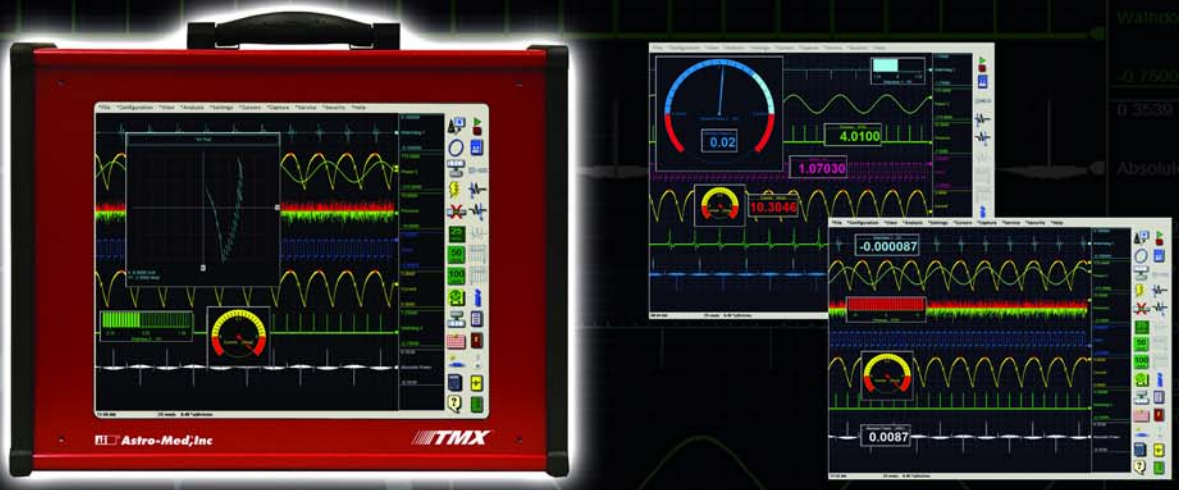
TMX™

Multi-Channel High Speed Data Acquisition System



Operations Manual

Astro-Med,® Inc
TEST & MEASUREMENT PRODUCT GROUP

Measurement has never been this easy

TMX
Operations Manual

Part Number: 22834578-EN-E
Manual Version 1.0

1/10

Astro-Med, Inc.
600 East Greenwich Avenue
West Warwick, RI 02893
401-828-4000

Technical Support 877-867-9783
techserv@astromed.com

Copyright © 2010 Astro-Med, Inc.

Trademarks

TMX and AstroVIEW X are trademarks of Astro-Med, Inc.

Microsoft and Microsoft Windows are trademarks of Microsoft Corporation.

Adobe and Adobe Acrobat are trademarks of Adobe Systems Inc.

Ethernet is a trademark of Xerox Corporation.

Symbols Displayed on this Product

Attention, refer to manual.

Off (system shutdown).

On (system connection to mains).

General Safety Information

Please review the following safety precautions to prevent personal injury or equipment damage.

- Service must be performed by qualified service personnel.
- The disconnecting device for the TMX is the AC power inlet connector. Note that the power system will be energized even when the power switch is switched off. To ensure that the power system is de-energized, the power cord must be removed from the unit.
- Use only the specified power cord and a proper outlet with protective earth ground connection.

- Never exceed the specified mains or signal input voltages specified in Appendix A.
- This equipment is designed for indoor use only. Never operate it in wet conditions, explosive atmospheres, or environments outside of the temperature and humidity specifications listed in Appendix A. Proper ventilation must be provided to keep this equipment within these specifications.
- Do not use the equipment if it has visible or detectable damage. Do not use the equipment if it has been exposed to stresses beyond the limits indicated in Appendix A.
- Using this product in a manner inconsistent with what is described in this manual may impair protections provided.

Limited Warranty

Astro-Med, Inc. warrants all portions of this hardware equipment against defects in materials or workmanship for a period of one year from the date of original purchase. If you discover a defect, Astro-Med will, at its option, repair or replace this product at no additional charge except as set forth below.

Repair parts and replacement parts will be furnished on an exchange basis and will be either reconditioned or new. All replaced parts become the property of Astro-Med. This warranty does not apply if the product has been damaged by accident, abuse, misuse, or misapplication or has been modified without the written permission of Astro-Med.

To obtain warranty services, contact Astro-Med at 401-828-4000. Astro-Med, Inc. is not responsible for your product if it is lost or damaged in transit. Astro-Med, Inc. makes no warranty either express or implied with respect to this product's fitness for a particular purpose.

Disclaimer of Warranty

This software and the accompanying files are sold "as is" and without warranties as to performance or merchantability or any other warranties whether expressed or implied. No warranty of fitness for a particular purpose is offered.

It is good practice to thoroughly test the software with non-critical data before relying on it. It is the customer's responsibility to follow proper file backup

procedures in the event of data loss. Astro-Med, Inc. is not responsible for any data loss experienced in operation of this software.

Astro-Med, Inc. does not warranty that the functions contained in the software will meet your requirements or that the operation of the software will be uninterrupted or error-free. The user must assume the entire risk of using the program. Any liability of the seller will be limited exclusively to product replacement or refund of purchase price.

Contact Astro-Med, Inc.

To contact the Astro-Med, Inc. Test & Measurement Group, send us an e-mail at mtgroup@astromed.com or call 401-828-4000.

Product Identification

The software provided with your equipment is the most current available. Record the model number, serial number, and software version installed on your equipment using the following spaces. Any upgrades to resident software should also be noted as they are installed.

If for any reason you need to contact Astro-Med, Inc. regarding your purchase, you will be asked to refer to this information.

Model Number:
Serial Number:
Original Software Version:

Upgraded Software Version:
Date Installed:

Upgraded Software Version:
Date Installed:

Upgraded Software Version:
Date Installed:

Contents

Chapter 1: Getting started

Introduction

The TMX is a versatile, high-frequency data acquisition recording system that provides the capability to display, record, and review waveform data.

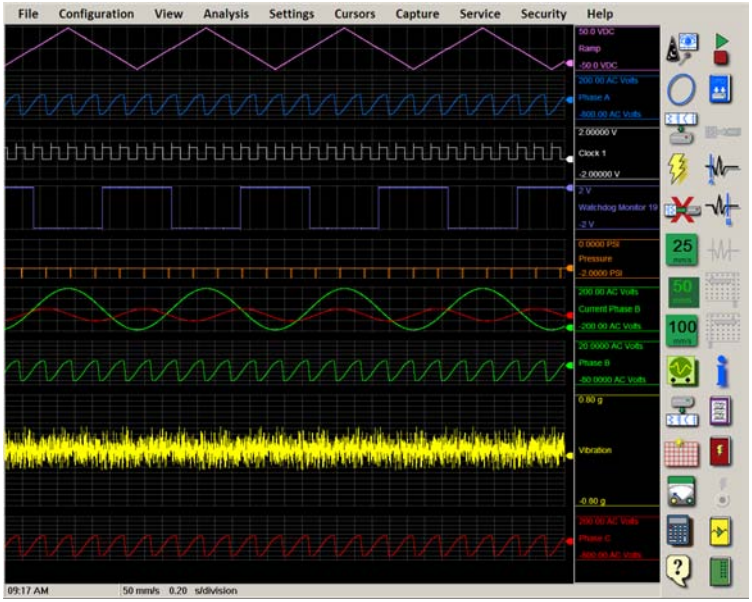
The TMX utilizes a touch-screen as the main user interface. Many of the control buttons are customizable, providing the capability to modify the display based on the needs of the user or application.

Modes of operation

The operating modes are Realtime, Scope, and Review modes.

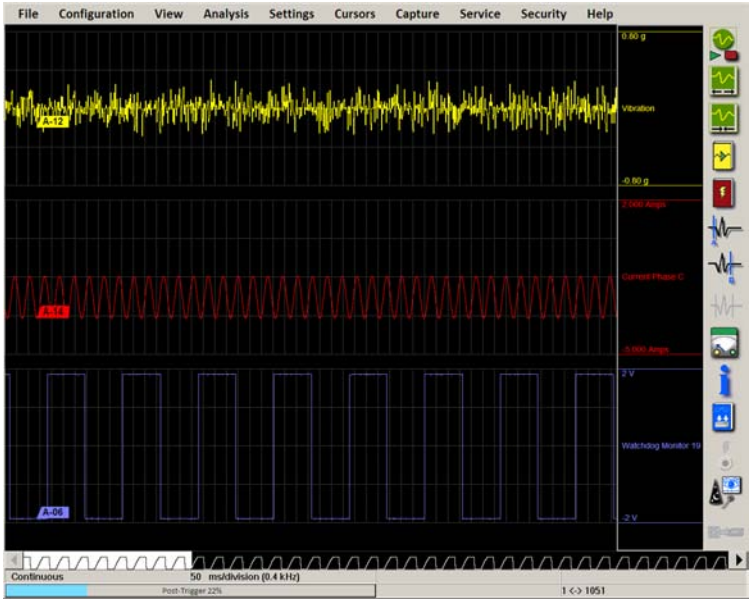
- **Realtime mode** - Realtime mode provides real-time waveform scrolling, monitoring, and data capture capabilities, typically used to view low frequency waveforms. Additionally, almost all system setup options are accessible from Realtime mode.

  Choose Configuration >> Realtime from the menu bar to access Realtime mode.



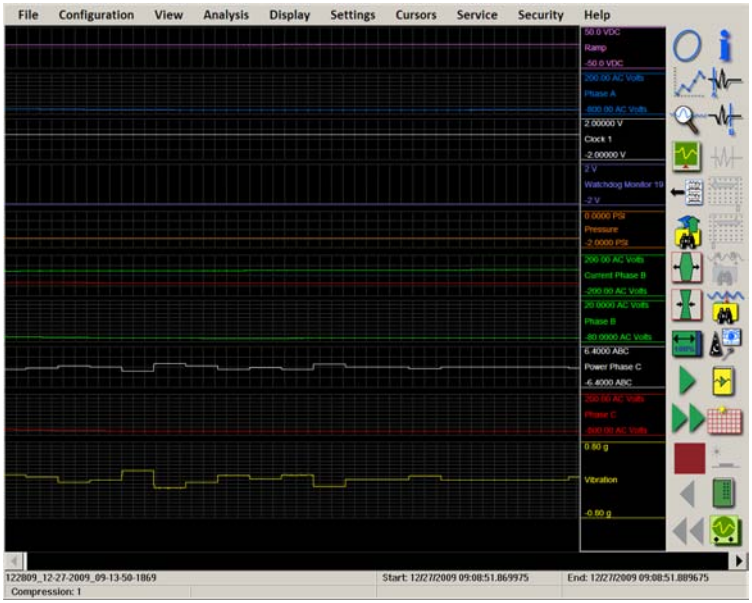
- **Scope mode** - Scope mode acts like a digital storage oscilloscope, providing high time-base resolution for viewing high-frequency signals. Scope mode is useful for timing and synchronization analysis, transient capture, and high-speed testing. It can be used while continuously capturing data and monitoring signals on the display.

  Choose Configuration >> Scope from the menu bar to access Scope mode.



- **Review mode** - Review mode provides the capability to review and analyze saved data capture and scope capture files. It also provides file management features.

  Choose Configuration >> Review from the menu bar. A file selection window will open. Select a file to review and choose OK.



Data capture introduction

There are two hard drives in the TMX. The system drive contains the Windows operating system of the recorder. The data capture drive is used to record streaming data.

During the data capture process, data is recorded directly to the data capture drive. When the data capture is complete, you can review the data file on the TMX using review mode.

If you want to export the data from the data capture drive, you can archive the data capture file to the system drive or directly to a USB storage device.

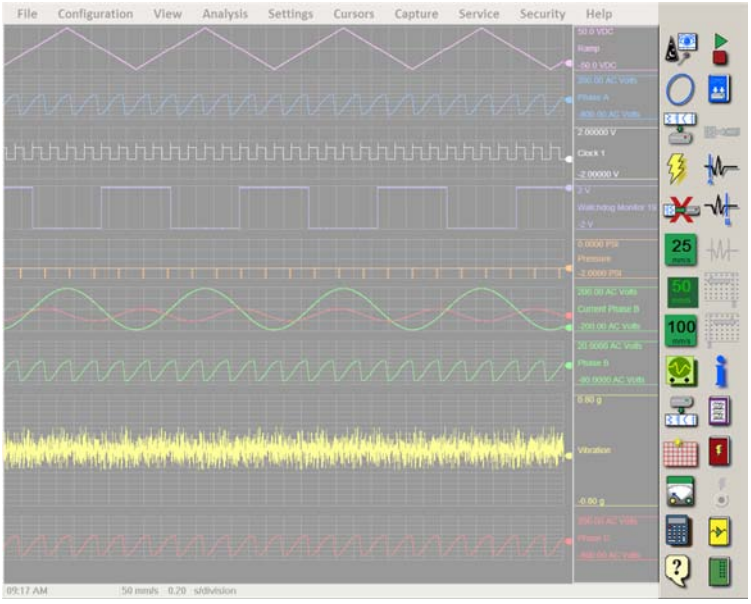
The TMX archive feature ensures data integrity and is the only method of exporting files from the data capture drive. The data capture drive is not accessible from Windows Explorer on the system drive.

The menu bar and control panel displayed on the touch-screen provide access to the controls and settings. Host control options are also available.

- **Menu bar** - The menu bar is a group of drop-down menus located across the top of the display. All modes and features can be accessed from this menu. Menu options will vary based on the mode of operation (Realtime, Scope, or Review) used.

Note: *The ">>" symbol in this manual indicates selections made using the menu bar. For example, "Configuration >> Realtime" indicates to choose Configuration, then Realtime from the menu bar.*

- **Control panel** - The control panel is a customizable group of icon buttons located on the right side of the display. It provides immediate access to virtually any function with one touch. Each mode of operation (Realtime, Scope, and Review) utilizes its own control panel.



- **Host control** - The recorder can be controlled remotely via a network connection and host commands. For more information, please contact Technical Support for the TMX Host Control Guide.

Help features

Context help, icon help, and the online manual are available to help you learn more about the recorder.

Using context help

The context help feature provides on-screen help based on the activity being performed in a window.

1. Choose the Context Help button from an open window.

   Context help instructions for the current task will appear.

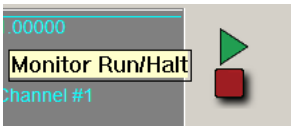
2. To close the help, choose Cancel in the help window.

Using icon help

The icon help feature provides brief on-screen descriptions for icons.

1. To use icon help in the main Realtime, Scope, or Review mode screens, choose Help >> Icon Help.

   Touch and hold a control panel icon. A short description of the icon will appear.



   To exit icon help, choose Help >> Icon Help again.

   Note: *Remember to deactivate the icon help function after using it, as icons will not perform their functions when icon help is active.*

2. To use icon help in an open window, choose the Icon Help button.

Touch and hold an icon in the window. A short description of the icon will appear.

To exit icon help, choose the Icon Help button again.

Note: *Remember to deactivate the icon help function after using it, as icons will not perform their functions when icon help is active.*

Viewing the operations manual PDF

The Adobe Portable Document Format (PDF) version of this manual is available for on-screen viewing.

1. Choose Help >> Operations Manual. Adobe® Acrobat® Reader will launch and the online version of this manual will be opened.

2. To exit the online manual, close Adobe Acrobat Reader.

Technical support

For additional assistance, contact Astro-Med, Inc. Technical Support via telephone toll-free at 1-877-867-9783 (U.S. and Canada only) or e-mail at techserv@astromed.com.

Please have the serial number of the unit available when contacting support.

Viewing the software version number

Use the About window to view the version number of the software currently installed on the system.

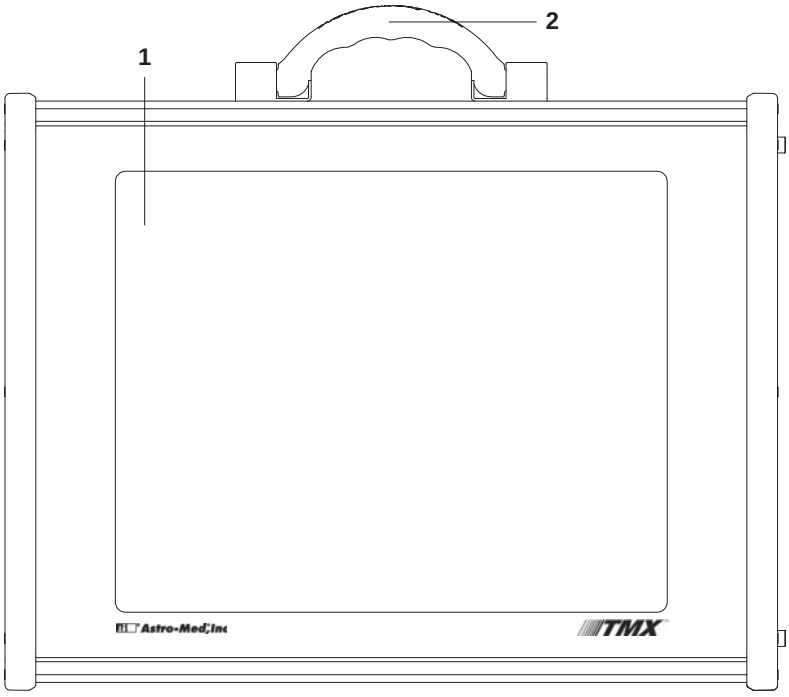
1. Choose Help >> About.

Chapter 2: Hardware overview

Hardware diagrams

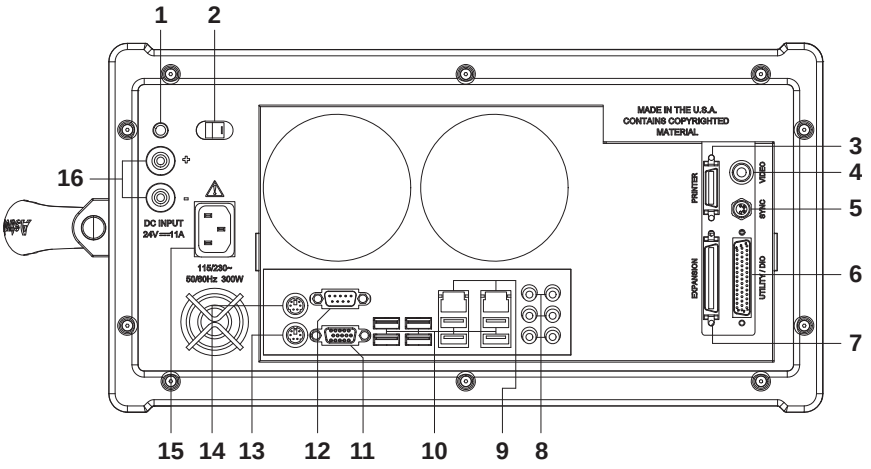
This section provides a visual overview of the hardware characteristics of the recorder. In addition to the diagrams, brief descriptions are included on various components.

Front view



#	Description
1	**Touch screen** The touch screen is the main user interface for the recorder. Touch the display with your fingertip or a stylus to select on-screen menus and buttons. ***Note:*** *To clean the touch-screen, dampen a soft cloth with window cleaner or water. Then gently clean the screen using the cloth. Cleaner should be sprayed on the cloth and not directly on the touch-screen. Spraying cleaner on the touch-screen could result in damage from liquid draining into the system.*
2	**Carry handle** Use the carry handle to move the recorder.

Left side view

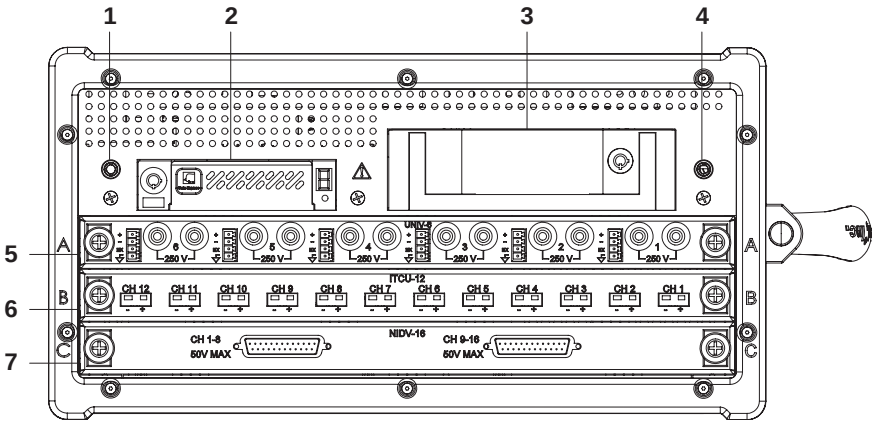


#	Description
1	**Power indicator light** This light visually indicates the power status of the recorder. When the recorder is operating properly with full power, the indicator will remain lit. During the power-down sequence, it will pulse for a few moments until the recorder turns off.

#	Description
2	**Power switch** Use the power switch to power up and power down the recorder. The power-up sequence requires less than one minute.
3	**Reserved (do not connect)** This port is reserved for factory use.
4	**Reserved (do not connect)** This port is reserved for factory use.
5	**Reserved (do not connect)** This port is reserved for factory use.
6	**Utility / D|O port** This port provides utility input/output functions and is used for event inputs.
7	**Reserved (do not connect)** This port is reserved for factory use.
8	**Audio ports** These ports provide a variety of audio outputs from the single-board computer within the unit.
9	**Ethernet ports (1000 Mbps)** The Ethernet port is used to provide network connectivity to the unit. Once the unit is properly connected to a network via Ethernet, file transfer and host control capabilities can be used. These 1000 Mbps ports are capable of automatically switching to 10/100 Mbps speeds.
10	**USB 2.0 ports** The USB 2.0 ports are used to connect USB accessories to the unit, such as a portable memory storage device, keyboard, or mouse.
11	**VGA port** The VGA port is used to connect the recorder to an external monitor. The monitor must be capable of displaying a 1024 x 768 screen resolution.
12	**Reserved (do not connect)** This port is reserved for factory use.

#	Description
13	**Keyboard port** The keyboard port is used to connect a standard keyboard to the unit. Once connected, the keyboard can be used as an alphanumeric input device.
14	**Mouse port** The mouse port is used to connect a standard mouse to the unit. Once connected, the mouse can be used as a pointing and selection device.
15	**Power inlet** The power inlet is used to attach the power cable to the unit.
16	**DC input** The DC input jacks are used to connect DC power (24 VDC) to the unit.

Right side view

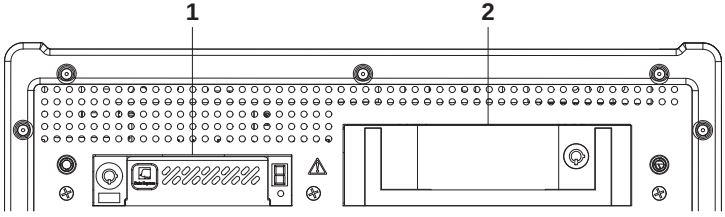


#	Description
1	**Ground connection** The ground connection is used to connect shields on input wiring, if needed.

#	Description
2	**System drive** The system drive contains the Windows operating system and can be used to archive files.
3	**Data capture drive** The data capture drive is used to record channel samples.
4	**Ground connection** The ground connection is used to connect shields on input wiring, if needed.
5	**Input module A slot** The input module installed in this slot is identified as module A in the software.
6	**Input module B slot** The input module installed in this slot is identified as module B in the software.
7	**Input module C slot** The input module installed in this slot is identified as module C in the software.

Drive module removal and installation

The system drive (1) and data capture drive (2) modules can be unlocked and removed from the recorder. This feature allows you to remove your data for storage in a secure location.



You can later reinstall the drive modules when needed. Both drive modules must be installed and locked or the recorder will not start up.

Removing drive modules

1. Turn the power off on the recorder.

Caution: *The hard drive modules are not hot-swappable, and should never be installed or removed while the recorder is powered on. Doing so may result in data loss or damage to the recorder.*

2. Insert the key into the lock and turn clockwise to unlock the drive bay.
3. Remove the drive.
   - To remove the system drive, press the eject button located below the lock to remove it from the bay.
   - To remove the data capture drive, pull the handle on the front of the drive to remove it from the bay.
4. Re-lock the drive by turning the drive key counter-clockwise.

Installing drive modules

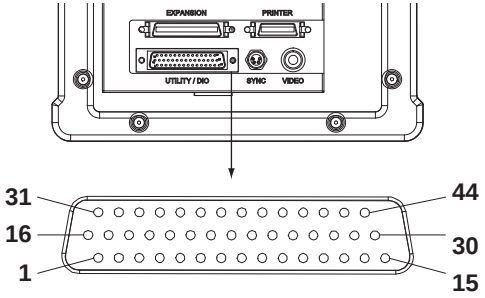
1. Turn the power off on the recorder.

   Caution: *The hard drive modules are not hot-swappable, and should never be installed or removed while the recorder is powered on. Doing so may result in data loss or damage to the recorder.*

2. Insert the key into the lock and turn clockwise to unlock the drive bay.
3. Slide the drive all the way into the drive bay.
4. Re-lock the drive by turning the drive key counter-clockwise.

Utility / D|O port pin configuration

The following diagram illustrates the utility / D|O port pin configuration. All inputs are TTL or switch closure compatible. Event signals have a high state (switch open) and a low state (switch closed).



Pin	Description
1	Event 1 input
2	Event 2 input
3	Event 3 input
4	Event 4 input
5	Event 5 input
6	Ground
7	Reserved (do not connect to this pin)
8	Reserved (do not connect to this pin)
9	Trigger output The trigger output pulses a TTL low for 100 usec in response to the recorder recognizing a trigger. This line can be connected to the external trigger input line of another TMX. In this case, the TMX receiving the external trigger input signal will trigger in response to the same event as the TMX sending the trigger output signal.
10	External abort input The external abort input line provides the capability to abort any currently running data capture. A TTL low for at least 100 ms will cause the recorder to abort any data capture in progress.
11	Reserved (do not connect to this pin)
12	Five volt (100 ohm resistor) This pin provides five volts through a 100 ohm resistor.
13	Ground
14	Reserved (do not connect to this pin)
15	Reserved (do not connect to this pin)
16	Event 6 input

Pin	Description
17	Event 7 input
18	Event 8 input
19	Event 9 input
20	Event 10 input
21	Event 11 input
22	Reserved (do not connect to this pin)
23	Reserved (do not connect to this pin)
24	Chart/page mark input The chart/page mark input provides the capability to place a full page mark across the display.
25	Alarm output The alarm output can be used to signal when alarm conditions for selected channels occur. This is an active low output. The signal is TTL low during alarm signals. This pin can sink 16 mA.
26	Print channel ID This input provides the capability to place channel IDs on the display. A logic level low or switch closure on this line for at least 100 ms will cause the selected item to be printed, provided the chart is running.
27	External run/halt input The external run/halt input provides the capability to start/stop the external chart printer. A logic level low or switch closure on this input will start/stop the printer.
28	External arm input The external arm input provides the capability to arm a data capture and/or scope capture using an outside signal. A logic level low or a switch closure on this line for at least 100 ms will cause a capture to be armed, provided there is storage available for the capture.

Pin	Description
29	Reserved (do not connect to this pin)
30	Reserved (do not connect to this pin)
31	Event 12 input
32	Event 13 input
33	Event 14 input
34	Event 15 input
35	Event 16 input
36	Reserved (do not connect to this pin)
37	Reserved (do not connect to this pin)
38	External trigger input The external trigger input provides the capability to invoke a trigger using an outside signal. A falling edge or switch closure on this line will cause an external trigger, if external triggers are enabled.
39	External sample rate input This pin provides the capability to enter a sample rate for data captures via an external signal. To use an external sample rate, external sample rate must be selected in the Capture Setup window. Sample clock must be between 10% - 90% duty cycle.
40	External motor clock input The external motor clock input provides the capability to use an external motor clock for controlling the Realtime chart speed. The external motor clock uses TTL voltage levels or switch closure inputs, and has a max input frequency of 680 Hz, which corresponds to 65 mm/s.

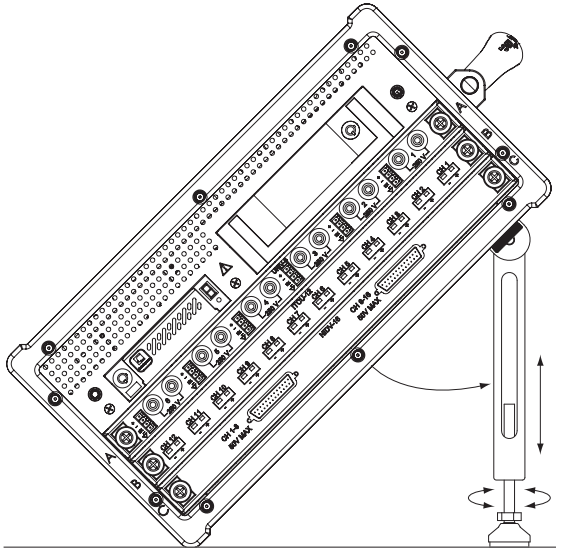
Pin	Description
41	Security input The security input pin provides the capability to disable the password protection security. A logic level low or switch closure on this line will disable passwords and allow full access to previously protected features.
42	Reserved (do not connect to this pin)
43	Reserved (do not connect to this pin)
44	Reserved (do not connect to this pin)

Using the stand

Use the adjustable stand to securely position the recorder at an angle.

1. Align the thumbscrews on the stand with the holes on the back of the recorder. Use the thumbscrews to securely fasten the stand to the recorder.

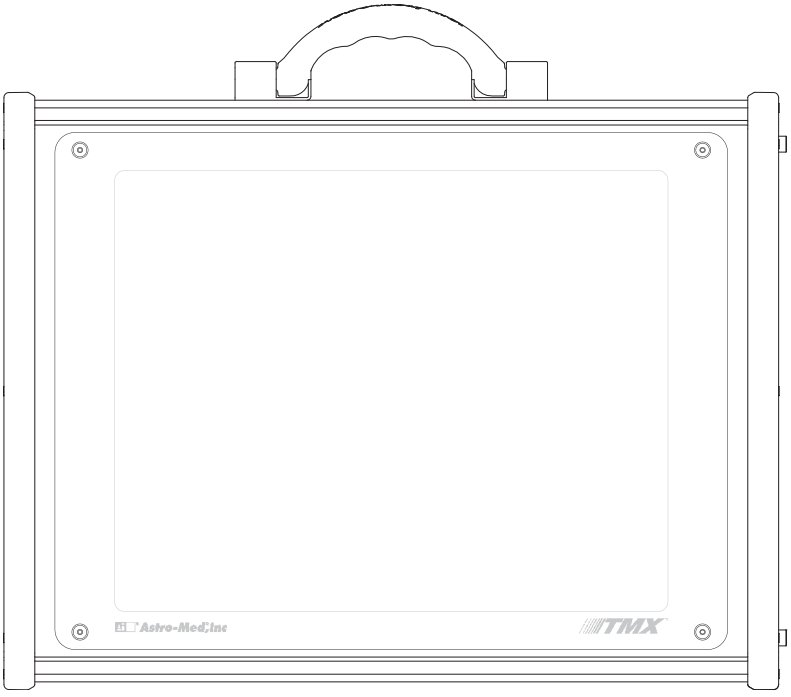
2. Open the stand fully and position the recorder as indicated in the following illustration. If necessary, you can adjust the stand height by rotating the leveling foot.



Using the display cover

If harsh conditions are part of your testing environment, you can protect the touch-screen with the transparent display cover.

1. Place the display cover over the touch-screen and align the thumbscrews with the holes on the front of the recorder.

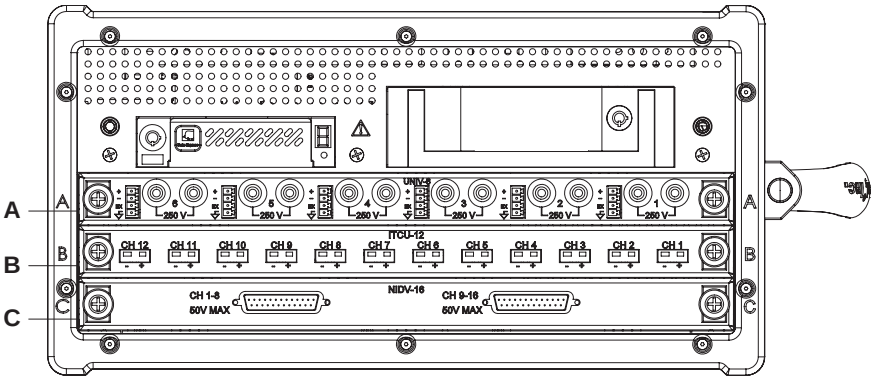


2. Use the thumbscrews to fasten the cover into position.

Chapter 3: Input modules

Input module locations

The system uses up to three modules for signal input connections. Input modules can be installed using the plug-in slots located on the side of the recorder. The slots are labeled A, B, and C for module identification in the software.

