **Channel Bar**.

Images of the raw data and computed channels can also be added to the **Journal**:

- Choosing **Add Image to the Journal** from the **Tools** menu sends an image of all the raw data and computed channels on the visible screen to the **Journal** from either the **Main** or **Analysis Window**.

- Choosing **Copy Graph** from any of the **Channel Menus** will send an image of that channel to the clipboard. The image can then be pasted into the **Journal** or an external application.

Printing and Saving the Journal

The **Journal** can be printed by using the **Print Journal** command in the **File** menu.

The **Journal** can be saved as a web page (in *.html format) or as an XML file (in *.xml format) by selecting **Save as...** in the **File** menu and choosing a format from the **Save as type:** drop-down menu in the **Save as...** dialog.

By moving images and calculated values to the **Journal** and adding typed comments from the keyboard, an entire lab report can be created without ever leaving *LabScribe2*.

Cutting, Copying and Pasting

LabScribe2 supports cutting and copying:

- Choosing **Copy** in the **Edit** menu will copy an image of all raw data and computed channels to the clipboard. This image can then be pasted into the **Journal** (by using the **Paste** icon in the **Journal's** toolbar) or into an external application.

- In *LabScribe2*, pasting is supported only in the **Journal**. To paste the copied image into the **Journal**, first **Copy** data from either the **Main** or **Analysis** windows. Open the **Journal** and click the **Paste** icon in the **Journal's** toolbar. Text and images from other sources can also be pasted into the **Journal**.

Saving and Saving As

Once recorded, *LabScribe2* can save data in its own binary format using the **Save** or **Save As** command found in the **File** menu. Selecting **Save As...** will open a dialog allowing the user to create a copy of the file with a new name. *LabScribe2* can save several types of documents:

- An *iWorx* data file (*.iwxdata format).

- An *iWorx* **Settings File** (*.iwxset format).

- A digital or analog **Output Sequence** (*.iwxseq).

- The **Journal** as a web page (*.html format) or XML file (*.xml format).
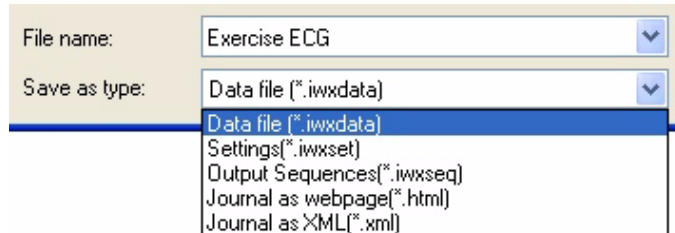
Save as... file type options in the Save as... dialog.

Printing

The **Print View** command in the **File** menu will print the raw and computed data channels exactly as they appear on the current screen. The **Print Preview** command previews the image to be printed.

The **Print Journal** command will preview the **Journal** pages as they will appear in print. A **Print...** command will appear over this preview which opens a dialog from which the user can choose the pages of the **Journal** to be printed.

It is important that you display the *LabScribe2* windows exactly as you want them to appear on the printed page before issuing a **Print** command.

Exporting Data

Data can be exported from *LabScribe2* as either data values 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.
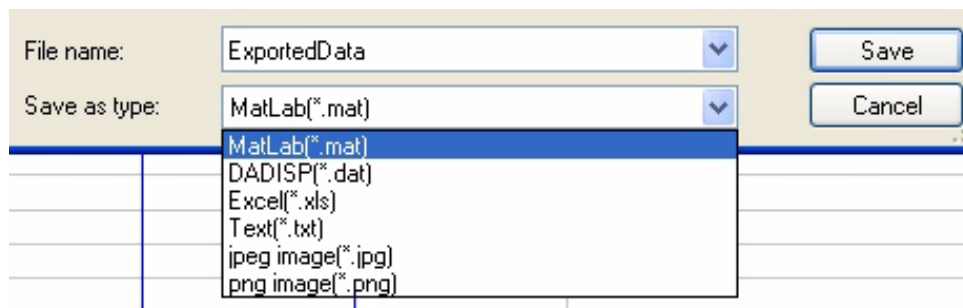
Export file format options.

Exporting Images

The active display can be exported as a JPEG (*.jpeg format) or a Portable Network Graphics (*.png) image. Portable Network Graphics is a high resolution format for images. Only the raw data and computed channels will be included in the image. If the **Journal** is open, it will not be part of the image.

Exporting Numerical Data

The numerical data points that make up the recorded data can be exported in Matlab (*.mat format), DADiSP (*.dat format), Text (*.txt format) or Excel (*.xls format).

If data are exported from the **Main Window**, the complete data file is exported. If data are exported from the **Analysis Window**, then only the data displayed in the visible window are exported.

Choosing any of these **Export** formats will open an **Export Parameters** dialog in which the user can choose whether to export every data point, a sampling of data points, the averages of a chosen number of data points, or the maximum or minimum values of a chosen number of data points.

The Export Parameters dialog.

Index