



WORLD
PRECISION
INSTRUMENTS

INSTRUCTION MANUAL

Lab-Trax-816-MDAC

LabTrax 8/16 with MDAC

Serial No. _____

www.wpiinc.com

041819

CONTENTS

ABOUT THIS MANUAL	1
INTRODUCTION	1
Parts List.....	1
Unpacking.....	1
SOFTWARE INSTALLATION	2
Installing MDAC/DAQ Software.....	2
Setting of Latency timer of COM Ports.....	6
INSTRUMENT SETUP.....	7
Setting Up the LabTrax 8/16 Data Acquisition Unit	7
Setting Sampling and Recording Preferences	11
Opening Recorded Data File	13
Deleting Recorded Data Files.....	14
Setting Channel Display Preferences.....	15
Plot Legend.....	16
Setting the Number of Channels	16
Resetting the Factory Defaults.....	16
Working with Annotations.....	16
Composing/Editing/Saving Motor Position Protocols (OUT 1-3).....	17
Editing Channel Math.....	21
Calibrating a Transducer	22
Unit Conversion.....	24
Setting Record Time	25
Editing Threshold Trigger	25
Online Filtering	26
Toolboxes.....	27
OPERATING INSTRUCTIONS.....	27
Data Recording.....	27
Displaying/Printing Data Files.....	31
Main Window Context Menu.....	32
Analyzing Data	33
SPECIFICATIONS.....	48
INDEX.....	50
WARRANTY	51
Claims and Returns	51
Repairs.....	51

Copyright © 2019 by World Precision Instruments, Inc. All rights reserved. No part of this publication may be reproduced or translated into any language, in any form, without prior written permission of World Precision Instruments, Inc.

Microsoft and Excel are registered trademarks of the Microsoft Corporation.

ABOUT THIS MANUAL

The following symbols are used in this guide:



This symbol indicates a CAUTION. Cautions warn against actions that can cause damage to equipment. Please read these carefully.



This symbol indicates a WARNING. Warnings alert you to actions that can cause personal injury or pose a physical threat. Please read these carefully.

NOTES and TIPS contain helpful information.



Fig. 1—The LabTrax 8/16 is an 8-channel data acquisition system.

INTRODUCTION

The **LabTrax 8/16** is a 16-bit analog to digital converter that can be programmed to activate as many as eight analog inputs, four analog outputs, 16 digital inputs and 16 digital outputs.

Parts List

After unpacking, verify that there is no visible damage to the instrument. Verify that all items are included:

- (1) **LabTrax 8/16**
- (1) MDAC Software CD
- (1) Power Supply cable
- (1) USB cable
- (1) Instruction Manual

Unpacking

Upon receipt of this instrument, make a thorough inspection of the contents and check for possible damage. Missing cartons or obvious damage to cartons should be noted on the delivery receipt before signing. Concealed damage should be reported at once to the carrier and an inspection requested. Please read the section entitled

“Claims and Returns” on page 51 of this manual. Please contact WPI Customer Service if any parts are missing at 941.371.1003 or customerservice@wpiinc.com.

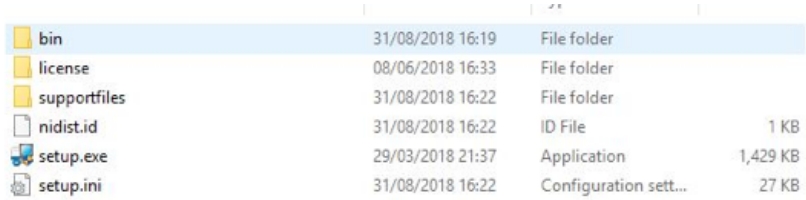
Returns: Do not return any goods to WPI without obtaining prior approval (RMA # required) and instructions from WPI’s Returns Department. Goods returned (unauthorized) by collect freight may be refused. If a return shipment is necessary, use the original container, if possible. If the original container is not available, use a suitable substitute that is rigid and of adequate size. Wrap the instrument in paper or plastic surrounded with at least 100mm (four inches) of shock absorbing material. For further details, please read the section entitled “Claims and Returns” on page 51 of this manual.

SOFTWARE INSTALLATION

Installing MDAC/DAQ Software

This installation installs both the **MDAC** software and the necessary National Instruments interface components.

1. Insert the MDAC installation CD into the computer CD drive and copy the CD content to your hard disk drive (C:\).
2. If the installation does not start automatically, use **Windows Explorer** to view the files on the CD. To open **Windows Explorer**, right click on the Windows **Start** button and choose **Explore**. Navigate to the CD drive.



Name	Modified	Type	Size
bin	31/08/2018 16:19	File folder	
license	08/06/2018 16:33	File folder	
supportfiles	31/08/2018 16:22	File folder	
nidist.id	31/08/2018 16:22	ID File	1 KB
setup.exe	29/03/2018 21:37	Application	1,429 KB
setup.ini	31/08/2018 16:22	Configuration sett...	27 KB

Fig. 2—Double click on the *Setup.exe* file to install the MDAC software.

3. Double-click on the **setup.exe** file to start the installation of the **MDAC** program. It is in the root directory of the CD. The Destination Directory window appears.

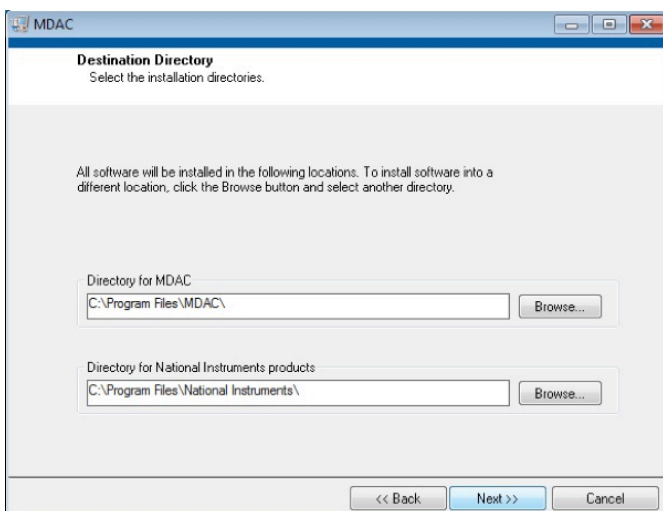


Fig. 3—The first install window lets you choose the installation directories.

- By default the software packages install in the *C:\Program Files* directory.

TIP: To change one of the installation directories, click on the **Browse...** button and choose another location.

Click the **Next >>** button. The license agreement displays.

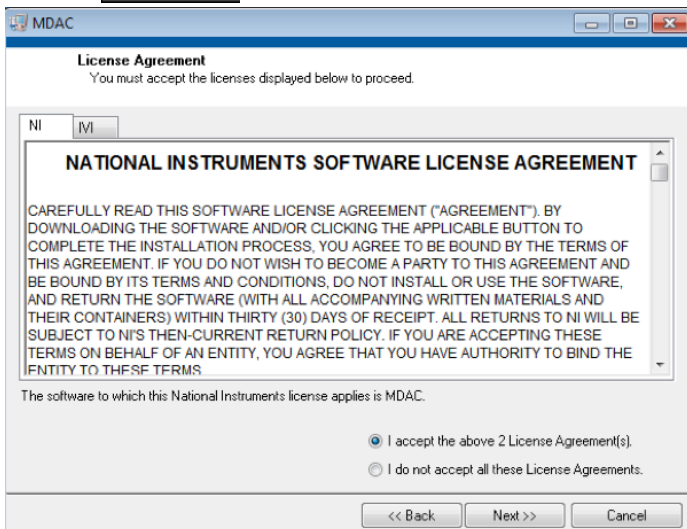


Fig. 4—The National Instruments license agreement is on the NI tab, and the MDAC license agreement is on the M tab.

5. After reading the license agreements for the National Instruments and **MDAC** software, click the **Next >>** button. The installation summary page displays.

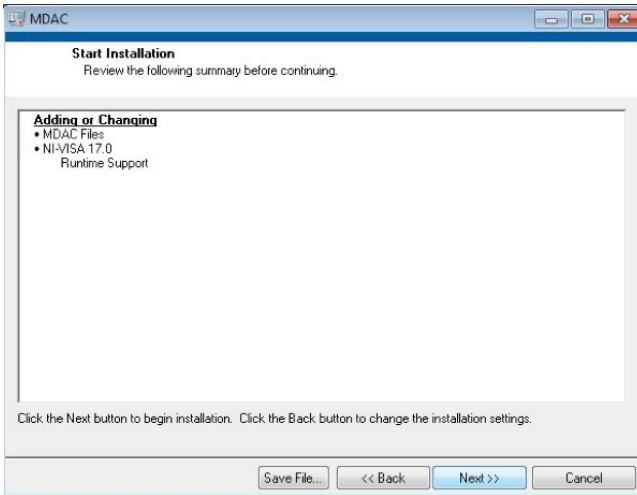


Fig. 5—The installation summary window shows the files that will be installed.

6. Review the summary. Click the **Next >>** button. The status indicator appears to show the progress of the installation.

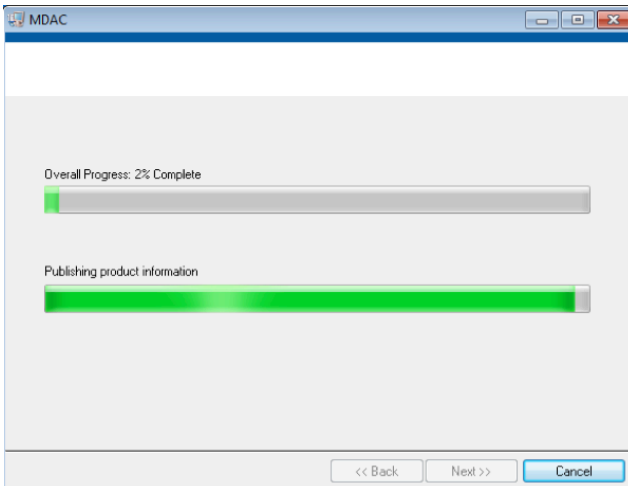


Fig. 6—The installation window shows the status of the installation.

- When the installation is complete, the Installation Complete message displays. Click the **Finish** button. A message box appears indicating that you must restart the computer to complete the installation.

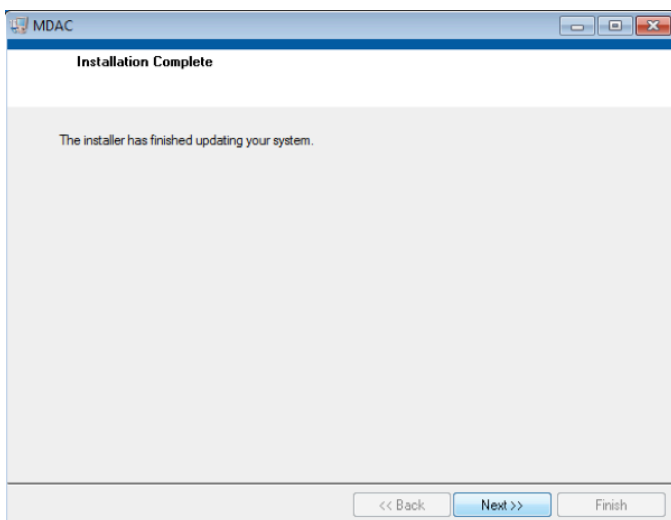


Fig. 7—When the installation is complete, click Finish.

- Before running the software, you must restart the computer. Close all open programs and then click the **Restart** button.

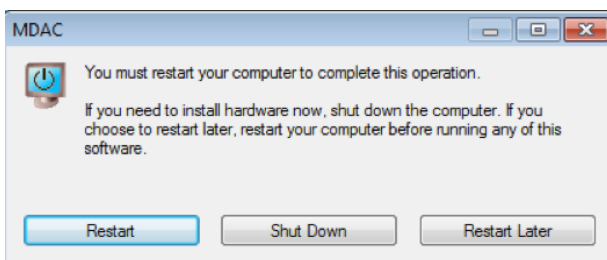


Fig. 8—Close all open programs and click the Restart button.

TIP: To locate the driver files if they are needed during the installation process, look in the directory labeled FTDI.

- To install driver for LabTrax-8/16, navigate to C:\Program Files\MDAC\FTDI_2018 and double click on FTDI_2018.exe to launch the FTDI driver installation. Follow the instructions for the driver installation.

- To start the MDAC software, click on the Windows **Start** button, choose **Programs>MDAC>MDAC**.

Setting of Latency timer of COM Ports

The COM ports need to be set from 16 ms to 1 ms in the Device Manager of the USB serial ports.

1. Open the Device Manager in the Windows Control Panel.

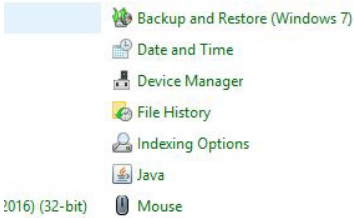


Fig. 9—Windows Control Panel

2. Go to Ports (COM & LPTI) and open the drop-down menu. Right click to open the properties of the first USB serial port (e.g. COM3).

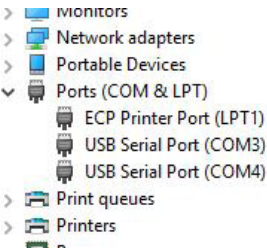


Fig. 10—Open the Ports submenu.