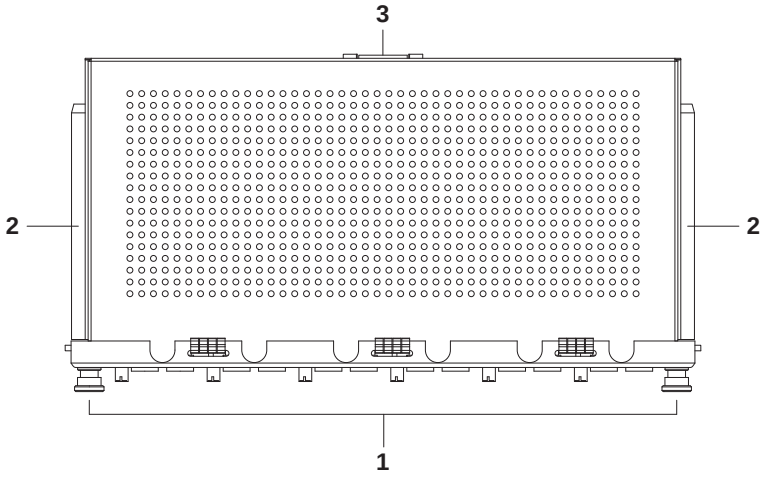


Note: *The input modules displayed in this diagram are used for illustration purposes. The appearance of input modules will vary based on the module types installed.*

Input module installation and removal

This section describes how to install and remove input modules. You will need a Phillips screwdriver to complete the module installation and removal tasks.

The following diagram illustrates the connection points on input modules.

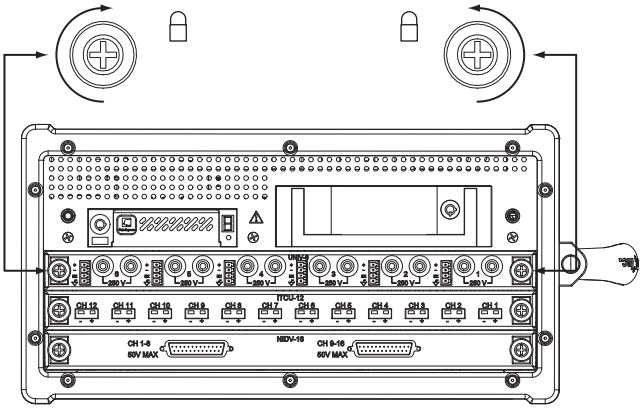


#	Description
1	**Locking knobs** The locking knobs are used to fasten the module into the recorder.
2	**Slot guides** The slot guides are used to align the module and guide it into position in the recorder.
3	**Connector** The connector is used to electronically connect the module to the recorder.

Installing input modules

1. Turn the power off on the recorder.
2. Position the recorder on its back with the touch-screen facing up.
3. Insert the input module into an empty module slot. Ensure the module slot guides are aligned with the corresponding slots in the recorder. Slide the module fully into the recorder.

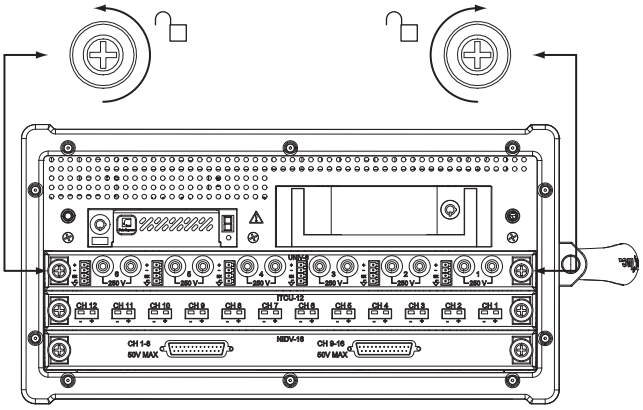
4. Using a Phillips screwdriver, lock the two knobs on the input module. Turn the left knob clockwise and the right knob counter clockwise.



5. Turn the recorder power on.

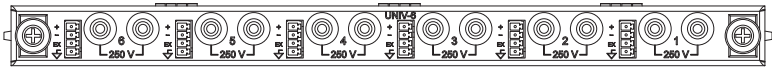
Removing input modules

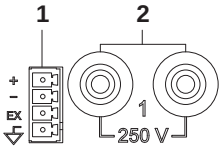
1. Turn the power off on the recorder.
2. Position the recorder on its back with the touch-screen facing up.
3. Using a Phillips screwdriver, unlock the two knobs on the input module you want to remove. Turn the left knob counter clockwise and the right knob clockwise.



4. Grasp the two knobs and slide the input module out of the recorder.

UNIV-6 - Universal voltage module with DC bridge



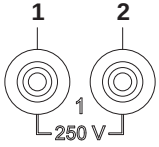


#	Description
1	Differential / DC bridge input
2	Single ended input

Warning: *You must only use one physical connection per channel at a time. Use either the 4-pin connector or the banana jack inputs.*

UNIV-6 Single ended inputs

Use the following diagram to connect to the single ended inputs.

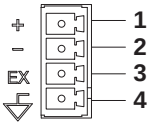


#	Description
1	Black (-) input
2	Red (+) input

Note: *The use of guarded banana test leads will eliminate the possibility of introducing a ground to the equipment under test if a regular banana test lead should come in contact with the recorder metal case.*

UNIV-6 Differential / DC bridge inputs

Use the following diagram to connect to the differential / DC bridge inputs.

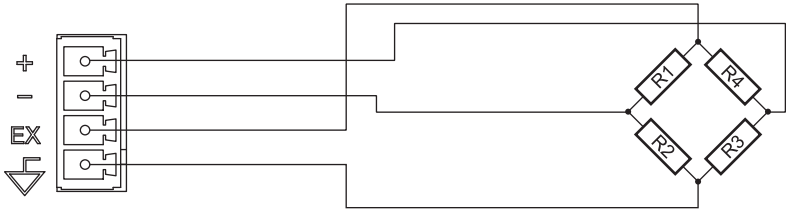


#	Description
1	Differential input (+)
2	Differential input (-)
3	Excitation output (0 to 10V DC, 30 mA max)
4	Excitation return (ISOCOM)

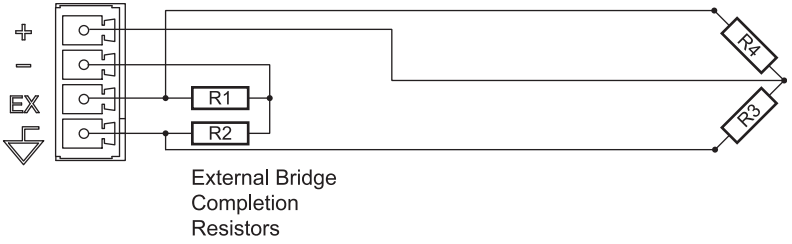
UNIV-6 DC bridge wiring diagrams

Use the following diagrams to set up DC bridge wiring on the four-pin connector on the UNIV-6.

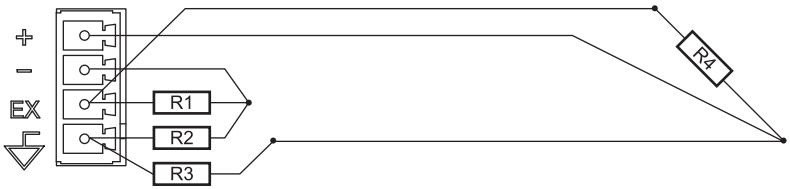
Four-wire full bridge



Three-wire half bridge

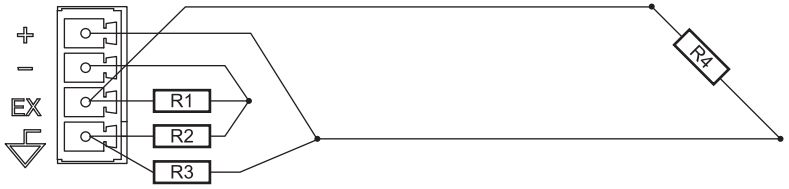


Three-wire quarter bridge



External Bridge
Completion
Resistors

Two-wire quarter bridge



External Bridge
Completion
Resistors

Chapter 4: Optional hardware

Installing a mouse and keyboard

The mouse and keyboard provide an additional method for controlling the recorder. The touch-screen will remain active, regardless of whether a mouse and keyboard are installed.

1. Turn the power off on the recorder.
2. Connect a mouse and/or keyboard to the recorder via the inputs on the side panel. The recorder provides built-in support for a variety of standard PS2 and USB mice and keyboards. Installing drivers is not necessary.
3. Turn the recorder power on.
4. Use the mouse and keyboard.
    - The mouse can be used as a pointing and selection device.
    - The keyboard can be used as an alphanumeric input device while a keypad or number pad is visible on the display.

USB memory devices

You can install a USB hard drive or flash drive using an available USB port on the recorder. USB memory devices can be used for a variety of purposes, including the following:

- Saving and loading setup files
- Saving archived data captures and scope captures
- Upgrading the system software

Installing the USB external hard drive (EHD-USB)

1. Turn the power off on the recorder.
2. Connect the EHD-USB to the recorder via the USB port.
3. Turn the EHD-USB power on.
4. Turn the recorder power on. The EHD-USB will be assigned a drive letter.

Installing the USB flash memory drive

1. Connect the drive to the recorder via the USB port.
2. The drive will be assigned a drive letter.

External monitor

An external monitor can be connected to the recorder via the VGA port. When installed, the monitor will display the contents of the recorder touch-screen.

Note: *Ensure that the monitor is capable of displaying a 1024 x 768 screen resolution.*

Installing an external monitor using the touch-screen

1. Turn the power off on the recorder. Then connect the external monitor to the VGA port located on the side of the recorder.
2. Turn the recorder power on.
3. Choose Configuration >> Utilities from the menu bar to open the utilities screen.

   From the Utilities screen, choose the Operating System button. The Enter Password window will open.

   Enter the system password and choose OK. The TMX software will shut down, providing access to the operating system.

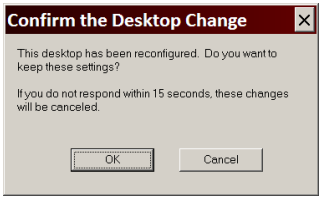
   Note: *The default system password is "tmx" (lowercase, without the quotation marks).*

4. Choose Start >> Control Panel to open the Control Panel window.

5. Double-click the Intel GMA Driver option. The Intel Graphics Media Accelerator Driver window will open.

In this window, “monitor” refers to the external display and “notebook” refers to the recorder touch-screen.

6. Choose the Intel Dual Display Clone option. Then choose Apply. The contents of the touch-screen will be displayed on the external monitor. A confirmation message will appear.



   Choose OK to confirm the change.

7. Choose OK in the Intel Graphics Media Accelerator Driver window. Then close the Control Panel window.

Related topics:

- *Changing passwords* on page 14-9

Installing an external monitor using a mouse

1. Turn the power off on the recorder. Then connect the external monitor to the VGA port located on the side of the recorder.

2. Connect a mouse to the mouse port on the side of the recorder.

3. Turn the recorder power on.

4. Choose Configuration >> Utilities from the menu bar to open the utilities screen.

   From the Utilities screen, choose the Operating System button. The Enter Password window will open.

   Enter the system password and choose OK. The TMX software will shut down, providing access to the operating system.

   Note: *The default system password is "tmx" (lowercase, without the quotation marks).*

5. Right-click the desktop. A menu will appear.

   Choose Graphics Options >> Output To >> Intel(R) Dual Display Clone >> Monitor + Notebook.

   In this menu, "monitor" refers to the external display and "notebook" refers to the recorder touch-screen.

Related topics:
- *Changing passwords* on page 14-9

Installing Windows-based printers

Windows-based printers are supported via USB port or network connection. Windows-based printers can be used to print screen captures.

Note: *Using a Windows-based printer may affect real-time displays, and some printers will perform better than others. Astro-Med will not guarantee full specifications while a Windows-based printer is printing.*

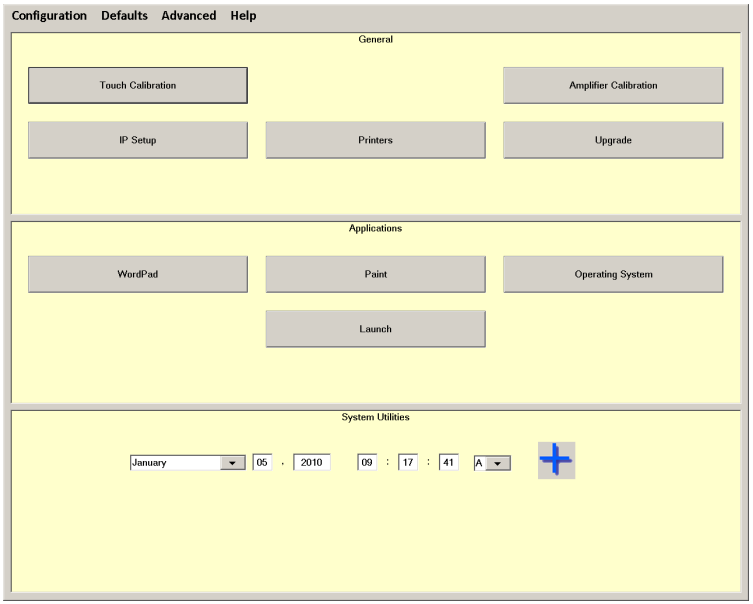
Many printer drivers are pre-installed as part of the operating system. If the necessary drivers are not pre-installed, they need to be made available to the recorder. A USB-based memory storage device can facilitate this process.

1. Turn the recorder power on.

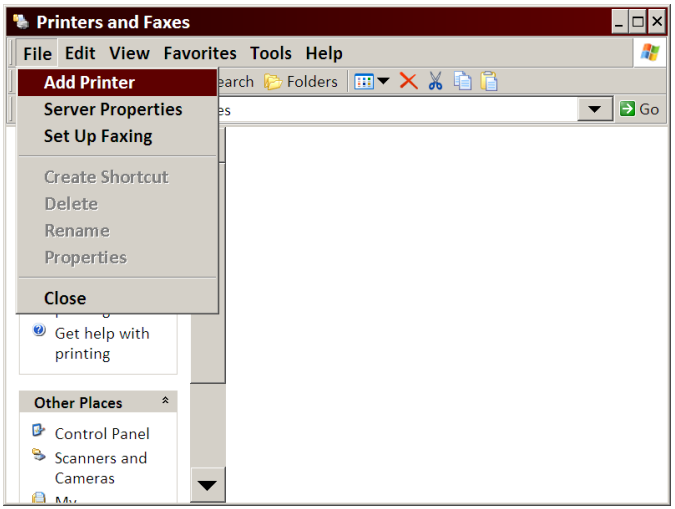
    - If you are installing a USB-based printer, connect the printer to the recorder using a USB cable.

    - If you are installing a network-based printer, connect the recorder to the network using a network cable.

2. Choose Configuration >> Utilities from the menu bar to open the utilities screen.



3. Choose the Printers button. The Printers and Faxes window will open. This window is a part of the Windows operating system.



4. Choose File >> Add Printer. The printer installation wizard will start.

5. Follow the printer wizard instructions to install the printer.

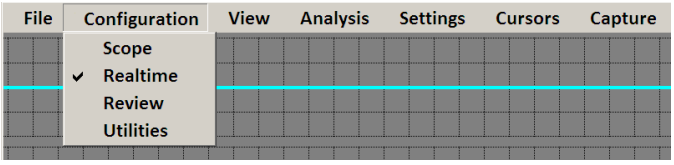
   If the operating system does not include the necessary printer driver, obtain the driver and copy it to a USB-based memory storage device. Then connect the storage device to the recorder via a USB port. When prompted by Windows to search for drivers, browse to the driver located on the USB storage device.

Chapter 5: Menus and icons

Menu bar

The menu bar is a group of drop-down menus located across the top of the display. All modes and features can be accessed from this menu. Menu options will vary based on the mode of operation (Realtime, Scope, or Review) used.

The following illustration displays the menu bar with the Configuration menu selected.



Locking the menu bar

The menu bar can be locked to prevent unauthorized users from accessing it. While locked, the menu bar cannot be used to access functions; the control panel must be used instead. The menu bar will remain locked, even if the recorder power is cycled, until it is unlocked by a user.

1. Choose Security >> Menu Lock. A keypad will appear. Enter the system password and choose OK. All menu options except Security and Help will be locked.

   Note: *The default system password is "tmx" (lowercase, without the quotation marks).*

2. To unlock the menu bar, choose Security >> Menu Lock. A keypad will appear. Enter the system password and choose OK. All menu options will be unlocked.

Related topics:

- *Changing passwords* on page 14-9

Common system icons

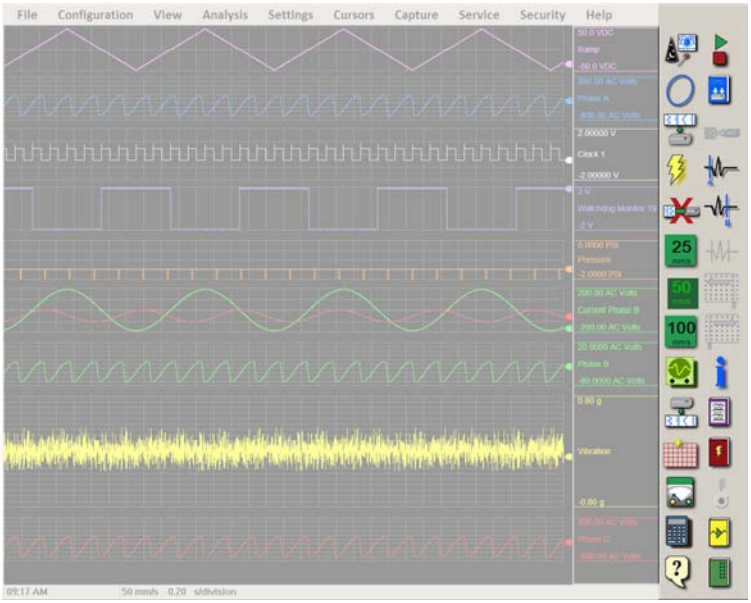
The following list describes the functions of common icons.

Icon	Description
	**OK** saves the information entered in a window and then closes the window.
	**Exit** cancels the action being performed in a window and closes the window without saving any changes.
	**Apply** saves the information modified in a window without closing the window.
	**Select All** selects/highlights all items in a list box.
	**Clear Selection** removes the selections/highlights from all items in a list box.

Control panel

The control panel is a customizable group of icon buttons located on the right side of the display. It provides immediate access to virtually any function with one touch. Each mode of operation (Realtime, Scope, and Review) utilizes its own control panel.

The following illustration displays the control panel.

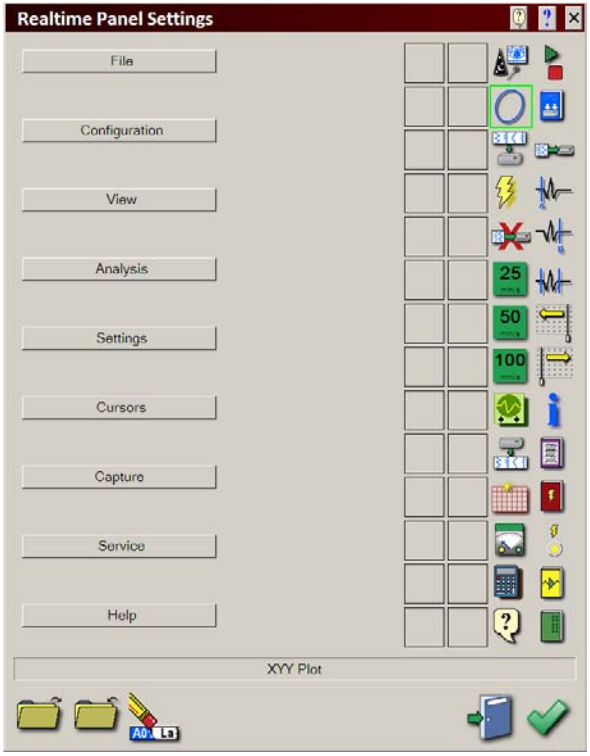


Most of the control panel buttons are one-touch equivalents of multi-step functions involving the menu bar and/or specific windows. Adding frequently used functions and removing unused functions can save time and effort while using the recorder.

For example, if the Channel Settings window is used frequently, it might be wise to add the associated icon button to the control panel. Instead of using the menu bar to choose Settings >> Channels, the function can be utilized by pressing one control panel button.

Customizing the control panel

1. Choose Settings >> Control Panel. The Panel Settings window will open.



   The buttons on the left provide functions that can be added to the control panel. Many of these functions correspond to menu bar options. The columns on the right display the layout of the control panel.

2. To add icons, select an empty control panel icon slot or existing icon. Then select a function from the buttons on the left.

   An icon for the chosen function will appear in the selected control panel location. If the location previously contained an icon, the icon will be replaced.

3. To remove icons, select an existing control panel icon. Then choose the Clear button.

You can also remove icons by pressing and dragging an empty icon over the icon you want to remove. Release it and the icon will be removed.

4. To arrange the icons, select a control panel icon. While pressing the icon, drag it to a different location in the control panel and release it. The icon will be moved to the new location. If the new location previously contained an icon, it will be replaced.

5. Choose OK.

Control panel setup files

Control panel setup files contain information about the icons used on the control panel. Once a control panel is set up for a particular application or user, the settings can be saved to a control panel setup file for later recall. These files can greatly decrease the amount of time spent customizing control panels before measurement.

It may be helpful to create a library of control panel setup files to accommodate multiple tasks and/or users. Additionally, control panel setup files are portable, so they can be shared with other TMX units.

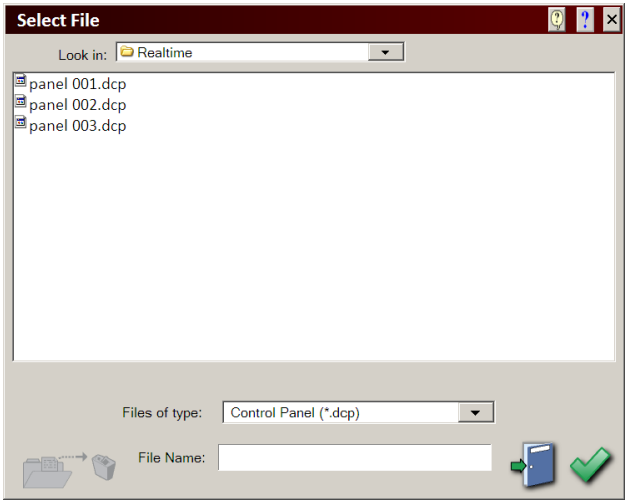
Saving control panel setup files

1. Open the Select File window using one of the following methods.

   - Choose File >> Save >> Control Panel.

   - Choose Settings >> Control Panel to open the Panel Settings window. Choose the Save Settings to File icon.

The Select File window will open.



2. Choose a destination for the control panel setup file. By default, the file will be saved on the system drive (C) in the selected folder. If necessary, you can save the file in a different location by choosing the folder list and browsing to a folder.

3. Choose the File Name field. A keypad will appear. Enter a file name and choose OK. The specified file name will appear in the field.

4. Choose OK.

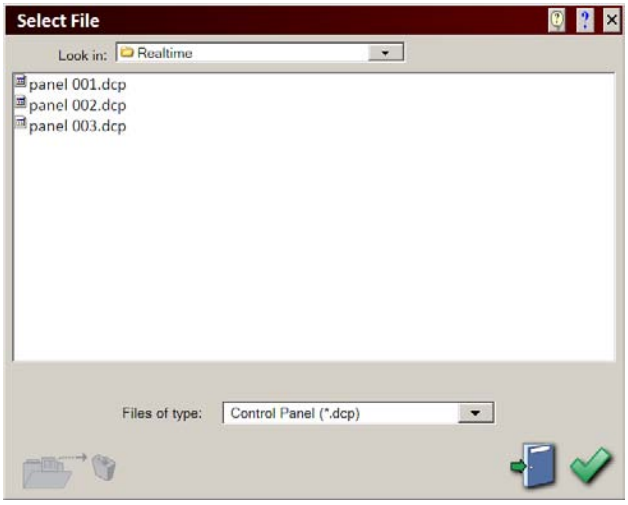
Loading control panel setup files

1. Open the Select File window using one of the following methods.

   - Choose File >> Load >> Control Panel.

   - Choose Settings >> Control Panel to open the Panel Settings window. Choose the Load Settings from File icon.

The Select File window will open.



2. Select a control panel setup file to load. By default, files in the default location on the system drive (C) will be displayed. If necessary, you can select a file from a different location by choosing the folder list and browsing to a folder. Choose OK.

Loading control panel setup files with the control panel

You can use control panel icons to quickly load control panel setup files.

1. Add the control panel setup file icon to the control panel.

   Choose Settings >> Control Panel to open the Panel Settings window.

   Choose the File button. A sub menu will appear. Choose Control Panel. The Select File window will open.

   Select a control panel setup file and choose OK. An icon for the selected file will be added to the control panel. Choose OK.

2. Load the control panel setup file from the control panel by choosing the appropriate control panel setup file icon.

Chapter 6: Channel setup

Channel setup concepts

The following concepts are commonly used during channel setup tasks.

Channels

Channels are single paths used for displaying waveforms. A waveform displayed in a channel can be generated directly from a signal input, or can result from filtering or math operations performed on a signal input.

Signals

Signals are voltage levels transmitted to the recorder from the measurement source. Signals enter the recorder via the input modules located on the side of the unit.

Channel labels

By default, waveforms are identified by labels corresponding to channel inputs. These labels can be modified to assign meaningful names to waveforms. For example, instead of using default labels:

- A-01: Channel #1
- A-02: Channel #2
- A-03: Channel #3

Descriptive labels can be assigned:

- A-01: Power
- A-02: Power (after filter)
- A-03: Pressure

Spans

The span indicates the size of the channel from top edge to bottom edge.

Bottoms/Centers

The bottom option indicates the bottom point of the channel span. The center option indicates the center point of the channel span.

Engineering units

Engineering units provide the capability to display user-selected units instead of voltage.

All signal information enters the recorder as voltage. However, converting the voltage unit to an alternative unit of measure may be desirable in applications that measure pressure, strain, or any other nonvoltage unit.

Note: *The relationship between the voltage and the engineering unit is assumed to be linear, characterized by a slope and offset (y = mx + b).*

After engineering units are defined and enabled, all appropriate menus will be displayed in the designated engineering unit values. For example, if pounds per square inch (PSI) are used as engineering units, the PSI label and value will be displayed instead of voltage.

Low and high alarm levels

Alarms provide a visual indicator when signals extend below or above specified boundaries. These boundaries are defined by setting up low and high alarm levels.

- **Low alarm level** - An alarm will occur when a signal is at or below the specified low alarm level. Portions of the waveform in the alarm area will be drawn in the selected alarm color.
- **High alarm level** - An alarm will occur when a signal is at or above the specified high alarm level. Portions of the waveform in the alarm area will be drawn in the selected alarm color.

The utility / D|O port provides an alarm output pin that can be used to signal when alarm conditions for selected signals occur.

Filters

Filters limit waveform amplitude based on frequency and thereby restrict or allow the waveforms to pass. Filters can be set up to filter out specific ranges of frequencies while allowing others to pass through for measurement.

There are four aspects to the filtering system.

Anti-aliasing filter – Limits input frequencies and prevents aliasing	
Implementation	Hardware
Type	Bessel lowpass
Cutoff (-3dB)	20 KHz

Order	4
Roll-off	-80 dB/decade

Fixed lowpass filter – Limits high-frequency noise	
Implementation	Digital signal processing
Topology	IIR Biquad
Type	Bessel lowpass
Cutoff (-3dB)	22.3 KHz
Order	2
Roll-off	-40 dB/decade

Built-in user-selectable filter – User-selectable filtering	
Implementation	Digital signal processing
Topology	IIR Biquad
Cutoff (-3dB)	User selectable
Type	Bessel Lowpass, Highpass, Bandpass, Notch
Order	2
Roll-off	-40 dB/decade

Attenuators

Attenuators limit the maximum signal input to the recorder. When the attenuator ranges are unlocked, the software will select the best attenuator setting. When the attenuator ranges are locked, possible span values will be limited based on the attenuator setting.

Grounding

A ground is generally considered an equipotential point or structure designed so the voltage between any two points is zero. In practice, there are no perfect grounds and all should be considered suspect.

Grounds may be used to:

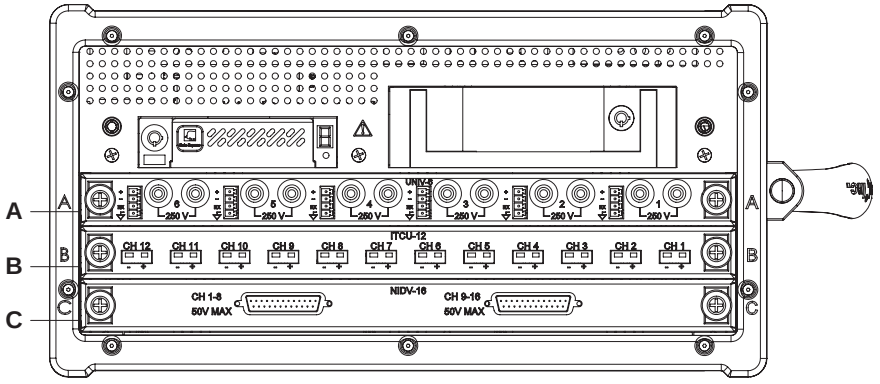
- Provide a safe return for excess current under fault conditions.
- Shield components from external sources.
- Provide a reference for voltage measurements. Poor grounding is a common cause of measurement errors.

A review of one of the many texts on the subject of grounding is advised. The following definitions are used in this document:

- **Earth Ground** is a low-impedance path to earth. In a properly installed 120-VAC outlet, the green wire is assumed to be earth ground.
- **Case Ground** refers to grounding achieved using an instrument's metal enclosure or frame. When the instrument is powered by line voltage, case ground is usually connected to earth ground for safety purposes.
- **Signal Ground** is an analog reference point for the measuring device.
- **Source Common** is a reference point at the voltage source.

Signal inputs

The term input refers to a signal connected to a physical input on the side of the recorder. The following diagram illustrates the layout of signal inputs.



Note: *The input modules displayed in this diagram are used for illustration purposes. The appearance of input modules will vary based on the module types installed.*

In the software, each signal input has a channel label for identification purposes. The input label corresponds to the physical location of the input. For example, "A-01: Channel #1" is the default input label of the signal connected to channel 1 of input module A.

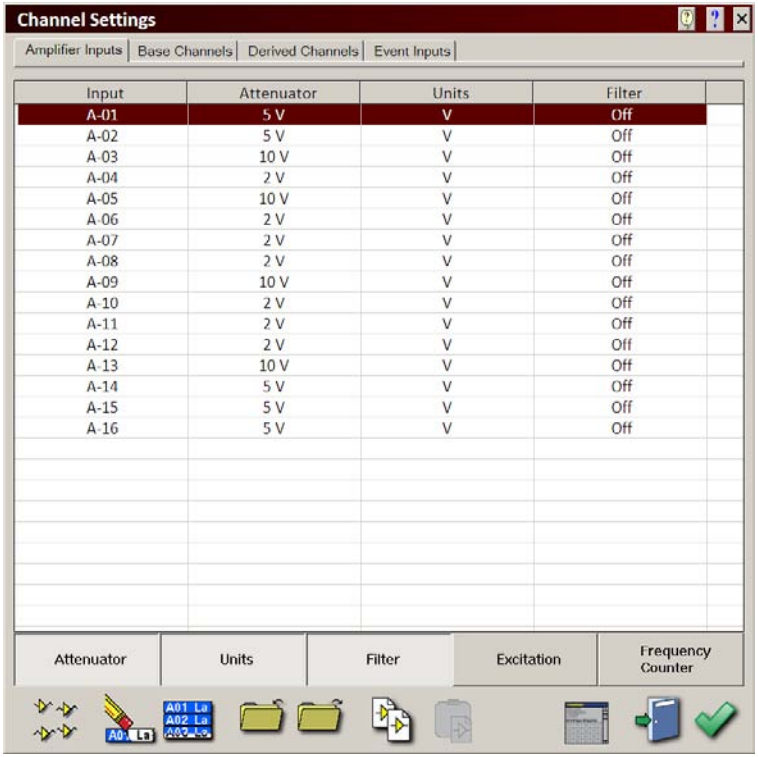
Channel Settings window overview

The Channel Settings window (Settings >> Channels) is used to set up channels and events. The Channel Settings window provides two layout options.

- **List view** - You can change settings in list view by selecting a channel and choosing the column heading for the setting you want to change. Some options can be configured for a group of channels at the same time by

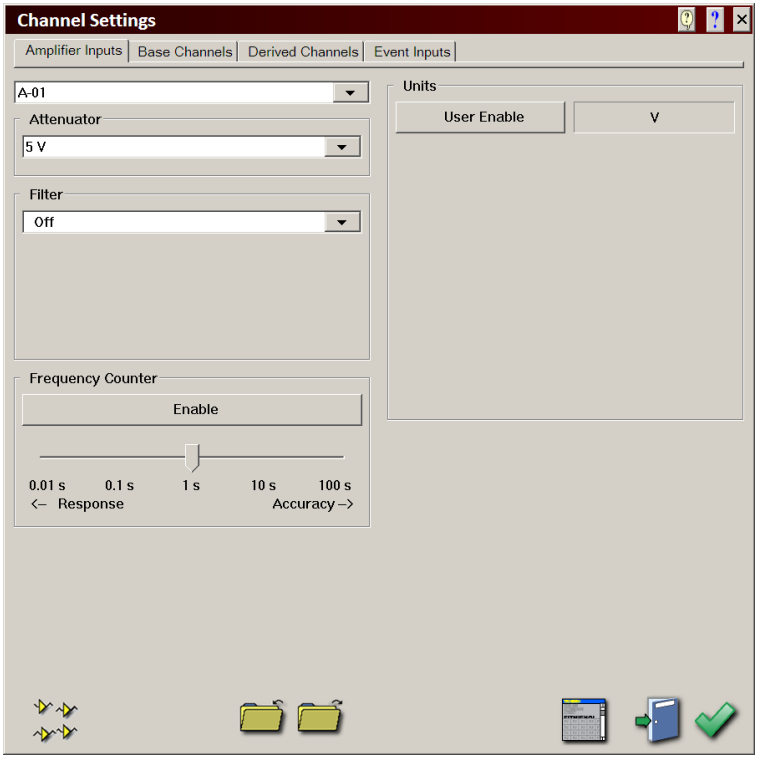
selecting a group of channels prior to choosing a column heading. In this case, setup information will be applied to all selected channels.