3. Click on Change Settings and then select Tab Port Settings.

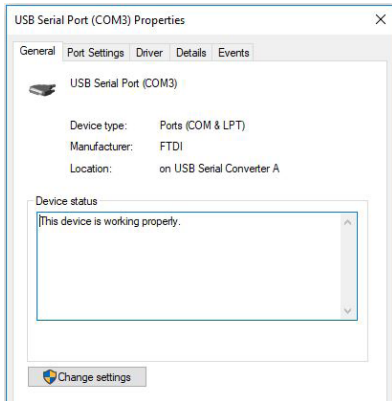


Fig. 11—Choose Change Settings from the popup menu.

- Click Advanced. The port settings pop up opens.

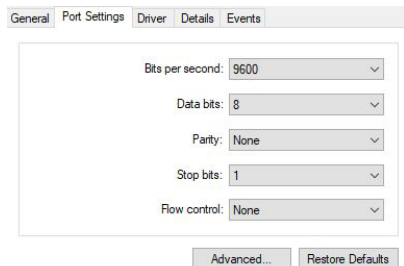


Fig. 12—Advanced settings for the COM port.

- Search the Latency Timer and scroll from the default 16 ms to 1 ms.

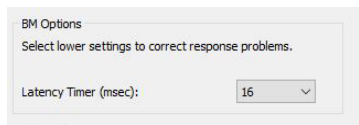


Fig. 13—Change the latency timer.

- Apply the changes and close the window. You may be prompted to restart your computer.
- Repeat the procedure for the second USB Serial Port (e.g. COM4).

INSTRUMENT SETUP

Setting Up the LabTrax 8/16 Data Acquisition Unit



Fig. 14—The front of the LabTrax 8/16 has eight analog inputs.

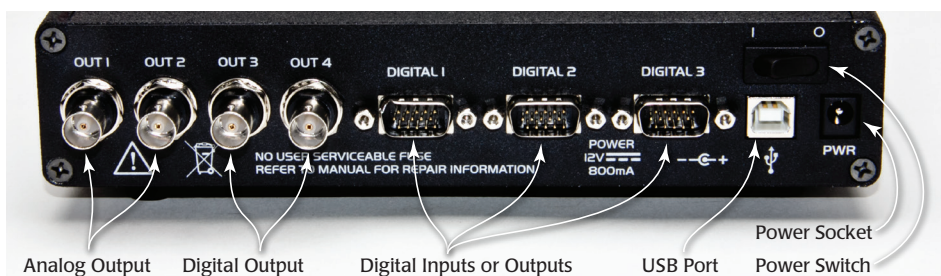


Fig. 15—The back panel of the LabTrax 8/16 has four analog outputs, digital inputs or outputs, a USB port, power socket and power switch.

1. Using a standard USB cable, connect the USB port on the back of the **LabTrax 8/16** data acquisition unit to one of the USB ports on the computer.
2. Plug the connector on the end of the **LabTrax 8/16** power supply cable into the power (**PWR**) socket on the back of the **LabTrax 8/16**.
3. Plug the power supply into an AC wall outlet or a power strip with a surge suppressor.
4. When using WPI's SI-H line muscle test devices do the following connection. Else setup your own connection.
5. Connect the **Force Output** of the **SI-BAM21-LCB** transducer amplifier, the **LPF Out** of the **SI-PF100** programmable filter module (if one is used) or the **Corrected Output** of the **SI-AOSUB** anti-oscillation module (if one is used) to the **IN 1** analog input using a BNC-BNC cable.
6. If electrical stimulation of the muscle tissue is required, connect a stimulating electrode assembly to the **OUT 2** analog output on the back of the **LabTrax 8/16**. **OUT 2** functions as a stimulator when it activated by a stimulation protocol.
7. If the muscle tissue will be stretched using a motor, two additional analog inputs on the front and another analog output on the back of the **LabTrax 8/16** need to be connected to the motor control module in the amplifier system:
 - A. Place a BNC T-adapter on the **Position In** connector of the **SI-MOTDB** Linear Motor Controller that is used with **SI-MTM** and **SI-MKBM** muscle tester systems, or the **Position In** connector on the **SI-CISB** Position Control module that is used with **SI-CTS** Cell Tester systems.
 - B. Connect a BNC-BNC cable between the **OUT 1** analog output of the **LabTrax 8/16** and one of open connectors on the BNC T-adapter attached to the **Position In** connector.
 - C. Connect another BNC-BNC cable between the **IN 2** analog input and the remaining open connector on the BNC T-adapter on the **Position In** connector. With this connection, the actual signal sent to the motor is recorded.
 - D. Connect a third BNC-BNC cable between the **IN 3** analog input and the **Position Out** connector on either the **SI-MOTDB** Linear Motor Controller or the

SI-CISB Position Control module. With this connection, the actual position of the motor is recorded. This signal provides important information on the responsiveness of the motor.

8. Turn on the **Power** switch on the back panel (I) of the **LabTrax 8/16** unit.
9. Open the **MDAC** software. When the main window opens (Fig. 16), four channels that are capable of displaying the recorded data appear.

CAUTION: MDAC is set on launch with a Zero volt signal to protect your electronic device. Therefore, first switch on LabTrax, and then launch MDAC. Now you can switch on your electronic device, with a Zero volt setting.



Fig. 16—The MDAC software main window.

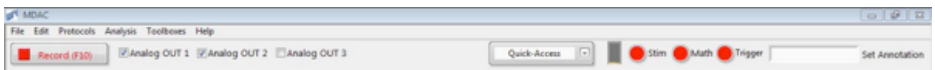


Fig. 17—The MDAC Main Window toolbar has new real-time control functions.

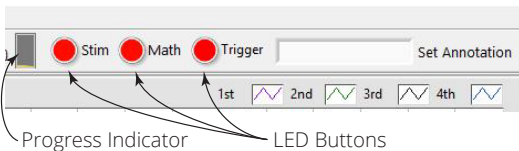


Fig. 18—The MDAC toolbar has new live Math and Trigger functions.

- LED button color display (green/red) to toggle the live **Math** or **Trigger** function on or off.

- The Progress Indicator shows the progress of repeat protocols.

If a communication error appears:

1. Select **Preferences** from the **Edit** pull down menu (right) on the **MDAC** main window.
2. In the **Preferences** window (Fig. 19), set the communications (VISA COM) port to one of the available ports (COMx) on the pull-down window. **LabTrax 8/16** will present two consecutive numbers (for example, COM3 and COM4). Always select the lowest number COM port (for example, COM3).

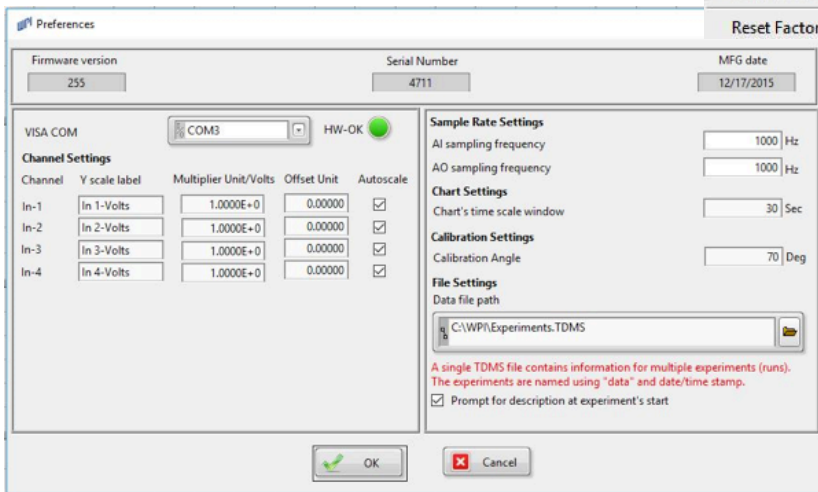
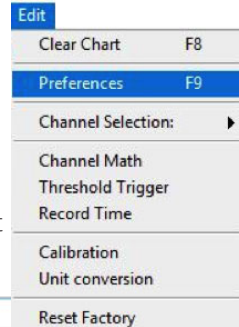


Fig. 19—The Preferences window has settings for using MDAC.

Setting Sampling and Recording Preferences

The sampling and recording parameters of the main window have been set to default values that will display data traces that are functional. These parameters can be customized to fit the requirements of the experimental procedure or your preferences.

The **Preferences** window contains display settings, variables and functions, which are used in MDAC:

- Firmware version, serial number, MFG date updated and VISA COM port (HW-OK LED) connection between MDAC and LabTrax 8/16 are now set or observed in the **Preferences** window. Visual display of the connected Hardware is given by the VISA COM port name and the green HW-OK LED. (If the connection failed or there is no connection, the VISA COM port name is disabled and grayed out, and HW-OK LED is red.) (See Fig. 20.)

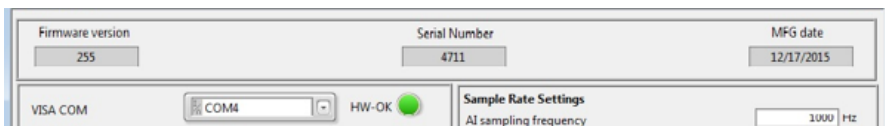


Fig. 20—This section of the Preferences window displays the VISA COM port connection state. Selecting the appropriate, available VISA COM port updates the state variables.

- You can change the Calibration Angle and saved it in the MDAC Settings file. The calibration angle is used during the calibration process to consider the force transducer angle of inclination (or any other sensor/transducer), when calibration is done on a plane surface (i.e. no inclined transducer support is available). Calibration angle compensation uses sine for calculation.

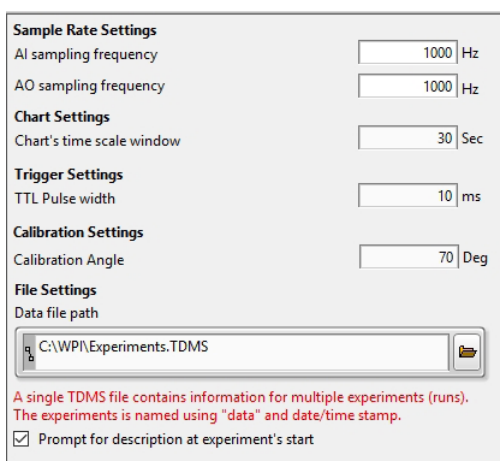
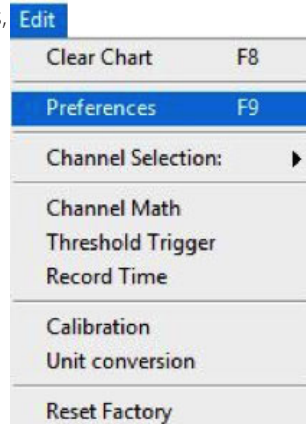


Fig. 21—The initial calibration angle of 70° was used in the Calibration menu.

NOTE: All calibration variables (Y scale label, Multiplier, Offset) are retrieved automatically from the **Calibration** menu and displayed in the **Preferences** menu or the chart axes. Multiplier values are given in Volts/Unit..

1. To customize the sampling and recording parameters, the **Edit** pull down menu on the main window (right). The **Preferences** window opens (Fig. 19).



2. In the **Preferences** window set these parameters:

For Analog Inputs:

- **Y scale label**—In the box next to the number of the channel, enter a word that reflects the function being recorded on the channel. For example, if contractile force is being recorded on the 1st channel (In1), type **Force** in the **Y scale label** box next to 1st channel. The default settings are In1 – In4, for channels 1 to 4.
- **Multiplier**—Enter the multiplier factor into the text box. This is needed to convert the recorded voltage into the units of the measured parameter. For example, if the calibration of a transducer determines that a 10g mass creates a 1V signal, place the value 10 (1.000E+1) in the multiplier box to change the y-values of amplitudes of the data points by 10X. The Y scale label should be altered to include the units, like **Force (in g)** in this example. The default setting is 1.000E+0 or 1.
- **Offset**—Enter a factor that will be added to or subtracted from the voltages recorded on each channel to reposition the trace up or down the Y-axis. The default setting is 0 (zero).
- **Autoscale**—Place a check in the box for each channel to adjust the Y scale automatically when voltages go out of range.
- **AI sampling frequency**—Select a frequency (in Hz) that is appropriate for the type of data being recorded. The default setting is 100 Hz.
- **Chart's time scale window**—Select the duration (in sec.) of data that will be displayed on the main window. The default setting is 30 sec.

For Analog Outputs:

- **AO sampling frequency**—Select a frequency (in Hz) that is appropriate for the time resolution of the stimulus and position protocols that will be used in the experiment. The default setting is 1000Hz.

For Data File Path:

- **Path to location of data file**—Type the path to the location where the data files will be recorded in the experiment. If you prefer, use the **Browse** (📁) button on the right-side of the path box to select the location of the file folder. When the Windows Explorer window appears, navigate to the folder where you want to store your files (Fig. 22). The default setting is: **C:\WP\Experiments.tdms**.

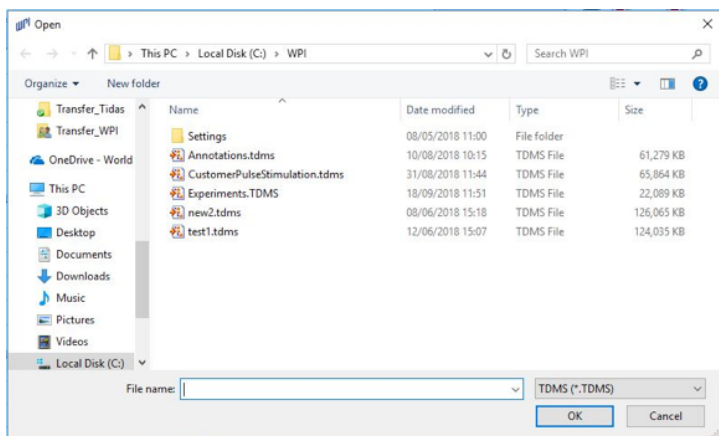


Fig. 22—Select the folder in which data files are saved during the experiments. Each TDMS file can contain multiple experimental runs.

- Prompt for description at experiment start**—Select this check box to allow you to enter a brief description of each experimental run and attach that notation to the recording. Entering descriptions for each run makes a file easier to locate when it is needed again. For example, the description entered in this text box appears in the *Open Data File* dialog box next to the file run number (Fig. 23).

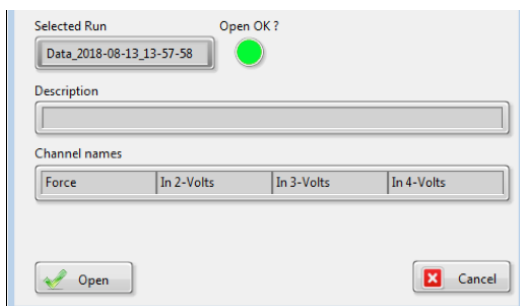


Fig. 23—The *Open Data File* dialog box displays the user-entered description in the *Description* text box.

- Click **OK** to activate the changes to the parameter settings.

Opening Recorded Data File

To open a TDMS run file, select the **File** menu and choose **Open TDMS Run**. The progress of file loading displays on screen as the TDMS file loads.

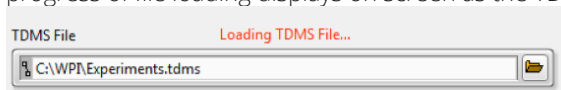


Fig. 24—The red text indicates that the selected TDMS file is loading.

When the file has loaded the name of the loaded TDMS file appears at the top of the window.

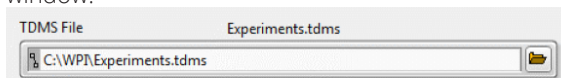


Fig. 25—The name of the selected TDMS file appears in the header area (Experiments.tdms).

Deleting Recorded Data Files

1. To delete a data run in a TDMS run file, select the **File** menu and choose **Delete TDMS Run**. The Delete TDMS Run window appears.

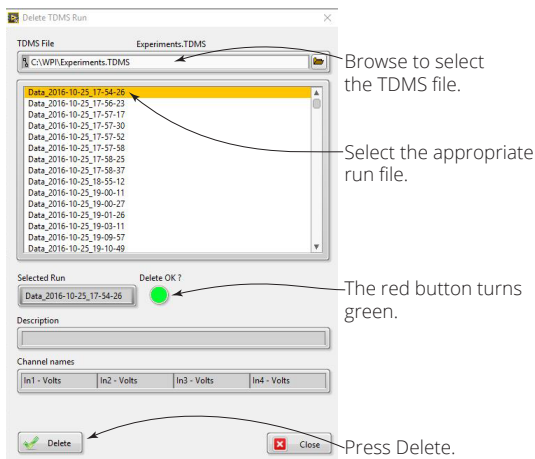
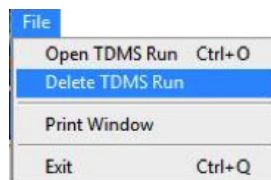


Fig. 26—The Delete TDMS Run window displays the run files, and the first one is selected.

2. To delete a TDMS run, browse at the top to select the appropriate TDMS file, and then select the desired data run in the center area of the window where all the runs are listed. The selected data run is highlighted and the LED switches from red to green indicating that the run is loaded and ready to be deleted. Press the **Delete** button.

Setting Channel Display Preferences

The display parameters of each channel have been set to default values that display data traces that are functional. These parameters may be customized for your preferences.

1. To customize the channel display parameters, right click on the appropriate channel icon located in the upper right margin of the Chart area (Fig. 27). The popup (context sensitive) menu (right) opens.

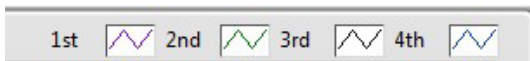


Fig. 27—The Channel icons are located in the top right corner of the main window.

2. Set the parameters, as desired.

On each channel, the following parameters can be set:




- **Plot Visible**—Check to display the recorded data trace on the selected channel. If it is not checked, the data channel appears blank.
- **Common Plots**—Select a plot other than the default smooth curve that is normally used for physiological recordings.
- **Color**—Choose a color for the data trace.
- **Line Style**—Select another line style for the data trace. The default is a continuous line.
- **Line Width**—Select another line width for the data trace.
- **Anti-Aliased**—Select to apply an anti-aliasing filter to the data recording. If this is not checked, the data channel is unfiltered.
- **Bar Plots**—If you want to display the data on one of five different bar graphs (in either a vertical or horizontal format), select an option here. The default setting is a line graph that is normally used for physiological recordings.
- **Fill Base Line**—Select how the area above or below the data trace is filled with color. Choices include None (no fill), zero, $-\infty$ and ∞ .
- **Interpolation**—Select the method used to fill the trace between the recorded data points. The default setting for interpolation is a smooth curve, which is best for most physiological signals.
- **Point Style**—Open this menu to select the style of point used to identify real data points on the trace.
- **X Scale**—Highlight X Scale to check the label used on the X-axis.
- **Y Scale**—Highlight Y Scale to check the labels used on the Y-axis of each channel.
- **Export**—To export the channel's data, select the destination/ location to export. Options available are clipboard, Excel and DIAdem.



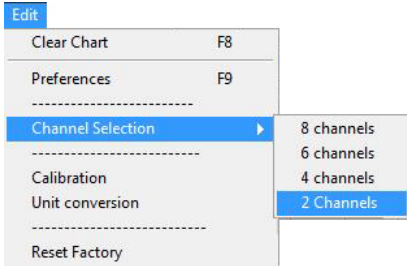
Plot Legend

The data run displayed on the main window can be optimized for viewing and printing



by using the data display controls   . These three controls appear on the upper left margin of the data window. For more information, see “Displaying/Printing Data Files” on page 31.

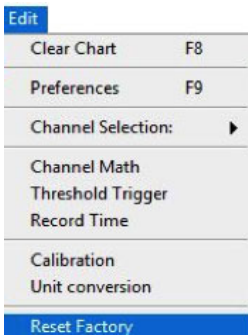
Setting the Number of Channels



You may select the number of channels you need for an application. Options include 4, 6 or 8 channels for acquisition. The acquired data chart on the main window adapts to the number of selected channels.

To change the number of channels, select the **Edit** menu (left), choose the **Channel Selection** menu item, and then select the number of channels from the submenu.

Resetting the Factory Defaults



To reset the factory defaults, select the **Edit** menu (left) and choose **Reset Factory**. This resets all the factory settings, as follows:

- Y-Label is set to +10V and -10V
- AO Frequency is set to 1000Hz
- AI Sampling Frequency to 100Hz
- Clears the graphs
- Resets the calibration multiplier to 1 and offset value to 0
- Y scale label to its default label name
- Chart's time scale window to 30 Sec
- Calibration Angle to 70 Deg

NOTE: Edited protocols are not reset.

Working with Annotations

Using annotations, you may enter notes that will appear on the graphs in the Analysis Window. Notes may be entered on the Main window during acquisition or on the Analysis window afterwards.

- Notes entered on the Main window (during acquisition) will display on the Analysis window, and they are permanently stored with the data. They can be moved but not deleted

Adding Notes to a Chart During Acquisition

1. To add notes to your chart, enter the note in the **Set Annotation** field in the upper right corner of the main window (Fig. 28).
2. As the data is acquired, press Enter to set the position of the note. Pressing enter attaches that note to a particular time during data acquisition.

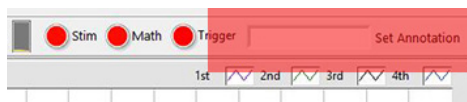


Fig. 28—Enter the note in the Set Annotation field and press Enter.

3. The annotations display in the Analysis window when you select the **Read Annotations** checkbox in the upper right corner of the Analysis window (Fig. 29).

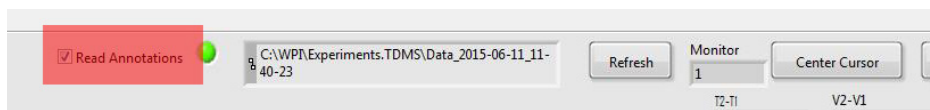


Fig. 29—Select the Read Annotations checkbox to display your notes on the graph.

NOTE: All annotations display in the first displayed graph, regardless of the channels.

Repositioning Notes

Notes added during acquisition may not be moved or deleted. They were written to the data file at specific times, and they are actually associated with those data points.

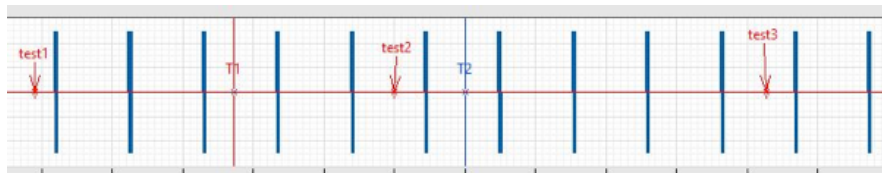
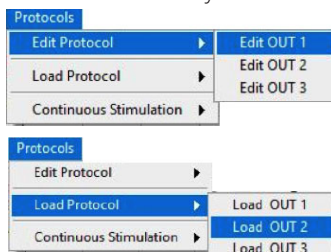


Fig. 30—When you drag the notes, arrows appear to indicate where the note was moved.

Composing/Editing/Saving Motor Position Protocols (OUT 1-3)

The motor used to stretch and relax muscle cells and fibers is controlled by the **OUT 1** analog output of the **LabTrax 8/16**. The amplitudes, durations and slopes of the signals that control the movements of the motor are programmed using an interactive window and saved for future use in a file known as a protocol.

1. To compose a position protocol, select **Edit Protocol** from the **Protocols** pull down menu on the main window (right).



- The **Edit Analog OUT 1** window opens (Fig. 31). Protocols are composed of segments that are linked together to move the motor according to the requirements of an experiment.

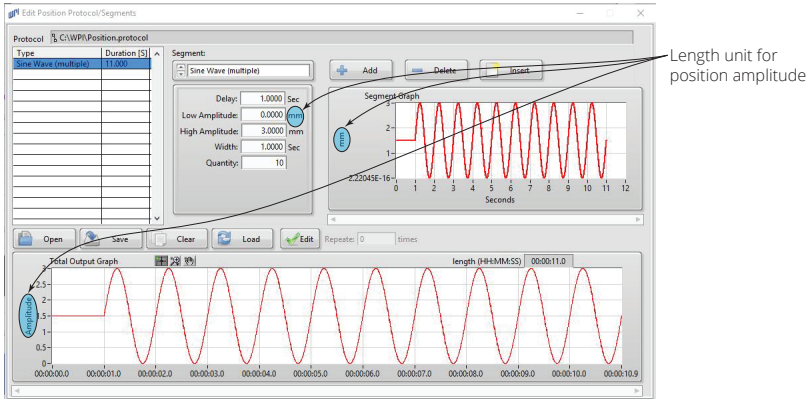


Fig. 31—The **Edit Analog OUT 1** window displays in the length units. This window shows the first segment added and type of segment, sine wavel, selected. The parameters of the segment are set at $\pm 1.5V$ for 11.00 seconds.


Amplitude for position protocol is programmed in factory length units, e.g. mm. The length unit may be user-defined. To define, click on Muscle Physiology from Toolboxes menu, then click on Motor Settings, which pops up Edit Motor Settings window. Then scroll and select the desired unit from OUT 1 Unit menu.

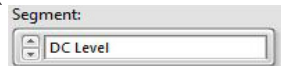
Click on Muscle Physiology/Motor Settings. The Motor Settings pop up displays for setting the following parameters:

- OUT1 UNIT – the unit used for editing protocols
- Out 1 Multiplier – the multiplier calibration factor
- OUT1 OFFSET – the offset calibration factor
- Set position (OUT 1) as In Channel for display purpose. Common is channel 1.

5. Set Motor Offset – this is the permanent motor OFFSET that may exist or can be set, despite setting the OUT1 Offset value. E.g. when your protocol includes an Offset value as starting position.