Some options must be configured for each channel individually, or in groups based on compatible channels.



Use the buttons below the list to display other columns.

- **Graphical view** - You can change settings in graphical view by selecting a single channel and the appropriate options for the channel.

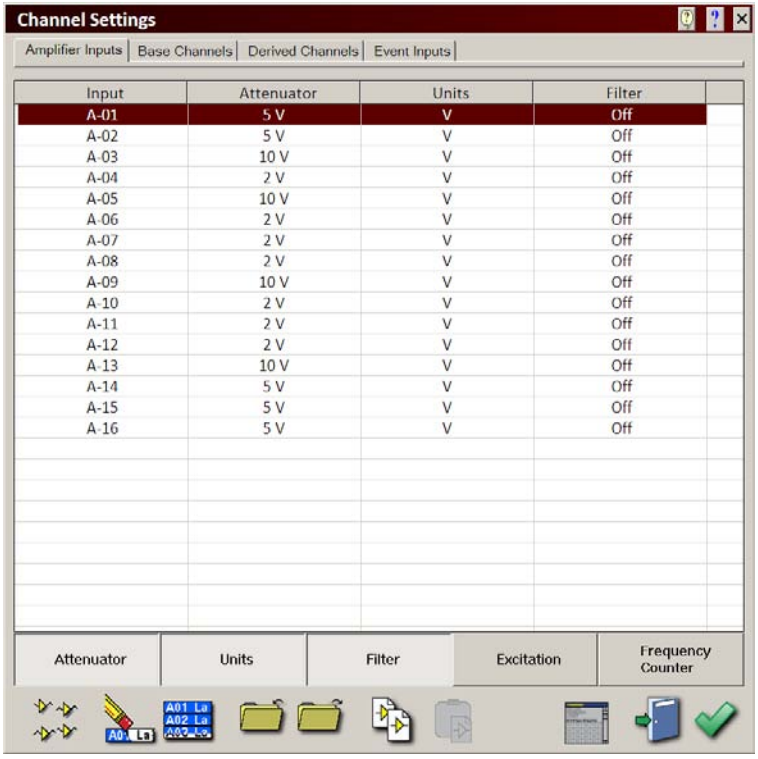


You can switch between list view and graphical view by choosing the Toggle Layout icon.

Setting up amplifier inputs

1. Choose Settings >> Channels. The Channel Settings window will open.

2. Choose the Amplifier Inputs tab.



3. Select an input. Inputs are labeled by module and input number. For example, “A-02” indicates input number two in input module A.

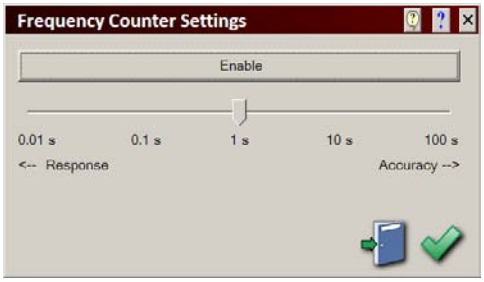
4. Choose the Attenuator column heading to specify the attenuator type for the input. The attenuator type refers to how the signal is physically connected to the recorder. Attenuator types vary based on input modules. The following attenuator types are available.

   - Single-ended (1, 10, 50, 200, 400 V full scale)
   - Differential (50, 500, 1000 mV full scale)
   - Bridge (50, 500, 1000 mV full scale)

   The attenuator setting limits the input signal to the selected maximum voltage.

5. If necessary, enable a filter for the input.

6. If you are setting up the first input of the input module, the frequency counter option will be available. Choose the Frequency Counter column heading. The Frequency Counter Settings window will open.



To use the current channel as a frequency counter, choose the Enable button. Use the slider to configure the counter. Move the slider to the left to increase response, and to the right to increase accuracy.

The frequency counter detects when signals cross the frequency counter reference threshold, which is located just above zero.

The frequency counter is effective only when measuring unipolar signals (0 to 5V for example) or bipolar signals (-5 to 5V for example). These signal types cross the frequency counter reference threshold, which is located just above zero.

The frequency counter cannot be used to measure signals that do not cross the reference threshold (5 - 50V or -20 to -5V for example).

7. If necessary, set up user engineering units.

8. If you are using a bridge input and attenuator, choose the Excitation column heading to select an excitation voltage for the input. This process is necessary only for bridge inputs.

9. Use the Attenuator Lock icon to specify whether to lock the attenuator ranges. When this button is pressed, the icon will change to indicate its current state.

   - This icon indicates that the attenuator ranges **are not** locked.

- This icon indicates that the attenuator ranges **are** locked.

When the attenuator ranges are unlocked, the software will select the best attenuator setting. When the attenuator ranges are locked, possible span values will be limited based on the attenuator setting.

10. Choose OK.

Related topics:
- *Setting up amplifier input filters* on page 6-10
- *Engineering units* on page 6-19

Setting up amplifier input filters

Filters are used to reduce the amplitude of certain waveforms based on their frequency. This allows you to restrict your data to certain frequencies and exclude extraneous data. Two types of filters are available.

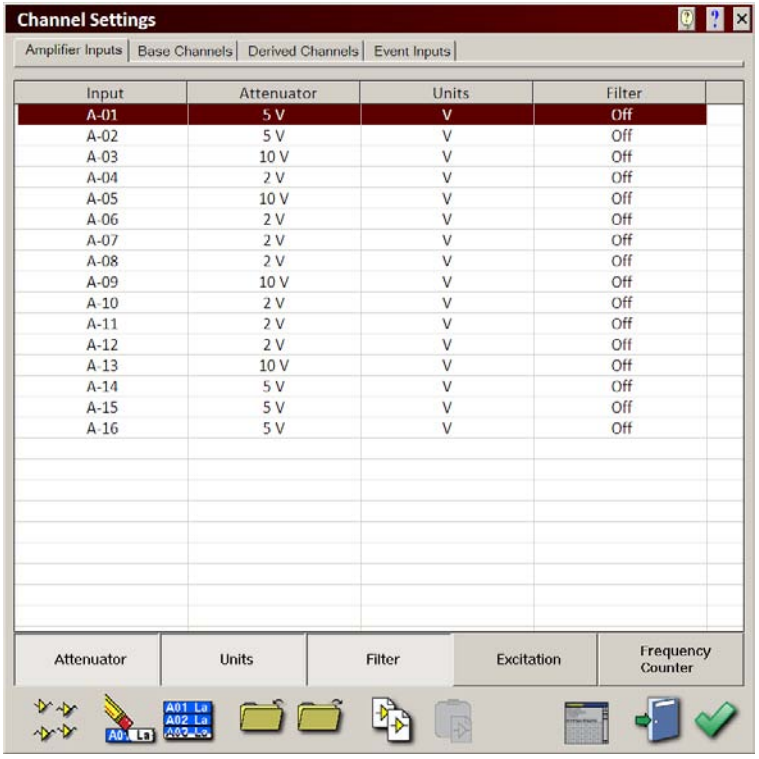
- **Amplifier input filters** are hardware based and affect the input data before it is captured. These filters are permanent and cannot be undone once the data is captured.

- **Review mode filters** are software based and affect how the data is viewed in Review mode. They temporarily modify the view without altering the captured data.

This section describes how to set up amplifier input filters.

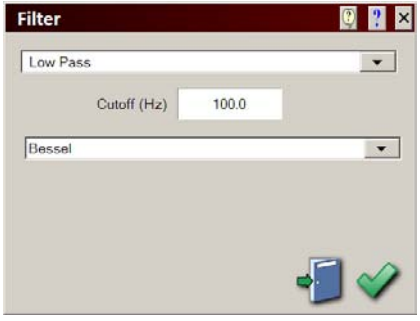
1. Choose Settings >> Channels. The Channel Settings window will open.

2. Choose the Amplifier Inputs tab.



3. Select an input. Inputs are labeled by module and input number. For example, "A-02" indicates input number two in input module A.

4. Choose the Filter column heading. The Filter window will open.



Select a filter type.

- **Low Pass** - Filters out frequencies above a specified cutoff point.
- **High Pass** - Filters out frequencies below a specified cutoff point.
- **Band Pass** - Creates a window based on two specified cutoff points, identified as upper and lower frequency boundaries. Anything outside this window is excluded.
- **Band Stop** - Creates a window based on two specified cutoff points, identified as upper and lower frequency boundaries. Anything inside this window is excluded.
- **RMS** - Filters the signal to its root-mean-square value. Because the RMS value is a measurement taken over some time period, a variety of response times are available to optimize the calculation for a particular input.

  The RMS calculation is performed as follows. First the input signal is squared. The squared signal is then sent into a first order low pass filter with a cutoff frequency that corresponds to the selected response time. The square root of the filter's output is then taken to finalize the RMS calculation.

  The response times are denoted by their "10-90" rise time. This is the time it takes for a square input to rise from 10% of its final value to 90% of its final value. Faster response times result in changes taking effect more quickly, but at the expense of more ripple.

  Note: *All filters use the 800 kHz sampling Rate.*

Response Time	Cutoff Frequency
2 Seconds	0.12 Hz
.2 Seconds	1.2 Hz
.02 Seconds	12 Hz
.002 Seconds	120 Hz

Once a filter type (other than RMS) is selected, filter setup options will appear.

5. Select a filter topology.

- **Bessel** filters are typically characterized by a nearly linear phase response in the pass band. They are commonly used in applications where little phase distortion is required. The trade off for this is a gentler roll off around the cutoff frequency.

- **Butterworth** filters produce the most "ideal" response, generating maximum flatness and unity gain in the pass band, and monotonic decrease of frequency after the cutoff.

- **Chebyshev** filters feature the sharpest transition band, but will have gain ripple in the pass band. A 4th order Chebyshev with 3 db of ripple, for instance, will drop at 100 db per decade.

  If the Chebyshev topology is used, the Ripple field will appear. Enter a desired ripple value from 0.1 to 10.0 into this field.

6. Enter a cutoff frequency. The available range of this value will vary, depending on the current filter type. If the Band Pass or Band Stop filter types are used, you'll need to enter two cutoff frequencies; an upper boundary and a lower boundary.

   Setting a cutoff value to 500 Hz or greater will result in a 4th order filter; a cutoff of less than 500 Hz will result in a 1st order filter.

7. Choose OK in the Filter window.

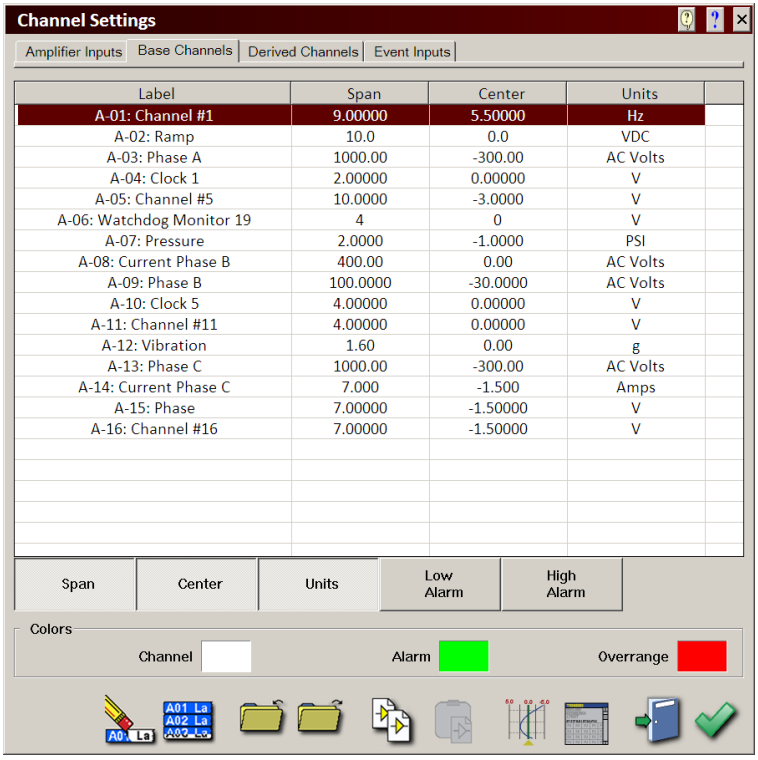
8. Choose OK.

Related topics:
- *Setting up Review mode filters* on page 11-18

Setting up base channels

1. Choose Settings >> Channels. The Channel Settings window will open.

2. Choose the Base Channels tab.



3. Select an input. Inputs are labeled by module and input number. For example, “A-02” indicates input number two in input module A. The channel label is also visible in this list.

4. To edit the channel label, choose the Label column heading.

   A keypad will appear. Enter a label for the channel and choose OK. The input module prefix, “A-02” for example, cannot be modified.

5. Choose whether to define the channel in terms of top/bottom or span/center. Use the following icon to switch methods.

   - **Top/Bottom** - If you select this method, choose the Top and Bottom column headings to enter the highest and lowest channel values.

- **Span/Center** - If you select this method, choose the Span and Center column headings to enter the total span of the channel and the center value.

6. To use low and/or high alarms, choose the Low Alarm or High Alarm column heading. The Alarms window will open.

   Choose the Low Alarm and/or High Alarm buttons. The selected alarms will become active and the alarm fields will be displayed. Use the alarm fields to enter alarm levels. Choose OK in the Alarms window.

7. Select a color scheme for the channel waveform.

   - **Channel** - This option sets the color of the waveform on the display. To modify the waveform color, choose the Channel color box. The Color window will open. Select a color and choose OK.

   - **Alarm** - This option sets the color of waveform portions that extend above or below the high and low alarm boundaries. To modify the alarm color, choose the Alarm color box. The Color window will open. Select a color and choose OK.

   - **Overrange** - This option sets the color of waveform portions that extend above or below the grid boundaries. To modify the overrange color, choose the Overrange color box. The Color window will open. Select a color and choose OK.

8. If necessary, set up user engineering units.

9. Choose OK.

Related topics:

- *Setting up engineering units* on page 6-19

Derived channels

Derived channels allow you to define flows of data that are not represented by a physical input, but instead result from operations performed on data from physical inputs. Once created, derived channels can be added to any grid on the display.

Derived channel syntax

Equations are used to define the content of derived channels. The equations indicate channels and the operations to perform on the channels. The following table describes the syntax of derived channel equations.

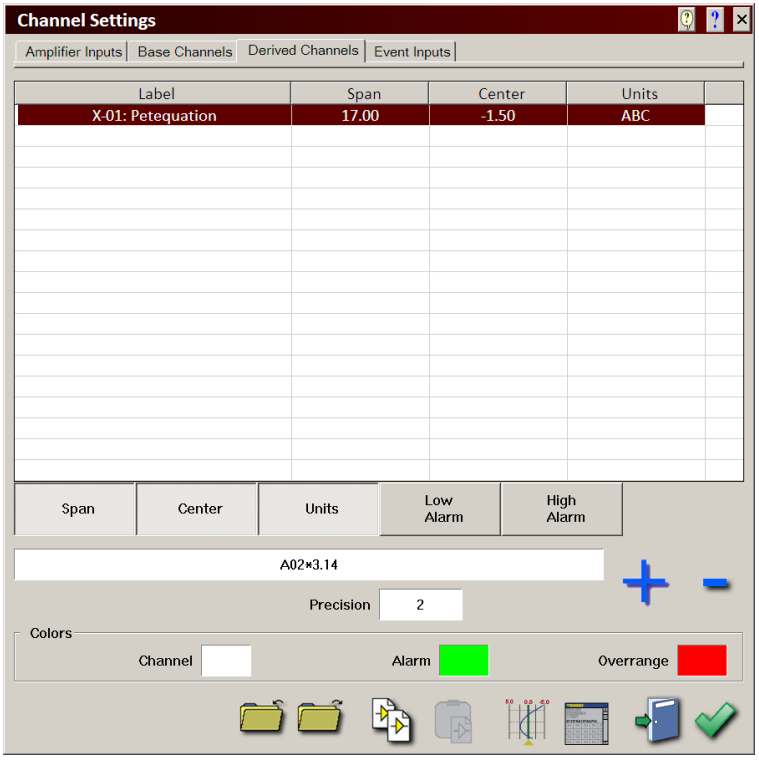
Function	Parameters	Example
Addition	Channel1, Channel2	A01 + A02
Subtraction	Channel1, Channel2	A01 - A02
Multiplication	Channel1, Channel2	A01 * A02
Division	Channel1, Channel2	A01 / A02

Parentheses must be used to denote the order of operations. Nested parentheses are allowed.

Setting up derived channels

1. Choose Settings >> Channels. The Channel Settings window will open.

2. Choose the Derived Channels tab.



3. To add a derived channel, choose the Add icon.

A keypad will appear. Enter a label for the derived channel and choose OK. Another keypad will appear. Enter the derived channel's equation and choose OK. The derived channel will be added. If necessary, you can later edit the equation by selecting the equation field.

Note: *You can remove a derived channel by selecting it from the derived channel list and choosing the Remove icon.*

4. Choose whether to define the channel in terms of top/bottom or span/center. Use the following icon to switch methods.

- **Top/Bottom** - If you select this method, choose the Top and Bottom column headings to enter the highest and lowest channel values.
- **Span/Center** - If you select this method, choose the Span and Center column headings to enter the total span of the channel and the center value.

5. To use low and/or high alarms, choose the Low Alarm or High Alarm column heading. The Alarms window will open.

Choose the Low Alarm and/or High Alarm buttons. The selected alarms will become active and the alarm fields will be displayed. Use the alarm fields to enter alarm levels. Choose OK in the Alarms window.

6. Select a color scheme for the channel waveform.

- **Channel** - This option sets the color of the waveform on the display. To modify the waveform color, choose the Channel color box. The Color window will open. Select a color and choose OK.
- **Alarm** - This option sets the color of waveform portions that extend above or below the high and low alarm boundaries. To modify the alarm color, choose the Alarm color box. The Color window will open. Select a color and choose OK.
- **Overrange** - This option sets the color of waveform portions that extend above or below the grid boundaries. To modify the overrange color, choose the Overrange color box. The Color window will open. Select a color and choose OK.

7. Choose the Units column heading to enter a name (display label) for the derived channel units.

8. Choose the Precision field and enter the number of decimal places for the derived channel unit. Choose OK.

9. Choose OK.

Engineering units

Engineering units provide the capability to display user-selected units instead of voltage.

All signal information enters the recorder as voltage. However, converting the voltage unit to an alternative unit of measure may be desirable in applications that measure pressure, strain, or any other nonvoltage unit.

Note: *The relationship between the voltage and the engineering unit is assumed to be linear, characterized by a slope and offset (y = mx + b).*

After engineering units are defined and enabled, all appropriate menus will be displayed in the designated engineering unit values. For example, if pounds per square inch (PSI) are used as engineering units, the PSI label and value will be displayed instead of voltage.

Setting up engineering units

1. Choose Settings >> Channels. The Channel Settings window will open.

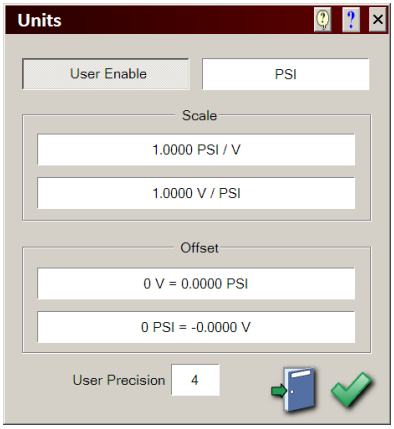
2. Engineering units can be configured in the Amplifier Inputs and Base Channels tabs. The tab you choose depends on whether derived channels will be used.

    - **Amplifier Inputs tab** - Configure engineering units in this tab only if the channel will be used in the definition of a derived channel in the Derived Channels tab.

    - **Base Channels tab** - Configure engineering units in this tab if the channel will not be used in the definition of a derived channel.

3. Select an input. Inputs are labeled by module and input number. For example, “A-02” indicates input number two in input module A.

4. Choose the Units column heading. The Units window will open. Choose the User Enable button to activate engineering units. Other engineering units options will be displayed.



5. Choose the Engineering Units field and enter a name (display label) for the units. For example, PSI would be an appropriate label denoting pounds per square inch.

6. Enter a scale for the engineering units by choosing one of the Scale fields. A number pad will appear. Enter the scale and choose OK. Scale can be specified as either of the following:

   - The waveform change in engineering units that is equal to one voltage unit.
   - The waveform change in voltage units that is equal to one engineering unit.

   Only one scale entry is required; the other is derived automatically.

7. Enter an offset for the engineering units by choosing one of the Offset fields. A number pad will appear. Enter the offset and choose OK. Offset can be specified as either of the following:

   - The number of engineering units equivalent to zero voltage units.
   - The number of voltage units equivalent to zero engineering units.

   Only one offset entry is required; the other is derived automatically.

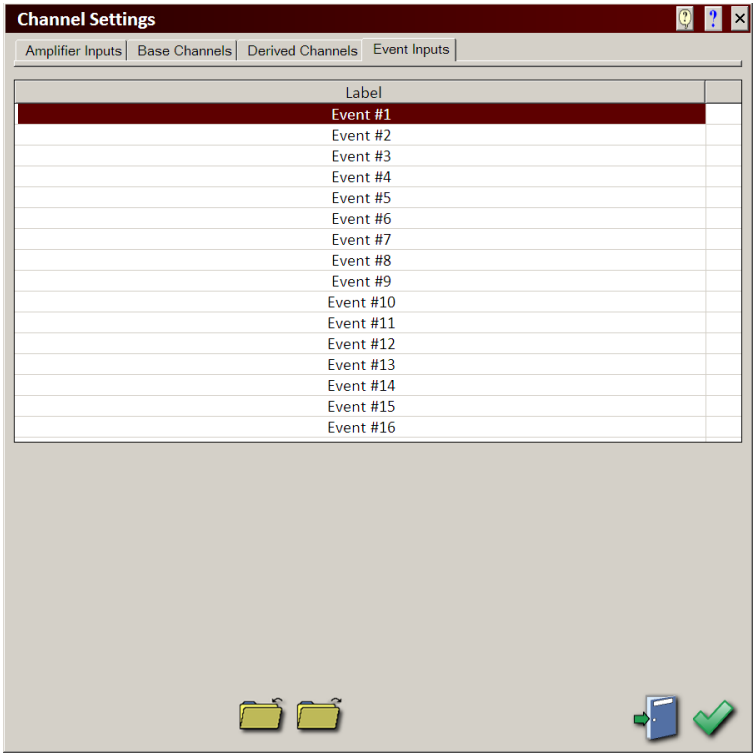
8. Choose the User Precision field and enter the number of decimal places for the engineering unit. Choose OK. Choose OK in the Units window.

9. Choose OK in the Channel Settings window.

Setting up event inputs

1. Choose Settings >> Channels. The Channel Settings window will open.

2. Choose the Event Inputs tab.



By default, events are labeled based on event input numbers.

3. To change an event label, select an event and choose the Label column heading. A keypad will appear. Enter an event label and choose OK.

4. Choose OK in the Channel Settings window.

Signal setup files

Signal setup files contain all setup information from the Channel Settings window. Once channels are set up for a particular application, the settings can be saved to a signal setup file for later recall. These files can greatly decrease the amount of time spent on setting up channels before measurement.

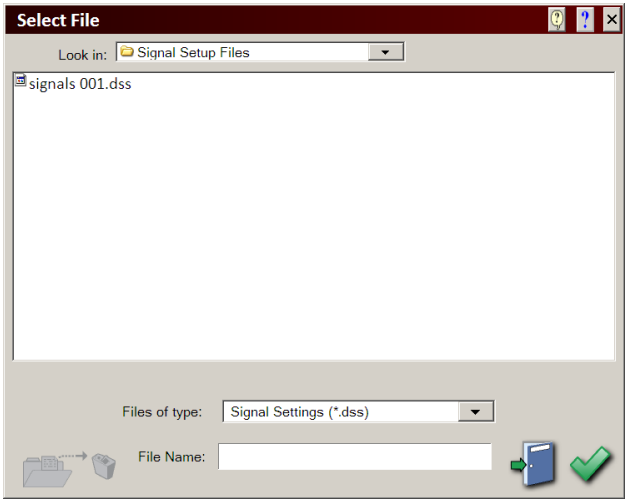
It may be helpful to create a library of signal setup files for commonly used measurement configurations. Additionally, signal setup files are portable, so they can be shared with other TMX units.

Saving signal setup files

1. Open the Select File window using one of the following methods.

   - Choose File >> Save >> Signals.
   - Choose Settings >> Channels. The Channel Settings window will open. Choose the Save Settings to File icon.

   The Select File window will open.



2. Choose a destination for the signal setup file. By default, the file will be saved on the system drive (C) in the selected folder. If necessary, you can save the file in a different location by choosing the folder list and browsing to a folder.

3. Choose the File Name field. A keypad will appear. Enter a file name and choose OK. The specified file name will appear in the field.

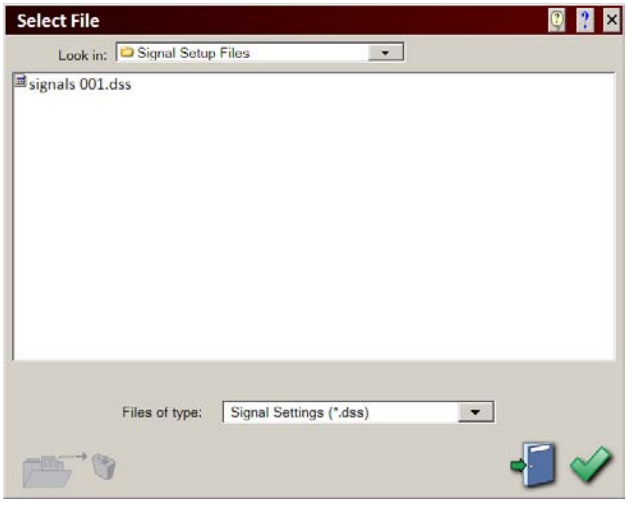
4. Choose OK.

Loading signal setup files

1. Open the Select File window using one of the following methods.

   - Choose File >> Load >> Signals.
   - Choose Settings >> Channels. The Channel Settings window will open. Choose the Load Settings from File icon.

   The Select File window will open.



2. Select a signal setup file to load. By default, files in the default location on the system drive (C) will be displayed. If necessary, you can select a file from a different location by choosing the folder list and browsing to a folder. Choose OK.

Loading signal setup files with the control panel

You can use control panel icons to quickly load signal setup files.

1. Add the signal setup file icon to the control panel.

Choose Settings >> Control Panel to open the Panel Settings window.

Choose the File button. A sub menu will appear. Choose Signals. The Select File window will open.

Select a signal setup file and choose OK. An icon for the selected file will be added to the control panel. Choose OK.

2. Load the signal setup file from the control panel by choosing the appropriate signal setup file icon.

Chapter 7: Display setup

Display Wizard

Use the Display Wizard to quickly set up or modify the display. You can add, edit, and remove grids and events. When you are finished, the grids and events will be sized to fit in the waveform display area.

Using the Display Wizard

1. Choose View >> Wizard. The Display Wizard window will open.

2. Choose the number field in the top of the grid options. A number pad will appear. Select the total number of grids you want to display.

   As an alternative, you can use the Basic Input Selection icon to quickly define a grid for each channel in an input module.

3. If necessary, add or remove grids.

   - To add a grid, choose the Add Grid icon.

- To remove a grid, select the grid and choose the Remove Grid icon.

4. If necessary, edit the properties of a grid by selecting the grid and choosing the Grid Properties icon.

The Grid Properties window will open.

- Select the channels to display by choosing them from the Available Channels list. Then choose the ">>>" button to add them to the Channels list.
- If necessary, select the channels to remove by choosing them from the Channels list. Then choose the "<<<" button to remove them from the Channels list.
- To change the number of grid divisions, choose the Divisions field. A number pad will appear. Enter the number of grid divisions and choose OK.
- To change the color of the grid, choose the Color box. The Color window will open. Select a color and choose OK.
- If there are two or more channels in the Channels list, select a master channel. This selection determines which channel's information will be displayed in the channel label on the right side of the grid.

5. Choose the number field in the top of the event options. A number pad will appear. Select the total number of events you want to display.
6. If necessary, add or remove events.
   - To add an event, choose the Add Event icon.

- To remove an event, select the event and choose the Remove Event icon.

7. If necessary, edit the properties of an event by selecting the event and choosing the Event Properties icon.

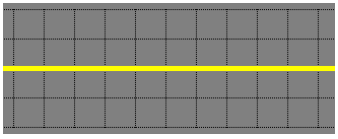
The Event Properties window will open.

- Select the event input to display by choosing it from the Event Input list.
- Select the event style from the Style list.
- To change the color of the event, choose the color box. The Color window will open. Select a color and choose OK.

8. Choose OK in the Display Wizard window.

Grids

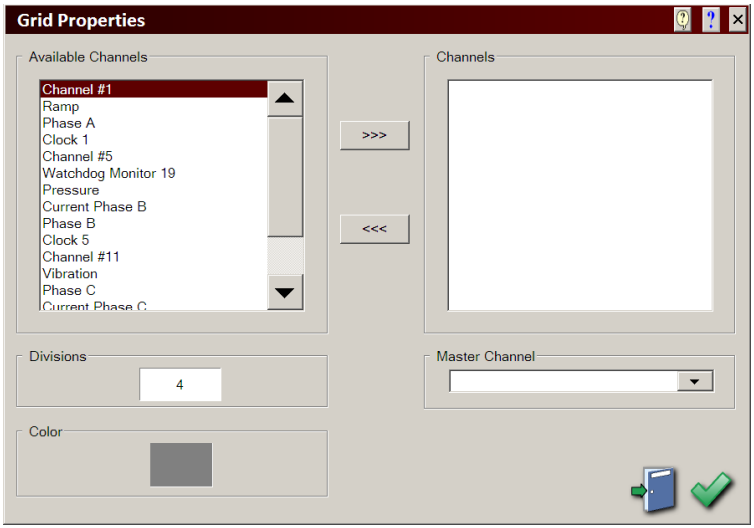
Channel waveforms are drawn on the display in customizable grids. The grids provide a visual aid that can be used for measurement purposes.



You can customize grid size, location, number of divisions, color, channel content, and various other settings to fit the needs of your application.

Adding grids

1. Choose View >> Add Grid. The Grid Properties window will open.



2. Select the channels to display by choosing them from the Available Channels list. Then choose the “>>>” button to add them to the Channels list.

3. If necessary, select the channels to remove by choosing them from the Channels list. Then choose the “<<<“ button to remove them from the Channels list.

4. To change the number of grid divisions, choose the Divisions field. A number pad will appear. Enter the number of grid divisions and choose OK.

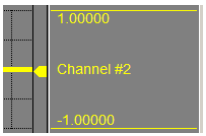
5. To change the color of the grid, choose the Color box. The Color window will open. Select a color and choose OK.

6. If there are two or more channels in the Channels list, select a master channel. This selection determines which channel’s information will be displayed in the channel label on the right side of the grid.

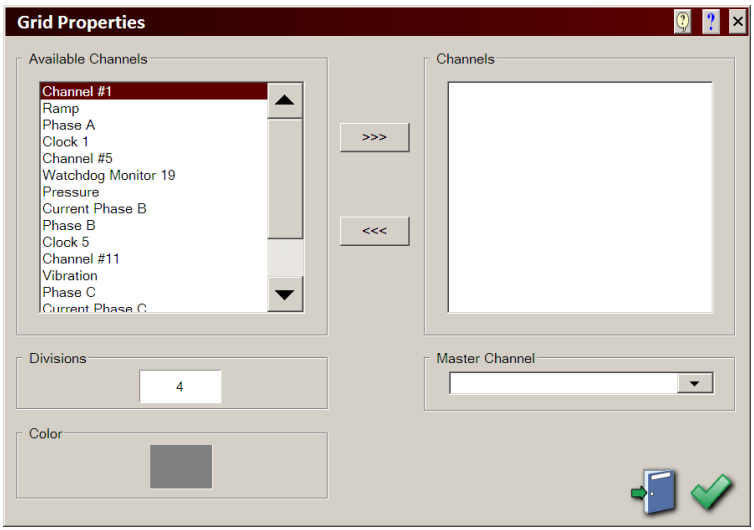
7. Choose OK.

Editing grids

1. Choose the channel label on the right side of the grid. A sub menu will appear.



2. Choose Properties. The Grid Properties window will open.



3. Select the channels to display by choosing them from the Available Channels list. Then choose the “>>>” button to add them to the Channels list.

4. If necessary, select the channels to remove by choosing them from the Channels list. Then choose the “<<<“ button to remove them from the Channels list.