NOTE: Changing the factory calibration unit (OUT1 Unit) needs a corresponding change in the Multiplier value (OUT1 Multiplier)

- Start building a protocol by selecting the type of waveform needed in the first segment of the protocol. Find the waveform selection box in the upper center of the **Edit Analog OUT 1** window. Click on either the up or down arrow () to scroll



through the list of waveforms available and select the waveform needed in the segment. Waveforms include:

- DC Level
 - Square Wave (Multiple)
 - Staircase
 - Sine Wave (Single/Multiple)
 - Ramp (Single/Multiple)
4. Set the parameters of the waveform by entering values into the text boxes to the right of each parameter. Depending on the waveform selected, parameters can include:
 - Amplitude (V)
 - Width (sec.)
 - Quantity (No.)
 - Delay (sec.)
 - Interpulse Distance (sec.)
 - more...
 5. Once the parameters of the first segment have been set, click on the **Add** button to add the segment to the protocol. The list of segments in a protocol is shown on the left side of the protocol window (Fig. 31).
 6. To add another segment to the protocol, click on the **Add** button. Select the waveform type from the selection box as described in Step 3. Set the parameters of the waveform in this segment by entering values for each parameter as described in Step 4. This segment will be the next segment on the list.
 7. Continue to add the needed segments to the protocol, using the same procedure used to add previous segments, until the protocol is completed (Fig. 32).

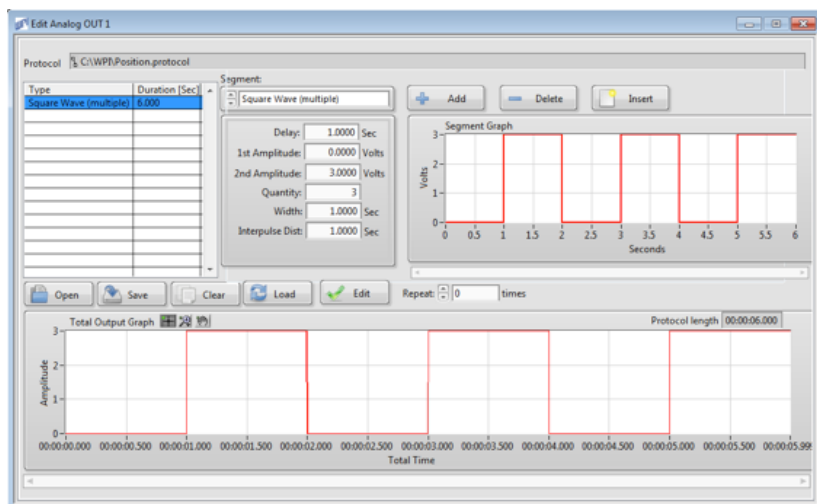


Fig. 32—It shows a square wave simulation protocol. As seen on the lower window, the motor is moved to specific positions at prescribed velocities by programming the voltages and duration of each segment.

8. To edit the type and the value of a waveform in a segment, highlight the segment on the list of segments. Make the changes to the waveform type and the values of the parameters. Click on the **Enter key** to enter the changes into the protocol.
9. To delete a segment from a protocol, highlight the segment to be deleted from the list of segments. Click on the **Delete** button and the segment will be deleted from the protocol.

10. To insert a segment into the list of segments, highlight the segment above the position of the new segment. Click on the **Insert** button to insert the new segment into the list. Highlight the newly-inserted segment. Set the waveform type and the values of the parameters. Click on the **Enter key** to enter the changes into the protocol.
11. To clear all the segments from the list and build a new protocol, click on the **Clear** button.
12. To save a protocol, click on the **Save** button. You will be directed to the folder designated as the storage site. Name the protocol, and click **OK**.
13. To edit a protocol that has already been saved, click the **Open** button in **Edit Analog OUT 1** window. Select the protocol to be altered from the list on the folder and click **OK**. Change the protocol using the tools explained above. Save the protocol using the same procedure outlined in Step 12.
14. Click the **Load** button and select a protocol to make a protocol active. The active protocol is the one that **MDAC** executes. The protocol will not run until it has been loaded into the software.
15. You may now use the **Repeat / Repeat for** utility, where a base protocol can be repeated x times or for a given time. This allows for programming only a base protocol (e.g. base stimulation burst) and letting the program repeat this base protocol to the desired protocol length.



Fig. 33—Repeat protocol selection option.

The mode of repetition may be changed by pressing the **Edit** button. A pop-up window opens where the mode of repetition can be chosen:

- Repeat x Times or
- Repeat x Seconds.

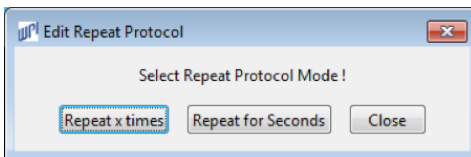


Fig. 34—Use the Edit Repeat Protocol window to configure the repeat protocol function.



CAUTION: If position and stimulation protocols are activated during a data recording, the recording will stop at the end of the longest active protocol. For example, if the position protocol is 20 seconds long and the longer of two active protocols, the recording will run for 20 seconds before turning off automatically. Make sure all protocols have the proper total durations. If no protocols are active, data recording will continue until you click the **STOP** button.

The same procedure is used for editing analog protocols for channels OUT2 or OUT3.

Editing Channel Math

Click on Channel Math under Edit drop down menu. This pops up Edit Channel Math window. The **Edit Channel Math** pop-up window (Fig. 35) allows you to run mathematical operations during online data sampling. Possible mathematical operations include adding, subtracting, multiplying or dividing two selected AI channels (In-Channel #1 and In-Channel #2) and displaying the result on a different selected AI channel (Out-Channel #). The online AI channel derivative is possible on a single selected AI channel (In-Channel #1 or In-Channel #2) and displaying the result on a different selected AI channel (Out-Channel #). Multiple mathematical operations are discarded. This is possible only when the selected mathematical operation is highlighted and all other operations are disabled (Fig. 36).

NOTE: AI In-Channel# and AI Out-Channel# should be selected in ascending order.

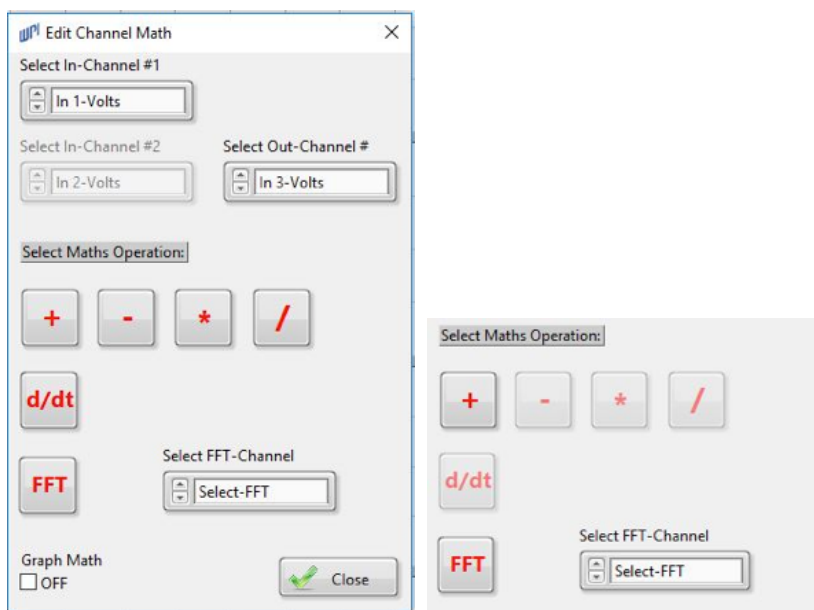


Fig. 35—The Edit Channel Math pop-up window shows the main mathematical functions.

Fig. 36—The Select Math Operation area shows the selected addition operation, while all other mathematical operations are disabled

Graph Math function can be turned ON/OFF either in the **Edit Channel Math** pop-up window by the **Graph Math** check box or in the MDAC Main window toolbar using the Math LED (green/red is ON/OFF).

In addition, you can run an online FFT analysis on single AI channel simultaneously, depending on the selected channel number. Channel number selection is considered in Select FFT-Channel Ring button.

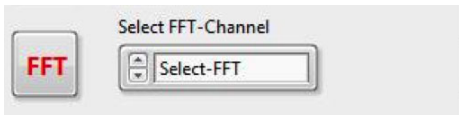
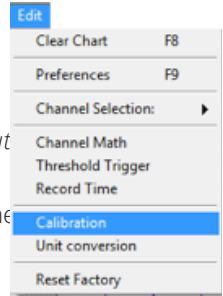


Fig. 37—The Select FFT-Channel is single selection and pressing FFT button with data sampling.



NOTE: Online FFT analyses can be done simultaneously with mathe

Calibrating a Transducer

1. Select **Calibration** from the **Edit** menu (Right). The *Edit Calibration* window appears (Fig. 38).

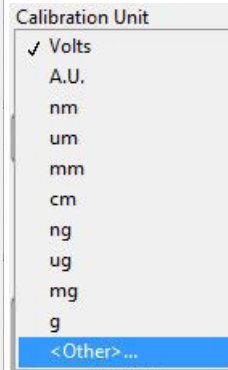
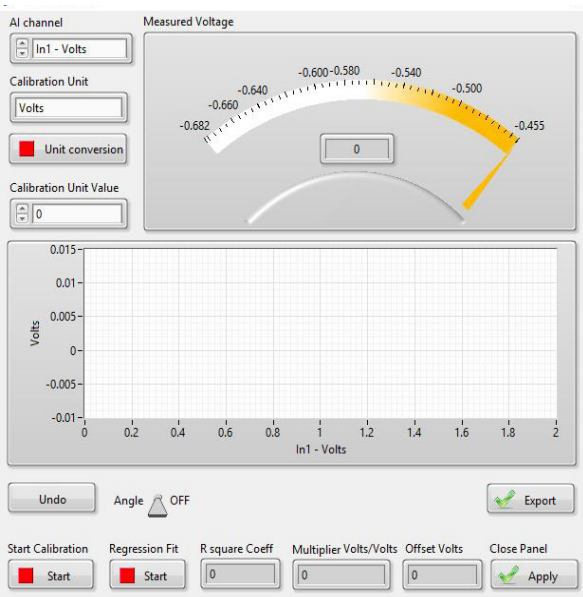


Fig. 38—(Left) The Edit Calibration window with main functions.

Fig. 39—(Right) Calibration units may be selected.

2. Select the AI channel to be calibrated. Switching between the AI channels empties the memory of its temporarily stored calibration values.

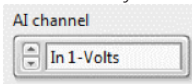


Fig. 40—Choose the AI Channel.

3. Select the Unit of your calibration value. You can select between g, mg, μg , ng, mm, μm , AU (arbitrary unit).

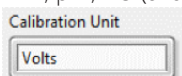


Fig. 41—Select your units of measurement.

4. Press the **Unit Conversion** button to use N, mN, μN or nN unit for force and g, mg, μg or ng unit for mass to obtain the conversion factor that is taken into consideration during the calibration process.
5. Enter the Unit Value, which is the value to be calibrated for the chosen unit.

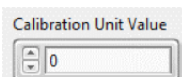


Fig. 42—Select the value to be calibrated.

6. Clear the calibration values from the memory by pressing the **Undo** button. You are prompted to select whether to clear the whole buffer of the calibration values or the last calibrated value.
7. When the calibration is in equilibrium, run the calibration, by pressing the **Start calibration** button. This acquires the voltage value for the chosen unit value.
8. Repeat the calibration for different unit values. The corresponding voltage is displayed in the Indicator graph as Measured Voltage. For each calibration the last 100 sampled points of the measurements at 128 samples/second are taken into consideration. When the calibration weight has stabilized, run the calibration measurement for at least 1–2 seconds.
9. To obtain the calibration factors, press the **Regression Fit** button. For the regression fit, you need at least two calibration points. The linear regression fit takes into account possible offset values of the transducer. A one point calibration is also possible. The calibration values and the regression fit line are displayed in the graph, as well as the R^2 coefficient, indicating the accuracy of the calibration. Calibration factors are calculated by using the linear equation $y = mx + b$ of the calibration values.

TIP: When you close the panel, MDAC takes the new calibration into consideration. Select Preferences from the Edit menu, and notice the y-axis label. If desired, you may customize the y-axis label.

10. Press the **Apply** button to save the calibration or press the X in the upper right corner to close the window without saving the new calibration information.

The **Calibration** menu includes the possibility to consider the angle inclination of a force transducer (or any sensor) when no force inclination stand is available.

Data measuring is done and obtained on raw values (i.e. display of the measured voltage without angle correction) while the regression fit transforms the transducer

calibration values to the corresponding Multiplier (Volts/Unit) and Offset (Volts) values, considering the possible angle compensation. These values are displayed in the Volts - In-channel relationship.

Angle compensation can be turned ON/OFF by switching the toggle button .Calibration values can be exported to Excel spreadsheet by pressing the **Export** button. (Data export includes the measured calibration values (compensated) and the data points used for the regression fit)

Calibration values, i.e. Multiplier of a desired sensor, are obtained by linear regression of the Calibration unit – Volts relationship to obtain a Unit/Volts calibration value. This value is automatically considered during data acquisition and displayed in the Preferences dialog.

- Display of calibrated Multiplier value in Unit/Volts.
- Display of calibrated Offset value in Volts.
- Display of R square coefficient to indicate goodness of fit (in **Edit Calibration** window)

It is also possible to include user-defined calibration units for additional sensors by selecting **Other**. The new user-defined calibration unit and its corresponding calibration values (Multiplier in Unit/Volts and Offset in Volts) are permanently stored and displayed in the Preferences dialog.

Unit Conversion

The Unit Conversion menu may be used to convert mass into force units.

1. Choose the **Edit** menu and select **Unit Conversion**. The Edit Unit Conversion window appears (Fig. 43).

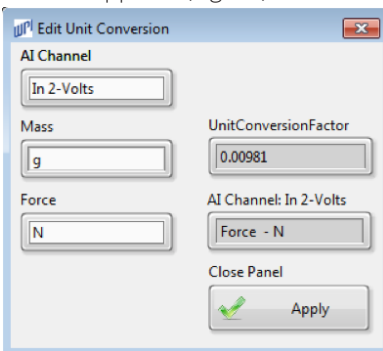


Fig. 43—Edit Unit Conversion window is used to convert mass into force units.

2. Select your AI channel and the Mass Unit to be converted into Force Unit. Once the Unit conversion is complete, the corresponding AI channel is grayed out and cannot be longer converted. The Conversion Factor and the Channel unit displayed are taken into consideration in Preferences window and the y-axis label of the corresponding chart.

NOTE: The conversion factor is taken into account with the Multiplier and Offset value shown on the Preferences window for the selected channel. For further conversion, use Preferences window.

Setting Record Time

For setting record time, click on **Record Time** from **Edit** drop down menu. This pops up **Edit Record Time** window.

The **Edit Record Time** pop-up window allows you to set a determined sample time during Free Run modus (no protocol active). Set parameters are the Record Time and the Time Unit (seconds, minutes or hours).

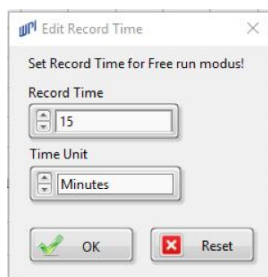


Fig. 44—Set Record Time pop-up window allows you to set a determined sampling time in the Free Run modus.

Editing Threshold Trigger

The **Threshold Trigger** under **Edit** menu allows online triggering of AI channels:

- For using the trigger function, select the trigger reference AI channel (In-channel #1) that corresponds to the reference signal of the **Absolute Threshold** trigger value, which in turn triggers the corresponding AI channel (In-Channel #2). Values below the trigger threshold value are cut and set to zero for the channel to be triggered.
- Display In-Ch#1 check box allows for triggering and displaying In-Ch #1 as its own reference and triggered signal on AI channel In-Channel #2.
- AI Trigger can be turned ON/OFF in **Edit Threshold Trigger** or by using the Trigger LED (ON/OFF is green/red) in MDAC Main Window.

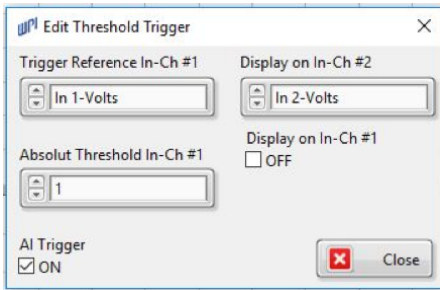


Fig. 45—The Edit Threshold Trigger pop-up window.

- Trigger operations are saved in user-settings and loaded on opening MDAC.

Online Filtering

A new function that has been included in MDAC - LabTrax-8/16 is the use of an online software filter (Butterworth – Lowpass, Highpass, Bandpass or Bandstop) during data acquisition for Analog Input signals (AI channels).

1. Press <F3> to open the AI Channel filter settings dialog window and for online filtering followings can be selected:
 - Desired channel to filter.
 - Filter parameters.
 - Choice to activate the filter (Filter ON – Filter OFF) for the desired channel.
 - Save all filter parameters each time a channel has been configured for filtering, for future use.

NOTE: Online digital filtering is done after channel math functions, allowing for online filtering of derivative signals. Similar for trigger functions.

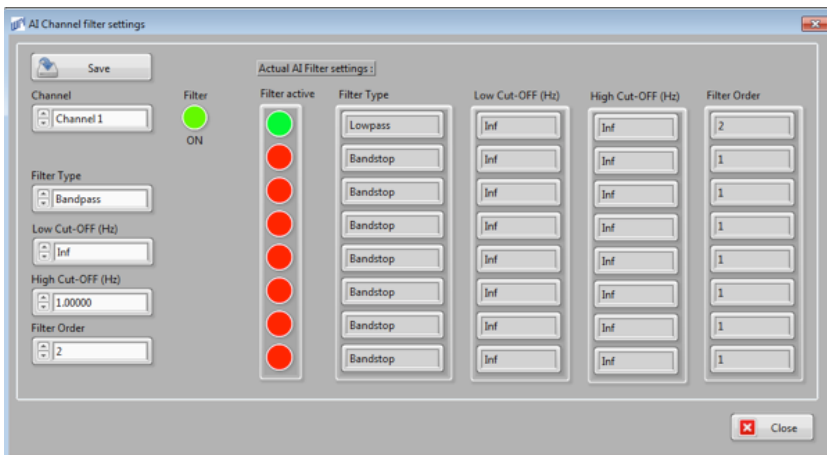


Fig. 46—The AI Channel filter pop-up window shows the settings dialog window for online filtering.

Toolboxes

Toolboxes

Muscle Physiology ▶

Motor Settings

The Muscle Physiology Toolbox is used to set Motor calibration values that are considered for protocol programming, i.e. use of length units for displacement protocols.

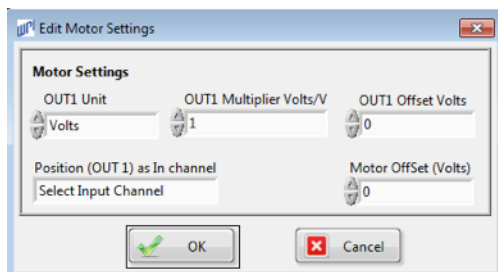


Fig. 47—<Edit Motor Settings> main window

Programmable variables are:

- OUT1 Unit as Volts, cm, mm, nm
- OUT1 Multiplier value in Volts/Units corresponds to the Motor Multiplier calibration value and allows for programming position protocols in length units.
- OUT1 Offset in Volts corresponds to the Motor Offset calibration value and allows for programming position protocols in length units.
- Motor Offset (Volts) avoids peak perturbation during position protocol.
- Position (OUT 1) as In channel sets the AI channel for sampling the real Motor response to a programmed position protocol (Observe to connect the Position Out BNC connection of Motor Control Unit to the selected AI channel of the LabTrax 8/16 board).

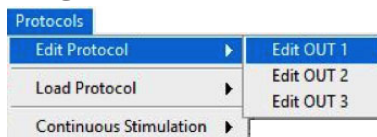
OPERATING INSTRUCTIONS

Data Recording

Once the preferences and parameters of the **LabTrax 8/16** data acquisition system are programmed and after the position, stimulation or digital output protocols are loaded, data recording can begin.

A. Selecting a Position Protocol (Analog OUT 1)

1. Select the position protocol, if any, to be used during the recording. Select **Load Position Protocol** from the **Protocols** pull down menu (right).



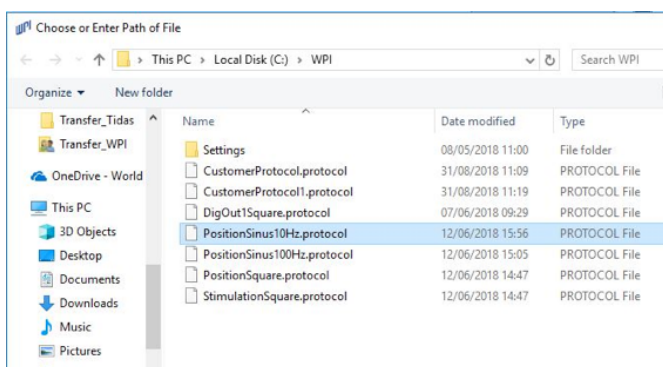


Fig. 48—Choose the location where the protocol is located and press OK.

2. On the next window (Fig. 48), navigate to the appropriate directory and select the position protocol needed in the experiment. Click **OK** to activate the selected protocol.
3. On the upper margin of the main window (Fig. 49), find the **Position (OUT1)** text box. The value to its right below the graph is the total duration of the selected position protocol.

NOTE: If the duration of the position protocol is shorter than other protocols, the final position output of the position protocol remain in effect until the last protocol completes.

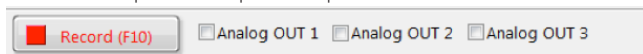
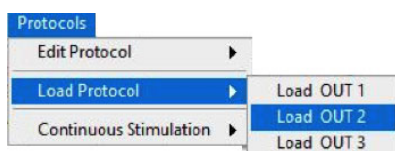


Fig. 49—The Position Output checkbox is in the upper left corner of the main window.

4. Select the check box left of the **Position Output** text box to mark the data recording.

B. Selecting a Stimulation Protocol (Analog OUT 2)

1. Select the stimulation protocol, if any, to be used during the recording. Select **Load Stimulation Protocol** from the **Protocols** pull down menu (right). The *Choose or Enter Path of File* window appears (Fig. 48).
2. Select the stimulation protocol needed in the experiment. Click **OK** to activate the selected protocol.
3. On the upper margin of the main window, find the **Stimulation (OUT2)** text box. The value to the right below the graph is the total duration of the selected stimulation protocol.



NOTE: If the duration of the stimulation protocol is shorter than the other protocols, the output is set to 0.0V when the protocol completes.

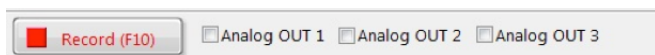


Fig. 50—The Stimulation Output checkbox is located to the right of the Position Output field.

4. Select the check box to the left of the **Stimulation Output** text box to mark the data recording.

C. Selecting Analog Output Protocols (Analog OUT 3)

1. Load any analog output protocols that are needed during the recording. Select **Load OUT 3** from **Load Protocol** under **Protocols** pull down menu (right). The **Choose or Enter Path of File** window appears (Fig. 48).
2. Select the digital output protocol needed in the experiment. Click **OK** to activate the selected protocol.
3. On the upper margin of the main window, find the **OUT #** text box .
NOTE: If the duration of the analog output 3 protocol is shorter than the other protocols, the output is set to 0.0V when the protocol completes.

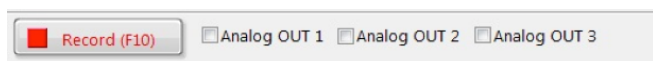
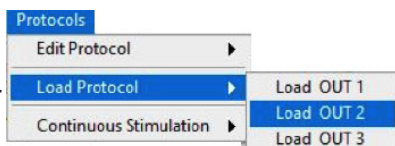


Fig. 51—Analog Output fields are located next to the Stimulation Output field.

4. Select the check box to the left of the **OUT3** text box to mark the data recording with lines that indicate the transition points between segments of this protocol (Fig. 52).

D. Setting up Continuous Stimulation

A continuous stimulation protocol allows you to run a predetermined stimulation protocol indefinitely.

1. From the **Protocols** pull down menu (right), select **Continuous Stimulation** then click **Edit Continuous** to edit a stimulation protocol. The **Edit Continuous Stimulation** window opens (Fig. 52).

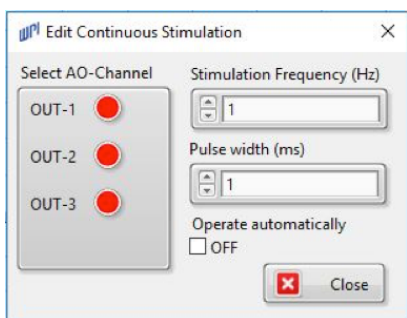
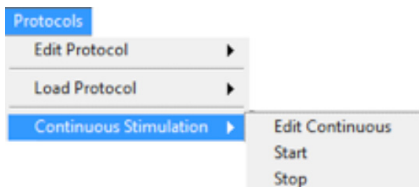
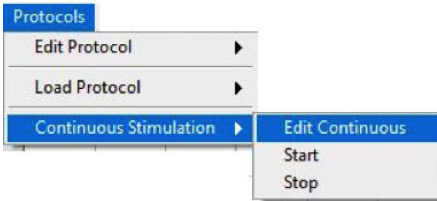


Fig. 52—The Edit Continuous Stimulation window allows you to change the frequency of the continuous stimulation protocol.

2. Select the desired analog output channel from the **AO channel** list box. Options include OUT1, OUT2 or OUT3.
3. In the **Stimulation Frequency** field, set the frequency in Hertz that will be used for continuous stimulation. The default value is 1Hz.

4. In the **Pulse Width** field, set the pulse width. The default is 1 ms.
5. Determine how you want to start and stop continuous stimulation:
 - If you would like for the software to turn the continuous stimulation on after each run and turn it off before each run, then select the **OFF/ON Operate automatically** checkbox.
 - Or, you can manually start and stop the continuous stimulation. From the **Protocols** drop down menu, select **Continuous Stimulation**. Then, choose **Start** or **Stop** from the submenu.



- It is possible to select all three AO channels to be used for continuous stimulation protocol, as a pacing function. Continuous Stimulation does not allow for data sampling.