5. To change the number of grid divisions, choose the Divisions field. A number pad will appear. Enter the number of grid divisions and choose OK.

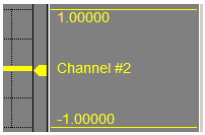
6. To change the color of the grid, choose the Color box. The Color window will open. Select a color and choose OK.

7. If there are two or more channels in the Channels list, select a master channel. This selection determines which channel's information will be displayed in the channel label on the right side of the grid.

8. Choose OK.

Moving grids

1. Choose the channel label on the right side of the grid. A sub menu will appear.



2. Choose Move. The channel label text color will change and the grid will be highlighted in the waveform display area.

3. Touch and drag up/down anywhere in the waveform display area. Move the grid highlight to the desired location. Choose the grid's channel label. The grid will be moved to the new position. To accommodate this movement, previous grids at and below this position on the display will be moved downward.

Resizing grids

1. Choose the channel label on the right side of the grid. A sub menu will appear.

2. To resize by moving the position of the grid's top, choose Size Top. The channel label text color will change and the grid top will be highlighted in the waveform display area.

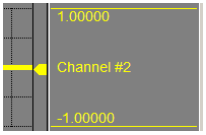
   Touch and drag up/down anywhere in the waveform display area. Move the grid top to the desired location. Choose the grid's channel label. The grid top will be moved to the new position. If necessary, other grids will be resized to accomoodate the new position.

3. To resize by moving the position of the grid's bottom, choose Size Bottom. The channel label text color will change and the grid bottom will be highlighted in the waveform display area.

   Touch and drag up/down anywhere in the waveform display area. Move the grid bottom to the desired location. Choose the grid's channel label. The grid bottom will be moved to the new position. If necessary, other grids will be resized to accomoodate the new position.

Removing grids

1. Choose the channel label on the right side of the grid. A sub menu will appear.



2. Choose Remove. The grid will be removed from the display.

Selecting a color for all grids

During the grid setup process, you assigned colors to individual grids using the Grid Properties window. As an alternative, you can quickly apply a color to all grids on the display.

1. Choose View >> Grid Color. The Color window will open. Select a color and choose OK. The selected color will be applied to all grids on the display.

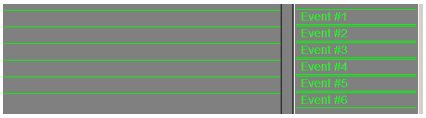
Selecting a background color

You can change the background color of the waveform display and channel label areas for contrast with your grid colors.

1. Choose View >> Background Color. The Color window will open. Select a color and choose OK. The selected color will be applied to the background.

Events

Events are binary signals that can be monitored and recorded along with waveform data. The state of an event signal is referred to as either high or low.



You can customize event location, style, and color to fit the needs of your application.

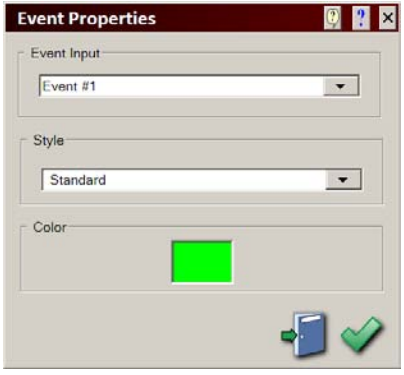
Event styles

During the event setup process, you will select styles for events. Event styles determine how event signals are drawn on the display. The following event styles are available.

Event Style	Example
Standard	
Bar/Line	
Bar/Off	
Tick	

Adding events

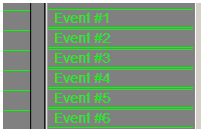
1. Choose View >> Add Event. The Event Properties window will open.



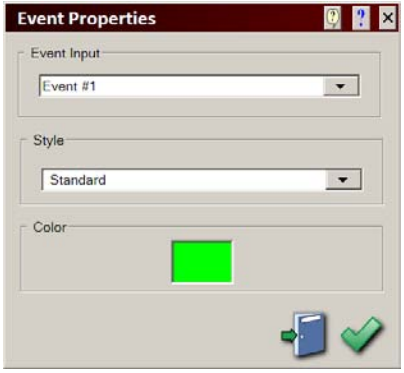
2. Select the event input to display by choosing it from the Event Input list.
3. Select the event style from the Style list.
4. To change the color of the event, choose the color box. The Color window will open. Select a color and choose OK.
5. Choose OK.

Editing events

1. Choose the event label on the right side of the event. A sub menu will appear.



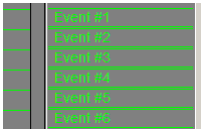
2. Choose Properties. The Event Properties window will open.



3. Select the event input to display by choosing it from the Event Input list.
4. Select the event style from the Style list.
5. To change the color of the event, choose the color box. The Color window will open. Select a color and choose OK.
6. Choose OK.

Moving events

1. Choose the event label on the right side of the event. A sub menu will appear.

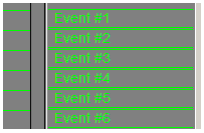


2. Choose Move. The event label text color will change and the event will be highlighted in the waveform display area.

3. Touch and drag up/down anywhere in the waveform display area. Move the event highlight to the desired location. Choose the event label. The event will be moved to the new position. To accommodate this movement, previous events at and below this position on the display will be moved downward.

Removing events

1. Choose the event label on the right side of the event. A sub menu will appear.



2. Choose Remove. The event will be removed from the display.

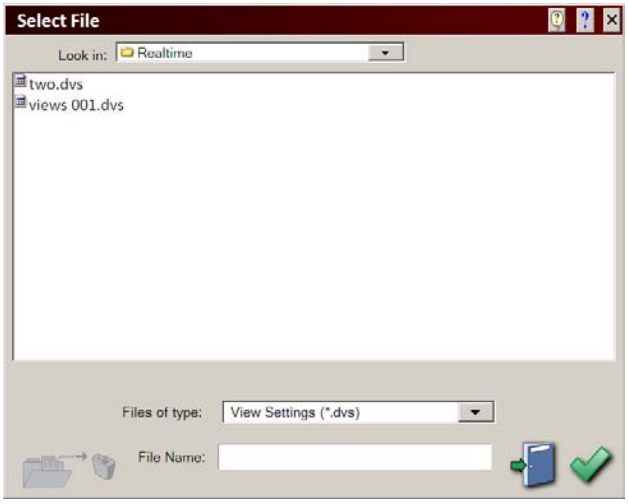
View setup files

View setup files contain information about the display settings. Once a display is set up for a particular application or user, the settings can be saved to a view setup file for later recall. These files can greatly decrease the amount of time spent customizing the display before measurement.

It may be helpful to create a library of view setup files to accommodate multiple tasks and/or users. Additionally, view setup files are portable, so they can be shared with other TMX units.

Saving view setup files

1. Choose File >> Save >> View. The Select File window will open.



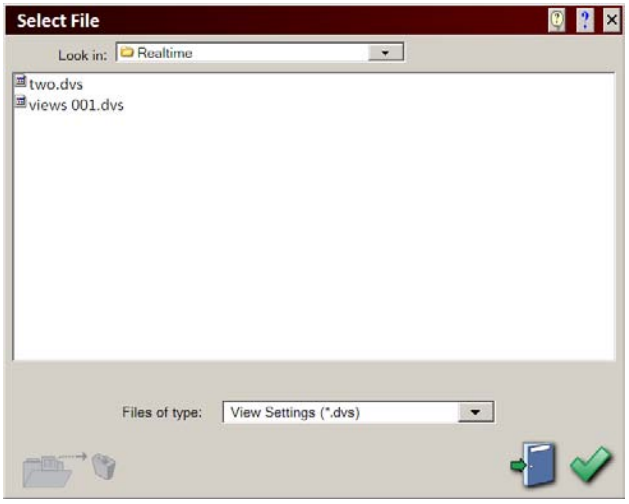
2. Choose a destination for the view setup file. By default, the file will be saved on the system drive (C) in the selected folder. If necessary, you can save the file in a different location by choosing the folder list and browsing to a folder.

3. Choose the File Name field. A keypad will appear. Enter a file name and choose OK. The specified file name will appear in the field.

4. Choose OK.

Loading view setup files

1. Choose File >> Load >> View. The Select File window will open.



2. Select a view setup file to load. By default, files in the default location on the system drive (C) will be displayed. If necessary, you can select a file from a different location by choosing the folder list and browsing to a folder. Choose OK.

Loading view setup files with the control panel

You can use control panel icons to quickly load view setup files.

1. Add the view setup file to the control panel.

   Choose Settings >> Control Panel to open the Panel Settings window.

   Choose the File button. A sub menu will appear. Choose View. The Select File window will open.

   Select a view setup file and choose OK. An icon for the selected file will be added to the control panel. Choose OK.

2. Load the view setup file from the control panel by choosing the appropriate view setup file icon.

Chapter 8: Realtime mode

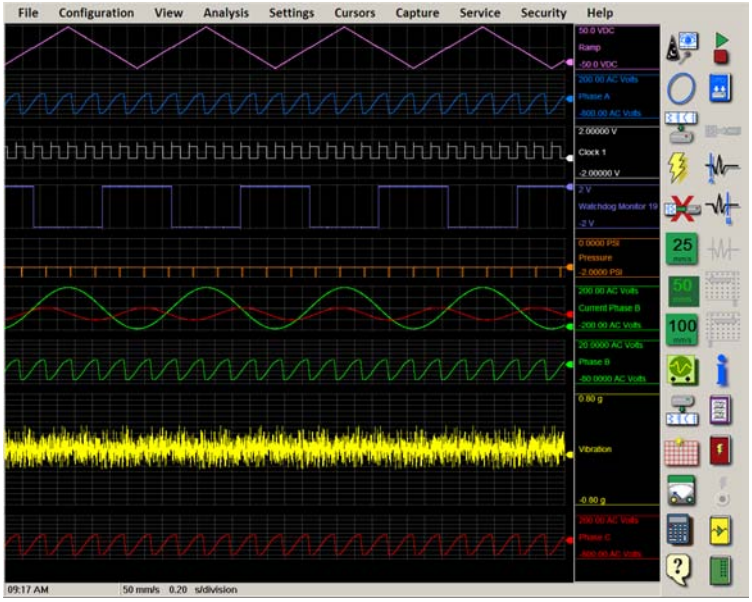
Realtime mode introduction

Realtime mode provides real-time waveform scrolling, monitoring, and data capture capabilities, typically used to view low frequency waveforms. Additionally, almost all system setup options are accessible from Realtime mode.

Accessing Realtime mode

1. Choose Configuration >> Realtime from the menu bar. Realtime mode will start.

   The following illustration displays a typical Realtime mode screen. Realtime mode screen appearances will vary based on the control panel configuration and other selected options.



Freezing and running the monitor

1. To freeze/run the monitor with the menu bar, choose View >> Freeze Display. Repeat this step to toggle between frozen and running monitor states.

2. To freeze/run the monitor with the control panel, choose the Monitor Run/Halt icon.

Repeat this step to toggle between frozen and running monitor states.

Realtime mode default control panel

The following list describes the functions of default control panel icons.

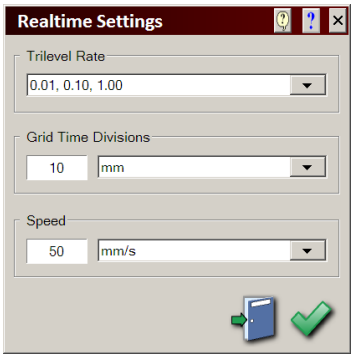
Note: *A selection path for each icon is indicated. When the Panel Settings window is open, you can use this path to add the icon to the control panel.*

Icon	Description
	**Monitor Run/Halt** starts and stops the on-screen display from running. *View >> Monitor Run/Halt*
	**Set Speed** changes the speed of the on-screen chart. In this example, the chart speed will be changed to 5 mm/s when the icon is pressed. Custom speed icons can be configured during the control panel setup process. *Settings >> Speed >> mm/s* or *mm/m*
	**Channel Settings** opens the Channel Settings window, which is used to set up amplifier inputs, base channels, derived channels, and event inputs. *Settings >> Channels*
	**Trigger Settings** opens the Trigger Settings window, which is used to set up data capture triggers and aborts. *Capture >> Trigger/Abort Settings*
	**Trigger Indicator** indicates when a trigger occurs by displaying a yellow circle. *Capture >> Trigger Indicator*
	**Realtime Settings** opens the Realtime Settings window, which is used to configure Realtime mode. *Settings >> Realtime*

Icon	Description
	**Arm Capture** arms (starts) a data capture using the configured data capture setup options. *Capture >> Arm*
	**Capture Indicator** indicates when a data capture is in progress by illuminating. *Capture >> Capture Indicator*

Setting up Realtime mode

1. Choose Settings >> Realtime.



2. Select a trilevel rate from the list.

   Each trilevel set represents a series of three time intervals (in seconds). The first, second, and third intervals indicate how often small, medium, and large marks will be printed respectively.

3. Set the size of the grid time divisions.

   - To set the size in millimeters, select the mm option from the list. Then choose the field to the left of the list. A number pad will appear. Enter the size of the grid time divisions and choose OK.
   - To set the size based on the trilevel rate, choose Trilevel Slow, Trilevel Medium, or Trilevel Fast from the list.

4. Set a chart speed by selecting millimeters per second or millimeters per minute from the list. Then choose the field to the left of the list. A number pad will appear. Enter the chart speed and choose OK.

5. Choose OK.

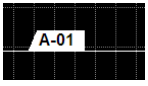
Realtime mode view options

This section describes the view options available in Realtime mode.

Printing signal IDs

Signal IDs are small visual indicators that identify channels. Signal IDs can be printed at any time. This feature is especially helpful for identifying multiple waveforms displayed in a single grid.

1. Choose View >> Print IDs.

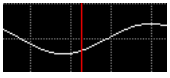


A signal ID indicator will be displayed and printed for each waveform and event.

Printing a full-page mark

Printing a full-page mark creates a vertical line that spans from the top to the bottom of the waveform display monitor.

1. Choose View >> Page Mark.



A mark will be printed vertically across all waveforms.

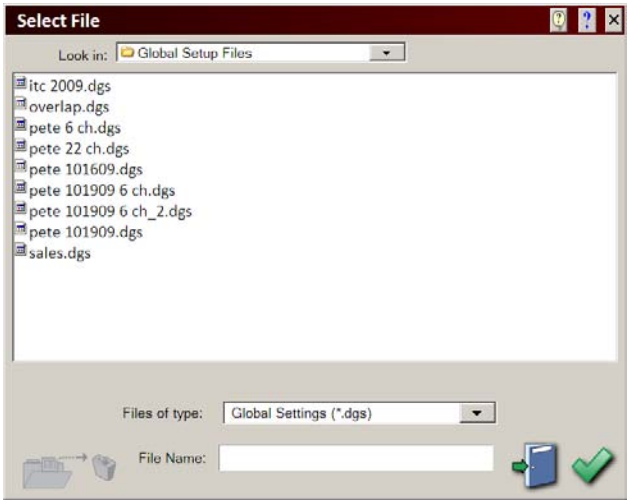
Global setup files

Global setup files contain all system setup information that can be saved to a file for later recall. These files can be considered a “complete setup” that can be saved and loaded as needed.

It may be helpful to create a library of global setup files to accommodate multiple tasks and/or users. Additionally, global setup files are portable, so they can be shared with other TMX units.

Saving global setup files

1. Choose File >> Save >> Global. The Select File window will open.



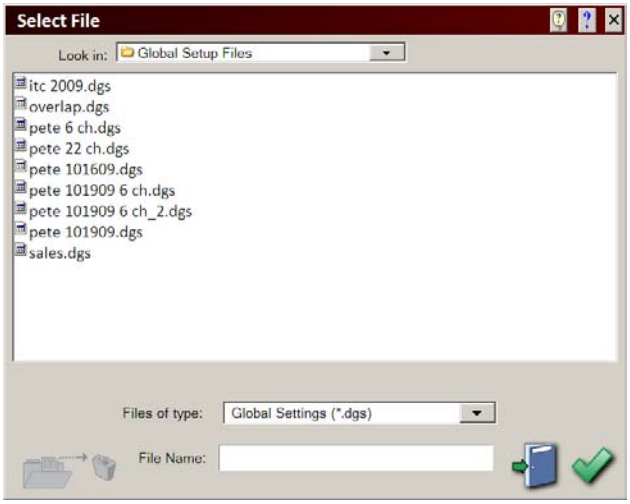
2. Choose a destination for the global setup file. By default, the file will be saved on the system drive (C) in the selected folder. If necessary, you can save the file in a different location by choosing the folder list and browsing to a folder.

3. Choose the File Name field. A keypad will appear. Enter a file name and choose OK. The specified file name will appear in the field.

4. Choose OK.

Loading global setup files

1. Choose File >> Load >> Global. The Select File window will open.



2. Select a global setup file to load. By default, files in the default location on the system drive (C) will be displayed. If necessary, you can select a file from a different location by choosing the folder list and browsing to a folder. Choose OK.

Loading global setup files with the control panel

You can use control panel icons to quickly load global setup files.

1. Add the global setup file to the control panel.

   Choose Settings >> Control Panel to open the Panel Settings window.

   Choose the File button. A sub menu will appear. Choose Global. The Select File window will open.

   Select a global setup file and choose OK. An icon for the selected file will be added to the control panel. Choose OK.

2. Load the global setup file from the control panel by choosing the appropriate global setup file icon.

Chapter 9: Data capture

Data capture overview

This section provides an overview of data capture concepts and the data capture process.

Data capture concepts

The following concepts are commonly used during data capture-related tasks.

Data capture

Data capture is the process of sampling signals and saving the sampled data to the data capture hard drive.

Data capture record (DCR) file

The recorder saves data capture information in a type of file format known as a DCR file. Base file names can be assigned to these files as part of the data capture setup process.

Sample rates

Sample rates define the speed at which signals are sampled. This rate is defined in units of Hz (hertz), the number of samples per second.

Trigger

A trigger is a user-defined event that starts the post-trigger recording phase of a data capture.

Data capture storage allocation

The data capture storage allocation determines the size of the data capture.

An estimate of needed disk space for the DCR file is derived based on the data capture storage allocation and sample rate specified.

Pre-trigger and post-trigger data

Pre-trigger data makes up the sequence of samples recorded prior to the occurrence of a trigger. Similarly, post-trigger data makes up the sequence of samples recorded after the occurrence of a trigger.

Pre-trigger percent

The pre-trigger percent is the amount of space in a data capture that is reserved for pre-trigger data.

This amount is defined by a percentage of the whole data capture size. For example, a pre-trigger percentage of 25% would result in a data capture file that contains 25% pre-trigger data and 75% post-trigger data.

Arm

Arming starts the data capture function. When armed, the system monitors trigger and abort conditions. If a trigger occurs, the post-trigger recording phase will begin. If an abort occurs, the data capture will be canceled.

- If a pre-trigger recording percentage is used, the pre-trigger recording phase begins when the system is armed. Pre-trigger samples will be acquired and stored up to the specified amount. When the specified amount of pre-trigger data has been stored, the oldest sample will be replaced by the most recent, creating a circular buffer.

- If no pre-trigger recording percentage is used, samples will be recorded up to the specified amount.

Auto re-arm

The auto re-arm feature automatically re-arms a new capture immediately after the current capture is complete. Automatically re-arming data captures is helpful when analyzing repetitive events, but it can produce a large number of captures depending on trigger conditions.

Archive

The recorder utilizes two separate hard drives: the data capture drive and the system drive. The data capture drive is used to record signal samples. The system drive contains the Windows operating system and can be used to archive files.

Archiving a data capture copies the DCR file from the capture drive to the system drive. The auto archive feature automatically archives the DCR file to the system drive immediately after the capture is complete.

Abort

An abort is a user-defined event that stops a data capture in progress. When an abort condition is detected, any currently running data capture will be stopped. All data captured up to this point is saved.

Data capture process

This section provides an overview of the data capture process.

1. **Set up the data capture**

   The data capture setup process involves entering a base file name for the data capture, specifying storage allocation options, defining and selecting channel sample rates, and activating the desired automation options.

2. **Set up triggers and aborts**

   The trigger and abort setup process involves defining conditions that initiate triggers and aborts.

3. **Arm the data capture**

   Arming starts the data capture function. When armed, the system monitors trigger and abort conditions. If a trigger occurs, the post-trigger recording phase will begin. If an abort occurs, the data capture will be canceled.

   - If a pre-trigger recording percentage is used, the pre-trigger recording phase begins when the system is armed. Pre-trigger samples will be acquired and stored up to the specified amount. When the specified amount of pre-trigger data has been stored, the oldest sample will be replaced by the most recent, creating a circular buffer.

   - If no pre-trigger recording percentage is used, samples will be recorded up to the specified amount.

4. **Post-trigger recording**

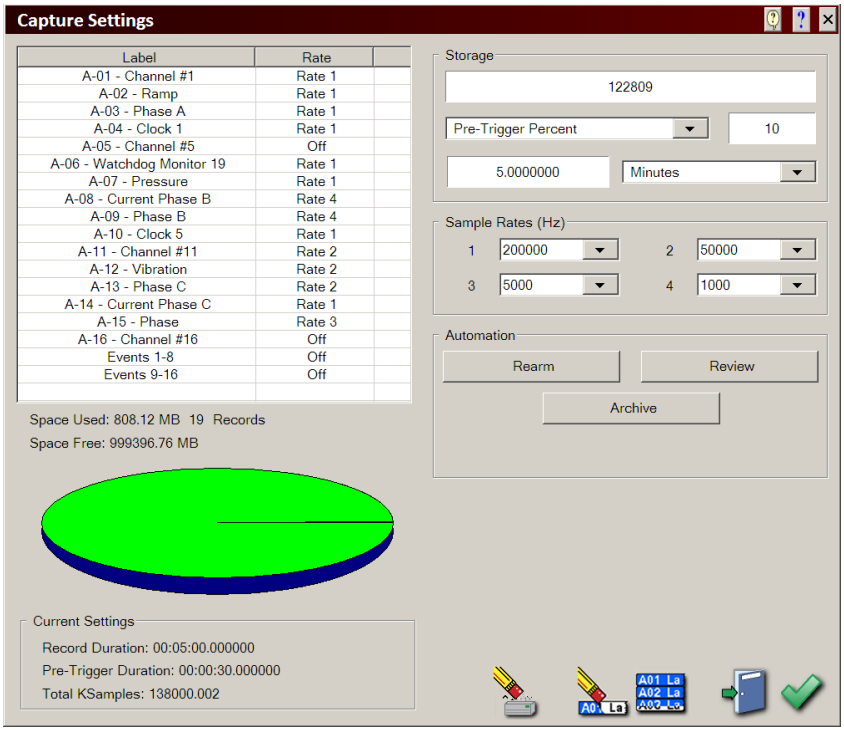
   When a trigger occurs, the post-trigger recording phase will begin. Samples will be acquired and saved until the data capture storage allocation is met or the capture is aborted.

Data capture setup

This section provides instructions for setting up data captures.

Setting up a data capture

1. Choose Capture >> Settings. The Capture Settings window will open.



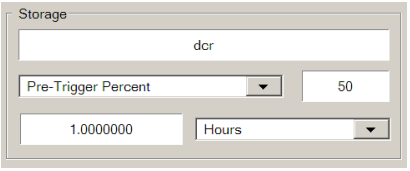
In the lower-left corner, a graphical representation of hard drive space is displayed. The following color key is used:

- **White** - Indicates the space available for data captures.
- **Blue** - Indicates the space already containing data captures.
- **Green** - Indicates the amount of space the next valid data capture will use.
- **Red** - Indicates that the next capture is too large for the hard drive.

2. Specify a DCR base file name by choosing the text field in the Storage options. A keypad will appear. Enter a base file name for the DCR file and choose OK.

   When a data capture is saved, the time and date of the capture are automatically appended to the end of the base file name.

3. Specify pre-trigger storage options by deciding if pre-trigger data should be included in the data capture.



   - If the data capture will contain pre-trigger data, select the Pre-Trigger Percent option. Then choose the Pre-Trigger Percent field. A number pad will appear.

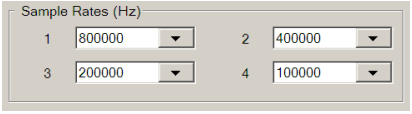
     Enter the percentage of the file that will be used for pre-trigger data and choose OK. If the trigger point represents the end of the data of interest, choose a high pre-trigger percentage. If it precedes the data of interest, choose a low number.

   - If the data capture will not contain pre-trigger data, select the No Trigger option.

4. Specify the size of the capture by defining a data capture storage allocation. The size can be defined in units of time (hours, minutes, or seconds) or number of samples (KS/channel).

   Select a unit of measure from the drop-down list. Then choose the quantity field to the left of the drop-down list. A number pad will appear. Enter the desired quantity and choose OK.

5. Specify sample rates to define the speeds at which channels are sampled. Sample rates are defined in units of Hz (hertz), the number of samples per second. You can select up to four sample rates.

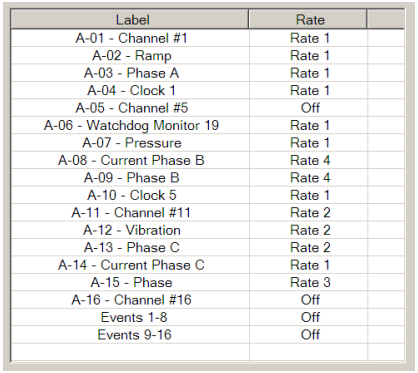


   Select the highest sample rate you plan to use from the first list. Then select the second, third, and fourth highest rates using the other lists.

   Subsequent sample rates can be set to a value up to half of the previous rate. For example, if you choose 80000 for rate 1, you can choose up to 40000 for rate 2.

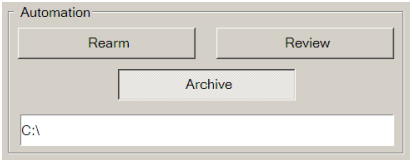
6. Select the channels to include in the data capture, and the sample rate for each channel.

   Select a channel and choose the Rate column heading. A list of sample rate options will appear. To include the channel in the data capture, select Rate 1, Rate 2, Rate 3, or Rate 4. To exclude the channel from the data capture, select Off.



Label	Rate
A-01 - Channel #1	Rate 1
A-02 - Ramp	Rate 1
A-03 - Phase A	Rate 1
A-04 - Clock 1	Rate 1
A-05 - Channel #5	Off
A-06 - Watchdog Monitor 19	Rate 1
A-07 - Pressure	Rate 1
A-08 - Current Phase B	Rate 4
A-09 - Phase B	Rate 4
A-10 - Clock 5	Rate 1
A-11 - Channel #11	Rate 2
A-12 - Vibration	Rate 2
A-13 - Phase C	Rate 2
A-14 - Current Phase C	Rate 1
A-15 - Phase	Rate 3
A-16 - Channel #16	Off
Events 1-8	Off
Events 9-16	Off

7. Specify automation preferences by choosing whether to enable automatic re-arm, review, and archive options.



   - To automatically re-arm the capture immediately after the current capture is complete, choose the Rearm option. This feature cannot be used if Auto Review is enabled.

     Automatically re-arming data captures is helpful when analyzing repetitive events, but it can produce a large number of captures depending on trigger conditions.

   - To automatically open the DCR file in Review mode immediately after the data capture is complete, choose the Review option. This feature cannot be used if Auto Rearm is enabled.

- To automatically archive the DCR file to the system or other drive immediately after the capture is complete, choose the Archive option. A file location field will appear.

  Choose the file location field. The Select Folder window will open. Browse to the desired archive folder and choose OK.

8. Choose OK.

Triggers and aborts

A trigger is a user-defined event that starts the post-trigger recording phase of a data capture. An abort is a user-defined event that stops a data capture in progress. When an abort condition is detected, any currently running data capture will be stopped. All data captured up to this point is saved.

Trigger and abort conditions can be defined as the manual push of a button, the receiving of an external signal, or when specific channels detect data at certain values.

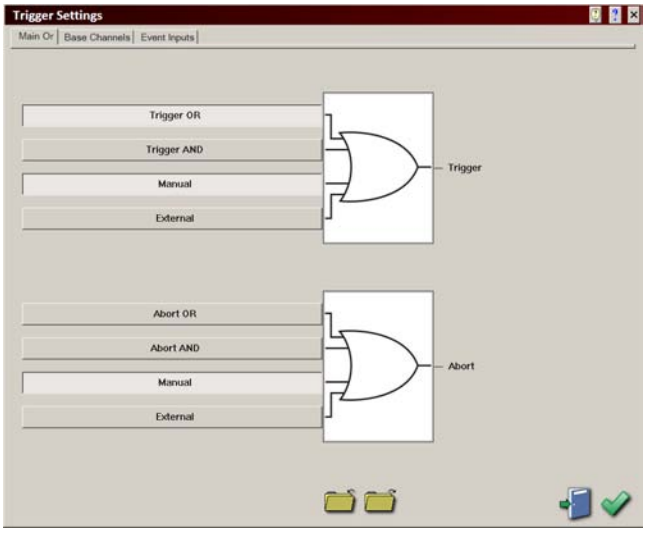
The following trigger/abort types are available:

- **Window Trigger/Abort** - This trigger/abort occurs when signals move inside or outside a predetermined window of values. The user provides the high and low level for the window.
- **Edge Trigger/Abort** - This trigger/abort occurs when signals move above or below a certain level. The user provides this level.
- **Slew Trigger/Abort** - This trigger/abort occurs when a signal's rate of change (known as "slew" or "slope") reaches or drops below a certain value. The user provides the change in amplitude and the length of time, which are used to calculate slew.
- **Manual Trigger/Abort** - This trigger/abort occurs when the user produces a trigger/abort via the menu bar or control panel.
- **External Trigger/Abort** - This trigger/abort uses an external signal via the Utility / D|O port to produce a trigger or abort. The external signal is a switch closure to the ground or TTL low. Process control signals can be used to produce this trigger/abort.
- **Event Trigger/Abort** - This trigger/abort uses external signals via the event port to produce a trigger or abort, based on the state of the events.

Enabling and disabling triggers and aborts

1. Choose Capture >> Trigger/Abort Settings. The Trigger Settings window will open.

2. Choose the Main Or tab.



3. Enable and disable triggers or aborts by choosing the trigger/abort buttons. Buttons that appear "pressed in" indicate enabled triggers/aborts.

   There are eight groups of triggers and aborts that can be enabled/disabled from this screen. All triggers/aborts fall into one of these categories.

   - Trigger OR, Abort OR
   - Trigger AND, Abort AND
   - Manual Trigger, Manual Abort
   - External Trigger, External Abort

   Note: *Event triggers/aborts are the exception to this rule and are configured separately in the Event Inputs tab.*

4. The next step in the setup process depends on what kind of trigger/abort you are setting up.

- **Manual** triggers/aborts require no additional setup. Simply choose OK to close the Trigger Settings window.
- **External** triggers/aborts require that you set up the Utility / D|O port, if you have not already done so.
- **AND/OR** triggers/aborts have their own setup procedures.

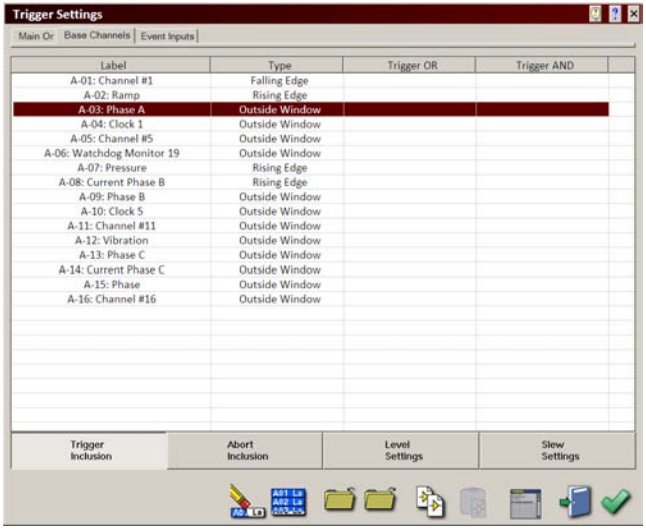
Related topics:
- *AND/OR triggers/aborts* on page 9-10
- *Viewing and modifying utility / D|O port settings* on page 14-10

Trigger Settings window overview

The Trigger Settings window (Capture >> Trigger/Abort Settings) is used to set up triggers and aborts. The Trigger Settings window provides two layout options.

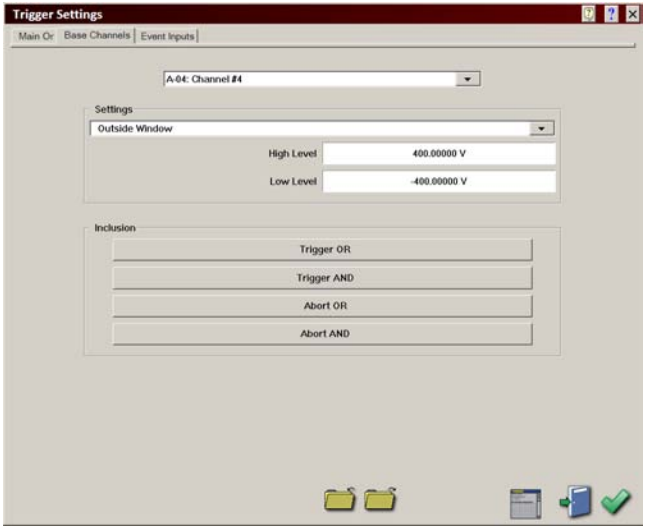
- **List view** - You can change settings in list view by selecting a channel and choosing the column heading for the setting you want to change. Some options can be configured for a group of channels at the same time by selecting a group of channels prior to choosing a column heading. In this case, setup information will be applied to all selected channels.

  Some options must be configured for each channel individually, or in groups based on compatible channels.



Use the buttons below the list to display other columns.

- **Graphical view** - You can change settings in graphical view by selecting a single channel and the appropriate options for the channel.



You can switch between list view and graphical view by choosing the Toggle Layout icon.

AND/OR triggers/aborts

All AND/OR triggers and aborts use AND/OR logic to determine when to activate; they can be based on an amplitude window, level (edge), or slew.

- **OR** triggers/aborts will activate if **any** OR conditions have been met.

  For example: A one-channel OR trigger/abort will activate as soon as its conditions are met; the status of the other channels is irrelevant. A two-channel OR trigger/abort will activate as soon as either of the channels' conditions are met

- **AND** triggers/aborts will activate if (and only if) **all** AND conditions have been met.

  For example: A two-channel AND trigger/abort will activate as soon as both channels' conditions are met; nothing will occur if one channel meets

its conditions without the other. A three-channel trigger/abort will activate as soon as all three channels' conditions are met.

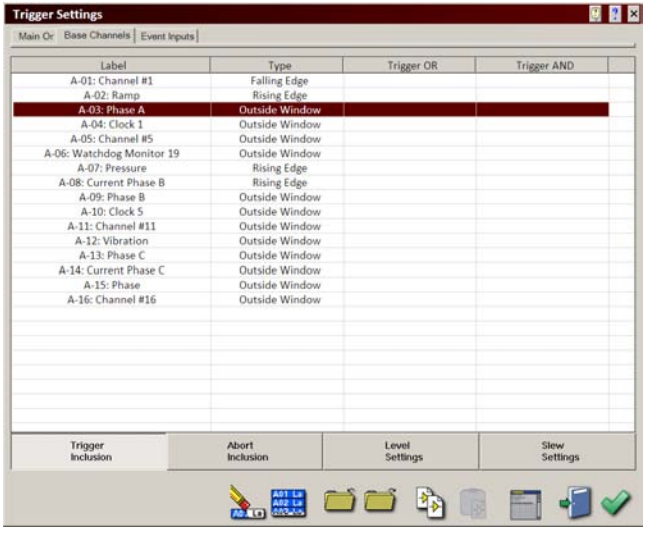
Any channel can be included or removed from either trigger/abort by adjusting its settings in the Trigger Settings window.

Setting up a Window trigger/abort

This trigger/abort occurs when signals move inside or outside a predetermined window of values. The user provides the high and low level for the window.

1. Choose Capture >> Trigger/Abort Settings. The Trigger Settings window will open.

2. Choose the Base Channels tab.



3. Select the channel you want to apply the trigger/abort to from the channel list.

4. Choose the Type column heading. Select the window type from the list.

   - **Outside window** - Choose this option to set the trigger/abort to occur when the signal exceeds the upper boundary or drops below the lower boundary.

- **Inside window** - Choose this option to set the trigger/abort to occur when the signal exceeds the lower boundary but remains below the upper boundary.

5. Define the amplitude window boundaries.

   - Choose the High Level column heading. A number pad will appear. Enter the high value for the amplitude window and choose OK.

   - Choose the Low Level column heading. A number pad will appear. Enter the low value for the amplitude window and choose OK.

6. Set the trigger/abort as AND, OR, or both.

   - **Trigger OR** - To include the channel in the OR trigger, choose the Trigger OR column heading and choose Include.

   - **Trigger AND** - To include the channel in the AND trigger, choose the Trigger AND column heading and choose Include.

   - **Abort OR** - To include the channel in the OR abort, choose the Abort OR column heading and choose Include.

   - **Abort AND** - To include the channel in the AND abort, choose the Abort AND column heading and choose Include.

7. Choose OK.

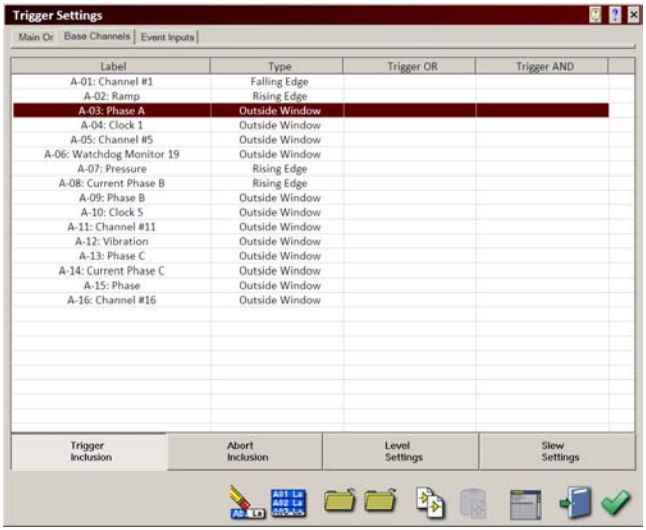
Related topics:
- *AND/OR triggers/aborts* on page 9-10

Setting up an Edge trigger/abort

This trigger/abort occurs when signals move above or below a certain level. The user provides this level.

1. Choose Capture >> Trigger/Abort Settings. The Trigger Settings window will open.

2. Choose the Base Channels tab.



3. Select the channel you want to apply the trigger/abort to from the channel list.

4. Choose the Type column heading. Select the edge type from the settings list.

   - **Rising Edge** - Choose this option to set the trigger/abort to occur when the signal rises above a specific level.

   - **Falling Edge** - Choose this option to set the trigger/abort to occur when the signal drops below a specific level.

5. Choose the High Level column heading. A number pad will appear. Enter the edge level and choose OK.

   Note: *The Low Level setting is not used in Rising Edge or Falling Edge triggers/aborts.*

6. Set the trigger/abort as AND, OR, or both.

   - **Trigger OR** - To include the channel in the OR trigger, choose the Trigger OR column heading and choose Include.

   - **Trigger AND** - To include the channel in the AND trigger, choose the Trigger AND column heading and choose Include.

- **Abort OR** - To include the channel in the OR abort, choose the Abort OR column heading and choose Include.
- **Abort AND** - To include the channel in the AND abort, choose the Abort AND column heading and choose Include.

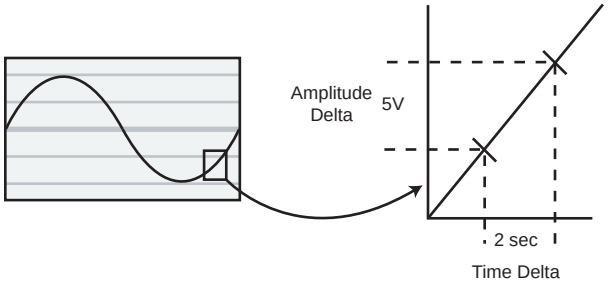
7. Choose OK.

Related topics:
- *AND/OR triggers/aborts* on page 9-10

About Slew triggers/aborts

This trigger/abort occurs when a signal's rate of change (known as "slew" or "slope") reaches or drops below a certain value. The user provides the change in amplitude and the length of time, which are used to calculate slew.

The following example illustrates important concepts related to this trigger/abort. You may want to refer to this example as you set up a Slew trigger/abort.



Amplitude Delta - The Amplitude Delta indicates the slew voltage span. The slope of this voltage span with respect to the Time Delta will set the trigger point.

For this example, Amplitude Delta is set to 5 V.

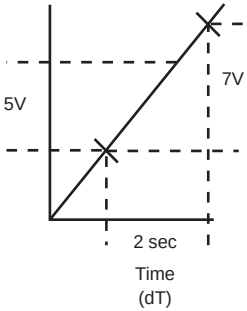
Time Delta - The Time Delta indicates the time to monitor the slope change of the Amplitude Delta span. This can be set from 500 ns to 8.3886075 seconds. This is done by choosing a number that when multiplied with 0.0000005 (500 ns, the shortest time you can set) results in the Time Delta.

For this example, Time Delta is set to 4000000, resulting in a Time Delta of 2 seconds.

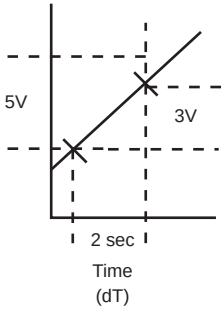
Slew Spikes/Dropouts - A slew spike occurs when the slew rate of the Amplitude Delta with respect to the Time Delta rises above the specified parameters.

Likewise, a slew dropout occurs when the slew rate of the Amplitude Delta with respect to the Time Delta falls below the specified parameters.

- If using slew spikes, the following scenario would result in a trigger/abort: Signal Slew Rate = 7V / 2 sec = 3.5 V/s



- If using slew dropouts, the following scenario would result in a trigger/abort: Signal Slew Rate = 3V / 2 sec = 1.5 V/s

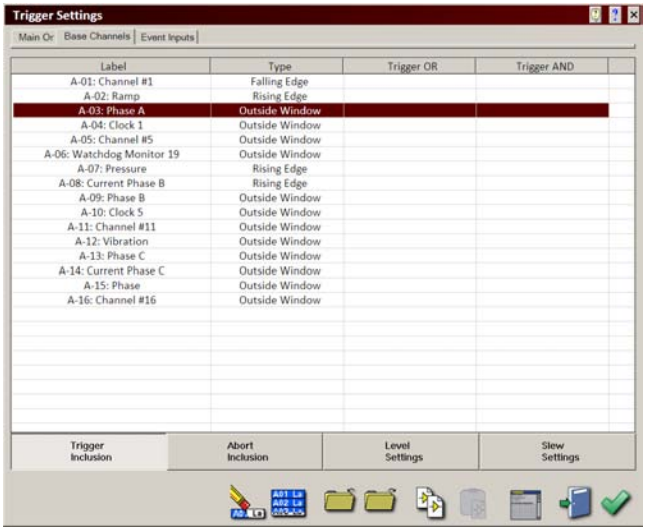


Setting up a Slew trigger/abort

This trigger/abort occurs when a signal's rate of change (known as "slew" or "slope") reaches or drops below a certain value. The user provides the change in amplitude and the length of time, which are used to calculate slew.

1. Choose Capture >> Trigger/Abort Settings. The Trigger Settings window will open.

2. Choose the Base Channels tab.



3. Select the channel you want to apply the trigger/abort to from the channel list.

4. Choose the Type column heading. Select the slew type from the list.

   - **Slew Spikes** - Choose this option to set the trigger/abort to occur when the slew rises above the chosen rate.

   - **Slew Dropout** - Choose this option to set the trigger/abort to occur when the slew falls below the chosen rate

5. Define the slew rate.

   - Choose the Time Delta column heading. A number pad will appear. Enter the time delta value and choose OK.

   - Choose the Amplitude Delta column heading. A number pad will appear. Enter the amplitude delta value and choose OK.

6. Set the trigger/abort as OR.

   - **Trigger OR** - To include the channel in the OR trigger, choose the Trigger OR column heading and choose Include.

   - **Abort OR** - To include the channel in the OR abort, choose the Abort OR column heading and choose Include.

Note: *Trigger/Abort AND logic cannot be used on slew triggers/aborts.*

7. Choose OK.

Related topics:
- *AND/OR triggers/aborts* on page 9-10

Event triggers/aborts

An Event Trigger/Abort occurs when event inputs meet their change in logic state. This is communicated through the Utility / D|O port, and is not affected by any of the channel waveforms.

There are two types of Event triggers/aborts:

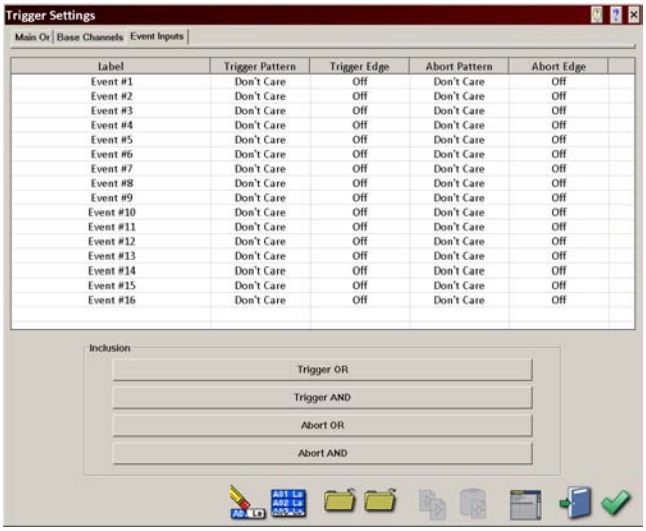
- **Pattern** - An event pattern trigger/abort occurs each time all events meet the selected logic state (high, low, glitch, don't care). The user provides the state for each event.

- **Edge** - An event edge trigger/abort occurs any time at least one of the events changes states in a specific direction (rising, falling, either). The user provides this direction for each event.

Setting up an Event trigger/abort

1. Choose Capture >> Trigger/Abort Settings. The Trigger Settings window will open.

2. Choose the Event Inputs tab.



3. Select the event you want to apply the trigger/abort to from the event list.

4. To set an event pattern trigger/abort, choose the Trigger Pattern or Abort Pattern column heading. Select a pattern option. Remember that all the events must meet their states in order for an event pattern trigger or abort to occur.

   - **Don't Care** - If the Don't Care option is chosen, changes in the selected event will not be considered in the trigger/abort pattern.
   - **Low** - If the Low option is chosen, a trigger/abort will occur when the selected event is in its low state (assuming all other event state trigger/abort conditions are met).
   - **High** - If the High option is chosen, a trigger/abort will occur when the selected event changes to its high state (assuming all other event state trigger/abort conditions are met).
   - **Glitch** - If the Glitch option is chosen, a trigger/abort will occur any time the selected event changes state in any way; low to high or high to low (assuming all other event state trigger/abort conditions are met).

5. To set an event edge trigger/abort, choose the Trigger Edge or Abort Edge column heading. Select an edge option. Remember that a trigger or abort will occur whenever any one of the events meets its state.

    - **Off** - If the Off option is chosen, changes in the selected event will not cause a trigger/abort to occur.

    - **Rising** - If the Rising option is chosen, a trigger/abort will occur whenever the selected event changes to its high state.

    - **Falling** - If the Falling option is chosen, a trigger/abort will occur whenever the selected event changes to its low state.

    - **Either** - If the Either option is chosen, a trigger/abort will occur any time the selected event changes state in any way; low to high or high to low.

6. Set the event triggers/aborts as AND, OR, or both.

    - **Trigger OR** - Choose this option to include events in the OR trigger.

    - **Trigger AND** - Choose this option to include events in the AND trigger.

    - **Abort OR** - Choose this option to include events in the OR abort.

    - **Abort AND** - Choose this option to include events in the AND abort.

7. Choose OK.

Related topics:
- *AND/OR triggers/aborts* on page 9-10

Performing a data capture

When the data capture setup, trigger setup, and abort setup tasks are complete, the data capture can be initiated by arming the system.

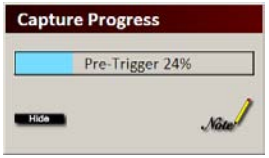
Arming starts the data capture function. When armed, the system monitors trigger and abort conditions. If a trigger occurs, the post-trigger recording phase will begin. If an abort occurs, the data capture will be canceled.

- If a pre-trigger recording percentage is used, the pre-trigger recording phase begins when the system is armed. Pre-trigger samples will be acquired and stored up to the specified amount. When the specified amount of pre-trigger data has been stored, the oldest sample will be replaced by the most recent, creating a circular buffer.

- If no pre-trigger recording percentage is used, samples will be recorded up to the specified amount.

When a trigger occurs, the post-trigger recording phase will begin. Samples will be acquired and saved until the data capture storage allocation is met or the capture is aborted.

1. Choose Capture >> Arm. The Capture Progress window will open.



   If the data capture includes pre-trigger data, the recorder will begin recording pre-trigger data.

2. Wait for a trigger or abort to occur based on the trigger and abort settings.

   - If manual triggers are used, manually trigger the data capture when appropriate by choosing Capture >> Trigger.

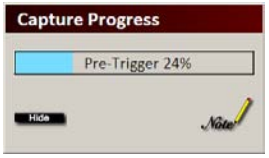
   After a trigger occurs, the data capture will complete automatically and save the results to a file.

   Note: *If automatic re-arming is enabled, the system will continually re-arm the data capture.*

   - If manual aborts are used, manually abort the data capture when appropriate by choosing Capture >> Abort.

Adding notes to data captures

1. When a data capture is in progress, the Capture Progress window will be open.



2. Choose the Note icon to add a text note. A keypad will appear.

3. Enter the note text and choose OK. The note will be saved as part of the data capture. To enter another note, repeat this process.

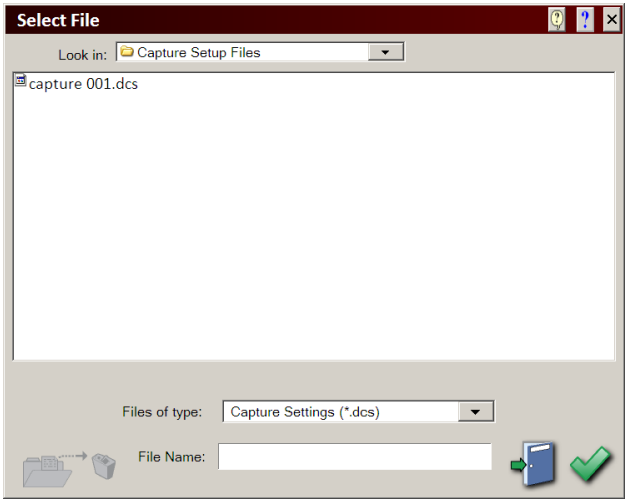
Capture setup files

Capture setup files contain data capture setup information from the Capture Settings window. Once a data capture is set up for a particular application, the settings can be saved to a capture setup file for later recall. These files can greatly decrease the amount of time spent on setting up captures before measurement.

It may be helpful to create a library of capture setup files for commonly used measurement configurations. Additionally, capture setup files are portable, so they can be shared with other TMX units.

Saving capture setup files

1. Choose File >> Save >> Capture. The Select File window will open.



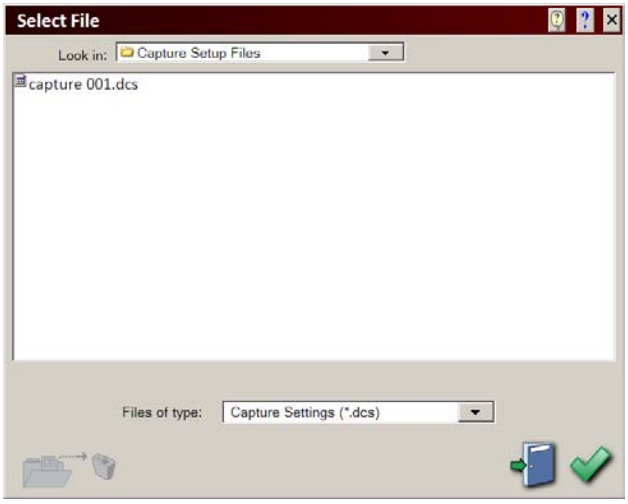
2. Choose a destination for the capture setup file. By default, the file will be saved on the system drive (C) in the selected folder. If necessary, you can save the file in a different location by choosing the folder list and browsing to a folder.

3. Choose the File Name field. A keypad will appear. Enter a file name and choose OK. The specified file name will appear in the field.

4. Choose OK.

Loading capture setup files

1. Choose File >> Load >> Capture. The Select File window will open.



2. Select a capture setup file to load. By default, files in the default location on the system drive (C) will be displayed. If necessary, you can select a file from a different location by choosing the folder list and browsing to a folder. Choose OK.

Loading capture setup files with the control panel

You can use control panel icons to quickly load capture setup files.

1. Add the capture setup file icon to the control panel.

   Choose Settings >> Control Panel to open the Panel Settings window.

   Choose the File button. A sub menu will appear. Choose Capture. The Select File window will open.

   Select a capture setup file and choose OK. An icon for the selected file will be added to the control panel. Choose OK.

2. Load the capture setup file from the control panel by choosing the appropriate capture setup file icon.

Trigger setup files

Trigger setup files contain all setup information from the Trigger Settings window. Once triggers are set up for a particular application, the settings can be saved to a trigger setup file for later recall. These files can greatly decrease the amount of time spent on setting up triggers before measurement.

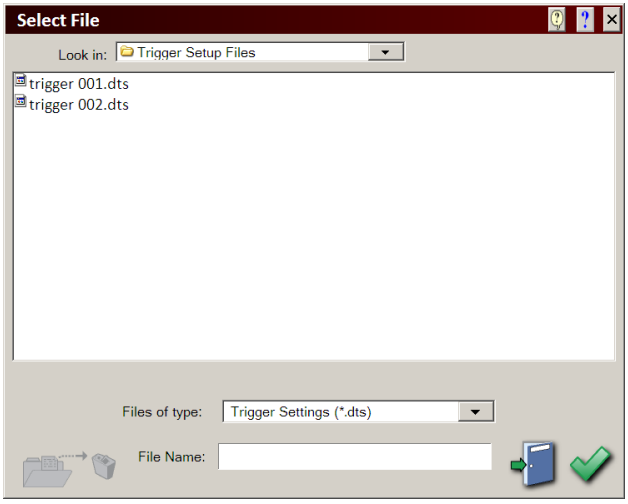
It may be helpful to create a library of trigger setup files for commonly used measurement configurations. Additionally, trigger setup files are portable, so they can be shared with other TMX units.

Saving trigger setup files

1. Open the Select File window using one of the following methods.

   - Choose File >> Save >> Trigger.
   - Choose Capture >> Trigger/Abort Settings. The Trigger Settings window will open. Choose the Save Settings to File icon.

   The Select File window will open.



2. Choose a destination for the trigger setup file. By default, the file will be saved on the system drive (C) in the selected folder. If necessary, you can save the file in a different location by choosing the folder list and browsing to a folder.

3. Choose the File Name field. A keypad will appear. Enter a file name and choose OK. The specified file name will appear in the field.

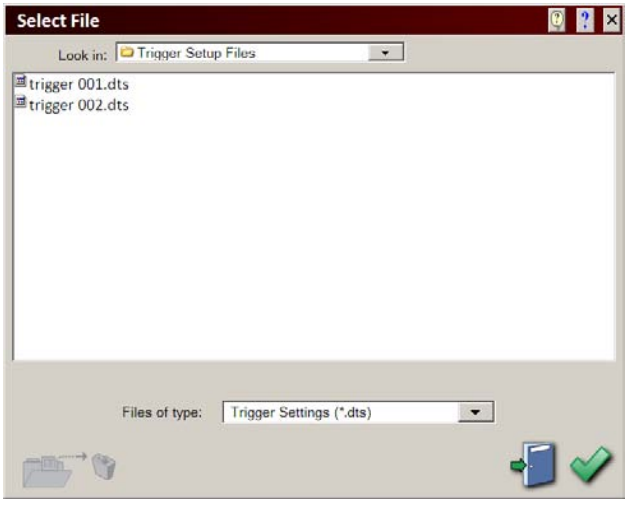
4. Choose OK.

Loading trigger setup files

1. Open the Select File window using one of the following methods.

   - Choose File >> Load >> Trigger.
   - Choose Capture >> Trigger/Abort Settings. The Trigger Settings window will open. Choose the Load Settings from File icon.

   The Select File window will open.



2. Select a trigger setup file to load. By default, files in the default location on the system drive (C) will be displayed. If necessary, you can select a file from a different location by choosing the folder list and browsing to a folder. Choose OK.

Loading trigger setup files with the control panel

You can use control panel icons to quickly load trigger setup files.

1. Add the trigger setup file icon to the control panel.

Choose Settings >> Control Panel to open the Panel Settings window.

Choose the File button. A sub menu will appear. Choose Trigger. The Select File window will open.

Select a trigger setup file and choose OK. An icon for the selected file will be added to the control panel. Choose OK.

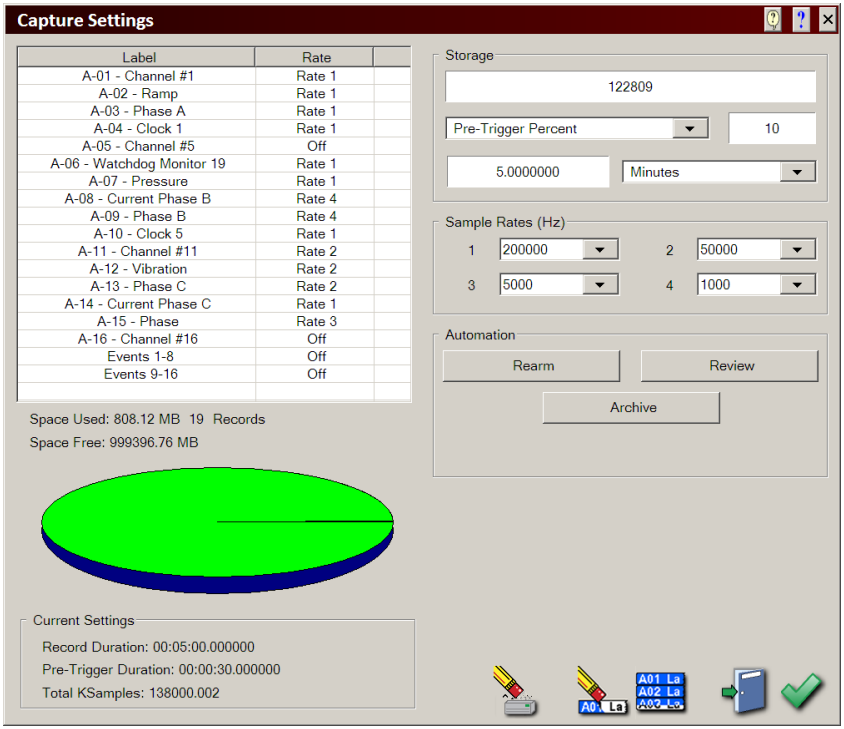
2. Load the trigger setup file from the control panel by choosing the appropriate trigger setup file icon.

Formatting the data capture drive

Records can be removed from the data capture drive by performing a format of the drive. This procedure applies to the data capture drive only; it does not affect the system (Windows) drive.

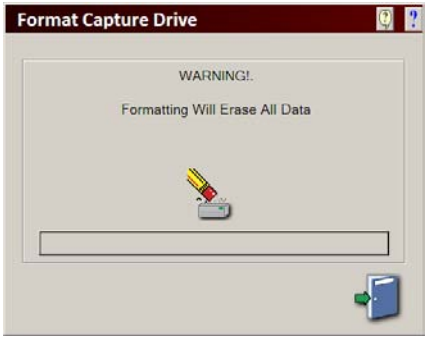
Caution: *Use caution when formatting the data capture drive because all data files are deleted. Ensure you have archived any needed files prior to formatting.*

1. Choose Capture >> Settings. The Capture Settings window will open.



2. Choose the Format Capture Disk button.

The Format Capture Drive window will open.



3. Choose the Format Drive button.

A confirmation message will appear. Choose OK. The data capture drive will be formatted

Chapter 10: Scope mode

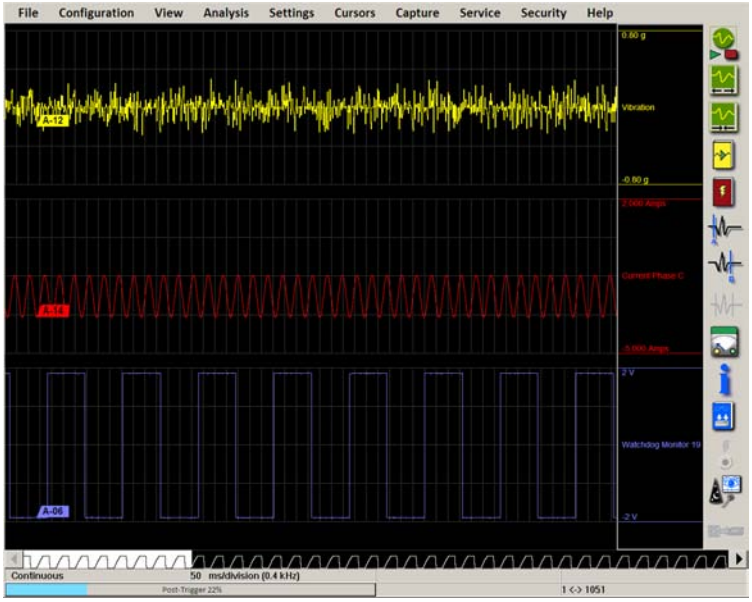
Scope mode introduction

Scope mode acts like a digital storage oscilloscope, providing high time-base resolution for viewing high-frequency signals. Scope mode is useful for timing and synchronization analysis, transient capture, and high-speed testing. It can be used while continuously capturing data and monitoring signals on the display.

Accessing Scope mode

1. Choose Configuration >> Scope from the menu bar. Scope mode will start.

   The following illustration displays a typical Scope mode screen. Scope mode screen appearances will vary based on the control panel configuration and other selected options.



Using the track view

The track view is a visual scroll bar located on the bottom of the display. It can be used to navigate to other areas of the capture.

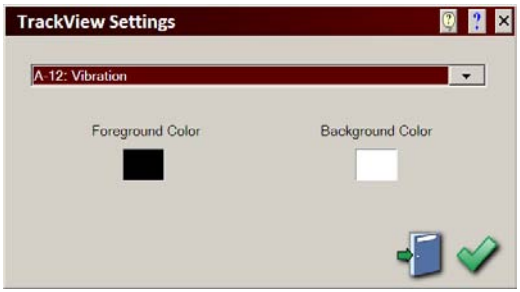
The track view displays a signal for a visual reference. The signal displayed in the track view can be changed, if desired. If the track view signal is changed, the selected signal will become the default until another signal is chosen.

1. Navigate though a file using the track view.

    - To scroll slowly through the file in a particular direction, choose one of the arrows on the sides of the track view.

    - To scroll quickly through the file in a particular direction, press and hold the shaded portion of the track view. Then drag it to the new location and release it. The file will scroll as the shaded bar is dragged.

2. If necessary, edit track view settings by choosing View >> Track View Channel. The TrackView Settings window will open.



    - Select a channel or event to display in the track view.

    - To select a track view foreground color, choose the Foreground Color box. The Color window will open. Select a color and choose OK.

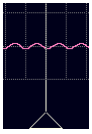
    - To select a track view background color, choose the Background Color box. The Color window will open. Select a color and choose OK.

3. Choose OK.

Displaying the trigger line

The trigger line is a vertical line drawn on the display that represents the trigger point of a capture. Use the following instructions to display the trigger line.

1. Choose View >> Show Trigger Line. The trigger line will be displayed.



2. To hide the trigger line, repeat this process.

Displaying channel IDs

Channel IDs are small visual indicators that identify channels. This feature is especially helpful for identifying multiple signals displayed in a capture.

1. Choose View >> Show Channel IDs. Channel IDs will be displayed.

2. To hide channel IDs, repeat this process.

Scope mode default control panel

The following list describes the functions of default control panel icons.

Note: *A selection path for each icon is indicated. When the Panel Settings window is open, you can use this path to add the icon to the control panel.*

Icon	Description
	**Scope Arm/Abort** arms and aborts scope captures. *Settings >> Arm/Abort*
	**Channel Settings** opens the Channel Settings window, which is used to set up amplifier inputs, base channels, derived channels, and event inputs. *Settings >> Channels*

Icon	Description
	**Trigger Settings** opens the Trigger Settings window, which is used to set up data capture triggers and aborts. *Capture >> Trigger/Abort Settings*
	**Cursor A** displays and hides cursor A. *Cursors >> Cursor A*
	**Cursor B** displays and hides cursor B. *Cursors >> Cursor B*
	**Move Cursor Left** moves the active cursor(s) to the left each time the icon is pressed. *Cursors >> Move Left*
	**Move Cursor Right** moves the active cursor(s) to the right each time the icon is pressed. *Cursors >> Move Right*
	**Trigger Indicator** indicates when a trigger occurs by displaying a yellow circle. *Capture >> Trigger Indicator*
	**Capture Indicator** indicates when a data capture is in progress by illuminating. *Capture >> Capture Indicator*

Scope captures

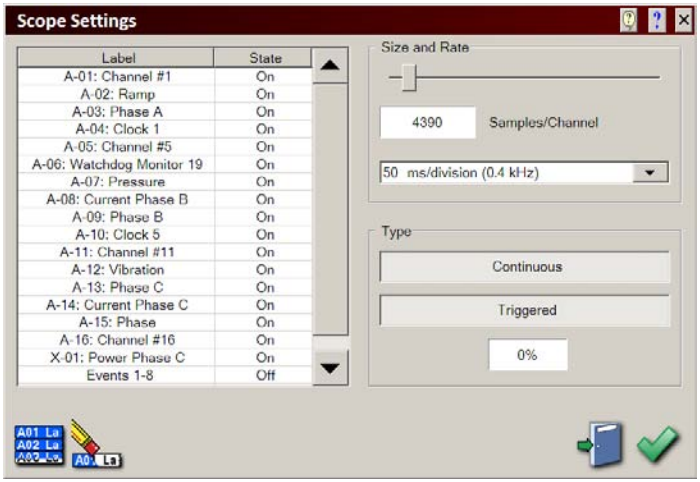
A scope capture is a high-speed snapshot of channel information. Every time you view data in Scope mode, you are viewing a scope capture. Scope captures are somewhat similar to data captures, and share much of the same terminology. For example, the concepts of arming, triggering, and aborting apply to scope captures as well as data captures.