When continuous stimulation is running, the **Stimulation LED** (located on the right side of the header area of the window) turns green. When there is no continuous stimulation, the **Stimulation LED** is red (Fig. 53). Simulation LED turns red during recording time.

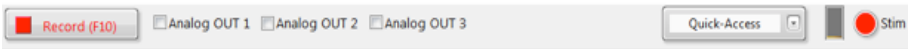


Fig. 53—The Stimulation LED turns red when no stimulation is taking place.

TIP: The four continuous stimulation channels may be activated and deactivated by pressing the Stimulation LED.

E. Recording Data

1. When you are ready to conduct an experiment, click the **Record (F10)** button on the upper left side of the main window. As soon as you click the button, it immediately toggles to a **Stop** button.
2. If a data run is still displayed on the main chart window, a dialog box appears as a reminder. To overwrite the data displayed on the main window, click **OK**.

NOTE: All previously recorded data runs are automatically saved in a data file if the data path was programmed on the **Preferences** window.

3. Another dialog window appears as the first closes. If needed, type a short description of the event being recorded on this data run to distinguish it from the other data runs in the data file. Click **OK** to close the dialog box, and the recording will begin in a few moments.


4. If position and/or stimulation protocols are activated, the recording stops at the end of the longest active protocol. For example, if the position protocol is 20 seconds long, and it is longer than the stimulation (or digital output) protocol, the recording will run for 20 seconds.


NOTE: If no protocols are active, data recording continues until you click the **STOP** button.

5. At any time, the recording can be halted by clicking the **STOP** button.
6. Continue recording runs until the experiment is complete.

Displaying/Printing Data Files

The data run displayed on the main window can be optimized for viewing and printing

by using the data display controls . These three controls appear on the upper left margin of the data window.

- To reposition the data on a channel, click the  button. This turns the normal cursor into a hand cursor. Place the hand cursor on the channel to be repositioned. Click and drag the hand to reposition the data on the selected channel. The other channels will move horizontally to synchronize the time bases on all channels, but they will not move vertically.
- To expand or compress the data displayed, use one of six functions on the **Zoom** menu.

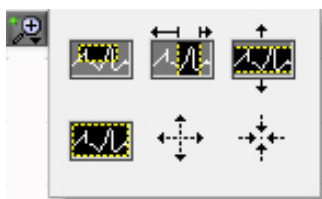








Fig. 54—Click the **Zoom** button to see the six expansion/compression functions on the **Zoom** menu.

Click on the **Loop** button in the array to open this menu (Fig. 54). Select the required function by clicking the symbol of the function. The selected function is highlighted by a blue box. Available functions include:

- **Zoom In**—Select the top left icon . The menu disappears when the icon is selected. On the data channel to be expanded, place the cursor above and to the left of the data to be expanded. Click and hold the left mouse button as you drag the cursor over the data area to be expanded. The selected area will expand to fill that channel display. The time base of the other channels also expands.
- **Horizontal Expansion**—Select the top center icon . On the data channel to be expanded, place the cursor to the left of the data to be expanded. Click and hold the left mouse button as you drag the cursor to the right and over

the data area to be expanded. The selected area will expand horizontally to fill that channel display. The time base of the other channels also expands.

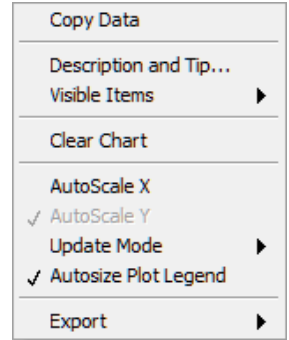
- **Vertical Expansion**—Select the top right icon . On the data channel to be expanded, place the cursor above the data to be expanded. Click and hold the left mouse button as you drag the cursor down and over the data area to be expanded. The selected area expands vertically to fill that channel display.
- **Zoom to Extent**—Select the bottom left icon . All the data in the run is compressed or expanded so that the whole run fits on a single window.
- **Step Expansion**—Select the bottom center icon . Place the cursor anywhere on any of the data channels. Click the left mouse button and the trace on each channel expands.
- **Step Compression**—Select the bottom right icon . Place the cursor anywhere on any of the data channels. Click the left mouse button and the trace on each channel compresses.

Main Window Context Menu

Context-sensitive menu available on the main window (right). To open the menu, right-click on the **Acquired Data** window.

Export submenu can be used to export data.

- Use this submenu to select the method needed to export the data points in the data run. Data can be exported to:
 - Clipboard
 - Microsoft Excel
 - DIAdem
 - Simplified image.



Analyzing Data

From the Analysis window, you have the opportunity to apply a variety of filters, mathematical calculations and statistical comparisons to your data.

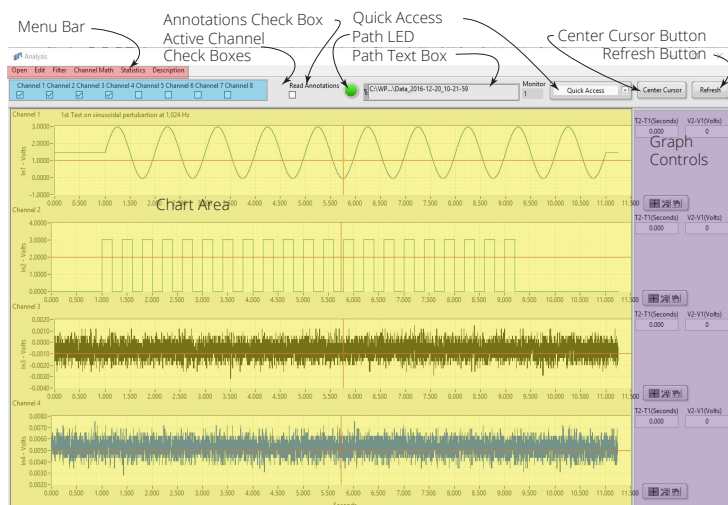



Fig. 55—The Analysis Window graphically displays data from a selected TDMS data file.

Menu Bar—Access all the pull down menus for the *Analysis* window

Active Channel Check boxes—A number of check boxes are arranged in the upper left corner of the window. These check boxes indicate which channels are active in the open data file.

Annotations Check box—Select this check box to display any notes that have been entered.

Path text field shows the path of the data file you selected. The **Path LED** next to it turns from red when the file is loading to green when the file is ready for use.

 —The **Refresh** button is used to update the display. For example, if you use more than one screen (monitor) on your PC, **Refresh** may be used to update the display.

Monitor Field—This is a read only field. If you are using more than one monitor screen with your computer, you may drag the *Analysis* window to a secondary monitor and resize it as desired. Then, press **Refresh** to update your display. The monitor which is actually used displays in this field, and the *Analysis* window dynamically adapts to the new monitor/window size.

Press the **Center** button to center the cursor for each channel in the **Chart Area**.

Chart Area—The data for each channel displays in this area, along with a control legend for each channel.

Graph Controls—When two cursors are used for a single channel, the time difference between the cursors in that channel appears in the field labeled **T2-T1**, and the value difference is displayed in the field labeled **V2-V1**. **Data Display Controls** are also available for each channel. See “Displaying/Printing Data Files” on page 31.

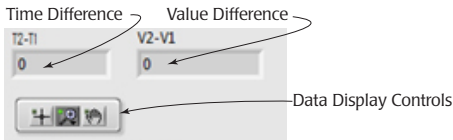


Fig. 56—The graph controls.

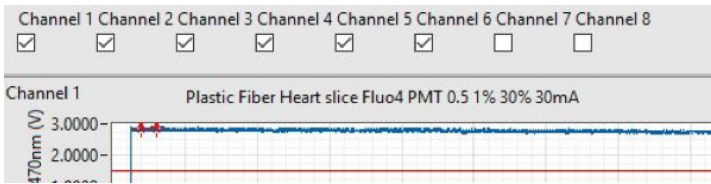


Fig. 57—The Description displays the file description above the first selected channel.

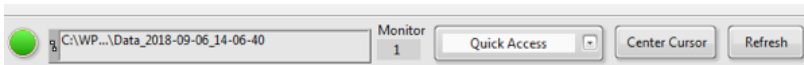


Fig. 58—The Refresh button, refreshes all data and axes changes like re-open the data file.



Fig. 59—The Quick Access menu allows for direct graph/signal processing using pre-defined parameters or formerly set parameters of the selected function.

Quick Access allows for direct access to a limited number of functions, which are applied directly either to all graphs/channels or single selected graph/channels, depending on the Time Scale selection and active channel.



Fig. 60—The Quick-Access pull-down menu selection.

Working with Data on the Graphs

After you open a data file in the Analysis window, you have options for working with the file using functions which may be found in the context sensitive menu. Display the graph pop-up menu (left) by right clicking in the chart area of the graph you want to work with.

NOTE: Most of the functions of this menu are activated/can be used only when MDAC main window is closed.

Visible Items—From this submenu (right), you may select the legends and labels to be displayed on the Analysis window. Items that may be displayed include:

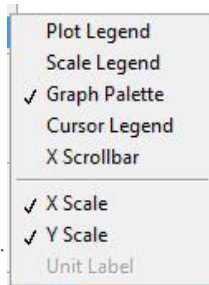
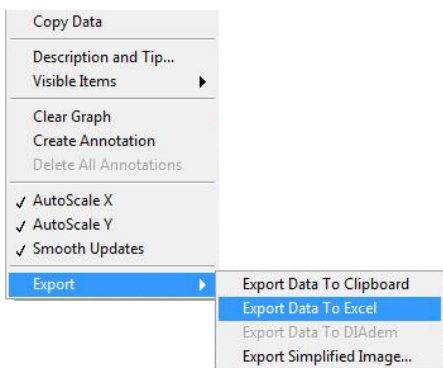
- Plot Legend
- Graph Palette
- X Scrollbar
- Y Scale
- Scale Legend
- Cursor Legend
- X Scale
- Unit Label

Clear Graph—Use this menu item removes the selected graph from the display. Cleared items are not deleted. To reload the data, restore the defaults. See “Restoring Defaults” on page 47.

Create Annotation—Use this menu item to set marks on the graph. Annotations are automatically saved with the data file.

Delete All Annotations—Selecting this menu item deletes all annotations that you have added to the graph.

AutoScale X—Selecting this option scales the X-axis so that all data points fit on the same display window.



AutoScale Y– Selecting this option automatically adjusts the Y scale when values go out of range.

Smooth Updates–Select this menu option to remove minor fluctuations from the display of the graph.

Export–From this submenu, you have several options to export your data. See “Exporting Data from a Single Graph” on page 36.

Exporting Data from a Single Graph

1. To export data from a single channel, right click on a graph in the chart area (of the *Analysis* window) to display the popup menu (above).
2. Select the **Export** submenu and choose one of the export options:
 - **Export Data to Clipboard**–The graph data is copied to the clipboard and may be pasted into a text or spreadsheet file.
 - **Export Data to Excel**–The file opens in Microsoft Excel with a generic name. If desired, you may select **Save** from the Excel **File** menu to save the file.
 - **Export Data to DIAdem**–The file is exported in a format that may be opened by National Instruments DIAdem program.
 - **Export Simplified Image**–An image of the graph (in bitmap, EPS or EMF format) may be captured on the clipboard or saved to a file. See “Saving an Image of a Graph” on page 36.

Saving an Image of a Graph

1. Right click on a graph in the chart area (of the *Analysis* window) to display the popup menu. Select the **Export** submenu and choose **Export Simplified Image**. The *Export Simplified Image* window appears (Fig. 61).

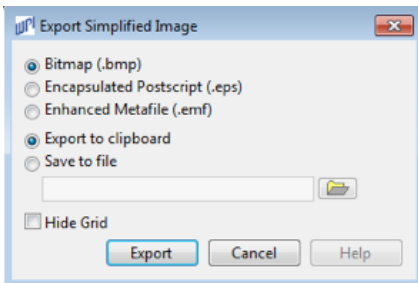



Fig. 61—The *Export Simplified Image* file lets you determine how the image exports.

2. Select the type of image file to export (Bitmap, Encapsulated Postscript or Enhanced Metafile).
3. Select **Export to clipboard** or **Save to file**. If you select **Save to file**, enter the directory path where you want to save the file in the text field or click on the  button to select a file location.
4. Click **Export**.

Saving a Processed File in a Binary Format

When you finish processing a file, you may save it in a binary format. Select the **File** menu (in **Analysis** window) and choose **Save Binary File**. The file will be saved in the same location and with the same name as the open data file. The file is saved with a .FLT extension

Opening a Data File for Analysis

1. To analyze collected data, select **Analysis** from the main menu. This opens **Analysis** window.

NOTE: If the *Analysis* window is already open, you may:

- Open a different data file by selecting **File>Open TDMS File**.
- Open a binary data file by selecting **File>Open Binary File**.
- Open a text data file by selection **File>Open Text File**.