Setting up a scope capture

1. If triggers or aborts will be used, set them up using the Trigger Settings window (Capture >> Trigger/Abort Settings).

   Note: *The trigger/abort setup process for scope captures is identical to the trigger/abort setup process for data captures.*

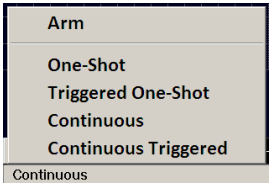
2. Choose Settings >> Scope. The Scope Settings window will open.



3. You can choose from four different types of captures by selecting various combinations of the Continuous and Triggered options.

   - **One-Shot** - Perform a one-time scope capture without using a trigger. (Select neither option)
   - **Continuous** - Perform multiple scope captures without using a trigger. (Select the Continuous option)
   - **Triggered One-Shot** - Perform a one-time scope capture that is initiated by a trigger. (Select the Triggered option)
   - **Triggered Continuous** - Perform continuous scope captures that are initiated by a trigger. (Select the Triggered and Continuous options)

   Note: *You can later change the capture type by choosing the Scope mode status bar along the bottom of the display.*



4. If the Triggered option is enabled, specify a pre-trigger percentage by selecting the percentage field. A number pad will appear. Enter the

percentage of the scope capture to record before the trigger and choose OK.

5. Select the channels to include in the scope capture. You can add or remove a channel from the capture by highlighting it, selecting the State column heading, and choosing On or Off.

6. Select a size for the scope capture by using the size slider bar. The currently selected number of samples will be displayed just below the slider. The maximum number of samples available will decrease as more channels are included in the capture.

   For an alternative method of selecting scope capture size, choose the Samples/Channel field. A number pad will appear. Enter the number of samples to include and choose OK.

   Scope capture size is based on the number of samples in the capture. If the scope capture exceeds the display size, a scrolling track view bar will appear on the bottom of the screen.

7. Select a time base from the drop-down list. The time base is specified in units of ms/Division or us/Division.

   Note: *You can later change the time base by choosing the Scope mode status bar along the bottom of the display.*

8. Choose OK.

Related topics:
- *Setting up a data capture* on page 9-4

Performing a scope capture

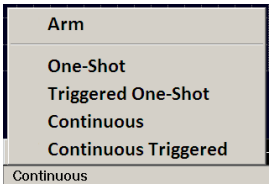
Scope captures are initiated using a similar process as data captures. The system is armed, pre-trigger data is collected (if triggers are used), a trigger occurs, and data is recorded until the capture is complete or aborted.

1. Set up the scope capture.

2. Arm the scope capture using the control panel or status bar.

- To use the control panel, choose the Scope Arm/Abort icon.

- To use the status bar along the bottom of the display, choose the Arm option.



3. The next system action depends on the type of scope capture being recorded. The following list describes the sequence of events for each scope capture type:

   - **One-Shot** - A one-time scope capture will occur and display when the scope capture is armed.

   - **Continuous** - Scope captures will occur and display continuously when the scope capture is armed.

   - **Triggered One-Shot** - The system will acquire pre-trigger data until all of the requested pre-trigger storage allocation is filled. When a trigger occurs, a one-time scope capture will occur and display.

   - **Triggered Continuous** - The system will acquire pre-trigger data until all of the requested pre-trigger storage allocation is filled. When a trigger occurs, a scope capture will occur and display. After each trigger, this process will be repeated.

4. To abort a scope capture, use the control panel or status bar.

   - To use the control panel, choose the Scope Arm/Abort icon.

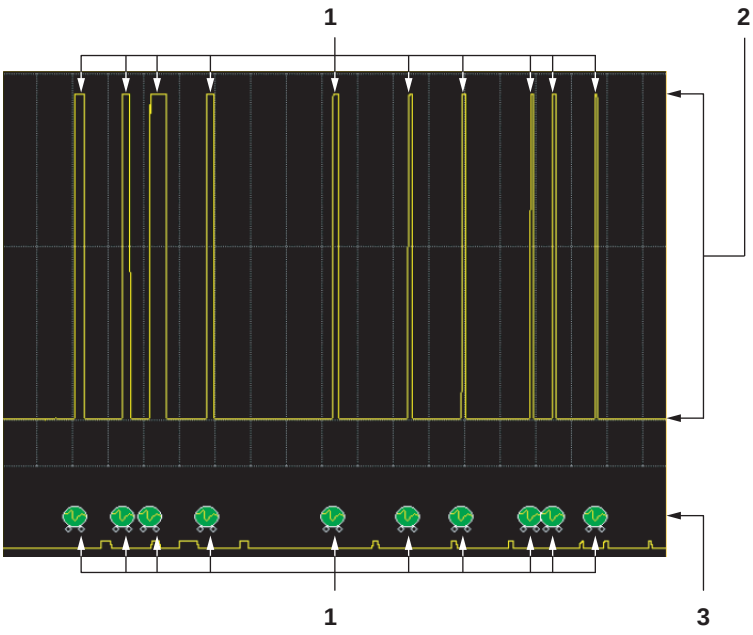
- To use the status bar along the bottom of the display, choose the Arm option.

The currently running scope capture will be aborted.

Embedded scope captures

Embedded scope captures allow you to save trend data at low sample rates while capturing transients at high sample rates. When you open the trend data capture in Review mode, the embedded scope captures will be viewable.

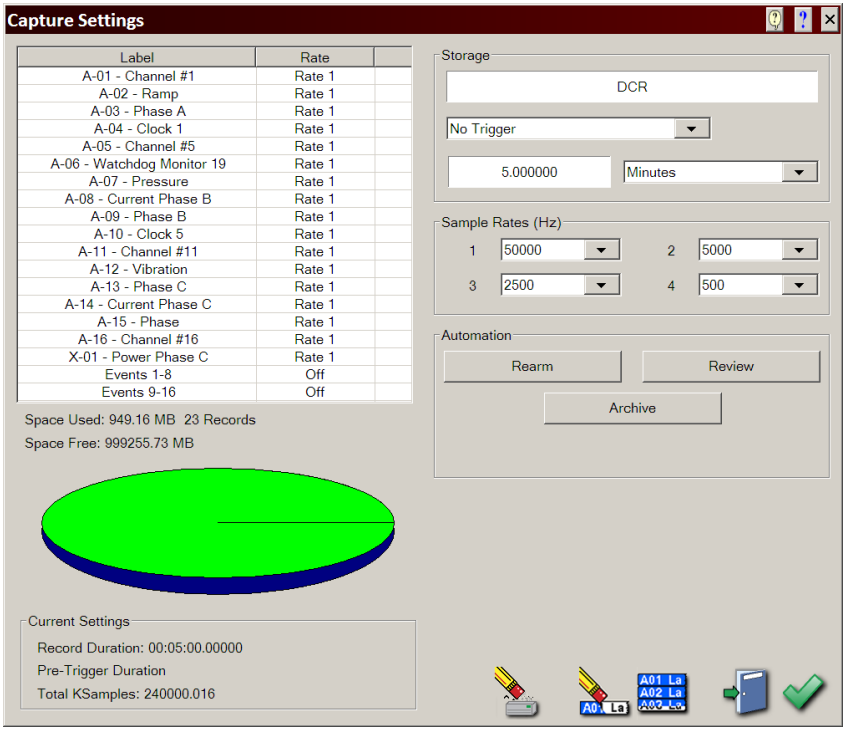
The following diagram illustrates the embedded scope capture concept. Trigger points initiate scope captures that are embedded into the trend data recording. During trend data review, you can open any of the embedded scope captures for high sample rate analysis.



#	Description
1	Transients
2	Trend data (low sample rate)
3	Embedded scope captures (high sample rate)

Setting up embedded scope captures

1. Choose Capture >> Settings. The Capture Settings window will open. You will use this window to set up the trend data capture.



2. Select the No Trigger option. Trend data captures do not use pre-trigger recording and they begin when manually armed. Triggers will later be used to initiate scope captures.

3. Specify the size of the trend data capture by defining a data capture storage allocation. The size can be defined in units of time (hours, minutes, or seconds) or number of samples (KS/channel).

4. Specify sample rates to define the speeds at which trend channels are sampled.

5. Select the channels to include in the trend data capture, and the sample rate for each channel.

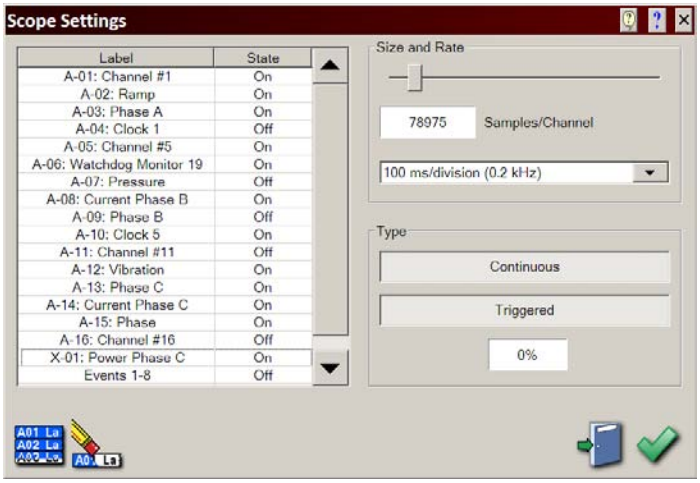
6. Choose OK to complete the trend data capture setup process.

7. Choose Capture >> Auto Save Scope with Capture. A check will appear next to this menu option to indicate it is enabled.

8. Set up the triggers (and aborts if necessary) using the Trigger Settings window (Capture >> Trigger/Abort Settings).

   In the case of embedded scope captures, the triggers are used to capture the high-speed transients and embed them into the slower trend data.

9. Choose Settings >> Scope. The Scope Settings window will open. You will use this window to set up scope captures that will be embedded in the trend data.



10. Select the Triggered and Continuous options.

11. Specify a pre-trigger percentage to indicate the percentage of the embedded scope capture to record before a trigger.

12. Select the channels to include in the embedded scope captures.

13. Select a size for the embedded scope captures by using the size slider bar or Samples/Channel field.

14. Select a time base for the embedded scope captures from the drop-down list.

15. When you are finished setting up the scope captures, choose OK to close the Scope Settings window.

16. Arm the scope capture using the control panel or status bar.

    - To use the control panel, choose the Scope Arm/Abort icon.

    - To use the status bar along the bottom of the display, choose the Arm option.

17. Arm the trend data by choosing Capture >> Arm. The trend data will begin recording.

    When the trend data meets the specified trigger conditions, a scope capture will be embedded into the trend data.

Related topics:
- *Setting up a scope capture* on page 10-4
- *Setting up a data capture* on page 9-4
- *Triggers and aborts* on page 9-7

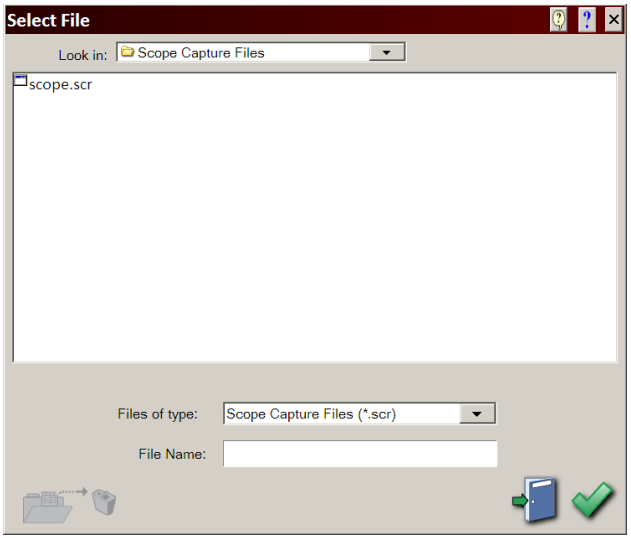
Archiving scope captures

Archiving a scope capture saves the capture to the system drive. This section provides instructions on archiving scope captures.

Archiving as data capture records in Scope mode

Use the following information to archive as data capture records. This binary format is preferable when using AstroVIEW X on a PC.

1. Choose File >> Archive as DCR. A sub menu will appear. Choose whether to archive the entire file, current page, or the area between cursors. The Select File window will open.



2. Choose a destination for the archived scope capture file. By default, the file will be saved on the system drive (C) in the selected folder. If necessary, you can save the file in a different location by choosing the folder list and browsing to a folder.

3. Choose the File Name field. A keypad will appear. Enter a file name and choose OK. The specified file name will appear in the field.

4. Choose OK.

Chapter 11: Review mode

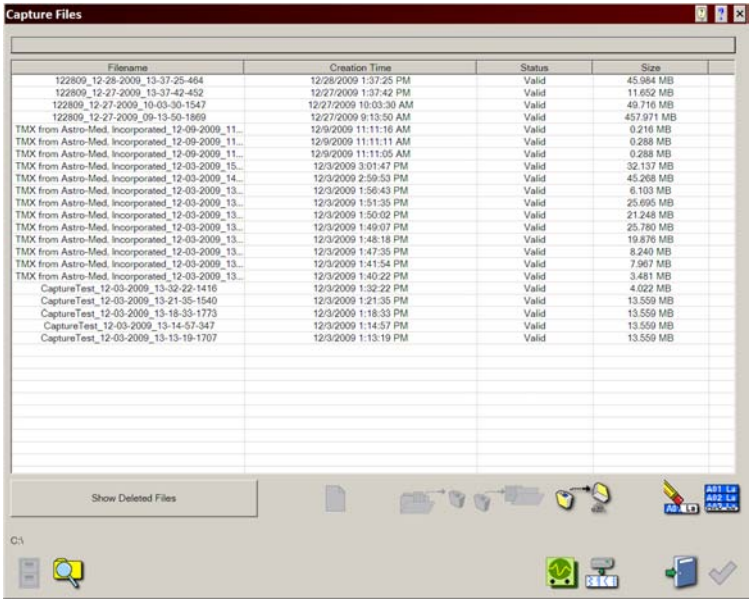
Review mode introduction

Review mode provides the capability to review and analyze saved data capture and scope capture files. It also provides file management features.

Accessing Review mode

1. Choose Configuration >> Review from the menu bar. A file selection window will open.

   Note: *You cannot use Review mode if a data capture is in progress.*



   This window provides a list of all files stored on the data capture drive.

2. Select a file to analyze in Review mode.

   - To open a data capture file from the data capture drive, select a file from the list and choose OK.

- To open an archived data capture file from the system drive, choose the Archived Data Capture icon.

  Select an archived data capture file and choose OK.

- To open a scope capture file from the system drive, choose the Scope Capture button.

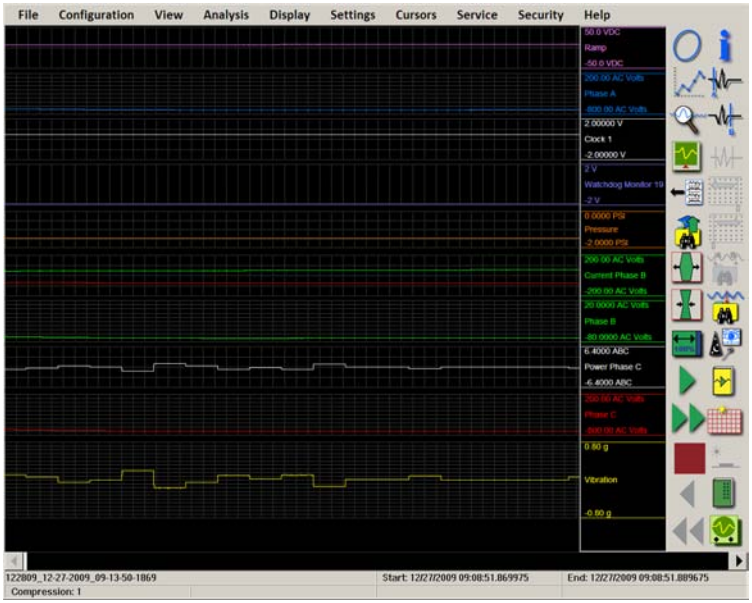
  Select a scope capture file and choose OK.

- To open a file located in a specific folder on the system drive, choose the Browse Folders icon.

  Browse to a file and choose OK.

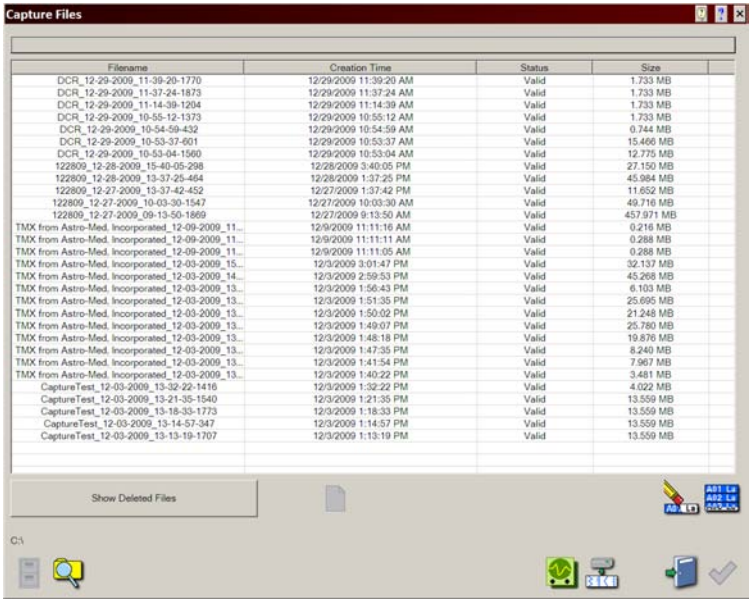
Review mode will start.

The following illustration displays a typical Review mode screen. Review mode screen appearances will vary based on the control panel configuration and other selected options.



Loading a new file

1. While in Review mode, choose File >> Load New File. A file selection window will open.



2. Select a file to open and choose OK.

Related topics:

- *Accessing Review mode* on page 11-1

Loading the next/previous file

You can quickly open the next or previous file on a specific drive during data capture review.

1. Set a sorting preference to determine the file next/previous sequence.

    - From Review mode, choose File >> Load New File to open the Review mode file selection window.

    - From any other mode of operation, choose Configuration >> Review to open the Review mode file selection window.

    Choose a column heading to apply a sorting order.

2. Open a file for review.

3. While reviewing the file, choose File >> Load Next File or Load Previous File to open the next or previous file.

   The next/previous file is determined by the last sorting order used in the Review mode file selection window for the drive.

Viewing file information

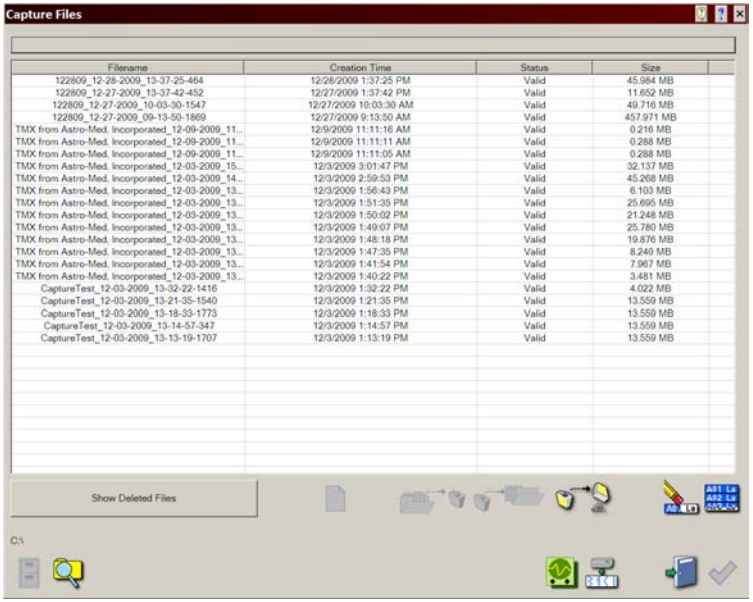
The File Summary window in Review mode provides additional information about capture files. It displays information including file name, number of samples, sample rates, data capture, and channel amplifier settings.

1. Open the File Summary window using one of the following methods.

   - With a file open in Review mode, choose File >> Summary.
   - From the Review mode file selection screen, select a file and choose the Summary button.

Deleting files

1. Choose Configuration >> Review from the menu bar. A file selection window will open.

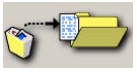


2. To send files to the trash, select the files and choose the Send to Trash icon.

The selected files will be transferred to the trash.

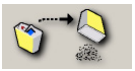
3. To retrieve files from the trash, choose the Show Deleted Files button. The file list will be refreshed to display deleted files. Then select the files from the list.

Choose the Retrieve from Trash icon.



The selected files will be retrieved from the trash.

4. To empty the trash and free the space it occupies on the data capture drive, choose the Empty Trash icon.



The trash will be emptied.

Note: *Emptying the trash permanently deletes all files currently in the trash. Use caution with this feature to ensure that needed files are not deleted. This process can take a significant amount of time. Ensure the unit can be allowed to finish without power interruption to prevent data loss.*

Review mode default control panel

The following list describes the functions of default control panel icons.

Note: *A selection path for each icon is indicated. When the Panel Settings window is open, you can use this path to add the icon to the control panel.*

Icon	Description
	**Scroll Fast Back** scrolls the chart backward quickly. *Scroll >> Rewind*
	**Scroll Back** scrolls the chart backward. *Scroll >> Scroll Backward*
	**Scroll Stop** stops the chart from scrolling. *Scroll >> Scroll Stop*
	**Scroll Forward** scrolls the chart forward. *Scroll >> Scroll Forward*
	**Scroll Fast Forward** scrolls the chart forward quickly. *Scroll >> Fast Forward*
	**Cursor A** displays and hides cursor A. *Cursors >> Cursor A*
	**Cursor B** displays and hides cursor B. *Cursors >> Cursor B*

Icon	Description
	**Active Cursor** changes the active cursor between A, B, or A & B. *Cursors >> Active Cursor*
	**Move Cursor Left** moves the active cursor(s) to the left each time the icon is pressed. *Cursors >> Move Left*
	**Move Cursor Right** moves the active cursor(s) to the right each time the icon is pressed. *Cursors >> Move Right*
	**Trigger Line** shows and hides the trigger line in the waveform display area. *View >> Trigger Line*

File scrolling and navigation

This section describes how to use the track view, control panel, and menu bar to scroll and navigate through capture files in Review mode.

Using the track view

The track view is a visual scroll bar located on the bottom of the display. It can be used to navigate to other areas of the capture.

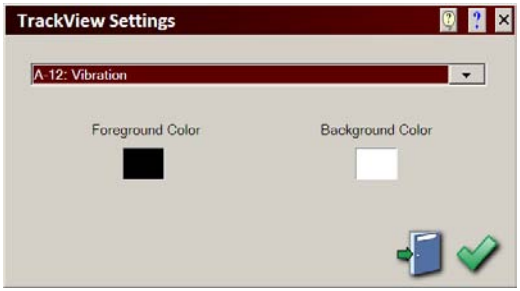
The track view displays a signal for a visual reference. The signal displayed in the track view can be changed, if desired. If the track view signal is changed, the selected signal will become the default until another signal is chosen.

1. Navigate though a file using the track view.

   - To scroll slowly through the file in a particular direction, choose one of the arrows on the sides of the track view.

   - To scroll quickly through the file in a particular direction, press and hold the shaded portion of the track view. Then drag it to the new location and release it. The file will scroll as the shaded bar is dragged.

2. If necessary, edit track view settings by choosing View >> Track View Channel. The TrackView Settings window will open.



   - Select a channel or event to display in the track view.
   - To select a track view foreground color, choose the Foreground Color box. The Color window will open. Select a color and choose OK.
   - To select a track view background color, choose the Background Color box. The Color window will open. Select a color and choose OK.

3. Choose OK.

File navigation control panel icons

The default control panel for Review mode provides a variety of buttons to scroll through the file.

Related topics:

- *Review mode default control panel* on page 11-7

File navigation menu options

The Display option on the menu bar provides the following methods of navigating through capture files.

- **Goto** scrolls directly to the start, end, trigger point, or cursor location in a file. An advanced search is also available that scrolls based on specified conditions.
- **Page Back** scrolls the chart back one page.
- **Page Forward** scrolls the chart forward one page.
- **Rewind** scrolls the chart backwards quickly.

- **Scroll Back** scrolls the chart backwards.
- **Scroll Stop** stops the chart from scrolling.
- **Scroll Forward** scrolls the chart forward.
- **Fast Forward** scrolls the chart forward quickly.

Many of the navigational features found on the menu bar are also included in the default control panel for Review mode.

Related topics:
- *Review mode default control panel* on page 11-7

File compression

Review mode displays files in a horizontally scrolling orientation. The file under review is typically larger than the display area, and scrolling is used to navigate through the file. Compression and expansion options are available to decrease or increase the horizontal size of a file.

Showing all of the file

The contents of a file can be compressed to fit the entire file in the display.

1. Choose Display >> Show All File. The file will be compressed to fit on one screen. The amount of compression will be displayed on the bottom of the screen.

Setting a compression

A specific compression ratio can be used to compress the file. The compression ratio is based on a factor of 1 to X, where X is that which will cause the entire file to fit on the display without scrolling. A compression ratio of 1 will result in no compression.

When a file is compressed, more of the entire file will fit on one screen, and less scrolling is necessary to navigate throughout the file.

Note: *Very large data captures may take some time to display if the compression value is high.*

1. Choose Display >> Compression. A number pad will appear.
2. Enter the desired compression ratio and choose OK. The file will be compressed.

Setting an expansion

A specific expansion ratio can be used to expand the file. The expansion ratio is based on a factor of 1 to 100. An expansion ratio of 1 will result in no expansion, while a ratio of 100 will expand the file by 100 times its original size.

When a file is expanded, less of the entire file will fit on one screen, and more scrolling is necessary to navigate throughout the file.

1. Choose Display >> Expansion. A number pad will appear.
2. Enter the desired expansion ratio and choose OK. The file will be expanded.

Review mode view options

This section describes how to configure the view and display options of Review mode.

Selecting a status text format

In the lower-right corner of the Review mode screen, a status text area is displayed.

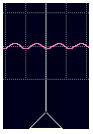
The type of information displayed in the status text area can be changed.

1. Choose View >> Status Text Format. A sub menu will appear.
2. Select a status text display option.
    - **Samples** - The sample numbers for the first and last samples displayed on the screen will be shown.
    - **Relative Time** - The times, relative to the start of the data capture, for the first and last samples displayed on the screen will be shown.
    - **Absolute Time** - The times, as recorded by the system clock, for the first and last samples displayed on the screen will be shown.
    - **Percent** - The percentage points, relative to the entire data capture, for the first and last samples displayed on the screen will be shown.

Displaying the trigger line

The trigger line is a vertical line drawn on the display that represents the trigger point of a capture. Use the following instructions to display the trigger line.

1. Choose View >> Show Trigger Line. The trigger line will be displayed.



2. To hide the trigger line, repeat this process.

Displaying channel IDs

Channel IDs are small visual indicators that identify channels. This feature is especially helpful for identifying multiple signals displayed in a capture.

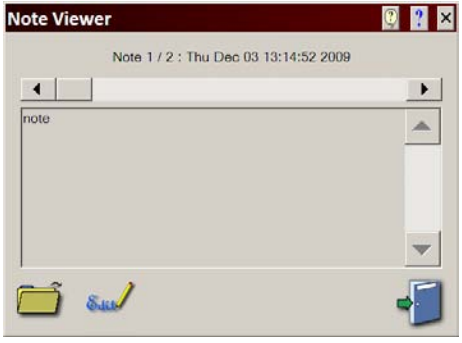
1. Choose View >> Show Channel IDs. Channel IDs will be displayed.

2. To hide channel IDs, repeat this process.

Viewing and editing notes

If notes are included in a capture, they can be viewed and edited in Review mode.

1. Chose Analysis >> Note Viewer. The Note Viewer window will open.



As an alternate method of opening the Note Viewer window, scroll to and choose a pencil icon in the file.

2. Use the scroll bar on the top of the window to view other notes located in the file.

   If necessary, choose the Find note button to scroll the file and display the area containing the selected note.

3. To edit the note, choose the Edit Note icon. A keypad will appear. Edit the note and choose OK.

Adding notes

You can add notes to files in Review mode.

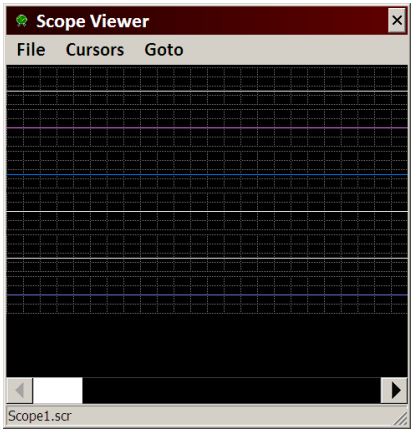
1. Choose Analysis >> Add Note. A sub menu will appear. Choose whether to add the note at the center of the screen or a cursor location. A keypad will appear.

2. Enter the note text and choose OK. The note will be saved as part of the capture file.

Viewing embedded scope captures

If embedded scope captures are included in a capture, they can be viewed in Review mode. You can view embedded scope captures in a window within Review mode, or in the full Review mode screen itself.

1. **To view embedded scope captures in the Scope Viewer window**, choose Settings >> Scope Viewer >> Viewer. Then choose Analysis >> Scope Viewer to open the Scope Viewer window.



   As an alternate method of opening embedded scope captures, scroll to and choose a scope icon in the file.

   - If necessary, choose the Cursors menu option to use cursors. The cursors in this window function similar to cursors in other functions.
   - If necessary, choose Goto >> Scope Location from the menu to scroll to the scope location in the file.
   - If necessary, move to the next or previous scope capture in the file by choosing File >> Load Next File or Load Previous File.
   - Choose the X in the upper-right corner to close the Scope window.

2. **To view embedded scope captures in the full Review mode screen**, choose Settings >> Scope Viewer >> Full Screen. Then choose Analysis >> Scope Viewer to review the embedded scope captures.

As an alternate method of opening embedded scope captures, scroll to and choose a scope icon in the file.

- While reviewing an embedded scope capture in the Review mode screen, the Show Next File and Show Previous File functions can be used to navigate to other embedded scope captures in the current data capture.
- When you are finished reviewing the embedded scope capture, choose File >> Return to DCR, or the Return to DCR control panel button, to close the scope capture and return to the original data capture file.

Saving the Review mode view as default

When you open a file in Review mode, the view will be the same as when the file was captured. You can modify the view and save it as the default for future review sessions with that file.

1. Choose File >> Save View as Default. The view will be saved.

   The next time the current file is opened in Review mode, this saved view will be used.

Review mode channel setup

This section describes how to modify channel settings for files during review.

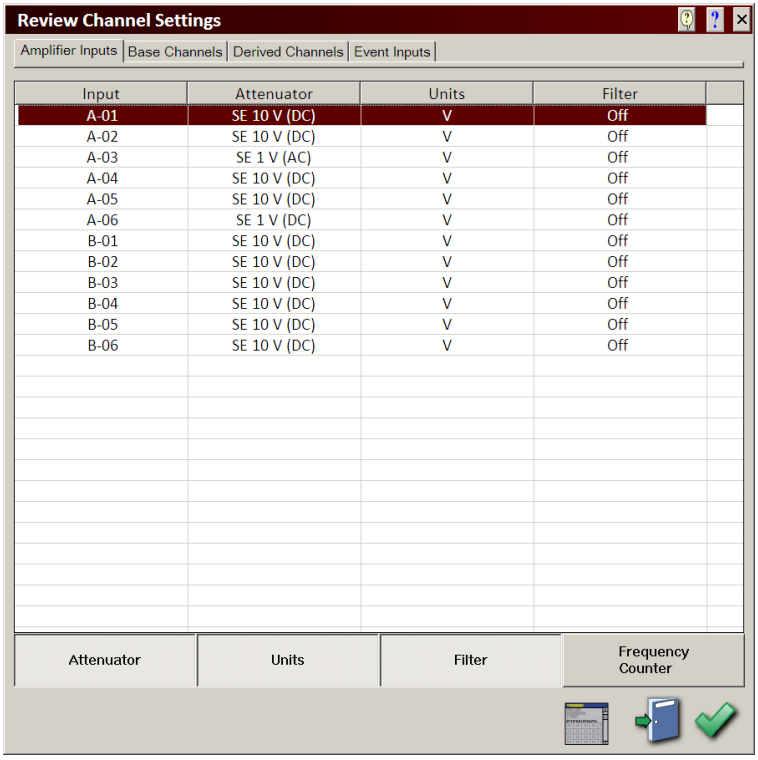
Review Channel Settings window overview

The Review Channel Settings window (Settings >> Channels) is used to set up channels and events in Review mode. The Review Channel Settings window provides two layout options.

- **List view** - You can change settings in list view by selecting a channel and choosing the column heading for the setting you want to change. Some options can be configured for a group of channels at the same time by

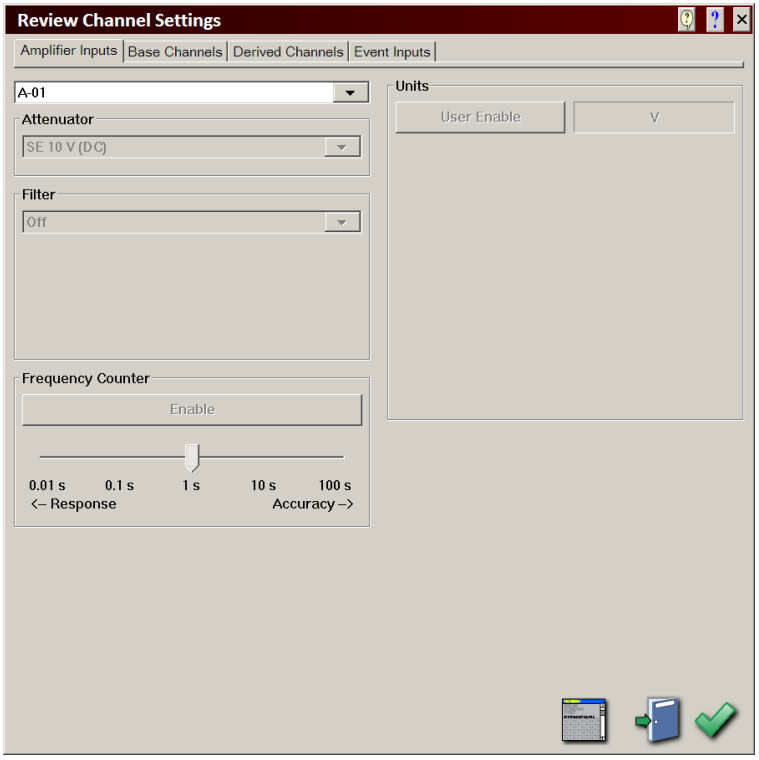
selecting a group of channels prior to choosing a column heading. In this case, setup information will be applied to all selected channels.

Some options must be configured for each channel individually, or in groups based on compatible channels.



Use the buttons below the list to display other columns.

- **Graphical view** - You can change settings in graphical view by selecting a single channel and the appropriate options for the channel.



You can switch between list view and graphical view by choosing the Toggle Layout icon.

Some items in the Review Channel Settings window cannot be changed. The following items are saved as part of a data capture and cannot be edited in Review mode.

- Amplifier Inputs - Items in this tab cannot be modified.
- Base Channels - Top/Bottom and Center/Span settings cannot be modified.

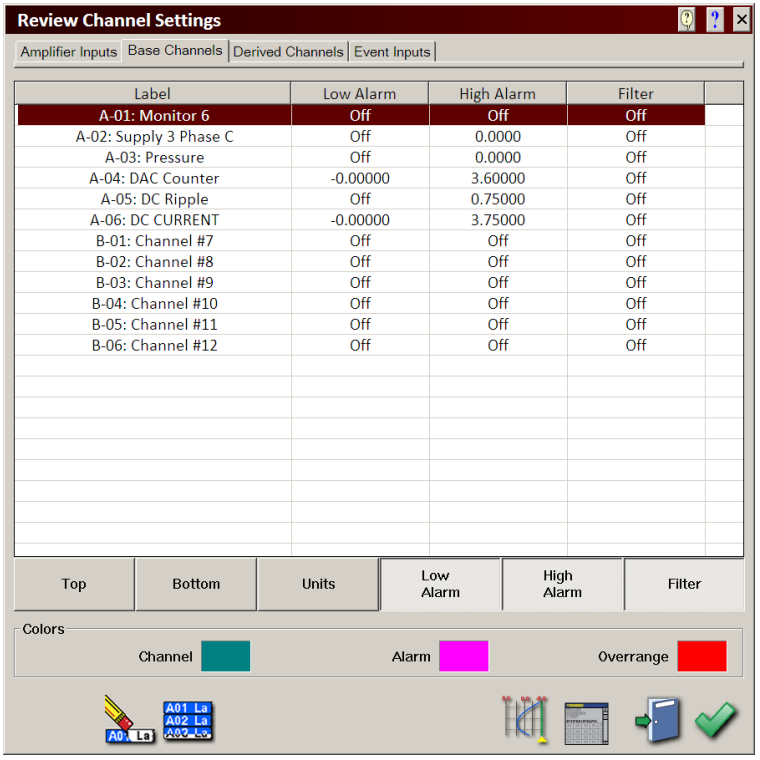
Setting up Review mode filters

Filters are used to reduce the amplitude of certain waveforms based on their frequency. This allows you to restrict your data to certain frequencies and exclude extraneous data. Two types of filters are available.

- **Amplifier input filters** are hardware based and affect the input data before it is captured. These filters are permanent and cannot be undone once the data is captured.
- **Review mode filters** are software based and affect how the data is viewed in Review mode. They temporarily modify the view without altering the captured data.

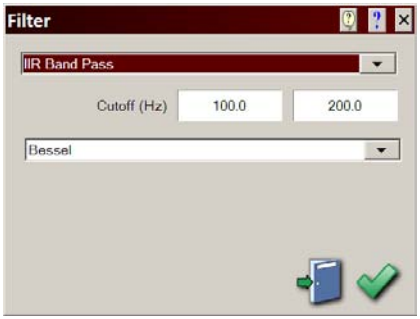
This section describes how to set up Review mode filters.

1. Choose Settings >> Channels. The Review Channel Settings window will open. Choose the Base Channels tab.



2. Select the channel or channels to apply a filter to. Note that more than one channel can be selected at a time. Any changes made will affect all selected channels.

3. Choose the Filter column heading. The Review mode Filter window will open.



4. Select an **IIR** (Infinite Impulse Response) or **FIR** (Finite Impulse Response) filter type.

   - IIR filters perform faster, and are generally easier to construct, but can suffer from non-linear phase response.

   - FIR filters have a known, linear phase response. This allows the phase change that results from filtering to be easily compensated for. The drawback is that they require more resources for similar frequency responses in smaller IIR filters.

   The following filter types are available.

   - **Low Pass** - Filters out frequencies above a specified cutoff point.

   - **High Pass** - Filters out frequencies below a specified cutoff point.

   - **Band Pass** - Creates a window based on two specified cutoff points, identified as upper and lower frequency boundaries. Anything outside this window is excluded.

   - **Band Stop** - Creates a window based on two specified cutoff points, identified as upper and lower frequency boundaries. Anything inside this window is excluded.

5. Select a filter topology.

- **Bessel** filters are typically characterized by a nearly linear phase response in the pass band. They are commonly used in applications where little phase distortion is required. The trade off for this is a gentler roll off around the cutoff frequency.

- **Butterworth** filters produce the most "ideal" response, generating maximum flatness and unity gain in the pass band, and monotonic decrease of frequency after the cutoff.

- **Chebyshev** filters feature the sharpest transition band, but will have gain ripple in the pass band. A 4th order Chebyshev with 3 db of ripple, for instance, will drop at 100 db per decade.

  If the Chebyshev topology is used, the Ripple field will appear. Enter a desired ripple value from 0.1 to 10.0 into this field.

6. Enter a cutoff frequency. The available range of this value will vary, depending on the current filter type. If the Band Pass or Band Stop filter types are used, you'll need to enter two cutoff frequencies; an upper boundary and a lower boundary.

   Setting a cutoff value to 500 Hz or greater will result in a 4th order filter; a cutoff of less than 500 Hz will result in a 1st order filter.

7. Choose OK.

Related topics:
- *Setting up amplifier input filters* on page 6-10

Review mode derived channels

Derived channels are saved as part of data captures, however, derived channel data is not saved directly. Instead, the following items are saved.

- The derived channel equation

- The channels that are used in the derived channel equation

This allows you to review derived channels and modify their equations during review. You can also create new derived channels based on channels in the capture.

Note: *If you modify derived channel equations in Review mode, and then use the Save Channels as Default feature, the original equations will be overwritten.*

Related topics:
- *Derived channels* on page 6-16

• *Saving the Review mode channels as default* on page 11-21

Saving the Review mode channels as default

When you open a file in Review mode, the channel settings will be the same as when the file was captured. You can modify the channel settings and save them as the default for future review sessions with that file.

1. Choose File >> Save Channels as Default. The channel settings will be saved.

   The next time the current file is opened in Review mode, the saved channel settings will be used.

Archiving files in Review mode

Archiving a file copies the file from the data capture drive to the system drive. Once files are archived, they can be copied, moved, and deleted using the Microsoft ® Windows operating system. Additionally, files must be archived in order to access them via Ethernet from a PC.

Note: *Archiving files does not remove them from the data capture drive.*

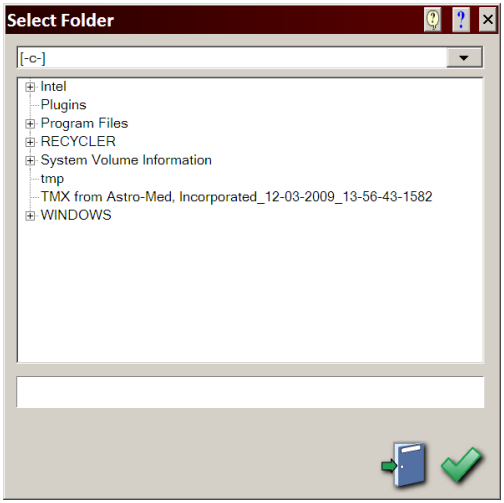
Archiving as data capture records in Review mode

Use the following information to archive as data capture records. This binary format is preferable when using AstroVIEW X on a PC.

Note: *Files can also be archived as data capture records in the file selection window prior to accessing Review mode. Choose the folder icon to select an archive location.*

Then select the files to archive and choose the file cabinet icon button. This method is helpful for archiving multiple files at the same time.

1. Choose File >> Archive as DCR. A sub menu will appear. Choose whether to archive the entire file, current page, or the area between cursors. The Select Folder window will open.



2. Select an archive drive with the drop-down list. A folder list will appear. Browse to the archive folder and choose OK. A keypad will appear.

3. Enter a name for the archive file and choose OK. The archive file will be saved.

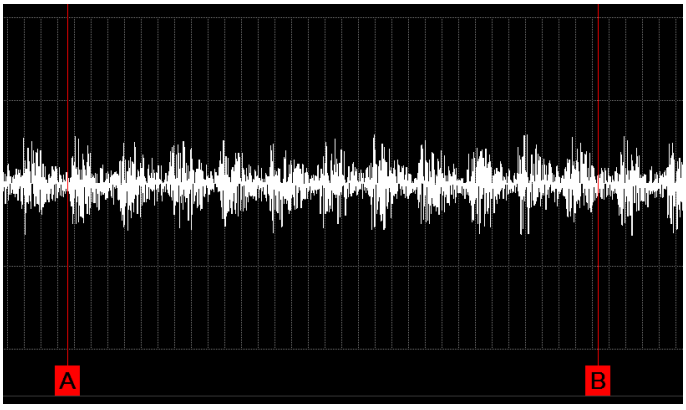
Chapter 12: Analysis tools

Cursors

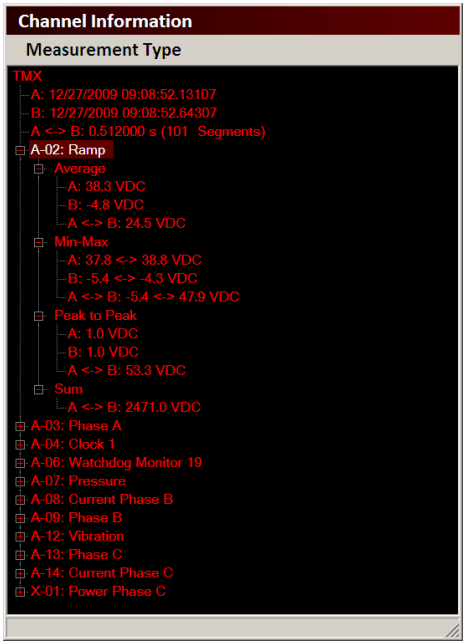
Cursors can be used to measure channels in Realtime, Scope, and Review modes. They are used in conjunction with the Channel Information window to view a variety of measurements.

Measuring channels with cursors

1. Show one or both of the cursors by choosing Cursors >> Show Cursor A and/or Cursors >> Show Cursor B from the menu bar.



2. Choose Analysis >> Channel Information to open the Channel Information window.



The Channel Information window displays the waveform values at each cursor based on the current measurement type.

3. Choose the Measurement Type menu option. Select a measurement type from the list.

   - **Average** displays the midpoint value of the data represented by the cursor. When using two cursors, it displays the average of the midpoint values of the data between cursors A and B (including cursor data).

   - **Minimum-Maximum** displays the maximum and minimum signal values of the data represented by the cursor. When using two cursors, it displays the maximum and minimum signal values represented by the data between cursors A and B (including cursor data).

   - **Peak-Peak** displays the difference between the maximum and minimum signal measurements between cursors A and B (including cursor data). This measurement is available only when both cursors are displayed.

- **Slope** uses linear regression of the average of the maximum and minimum of each point between cursors A and B (including cursor data) to create a line which best represents the data. This measurement is available only when both cursors are displayed.
- **RMS** displays the square root of the sum of squares divided by the number of samples between cursors A and B (including cursor data). This measurement is available only when both cursors are displayed.
- **Sum** displays the sum of the average of the maximum and minimum of each point between cursors A and B (including cursor data). This measurement is available only when both cursors are displayed.
- **Sum of Squares** displays the sum of the average of the maximum and minimum signal values represented by the data between cursors A and B (including cursor data) squared. This measurement is available only when both cursors are displayed.
- **Variance** measures how the set of data between cursors A and B (including cursor data) is dispersed about the mean.

$$\text{Variance} = (\Sigma y^2 - ((\Sigma y)^2 / n)) / n$$

Where y = sample value (average of maximum and minimum), n = number of points between cursors.

- **Standard Deviation** displays the square root of the variance. This measurement is available only when both cursors are displayed.
- **Area** displays the area under the curve. This measurement is available only when both cursors are displayed.

$$\text{Area} = \sum xy$$

Where x = time delta for a sample and y = sample value (average of maximum and minimum)

4. If necessary, move one or both of the cursors. Cursors must be activated before they can be moved. In the following illustration, cursor A is active and cursor B is inactive.

Activate the cursors you want to move by choosing Cursors >> Active >> A or B. You can also choose the cursor labels to quickly activate/deactivate the cursors.

Move the active cursor or cursors by touching and dragging the bottom of the waveform display area. Ensure you do not touch a cursor label, as the cursor's active status will be changed. Instead, touch an area to the side of the cursor labels.

As cursors are moved, the values in the Channel Information window will update based on the signal values at the new cursor locations.

Note: *In Realtime mode, cursors cannot be moved outside of the viewing window.*

5. To modify the cursor color, choose Cursors >> Color. The Color window will open. Select a color and choose OK.

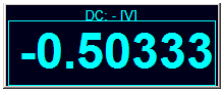
Channel meters

Channel meters provide a variety of ways to visually indicate channel activity.

Channel meter types

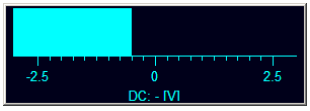
Numeric

The numeric meter displays the channel value.



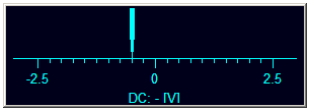
Horizontal Bar

The horizontal bar meter visually represents channel activity in a bar format.



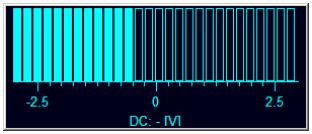
Horizontal Needle

The horizontal needle meter visually represents channel activity with a moving needle.



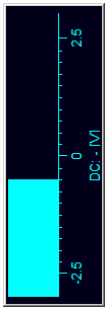
Horizontal LED

The horizontal LED meter visually represents channel activity with LED style bars.



Vertical Bar

The vertical bar meter visually represents channel activity in a bar format.

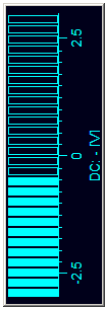


Vertical Needle

The vertical needle meter visually represents channel activity with a moving needle.

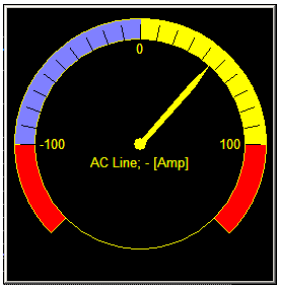
Vertical LED

The vertical LED meter visually represents channel activity with LED style bars.



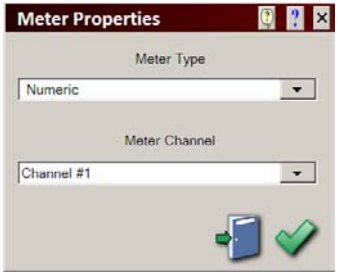
Gauge

The gauge meter visually represents channel activity with a rotating needle.



Adding channel meters

1. Choose Analysis >> Meter. The Meter Properties window will open.



2. Select a meter type and the channel that will be displayed in the meter. Choose OK. The meter will be added to the display.

Editing channel meters

1. Choose the channel meter you want to edit. A sub menu will appear.

   - To change the meter type or channel, choose Properties. The Meter Properties window will open. Edit the meter options and choose OK.

   - To change the measurement point in Scope and Review modes, choose Screen Left, Cursor A, or Cursor B. The cursor options are available only if cursors are displayed.

Moving channel meters

1. Choose the channel meter you want to move. A sub menu will appear. Choose Move.

2. Touch the meter and drag it to the new location. Release the meter to lock it into position.

Resizing channel meters

1. Choose the channel meter you want to resize. A sub menu will appear.

   Note: *Numeric meters cannot be resized.*

2. Resize the meter using one of the following options.

   - **Size Left** - Choose this option to resize by moving the left side of the meter. Touch within the meter and drag horizontally to move the left side.

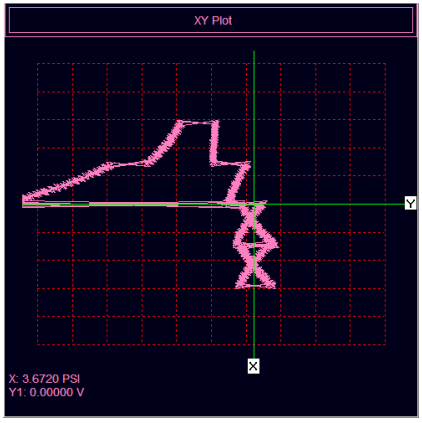
- **Size Right** - Choose this option to resize by moving the right side of the meter. Touch within the meter and drag horizontally to move the right side.
- **Size Top** - Choose this option to resize by moving the top of the meter. Touch within the meter and drag vertically to move the top.
- **Size Bottom** - Choose this option to resize by moving the bottom of the meter. Touch within the meter and drag vertically to move the bottom.

Removing channel meters

1. Choose the channel meter you want to remove. A sub menu will appear.
2. Choose Close.

XYY plots

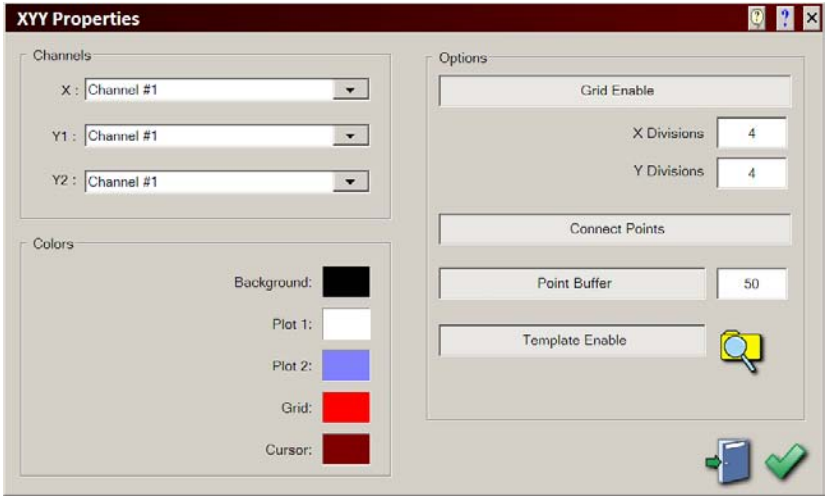
XYY plots are available for viewing one X channel and up to two Y channels.



Note: *The Review mode plot is generated based upon points in the DCR file being reviewed. When cursors are enabled, however, the plot will be generated based upon review mode screen points.*

Adding XYY plots

1. Choose Analysis >> XYY Plot. The XYY Properties window will open.



2. Use the X, Y1, and Y2 options to select channels to display in the XYY plot.

3. Select a color scheme for the XYY plot.

   - To specify a background color, choose the Background color box. The Color window will open. Select a color and choose OK.
   - To specify a color for plot 1, choose the Plot 1 color box. The Color window will open. Select a color and choose OK.
   - To specify a color for plot 2, choose the Plot 2 color box. The Color window will open. Select a color and choose OK.
   - To specify a color for the grid, choose the Grid color box. The Color window will open. Select a color and choose OK.
   - To specify a color for cursors, choose the Cursor color box. The Color window will open. Select a color and choose OK.

4. To enable a grid in the XYY plot, choose the Grid Enable option. Grid X and Y division fields will appear. Enter the number of divisions to display in the grid.

5. To connect the sample points drawn on the plot with line segments, choose the Connect Points option.

6. To enable the point buffer, which simulates persistence, choose the point buffer option. A point buffer size field will appear. Enter a point buffer size to indicate how many data points should be used to make the plot.

7. Choose OK. The XYY plot will be displayed.

8. If necessary, you can clear the plot by choosing the top area of the XYY plot. A sub menu will appear. Choose Clear.

9. If necessary, you can print the plot by choosing the top area of the XYY plot. A sub menu will appear. Choose Print.

Editing XYY plots

1. Choose the top area of the XYY plot. A sub menu will appear.

2. Choose Properties. The XYY Properties window will open.

3. Edit the XYY plot options and choose OK.

Moving XYY plots

1. Choose the top area of the XYY plot. A sub menu will appear. Choose Move.

2. Touch the XYY plot and drag it to the new location. Release the XYY plot to lock it into position.

Resizing XYY plots

1. Choose the top area of the XYY plot. A sub menu will appear.

2. Resize the XYY plot using one of the following options.

    - **Size Left** - Choose this option to resize by moving the left side of the plot. Touch within the plot and drag horizontally to move the left side.

    - **Size Right** - Choose this option to resize by moving the right side of the plot. Touch within the plot and drag horizontally to move the right side.

    - **Size Top** - Choose this option to resize by moving the top of the plot. Touch within the plot and drag vertically to move the top.

    - **Size Bottom** - Choose this option to resize by moving the bottom of the plot. Touch within the plot and drag vertically to move the bottom.

Using XYY plot cursors

1. Choose the top area of the XYY plot. A sub menu will appear.

2. Choose Cursor X or Cursor Y to toggle the cursor display.

   Cursor measurements will be displayed in the bottom of the XYY plot.

   Note: *In Review mode, the cursor information shown at the bottom of the review screen and the Channel Information window will reflect the cursor measurements in either the XYY Plot, Zoom, or Fourier Transform window that is active and selected. If you have cursors on the review screen and the XYY Plot, Zoom, or Fourier Transform windows, you can toggle between measurements by touching the review screen and the appropriate analysis window.*

3. To move cursors, choose the top area of the XYY plot. Then choose Move Cursor X or Move Cursor Y. The cursor color will change.

   Touch and drag in the XYY plot to move the cursor to a new position, and then release it. To lock the cursor into position, choose the top area of the XYY plot or wait five seconds.

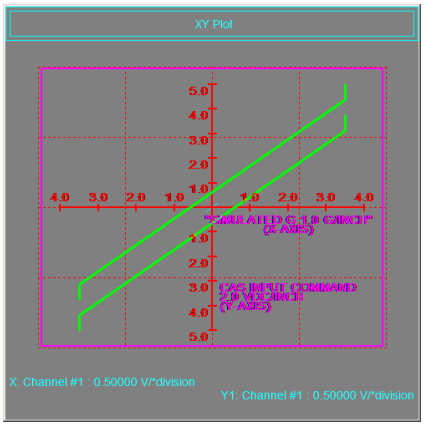
Removing XYY plots

1. Choose the top area of the XYY plot. A sub menu will appear.

2. Choose Close.

XYY plot templates

XYY plot templates allow you to display a customized background template in the XYY plot window. This background is a visual aid you can use for comparing plot results against a standard template you define.



Creating XYY plot templates

An XYY plot template file (*.xyt) specifies text strings and a series of curves created by connecting points. Each curve or text string is positioned based on a coordinate system. The template file creates an image to overlay on an XYY plot.

1. Use a plain text editor, such as Notepad, on a personal computer to create and save a template file. Save the file with the extension XYT.

2. Define the scaling used in the file by specifying the minimum and maximum values for both the X axis and Y axis. Values range from -32767 to +32767. Refer to the following example:

```
[Scale]
Min=-32767
Max=32767
```

3. Define the color that will be used as the transparent color. Choose a color that is unique to all colors used for curves or text. The color is specified in RGB format with values ranging from 0 to 255. Refer to the following example:

```
[Transparent]
Color=55,55,55
```

4. Enter curve information in the template file. You can enter up to ten curves. Refer to the following example:

```
[Curve 1]
Thickness=2
Color=255,0,0
Pt1=-26214,-655
Pt2=-26214,655
Pt3=-26214,0
```

- **[Curve Number]** - Enter the curve number between brackets. Curve numbers are specified in the following format: [Curve 1], [Curve 2], [Curve 3], etc. up to [Curve 10].
- **Thickness** - Enter a thickness value for the curve. Curve thickness is specified in pixels and ranges from 1 to 50.
- **Color** - Enter a color for the curve. The color is specified in RGB format with values ranging from 0 to 255.
- **Points** - Enter a list of points to describe the curve. Up to 300 individual points can be specified. Points are specified in the following format: Ptn=x,y

  Values for n range from 1 to 300. Values for x and y range from -32767 to +32767.

  The points are to be scaled from -32767 to +32767 and are contained in the first quadrant (both X and Y positive).

5. Enter text fields in the template file. You can enter up to 100 text fields. Refer to the following example:

```
[Text 1]
Color=255,0,0
Horizontal=0
Vertical=-1
Pt=-26214,786
String=4.0
```

- **[Text Number]** - Enter the text field number between brackets. Text field numbers are specified in the following format: [Text 1], [Text 2], [Text 3], etc. up to [Text 100].
- **Color** - Enter a color for the text. The color is specified in RGB format with values ranging from 0 to 255.

- **Horizontal** - Specify the horizontal alignment of the text: (-1) left justified, (0) centered, (1) right justified.
- **Vertical** - Specify the vertical alignment of the text: (-1) bottom, (0) centered, (1) top.
- **Pt** - Enter the point at which to position the text field. Points are specified in the following format: Pt=x,y

  Values for x and y range from -32767 to +32767. Alignment is based on this point.
- **String** - Enter the text string to display in the field. You can enter up to 100 characters.

6. Save the template file. Ensure you save it with a XYT extension. Then copy it to a USB-based memory storage device.

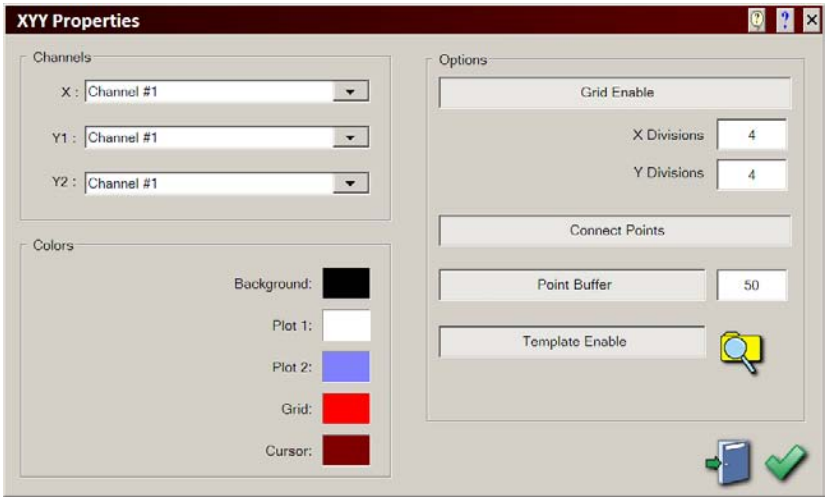
Loading XYY plot templates

1. Insert the USB-based memory storage device that contains the template (*.xyt) file into the recorder USB port.

   Note: *If you plan to use the template file often, you may want to copy it to the C:\tmx\XY Templates folder using the Operating System utility.*

2. Choose whether to add an XYY plot template to a new XYY plot or an existing one.
   - To use an XYY plot template on a new plot, choose Analysis >> XYY Plot. The XYY Properties window will open.

- To use an XYY plot template on an existing plot, choose the top area of the XYY plot. A sub menu will appear. Choose Properties. The XYY Properties window will open.



3. Configure any other XYY template options if needed.

4. Choose the Template Enable option. Then choose the Browse Folders icon and select an XYY template to load. By default, files in the default location on the system drive (C) will be displayed. If necessary, you can select a file from a different location by choosing the folder list and browsing to a folder. Choose OK.

5. Choose OK in the XYY Properties window. The XYY plot will be displayed with the selected template.

Adjusting XYY plot templates

1. Choose the top area of the XYY plot. A sub menu will appear.

2. To move the template, choose Shift Template.

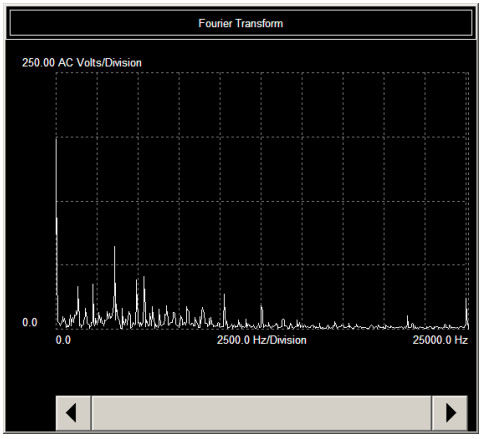
   Touch and drag in the XYY plot to move the template to a new position, and then release it. To lock the template into position, choose the top area of the XYY plot or wait five seconds.

3. To rotate the template, choose Rotate Template.

Touch and drag in the XYY plot to rotate the template, and then release it. To lock the template into position, choose the top area of the XYY plot or wait five seconds.

Fourier Transform window

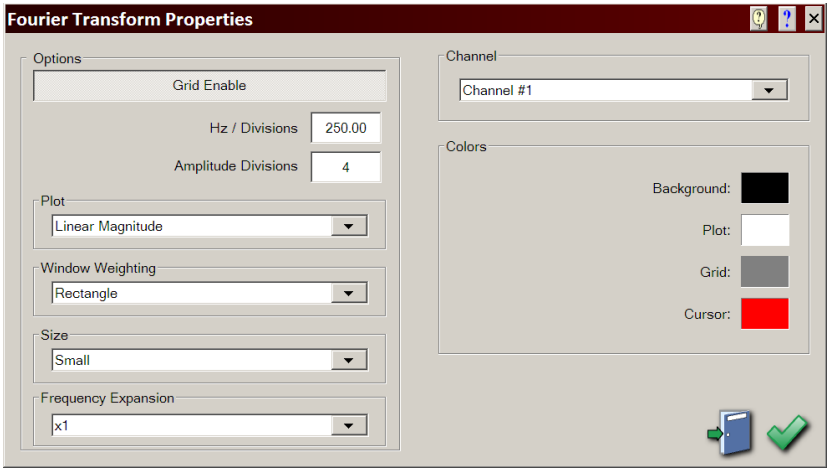
The Fourier Transform is created from the data displayed in the waveform display area. This data can consist of up to 1024 segments. Each segment consists of two points, a minimum and maximum, which represent the minimum and maximum signal values for each period.



Fourier Transform calculations require a single point for each period. The point used for the Fourier Transform calculation is the midpoint of the minimum and maximum. Therefore, a signal overlay compressed may not yield accurate Fourier Transform results.

Adding Fourier Transform windows

1. Choose Analysis >> Fourier Transform. The Fourier Transform Properties window will open.



2. Select the channel to display in the Fourier Transform window.

3. To display a grid, choose the Grid Enable option. Then specify Hz/divisions and amplitude divisions values.

4. Use the plot option to choose between linear magnitude, logarithmic magnitude, or magnitude^2.

5. Choose a window weight option. The following options are available: Hanning, Hamming, Blackman, Barlett, Triangle, Kaiser, Bman-Harris, and Rectangle.

6. Select whether to display a small, medium, or large size Fourier Transform window.

7. If necessary, use the Frequency Expansion option to expand the frequency axis and provide a more detailed view.

8. Select a color scheme for the Fourier Transform window.

   - To specify a background color, choose the Background color box. The Color window will open. Select a color and choose OK.

   - To specify a plot color, choose the Plot color box. The Color window will open. Select a color and choose OK.

- To specify a color for the grid, choose the Grid color box. The Color window will open. Select a color and choose OK.
- To specify a color for cursors, choose the Cursor color box. The Color window will open. Select a color and choose OK.

9. Choose OK. The Fourier Transform window will be displayed.
10. If necessary, you can print the Fourier Transform by choosing the top area of the Fourier Transform window. A sub menu will appear. Choose Print.