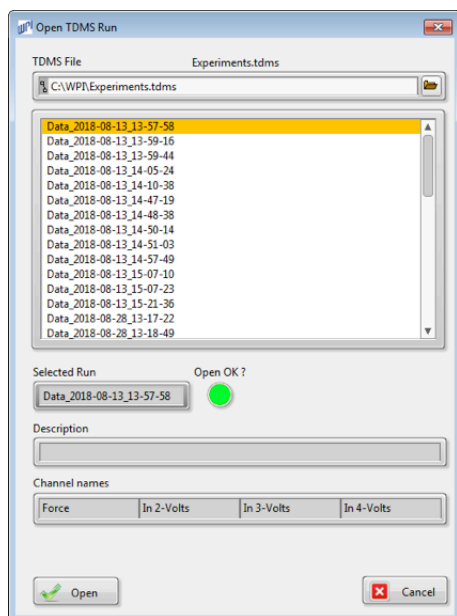



Fig. 62—The *Open TDMS Run* window allows you to select a file for analysis.

2. The main section of the window displays the data runs saved in that folder (Fig. 62). To open another folder location, click on the **Browse** button  to the right of the **TDMS file** text box. On the next window, browse the computer to locate and highlight the required folder. Click the **OK** button to open it.
3. Click on a file to highlight it. If the file is not ready to be opened for analysis, the **Open OK** LED turns red. If the file is ready to open, the **Open OK** LED turns green.

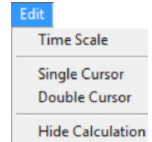
- Press **Open** button. The **Analysis** window displays the original data set and the data from all the recorded channels are displayed.

TIP: If you do not wish to see the data from all the channels in the data file, you may un-select the **Active Channel Check boxes** of the channels you do not wish to analyze.

Changing the Display of a Data File

The options available for changing the display of a data file can be found in the **Edit** menu (of the **Analysis** window).

- "Changing the Time Scale" on page 38.
- "Hiding Calculations" on page 39.
- "Using a Single or Double Cursor" on page 39.

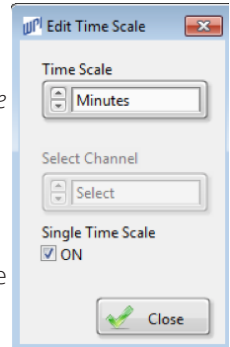


Changing the Time Scale

- To change the time scale of the displayed channels, click on the **Edit** menu (of the **Analysis** window) and select **Edit Time Scale**. The **Select Time Scale** window appears.

Fig. 63—Select the Time Scale unit and check box to switch Single Time Scale ON/OFF.

- Use the arrow keys to select the time scale. Options include Hours, Minutes and Seconds.
- This menu allows for selecting either Single Time Scale ON to use all cursors simultaneously or Single Time Scale OFF to use a given cursor for one single active graph/channel.



In Single Time Scale OFF mode, the active graph/channel is highlighted by the red Channel Name (e.g. Channel 1 Active). Place the cursor of the desired channel to activate a channel/graph. This is only used when you are moving the cursor on the selected active channel/graph. It may also be used in Quick Access to realize data/signal processing directly on only the active channel.

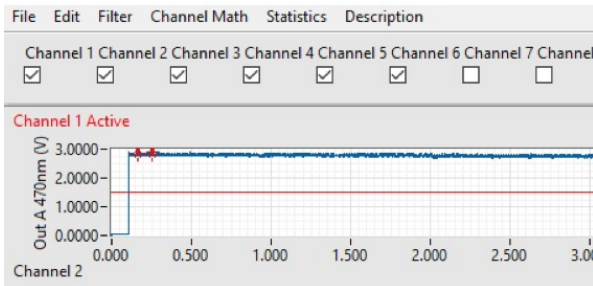
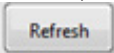


Fig. 64—The channel name indicator switches to Channel 1 Active (red display), when Single Time Scale is set to OFF.

- Click **Close**

Hiding Calculations

To hide the calculation windows, click on the **Edit** menu (of the *Analysis* window) and select **Hide Calculation**. To reveal the calculation windows again, press .

Using a Single or Double Cursor

To display a single cursor in each graph, click on the **Edit** menu (of the *Analysis* window) and select **Single Cursor**.

To display two cursors on each graph, click on the **Edit** menu (of the *Analysis* window) and select **Double Cursor**. The red cursor is the first cursor. It has the coordinates (T1, V1), where T1 is the first time and V1 is the first value. The second cursor is blue with coordinates (T2, V2).

Info Data shows the file description just above the first selected channel. Can be turned-on by first selection and turned-off by second selection.

Applying a Butterworth Filter

The Butterworth filter is an FIR (finite impulse response) filter which can be used as a low pass filter, a high pass filter, a band pass filter or a band stop filter.

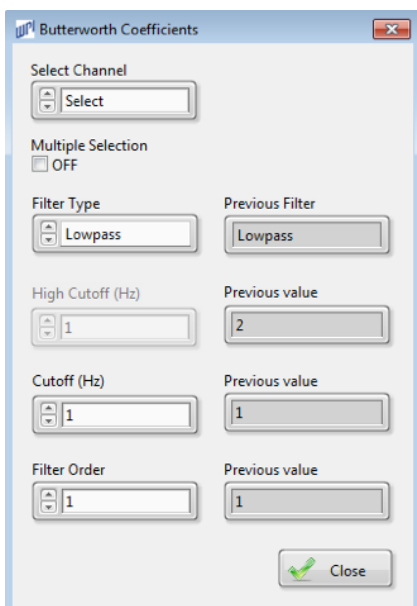


Fig. 65—The Butterworth Coefficient window lets you set the parameters of your filter.

1. To apply a Butterworth filter, click on the **Filter** menu (of the *Analysis* window) and select **Butterworth**. The *Butterworth Coefficients* window appears (Fig. 65).

2. Use the arrows next to the **Select Channel** field to select a single channel or all channels. By setting up **Multiple Selection ON**, you can filter as many channels or combination of channels that you like.
3. Use the arrows next to the **Filter Type** field to select a filter. Options include Low Pass Filter, High Pass Filter, Band Pass Filter and Band Stop Filter.
4. Use the arrows next to the **Cutoff** fields to apply the appropriate cutoff frequencies. Options available depend upon the type of filter selected.
5. Use the arrows next to the **Filter Order** fields to set the filter order. The higher the filter order the steeper the response roll off.
7. Click **Close button**

Filter or **Channel math** effect of selected parameters is displayed directly in MDAC Analysis Window to give a preview of the effect of the selected configuration. Use the Select option in **Select Channel** ring menu to undo filter or channel math configuration's direct effect on data/signals. This filter configuration functioning is like all data/signal processing methods.

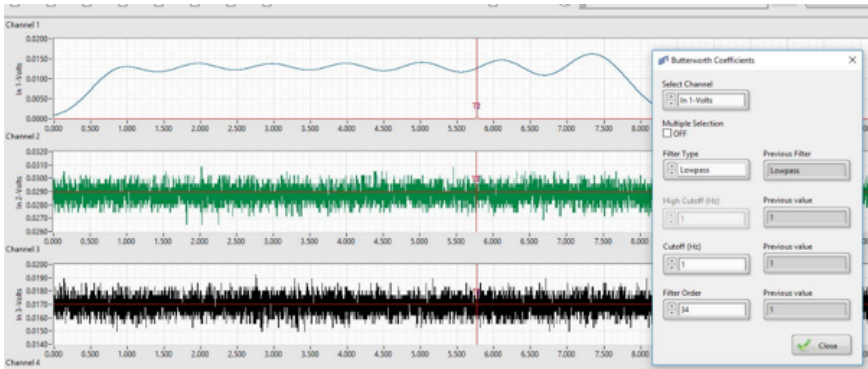


Fig. 66—The Butterworth filter configuration effect and preview on direct processed data/signals in MDAC Analysis Window.

NOTE: Data/signal processing is done on the entire data set of the selected channel, while only the processed signal of the formerly selected time range is displayed.

Applying a Savitzky-Golay Filter

The Savitzky-Golay filter is a polynomial filter.

1. To apply a Savitzky-Golay filter, click on the **Filter** menu (of the *Analysis* window) and select **Savitzky-Golay**. The *Savitzky-Golay Coefficients* window appears (Fig. 67).

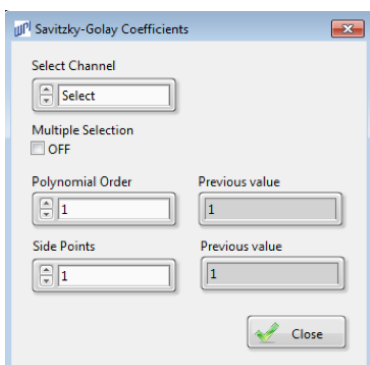


Fig. 67—The Savitzky-Golay Coefficient window lets you establish the parameters of your filter.

2. Use the arrows next to the **Select Channel** field to select a single channel or all channels. By setting up **Multiple Selection ON**, you can filter as many channels or combination of channels that you like.
3. Use the arrows next to the **Polynomial Order** field to establish the order that is applied.
4. Use the arrows next to the **Side Points** field to set the number side points to be filtered.
6. Click **Close** button

Applying a Moving Average Filter

The moving average filter calculate the mean value over a set period of time (averaging length) for each sampled data point. No data points are missing in the sampled data. This filter is effective for small data sets, but need calculation times for larger data sets.

1. To apply a moving average filter, click on the **Filter** menu (of the *Analysis* window) and select **Moving Average**. The *Moving Average Coefficients* window appears (Fig. 68).

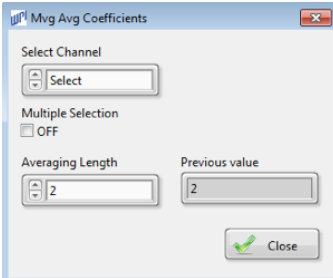


Fig. 68—The Moving Average Filter Coefficient window lets you establish the parameters of your filter.

2. Use the arrows next to the **Select Channel** field to select a single channel or all channels. By setting up **Multiple Selection ON**, you can filter as many channels or combination of channels that you like.
3. Use the arrows next to the **Averaging Length** field to establish the number of data points for averaging. In the sample (Fig. 68), two data points are averaged.
4. Click **Close** button

Down Sampling Data

You can down sample data. Down sampled data becomes the default data that is used for any other analysis. If you need to restore the original data see “Restoring Defaults” on page 47.

1. To down sample the data to a specific sample frequency, click on the **Filter** menu (of the *Analysis* window) and select **Downsample Data**. The *Sample Coefficients* window appears (Fig. 69).

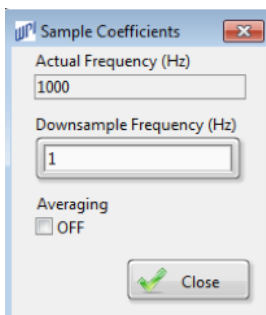


Fig. 69—The *Sample Coefficients* window lets you establish the parameters of your filter.

2. Click **Close** button

Removing Filters

When necessary, you may reset the graphs and the cursors to their original portions and undo any resampling.

1. Click on the **Filter** menu (of the *Analysis* window) and select **Default**. The *Default Signals* window appears (Fig. 70).

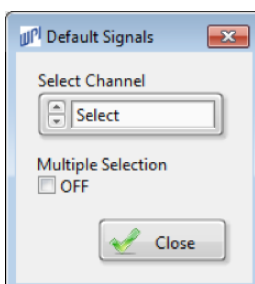


Fig. 70—The *Default Signals* window lets you return to your original data display.

2. Use the arrows next to the **Select Channel** field to select a single channel or all channels. By selecting **Multiple Selection** checkbox **ON**, you can restore the data display of the channels that you like.
3. Click **Close** button

Running Mathematical Operations on Data

You may run mathematical analyses on the data. Options include:

- “Calculating Derivatives” on page 44.
- “Calculating Integrals” on page 45.
- “Calculating Slopes” on page 46.
- “Calculating Latency” on page 46.
- “Restoring Defaults” on page 47.

Calculating Derivatives

You have four options when calculating the derivatives of the data: Signal, Maximum Value, Minimum Value and Cursor Value.

1. To calculate a derivative of data on a selected graph, click on the **Channel Math** menu (of the **Analysis** window) and select **Derivative**. Then, choose:
 - **Signal** to calculate the derivative of the signal. The derivative displays as a line on the graph.
 - **Max Value** to calculate the derivative of the maximum value. The **Max Derivative** window appears (Fig. 71).
 - **Min Value** to calculate the derivative of the minimum value. The **Min Derivative** window appears (Fig. 73).
 - **Cursor Value** to calculate the derivative of the value where the cursor is located. The **Cursor Derivative** window appears (Fig. 74).

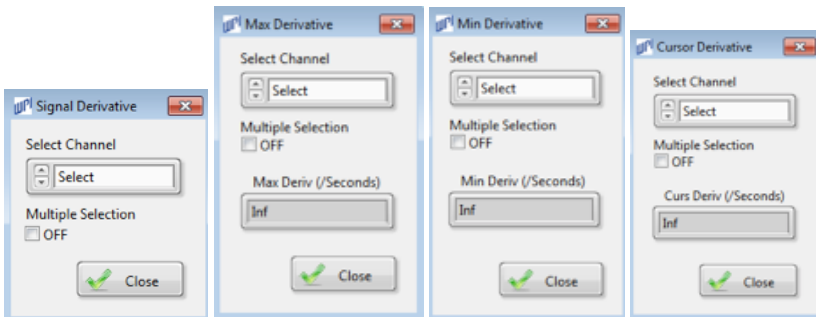


Fig. 71—(Left) Set the parameters for getting the derivative of the signal.