Editing Fourier Transform windows

1. Choose the top area of the Fourier Transform window. A sub menu will appear.
2. Choose Properties. The Fourier Transform Properties window will open.
3. Edit the Fourier Transform window options and choose OK.

Moving Fourier Transform windows

1. Choose the top area of the Fourier Transform window. A sub menu will appear.
2. Touch the Fourier Transform window and drag it to the new location. Release the window to lock it into position.

Resizing Fourier Transform windows

1. Choose the top area of the Fourier Transform window. A sub menu will appear.
2. Choose Small Size, Medium Size, or Large Size to resize the window

Using Fourier Transform window cursors

1. Choose the top area of the Fourier Transform window. A sub menu will appear.
2. Choose Cursor A or Cursor B to toggle the cursor display.

   Cursor measurements will be displayed in the bottom of the Fourier Transform window.

   Note: *In Review mode, the cursor information shown at the bottom of the review screen and the Channel Information window will reflect the cursor measurements in either the XYY Plot, Zoom, or Fourier Transform window that is active and selected. If you have cursors on the review screen and the XYY Plot,*

Zoom, or Fourier Transform windows, you can toggle between measurements by touching the review screen and the appropriate analysis window.

3. To move cursors, choose the top area of the Fourier Transform window. Then choose Move Cursor A or Move Cursor B. The cursor color will change.

   Touch and drag in the Fourier Transform window to move the cursor to a new position, and then release it. To lock the cursor into position, choose the top area of the Fourier Transform window or wait five seconds.

Removing Fourier Transform windows

1. Choose the top area of the Fourier Transform window. A sub menu will appear.
2. Choose Close.

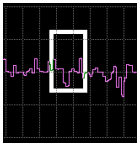
Zoom window

The Zoom window magnifies a portion of a capture for detailed analysis. Data displayed in the Zoom window is generated based upon points in the DCR file being reviewed.

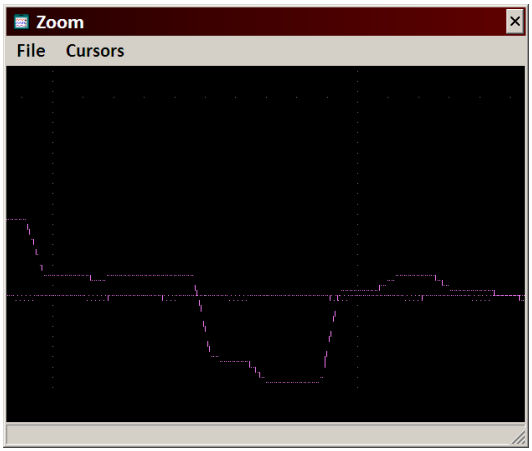
Adding Zoom windows

1. Choose Analysis >> Zoom. The Select Zoom Area window will open.

2. Create the zoom area by pressing anywhere on the waveform display area. While pressing, drag diagonally away from the first point to create a rectangular area.



3. Choose OK in the Select Zoom Area window. The Zoom window will open.



The Zoom window displays the rectangular area selected. Cursors are available via the Cursors menu.

Note: *In Review mode, the cursor information shown at the bottom of the review screen and the Channel Information window will reflect the cursor measurements in either the XYY Plot, Zoom, or Fourier Transform window that is active and selected. If you have cursors on the review screen and the XYY Plot, Zoom, or Fourier Transform windows, you can toggle between measurements by touching the review screen and the appropriate analysis window.*

4. If necessary, you can scroll the zoom area by pressing and dragging the outlined area in the waveform display.

5. If necessary, you can print the zoom area by choosing File >> Print in the Zoom window.

Chapter 13: Networking and communications

IP information

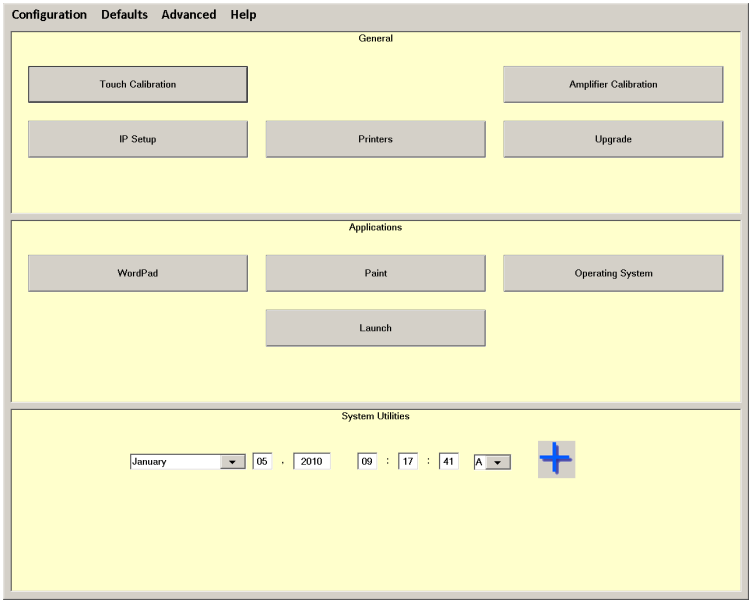
The recorder can be connected to a network and assigned an IP address for TCP/IP communication. The factory default IP settings are indicated below.

- IP Address: 192.168.255.1
- Subnet Mask: 255.255.255.0
- Gateway: None

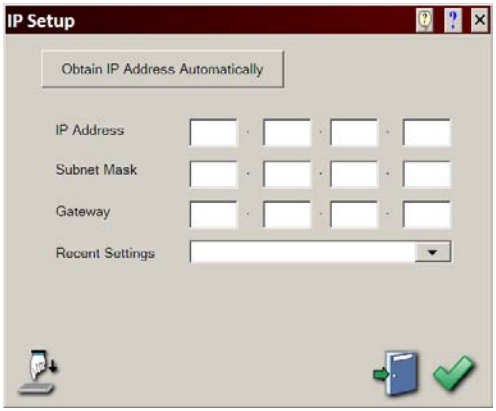
Modifying IP information

The recorder can be assigned a specific IP address or it can obtain one automatically using Dynamic Host Configuration Protocol (DHCP).

1. Choose Configuration >> Utilities from the menu bar to open the utilities screen.



2. Choose the IP Setup button. The IP Setup window will open.



3. To use DHCP, choose the Obtain IP address automatically button.

4. To specify specific values for the IP address, enter values for the IP address, subnet mask, and gateway. Enter an address by choosing a field and entering a value using the number pad. Repeat this process until the entire address is specified.

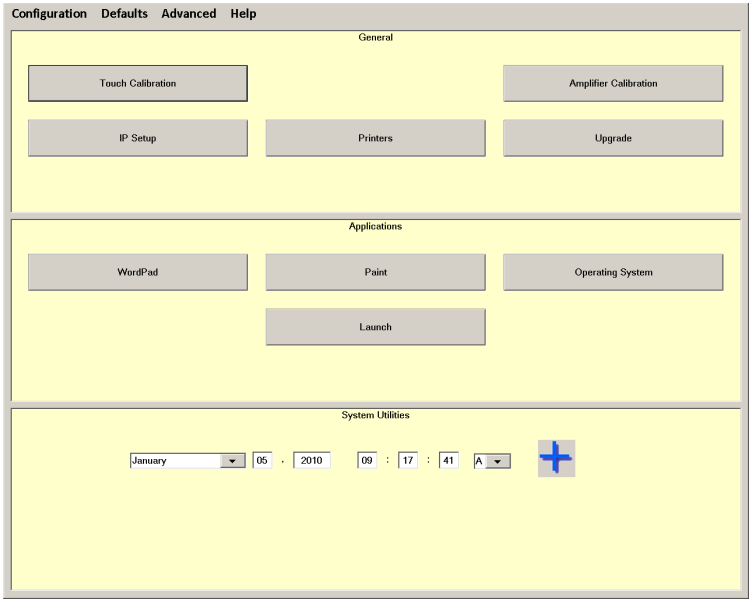
   When an IP address is used, it will be added to the Recent Settings drop-down list. To select this address in the future, select it from the list instead of entering it again.

5. Choose OK.

Chapter 14: Utilities and service options

Utilities screen

Choose Configuration >> Utilities from the menu bar to open the utilities screen.



Utilities and other options are available from this screen.

Restoring default settings

In some situations, it may be helpful to restore settings to the factory-default state.

1. Choose Configuration >> Utilities from the menu bar to open the utilities screen.

2. Choose Defaults. A sub menu will appear. Choose a default option.

    - **Channel Settings** - This option restores default settings for channel amplifier inputs, base channels, derived channels, and event inputs.

    - **Realtime View** - This option restores the default view settings for Realtime mode.

- **Realtime Panel** - This option restores the default control panel for Realtime mode.
- **Scope View** - This option restores the default view settings for Scope mode.
- **Scope Panel** - This option restores the default control panel for Scope mode.
- **Review Panel** - This option restores the default control panel for Review mode.
- **Trigger Settings** - This option restores the default trigger settings.
- **Capture Settings** - This option restores the default data capture settings.
- **System** - This option restores the default settings for all previous options.

A confirmation message will appear.

3. To confirm the default settings, choose OK. To cancel the default settings, choose Cancel.

Showing and hiding the cursor

By default, the cursor/pointer is not visible on the display. However, you can show/hide the cursor from the utilities screen. When the cursor is visible, an arrow icon will indicate the current focus on the touch screen. The arrow icon will not be displayed when the cursor is set to hidden.

1. Choose Configuration >> Utilities from the menu bar to open the utilities screen.
2. Choose Advanced >> Show Cursor to toggle between the visible and hidden cursor.

Selecting a language

1. Choose Configuration >> Utilities from the menu bar to open the utilities screen.
2. Choose Help >> Language from the menu bar. A sub menu will appear. The following language options are available:

- American
- British
- French
- Italian
- German

3. Select a language. The selected language will be used for all text in the software.

General utilities

The following general utilities are available in the utilities screen.

Touch Calibration
The Touch Calibration utility is used to set up the touch screen calibration and other display options.

Amplifier Calibration
The Amplifier Calibration utility is used to calibrate input channels.

IP Setup
The IP Setup utility is used to specify IP address information for the recorder in network environments.

Printers
The Printers utility is used to install an external printer.

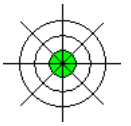
Upgrade
The Upgrade utility is used to upgrade the system software.

Related topics:
- *Calibrating the touch screen* on page 14-3
- *Calibrating channels* on page 14-4
- *Modifying IP information* on page 13-1
- *Installing Windows-based printers* on page 4-5
- *Upgrading the system software* on page 14-4

Calibrating the touch screen

1. Choose Configuration >> Utilities from the menu bar to open the utilities screen.

2. Choose the Touch Calibration button. The touch-screen calibration process will start and a touch target will be displayed.



3. Touch and hold the center of the target for approximately 2-3 seconds. Then release the target. If a new touch target is displayed, repeat this process.

   When the touch-screen calibration process is complete, a confirmation message will appear. Choose the Accept option.

Upgrading the system software

Software upgrades may be released in the future for the recorder. Use the following instructions to upgrade the system software.

1. Insert a USB flash memory drive containing the software upgrade.
2. Choose Configuration >> Utilities from the menu bar to open the utilities screen.
3. Choose Upgrade. The system software will be upgraded.

Calibrating channels

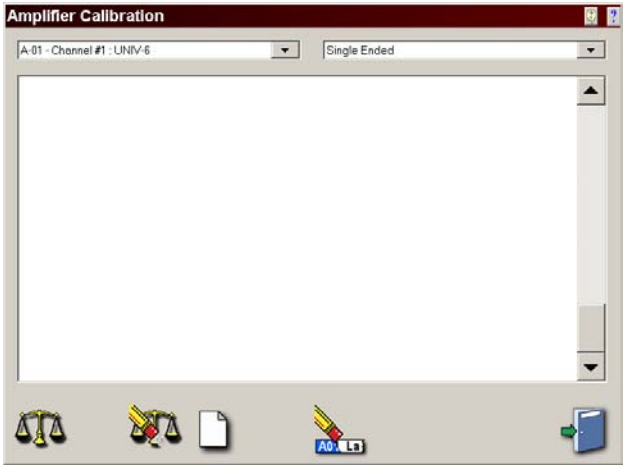
The calibration function adjusts the calibration of the signal input channels based on known standards. Calibration should be performed approximately once per year. New recorders are calibrated at the factory.

Note: *Known voltages must be applied during the calibration process. Ensure that all required voltages are available prior to starting the calibration process.*

1. Choose Configuration >> Utilities from the menu bar to open the utilities screen.
2. Choose the Amplifier Calibration button. The Enter Password window will open. Enter the calibration password and choose OK.

   Note: *The default calibration password is "tmx" (lowercase, without the quotation marks).*

The Analog Calibration window will open.



3. Select a channel and the input settings for the channel.

4. If necessary, you can view the calibration factors for the selected channel by choosing the Show Factors icon.

5. If necessary, you can clear the calibration settings for the selected channel by choosing the Clear Factors icon.

Clearing a channel's calibration settings will set these values to null. After being cleared, these channels will need to be calibrated before they will work properly.

6. Choose the Start Calibration icon.

7. The system will prompt you to short the inputs of the channel. This enables the baseline calibration for each attenuator to be performed.

8. The recorder will prompt for specific voltages to be placed at the selected input channel. These voltages will differ based on the input module being calibrated.

   Note: *The calibration process for each voltage may require several minutes.*

Module	Required Voltages
UNIV-6 (Single Ended)	0.5, 5.0, 25.0, 50.0, 100.0 V
UNIV-6 (Differential)	25.0, 100.0, 500.0 mV

9. When all attenuators have been calibrated, calibrate another channel if necessary.

10. If necessary, you can cancel calibration by choosing the Abort Calibration icon.

    Pressing this icon will stop the calibration process, however, the process must be completed from start to finish to ensure that the channel's attenuators will work properly.

11. When finished, choose the X icon to close the Calibration window.

Related topics:
- *Changing passwords* on page 14-9

Application utilities

The following application utilities are available in the utilities screen.

WordPad

The WordPad utility is used to launch Microsoft ® WordPad. This application is a text editor that can be used to create and save basic documents. A keyboard must be used to allow typing while in WordPad.

To launch WordPad, choose the WordPad button. When WordPad is open, all other utility options will become unavailable.

Paint

The Paint utility is used to launch Microsoft ® Paint. This application is a graphics editor that can be used to create and save basic graphic and screen captures.

To launch Paint, choose the Paint button. When Paint is open, all other utility options will become unavailable.

Launch

The Launch utility is used to launch other applications while the TMX software is running.

Operating System

The Operating System utility is used to exit the TMX software and access the Windows operating system.

Related topics:

- *Using the Launch utility* on page 14-7
- *Using the Operating System utility* on page 14-8

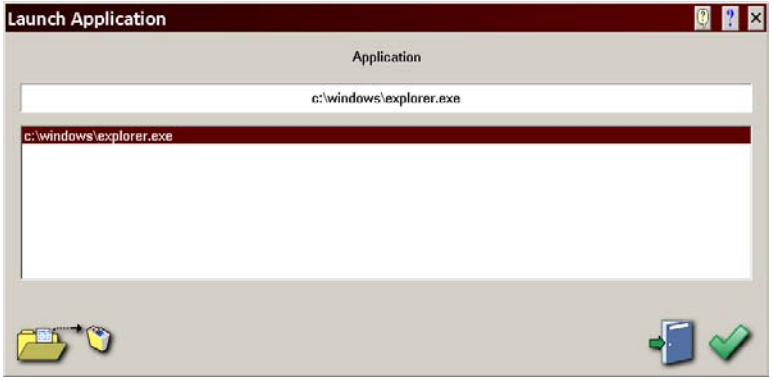
Using the Launch utility

The Launch utility is used to launch other applications while the TMX software is running.

Note: *Astro-Med, Inc. does not guarantee that any application can be used while running the TMX.*

1. Choose Configuration >> Utilities from the menu bar to open the utilities screen.

2. Choose the Launch button. The Launch Application window will open.



3. Choose the Application field. Then enter the full path of a program's ".exe" file using the keypad and choose the OK button. Choose OK in the Launch Application window. The program will be launched.

   When a program is successfully launched, the path will be added to the list in this window. To launch the program in the future, select it from the list and choose OK instead of entering the path again.

   Paths can be removed from this list by selecting the path and choosing the delete button.

Using the Operating System utility

The Operating System utility is used to exit the TMX software and access the Windows operating system.

Note: *You will need to use this procedure to access any applications that aren't listed on the Utilities screen.*

1. Choose Configuration >> Utilities from the menu bar to open the utilities screen.

2. Choose the Operating System button. The Enter Password window will open.

3. Enter the system password and choose OK. The TMX software will shut down, providing access to the operating system.

   Note: *The default system password is "tmx" (lowercase, without the quotation marks).*

Related topics:

- *Changing passwords* on page 14-9

Setting the system time

1. Choose Configuration >> Utilities from the menu bar to open the utilities screen.

2. Use the time fields and drop-down lists to enter the current date and time.

Changing passwords

This section describes how to change the system and calibration passwords.

Changing the system password

The system password is a security tool that prevents unauthorized access to the menu bar, operating system, and other functions.

1. From Realtime, Scope, or Review mode, choose Security >> Set Password. The Enter Current Password window will open.

2. Enter the current system password and choose OK. The Enter New Password window will open.

   Note: *The default system password is "tmx" (lowercase, without the quotation marks).*

3. Enter the new system password and choose OK. The Verify New Password window will open.

4. Re-enter the new system password and choose OK.

Changing the calibration password

You are required to enter a password in order to access the Analog Calibration window. By default, this password is the same as the system password. However, you can assign a unique password to the Analog Calibration window.

1. Choose Configuration >> Utilities from the menu bar to open the utilities screen.

2. Choose Advanced >> Set Calibration Password. The Enter Current Password window will open.

3. Enter the current calibration password and choose OK. The Enter New Password window will open.

   Note: *The default calibration password is "tmx" (lowercase, without the quotation marks).*

4. Enter the new calibration password and choose OK. The Verify New Password window will open.

5. Re-enter the new calibration password and choose OK.

Viewing and modifying utility / D|O port settings

1. From Realtime, Scope, or Review mode, choose Service >> Utility Port. The Utility Port Settings window will open.

   This window provides a graphical representation of the utility port pins and setup options.

2. Specify a chart/page mark input option.

   - **Chart Mark** - The Chart Mark option prints a mark on the printed chart.

   - **Page Mark** - The Page Mark option prints a full-page mark on the scrolling waveform display area.

3. To modify alarm output settings, select a channel (or multiple channels with "or" logic). When an alarm condition is met for the selected channels, the alarm output will occur.

4. Specify the print input option.

   - **Print Demand** - The Print Demand option will print the demand buffer.

   - **Print ID** - The Print ID option will print the channel IDs.

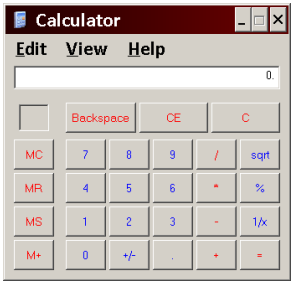
   - **Print Channel Labels** - The Print Channel Labels option will print the labels assigned to each channel.

5. Choose OK.

Using the calculator

Use the following instructions to access the calculator program, which is part of the Windows operating system.

1. From Realtime, Scope, or Review mode, choose Service >> Calculator to open the Calculator window.



2. Choose the X button to close the Calculator window.

Appendix A: Specifications

Color display

Type	Active matrix color LCD (TFT)
Viewing Area	17.0 inch (diagonal)
Resolution	1280 x 1024
Touch	Full screen, resistive

Signal modules

Maximum Modules	6 (3 in main chassis, 3 in external chassis)
Maximum Waveforms	96 (48 in main chassis, 48 in external chassis)
Event inputs	32 (16 in main chassis, 16 in external chassis)
Engineering Units	User defined units with y=mx+b scaling
Pre-capture Filter	Lowpass, highpass, bandpass, bandstop, RMS
Post-capture Filter	Lowpass, highpass, bandpass, bandstop, RMS
Calibration	Semi-automated to external reference

Data Acquisition

Recording Method	Internal SATA disk drive
Maximum Sample Rate	800,000 samples/second/channel (module dependent)
Minimum Sample Rate	1 sample/second
Multiple Sample Rates	Up to 4 integer divider sample rates

Total Capacity	1 Terabyte (over 400 million samples)
Time Stamp	Time and Date automatically saved with data
Header	Information on units, range, sample rates, etc. saved with data
Events	Recorded with data
Trigger Types	Edge, window, slew, event, manual, external
Trigger Point	Pre and post trigger is user adjustable (single sample rate)
Auto Re-Arm	Allows automatic stacking of captures
Shadow File	Recorded simultaneously with data capture

Power

Input Voltage Range	100 to 264 VAC
Frequency Range	47 Hz to 63 Hz
Power Factor	0.99

Physical

Enclosure	Aluminum with thermoplastic endcaps
Dimensions (inches)	14.5" x 18.8" x 7.5"

Appendix B: Icon descriptions

Common system icons

The following list describes the functions of common icons.

Icon	Description
	**OK** saves the information entered in a window and then closes the window.
	**Exit** cancels the action being performed in a window and closes the window without saving any changes.
	**Apply** saves the information modified in a window without closing the window.
	**Select All** selects/highlights all items in a list box.
	**Clear Selection** removes the selections/highlights from all items in a list box.

Realtime mode control panel

This section describes all icon buttons that can be added to the Realtime mode control panel.

Icon	Description
	**Load Signals File** loads a signal setup file. *File >> Signals*
	**Load View File** loads a view setup file. *File >> View*
	**Load Control Panel File** loads a control panel setup file. *File >> Control Panel*
	**Load Capture File** loads a data capture setup file. *File >> Capture*

Icon	Description
	**Load Trigger File** loads a trigger setup file. *File >> Trigger*
	**Load Global File** loads a global setup file. *File >> Global*
	**Print Screen** prints the contents of the display. This function is available only when a printer is installed. *File >> Print Screen*
	**Go to Scope** launches Scope mode. *Configuration >> Scope*
	**Go to Review** launches Review mode. *Configuration >> Review*
	**Go to Utilities** launches the Utilities screen. *Configuration >> Utilities*
	**View Wizard** opens the Display Wizard window, which is used to set up grid and event display options. *View >> Wizard*
	**Add Grid** opens the Grid Properties window, which is used to set up a new grid. *View >> Add Grid*
	**Add Event** opens the Event Properties window, which is used to set up a new event. *View >> Add Event*
	**Monitor Run/Halt** starts and stops the on-screen display from running. *View >> Monitor Run/Halt*
	**Channel Information** displays and hides the Channel Information window, which is used in conjunction with cursors to measure signals. *Analysis >> Channel Information*

Icon	Description
	**Meter** opens the Meter Properties window, which is used to set up a meter. *Analysis >> Meter*
	**XYY Plot** opens the XYY Properties window, which is used to set up an XYY plot. *Analysis >> XYY*
	**Realtime Settings** opens the Realtime Settings window, which is used to configure Realtime mode. *Settings >> Realtime*
	**Channel Settings** opens the Channel Settings window, which is used to set up amplifier inputs, base channels, derived channels, and event inputs. *Settings >> Channels*
	**Control Panel Settings** opens the Panel Settings window, which is used to set up control panel icons. *Settings >> Control Panel*
	**Set Speed** changes the speed of the on-screen chart. In this example, the chart speed will be changed to 5 mm/s when the icon is pressed. Custom speed icons can be configured during the control panel setup process. *Settings >> Speed >> mm/s* or *mm/m*
	**Cursor A** displays and hides cursor A. *Cursors >> Cursor A*
	**Cursor B** displays and hides cursor B. *Cursors >> Cursor B*
	**Active Cursor** changes the active cursor between A, B, or A & B. *Cursors >> Active Cursor*
	**Move Cursor Left** moves the active cursor(s) to the left each time the icon is pressed. *Cursors >> Move Left*

Icon	Description
	**Move Cursor Right** moves the active cursor(s) to the right each time the icon is pressed. *Cursors >> Move Right*
	**Capture Settings** opens the Capture Settings window, which is used to configure data capture setup options. *Capture >> Capture Settings*
	**Trigger Settings** opens the Trigger Settings window, which is used to set up data capture triggers and aborts. *Capture >> Trigger/Abort Settings*
	**Arm Capture** arms (starts) a data capture using the configured data capture setup options. *Capture >> Arm*
	**Manual Trigger** triggers a data capture manually. *Capture >> Manual Trigger*
	**Abort Capture** cancels the data capture in progress. *Capture >> Manual Abort*
	**Trigger Indicator** indicates when a trigger occurs by displaying a yellow circle. *Capture >> Trigger Indicator*
	**Capture Indicator** indicates when a data capture is in progress by illuminating. *Capture >> Capture Indicator*
	**Calculator** launches the Calculator application, which is part of the Windows operating system. *Service >> Calculator*
	**Operations Manual** opens the operations manual. *Help >> Operations Manual*

Scope mode control panel

This section describes all icon buttons that can be added to the Scope mode control panel.

Icon	Description
	**Load Signals File** loads a signal setup file. *File >> Signals*
	**Load View File** loads a view setup file. *File >> View*
	**Load Control Panel File** loads a control panel setup file. *File >> Control Panel*
	**Load Capture File** loads a data capture setup file. *File >> Capture*
	**Load Trigger File** loads a trigger setup file. *File >> Trigger*
	**Load Global File** loads a global setup file. *File >> Global*
	**Print Screen** prints the contents of the display. This function is available only when a printer is installed. *File >> Print Screen*
	**Save as Data Capture Record** archives an entire scope capture, the currently displayed page of the capture, or the portion of the capture between cursors. *File >> Save as Data Capture Record*
	**Go to Realtime** launches Realtime mode. *Configuration >> Realtime*
	**Go to Review** launches Review mode. *Configuration >> Review*
	**Go to Utilities** launches the Utilities screen. *Configuration >> Utilities*

Icon	Description
	**View Wizard** opens the Display Wizard window, which is used to set up grid and event display options. *View >> Wizard*
	**Add Grid** opens the Grid Properties window, which is used to set up a new grid. *View >> Add Grid*
	**Add Event** opens the Event Properties window, which is used to set up a new event. *View >> Add Event*
	**Trigger Line** shows and hides the trigger line in the waveform display area. *View >> Trigger Line*
	**IDs** displays an identification label for each channel in the waveform display area. *View >> IDs*
	**Channel Information** displays and hides the Channel Information window, which is used in conjunction with cursors to measure signals. *Analysis >> Channel Information*
	**Meter** opens the Meter Properties window, which is used to set up a meter. *Analysis >> Meter*
	**XYY Plot** opens the XYY Properties window, which is used to set up an XYY plot. *Analysis >> XYY*
	**Fourier Transform** opens the Fourier Transform Properties window. *Analysis >> Fourier Transform*
	**Scope Settings** opens the Scope Settings window, which is used to configure scope captures. *Settings >> Scope*

Icon	Description
	**Channel Settings** opens the Channel Settings window, which is used to set up amplifier inputs, base channels, derived channels, and event inputs. *Settings >> Channels*
	**Control Panel Settings** opens the Panel Settings window, which is used to set up control panel icons. *Settings >> Control Panel*
	**Increase Timebase** increases the timebase used for scope captures. *Settings >> Timebase Up*
	**Decrease Timebase** decreases the timebase used for scope captures. *Settings >> Timebase Down*
	**Pre-Trigger Percent** opens the Pre-Trigger Percent window, which is used to enter the percent of the scope capture allocated for pre-trigger data. *Settings >> Pre-Trigger Percent*
	**Scope Arm/Abort** arms and aborts scope captures. *Settings >> Arm/Abort*
	**Cursor A** displays and hides cursor A. *Cursors >> Cursor A*
	**Cursor B** displays and hides cursor B. *Cursors >> Cursor B*
	**Active Cursor** changes the active cursor between A, B, or A & B. *Cursors >> Active Cursor*
	**Move Cursor Left** moves the active cursor(s) to the left each time the icon is pressed. *Cursors >> Move Left*

Icon	Description
	**Move Cursor Right** moves the active cursor(s) to the right each time the icon is pressed. *Cursors >> Move Right*
	**Capture Settings** opens the Capture Settings window, which is used to configure data capture setup options. *Capture >> Capture Settings*
	**Trigger Settings** opens the Trigger Settings window, which is used to set up data capture triggers and aborts. *Capture >> Trigger/Abort Settings*
	**Arm Capture** arms (starts) a data capture using the configured data capture setup options. *Capture >> Arm*
	**Manual Trigger** triggers a data capture manually. *Capture >> Manual Trigger*
	**Abort Capture** cancels the data capture in progress. *Capture >> Manual Abort*
	**Trigger Indicator** indicates when a trigger occurs by displaying a yellow circle. *Capture >> Trigger Indicator*
	**Capture Indicator** indicates when a data capture is in progress by illuminating. *Capture >> Capture Indicator*
	**Calculator** launches the Calculator application, which is part of the Windows operating system. *Service >> Calculator*
	**Operations Manual** opens the operations manual. *Help >> Operations Manual*

Review mode control panel

This section describes all icon buttons that can be added to the Review mode control panel.

Icon	Description
	**Archive File** archives an entire capture, the currently displayed page of the capture, or the portion of the capture between cursors. *File >> Archive File*
	**Print Screen** prints the contents of the display. This function is available only when a printer is installed. *File >> Print Screen*
	**Show Next File** opens the next file on the selected drive during review. *File >> Show Next File*
	**Show Previous File** opens the previous file on the selected drive during review. *File >> Show Previous File*
	**Return to Data Capture Record** closes the embedded scope capture and returns to the original data capture file. *File >> Return to DCR*
	**Go to Scope** launches Scope mode. *Configuration >> Scope*
	**Go to Realtime** launches Realtime mode. *Configuration >> Realtime*
	**Go to Utilities** launches the Utilities screen. *Configuration >> Utilities*
	**View Wizard** opens the Display Wizard window, which is used to set up grid and event display options. *View >> Wizard*

Icon	Description
	**Add Grid** opens the Grid Properties window, which is used to set up a new grid. *View >> Add Grid*
	**Add Event** opens the Event Properties window, which is used to set up a new event. *View >> Add Event*
	**Trigger Line** shows and hides the trigger line in the waveform display area. *View >> Trigger Line*
	**IDs** displays an identification label for each channel in the waveform display area. *View >> IDs*
	**Channel Information** displays and hides the Channel Information window, which is used in conjunction with cursors to measure signals. *Analysis >> Channel Information*
	**Meter** opens the Meter Properties window, which is used to set up a meter. *Analysis >> Meter*
	**XYY Plot** opens the XYY Properties window, which is used to set up an XYY plot. *Analysis >> XYY*
	**Fourier Transform** opens the Fourier Transform Properties window. *Analysis >> Fourier Transform*
	**Zoom** opens the Select Zoom Area window. *Analysis >> Open Zoom Window*
	**Channel Settings** opens the Channel Settings window, which is used to set up amplifier inputs, base channels, derived channels, and event inputs. *Settings >> Channels*

Icon	Description
	**Control Panel Settings** opens the Panel Settings window, which is used to set up control panel icons. *Settings >> Control Panel*
	**Go to Start** scrolls the display to the start of the file. *Display >> Goto Start*
	**Go to End** scrolls the display to the end of the file. *Display >> Goto End*
	**Go to Trigger** scrolls the display to the trigger point in a file. *Display >> Goto Trigger*
	**Go to Cursor A** scrolls the display to the location of cursor A. *Display >> Goto Cursor A*
	**Go to Cursor B** scrolls the display to the location of cursor B. *Display >> Goto Cursor B*
	**Go to Cursors A and B** compresses or expands the display to the portion of the file between cursors A and B. *Display >> Goto Cursors A <-> B*
	**Go to Advanced** opens the Advanced Search window, which is used to navigate through the file using specific search criteria. *Display >> Goto Advanced*
	**Show All** compresses the file to fit the entire file on the display. *Display >> Show All*
	**Compress** compresses the display of files to fit more of the file on the screen. *Display >> Compress*
	**Expand** expands the display of files to fit less of the file on the screen. *Display >> Expand*

Icon	Description
	**Scroll Forward** scrolls the chart forward. *Scroll >> Scroll Forward*
	**Scroll Fast Forward** scrolls the chart forward quickly. *Scroll >> Fast Forward*
	**Scroll Stop** stops the chart from scrolling. *Scroll >> Scroll Stop*
	**Scroll Back** scrolls the chart backward. *Scroll >> Scroll Backward*
	**Scroll Fast Back** scrolls the chart backward quickly. *Scroll >> Rewind*
	**Cursor A** displays and hides cursor A. *Cursors >> Cursor A*
	**Cursor B** displays and hides cursor B. *Cursors >> Cursor B*
	**Active Cursor** changes the active cursor between A, B, or A & B. *Cursors >> Active Cursor*
	**Move Cursor Left** moves the active cursor(s) to the left each time the icon is pressed. *Cursors >> Move Left*
	**Move Cursor Right** moves the active cursor(s) to the right each time the icon is pressed. *Cursors >> Move Right*
	**Calculator** launches the Calculator application, which is part of the Windows operating system. *Service >> Calculator*
	**Operations Manual** opens the operations manual. *Help >> Operations Manual*

Index

G

H

I

K

L

M

N

O

P

R

S

T

U

V

W

X

Z