Fig. 72—(Center Left) Set the parameters for getting the derivative of the maximum value.

Fig. 73—(Center Right) Set the parameters for getting the derivative of the minimum value.

Fig. 74—(Right) Set the parameters for getting the derivative of the value of the point where the cursor is positioned.

2. Use the arrows next to the **Select Channel** field to select the desired channel.
3. If you want derivative of multiple channels. Set Multiple Selection checkbox ON
4. In case of max, min and cursor derivative, the derivative value is seen in its respective derivative window (except for All option).
5. Press Close to close the window.

Calculating Integrals

1. To calculate an integral of a selected graph, click on the **Channel Math** menu (of the *Analysis* window) and select **Integral**. The *Signal Integral* window appears.

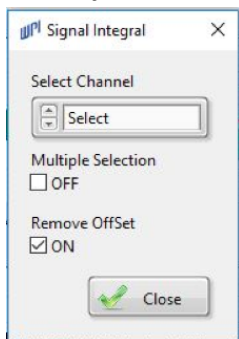


Fig. 75—The *Signal Integral* window lets you choose the channel to integrate.

2. Use the arrows next to the **Select Channel** field to select the desired channel.
3. When the **Default Data** checkbox is selected, the filter is applied to the original data set. Unselect the checkbox to apply the filter to the processed data.
4. When the **Remove Offset** checkbox is selected, the signal offset is removed to avoid possible drift. This is the default setting. You do not need to change it when drift should be avoided.

However, if you want to use the unprocessed data with its offset, unselect the checkbox. This may be necessary when a derivative is used or when you are integrating a signal. For example, when a DC or ramp signal needs to be integrated to obtain the mathematical corresponding signal like a slope or square signal, you want to include the offset.

5. Press **Close**. The integral of the selected channel displays in the chart area of the graph.



Fig. 76—The integral displays on the chart area.

Calculating Slopes

This option may be used when you are operating with double cursors. It is used to calculate the slope value between the two cursor values. The coordinates of the red cursor are (T1, V1) and the coordinates of the blue cursor are (T2, V2). So, the slope is calculated using the formula:

$$\text{Slope} = (V2-V1)/(T2-T1)$$

1. To calculate the slope value between two cursor values on a selected graph, position your cursors as desired. Then, press **Close**. Then, click on the **Graph Math** menu (of the **Analysis** window) and select **Slope**. The Slope window appears (Fig. 77).

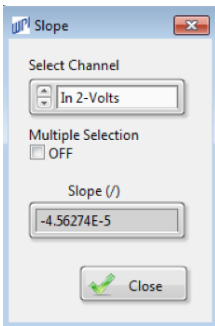


Fig. 77—The Slope Value window allows you to calculate the slope between two points on a selected channel.

2. Use the arrows next to the **Select Channel** field to select the desired channel.
3. You may set **Multiple Selection** checkbox **ON** to get slop of multiple channels.
4. Slope is displayed in **Edit Slope** window (but not in the case when All channel is selected). Slop is also available in the right top of all the corresponding channels.
5. Press **Close** to close the window.

Calculating Latency

Using the latency option, you can calculate the latency between two cursor time values of two selected graphs. For example, the first value could be T1 of Channel 2 and the second could be T1 of Channel 1.

1. To calculate the latency value between two selected points, click **Channel Math** menu (from **Analysis** window) and select **Latency**. This pops up a window. Click **OK**. Now, you can place **T1** cursor independently in different channels. Place the cursor as desired.
2. To calculate the latency value between the two selected points, click on the **Graph Math** menu (of the **Analysis** window) and select **Latency**. The **Latency Value** window appears.
3. Latency value is displayed in Latency window. You can further find the value on the right top of corresponding channel.
4. Press **Close** to close Latency Value window.

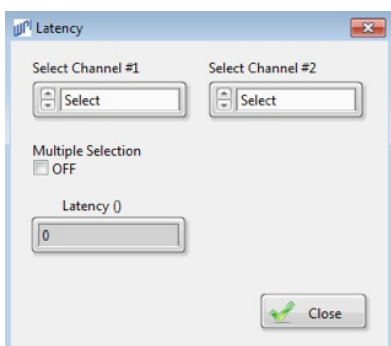


Fig. 78—The Latency Value window allows you to calculate the latency value between two points on two different channels.

Restoring Defaults

When necessary, you may reset the graphs and the cursors to their original portions. This deactivates any cursors that have been set.

1. Click on the **Channel Math** menu (of the *Analysis* window) and select **Default**. The *Default Signals* window appears (Fig. 79).

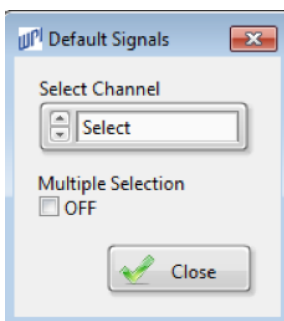


Fig. 79—The Default window lets you return to your original data display.

2. Use the arrows next to the **Select Channel** field to select a single channel or all channels. By setting **Multiple Channel** checkbox **ON**, you can restore the data display of the multiple channels that you like.
3. Click **Close**.

Running Standard Statistical Calculations

To calculate the standard statistical values of all of the sampled data, click on the **Statistics** menu (of the *Analysis* window). The calculation is performed on the data between the selected (positioned) cursors. This calculates the mean, minimum, maximum, standard deviation and median values of all graphs. The values display in the **Get Statistics** window (Fig. 80).

Click **Save** button to save the file or click **Close** button to close the window.

	Max Value	Min Value	Mean	Median	Standard Deviation
In 1-Volts	2.97089	0.01312	1.48744	1.46042	1.03513
In 2-Volts	0.03204	0.02838	0.03026	0.03021	0.00058
In 3-Volts	0.02014	0.01556	0.01781	0.0177	0.00061
In 4-Volts	0.01862	0.01434	0.01676	0.01678	0.0006

Fig. 80—The Get Statistics Window displays the standard statistical values for all sampled channels.

SPECIFICATIONS

This unit conforms to the following specifications:

Number of Inputs	8 Analog
ADC Resolution	16 bit, 305 μ V/bit
Maximum Sampling Rate.....	400 ksps aggregate in AI mode 10 ksps/channel (software limited)
Input Connections.....	8 BNC
Input Impedance	1 M Ω
Input Range.....	+10V to -10V
System Noise	<1mV RMS
Digital Input/Output.....	16/16
Power Source	12VDC
Operating Current.....	800mA maximum
PC Interface.....	USB 2.0
Digital and Analog Inputs.....	Maximum: 30V Minimum:-30V
Logic High Voltage	2.3V minimum
Logic Low Voltage.....	1.0V maximum
Analog Outputs.....	3
Resolution.....	16 bits
Output Range	+10V to -10V

INDEX

Symbols

16-bit 1

A

AI sampling frequency 12
amplitude 19
analog to digital converter 1
analyze collected data 37
analyzing data 33
AO sampling frequency 12
autoscale 12
autoscale X 35
autoscale Y 36

B

band pass filter 39
band stop filter 39
bar plots 15
binary format 37
Butterworth filter 39

C

center the cursor 33
channels 16
chart area 34
clear graph 35
clipboard 36
color 15
COM 10
common plots 15
communication error 10
compress the data 31
continuous stimulation 29
create annotation 35
cursor legend 35

D

data recording 27
DC Level 18, 19
defaults 16
delay 19
delete all annotations 35

derivative 44
DIAdem 32, 36
double cursor 39, 46
down sample 43

E

Excel 36
expand the data 31
export 15, 36

F

factory defaults 16
fill base line 15
finite impulse response 39
FIR 39
FTDI 5

G

graph controls 34
graph palette 35

H

hide the calculation 39
high pass filter 39
horizontal expansion 31

I

interpolation 15
installation 2
integral 45
interpulse distance 19

L

latency 46
license 3
line width 15
low pass filter 39

M

main window 32
mathematical analyses 44
mathematical calculations 33
maximum 47
mean 47
median values 47
minimum 47
monitor 33

moving average filter 42
multiplier 12

N

National Instruments 3, 4

O

offset 12

P

parts list 1
plot legend 35
plot visible 15
point style 15
polynomial filter 41
position protocols 17
power supply 8
preferences 10
print data file 16, 31
Prompt for description at experiment start 13
protocols, position 17

Q

quantity 19

R

ramp 19
record data 30
recording 11
refresh 33
remove filters 43
reposition the data 31
restoring defaults 47
returns 2

S

sampling 11
sampling frequency 12
Savitzky-Golay filter 41
scale legend 35
setup.exe 2
SI-AOSUB 8
SI-BAM21-LCB 8
SI-CISB 8
SI-CTS 8
SI-MKBM 8

SI-MOTDB 8
simplified image 36
SI-MTM 8
sine wave 19
single cursor 39
slope value 46
smooth updates 36
square wave 19
staircase 19
standard deviation 47
statistical values 47
step compression 32
step expansion 32
stimulation 8
stimulation, continuous 29
stimulation protocol 28

T

time scale of the displayed channels, click on the Edit menu (of the Analysis window) and select 38

U

unit label 35
unpacking 1
USB 8

V

vertical expansion 32

W

waveform 18
width 15, 19

X

X scale 15, 35
X scrollbar 35

Y

Y scale 15, 35
Y scale label 12

Z

zoom 31
zoom in 31
zoom to extent 32

WARRANTY

WPI (World Precision Instruments, Inc.) warrants to the original purchaser that this equipment, including its components and parts, shall be free from defects in material and workmanship for a period of one year* from the date of receipt. WPI's obligation under this warranty shall be limited to repair or replacement, at WPI's option, of the equipment or defective components or parts upon receipt thereof f.o.b. WPI, Sarasota, Florida U.S.A. Return of a repaired instrument shall be f.o.b. Sarasota.

The above warranty is contingent upon normal usage and does not cover products which have been modified without WPI's approval or which have been subjected to unusual physical or electrical stress or on which the original identification marks have been removed or altered. The above warranty will not apply if adjustment, repair or parts replacement is required because of accident, neglect, misuse, failure of electric power, air conditioning, humidity control, or causes other than normal and ordinary usage.

To the extent that any of its equipment is furnished by a manufacturer other than WPI, the foregoing warranty shall be applicable only to the extent of the warranty furnished by such other manufacturer. This warranty will not apply to appearance terms, such as knobs, handles, dials or the like.

WPI makes no warranty of any kind, express or implied or statutory, including without limitation any warranties of merchantability and/or fitness for a particular purpose. WPI shall not be liable for any damages, whether direct, indirect, special or consequential arising from a failure of this product to operate in the manner desired by the user. WPI shall not be liable for any damage to data or property that may be caused directly or indirectly by use of this product.

Claims and Returns

Inspect all shipments upon receipt. Missing cartons or obvious damage to cartons should be noted on the delivery receipt before signing. Concealed loss or damage should be reported at once to the carrier and an inspection requested. All claims for shortage or damage must be made within ten (10) days after receipt of shipment. Claims for lost shipments must be made within thirty (30) days of receipt of invoice or other notification of shipment. Please save damaged or pilfered cartons until claim is settled. In some instances, photographic documentation may be required. Some items are time-sensitive; WPI assumes no extended warranty or any liability for use beyond the date specified on the container

Do not return any goods to us without obtaining prior approval and instructions from our Returns Department. Goods returned (unauthorized) by collect freight may be refused. Goods accepted for restocking will be exchanged or credited to your WPI account. Goods returned which were ordered by customers in error are subject to a 25% restocking charge. Equipment which was built as a special order cannot be returned.

Repairs

Contact our Customer Service Department for assistance in the repair of apparatus. Do not return goods until instructions have been received. Returned items must be securely packed to prevent further damage in transit. The Customer is responsible for paying shipping expenses, including adequate insurance on all items returned for repairs. Identification of the item(s) by model number, name, as well as complete description of the difficulties experienced should be written on the repair purchase order and on a tag attached to the item.

** Electrodes, batteries and other consumable parts are warranted for 30 days only from the date on which the customer receives these items.*

USA

International Trade Center, 175 Sarasota Center Blvd., Sarasota FL 34240-9258
Tel: 941-371-1003 • Fax: 941-377-5428 • E-mail: sales@wpiinc.com

UK

1 Hunting Gate, Hitchin, Hertfordshire SG4 0TJ
Tel: 44 (0)1462 424700 • Fax: 44 (0)1462 424701 • E-mail: wpiuk@wpi-europe.com

Germany

Saarstraße 23, D-61169 Friedberg (Hesson), Germany
Tel: +49 (0)6031 1602171 • Fax: +49 (0)6031 1602180 • E-mail: wpide@wpi-europe.com

China & Hong Kong

WPI Shanghai Trading Co., Ltd.
Rm 18a, No8 Dongfang Rd., Pudong District, Shanghai, 200120 PR China
Tel: +86 21 6888 5517 • E-mail: chinasales@china.wpiinc.com

Brazil

Av. Conselheiro Nébias, 756 sala 2611, Santos-CEP: 11045-002, São Paulo Brazil Tel: (013) 406-29703 • E-mail: info@brazil.wpiinc.com

Internet

www.wpiinc.com • www.wpi-europe.com • www.wpiinc.cn • www.wpi brasil.com.br